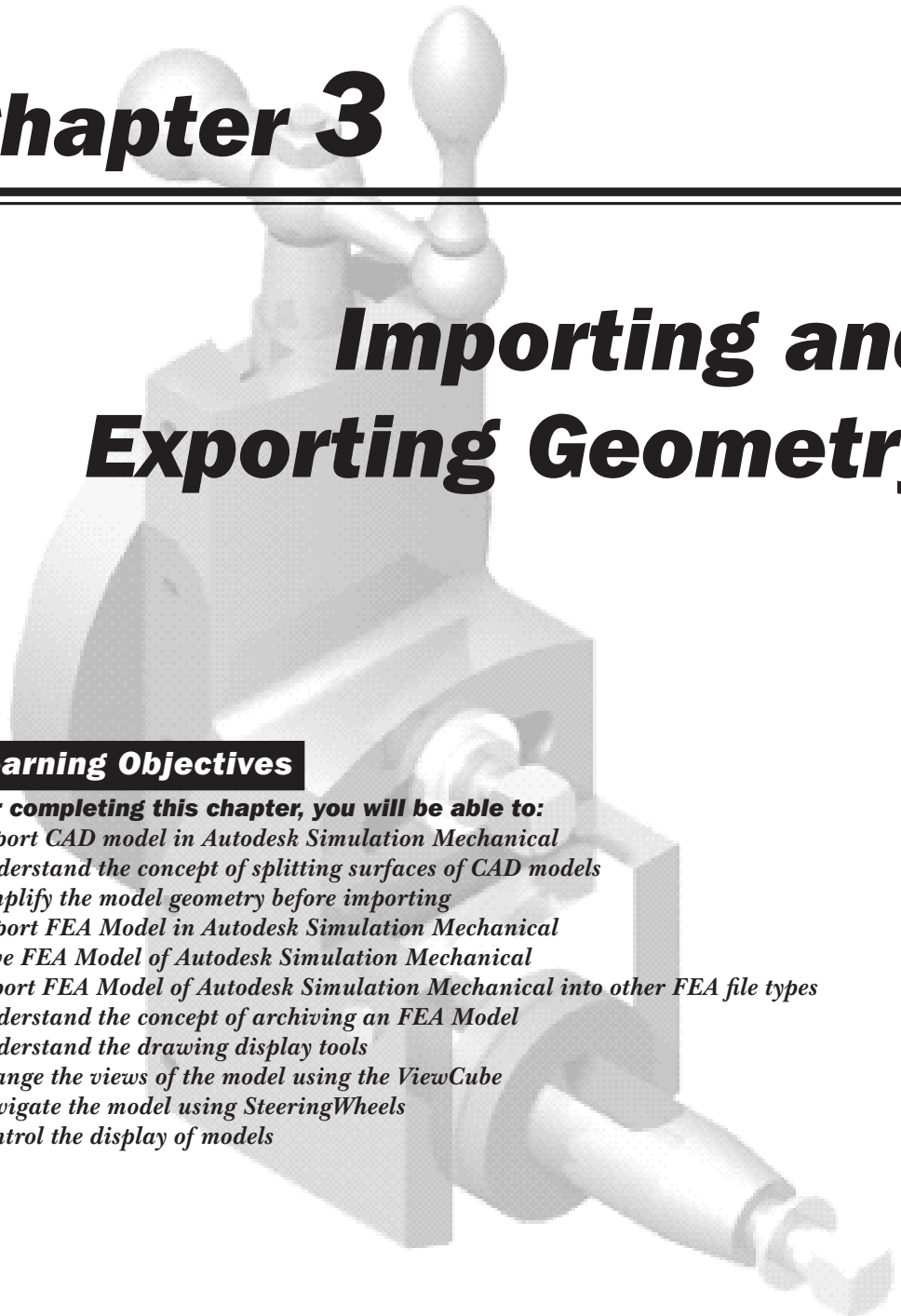
Chapter 3

Importing and Exporting Geometry



Learning Objectives

After completing this chapter, you will be able to:

- *Import CAD model in Autodesk Simulation Mechanical*
- *Understand the concept of splitting surfaces of CAD models*
- *Simplify the model geometry before importing*
- *Import FEA Model in Autodesk Simulation Mechanical*
- *Save FEA Model of Autodesk Simulation Mechanical*
- *Export FEA Model of Autodesk Simulation Mechanical into other FEA file types*
- *Understand the concept of archiving an FEA Model*
- *Understand the drawing display tools*
- *Change the views of the model using the ViewCube*
- *Navigate the model using SteeringWheels*
- *Control the display of models*

INTRODUCTION

In Autodesk Simulation Mechanical, you can open the existing Autodesk simulation FEA models or CAD models to run the analysis. Autodesk Simulation Mechanical provides a geometry data translation interface to various leading CAD systems. As a result, you can directly open the CAD models saved in AutoCAD DWG (*.dwg), AutoCAD DXF (*.dxf), Autodesk Inventor (*.iam, *.ipt), Autodesk Fusion (*.dwg), SolidWorks (*.sldasm, *.asm, *.sldprt, *.prt), Pro/ENGINEER (*.prt, *.asm) and many other file formats in this software. Similarly, it provides data translation interface to various other FEA software such as Abaqus, ANSYS, Nastran, Stereolithography, Blue Ridge Numerics, and so on. In addition, you can also open the models saved in neutral file format such as *.SAT, *.STP, *.IGES, and *.STEP.

The FEA models created in Autodesk Simulation Mechanical are saved in the *.fem* file format. Also, if you open a CAD model in Autodesk Simulation Mechanical and then save it, the model will be saved in the *.fem* file format. You can export an FEA model created in Autodesk Simulation Mechanical to other FEA file formats such as ABAQUS (*.inp), ANSYS (*.cdb, *.ans), Blue Ridge Numeric (*.neu), FEMAP Neutral (*.neu), NASTRAN (*.nas, *.bdf, *.dat), PATRAN (*.pat), and SDRC Universal (*.unv).

IMPORTING CAD MODEL

As discussed earlier, the Autodesk Simulation Mechanical provides data translation interface to leading CAD systems for directly opening the CAD models in it. To open a CAD model, invoke the **Open** dialog box, if not invoked already. To do so, choose the **Open** tool from the **Quick Access Toolbar** or from the **Launch** panel of the **Start & Learn** tab in the **Ribbon**; the **Open** dialog box will be invoked, as shown in Figure 3-1. In this dialog box, by default, the **Autodesk Simulation FEA Model (*.fem)** file type is selected in the **File of type** drop-down list, refer to Figure 3-1. As a result, the **Open** dialog box will list only those models that have *.*fem* file extension. The *.*fem* is the file extension of Autodesk Simulation Mechanical. Select the required CAD file type from the **Files of type** drop-down list of the **Open** dialog box. Figure 3-2 shows the **Open** dialog box with the **Files of type** drop-down list displayed. To open the model created in Autodesk Inventor, select the **Autodesk Inventor Files (*.ipt;*.iam)** from the **Files of type** drop-down list. Next, browse to the location where the Inventor part files are saved and then select the file to be imported. After selecting the required file, choose the **Open** button from the dialog box; the **Import Inventor Work Points** message window appears, prompting you to specify whether you want to import work points along with the model. Choose the **Yes** button to import the work points while meshing as it helps to place a node at the work point of the imported model. On doing so, the **Choose Analysis Type** dialog box will be displayed, as shown in Figure 3-3. Now, select the type of analysis that you want to carry out on the model by using the options in this dialog box. After defining the analysis type, choose **OK**; the selected model will be imported and opened in Autodesk Simulation Mechanical, refer to Figure 3-4. Similarly, you can select a CAD file type of other CAD software whose model is to be imported in Autodesk Simulation Mechanical.

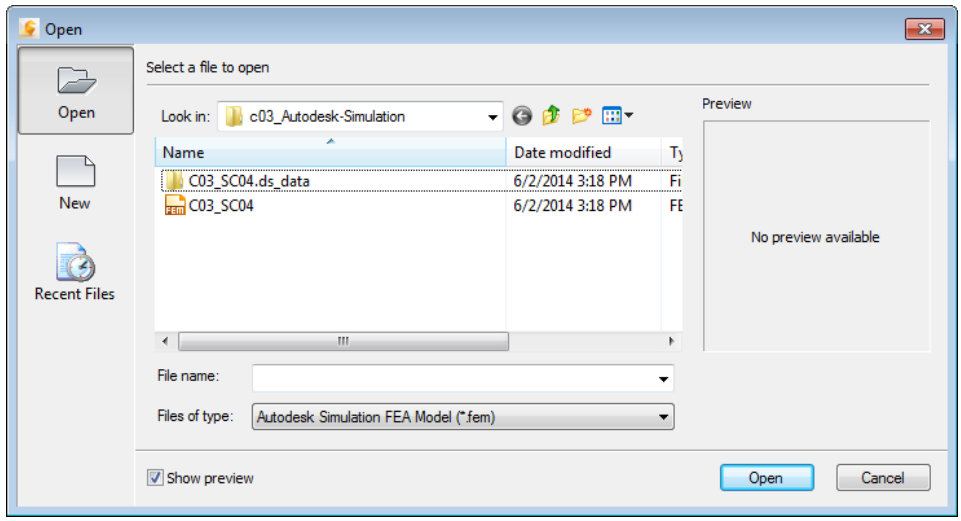


Figure 3-1 *The* ***Open*** *dialog box*

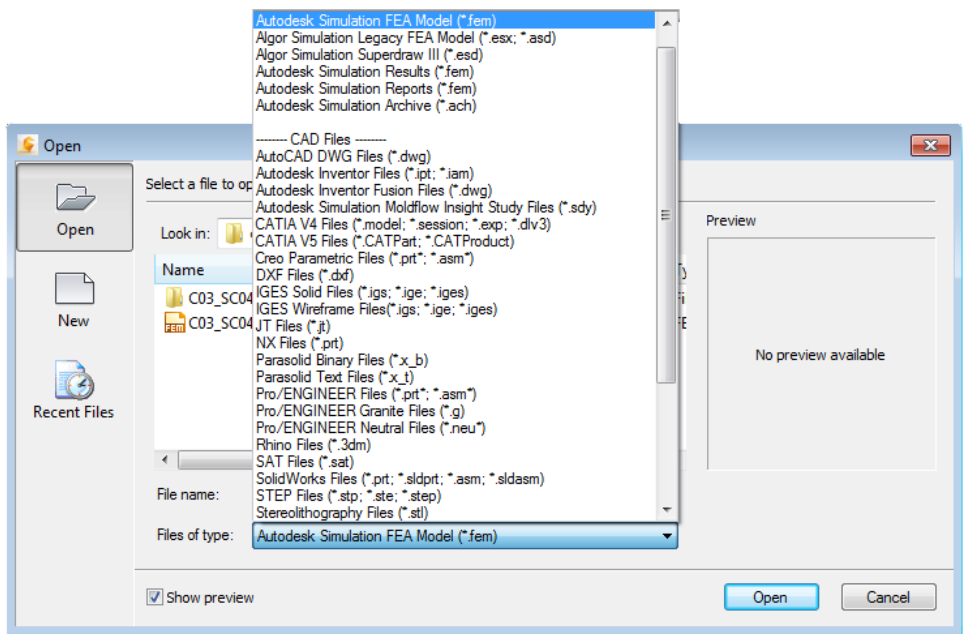


Figure 3-2 *The* ***Open*** *dialog box with the expanded view of the* ***Files of type*** *drop-down list*

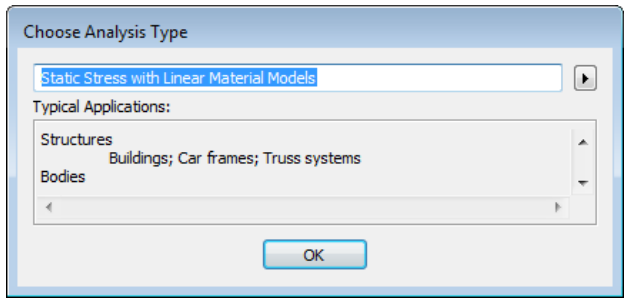


Figure 3-3 *The* ***Choose Analysis Type*** *dialog box*

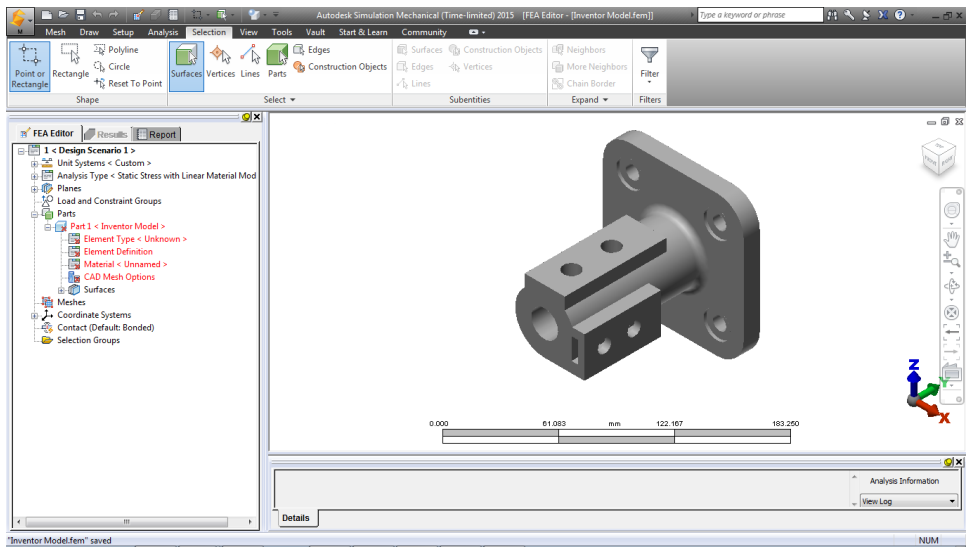


Figure 3-4 *The Inventor model opened in Autodesk Simulation Mechanical*

When you import any CAD model, 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 will be the same as that of the imported CAD model, but the units for force, time, and so on, may not be the same. However, you can modify the default unit system by using the **Unit System** dialog box that will be displayed by right-clicking on the **Unit Systems** node of the **Tree View** and choosing the **New** option from the shortcut menu displayed, refer to Figure 3-5. The method of setting unit system is discussed in Chapter 2.

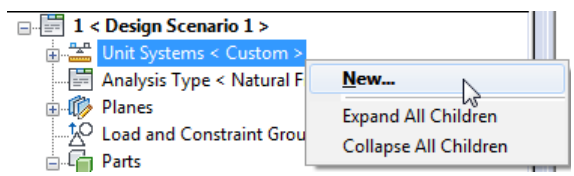


Figure 3-5 *Shortcut menu displayed*

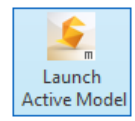
Note

*In addition to importing CAD models by using the **Open** dialog box, you can directly transfer them from their respective software to Autodesk Simulation Mechanical.*

Importing 3D Models from Autodesk Inventor

As discussed earlier, you can open the models created in Autodesk Inventor by selecting the **Autodesk Inventor Files (*.ipt;*.iam)** file type from the **Files of type** drop-down list in the **Open** dialog box of Autodesk Simulation Mechanical. However, if the model to be imported is created in the lower version of Autodesk Inventor then it will be updated to the current version of Autodesk Simulation Mechanical and cannot be opened back in the same version of Autodesk Inventor. For example, if the model is created in Autodesk Inventor 2008 and you open it in the Autodesk Simulation Mechanical 2015 for analysis, then the version of that model will be updated from 2008 to 2015 and you cannot open it again in Autodesk Inventor 2008 version. To avoid facing such problems, you can directly import the model from Autodesk Inventor to Autodesk Simulation Mechanical.

To import the model directly from Autodesk Inventor to Autodesk Simulation Mechanical, Autodesk Inventor must be installed on your system. Next, start Autodesk Inventor and then open the model that needs to be transferred.



Once the model is opened into Autodesk Inventor, choose the **Simulation** tab in the **Ribbon**. Next, choose the **Launch Active Model** tool from the **Simulation Mechanical 2015** panel of this tab; the **Import Inventor Work Points** message window will be displayed. Choose the **Yes** button from this message window; the starting screen of Autodesk Simulation Mechanical with the **Choose Analysis Type** dialog box will be displayed, refer to Figure 3-6.

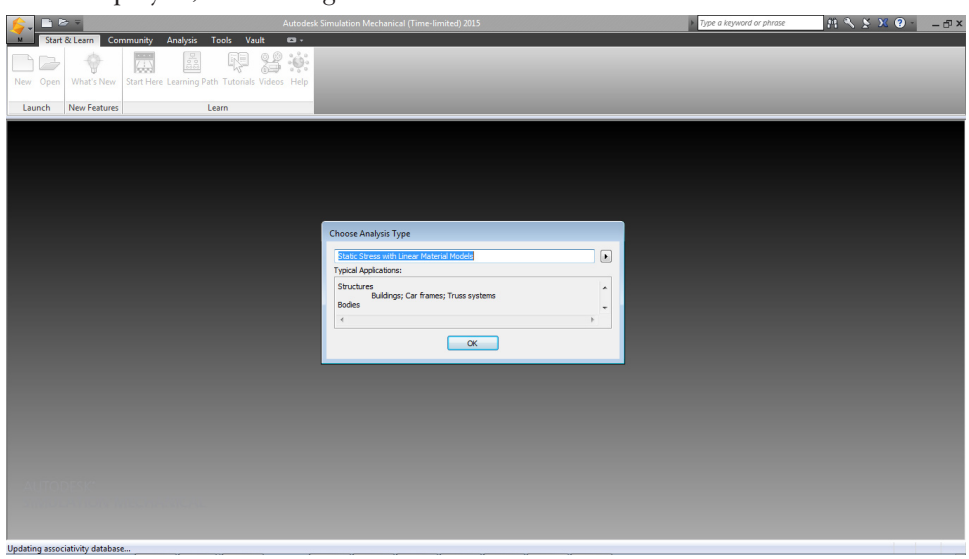


Figure 3-6 *Autodesk Simulation Mechanical with the **Choose Analysis Type** dialog box*

Tip. *You can also simplify the geometry of a model before transferring it from Autodesk Inventor to Autodesk Simulation Mechanical by suppressing some of its features such as fillets, chamfers, holes, and so on, if they are not required in the analysis process. The simplification of the Inventor model will be done by using the* ***Simplify Model*** *tool that is available in the* ***Simulation Mechanical 2015*** *panel of the* ***Simulation*** *tab in the* ***Ribbon****. You will learn more about simplifying the Inventor models later in this chapter.*

Select the required analysis type from the **Choose Analysis Type** dialog box. By default, the **Static Stress with Linear Material Models** analysis type is selected in this dialog box. After selecting the required analysis type, choose the **OK** button; the Inventor model will be opened in Autodesk Simulation Mechanical, as shown in Figure 3-7.

Note

If you have assigned any material property to the inventor model, then while transferring the model from Autodesk Inventor to Autodesk Simulation Mechanical, the material properties such as mass density, modulus of elasticity, poisson's ratio, thermal expansion coefficient, yield strength, ultimate tensile strength assigned to the Inventor model will also be transferred to Autodesk Simulation Mechanical.

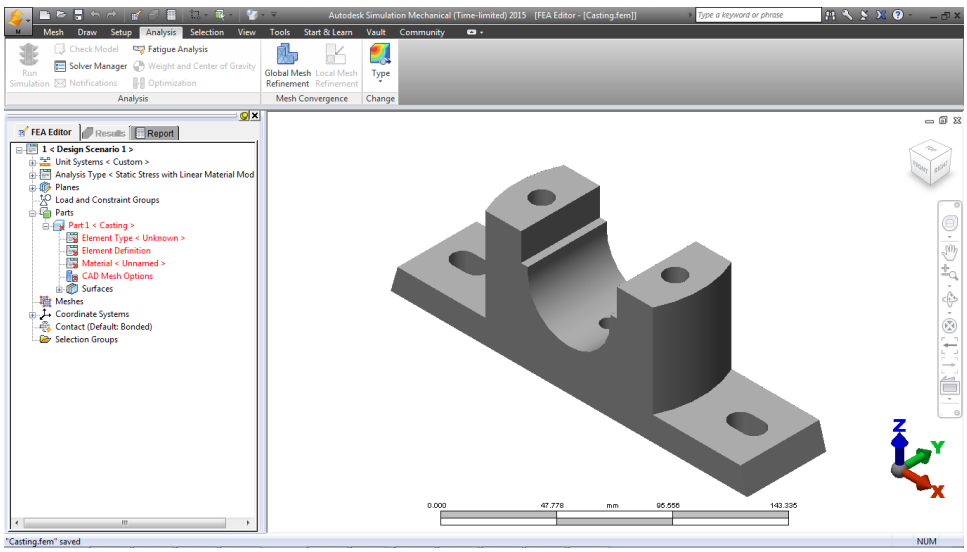
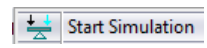


Figure 3-7 *Autodesk Inventor model imported to Autodesk Simulation Mechanical*

Importing 3D Models from SolidWorks

Similar to importing inventor model directly from Autodesk Inventor software to Autodesk Simulation Mechanical, you can also import the SolidWorks models directly from SolidWorks software to Autodesk Simulation Mechanical. To import the model directly from SolidWorks to Autodesk Simulation Mechanical, SolidWorks software must be installed on your system. Next, start SolidWorks and then open the model that needs to be transferred to Autodesk Simulation Mechanical, refer to Figure 3-8.



Once the model has been opened in SolidWorks, move the cursor over the SolidWorks logo that is available at the upper left corner of the screen; the SolidWorks menus will be displayed, refer to Figure 3-8. Next, from the SolidWorks menus, choose **Autodesk Simulation > Start Simulation**; the starting screen of Autodesk Simulation Mechanical with the **Choose Analysis Type** dialog box will be displayed.

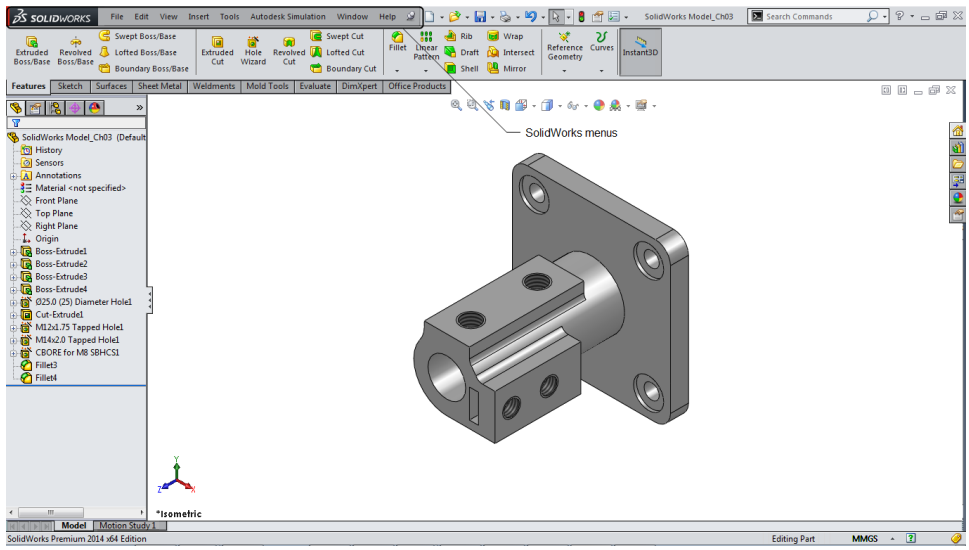


Figure 3-8 *A model opened in SolidWorks*

Select the required analysis type from the **Choose Analysis Type** dialog box. By default, the **Static Stress with Linear Material Models** analysis type is selected in this dialog box. After selecting the required analysis type, choose the **OK** button from the dialog box; the SolidWorks model will open in Autodesk Simulation Mechanical, as shown in Figure 3-9.

Note

Similar to Autodesk Inventor and SolidWorks, you can also transfer a model directly to Autodesk Simulation Mechanical from other software such as Pro/Engineer/Creo Parametric, Solid Edge, and Autodesk Mechanical Desktop.

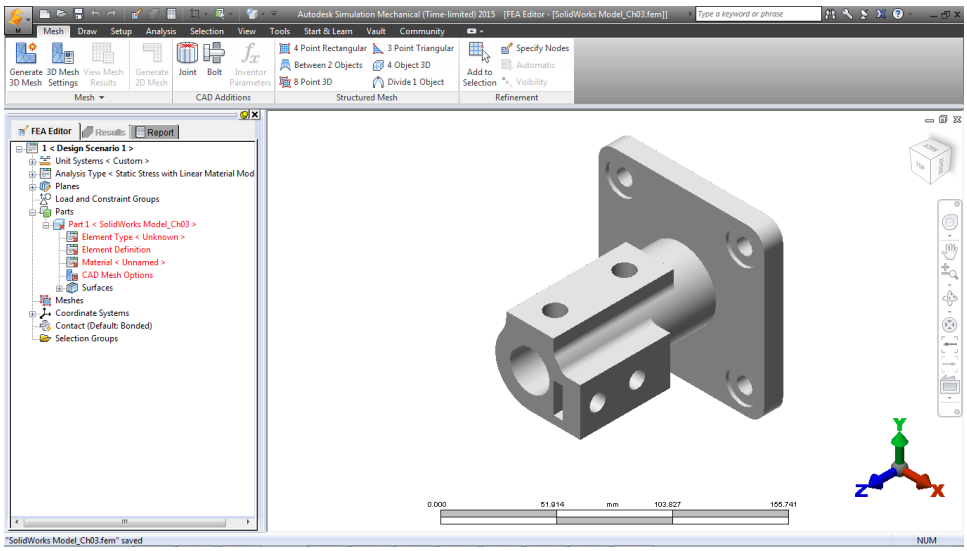


Figure 3-9 *SolidWorks model imported in Autodesk Simulation Mechanical*

SPLITTING SURFACES OF CAD MODELS

The splitting surfaces is a process of recognizing where two parts of a model have surfaces in contact and then splitting those surfaces, if necessary, to get identical surfaces. You can specify whether you want to run the process of splitting surfaces on the CAD model being imported by choosing the **Yes** or **No** button from the **Surface Splitting** dialog box, refer to Figure 3-10. The **Surface Splitting** dialog box is displayed automatically once you import a model having two or more than two connecting components.

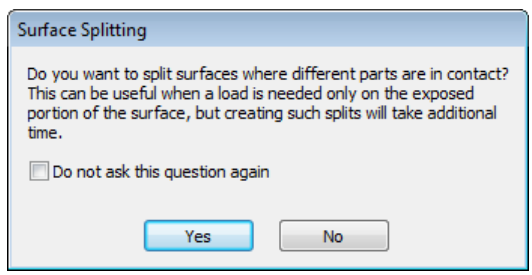


Figure 3-10 *The **Surface Splitting** dialog box*

Note that while importing a CAD model, the **Surface Splitting** dialog box may not be displayed. To display the **Surface Splitting** dialog box on importing a model, choose the **Application Options** button from the **Options** panel of the **Tools** tab in the **Ribbon**; the **Options** dialog box will be displayed, refer to Figure 3-11. Next, choose the **CAD Import** tab from the **Options** dialog box; the options related to importing a CAD geometry will be displayed, as shown in Figure 3-11.

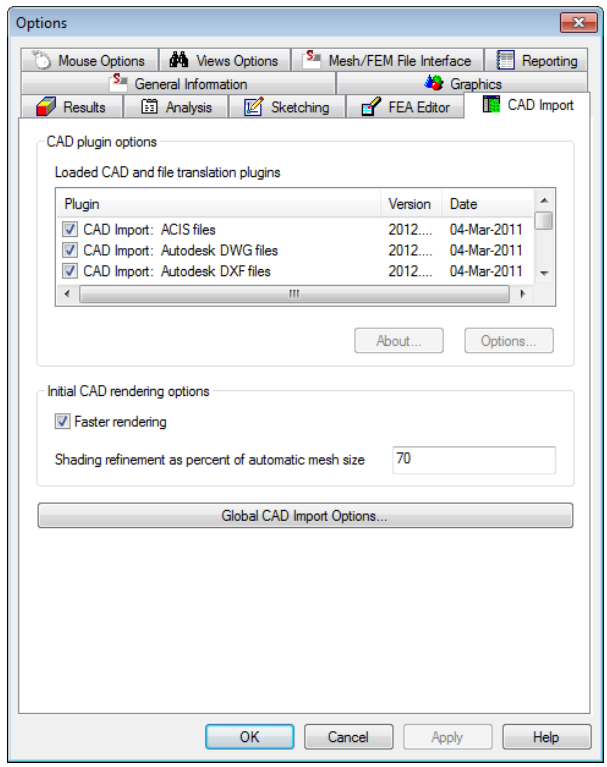


Figure 3-11 *The* ***Options*** *dialog box with the* ***CAD Import*** *tab chosen*

Next, choose the **Global CAD Import Options** button from this dialog box; the **Global CAD Import Options** dialog box will be displayed, refer to Figure 3-12. If you select the **Yes** radio button next to the **Split surface on import** option in this dialog box then on importing a CAD model into Autodesk Simulation Mechanical, the contact surfaces of the model being imported will split automatically. On the other hand, if the **No** radio button is selected for the **Split surface on import** option then the process of splitting surfaces on the model being imported will not be carried out. However, if you select the **Ask each time** radio button then the **Surface Splitting** dialog box will be displayed each time you import a CAD model in Autodesk Simulation Mechanical.

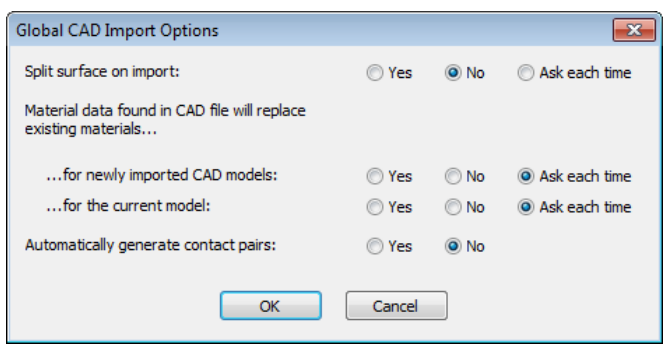


Figure 3-12 *The* ***Global CAD Import Options*** *dialog box*

SIMPLIFYING THE MODEL GEOMETRY BEFORE IMPORTING

While setting the model for analysis, most of the time you may need to first simplify the geometry to be analyzed by suppressing some of its features such as fillets, chamfers, holes, and so on. You can suppress those features that do not have major role in the analysis process or will not affect the analysis result. By suppressing some of the features, you can avoid the extra time that is involved in processes like meshing and computing results. You can simplify the geometry of the model in the software in which it is created before importing it in the Autodesk Simulation Mechanical. The process of simplifying the model geometry is discussed next.

Simplifying the Model Geometry in Autodesk Inventor

In Autodesk Inventor, you can simplify the geometry of the model before transferring it to Autodesk Simulation Mechanical. To do so, after opening the model in the Autodesk Inventor, choose the **Simplify Model** tool from the **Simulation Mechanical 2015** panel of the **Simulation** tab in the **Ribbon**; the **Model Simplification** dialog box will be displayed, as shown in Figure 3-13.

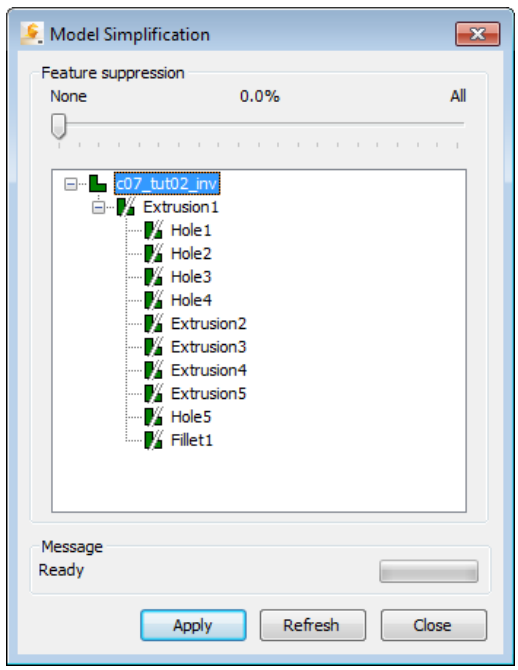


Figure 3-13 *The* ***Model Simplification*** *dialog box*

The **Model Simplification** dialog box displays a list of features of the active inventor model in the form of the tree view and is used to suppress features that are not necessary for the FEA model. To suppress a feature of a model to be imported in the Autodesk Simulation Mechanical for analysis, select it from the tree view of the **Model Simplification** dialog box. Next, drag the slider available at the top of the tree view in the dialog box toward the right upto 100% and then choose the **Apply** button from the dialog box; the selected feature will be suppressed. Figure 3-14 shows the **Model Simplification** dialog box with the fillet feature suppressed.

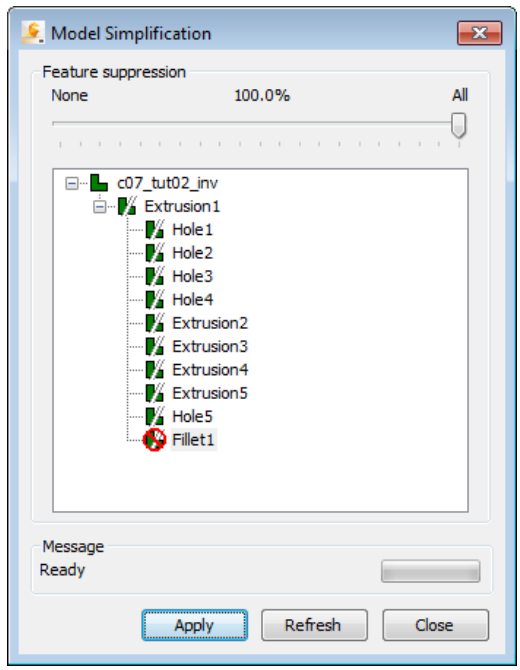


Figure 3-14 *The* ***Model Simplification*** *dialog box with fillet feature suppressed*

Simplifying the Model Geometry in SolidWorks

In SolidWorks, you can simplify the geometry of the model before transferring it to Autodesk Simulation Mechanical. To do so, open the model in SolidWorks and then choose **Autodesk Simulation > Simplify Model** from the SolidWorks menus; the **Model Simplification** dialog box will be displayed, refer to Figure 3-13.

To suppress a feature of a model to be imported in Autodesk Simulation Mechanical for analysis, select it from the tree view of the **Model Simplification** dialog box. Next, drag the slider available at the top of the tree view in the dialog box towards the right upto 100% and then choose the **Apply** button from the dialog box; the selected feature will be suppressed.

Note

Similar to the process of simplifying the geometry of a model in Autodesk Inventor and SolidWorks before transferring them to Autodesk Simulation Mechanical, you can simplify the geometry of models in the other CAD software as well.

IMPORTING FEA MODEL

Autodesk Simulation Mechanical also provides data translation interface to various leading FEA sofware for directly opening FEA models into Autodesk Simulation Mechanical. To open an FEA model, choose the **Open** tool from the **Quick Access Toolbar** of the Autodesk Simulation Mechanical; the **Open** dialog box will be displayed. By default, in the **Files of type** drop-down list of the **Open** dialog box, the **Autodesk Simulation FEA Model (*.fem)** file type is selected. As a result, the **Open** dialog box will list only those models that have **.fem* file extension. Select the required FEA file type from the **Files of type** drop-down list of the **Open** dialog box. For example, if you want to open the model created in ANSYS ANS Files of version 5.4 and ANSYS CDB Files of version 5 then select the **ANSYS (*.cdb, *.ans)** from the **Files of type** drop-down list of the **Open** dialog box. Next, browse to the location where the ANSYS models of version 5.4 and 5.5 are saved. Select the required ANSYS model in Autodesk Simulation Mechanical and then choose the **Open** button from the dialog box; the **Unit System** dialog box will be displayed. Select the required units for the model and then choose the **OK** button; the selected ANSYS model will be opened in Autodesk Simulation Mechanical. Similarly, you can open the ABAQUS INP files of version 5.3, FEMAP Neutral Files (NEU) of version 6.0, NASTRAN NAS, BDF and DAT files of version 2001, NASTRAN OP2 files of version 2001, PATRAN 2.5 Neutral Files (PAT), SDRC I-DEAS Universal Files (UNV), Stereolithography Files (STL), and Blue Ridge Numerics files (NEU).

Note

*Similar to the process of importing the CAD and FEA models into the Autodesk Simulation Mechanical by selecting the required file type from the **Files of type** drop-down list of the **Open** dialog box, you can also import the models that are saved in neutral file format such as **.sat, *.stp, *.iges, and *.step.*

SAVING FEA MODEL

When you open a model of any file type in Autodesk Simulation Mechanical, the system automatically saves the model in the **.fem* file extension with the same name as that of the original file at the same location. Now, after opening the model in Autodesk Simulation Mechanical, if you make any changes in it for carrying out the analysis process, you need to save those changes by choosing the **Save** button from the **Quick Access Toolbar**.

EXPORTING FEA MODEL

You can also export the FEA model of Autodesk Simulation Mechanical into the other FEA file formats such as ABAQUS (*.inp), ANSYS (*.cdb, *.ans), NASTRAN (*.nas, *.bdf, *.dat), and so on. To export Autodesk Simulation FEA model to other FEA file formats, choose the **Export > Third-Party FEA** from the **Application Menu**; the **Export** dialog box will be displayed, as shown in Figure 3-15.

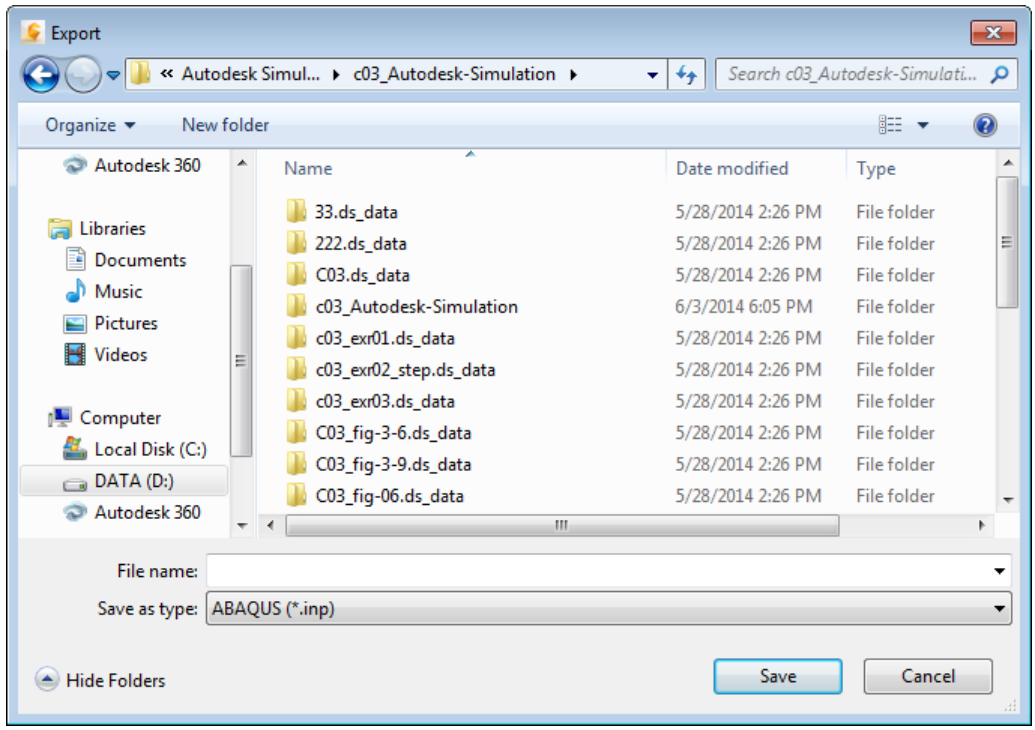


Figure 3-15 *The* ***Export*** *dialog box*

Now, browse to the location where you want to export the current FEA model. Next, specify a name for the FEA model in the **File name** edit box of the **Export** dialog box. After specifying the name, select the required FEA file type from the **Save as type** drop-down list of the dialog box and then choose the **Save** button; the FEA model will be exported to the selected file format. For example, if you want to export the FEA model of Autodesk Simulation Mechanical into Nastran file format then you need to select the **NASTRAN (*.nas, *.bdf, *.dat)** file type from the **Save as type** drop-down list of the **Export** dialog box.

ARCHIVING FEA MODEL

In Autodesk Simulation, you can save all files of an FEA model in an archive file. Archive files are compressed files which contain all the input files of the model as well as its results. As archive files are compressed files they use less disk space. Also, compressing files of an FEA model makes it easier to store them or to transfer them from one location to another. The *.*ach* is the file extension of the archive file. In Autodesk Simulation Mechanical, you can create, retrieve, repair, delete, and manage the existing archive files.

Creating an Archive File

To create an archive file, choose **Archive > Create** from the **Application Menu**; the **Create Archive** dialog box will be displayed, refer to Figure 3-16. Now, browse to the location where you want to save the archive file of the currently opened FEA model. You can also specify a new name for the archive file by entering a new name

in the **File name** edit box of the **Create Archive** dialog box. After specifying the location and name for the archive file, choose the **Save** button from the dialog box; the **Archive Creation Options** dialog box will be displayed, as shown in Figure 3-17. By default, the **Model only** radio button is selected in this dialog box. Note that, if you save the archive file with the **Model only** radio button selected in the **Archive Creation Options** dialog box then only the geometry of the FEA model will be saved in the archive file. However, if you select the **Model and results** radio button from the **Archive Creation Options** dialog box, all the input files of the model as well as its results will be saved in the archive file. After selecting the required radio button from the dialog box, choose the **OK** button; the archive file will be created in the specified location and the **Autodesk Simulation Mechanical** message window will be displayed on your screen informing you that the archive file has been created successfully. Choose the **OK** button from this message window.

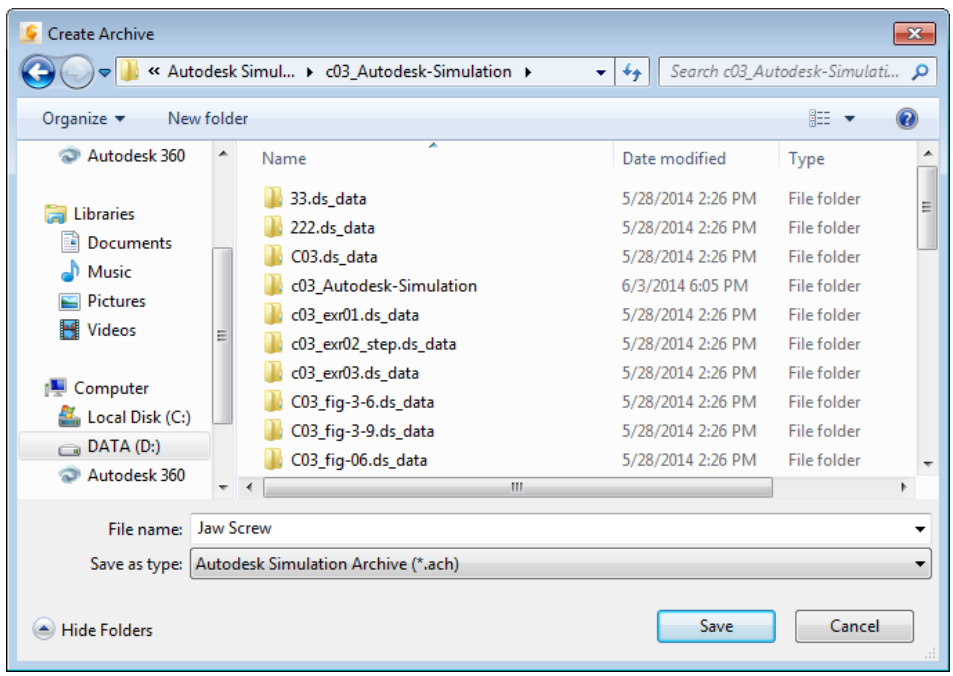


Figure 3-16 *The* ***Create Archive*** *dialog box*

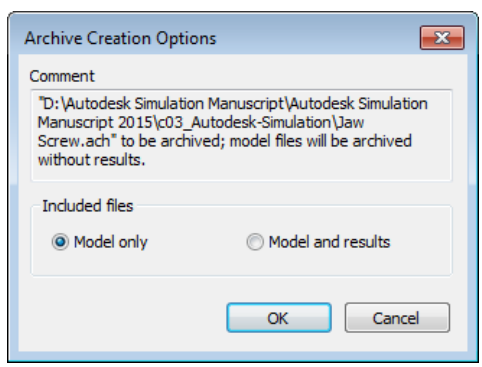


Figure 3-17 *The* ***Archive Creation Options*** *dialog box*

Retrieving an Archive File

You can retrieve an archive file and open it in Autodesk Simulation Mechanical. To retrieve an existing archive file, choose **Archive > Retrieve** from the **Application Menu**; the **Extract Archive** dialog box will be displayed. Browse to the required location and then select the archive file to be retrieved. After selecting the archive file, choose the **Open** dialog box; the **Browse for Folder** dialog box will be displayed. By using this dialog box, you can browse to the location where you want to extract all the data of the archived file. By default, in the **Browse for Folder** dialog box, the same folder in which the archive file is saved, is selected as the folder to extract the data. Choose the **OK** button to extract the data; the selected archive file is retrieved and the model is opened in Autodesk Simulation Mechanical.

Note

*If the folder selected for extracting the data of an archive file already has all the data of the same archive file then on choosing the **OK** button from the **Browse for Folder** dialog box, the **Confirm File Overwrite** message window will be displayed, refer to Figure 3-18. This message window informs you that the file already exists in this folder and do you want to replace it with the existing file.*

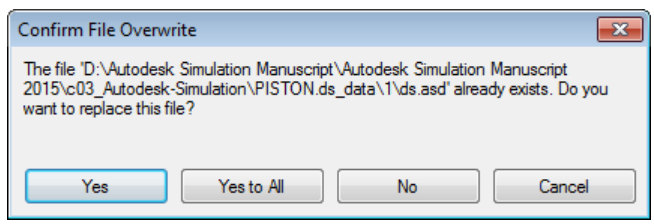


***Figure 3-18** The **Confirm File Overwrite** message window*

Repairing an Archive File

In Autodesk Simulation Mechanical, you can also repair a corrupt archive file. To do so, choose **Archive > Repair** from the **Application Menu**; the **Repair Archive** dialog box will be displayed. Browse to the required location and select the required archive file. Next, choose the **Open** button from the **Repair Archive** dialog box; the process for repairing the corrupted file will start. As soon as the repairing process is completed, an **Autodesk Simulation Mechanical** message window will be displayed with the information that the selected corrupted file has been repaired successfully. Next, choose the **OK** button from this message box to complete the process.

Managing an Existing Archive File

In Autodesk Simulation Mechanical, you can also manage an existing archive file by adding, deleting, restoring, or updating the data files of the selected archive file. To do so, choose **Archive > Manage Existing** from the **Application Menu**; the **Manage Archive** dialog box will be displayed. Browse to the location where the archive files are saved and then select the required archive file. Next, choose the **Open** button from the dialog box; the **Autodesk Simulation Archive Utility** window will be displayed which lists all the data files of the selected archive file, as shown in Figure 3-19. Now, you can add, delete, restore, or update the data files of the archive file by choosing **Edit > Add/Delete/Restore/**

Update from the Menu Bar of the **Autodesk Simulation Archive Utility** window. To exit from this window, choose **File > Exit** from the window.

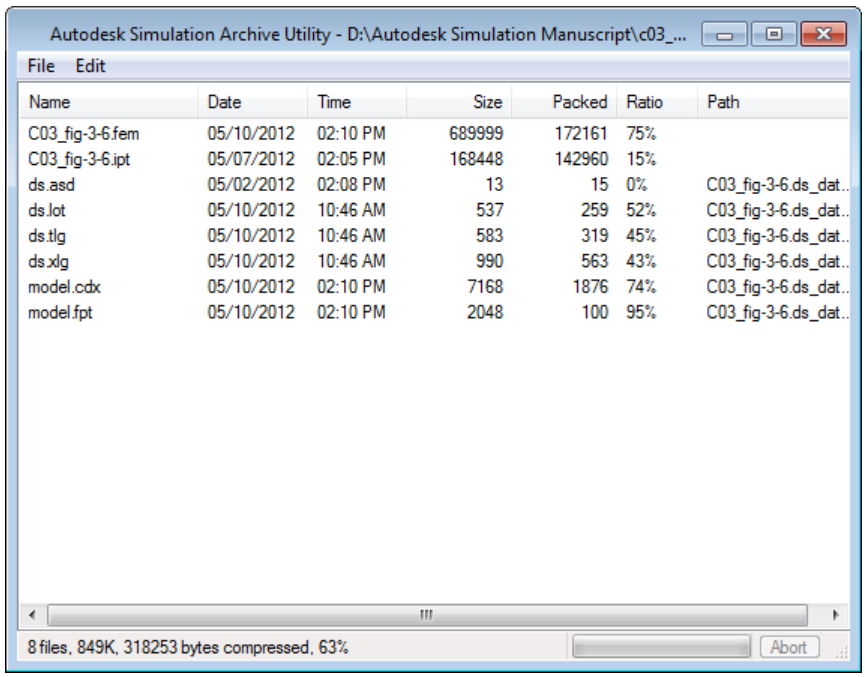


***Figure 3-19** The **Autodesk Simulation Archive Utility** window*

Deleting an Archive File

To delete an existing archive file, choose **Archive > Delete** from the **Application Menu**; the **Delete Archive** dialog box will be displayed. Browse to the required location, where the archive files are saved. Next, select the archive file to be deleted and then choose the **Open** button from the dialog box; the **Autodesk Simulation Mechanical** message window will be displayed prompting you to delete the selected archive file. Choose the **Yes** button from this message box; the **Autodesk Simulation Mechanical** message window will be displayed again informing you that the selected archive file has been deleted. Choose the **OK** button from this message box to complete the process.

UNDERSTANDING THE DRAWING DISPLAY TOOLS

The drawing display tools or navigation tools are integral part of any design or analysis software. These tools are extensively used during the design or analysis process. These tools are available in the **Navigation Bar** located on the right in the graphics window and also in the **Navigate** panel of the **View** tab in the **Ribbon**. Some of the drawing display tools in Autodesk Simulation Mechanical are discussed next and rest of these tools will be discussed in the later chapters.

Enclose (Fit All)/Zoom (Fit All)

Ribbon: View > Navigate > Enclose (Fit All)
Navigation Bar: Zoom flyout > Zoom (Fit All)

This tool is used to increase the drawing display area to display all the objects in the current display view. You can invoke this tool from the **Ribbon** by choosing the **Enclose (Fit All)** button from the **Navigate** panel of the **View** tab. You can also invoke this tool from the **Navigation Bar** by choosing **Zoom (Fit All)** from the **Zoom** flyout.

Zoom

Ribbon: View > Navigate > Zoom flyout > Zoom
Navigation Bar: Zoom flyout > Zoom

The **Zoom** tool is used to interactively zoom in and zoom out the drawing view. When you choose this tool, the default cursor is replaced by the zoom cursor. You can zoom out the drawing by pressing the left mouse button and dragging the cursor downward. Similarly, you can zoom in the drawing by pressing the left mouse button and then dragging the cursor in the upward direction. You can exit this tool by choosing another tool or by pressing ESC. You can also zoom in the drawing by rolling the scroll wheel of the mouse in the downward direction. Similarly, you can zoom out the drawing by rolling the scroll wheel in the upward direction.

Window/Zoom (Window)

Ribbon: View > Navigate > Zoom flyout > Window
Navigation Bar: Zoom flyout > Zoom (Window)

This tool is used to define an area to be magnified and viewed in the current drawing. The area is defined using two diagonal points of a box (called window) in the graphics window. The area inscribed in the window will be magnified and displayed on the screen. You can invoke this tool from the **Ribbon** by choosing **Window** from the **Zoom** flyout in the **Navigate** panel of the **View** tab. You can also invoke this tool from the **Navigation Bar** by choosing **Zoom (Window)** from the **Zoom** flyout.

Selected/Zoom (Selected)

Ribbon: View > Navigate > Zoom flyout > Selected
Navigation Bar: Zoom flyout > Zoom (Selected)

This tool is used to magnify the selected entity in the graphics area upto the maximum extent and place it at the center of the graphic window. To magnify the selected entity, select an entity to be magnified from the graphics area and then invoke this tool. You can invoke this tool from the **Ribbon** by choosing the **Selected** option from the **Zoom** flyout in the **Navigate** panel of the **View** tab. You can also invoke this tool from the **Navigation Bar** by choosing the **Zoom (Selected)** option from the **Zoom** flyout.

Pan

Ribbon:	View > Navigate > Pan
Navigation Bar:	Pan

The **Pan** tool is used to drag the current view in the graphics window. This tool is generally used to display the contents that lie outside the display area, without actually changing the magnification scale of the current drawing. It is similar to holding a geometry and dragging it across the graphics window. You can also invoke the **Pan** tool by pressing and holding the CTRL key and scrolling the middle mouse button.

Orbit

Ribbon:	View > Navigate > Orbit flyout > Orbit
Navigation Bar:	Orbit flyout > Orbit

The **Orbit** tool is used to rotate a model freely about any axis. It is useful when you want to rotate a model to any position. It is a transparent tool as it can be invoked inside any other command. You can invoke this tool by choosing the **Orbit** tool from the **Navigate** panel in the **View** tab or from the **Orbit** flyout in the **Navigation Bar**. On invoking this tool, the cursor will change to an orbit cursor. You can orbit the geometry into the graphics area by pressing the left mouse button and dragging the orbit cursor.

Constrained Orbit/Orbit (Constrained)

Ribbon:	View > Navigate > Orbit flyout > Constrained Orbit
Navigation Bar:	Orbit flyout > Orbit (Constrained)

This is one of the most important tools used for rotating the model around the vertical axis about a pivot point. When the **Constrained Orbit** tool is invoked, the cursor changes to the orbit cursor. You can press the left mouse button and drag the cursor to rotate the model towards left or right around the vertical axis about a previously defined pivot point. You can also define a new pivot point about which you want to rotate the model vertically. To define a pivot point, choose the **Center** tool from the **Navigation Bar** and click anywhere on the model in the graphic area. You can invoke this tool from the **Ribbon** by choosing the **Constrained Orbit** option from the **Orbit** flyout in the **Navigate** panel of the **View** tab. You can also invoke this tool from the **Navigation Bar** by choosing **Orbit (Constrained)** from the **Orbit** flyout.

Note

*While using the **Orbit (Constrained)** tool, the vertical axis about which the model will rotate will be normal to the Top face of the ViewCube.*

Previous View

Ribbon: View > Navigate > Previous View
Navigation Bar: Previous View

The **Previous View** tool in the **Navigate** panel of the **View** tab in the **Ribbon** is used to view the previous orientations of the model. You can also invoke this tool from the **Navigation Bar**.

Next View

Ribbon: View > Navigate > Next View
Navigation Bar: Next View

The **Next View** tool in the **Navigate** panel of the **View** tab in the **Ribbon** is used to activate the view that was current before you chose the **Previous View** tool. You can also invoke this tool from the **Navigation Bar**.

CHANGING THE VIEW USING THE VIEWCUBE

Autodesk Simulation Mechanical provides you with an option to change the view of a 3D model freely in 3D space using the ViewCube. A ViewCube is a 3D navigation tool which allows you to switch between the standard and isometric views in a single click. By default, it is in the inactive state, as shown in Figure 3-20. When you move the cursor closer to the ViewCube, it gets activated, refer to Figure 3-21.

Figure 3-20 *Inactive ViewCube*

Figure 3-21 *Active ViewCube*

Note

*You can control the visibility of the ViewCube by using the **Ribbon**. To do so, click on the down arrow displayed on the right of the **User Interface** tool in the **Appearance** panel of the **View** tab in the **Ribbon**; a drop-down list will be displayed, as shown in Figure 3-22. Select or clear the ViewCube check box from this drop-down list to display or hide the ViewCube, respectively in the graphics window.*

The faces, vertices, and edges of the ViewCube are called clickable areas. If you place the cursor on any of the clickable areas of the ViewCube, the corresponding area will be highlighted. Click on the required area to orient the model such that the clicked area and the model become parallel to the screen. If you press and drag the left mouse button over the ViewCube, it will provide a visual feedback of the current viewpoint of the model. When you right-click on the ViewCube, a shortcut menu will be displayed, as shown in Figure 3-23. The options in this shortcut menu are discussed next.

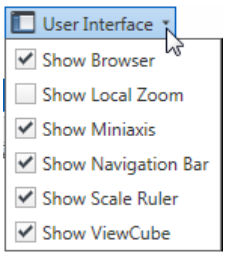


Figure 3-22 *The* ***User Interface*** *drop-down list*

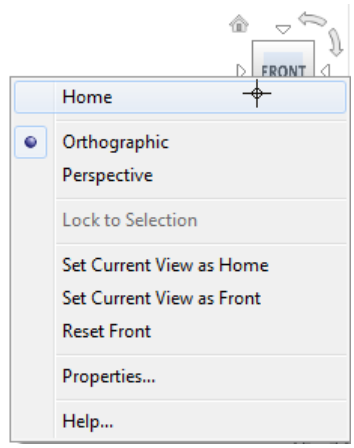


Figure 3-23 *The shortcut menu displayed after right-clicking on the ViewCube*

Home

It is used to display the default view of the model. On choosing this option, you can switch over to the home or isometric view of the model.

Orthographic

Choose this option to display the model in the orthographic view.

Perspective

Choose this option to display the model in the perspective view.

Lock to Selection

If you choose this option in a particular view, then that view gets locked and further manipulation of the view of the object will occur with respect to the locked position.

Set Current View as Home

This option is used to set the current view as the default or home view.

Set Current View as Front

This option is used to set the current view of the model as front view. When you select this option, the current view of the model will become the front view.

Reset Front

This option is used to reset the front view to the default setting.

Properties

Choose this option to invoke the **ViewCube Properties** dialog box, refer to Figure 3-24. The options in this dialog box are discussed next.

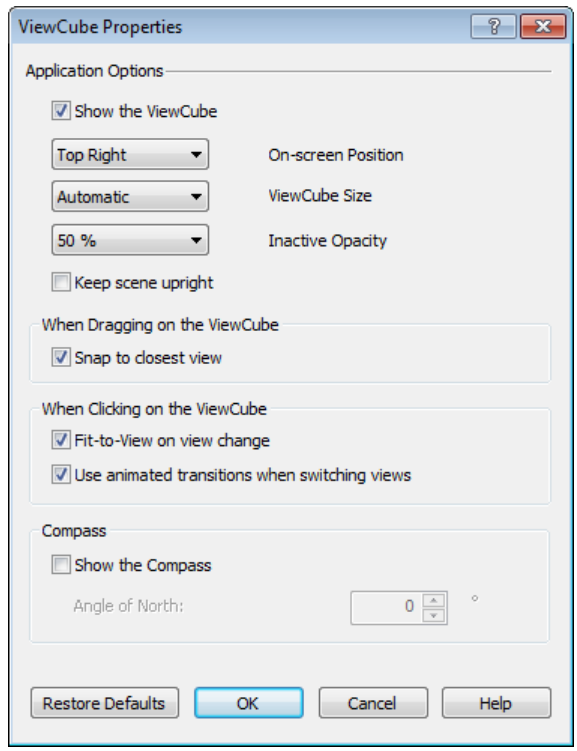


Figure 3-24 *The* ***ViewCube Properties*** *dialog box*

Show the ViewCube

By default, this check box is selected. As a result, the ViewCube will be displayed in the graphics window.

On-Screen Position Drop-down List

This drop-down list is used to display the on-screen position of the ViewCube. You can select the required option such as **Top Right**, **Bottom Right**, **Top Left**, or **Bottom Left** from the **On-Screen Position** drop-down list to set the position of ViewCube at any corner of the window. By default, the **Top Right** option is selected in this drop-down list. As a result, the ViewCube is displayed at the top right corner of the graphic window by default.

ViewCube Size Drop-down List

This drop-down list is used to set the size of the ViewCube. You can select the required option such as **Automatic**, **Tiny**, **Small**, **Medium**, or **Large** from the **ViewCube Size** drop-down list to set the size of the ViewCube. By default, the Automatic option is selected in this drop-down list. As a result, the size of the ViewCube is automatically adjusted according to the display area of the window.

Inactive Opacity Drop-down List

This drop-down list is used to set the opacity of the ViewCube in the inactive state. You must have noticed that when you move the cursor over the ViewCube in the graphics window, all additional controls of the ViewCube are displayed with full opacity. However, when the cursor is at some distance from the ViewCube, the ViewCube will be displayed with low opacity. This is because by default the **50%** option is selected in the **Inactive Opacity** drop-down list. As a result the opacity of the ViewCube in the inactive stage is set to fifty percent by default.

Keep scene upright

If this check box is selected then on clicking the edges, corners, or faces of the ViewCube, the ViewCube attempts to turn such that the upside-down orientations of the scene are avoided.

Snap to closest view

If the **Snap to closest view** check box is selected in the **When Dragging on the ViewCube** area of the dialog box, the view point will snap to one of the fixed views when the model is rotated using the ViewCube.

When Clicking on the ViewCube Area

The check boxes in this area are used for setting the preferences while clicking on the ViewCube.

Fit-to-View on view change

By default the **Fit-to-View on view change** check box is selected. As a result, on clicking on the edges, corners, or faces of the ViewCube, the orientation of the model will be changed accordingly. Also, it zooms out or zooms in the current display to fit the model into the graphics window. If this check box is clear, the model will not fit in the current graphics window when the orientation of the model is changed using ViewCube.

Use animated transitions when switching views

By default the **Use animated transitions when switching views** check box is selected. As a result, on changing the view of the model by using the ViewCube, an animated transition is displayed to help you visualize the spatial relationship between views.

Compass Area

This area is used to set the preference for the default display of the compass. If you select the **Show the Compass** check box, the compass will be displayed around the ViewCube in the graphics window, as shown in Figure 3-25. The **Angle of North** spinner of this area is used to set the angle between the **FRONT** face of the ViewCube and the **N** (North direction) of the compass.

Figure 3-25 *The ViewCube with Compass*

NAVIGATING THE MODEL USING STEERINGWHEELS

Ribbon: View > Navigate > Steering Wheels flyout > Full Navigation Wheel
Navigation Bar: Steering Wheels flyout > Full Navigation Wheel

SteeringWheels is one of the most useful and convenient way to navigate the model. To display SteeringWheels, choose the **Full Navigation Wheel** tool from the **Navigation Bar**; the SteeringWheels will be displayed attached to the cursor, as shown in Figure 3-26. The SteeringWheels are the tracking tools that are divided into different wedges. Each wedge represents a single navigation tool such as **PAN**, **ORBIT**, **ZOOM**, **REWIND**, **LOOK**, **CENTER**, **WALK**, and **UP/DOWN**. You can activate any wedge of the SteeringWheels by pressing and holding the cursor over it. The SteeringWheels travels along with the cursor to provide a quick access to common navigation controls. Right-click on the SteeringWheels; a shortcut menu will be displayed. Different types of SteeringWheels are available in this shortcut menu. You can choose any type of SteeringWheels by clicking on it. You can also invoke different types of SteeringWheels by selecting the corresponding option from the **SteeringWheels** flyout in the **Navigation Bar**, refer to Figure 3-27. To display the **SteeringWheels** flyout, click on the down arrow available below the active SteeringWheels tool in the **Navigation Bar**, refer to Figure 3-27. You can also invoke the SteeringWheels from the **Navigate** panel of the **View** tab in the **Ribbon**.

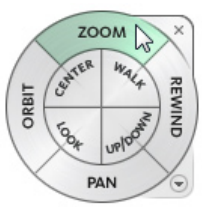


Figure 3-26 *The* ***Full Navigation Wheel*** *SteeringWheels*

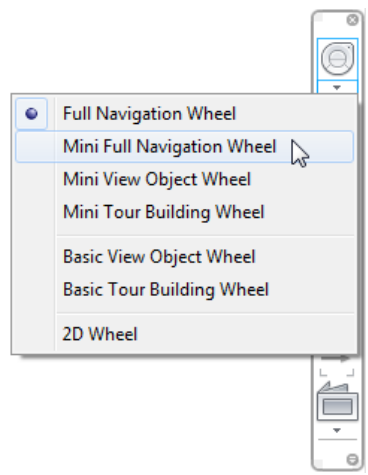


Figure 3-27 *The* ***SteeringWheels*** *flyout*

CONTROLLING THE DISPLAY OF MODELS

Autodesk Simulation Mechanical allows you to control the display of the models by setting various display modes and setting the camera type. The options for controlling the display of the models are discussed next.

Setting the Visual Styles

Ribbon: View > Appearance > Visual Style flyout

Visual style of a model determines the display of edges and face of a model in the graphics window. You can set the visual style for the solid models by using the **Visual Style** flyout provided in the **Appearance** panel of the **View** tab, refer to Figure 3-28. Various visual styles available in this flyout are discussed next.

Shaded

This style is used to shade the faces of a model with standard materials and colors. To apply this visual style, choose the **Shaded** tool from the **Visual Style** flyout of the **Appearance** panel in the **View** tab of the **Ribbon**. In this style, the visibility of the visible and hidden edges is turned off.

Edges

This style is used to display only the model edges with the shading turned off. To apply this visual style, choose the **Edges** tool from the **Visual Style** flyout of the **Appearance** panel in the **View** tab of the **Ribbon**. In this style, the visibility of both the visible and hidden edges is turned on and these edges are displayed as solid lines.

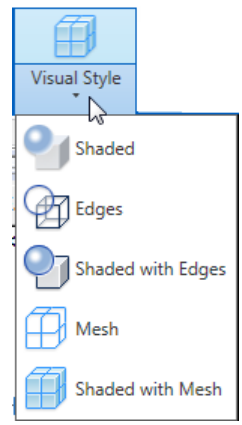


***Figure 3-28** Tools in the **Visual Style** flyout*

Shaded with Edges

In this style, the model appears shaded with all external edges clearly visible. To apply this visual style, choose the **Shaded with Edges** tool from the **Visual Style** flyout of the **Appearance** panel in the **View** tab of the **Ribbon**. In this style, the standard materials and colors are assigned to the model.

Mesh

This style is used to display the model with only the meshing lines visible. To apply this visual style, choose the **Mesh** tool from the **Visual Style** flyout of the **Appearance** panel in the **View** tab of the **Ribbon**.

Shaded with Mesh

In this style, the model appears shaded with meshing line visible. To apply this visual style, choose the **Shaded with Mesh** tool from the **Visual Style** flyout of the **Appearance** panel in the **View** tab of the **Ribbon**.

Setting the Camera Type

By default, the models are displayed in the orthographic camera type. You can change the camera type from the default orthographic to the perspective camera. To change the camera view to perspective, click on the down arrow next to the **Appearance** panel bar of the **View** tab; the **Appearance** panel will expand showing some other tools. Next, choose the **Perspective** tool from the expanded **Appearance** panel; the model will be displayed in the perspective camera.

TUTORIALS

Tutorial 1

In this tutorial, you will import the Inventor assembly model shown in Figure 3-29 to Autodesk Simulation Mechanical. Select the default analysis type that is **Static Stress with Linear Material Models** to carry out the analysis process. After opening the assembly, you will change its visual style to the **Shaded with Edges** style. You will also use the SteeringWheels to navigate the model. (**Expected time: 20 min**)

Note

Download the input files of tutorials used in this chapter from www.cadcim.com. The complete path for downloading the file is given below:

Textbooks > CAE Simulation > Autodesk Simulation Mechanical > Autodesk Simulation Mechanical 2015 for Designers > Input Files > c03_simulation_2015_input.zip .

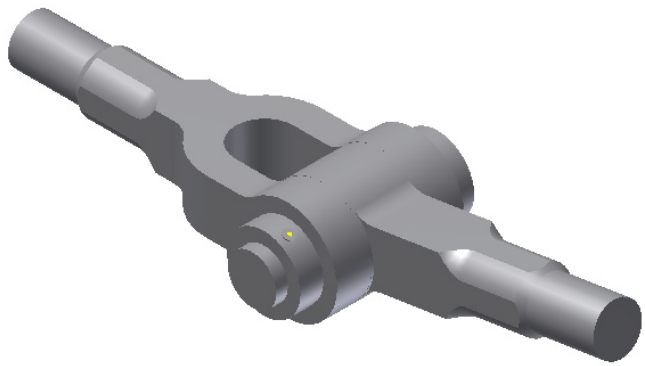


Figure 3-29 *The Knuckle joint*

The following steps are required to complete this tutorial:

a. Download the zipped file containing input files of Chapter 3.
b. Start Autodesk Simulation Mechanical.
c. Set the **Split surface on import** setting to **Ask each time**.
d. Open the Inventor model.
e. Change the visual style of the model to Shaded with Edges style.
f. Navigate the model using the SheeringWheels.
g. Save and close the model.

Downloading the File

1. Create a new folder with the name *Autodesk Simulation Mechanical* on the *C:|* drive of your computer. Next, create another folder with the name **c03** at the location *C:\Autodesk Simulation Mechanical*.

2. Download the zipped file named as *c03_simulation_2015_input* from the **Input Files** section of *www.cadcim.com*. The complete path for downloading the file is:

   Textbooks > CAE Simulation > Autodesk Simulation Mechanical > Autodesk Simulation Mechanical 2015 for Designers > Input Files

3. Extract the downloaded *c03_simulation_2015_input* zipped file. Next, copy the **Tut01** folder from the extracted folder and paste it at the location *C:\Autodesk Simulation Mechanical\c03*.

Importing the Inventor Assembly File

1. Double-click on the shortcut icon of **Autodesk Simulation Mechanical 2015** on the desktop of your computer; the initial screen of Autodesk Simulation is displayed with either the **Open** or **New** dialog box.

Note

*As discussed earlier, when you start Autodesk Simulation Mechanical, by default the initial screen of Autodesk Simulation Mechanical is displayed with either the **Open** or **New** dialog box depending upon the last used dialog box in the last session of Autodesk Simulation Mechanical.*

*You can turn off the automatic display of dialog box at the start up of Autodesk Simulation Mechanical. To do so, first close the opened dialog box, if any, and then 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. Next, choose the **Apply** button and then the **Close** button from the dialog box.*

In this tutorial, before opening the model in the Autodesk Simulation, you need to set the splitting surfaces setting to **Ask each time** so that whenever you import a model, the system asks you whether you want to split the surfaces of the model.

2. Close the **Open** or **New** dialog box, if displayed, by choosing the **Cancel** button from it.

3. Choose the **Tools** tab of the **Ribbon** and then choose the **Application Options** button from the **Options** panel of the **Tools** tab; the **Options** dialog box is displayed.

4. Choose the **CAD Import** tab from the **Options** dialog box; the options related to importing the CAD models are displayed in the dialog box.

5. Choose the **Global CAD Import Options** button from the dialog box; the **Global CAD Import Options** dialog is displayed, refer to Figure 3-30.

6. Select the **Ask each time** radio button for the **Split surface on import** option and then choose the **OK** button from the dialog box. Next, exit the **Options** dialog box by choosing **OK**.

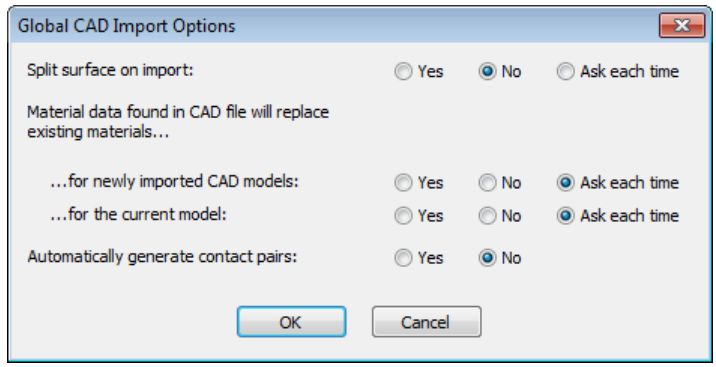


Figure 3-30 *The* ***Global CAD Import Options*** *dialog box*

Now, you need to open the inventor model of the Knuckle joint assembly in Autodesk Simulation Mechanical.

7. Choose the **Open** button from the **Launch** panel of the **Start & Learn** tab in the **Ribbon**; the **Open** dialog box is displayed.

As mentioned in the tutorial description that the assembly model to be used for this tutorial is created in Autodesk Inventor. Therefore, you need to select the file extension of the assembly file created in Autodesk Inventor from the **Files of type** drop-down list of the **Open** dialog box to open the Inventor assembly in to the Autodesk Simulation Mechanical.

8. Select the **Autodesk Inventor Files(*.ipt; *.iam)** file type from the **Files of type** drop-down list of the **Open** dialog box.

9. Browse to the location *C:\Autodesk Simulation Mechanical\c03\Tut01*; the inventor files of this tutorial are displayed in the **Open** dialog box.

10. Select the **Knuckle Joint.iam** file from the **Open** dialog box and then choose the **Open** button; the **Import Inventor Work Points** window is displayed.

11. Choose the **Yes** button from this window; the **Surface Splitting** dialog box is displayed, as shown in Figure 3-31.

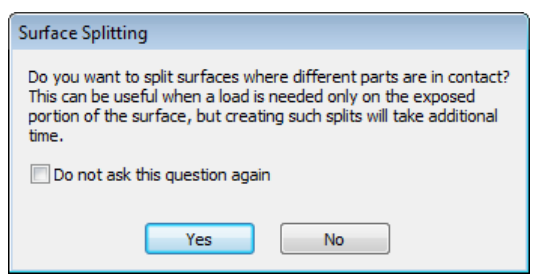


Figure 3-31 *The* ***Surface Splitting*** *dialog box*

12. Choose the **Yes** button from the **Surface Splitting** dialog box; the **Choose Analysis Type** dialog box is displayed with the **Stress with Linear Material Models** analysis type selected by default, as shown in Figure 3-32.

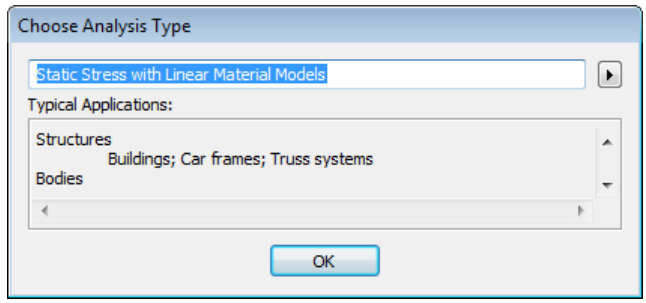


Figure 3-32 *The* ***Choose Analysis Type*** *dialog box*

13. Accept the default selection and choose the **OK** button from the dialog box; the Inventor assembly file is opened into Autodesk Simulation Mechanical, as shown in Figure 3-33. As soon as the model is opened in Autodesk Simulation Mechanical, a copy of the model is automatically saved as an FEA model with the file extension *.fem* in the same folder where the Inventor model is saved.

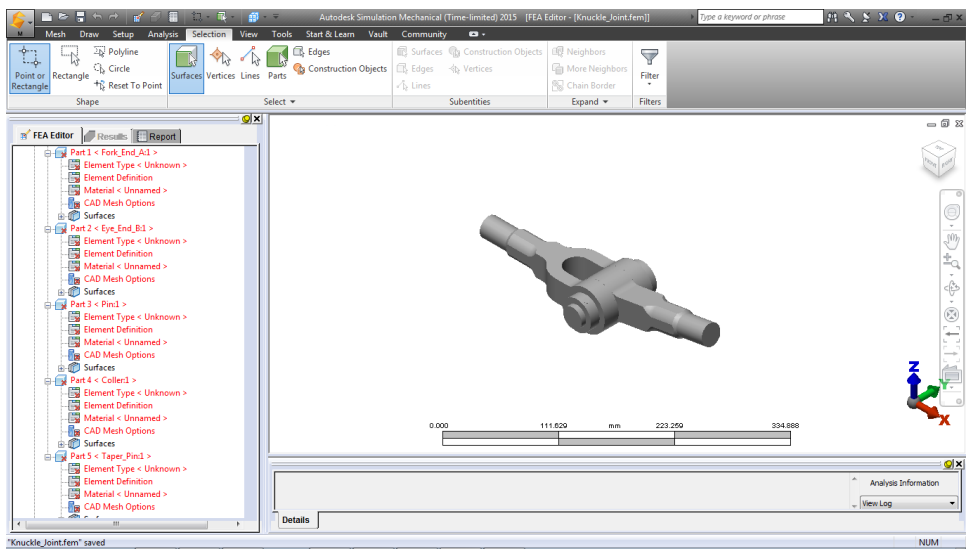


Figure 3-33 *Inventor assembly imported into Autodesk Simulation Mechanical*

Changing the Visual Style

In this section, you need to change the default visual style of the assembly to the Shaded with Edges visual style.

1. Choose the **View** tab of the **Ribbon**; all tools that are grouped together into the **View** tab of the **Ribbon** are displayed.

2. Click on the down arrow available at the bottom of the **Visual Style** button in the **Appearance** panel; the **Visual Style** flyout is displayed, as shown in Figure 3-34.

3. Choose the **Shaded with Edges** tool from this flyout; the current visual style of the assembly is changed to Shaded with Edges style, as shown in Figure 3-35.

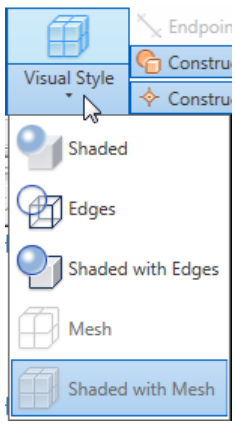


Figure 3-34 *The* ***Visual Style*** *flyout*

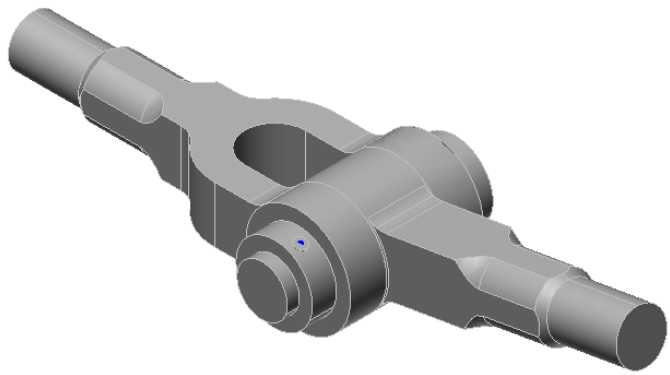


Figure 3-35 *The assembly displayed in Shaded with Edges visual style*

Navigating the Assembly by Using the SteeringWheel

Now, you need to navigate the assembly by using the SteeringWheels.

1. Choose the **Full Navigation Wheel** tool from the **Navigation Bar**; the SteeringWheels attached with the cursor is displayed, as shown in Figure 3-36.

2. Move the SteeringWheels over the assembly by moving the cursor.

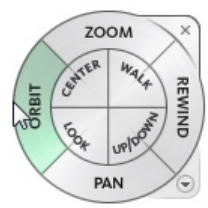


Figure 3-36 *The SteeringWheels*

3. Move the cursor over the **ORBIT** wedge of the SteeringWheels; the **ORBIT** wedge is highlighted. Next, press and hold the left mouse button; the **Orbit** tool is activated. Now, drag the cursor; the assembly rotates about the **PIVOT** point, refer to Figure 3-37.

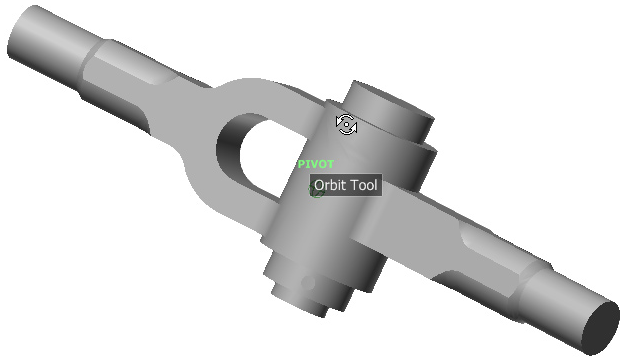


Figure 3-37 *The **Orbit** tool activated by using the SteeringWheels*

4. Move the cursor over the **PAN** wedge of the SteeringWheels; the **PAN** wedge is highlighted. Next, press and hold the left mouse button; the **Pan** tool is activated. Now drag the cursor; the assembly moves along the cursor.

5. Move the cursor over the **REWIND** wedge of the SteeringWheels and press and hold the left mouse button; the previous views are displayed in small boxes in a row. Drag the cursor along the row and select a view.

6. Move the SteeringWheels over the assembly by moving the cursor. Next, move the cursor over the **CENTER** wedge; a pivot is attached to the cursor. Next, release the left mouse button over the assembly at the point where you want to create a pivot point. Note that as soon as you release the left mouse button, a pivot point is created in the assembly and moved to the center of the graphics area.

7. Move the cursor over the **ZOOM** wedge of the SteeringWheels and press and hold the left mouse button; the **Zoom** tool is activated. Now, drag the cursor in the upward/downward direction to zoom in/out the graphics area.

8. Click on the cross mark at the top right corner of the SteeringWheels to exit this tool. Alternatively, right-click and choose the **Close Wheel** option from the shortcut menu displayed to exit the **SteeringWheels** tool.

9. Move the cursor over the ViewCube to activate it and then choose the **Home** button; the current view of the assembly is changed to the isometric view.

Saving the Model

Now, you need to save changes made in the FEA model.

1. Choose the **Save** button from the **Quick Access Toolbar**; the changes made in the assembly are saved.

2. Choose **Close** from the **Application Menu** to close the assembly.

Tutorial 2

In this tutorial, you will import a SolidWorks assembly file shown in Figure 3-38 into Autodesk Simulation Mechanical. Select the default analysis type which is **Static Stress with Linear Material Models**. After opening the assembly, you need to change the views of the assembly by using the ViewCube. (**Expected time: 15 min**)

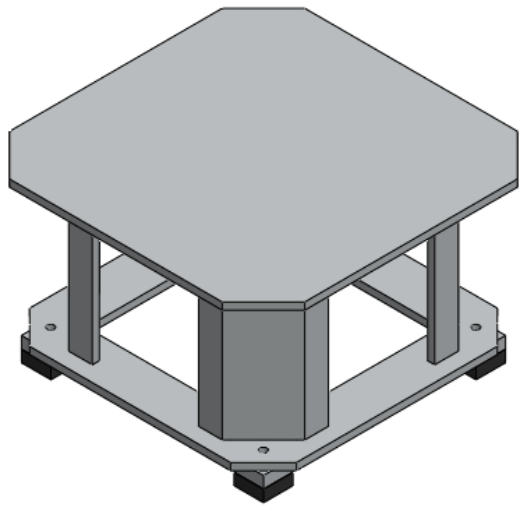


Figure 3-38 *SolidWorks assembly*

Note

You can download the input files of the tutorials used in this chapter from www.cadcim.com, if not already downloaded in Tutorial 1. The complete path for downloading the file is given below:

Textbooks > CAE Simulation > Autodesk Simulation Mechanical > Autodesk Simulation Mechanical 2015 for Designers > Input Files > c03_simulation_2015_input.zip .

The following steps are required to complete this tutorial:

a. Download the zipped file containing files of Chapter 3.
b. Start Autodesk Simulation Mechanical.
c. Set the **Split surfaces on import** setting **Yes** or **No** to **Ask each time**.
d. Open the SolidWorks model.
e. Navigate the model using the ViewCube.
f. Save and close the model.

Downloading the File

1. Create a new folder with the name *Autodesk Simulation Mechanical* on the *C:/* drive of your computer, if not already created. Next, create another folder with the name **c03** at the location *C:\Autodesk Simulation Mechanical*, if not already created in Tutorial 1 of this chapter.

2. Download the zipped file named as *c03_simulation_2015_input* from the **Input Files** section of *www.cadcim.com*, if not downloaded already in Tutorial 1 of this chapter. The complete path for downloading the file is:

   Textbooks > CAE Simulation > Autodesk Simulation Mechanical > Autodesk Simulation Mechanical 2015 for Designers > Input Files.

3. Extract the downloaded **c03_simulation_2015_input** zipped file. Next, copy the **Tut02** folder from the extracted folder and paste it at the location *C:/Autodesk Simulation Mechanical/c03*.

Importing the SolidWorks Assembly File

1. Start Autodesk Simulation Mechanical by double-clicking on the shortcut icon of **Autodesk Simulation Mechanical 2015** on the desktop of your computer.

   Now, you need to open the SolidWorks assembly into the Autodesk Simulation Mechanical.

2. Choose the **Open** button from the **Launch** panel of the **Start & Learn** tab in the **Ribbon**; the **Open** dialog box is displayed.

Note

*In this textbook, the automatic display of the **Open** or **New** dialog box on the start up of Autodesk Simulation Mechanical has been turned off.*

As mentioned in the tutorial description that the assembly model being used for this tutorial is created in SolidWorks. Therefore, you need to select the file extension of the assembly file created in SolidWorks from the **Files of type** drop-down list of the **Open** dialog box to open the SolidWorks assembly in to the Autodesk Simulation Mechanical.

3. Select the **SolidWorks Files (*.prt; *.sldprt; *.asm; *.sldasm)** file type from the **Files of type** drop-down list of the **Open** dialog box.

4. Browse to the location *C:\Autodesk Simulation Mechanical\c03\Tut02*; the SolidWorks assembly file of this tutorial is displayed in the **Open** dialog box.

5. Select the **Table.SLDASM** file and then choose the **Open** button from the **Open** dialog box; the **Surface Splitting** dialog box is displayed. Choose the **No** button from the **Surface Splitting** dialog box; the **Choose Analysis Type** dialog box is displayed with the **Stress with Linear Material Models** analysis type selected by default in it, as shown in Figure 3-39.

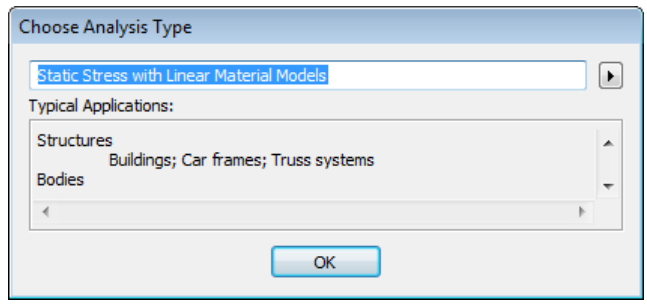


Figure 3-39 *The **Choose Analysis Type** dialog box*

Note

*The **Surface Splitting** dialog box will be displayed only if the **Ask each time** radio button in the **Split surface on import** area of the **Global CAD Import Options** dialog box is selected. If the **Yes** or **No** radio button for **Split surface on import** is selected in this dialog box then after choosing the **Open** button from the **Open** dialog box, the **Choose Analysis Type** dialog box will be displayed directly.*

6. Accept the default selection and choose the **OK** button from the dialog box; the SolidWorks assembly file is opened into the Autodesk Simulation Mechanical, as shown in Figure 3-40.

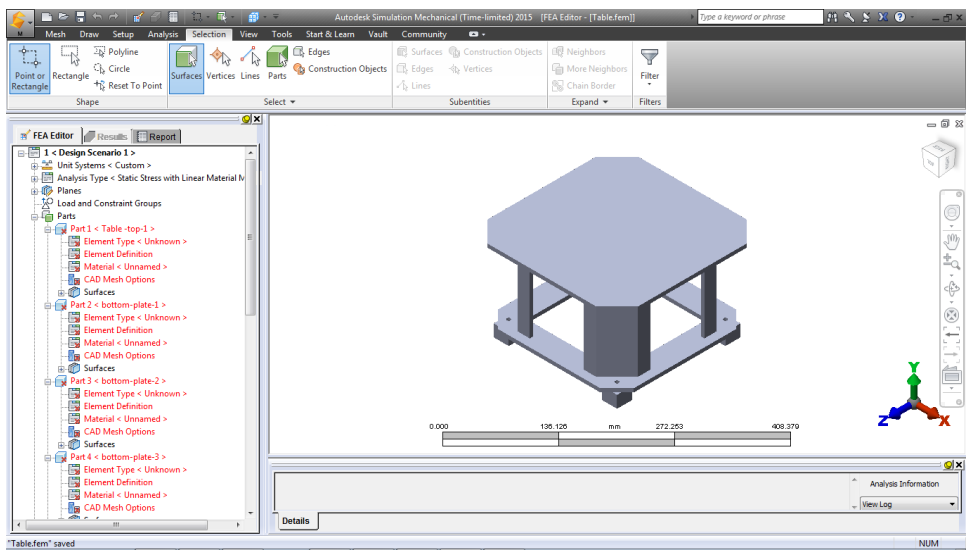


Figure 3-40 *SolidWorks assembly imported into Autodesk Simulation Mechanical*

Changing Views of the Assembly by Using the ViewCube

Now, you need to change the views of the assembly by using the ViewCube.

1. In case, the ViewCube is hidden or not visible in the graphics area, you need to first turn on its visibility. To do so, click on the down arrow displayed on the right of the **User Interface** tool of the **Visibility** panel in the **View** tab of the **Ribbon**; the **User Interface** flyout is displayed, refer to Figure 3-41.

2. Select the **Show ViewCube** check box from this flyout; the ViewCube is displayed at the upper right corner of the graphics area.

Note

If the ViewCube is already displayed at the upper right corner of the graphics area then you can skip step 1 and 2.

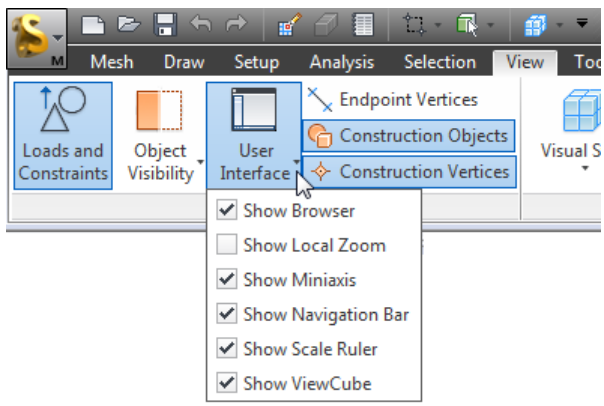


Figure 3-41 *The* ***User Interface*** *flyout*

3. Move the cursor over the **FRONT** clickable area of the ViewCube; the **FRONT** clickable area is highlighted, refer to Figure 3-42.

4. Click on the **FRONT** clickable area to display the front view of the model; the model and clicked area become parallel to the screen. Figure 3-43 shows the Front view of the model displayed after clicking on the **FRONT** clickable area of the ViewCube.

5. Choose the **Home** button of the ViewCube; the current view of the model is changed to the isometric view.

Figure 3-42 *The ViewCube with* ***FRONT*** *clickable area highlighted*

Figure 3-43 *The Front view of the model*

6. Move the cursor over the top right corner of the ViewCube, refer to Figure 3-44 and when it is highlighted, press and hold the left mouse button and drag the cursor. As the cursor moves, the model re-orients to give you a better view of the model.

Figure 3-44 *The ViewCube*

7. Choose the **Home** button of the ViewCube again to display the model to the isometric view.

Saving the Model

Now, you need to save the changes made in the FEA model.

1. Choose the **Save** button from the **Quick Access Toolbar**; the changes made in the assembly are saved.

2. Choose **Close** from the **Application Menu** to close the assembly.

Tutorial 3

In this tutorial, you will import a STEP file of the model shown in Figure 3-45. Select the default analysis type which is **Static Stress with Linear Material Models**. After opening the assembly, you need to change the views of the assembly by using the ViewCube.

(**Expected time: 15 min**)

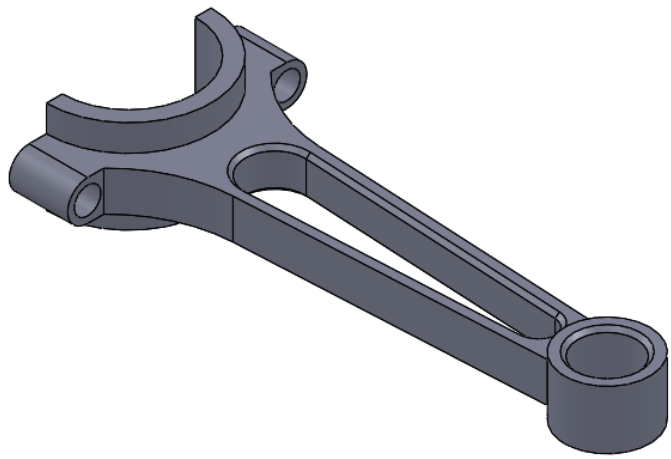


Figure 3-45 *STEP file model*

Note

You can download the input files of the tutorials used in this chapter from www.cadcim.com, if not already downloaded in Tutorial 1. The complete path for downloading the file is given below:

Textbooks > CAE Simulation > Autodesk Simulation Mechanical > Autodesk Simulation Mechanical 2015 for Designers > Input Files > c03_simulation_2015_input.zip .

The following steps are required to complete this tutorial:

a. Download the zipped file containing files of Chapter 3.
b. Start Autodesk Simulation Mechanical.
c. Set the **Split surfaces on import** setting **Yes** or **No** to **Ask each time**.
d. Open the SolidWorks model.
e. Navigate the model using the ViewCube.
f. Save and close the model.

Downloading the File

1. Create a new folder with the name *Autodesk Simulation Mechanical* on the *C:/* drive of your computer, if not already created. Next, create another folder with the name **c03** at the location *C:\Autodesk Simulation Mechanical*, if not already created in Tutorial 1 or 2 of this chapter.

2. Download the zipped file named as *c03_simulation_2015_input* from the **Input Files** section of *www.cadcim.com*, if not downloaded already in Tutorial 1 or 2 of this chapter. The complete path for downloading the file is:

   Textbooks > CAE Simulation > Autodesk Simulation Mechanical > Autodesk Simulation Mechanical 2015 for Designers > Input Files.

3. Extract the downloaded **c03_simulation_2015_input** zipped file. Next, copy the **Tut03** folder from the extracted folder and paste it at the location *C:/Autodesk Simulation Mechanical/c03*.

Importing the SolidWorks Assembly File

1. Start Autodesk Simulation Mechanical by double-clicking on the shortcut icon of **Autodesk Simulation Mechanical 2015** on the desktop of your computer.

   Now, you need to open the SolidWorks assembly into the Autodesk Simulation Mechanical.

2. Choose the **Open** button from the **Launch** panel of the **Start & Learn** tab in the **Ribbon**; the **Open** dialog box is displayed.

Note

*In this textbook, the automatic display of the **Open** or **New** dialog box on the start up of Autodesk Simulation Mechanical has been turned off.*

   As mentioned in the tutorial description that the model being used for this tutorial is a STEP file. Therefore, you need to select the file extension of the STEP file format from the **Files of type** drop-down list of the **Open** dialog box.

3. Select the **STEP Files (*.stp; *.ste; *.step)** file type from the **Files of type** drop-down list of the **Open** dialog box.

4. Browse to the location *C:\Autodesk Simulation Mechanical\c03\Tut03*; the STEP file of this tutorial is displayed in the **Open** dialog box.

5. Select the **C03_Tut03** file and then choose the **Open** button from the **Open** dialog box; the **Choose Analysis Type** dialog box is displayed with the **Stress with Linear Material Models** analysis type selected by default in it, as shown in Figure 3-46.

6. Accept the default selection and choose the **OK** button from the dialog box; the STEP file is opened into the Autodesk Simulation Mechanical, as shown in Figure 3-47.

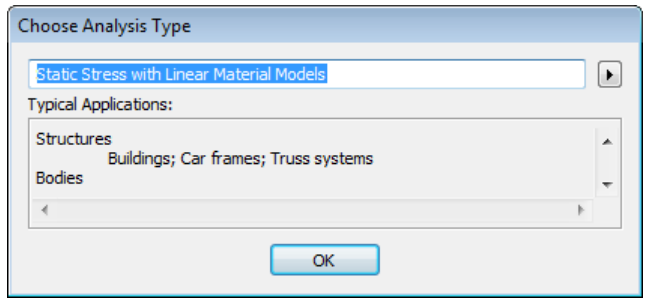


Figure 3-46 *The* ***Choose Analysis Type*** *dialog box*

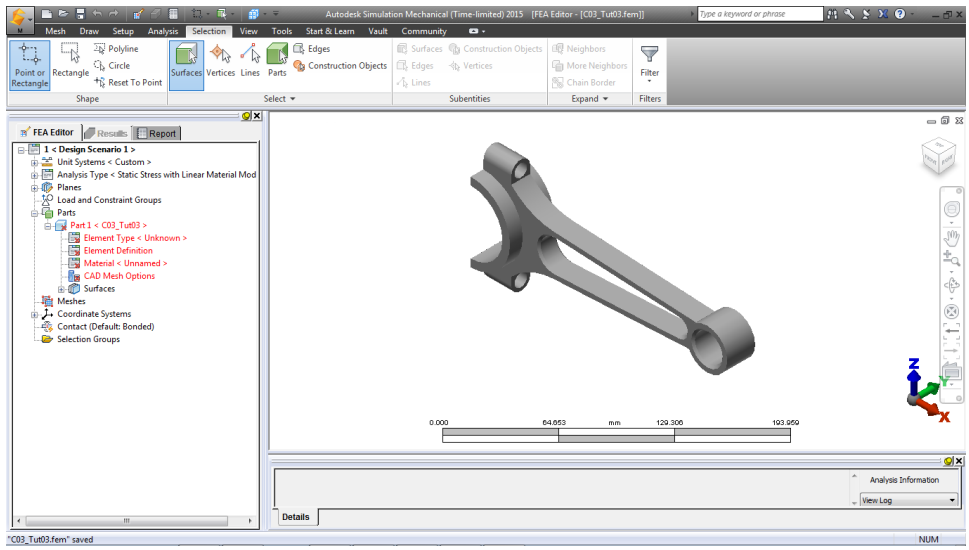


Figure 3-47 *The STEP file is imported into Autodesk Simulation Mechanical*

Changing Views of the Assembly by Using the ViewCube

Now, you need to change the views of the assembly by using the ViewCube.

1. Press and hold the middle mouse button and then drag the cursor; the model changes its orientation as you drag the cursor. Orient the model such that it is displayed similar to one shown in Figure 3-48.

2. Move the cursor over the top right corner of the ViewCube, refer to Figure 3-49 and then click the left mouse button when it highlighted; the model gets reoriented and gives you a better view.

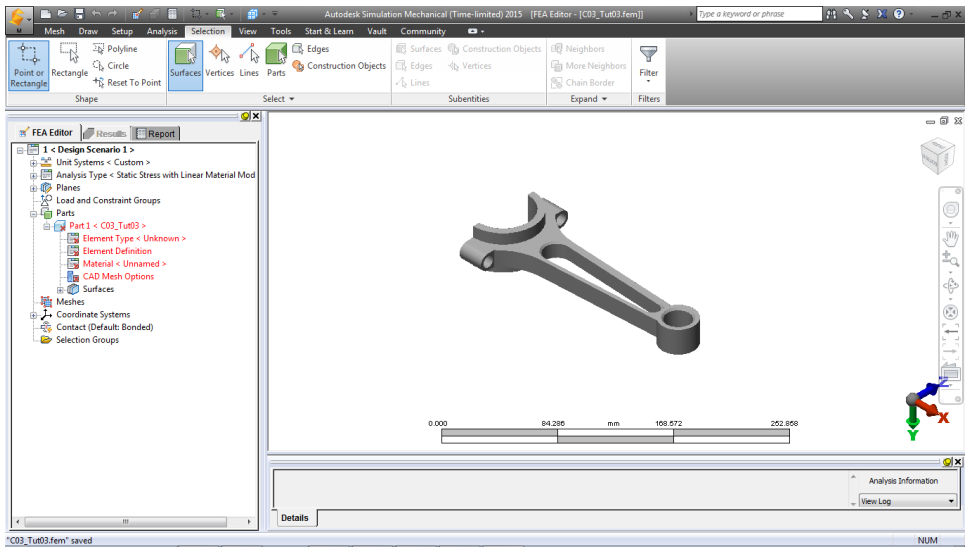


Figure 3-48 *The model after changing its default orientation*

Figure 3-49 *The ViewCube*

Saving the Model

Now, you need to save the changes made in the FEA model.

1. Choose the **Save** button from the **Quick Access Toolbar**; the changes made in the assembly are saved.

2. Choose **Close** from the **Application Menu** to close the assembly.

Self-Evaluation Test

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

1. In Autodesk Simulation Mechanical, you can directly import CAD model from the software on which it is created. (T/F)

2. To import a model directly from Autodesk Inventor to Autodesk Simulation Mechanical, Autodesk Inventor must be installed on your system. (T/F)

3. You cannot simplify the geometry of the inventor model before transferring it from Autodesk Inventor to Autodesk Simulation Mechanical. (T/F)

4. By default, the **Static Stress with Linear Material Models** analysis type is selected in the **Choose Analysis Type** dialog box. (T/F)

5. You cannot export the FEA model of Autodesk Simulation Mechanical into any other FEA file type. (T/F)

6. In Autodesk Simulation Mechanical, you can save the files of an FEA model to an archive file. (T/F)

7. The _____ is the file extension of Autodesk Simulation Mechanical.

8. The _____ is a process of recognizing contact surfaces of a model and then splitting them to get identical surfaces, if necessary.

9. The _____ dialog box displays the list of all the features of the active model in the form a tree view, so that you can suppress some of the features of the model that are not necessary for the FEA model.

10. The _____ files are compressed files which contain numerous input files of the model as well as the results of the model.

Review Questions

Answer the following questions:

1. In Autodesk Simulation Mechanical, you can turn on or off the visibility of the ViewCube. (T/F)

2. The SteeringWheels are the tracking tools that are divided into different wedges. (T/F)

3. In Autodesk Simulation Mechanical, you cannot export a FEA model to the file extension of ANSYS. (T/F)

4. When you open a model of any file type in Autodesk Simulation Mechanical, the system automatically saves the model with its original name and at original place with the *.fem* file extension. (T/F)

5. Autodesk Simulation Mechanical provides with data translation interface to various leading FEA software for directly opening FEA models created in other FEA software. (T/F)

6. Which of the following dialog boxes is used to select an analysis type?

(a) **Choose Analysis Type** (b) **Choose Analysis**
(c) **Select Analysis Type** (d) None of these

7. Which of the following dialog boxes is displayed when you choose **Archive > Create** from the Application Menu of Autodesk Simulation Mechanical?

(a) **Archive** (b) **Create Archive**
(c) **Create** (d) **Create Arch**

8. Which of the following tools is used to view the previous orientation of the model?

(a) **Previous** (b) **Previous Orientation**
(c) **Previous View** (d) None of these

9. Which of the following tools is used to increase the drawing display area to display all the objects in the current display view?

(a) **Enclose (Fit All)** (b) **Fit All**
(c) **Zoom (Fit All)** (d) Both a and c

10. Which of the following dialog boxes is displayed when you choose **Archive > Retrieve** from the Application Menu?

(a) **Extract Archive** (b) **Archive Retrieve**
(c) **Retrieve Archive** (d) None of these

EXERCISES

The input files used in the exercises are available in their respective chapter folder, where the input files of the tutorials are available. You can download the input files of the exercises used in this chapter from www.cadcim.com, if not already downloaded earlier. The complete path for downloading the files is given below:

Textbooks > CAE Simulation > Autodesk Simulation Mechanical > Autodesk Simulation Mechanical 2015 for Designers > Input Files > c03_simulation_2015_input.zip .

Exercise 1

In this exercise, you will import the *c03_exr01.igs* file from the **03_simulation_2015_input** extracted folder to Autodesk Simulation Mechanical. The model for this exercise is shown in Figure 3-50. Select the default analysis type which is **Static Stress with Linear Material Models**. After opening the assembly, you need to change the view of the model to Front view by using the ViewCube and then save the FEA model at the location *C:/Autodesk Simulation Mechanical/c03\Exr01*. (**Expected time: 10 min**)

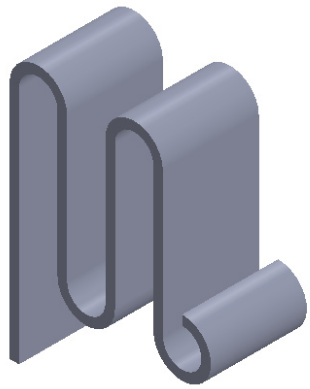


Figure 3-50 *IGS model for Exercise 1*

Exercise 2

In this exercise, you will import the *c03_exr02.STEP* file from the **03_simulation_2015_input** extracted folder to Autodesk Simulation Mechanical. The model is shown in Figure 3-51. Select the default analysis type which is **Static Stress with Linear Material Models** to carry out the analysis process. After opening the model, navigate it using the SteeringWheels. Save the FEA model at the location *C:\Autodesk Simulation Mechanical\c03\Exr02*.

(**Expected time: 10 min**)

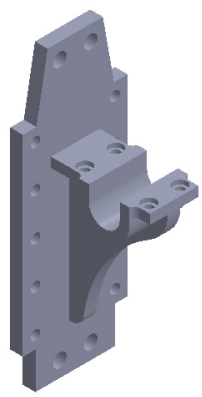


Figure 3-51 *STEP model for Exercise 2*

Exercise 3

In this exercise, you will import the *c03_exr03.IGS* file from the **03_simulation_2015_input** extracted folder to Autodesk Simulation Mechanical. The model is shown in Figure 3-52. Select the default analysis type which is **Static Stress with Linear Material Models** to carry out the analysis process. After opening the model, change its visual style to the Shaded with Edges style. Also, save the FEA model at the location *C:\Autodesk Simulation Mechanical\c03\Exr03*.

(**Expected time: 10 min**)

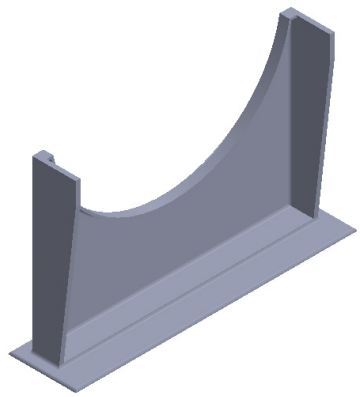


Figure 3-52 *IGS model for Exercise 3*

Answers to Self-Evaluation Test

1. T, **2.** T, **3.** F, **4.** T, **5.** F, **6.** T, **7.** .fem, **8.** Surface splitting, **9. Model Simplification**, **10.** archive