Chapter 8

Defining Materials and Boundary Conditions

Learning Objectives

After completing this chapter, you will be able to:

- *Define materials and boundary conditions in FEA models*
- *Assign material to FEA Models*
- *Manage material libraries in Autodesk Simulation Mechanical*
- *Apply boundary conditions*
- *Understand different types of constraints*
- *Apply constraints to an FEA model*
- *Understand different types of loads*
- *Define loading conditions for an FEA Model*

INTRODUCTION

Defining material and boundary conditions of an FEA model is one of the most important steps in an analysis process. In Autodesk Simulation Mechanical, you are provided with Autodesk Simulation Material Library and Autodesk Simulation Plastics Library that contain almost all the standard materials. You can select the required material from these libraries and assign it to the selected part of the model. You can also edit some of the properties of the standard materials available in the libraries such as mass density, modulus of elasticity, poisson's ratio, thermal coefficient of expansion, and shear modulus of elasticity as per the project requirement. Additionally, in Autodesk Simulation Mechanical, you can also create a new library with user-defined materials.

ASSIGNING MATERIAL

As discussed earlier, material properties must be assigned for each part in the model on which the analysis process is to be carried out. When you assign a material to a part, all physical properties of the selected material will be assigned to the part. To assign a material to a part of the model, right-click on the **Material <Unnamed>** option available under the **Part** node in the **Tree View** and then select the **Edit Material** option from the shortcut menu displayed, refer to Figure 8-1; the **Element Material Selection** dialog box will be displayed, as shown in Figure 8-2.

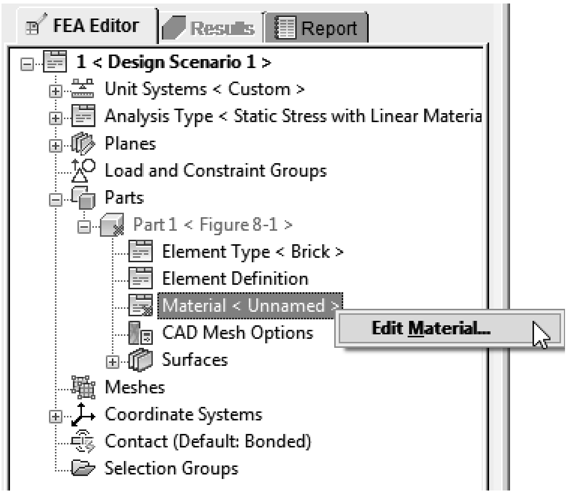


Figure 8-1 *Selecting the **Edit Material** option*

Note

*The **Edit Material** option is enabled in the shortcut menu only when the part is meshed.*

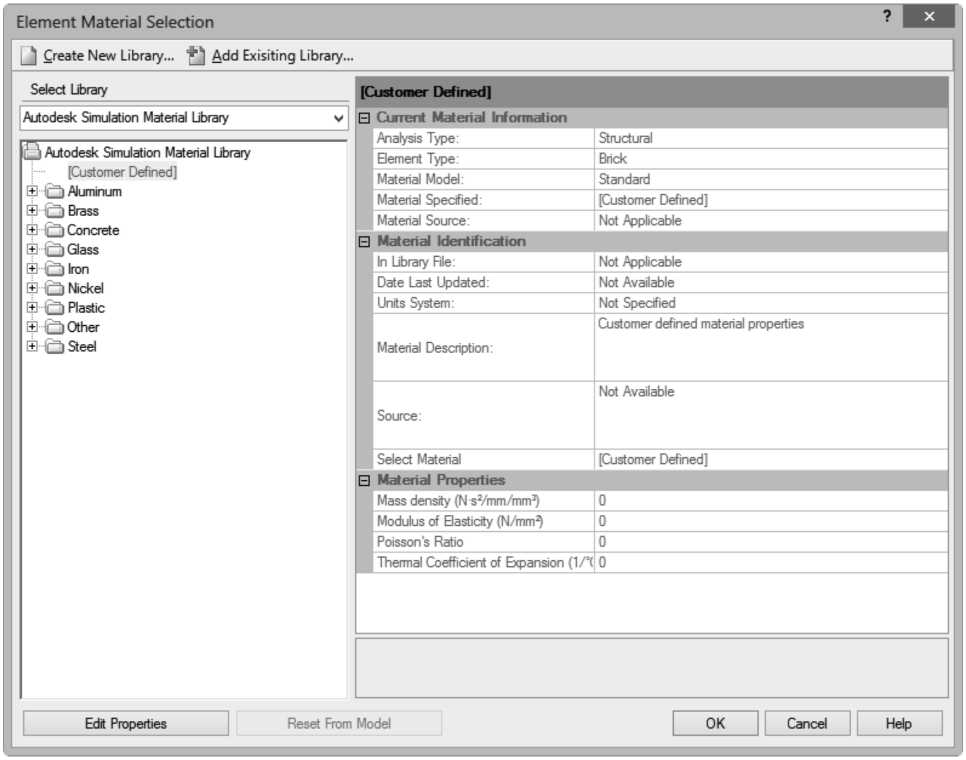


Figure 8-2 *The* ***Element Material Selection*** *dialog box*

By default, **Autodesk Simulation Material Library** is selected in the **Select Library** drop-down list of the **Element Material Selection** dialog box. As a result, a number of material families that are available in this library are listed in the left area of the dialog box. Click on the **+** sign located on the left of the desired material family to display all materials under that family. Figure 8-3 shows the **Element Material Selection** dialog box with the **Aluminum** material family expanded. Select the required material from the expanded material family from the left area of the dialog box; all the properties of the selected material will be displayed on the right of the dialog box. Next, choose the **OK** button; the material properties of the selected material will be assigned to the selected part.

In Autodesk Simulation Mechanical, you can also change or modify the material properties of the selected material. To do so, select the required material from the left of the **Element Material Selection** dialog box. Next, choose the **Edit Properties** button available on the bottom of the dialog box; the **Element Material Specification** dialog box will be displayed, refer to Figure 8-4. In this dialog box, you can specify new mass density, modulus of elasticity, poisson's ratio, and thermal coefficient of expansion for the selected material. After specifying the new material properties for the selected material in the dialog box, choose the **OK** button; the material properties of the selected material will be modified accordingly.

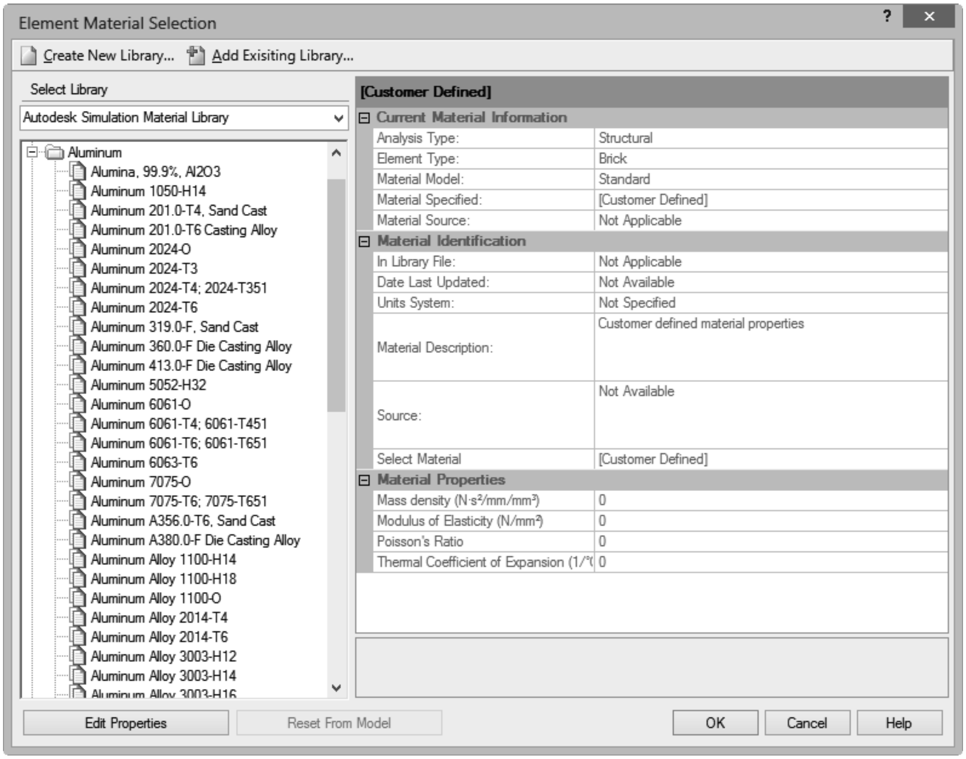


Figure 8-3 *The expanded **Aluminum** category in the **Element Material Selection** dialog box*

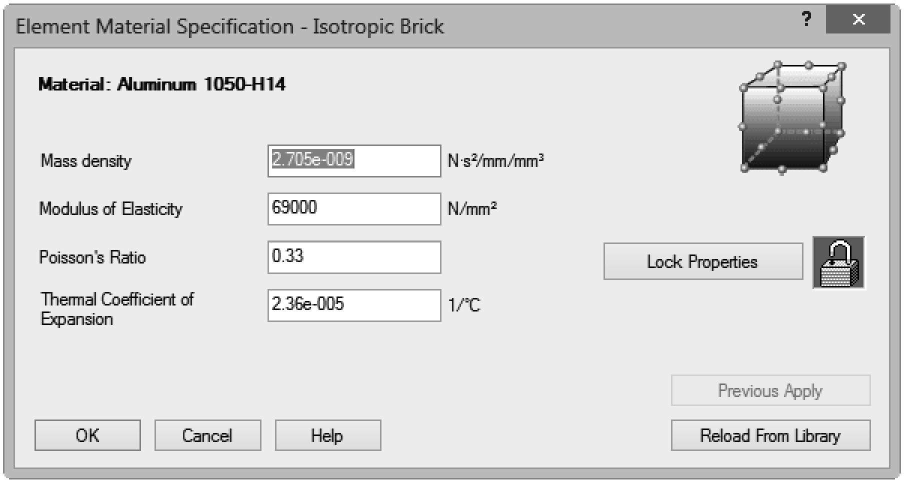


Figure 8-4 *The **Element Material Specification** dialog box*

Note

*The changes made in the material properties of the selected material by using the **Element Material Specification** dialog box will be saved in the model database only. In other words, there will be no modification made in the material library of Autodesk Simulation Mechanical. As a result, when you invoke the same material next time, its default or standard properties will be displayed.*

MANAGING MATERIAL LIBRARIES

Autodesk Simulation Mechanical allows you to manage material libraries by creating new material libraries, material categories, user-defined materials, setting selected library as default library, and so on. The procedure to perform all these operations for managing material libraries is discussed next.

Creating New Material Library

By default, in Autodesk Simulation Mechanical, you are provided with **Autodesk Simulation Material Library** and **Autodesk Simulation Plastics Library**. In addition to the default material libraries in Autodesk Simulation Mechanical, you can also create new material libraries. To do so, click on the **Tools** tab in the **Ribbon** to display all the tools available in this tab. Next, choose the **Manage Material Library** tool from the **Options** panel of the **Tools** tab; the **Autodesk Material Library Manager** dialog box will be displayed, as shown in Figure 8-5.

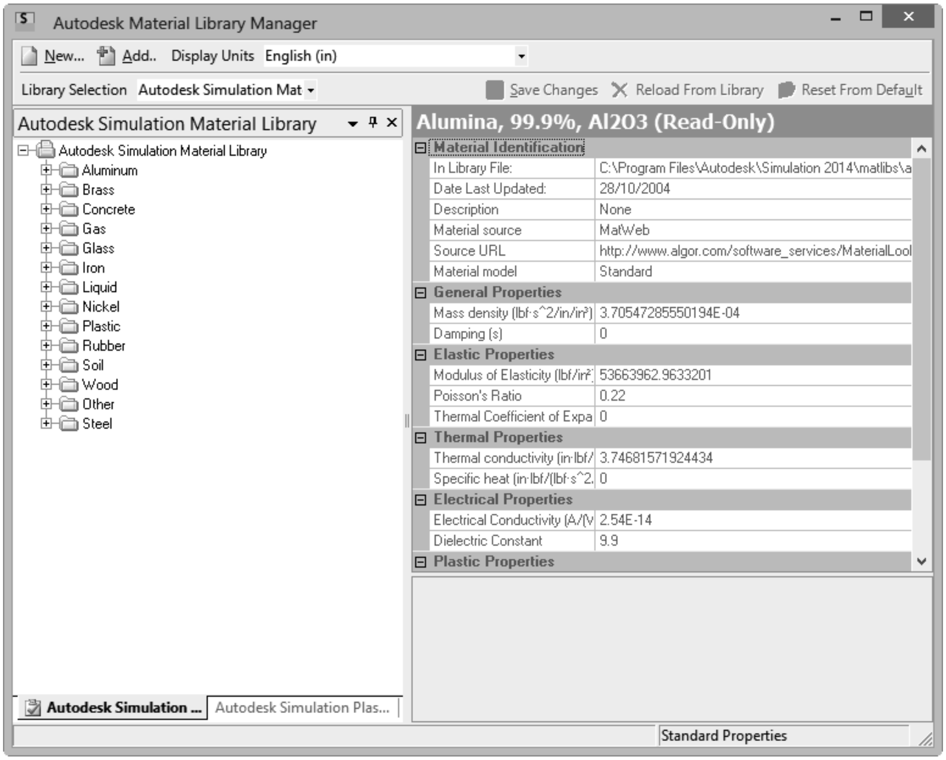


*Figure 8-5 The **Autodesk Material Library Manager** dialog box*

Choose the **New** button from the dialog box; the **Create Material Library** dialog box will be displayed. Browse to the location where you want to save the material library file and then specify a name to it in the **File name** edit box of the dialog box. Next, choose the **Save** button; the **Create Library** window will be displayed, refer to Figure 8-6. By default, the name entered for the previously created material library file will be displayed in the edit box of the **Create Library** window. You can enter a unique name for the material library in this edit box. Note that the material library created will be displayed in the list of available material libraries with the specified name. Enter the name for the material library and then choose the **OK** button; a new material library of the specified name will be created and selected in the **Library Selection** drop-down list of the **Autodesk Material Library Manager** dialog box. Figure 8-7 shows the **Autodesk Material Library Manager** dialog box with the new material library created.

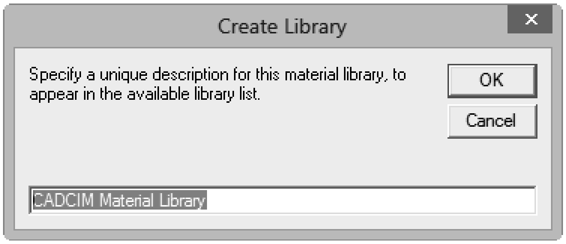


Figure 8-6 *The* ***Create Library*** *window*

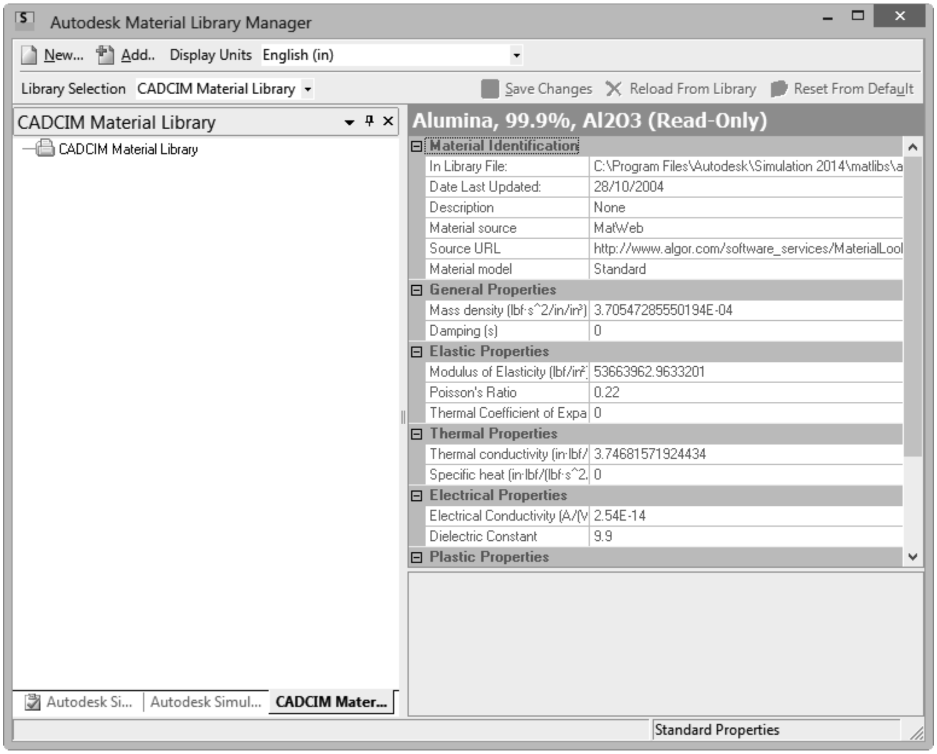


Figure 8-7 *The new material library created and selected in the* ***Autodesk Material Library Manager*** *dialog box*

Adding Material Categories

In a library, you can group materials of same family in a category. To create a category inside the library, invoke the **Autodesk Material Library Manager** dialog box by choosing the **Manage Material Library** tool from the **Options** panel of the **Tools** tab. Next, select the required library from the left of the **Autodesk Material Library Manager** dialog box and right-click; a shortcut menu will be displayed, refer to Figure 8-8.

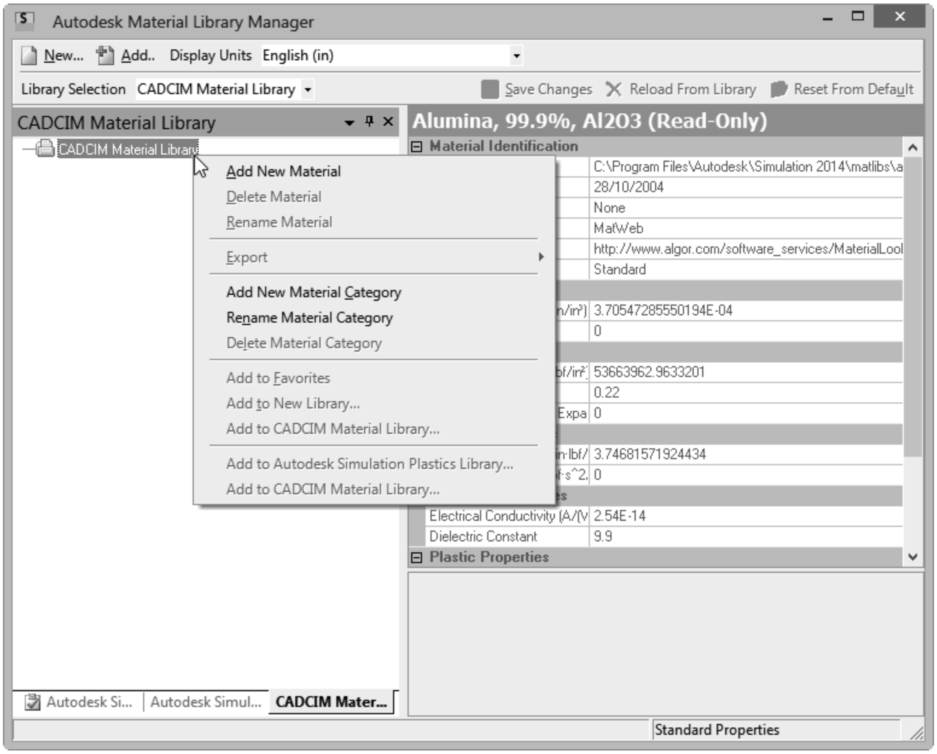


***Figure 8-8** A shortcut menu displayed in the **Autodesk Material Library Manager** dialog box*

Select the **Add New Material Category** option from the shortcut menu displayed; the **Material Category Name** window will be displayed, as shown in Figure 8-9. Enter the name of the category in the edit box of the window and then choose the **OK** button; the category of the specified name is created. Similarly, you can create multiple categories in the library. Figure 8-10 shows the **Autodesk Material Library Manager** dialog box with categories named AB, GA, and KV created under the **CADCIM Material Library** library.

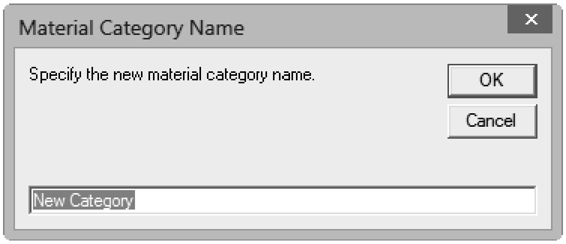


*Figure 8-9 The **Material Category Name** window*

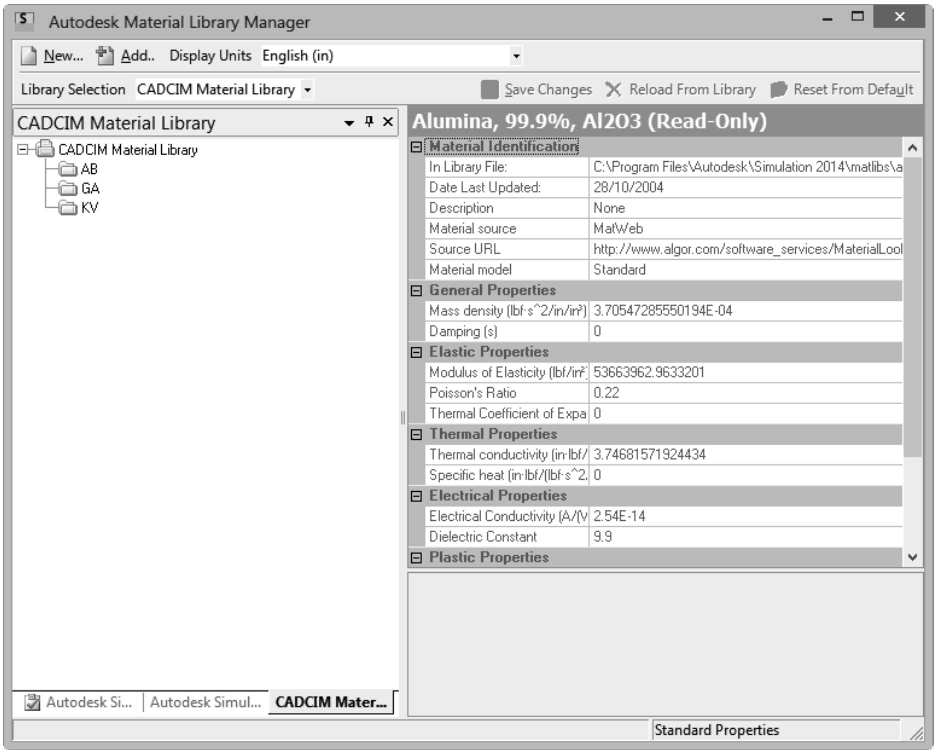


*Figure 8-10 The **Autodesk Material Library Manager** dialog box with categories named **AB**, **GA**, and **KV** created under the **CADCIM Material Library***

Adding Material Under a Material Category

To add material under a category of a library, invoke the **Autodesk Material Library Manager** dialog box and then select the category for which you want to add the material. Next, right-click and select the **Add New Material** option from the shortcut menu displayed; the **New Material** dialog box will be displayed, refer to Figure 8-11.

Specify the name of the material, type of material, material description, material source description (if any), and material source URL in their respective fields available on the left of the **New Material** dialog box. After specifying all the details, choose the **OK** button; the material with the specified details will be created and listed under the selected category in the **Autodesk Material Library Manager** dialog box, refer to Figure 8-12.

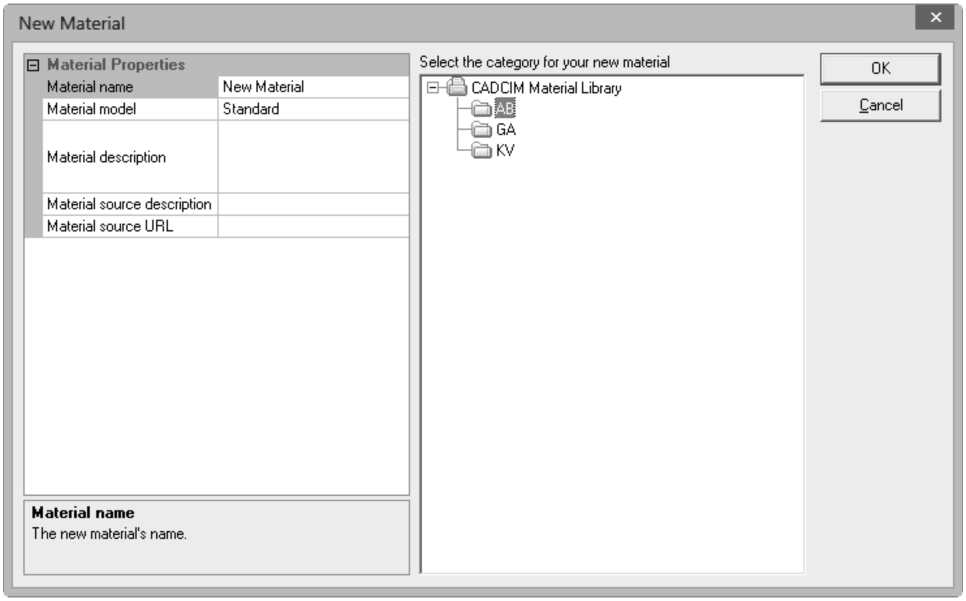


*Figure 8-11 The **New Material** dialog box*

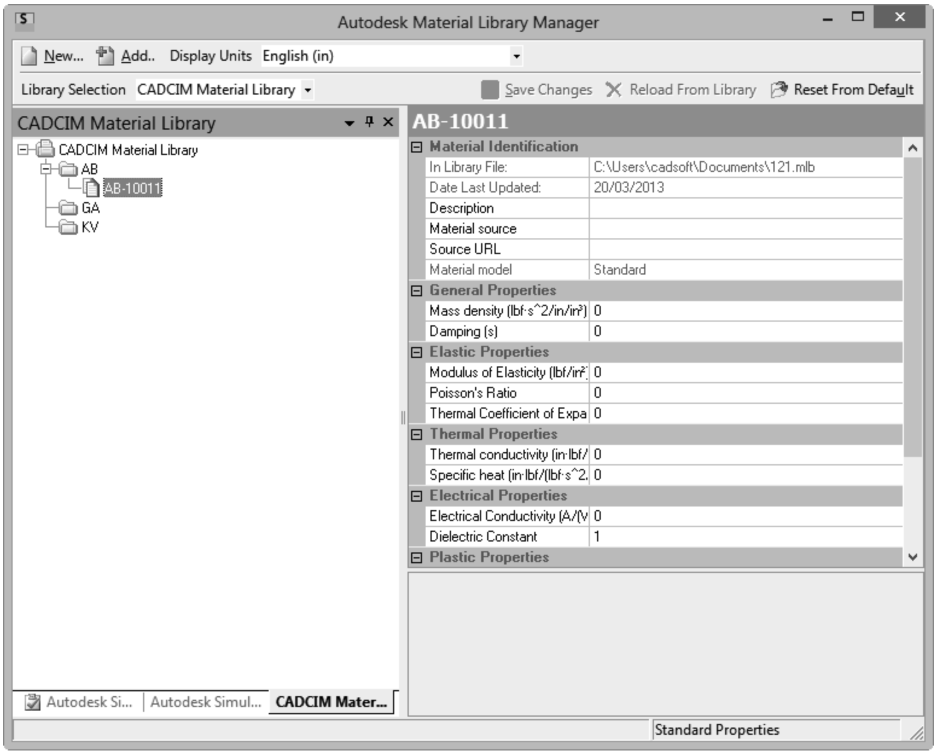


***Figure 8-12** The new material AB-10011 is added under the AB category of the library*

Now you need to specify the material properties for the newly created material. To do so, select the newly created material from the left of the **Autodesk Material Library Manager** dialog box; all the default material properties of the selected material will be displayed on the right of the dialog box. Specify the mass density, modulus of elasticity, poisson's ratio, shear modulus of elasticity, thermal conductivity, and so on of the material in their respective fields available on the right of the dialog box. After specifying all the material properties for the added material, choose the **Save Changes** button available at the top of the dialog box; the specified material properties will be saved in the material library file. Similarly, you can add a number of materials with user-defined material properties as per the requirement.

Deleting Material

In Autodesk Simulation Mechanical, you can delete a material from a material library. To do so, invoke the **Autodesk Material Library Manager** dialog box and then select the material library from the **Library Selection** drop-down list of the dialog box whose material is to be deleted. Next, select the material to be deleted from the material library list and right-click to display a shortcut menu. Choose the **Delete Material** option from the shortcut menu displayed; the **Delete Material** window will be displayed asking you to confirm whether you want to delete the selected material and its properties or not. Choose the **Yes** button from the window; the selected material will be deleted from the list of materials available in the library.

Deleting Material Category

Similar to deleting material from the library, you can also delete a material category from a material library. To do so, invoke the **Autodesk Material Library Manager** dialog box and then select the material library from the **Library Selection** drop-down list of the dialog box whose material category is to be deleted. Next, select the material category to be deleted from the material library list and right-click to display a shortcut menu. Choose the **Delete Material Category** option from the shortcut menu; the **Delete Category and Materials** window will be displayed asking you to confirm whether you want to delete the selected category and all its materials or not. Choose the **Yes** button from the window; the selected material category and all the materials available under it will be deleted from the library.

Loading Material Library File

In Autodesk Simulation Mechanical, the material library file is saved in *.mlb* format. The *.mlb* is the file extension for material library files. If you have a material library file (*.mlb*) which is currently not loaded in Autodesk Simulation Mechanical, you can load it by using the **Add** button available in the **Autodesk Material Library Manager** dialog box. To load a material library file, invoke the **Autodesk Material Library Manager** dialog box and choose the **Add** button from it; the **Add Library to Available Library List** dialog box will be displayed. Browse to the location where the library file (*.mlb*) is saved. Next, select the library file to be loaded and then choose the **Open** button from the dialog box; the **Add Library** dialog box will be displayed. Choose the **OK** button from the dialog box; the selected material file will be uploaded and a list of all the materials available in library file will be displayed in the left of the dialog box.

Setting Material Library as Default Library

By default, the **Autodesk Simulation Material Library** library is set as the default library. As a result, whenever you invoke the **Element Material Selection** dialog box to assign material to a model, the **Autodesk Simulation Material Library** is selected in the **Select Library** drop-down list and displays all the available material categories on the left in the dialog box. You can also set any other material library as the default library as per the requirement. To do so, invoke the **Autodesk Material Library Manager** dialog box by choosing the **Manage Material Library** tool from the **Options** panel of the **Tools** tab in the **Ribbon**. Next, select the material library that you want to set as the default library from the **Library Selection** drop-down list of the dialog box. Click on the down arrow available on the right of the selected library title in the dialog box to display a menu, refer to Figure 8-13. Select the **Set as Default** option from the menu displayed; the selected library will be set as the default library.

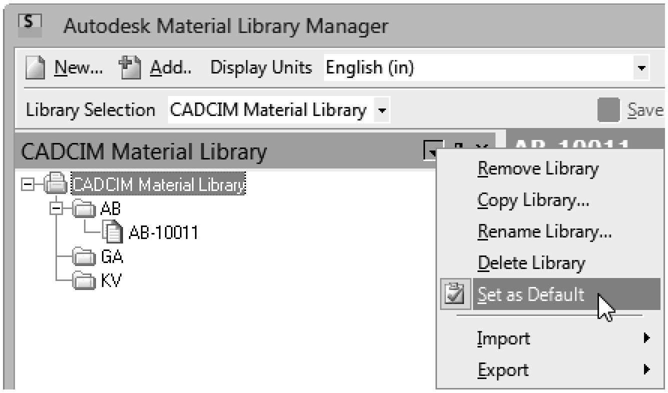


*Figure 8-13 Partial view of the **Autodesk Material Library Manager** dialog box with a menu displayed by clicking on the down arrow*

Removing/Deleting Library

In Autodesk Simulation Mechanical, you can remove material libraries from the list of libraries that are no longer required. To do so, invoke the **Autodesk Material Library Manager** dialog box and then select the material library that you want to remove from the list of libraries. Next, click on the down arrow available on the right of the selected library title in the dialog box to display a menu, refer to Figure 8-13. Choose the **Remove Library** option from the menu displayed; the **Database Removal** window will be displayed asking you whether you want to remove the library or not, refer to Figure 8-14.

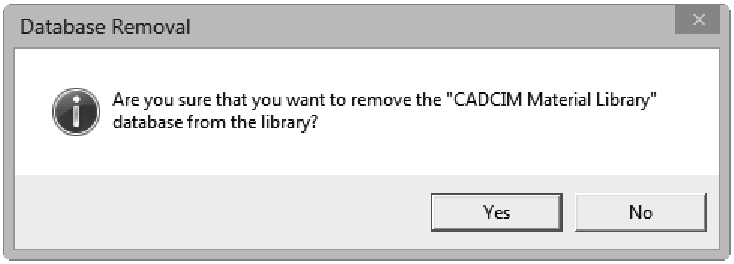


*Figure 8-14 The **Database Removal** window*

Choose the **Yes** button from the **Database Removal** window; the selected library will be removed. However, its data files will still be available in your system hard disk. As a result, you can load the removal material library again by loading its data files as discussed earlier in the section **Loading Material Library File**.

To permanently remove the library, select the **Delete Library** option from the menu displayed instead of selecting the **Remove Library** option. On selecting this option, the **Database Deletion** window will be displayed asking you whether you want to remove the library and its database files from the disk or not, refer to Figure 8-15. Choose the **Yes** button, the selected library will be removed from the list of libraries. Also, its data files will be deleted from the system hard disk.

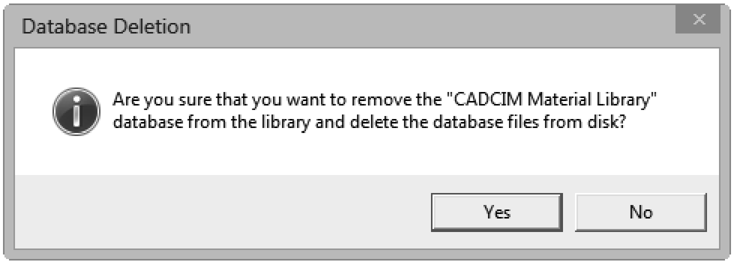


Figure 8-15 *The **Database Deletion** window*

Copying Material Library

In Autodesk Simulation Mechanical, you can create a copy of a material library to create a new material library by editing the material properties of the materials available in the copied library. To create a copy of an existing library, invoke the **Autodesk Material Library Manager** dialog box and then select the material library to be copied from the **Library Selection** drop-down list. Next, click on the down arrow available on the right of the selected library title in the dialog box to display a menu, refer to Figure 8-16. Select the **Copy Library** option from the menu displayed; the **Copy Library To** dialog box will be displayed. Browse to the location where you want to copy the library file and then specify a name for the new library file in the **File name** edit box of the dialog box. Next, choose the **Save** button; the **Create Library** window will be displayed. By default, the name entered previously copied for the material library file will be displayed in the edit box of this window. You can specify a unique name for the new material library in this edit box. Note that the material library created will be displayed in the list of available material libraries with the name entered in this edit box. Enter the name for the material library and then choose the **OK** button; a new material library with the specified name will be created and selected in the **Library Selection** drop-down list of the **Autodesk Material Library Manager** dialog box. Note that in the material library created, all the materials and their material properties are same as that of the parent material library. You can change the material properties as discussed earlier and then choose the **Save Change** button.

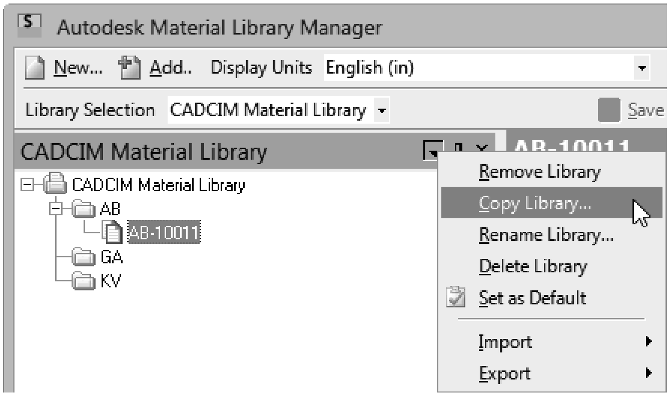


*Figure 8-16 A menu displayed in the **Autodesk Material Library Manager** dialog box*

BOUNDARY CONDITIONS

Boundary conditions are defined on the boundary of an FEA model to represent the effect of the surrounding environment. It includes forces, pressures, displacement, and so on. In Autodesk Simulation Mechanical, the boundary conditions are divided into two categories: loads and constraints. Both are discussed next.

Constraints

Constraints are defined as the boundary conditions that prevent the model from translation or rotational movement. In other words, the constraints are used to restrict the degree of freedom of a model. The applied constraints act as rigid supports between the model and the ground. You can select nodes, edges, and surfaces of a model to apply constraints. The tools used for applying constraints are discussed next.

General Constraint

Ribbon:	Setup > Constraints > General Constraint

The **General Constraint** tool is used to constrain the degree of freedom of nodes. You can select nodes, edges, or surfaces of a model to apply constraint by using the **General Constraint** tool. Note that depending upon the geometry selected for applying the constraint, the degree of freedom of the nodes of the selected geometry will be constrained. The procedure of applying general constraint on nodes, edges, and surfaces of the model are discussed next.

Applying Nodal General Constraint

General constraint applied on a node is called nodal general constraint. To apply a nodal general constraint, select a node of the model on which you want to apply the constraint and then choose the **General Constraint** tool from the **Constraints** panel of the **Setup** tab in the **Ribbon**; the **Creating Nodal General Constraint Object** dialog box will be displayed, as shown in Figure 8-17. Alternatively, you can invoke this dialog box by selecting a node and right-clicking in the drawing area to display a shortcut menu. Next, select **Add >**

Nodal General Constraint from the shortcut menu displayed. The options available in this dialog box are discussed next.

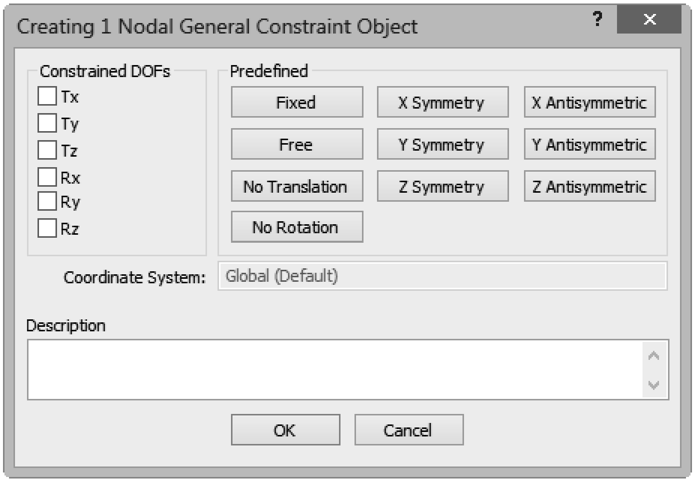


*Figure 8-17 The **Creating Nodal General Constraint Object** dialog box*

> **Tip**. *You can also select multiple nodes of a model to constrain by using the CTRL key. On selecting more than one node, the name of the dialog box invoked on choosing the **General Constraint** tool will be changed to **Creating** (number of nodes selected) **Nodal General Constraint Objects**.*

Constrained DOFs Area

The **Constrained DOFs** area of the dialog box is used to control or specify the constraints for the selected node. By default, all the check boxes available in this area are cleared. As a result, all degrees of freedom of the selected node are free. On selecting the **Tx** check box, the translation degree of freedom along the X-axis of the selected node will be constrained. Similarly, on selecting the **Ty** and **Tz** check boxes, the translation degree of freedom along the Y and Z axes will be constrained.

To constrain the rotational degree of freedom along the X, Y, and Z axes, you can select the **Rx**, **Ry**, and **Rz** check boxes, respectively, from the **Constrained DOFs** area of the dialog box.

Predefined Area

The buttons available in the **Predefined** area of the dialog box are used to select predefined sets of constraints to be applied to the selected nodes. On choosing the **Fixed** button from the **Predefined** area, the degree of freedom of the selected nodes will be fixed and all the check boxes available in the **Constrained DOFs** area of the dialog box will be selected.

On choosing the **Free** button from the **Predefined** area, all the selected check boxes of the **Constrained DOFs** area will be cleared and the degree of freedom of the selected nodes will be free. If you choose the **No Translation** button, the translation degree of freedom along X, Y, and Z axes of the selected nodes will be constrained and the rotational degree of freedom along the X, Y, and Z axes will be free to rotate. On selecting the **No Rotation** button, the rotational degree of freedom of the selected nodes along X, Y, and Z axes will be constrained and the translation degree of freedom along X, Y, and Z axes will be free.

Similarly, you can choose the **X Symmetry**, **Y Symmetry**, **Z Symmetry**, **X Antisymmetric**, **Y Antisymmetric**, and **Z Antisymmetric** buttons from the **Predefined** area of the dialog box to apply the predefined sets of symmetric or antisymmetric constraints to the selected nodes.

Description

The **Description** area of the dialog box is used to enter a comment or description about the constraint being applied.

After specifying the required set of constraints to be applied to the selected nodes, choose the **OK** button from the dialog box; the constraint will be applied. Figure 8-18 shows a meshed model with fixed constraint applied on some of the nodes.

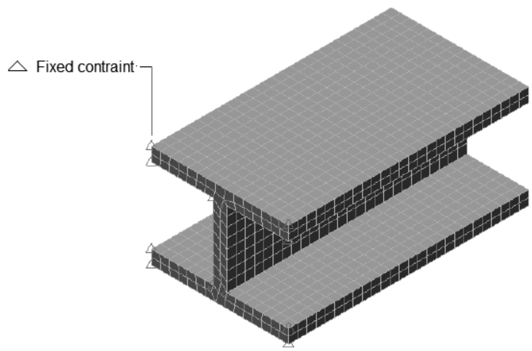


Figure 8-18 *The fixed constraint applied to some of the nodes of the model*

Applying Surface Constraint

General constraint applied on a surface is called surface general constraint. To apply surface general constraint, select the surface of the model on which you want to apply constraint and then choose the **General Constraint** tool from the **Constraints** panel of the **Setup** tab in the **Ribbon**; the **Creating Surface General Constraint Object** dialog box will be displayed, as shown in Figure 8-19. Alternatively, you can invoke this dialog box by selecting a surface and right-clicking in the drawing area to display a shortcut menu. Next, select **Add > Surface General Constraint** from the shortcut menu displayed. The options available in this dialog box are same as discussed earlier. Figure 8-20 shows a meshed model with the fixed constraint applied on a surface.

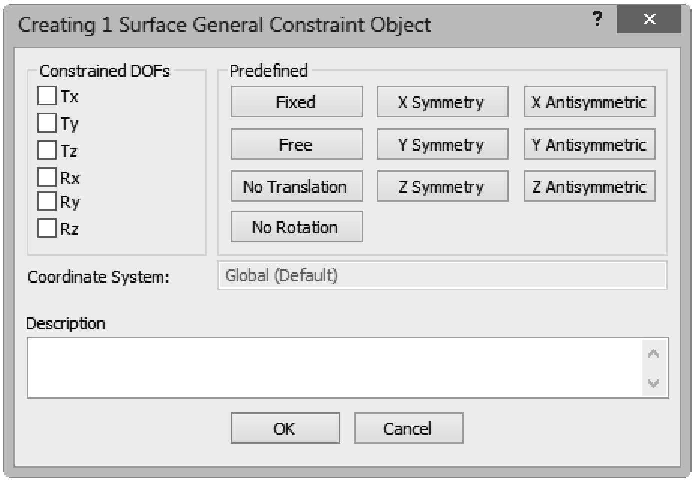


*Figure 8-19 The **Creating Surface General Constraint Object** dialog box*

Applying Edge Constraint

The method of applying constraints on edges is similar to applying constraints on surfaces and nodes. Figure 8-21 shows a meshed model with fixed constraint applied on an edge.

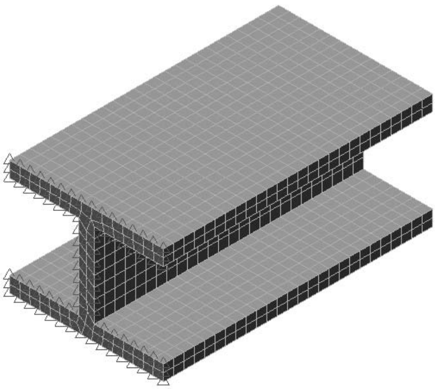


Figure 8-20 The fixed constraint applied on a surface of the model

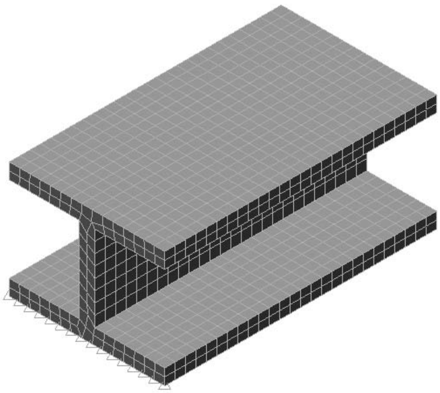


Figure 8-21 The fixed constraint applied on an edge of the model

Note

*To select a node/surface/edge of a model, click on the **Selection** tab in the **Ribbon**. Next, choose the **Point or Rectangle** tool from the **Shape** panel and the **Vertices/Surfaces/Edges** tool from the **Select** panel of the **Selection** tab in the **Ribbon**.*

Tip. *You can select nodes, surfaces, or edges for applying constraint either before or after invoking the dialog box.*

Pin Constraint

Ribbon:	Setup > Constraints > Pin Constraint

The **Pin Constraint** tool is used to constrain the radial, tangential, and axial degrees of freedom of the selected cylindrical surface. This constraint is widely used to simulate pin connection. You can apply pin constraint only to a cylindrical surface. To apply the pin constraint, select a cylindrical surface of the model on which you want to apply the pin constraint and then choose the **Pin Constraint** tool from the **Constraints** panel of the **Setup** tab in the **Ribbon**; the **Creating Pin Constraint Object** dialog box will be displayed, as shown in Figure 8-22. Alternatively, you can invoke this dialog box by selecting a cylindrical surface and right-clicking in the drawing area to display a shortcut menu. Next, select **Add > Surface Pin Constraint** from the shortcut menu displayed. The options available in this dialog box are discussed next.

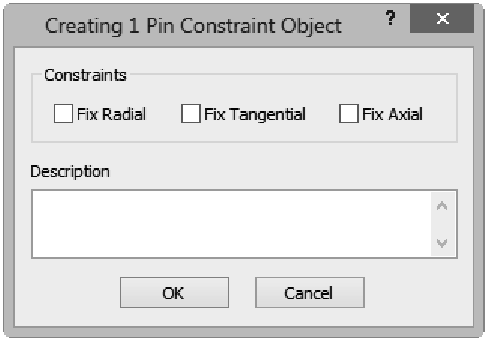


***Figure 8-22** The **Creating Pin Constraint Object** dialog box*

Constraints Area

The check boxes available in this area are used to constrain the radial, tangential, and axial degrees of freedom of the selected surface. These check boxes are discussed next.

Fix Radial

On selecting the **Fix Radial** check box, the selected cylindrical surface will be restricted to move or deform radially.

Fix Tangential

On selecting the **Fix Tangential** check box, the selected cylindrical surface will be restricted to rotate tangentially around the circumference of the cylindrical surface selected for applying the constraint.

Fix Axial

On selecting the **Fix Axial** check box, the selected cylindrical surface will be restricted to move axially along the axis of the cylindrical surface selected for applying the constraint.

Description Area

The **Description** area of the dialog box is used to enter a comment or description about the applied constraint.

After fixing the required degree of freedom of the selected cylindrical surface to simulate the pin connection, choose the **OK** button from the dialog box; the pin constraint will be applied. Figure 8-23 shows a model with the pin constraint applied.

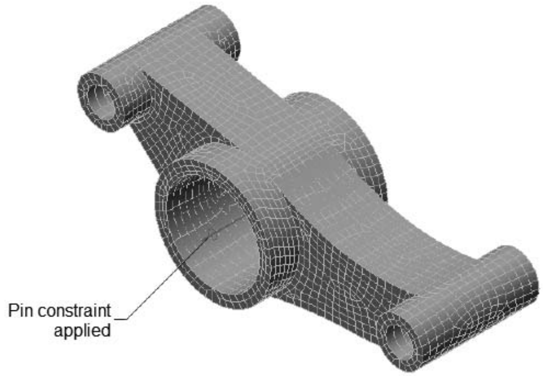


Figure 8-23 *The pin constraint applied to the cylindrical surface*

1D Spring Support

Ribbon:	Setup > Constraints > 1D Spring Support

The **1D Spring Support** tool is used to apply stiffness to nodes, edges, or surfaces of a model to resist their translational or rotational degree of freedom along or about any specified direction vector. Note that the deformation that occurs after applying the 1D spring support constraint to a node, edge, or a surface is calculated based on the spring equation F = KX. The greater the stiffness value, the lower will be the deformation. The procedure of applying 1D spring support constraint on nodes, edges, and surfaces of a model are discussed next.

Applying Nodal 1D Spring Support

The 1D Spring support applied on a node is called nodal 1D Spring support. To apply nodal 1D Spring support, select a node of the model and then choose the **1D Spring Support** tool from the **Constraints** panel of the **Setup** tab in the **Ribbon**; the **Creating Nodal 1D Spring Support Element** dialog box will be displayed, as shown in Figure 8-24. Alternatively, you can invoke this dialog box by selecting a node and right-clicking in the drawing area to display a shortcut menu. Next, select **Add > Nodal 1D Spring Support** from the shortcut menu displayed. You can also select multiple nodes by pressing the CTRL key. The options available in the **Creating Nodal 1D Spring Support Element** dialog box are discussed next.

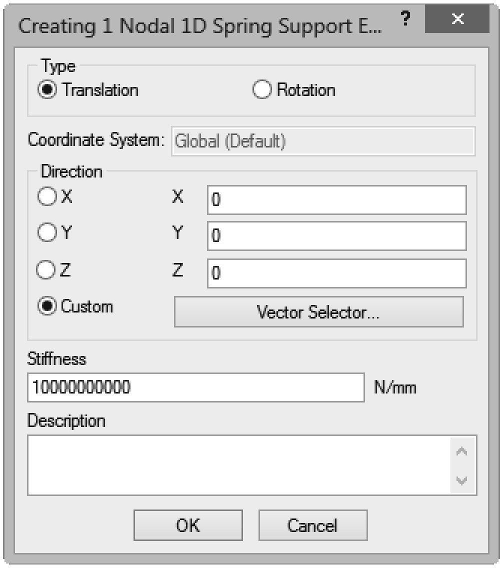


***Figure 8-24** The **Creating Nodal 1D Spring Support Element** dialog box*

Type Area

On selecting the **Translation** radio button in the **Type** area of the dialog box, you can restrict the translation movement of the selected node or nodes. To restrict the rotational movement of the selected node or nodes, select the **Rotation** radio button.

Direction Area

The **Direction** area of the dialog box allows you to select the required direction vector along/about which you want to restrict the translational/rotational movement. You can select the **X**, **Y**, or **Z** radio button to restrict the translational/rotational movement along/about X, Y, or Z axis. You can also select the **Custom** radio button to specify the direction vector other than the X, Y, and Z axes directions of the coordinate system.

To specify the direction vector other than X, Y, and Z directions, select the **Custom** radio button; the **X**, **Y**, and **Z** edit boxes of the dialog box will be enabled. In these edit boxes, you can specify the X, Y, and Z coordinates of the direction vector.

Moreover, you can specify the direction vector by selecting two points. To do so, choose the **Vector Selector** button available on the right of the **Custom** radio button in the dialog box; the dialog box will disappear and you will be prompted to specify the direction vector. Specify the start point and then the endpoint toward which the direction will point. As soon as you specify the endpoint for the direction vector, the **Creating Nodal 1D Spring Support Element** dialog box will be displayed again with the coordinate value of the specifying direction vector displayed in the X, Y, and Z edit boxes of the dialog box.

Stiffness Area

The **Stiffness** edit box is used to specify the stiffness of the elements for resisting the movement in the specified direction.

Description Area

The **Description** area of the dialog box is used to enter a comment or description about the applied constraint.

After specifying all the parameters to define 1D Spring support in a particular direction, choose the **OK** button from the dialog box; the 1D Spring support will be applied.

Applying Surface 1D Spring Support

The 1D Spring support applied on a surface is called surface 1D Spring support. To apply Surface 1D Spring support, select a surface of the model and then choose the **1D Spring Support** tool from the **Constraints** panel of the **Setup** tab in the **Ribbon**; the **Creating Surface 1D Spring Support Element** dialog box will be displayed, as shown in Figure 8-25. Alternatively, you can invoke this dialog box by selecting a surface and right-clicking in the drawing area to display a shortcut menu. Next, select **Add > Surface 1D Spring Support** from the shortcut menu. You can also select multiple surfaces by using the CTRL key. The options available in the **Creating Surface 1D Spring Support Element** dialog box are same as discussed earlier.

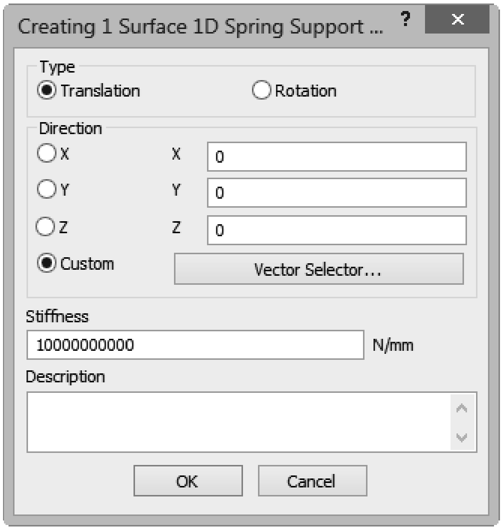


*Figure 8-25 The **Creating Surface 1D Spring Support Element** dialog box*

Applying Edge 1D Spring Support

The 1D Spring support applied on an edge is called edge 1D Spring support. To apply edge 1D Spring support, select an edge of the model and then choose the **1D Spring Support** tool; the **Creating Edge 1D Spring Support Element** dialog box will be displayed. Alternatively, you can invoke this dialog box by selecting an edge and right-clicking on the drawing area to display a shortcut menu. Next, select **Add > Edge 1D Spring Support** from the shortcut menu. The options available in this dialog box are same as discussed earlier.

Figure 8-26 shows a model with 1D spring support applied on the bottom front edge of the top plate.

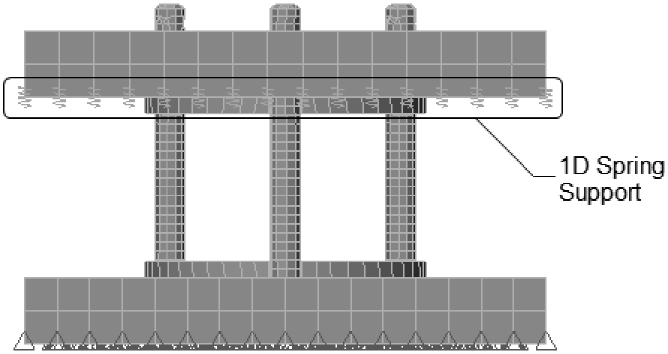


Figure 8-26 *The edge 1D spring support applied*

3D Spring Support

Ribbon:	Setup > Constraints > 3D Spring Support

The **3D Spring Support** tool is used to apply stiffness to nodes, edges, or surfaces of a model to restrict their translational/rotational degree of freedom along/about the X, Y, or Z global direction. You can also restrict the translational/rotational degree of freedom along/about more than one (X, Y, and Z) global directions by using this tool. Note that the deformation that occurs after applying the 3D spring support constraint to a node, edge, or a surface is based on the spring equation F = KX. The greater the stiffness value, the lower will be the deformation.

The procedure of applying 3D spring support constraint on nodes, edges, and surfaces of a model is same as discussed for applying 1D spring support. The only difference in 3D spring support is that you can restrict the translational or rotational movements in all directions globally.

To apply 3D spring support, select a node/edge/surface and then choose the **3D Spring Support** tool from the **Constraints** panel of the **Setup** tab in the **Ribbon**; the **Creating Nodal 3D Spring Support Element/Creating Edge 3D Spring Support Element/Creating Surface 3D Spring Support Element** dialog box will be displayed. Figure 8-27 shows the **Creating Edge 3D Spring Support Element** dialog box displayed on choosing the **3D Spring Support** tool after selecting an edge.

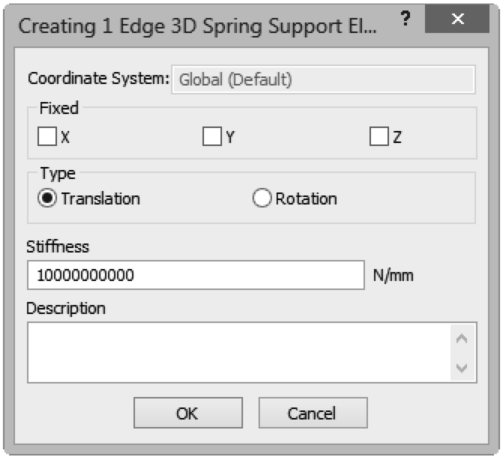


Figure 8-27 The ***Creating Edge 3D Spring Support Element*** *dialog box*

The options available in this dialog box are same as discussed while applying 1D spring support, except the options in the **Fixed** area. The **Fixed** area and its options are discussed next.

Fixed Area

The **Fixed** area of the dialog box is used to select the required directions to be resisted for translational or rotational movements along or about the X, Y, and Z direction. Select the required check boxes from the **Fixed** area of the dialog box along or about which you want to restrict the translational or rotational movements.

Figure 8-28 shows a model with 3D spring support applied on the bottom front edge of the top plate.

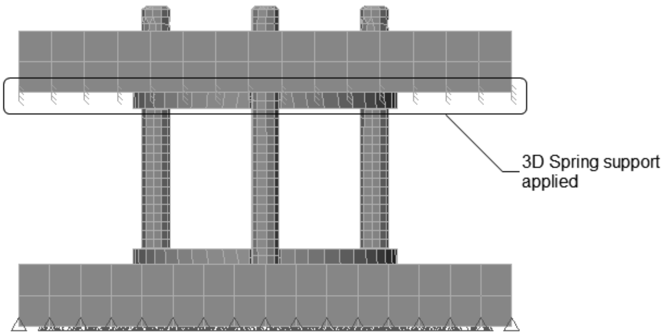


Figure 8-28 The 3D spring support applied

Prescribed Displacement Constraint

Ribbon: Setup > Constraints > Prescribed Displacement

A prescribed displacement allows nodes to translate or rotate through a specified distance in a specified vector direction. You can select nodes, edges, or surfaces of a model to apply prescribed displacement by using the **Prescribed Displacement** tool. Note that on selecting an edge or a surface, the prescribed displacement will be applied to all the nodes of the selected edge or surface.

To apply prescribed displacement to a surface, select the surface of a model and then choose the **Prescribed Displacement** tool from the **Constraints** panel of the **Setup** tab in the **Ribbon**; the **Creating Surface Prescribed Displacement Element** dialog box will be displayed, refer to Figure 8-29. The options available in this dialog box are discussed next.

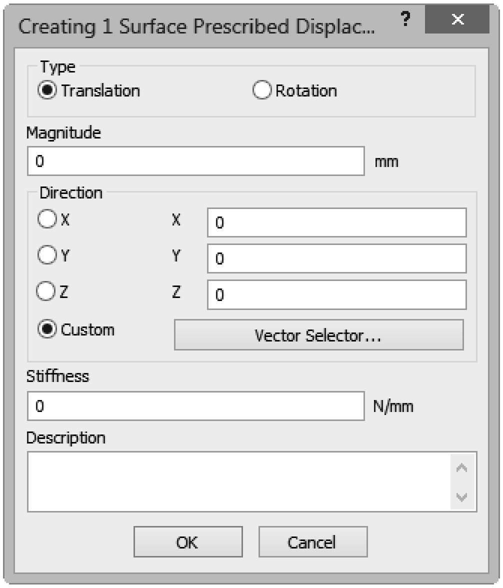


*Figure 8-29 The **Creating Surface Prescribed Displacement Element** dialog box*

Note

*The name of the dialog box displayed on choosing the **Prescribed Displacement** tool depends upon the geometry selected. If you select a node and choose the **Prescribed Displacement** tool, the **Creating Nodal Prescribed Displacement Element** dialog box will be displayed and if you select an edge, the **Creating Edge Prescribed Displacement Element** dialog box will be displayed. However, the options available in the dialog boxes are same.*

Type Area

The **Translation** radio button of the **Type** area in the dialog box is used to allow the translation movement upto the specified magnitude in a specified direction. To allow the rotational movement of the selected geometry upto the specified magnitude about an axis, select the **Rotation** radio button from the **Type** area of the dialog box.

Note

The rotational movement can only be possible for the elements that have rotational degrees of freedom such as shell and beam elements.

Magnitude Area

The **Magnitude** area of the dialog box is used to specify the magnitude for the translation or rotational movements.

Direction Area

The **Direction** area of the dialog box is used to select the required direction vector along or about which you want to allow the translational or rotational movements. You can select the **X**, **Y**, or **Z** radio button to allow the translational or rotational movement along or about X, Y, or Z axis. You can also specify the direction vector other than along the X, Y, and Z axes by using the **Custom** radio button and the **Vector Selector** button of the dialog box, as discussed earlier.

Stiffness Area

The **Stiffness** edit box is used to specify the stiffness of the elements to allow the movement in the specified direction. Note that the translational or rotational movement of the nodes of the model will also depend upon the stiffness value specified in this edit box. The higher the stiffness value specified, the greater will be the displacement.

Description Area

The **Description** area of the dialog box is used to enter a comment or description about the applied prescribed displacement.

Similarly, you can select nodes or edges to apply prescribed displacement. Figure 8-30 shows a model with prescribed displacement applied on the curved surface.

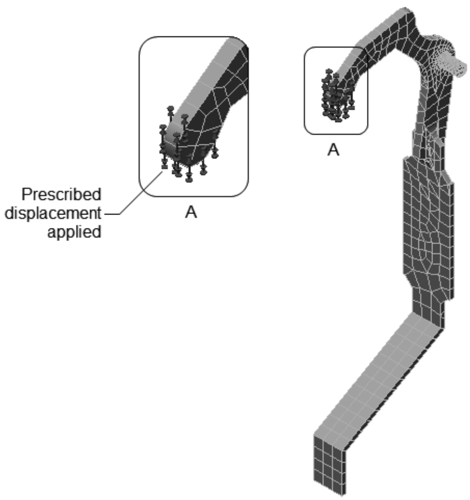


Figure 8-30 *The prescribed displacement applied on the curved surface*

Frictionless Constraint

Ribbon:	Setup > Constraints > Frictionless

The frictionless constraint is used to restrict the deformation of the selected surface along its normal direction due to the force applied on the model. You can select planar and cylindrical surfaces of a model to apply frictionless constraint by using the **Frictionless** tool.

To apply frictionless constraint to a surface of the model, select a surface and then choose the **Frictionless** tool from the expanded **Constraints** panel of the **Setup** tab in the **Ribbon**; the **Creating Frictionless Constraint Object** dialog box will be displayed, refer to Figure 8-31. In this dialog box, you can enter a comment or description about the frictionless constraint being applied in the **Description** edit box. Next, choose the **OK** button; the frictionless constraint will be applied to the selected surface.

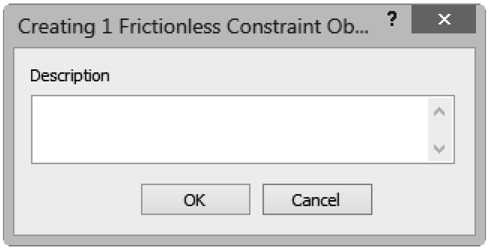


Figure 8-31 *The **Creating Frictionless Constraint Object** dialog box*

Figure 8-32 shows a model with frictionless constraint applied to the bottom planar surface of a structure member in FEA editor environment and Figure 8-33 shows the model in the Result environment of Autodesk Simulation Mechanical after performing the analysis. Note that the frictionless constraint does not allow the selected surface to deform along its normal direction, refer to Figure 8-33.

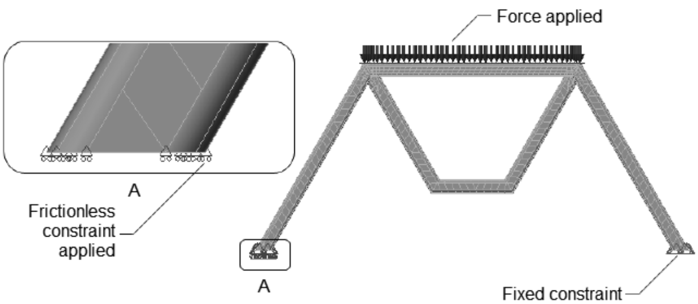


Figure 8-32 *The frictionless constraint applied to the bottom planar surface of a structure member*

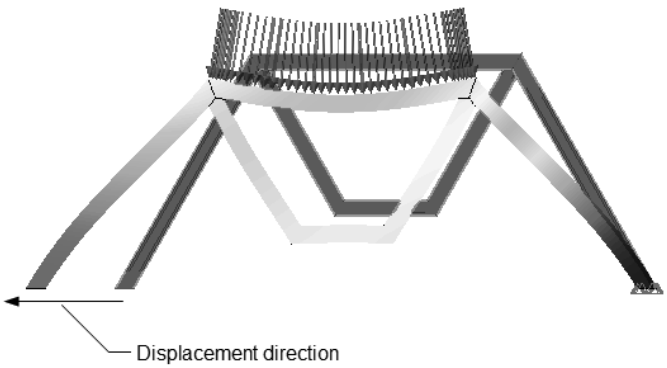


Figure 8-33 The model with displacement along the parallel direction after applying the frictionless constraint

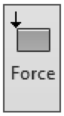
Note
You will learn more about Result environment of Autodesk Simulation Mechanical in later chapters.

Loads

Load is defined as the external force acting on the body. It includes forces, pressures, movement, and so on. The tools that are used for applying different loading conditions are discussed next.

Force

Ribbon:	Setup > Loads > Force



A force is an influence that causes an object to undergo a certain change, either concerning its movement, direction, or geometrical construction. In Autodesk Simulation Mechanical, you can apply force to a node, edge, and surface.

Applying Nodal Force

The force which is applied on a node is called as nodal force. To apply nodal force, select the node of the model on which you want to apply the force and then choose the **Force** tool from the **Loads** panel of the **Setup** tab in the **Ribbon**; the **Creating Nodal Force Object** dialog box will be displayed, as shown in Figure 8-34. Alternatively, select a node and right-click to display a shortcut menu. Next, select **Add > Nodal Force** from the shortcut menu displayed, refer to Figure 8-35; the **Creating Nodal Force Object** dialog box will be displayed.

Note
*To select a node of a model, click on the **Selection** tab in the **Ribbon**. Next, choose the **Point or Rectangle** tool from the **Shape** panel and the **Vertices** tool from the **Select** panel of the **Selection** tab in the **Ribbon**.*

Tip. *If you invoke the* **Force** *tool to apply force while multiple nodes are selected, the name of the dialog box displayed will be* ***Creating (number of nodes selected) Nodal Force Objects****.*

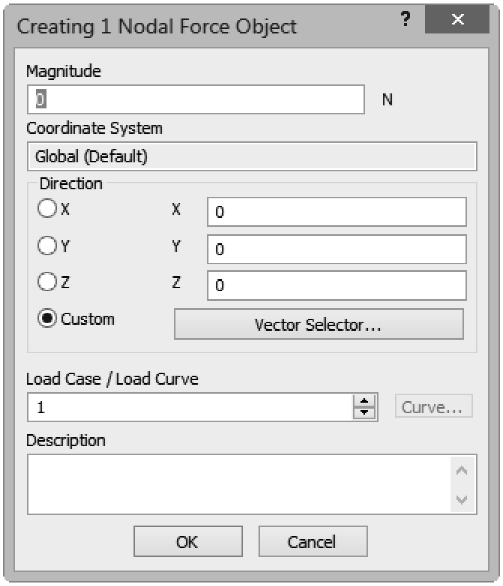


Figure 8-34 *The* ***Creating Nodal Force Object*** *dialog box*

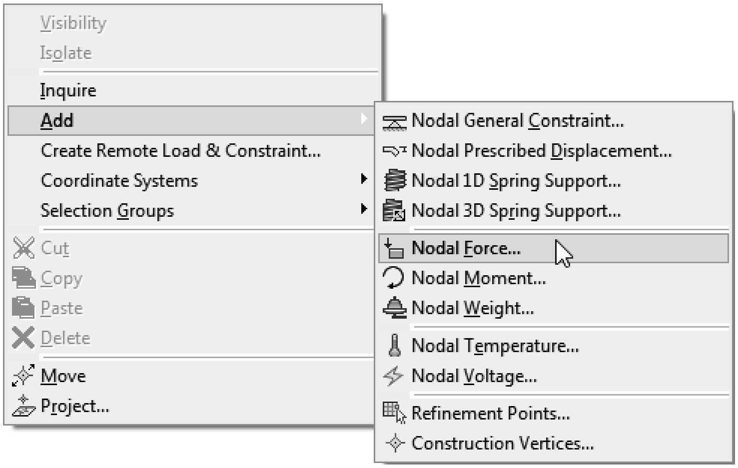


Figure 8-35 *Choosing the* ***Nodal Force*** *option from the shortcut menu*

The options available in the **Creating Nodal Force Object** dialog box are discussed next.

Magnitude Edit Box

The **Magnitude** edit box is used to specify the magnitude of the force to be applied on the selected object.

Direction Area

The **Direction** area of the dialog box is used to specify the direction of force. You can specify the direction of force normal to the X, Y, or Z direction by selecting the respective radio button from this area. For example, on selecting the **X** radio button, the direction of force will be toward the X direction with respect to the coordinate system displayed in the **Coordinate System** display area of the dialog box. The Global coordinate system is the default coordinate system.

You can also specify the direction of force other than the X, Y, and Z directions by using the **Custom** radio button. To do so, select the **Custom** radio button; the **X**, **Y**, and **Z** edit boxes of the dialog box will be enabled. In these edit boxes, you can specify the coordinates of the force direction.

Moreover, you can also specify the direction of force by specifying the direction vector. To do so, choose the **Vector Selector** button in the dialog box; the dialog box will disappear and you will be prompted to specify the direction vector for the force to be applied. Specify the start point and then the endpoint toward which the direction should point, refer to Figure 8-36. As soon as you specify the endpoint for the direction vector, the **Creating Nodal Force Object** dialog box will be displayed again with the coordinate value of the specified direction vector entered in the **X**, **Y**, and **Z** edit boxes.

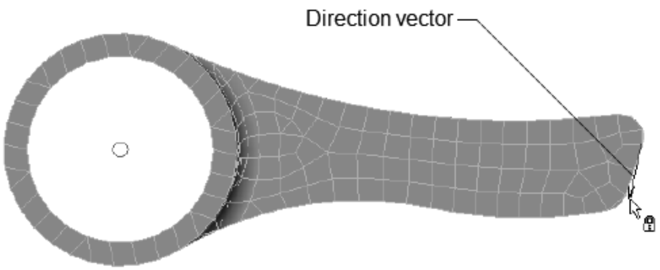


Figure 8-36 *The direction vector for specifying the direction of force in a model*

Load Case/Load Curve Spinner

The **Load Case/Load Curve** spinner of the dialog box is used to select the required load case/load curve to associate with the force being applied. You will learn more about load case and load curve later in this chapter.

Note

*The **Load Case/Load Curve** spinner will be available only while applying nodal and edge forces in their respective dialog boxes.*

*Also the **Curve** button, which is available on the right of the **Load Case / Load Curve** spinner in the dialog box, will be enabled only at the time of performing MES with nonlinear material analysis and the static stress with nonlinear material analysis. This button is used to define the load curve to be followed by the force being applied.*

Description

The **Description** area of the dialog box is used to enter a comment or description about the force being applied.

After specifying all the parameters for applying the nodal force, choose the **OK** button from the dialog box; the force will be applied on the selected node and an arrow representing the force will be displayed in the drawing area.

Figure 8-37 shows a model with nodal force applied along the direction vector shown in Figure 8-36.

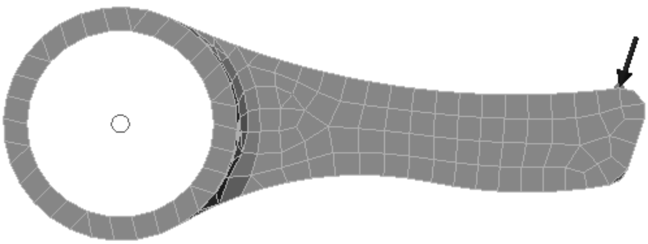


Figure 8-37 *Arrow representing the applied nodal force*

Note

You can select multiple nodes to apply force by using the CTRL key. However, in that case, the total magnitude of the force applied on the model will be equal to the number of nodes selected multiplied by the value of magnitude specified. For example, if you select 4 nodes to apply force of 100 N, then the total magnitude of the force to be applied on the model will be 400 N (4 X 100).

Applying Edge Force

The force applied on an edge is called edge force. To apply the edge force, select the edge of the model on which you want to apply the force and then choose the **Force** tool from the **Loads** panel of the **Setup** tab in the **Ribbon**; the **Creating Edge Force Object** dialog box will be displayed, as shown in Figure 8-38. Alternatively, select the edge and right-click to display a shortcut menu. Next, select **Add > Edge Force** from the shortcut menu to invoke this dialog box. The options available in the **Creating Edge Force Object** dialog box are same as discussed earlier.

Note

*To select an edge of a model, click on the **Selection** tab in the **Ribbon**. Next, choose the **Point or Rectangle** tool from the **Shape** panel and the **Edges** tool from the **Select** panel of the **Selection** tab in the **Ribbon**.*

Applying Surface Force

The force applied on a surface is called surface force. To apply the surface force, select the surface of the model on which you want to apply the force and then choose the **Force** tool from the **Loads** panel of the **Setup** tab in the **Ribbon**; the **Creating Surface Force Object** dialog box will be displayed, as shown in Figure 8-39. Alternatively, select the surface and right-click to display a shortcut menu. Next, select **Add > Surface Force** from the shortcut menu to invoke this dialog box.

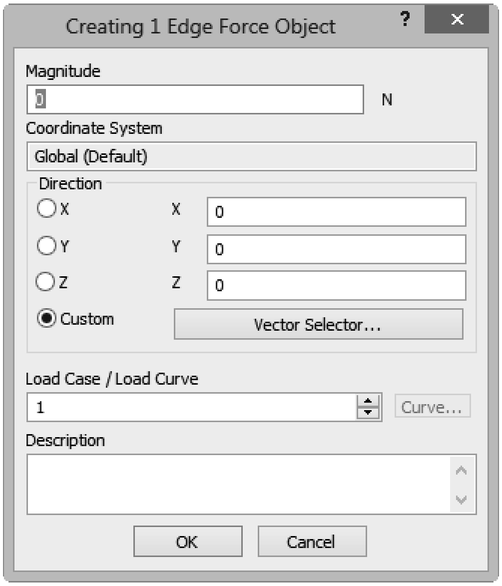


*Figure 8-38 The **Creating Edge Force Object** dialog box*

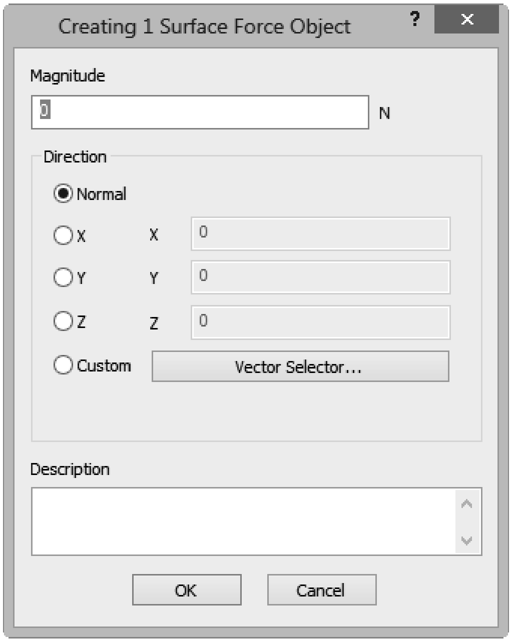


*Figure 8-39 The **Creating Surface Force Object** dialog box*

The options available in the **Creating Surface Force Object** dialog box are same as in the **Creating Nodal Force Object** and **Creating Edge Force Object** dialog boxes except the **Normal** radio button.

Note

To select a surface of a model, click on the ***Selection*** *tab in the* ***Ribbon****. Next, choose the* ***Point or Rectangle*** *tool from the* ***Shape*** *panel and the* ***Surface*** *tool from the* ***Select*** *panel of the* ***Selection*** *tab in the* ***Ribbon****.*

On selecting the **Normal** radio button available in the **Direction** area of the **Creating Surface Force Object** dialog box, you can specify the direction of force normal to the selected surface. Note that after applying the force normal to a surface, the direction arrows representing the direction of force may not be displayed normal to the surface in the FEA Editor environment. However, in the Result environment, the arrows representing the direction of force will be displayed normal to the applied surface.

Figure 8-40 shows the direction of arrows in the FEA Editor environment on applying the surface force normal to the curved surface of the model. Figure 8-41 shows the same model in Result environment with the direction of arrows normal to the curved surface.

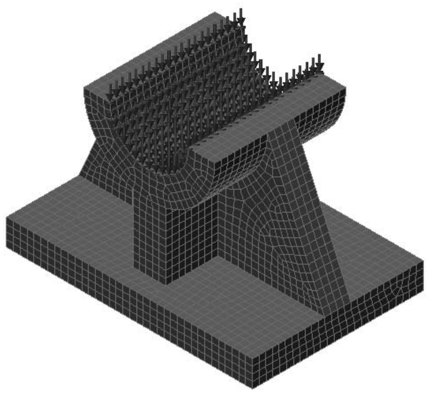


Figure 8-40 *Direction of arrows displayed in the FEA Editor environment on applying surface force normal to the curved surface of the model*

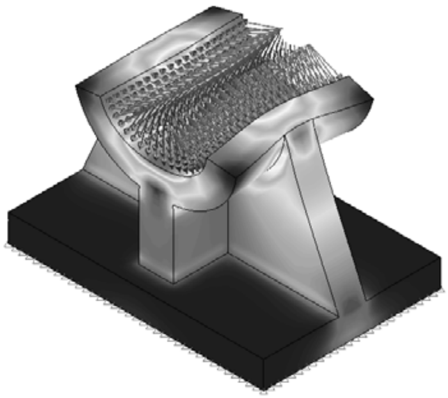


Figure 8-41 *Direction of arrows displayed in the Result environment on applying surface force normal to the curved surface of the model*

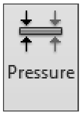
Note

You will learn more about the Result environment of Autodesk Simulation Mechanical in later chapter.

Tip. *You can select nodes, surfaces, or edges for applying force either before or after invoking the dialog box.*

Pressure

Ribbon: Setup > Loads > Pressure



Pressure is defined as the force per unit area. In Autodesk Simulation Mechanical, you can apply normal/traction pressure to the selected surface by using the options available in the **Creating Surface Pressure/Traction Object** dialog box. A normal pressure refers to the pressure that is applied normal to the selected surface whereas a traction pressure is applied in a specific direction.

To apply normal/traction surface pressure, select the surface of the model on which you want to apply pressure and then choose the **Pressure** tool from the **Loads** panel of the **Setup** tab in the **Ribbon**; the **Creating Surface Pressure/Traction Object** dialog box will be displayed, as shown in Figure 8-42. Alternatively, select the surface and right-click to display a shortcut menu. Next, select **Add > Surface Pressure/Traction** from the shortcut menu to invoke this dialog box.

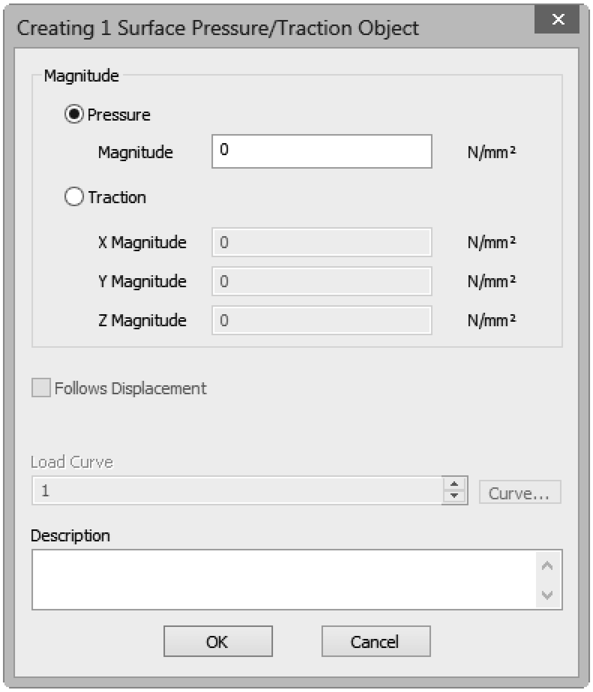


***Figure 8-42** The **Creating Surface Pressure/Traction Object** dialog box*

The options available in the **Creating Surface Pressure/Traction Object** dialog box are discussed next.

Pressure Radio Button

The **Pressure** radio button of the dialog box is used to apply pressure normal to the selected surface. To apply the normal pressure, select the **Pressure** radio button and then enter the magnitude of the pressure in the **Magnitude** edit box available below this radio button.

Traction Radio Button

The **Traction** radio button of the dialog box is used to apply a pressure that is oriented toward specific direction. To apply the traction pressure, select the **Traction** radio button; the **X Magnitude**, **Y Magnitude**, and **Z Magnitude** edit boxes will be enabled. In these edit boxes, you can specify the magnitude of the pressure in each of the global directions.

Multiplier Edit Box

The **Multiplier** edit box of the dialog box is used to enter the multiplier value for the load curve selected in the **Load Curve** field of the dialog box. This edit box will be enabled while performing MES with nonlinear material analysis and static stress with nonlinear material analysis.

Load Curve Spinner

The **Load Curve** spinner of the dialog box is used to set the load case to be used for the pressure being applied. After setting the load case to be used, you can modify the load curve of the respective load case by using the **Curve** button available next to the **Load Curve** spinner. On choosing the **Curve** button; the **Multiplier Table Editor** dialog box will be displayed, as shown in Figure 8-43.

Note

*The **Load Curve** spinner and the **Curve** button of the **Creating Surface Pressure/Traction Object** dialog box will be enabled only while performing MES with nonlinear material analysis and the static stress with nonlinear material analysis.*

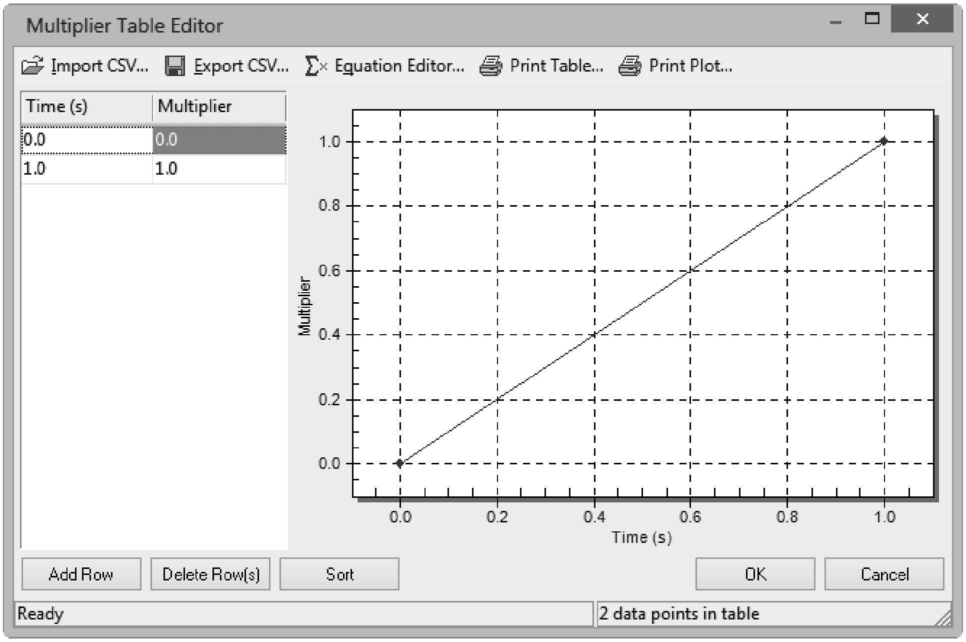


*Figure 8-43 The **Multiplier Table Editor** dialog box*

In the **Multiplier Table Editor** dialog box, the load curve is defined as the relationship between the time and multiplier. By default, two data points are defined for the load curve. You can define multiple data points for the load curve. To do so, choose the **Add Row** button from the dialog box; a new row will be added to the left of the dialog box with the default time and multiplier values. Also, a new point will be added in the load curve with the default values. To specify new values for time and multiplier in a particular row, click on the field corresponding to the **Time (s)** / **Multiplier** column of that row to activate its exit mode and then enter the required value in it. Next, press ENTER. If you want to delete a row, select the row to be deleted from the left side of the dialog box and then choose the **Delete Row(s)** button from the **Multiplier Table Editor** dialog box.

You can import the data of an existing load curve which is saved in the .csv file format, using the **Import CSV** button of the dialog box. Similarly, you can export the current load curve data to a .csv file format, using the **Export CSV** button of the dialog box. You can also print the table containing the time and multiplier values of the data points by using the **Print Table** and **Print Plot** buttons in the dialog box. After specifying all the data points for defining the load curve, exit from the dialog box by choosing the **OK** button.

Description Area

The **Description** area of the **Creating Surface Pressure/Traction Object** dialog box is used to enter a comment or description about the pressure being applied.

Figure 8-44 shows a mesh model with pressure applied on its all the inner surfaces. Figure 8-45 shows the same model after performing analysis with normal pressure applied and Figure 8-46 shows the model with traction pressure of same magnitude applied towards the Y axis direction.

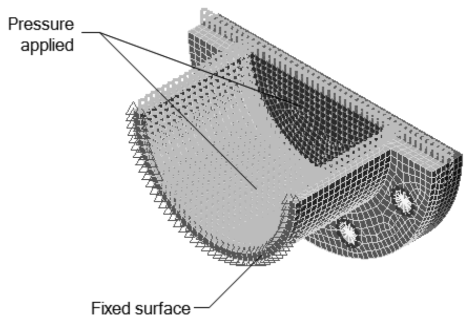


Figure 8-44 *The pressure applied on the inner surfaces of the model*

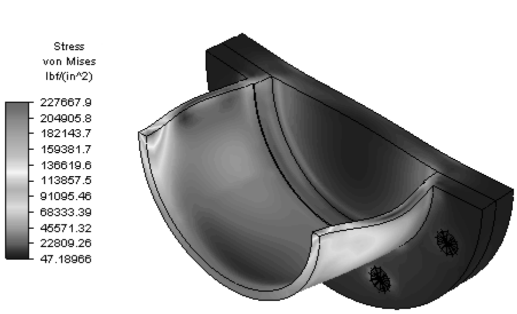


Figure 8-45 Model after performing analysis with normal pressure applied on all its inner surfaces

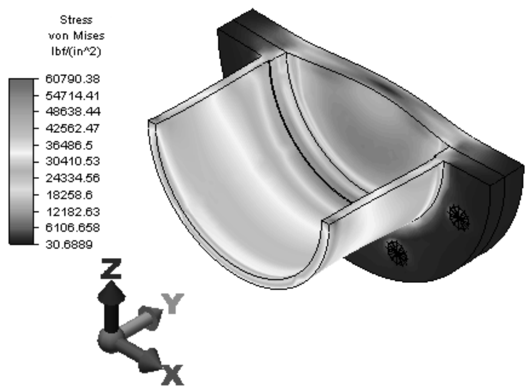


Figure 8-46 *Model after performing analysis with traction pressure applied in the direction of Y axis*

Variable Pressure

Ribbon:	Setup > Loads > Variable Pressure

The variable pressure is applied by specifying different pressures at uniform intervals along the direction of the selected surface. To apply variable pressure to a surface, select the surface of the model on which you want to apply variable pressure. Next, choose the **Variable Pressure** tool from the expanded **Loads** panel of the **Setup** tab in the **Ribbon**, refer to Figure 8-47; the **Creating Surface Variable Pressure Object** dialog box will be displayed, as shown in Figure 8-48.

Alternatively, select the surface and right-click to display a shortcut menu. Next, select **Add > Surface Variable Pressure** from the shortcut menu displayed to invoke the **Creating Surface Variable Pressure Object** dialog box. The options available in the **Creating Surface Variable Pressure Object** dialog box are discussed next.

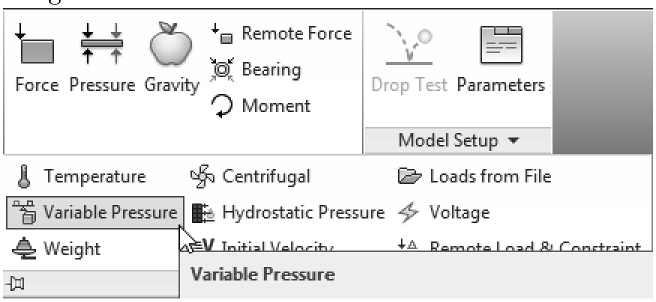


Figure 8-47 *The* ***Variable Pressure*** *tool in the expanded* ***Loads*** *panel*

Coordinate system

The **Coordinate system** drop-down list displays the list of all the coordinate systems defined in the current FEA Editor environment in addition to the global coordinate system. You can select the required coordinate system from this drop-down list to be followed by the variable loads. Note that if you want the starting magnitude of the variable pressure to be zero and starts from the starting node of an edge of the surface then you need to create a coordinate system that has the origin at the starting node of the edge.

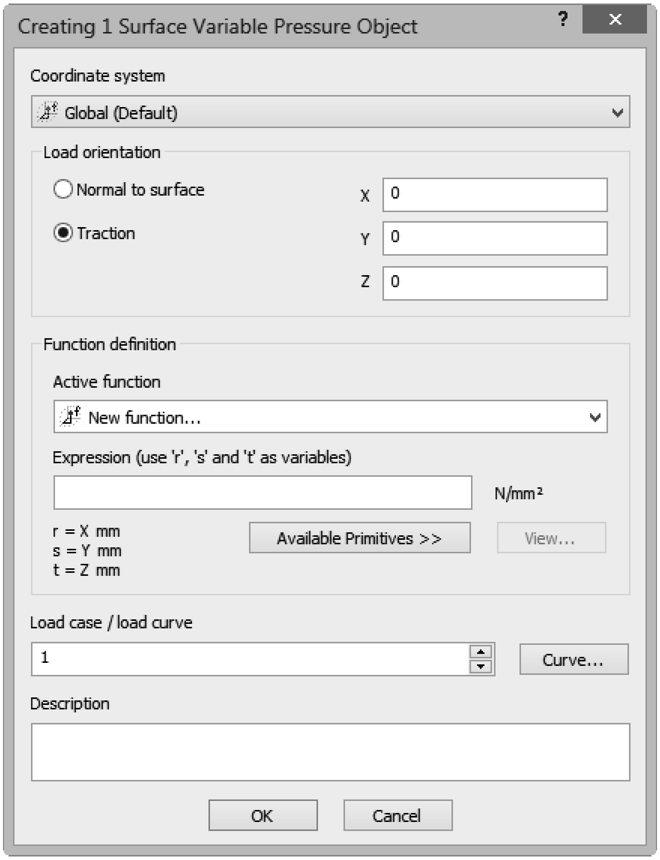


***Figure 8-48** The **Creating Surface Variable Pressure Object** dialog box*

Load orientation Area

The options in the **Load orientation** area are used to specify the direction of pressure. On selecting the **Normal to surface** radio button, the direction of pressure being applied will be perpendicular to the surface selected for applying the pressure. On selecting the **Traction** radio button, the **X**, **Y**, and **Z** edit boxes on the right of this radio button will be enabled. By using these edit boxes, you can specify the direction vector for the variable pressure to be followed.

Function definition Area

The options in the **Function definition** area are used to specify a function that defines the variable load. The **Active function** drop-down list of this area displays the list of all the existing functions that are already specified in the current session of Autodesk Simulation

Mechanical. From this drop-down list, you can select an existing function to be used for the variable load being applied. By default, the **New Function** option is selected in this drop-down list. You can also enter a new name in the edit field of the **Active function** drop-down list to specify a new function for the variable load.

The **Expression** field in the **Function definition** area is used to specify the expression for the function selected in the **Active function** drop-down list. You can write the equation in the **Expression** field for representing the pressure to be applied to the selected surface. You can use the variables r, s, and t in the expression, where r represents the distance in the X direction, s represents the distance in the Y direction, and t represents the distance in the Z direction. Also, you can use the basic operators such as +, -, * ,/, (,), and ^ in the expression. For example, if **25*(r)** is the expression entered in the **Expression** field and the unit system is set to **English (in)**, then the pressure at the distance of 50 inches along the X direction is equal to 1250 psi (25 x 50). In other words, as per the expression **25*(r)**, the pressure 25 psi is increasing per inch distance toward the X direction. If the **25*(r ^ 2)** will be the expression entered in the **Expression** field, then the pressure at the distance of 50 inches along the X direction will be equal to 62500 psi [25 x (50x50)]. Similarly, 250000 psi pressure at the distance of 100 inches along the X direction.

You can also use some commonly used functions for defining the variable pressure. To do so, click on the **Available Primitives** button of the dialog box; a flyout will be displayed. You can select the required functions from the flyout displayed.

To view the graphical representation of the variable pressure as per the expression entered in the **Expression** field along a specific direction, choose the **View** button from the dialog box. As soon as you choose the **View** button, the **Variable Load Viewer** window will be displayed, refer to Figure 8-49.

Load case/load curve Spinner

The **Load case/load curve** spinner is used to set the load case to be used for the pressure being applied. After setting the load case to be used, you can modify the load curve of the respective load case by using the **Curve** button available next to the **Load Curve** spinner. On choosing the **Curve** button; the **Multiplier Table Editor** dialog box will be displayed. The options in this dialog box area are same as discussed earlier.

Note

*The **Load case/load curve** spinner and the **Curve** button of the **Surface Variable Load Object** dialog box will be enabled only in case of performing MES with nonlinear material analysis and the static stress with nonlinear material analysis.*

Description Area

The **Description** area of the dialog box is used to enter a comment or description about the pressure being applied.

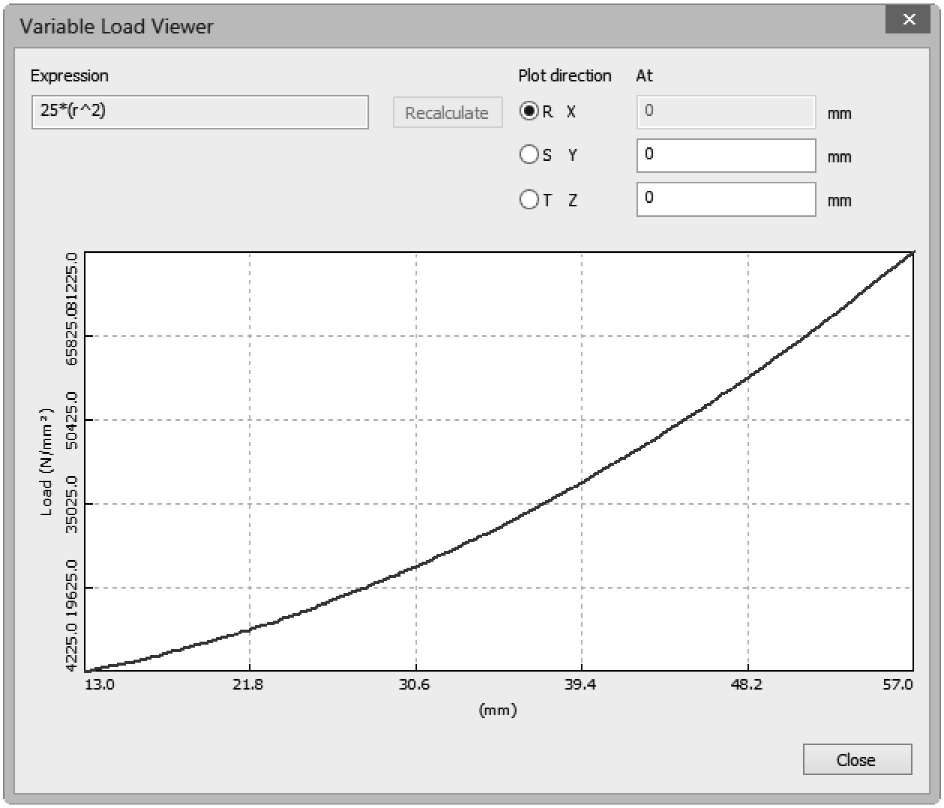


Figure 8-49 *The **Variable Load Viewer** dialog box*

Figure 8-50 shows the front view of a model in the Result environment with variable pressure that varies along the x axis by using the expression **25*(r^2)**. Note that the origin of the coordinate system used for the variable pressure applied in the model shown in Figure 8-50 is at the starting node of the left side of the top edge. You will learn more about the Result environment in the later chapter.

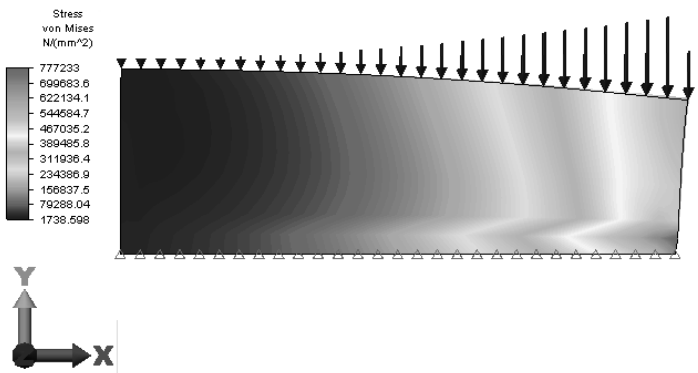


Figure 8-50 *Variable pressure applied on the top surface of the model*

Hydrostatic Pressure

Ribbon: Setup > Loads > Hydrostatic Pressure

Hydrostatic pressure is the pressure that is exerted by a fluid at equilibrium state. It varies linearly from the level of the fluid in the direction of increasing depth of the fluid. The magnitude of the hydrostatic pressure is directly proportional to the depth of the fluid and to the density of the fluid.

To apply hydrostatic pressure, select the surface of the model. You can also select multiple surfaces by pressing the CTRL key. After selecting the surface, choose the **Hydrostatic Pressure** tool from the expanded **Loads** panel of the **Setup** tab in the **Ribbon**; the **Creating Surface Hydrostatic Pressure Object** dialog box will be displayed, refer to Figure 8-51. Alternatively, select the surface and right-click to display a shortcut menu. Next, select **Add > Surface Hydrostatic Pressure** from the shortcut menu displayed to invoke this dialog box. The options available in this dialog box are discussed next.

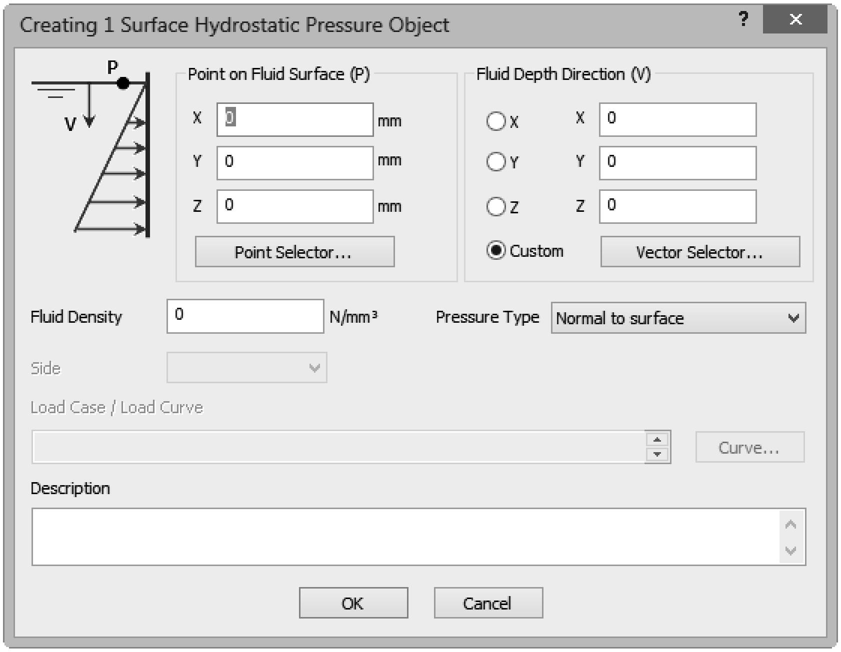


*Figure 8-51 The **Creating Surface Hydrostatic Pressure Objects** dialog box*

Point on Fluid Surface (P) Area

The **Point on Fluid Surface (P)** area of this dialog box is used to specify the X, Y, and Z coordinates of the point that is on the top of the fluid surface for creating the hydrostatic pressure. You can specify these coordinates in their respective **X**, **Y**, and **Z** edit boxes. Note that only the elements below this point will be affected by the magnitude of the pressure. The magnitude of the pressure on the elements of surfaces below this point will vary and will depend upon the distance from the point on the fluid surface and the density of fluid. The distance from the point on the top of the fluid surface will be multiplied by the value of the density of the fluid to calculate the magnitude of pressure on an element.

By using the **Point Selector** button in this area of the dialog box, you can also select a point directly from the drawing area as the top point on the fluid surface.

Fluid Depth Direction (V) Area

The **Fluid Depth Direction (V)** area of the dialog box is used to specify the direction vector for the depth of fluid. You can specify the direction vector normal to the X, Y, or Z direction by selecting the respective radio button from this area. For example, on selecting the **X** radio button, the direction of fluid depth will be toward the X direction with respect to the global coordinate system.

You can also specify the direction of fluid depth other than the X, Y, and Z directions by using the **Custom** radio button. To do so, select the **Custom** radio button; the **X**, **Y**, and **Z** edit boxes of the dialog box will be enabled. In these edit boxes, you can specify the X, Y, and Z coordinates of the direction vector.

In addition to this, you can also specify the direction of fluid depth by selecting two points in the drawing area. To do so, choose the **Vector Selector** button available on the right of the **Custom** radio button in the dialog box; the dialog box will disappear. Now, you can specify direction vector by selecting two points. Specify the start point and then the end points toward which the direction will point. As soon as you specify the endpoint for the direction vector, the dialog box will be displayed again with the coordinate values of the direction vector entered in the **X**, **Y**, and **Z** edit boxes.

Fluid Density Edit Box

The **Fluid Density** edit box is used to specify the density of the fluid which will create the hydrostatic pressure.

Pressure Type Drop-down List

The **Pressure Type** drop-down list is used to select the way the hydrostatic pressure will be applied on the selected surface. By default, the **Normal to surface** option is selected in this drop-down list. As a result, the direction of the magnitude applied will be normal to the selected surfaces.

On selecting the **Full pressure in horizontal** option from this drop-down list, the magnitude of the hydrostatic pressure will be applied only in the horizontal direction. In other words, no pressure will be applied parallel to the direction vector of the fluid depth.

Figure 8-52 shows the representation of hydrostatic pressure applied on selecting the **Normal to surface** option from the **Pressure Type** drop-down list. Figure 8-53 shows the hydrostatic pressure applied horizontally by selecting the **Full pressure in horizontal** option.

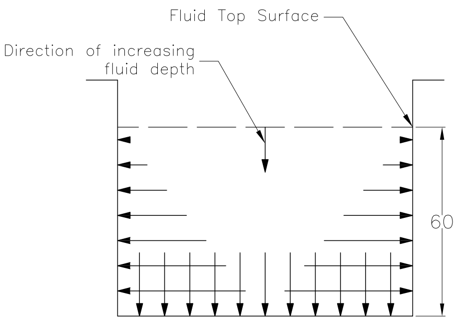


*Figure 8-52 Hydrostatic pressure applied on surfaces with the **Normal to surface** option selected*

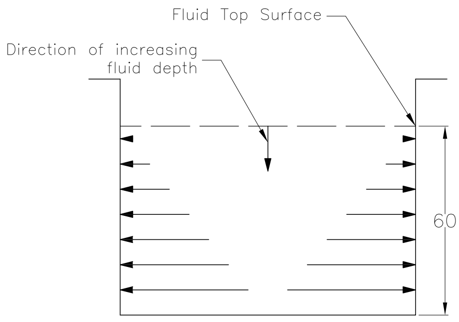


*Figure 8-53 Hydrostatic pressure applied on surfaces with the **Full pressure in horizontal** option selected*

If you select the **Horizontal component only** option from the **Pressure Type** drop-down list, the hydrostatic pressure will be applied horizontally only on the horizontal components of the selected surfaces.

Note

*Note that if you select the **Horizontal component only** option, the magnitude of the hydrostatic pressure will be equal to the fluid density multiplied by the depth of fluid from the top surface and sin(θ).*

Where θ is the angle between the normal to the surface selected and the normal to the direction vector of the fluid depth.

After specifying all the parameters for the hydrostatic pressure, choose the **OK** button from the dialog box.

Remote Load & Constraint

Ribbon: Setup > Loads > Remote Load & Constraint

The **Remote Load & Constraint** tool is used to apply nodal load and boundary condition that originates from a point which is located in the space, not on the model. In other words, the load that originates from a point located in the space and is transmitted to the model through line elements such as beam, truss, or similar line elements is called as remote load. You can apply a remote load similar to applying force on the model, with the only difference that the location of the remote load origin can be anywhere in the space.

You can select vertices, lines, edges, surfaces, and parts for applying remote load. Regardless of the element selected, the remote load will be applied to the vertices of the selection. To apply remote load, choose the **Remote Load & Constraint** tool from the expanded **Loads** panel of the **Setup** tab in the **Ribbon**; the **Create Remote Load** dialog box will be displayed, as shown in Figure 8-54. The options in this dialog box are discussed next.

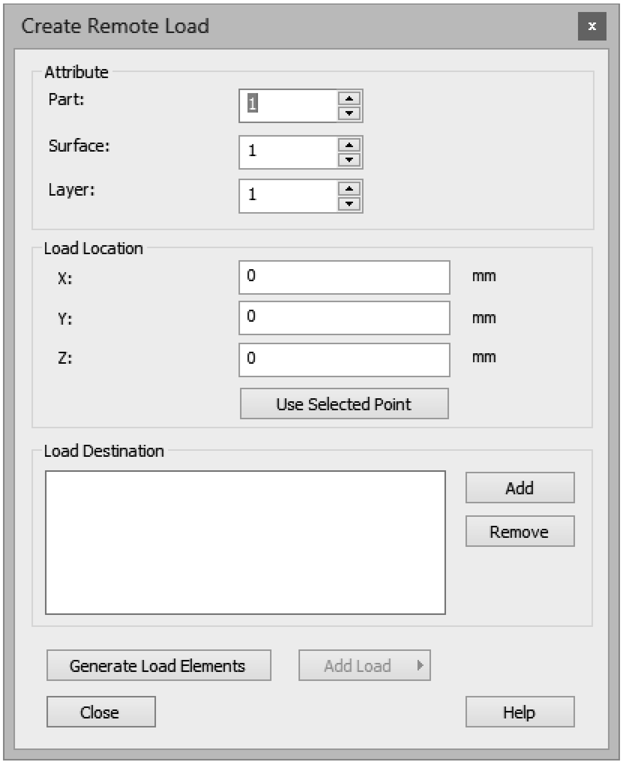


***Figure 8-54** The **Create Remote Load** dialog box*

Attribute Area

The **Part**, **Surface**, and **Layer** spinners of the **Attribute** area are used to specify the part, surface, and layer attributes for the load elements. Make sure that the part number specified in the **Part** spinner has not already been specified to other part of the model.

Load Location Area

The **Load Location** area is used to specify the location from where the remote load will originate. You can specify the X, Y, and Z coordinates of the point in their respective **X**, **Y**, and **Z** edit boxes of the **Load Location** area. Note that this point will act as the point of originating remote load. Alternatively, you can select a vertex or construction vertex from the graphic area and choose the **Use Selected Point** button from the **Load Location** area of the dialog box; the coordinates of the selected point will be entered in the **X**, **Y**, and **Z** edit boxes of the dialog box.

Load Destination Area

The **Load Destination** area is used to specify the destination for the remote load. To specify the destination for the remote load, select the destination for it from the graphic area. The destination can be vertices, lines, edges, surfaces, or parts. Next, choose the **Add** button from the **Load Destination** area; the selected geometry will be selected as the destination for the remote load. Also, the name of the selected geometry will be displayed in the selection area of the **Load Destination** area of the dialog box. Similarly, by using the **Add** button, you can also add multiple geometries as destinations for the remote load. Note that all the vertices of the selections will be calculated for distributing the remote load.

You can also remove the selected geometry from the list of geometries added in the **Load Destination** area. To do so, select the geometry to be removed from the selection area of the **Load Destination** area and then choose the **Remove** button; the selected geometry will be removed.

Note

*To select a vertex, line, edge, surface, or part, click on the **Selection** tab in the **Ribbon**. Next, choose the **Point** tool from the **Shape** panel and then the **Vertices**, **Lines**, **Edges**, **Surfaces**, or **Parts** tool from the **Select** panel of the **Selection** tab in the **Ribbon**.*

Generate Load Elements Button

The **Generate Load Elements** button of the dialog box is used to generate the load elements that transmit the load from its originating location to the nodes of the model. As soon as you choose this button, a new geometry and a node at the specified point in the space will be created. Also, the lines representing the load elements from the remote load location to the model will be displayed in the graphic area, refer to Figure 8-55.

Note

*The new geometry created on choosing the **Generate Load Elements** button will also be displayed in the **Tree View**. Therefore, you need to define its element type, element definition, and the material to continue with the analysis process.*

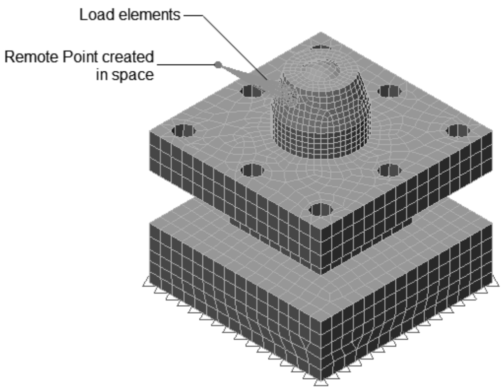


Figure 8-55 *The remote load applied on the model*

Add Load Button

The **Add Load** button of the dialog box is used to select the type of load or the boundary conditions to be applied at the remote load location. Note that this button will be enabled only after the load elements are generated.

To select the type of load or the boundary condition to be applied, click on the side arrow available on the **Add Load** button; a flyout will be displayed, as shown in Figure 8-56. You can select the required type of remote load or the boundary conditions from this flyout. Note that depending upon the type of remote load or boundary condition you select from the flyout, the dialog box for specifying the parameters will be displayed.

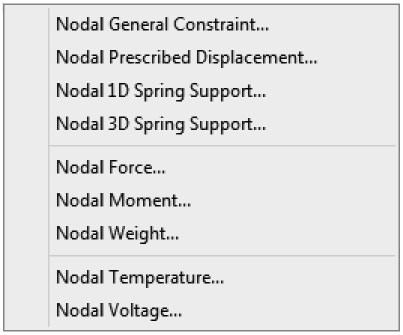


Figure 8-56 *The flyout displayed on clicking on the arrow on the **Add Load** button*

After defining all the parameters for creating the remote load, exit from the **Create Remote Load** dialog box by choosing the **Close** button. The new part with the specified part number will be displayed in the **Tree View** in red, refer to Figure 8-57. Also, the graphic representation of the line elements transforming the load from the point in space to the model, refer to the Figure 8-58.

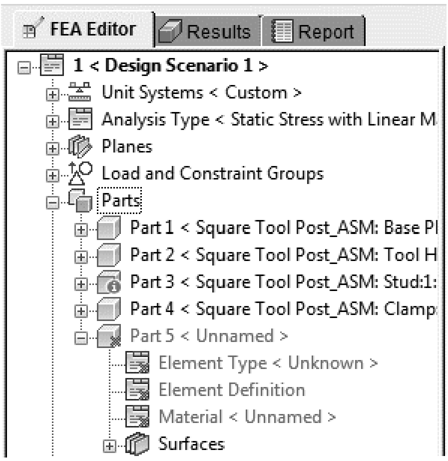


Figure 8-57 *The new part displayed in the **Tree View** in red*

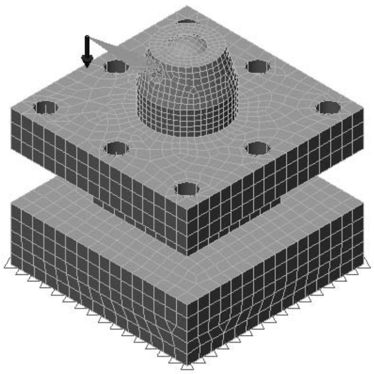


Figure 8-58 *The graphical representation of the remote load applied on the model*

Now, you need to define the element type, element definition, and the material for the newly created part in order to continue with the analysis process.

To define the element type, select the **Element Type** sub-node under the newly created part node in the **Tree View** and right-click on it; a shortcut menu will be displayed with the options of different element types, refer to Figure 8-59. You can select the required element type from the shortcut menu displayed.

***Figure 8-59** A shortcut menu displayed with different element type options*

Note

If you have applied moment as the remote load, then you need to select the beam elements because the moment can be transmitted only through the beam elements. The other elements such as truss, gap, and so on do not have rotational degrees of freedom and cannot transmit moments and torques.

Similarly, you can define the element definition and the material for the line elements that define the remote load.

Remote Force

Ribbon:	Setup > Loads > Remote Force

The **Remote Force** tool is used to apply force that originates from a point which is located in the space, not on the model. You can apply remote force similar to applying a force on to the model with the only difference that the location of the remote force origin can be anywhere in the space.

You can select surfaces of the model for applying remote force. Note that regardless to the surface selected, the remote force will be distributed to all its nodes. To apply remote force, select the surface on which you want to simulate the effects of remote force and then choose the **Remote Force** tool from the **Loads** panel of the **Setup** tab in the **Ribbon**; the **Creating Surface Remote Force** dialog box will be displayed, as shown in Figure 8-60. The options in this dialog box are discussed next.

Magnitude Edit Box

The **Magnitude** edit box of the dialog box is used to specify the magnitude of the remote force.

Remote Point Area

The **Remote Point** area of the dialog box is used to specify the location from where the remote force will be applied. You can specify a point for originating remote force by specifying its X, Y, and Z coordinates in their respective **X**, **Y**, and **Z** edit boxes of the dialog box.

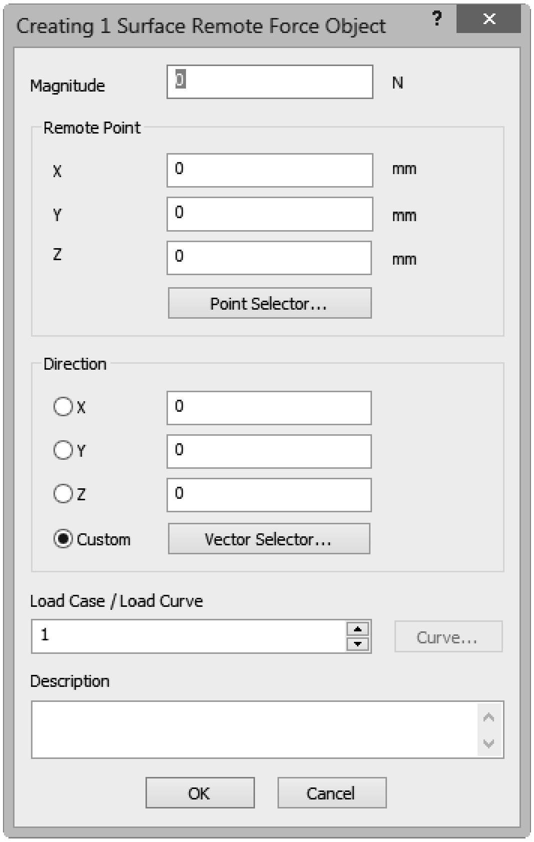


Figure 8-60 *The* ***Create Surface Remote Force*** *dialog box*

Alternatively, you can select a vertex or a construction vertex from the graphic area and choose the **Point Selector** button from this area of the dialog box; the coordinates of the point will be displayed in the **X**, **Y**, and **Z** edit boxes of the dialog box. Also, the point will be selected as the point of origination of the remote force.

Direction Area

The **Direction** area of the dialog box is used to specify the direction of force. You can specify the direction of force along the X, Y, or Z direction by selecting the respective radio button from this area. For example, on selecting the **X** radio button, the direction of force will be toward the X direction with respect to the Global Coordinate System.

You can also specify the direction of force other than the X, Y, or Z directions by using the **Custom** radio button. To do so, select the **Custom** radio button; the **X**, **Y**, and **Z** edit boxes of the dialog box will be enabled. In these exit boxes, you can specify the X, Y, and Z coordinates of the force direction.

In addition to this, you can also specify the direction of force by specifying two points in the graphic area. To do so, choose the **Vector Selector** button available in this area; the dialog box will disappear and you will be prompted to specify the direction vector for the force to be applied. Specify the start point and then the endpoint for defining the direction. As soon as you specify the endpoint for the direction vector, the **Create Surface Remote Force** dialog box will be displayed again. Note that the coordinate values of the specified direction vector are entered in the X, Y, and Z edit boxes.

Load Case/Load Curve Spinner

The **Load Case/Load Curve** spinner is used to select the required load case / load curve to be associated with the force being applied.

Note

*The **Curve** button which is available on the right of the **Load Case/Load Curve** spinner of the dialog box will be enabled only on performing MES with nonlinear material analysis and the static stress with nonlinear material analysis. This **Curve** button is used to define the load curve to be followed by the force.*

Description

The **Description** area of the dialog box is used to enter a comment or description about the force being applied.

After specifying all the parameters for applying the remote force, choose the **OK** button from the dialog box; the remote force will be applied on the selected surfaces and an arrow representing the direction of force will be displayed in the drawing area.

Figure 8-61 shows a model with remote force applied on the circular face of the pulley.

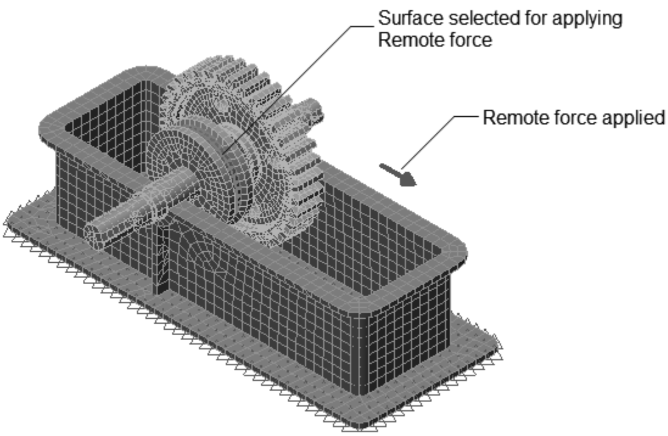


Figure 8-61 *Remote force applied on the model*

Moment

Ribbon:	Setup > Loads > Moment

A moment is a load or force that causes rotation of the nodes to which it is applied in a model. In Autodesk Simulation Mechanical, you can select nodes or surfaces for applying moment along a direction specified by a vector. Note that the nodes connected with the nodes on which moment is applied will also be affected by the applied moment and tend to rotate about the specified vector. The procedure of applying nodal or surface moment is discussed next.

Applying Nodal Moment

To apply nodal moment, select the node of the model and then choose the **Moment** tool from the **Loads** panel of the **Setup** tab in the **Ribbon**; the **Creating Nodal Moment Object** dialog box will be displayed, refer to Figure 8-62. You can also select multiple nodes for applying nodal moment by pressing the CTRL key.

Alternatively, you can also invoke this dialog box by selecting a node of the model and right clicking in the drawing area to display a shortcut menu. Next, select **Add > Nodal Moments** from the shortcut menu displayed.

Note

*On selecting more than one node, the name of the dialog box invoked on choosing the **Moment** tool will be changed to **Creating** (number of nodes selected) **Nodal Moment Objects**.*

You can select nodes for applying moment either before or after invoking the dialog box.

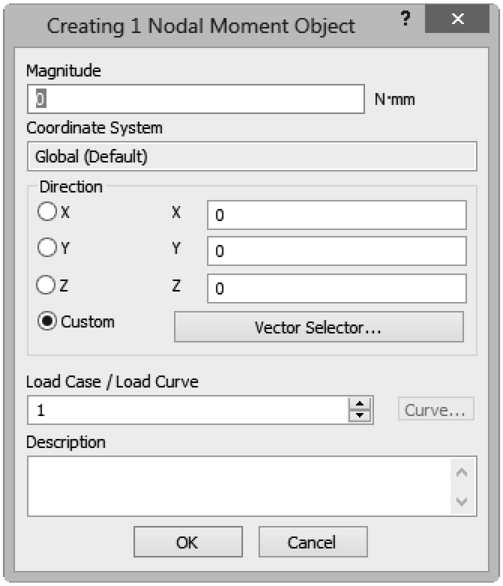


Figure 8-62 *The **Creating Nodal Moment Object** dialog box*

The options available in the dialog box are same as discussed earlier. Specify the parameters of the nodal moment such as magnitude, axis of rotation, and so on in their respective edit boxes. Note that the direction of rotation of moment can be controlled by specifying the positive or negative magnitude value. On specifying the positive magnitude value, the rotation of moment will be in the clockwise direction and on specifying the negative magnitude value, the rotation of moment will be in the counterclockwise direction. After specifying all the parameters, choose the **OK** button. Figure 8-63 shows a meshed model in FEA Editor environment with moment applied on a corner node of top right edge of the model. Figure 8-64 shows the same model in the Result environment with displacement representation that occurred due to the moment applied on the node. Note that the magnitude of the displacement is higher on the node where moment has been applied than the other nodes. In other words, the more the distance travelled by moment, the less will be its effect.

Note

*You can apply nodal moment only on plate and beam elements. Also, if you select multiple nodes to apply moment in a model, the magnitude of the moment will be applied to each selected node. For example, if you select 5 nodes and specify 10 as the magnitude of moment in the model units about the X axis, then the applied magnitude to the model will be 50 (5*10) in the model units.*

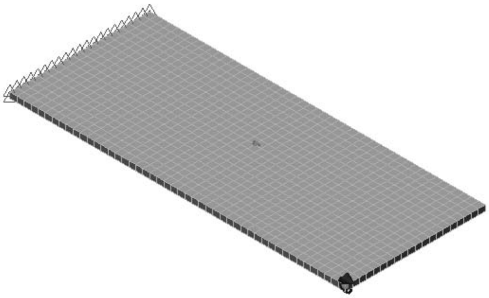


Figure 8-63 *The nodal moment applied on a node of the top right edge of the model*

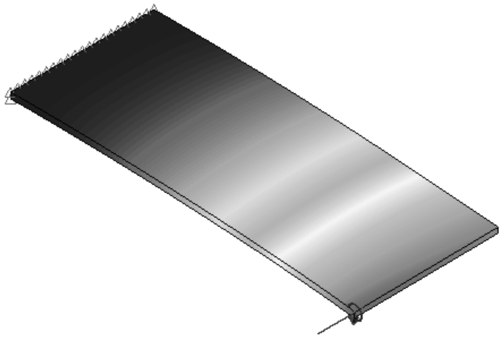


Figure 8-64 *Model in the Result environment after applying moment on a node of the model*

Applying Surface Moment

Similar to applying nodal moment on the selected nodes you can also apply moment on surfaces. To do so, select the surface on which you want to apply moment and then choose the **Moment** tool from the **Loads** panel of the **Setup** tab in the **Ribbon**; the **Creating Surface Moment Object** dialog box will be displayed, refer to Figure 8-65. Alternatively, you can invoke this dialog box by selecting the surface of the model and right-click in the drawing area to display a shortcut menu. Next, select **Add > Surface Moment** from the shortcut menu displayed.

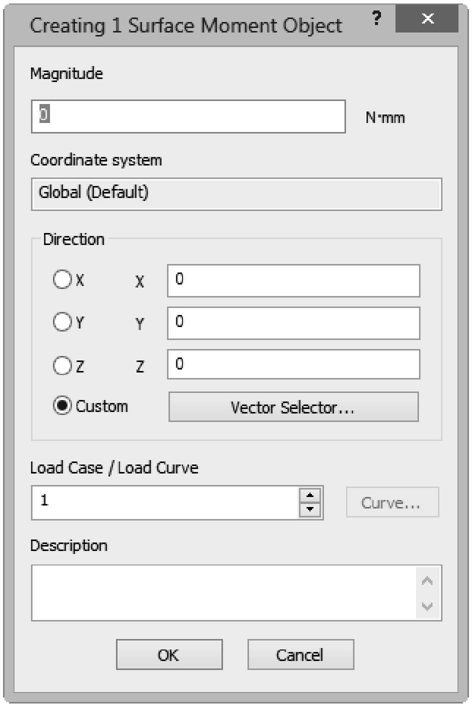


Figure 8-65 *The* ***Creating Surface Moment Object*** *dialog box*

The options available in the dialog box are same as discussed earlier. Specify the parameters of the surface moment, such as magnitude, axis of rotation, and so on in their respective edit boxes. Next, choose the **OK** button. Figure 8-66 shows a meshed model in the FEA Editor environment with the moment applied on its front face and Figure 8-67 shows the model in the Result environment. You will learn more about the Result environment of Autodesk Simulation Mechanical in the later chapters.

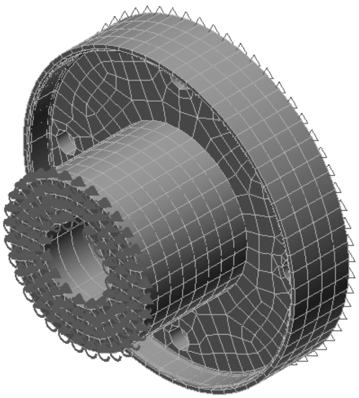


Figure 8-66 *Moment applied on the front surface of the model*

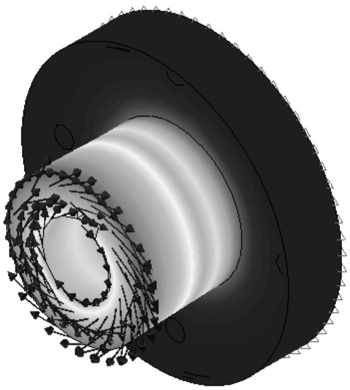


Figure 8-67 *The model in the Result environment*

Note

You can apply surface moment only on plate, brick, and tetrahedron elements.

Bearing Load

Ribbon:	Setup > Loads > Bearing

A bearing load is defined as a compressive load that causes radial, thrust, or combination of radial and thrust loading between the contact areas of shafts and bearing or bushings. To apply the bearing load: radial and thrust, select a cylindrical surface of the model where you want to apply bearing load from the drawing area and then choose the **Bearing** tool from the **Loads** panel of the **Setup** tab in the **Ribbon**; the **Creating Bearing Load Object** dialog box will be displayed, refer to Figure 8-68. In this dialog box, you can specify the parameters for applying thrust and radial loading. The left half of the dialog box displays the options for specifying the thrust load and right half of the dialog box displays the options for specifying the radial load.

Note

*You can also select multiple cylindrical surfaces for applying the bearing load by pressing the CTRL key. However, depending upon the number of surfaces selected, name of the dialog box invoked will be modified and will display count number. For example, on selecting one surface, the name of the dialog box invoked will be displayed as **Creating 1 Bearing Load Object**, refer to Figure 8-66. On the other hand, on selecting two surfaces, the name of the dialog box invoked will be displayed as **Creating 2 Bearing Load Objects**.*

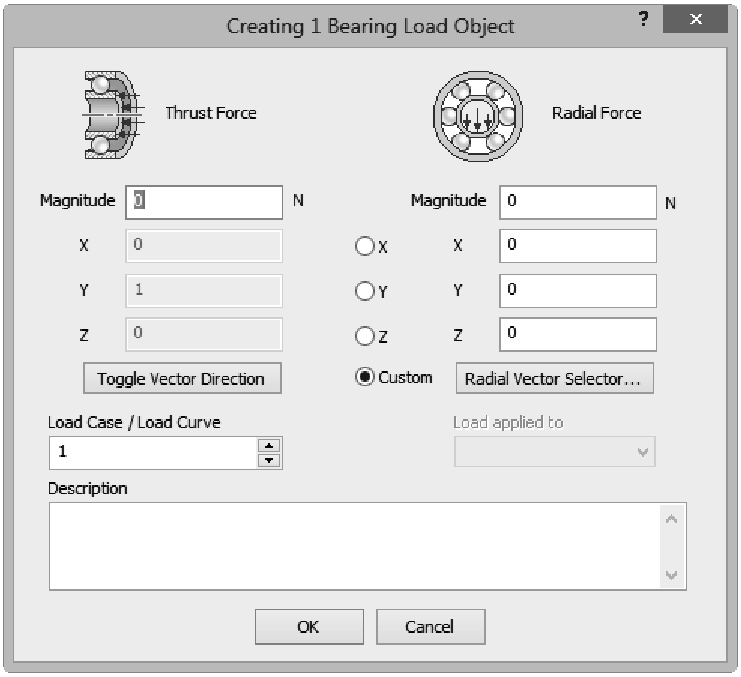


Figure 8-68 The ***Creating Bearing Load Object*** *dialog box*

To apply the thrust/radial load on the selected surface, specify the magnitude of thrust/radial load in their respective **Magnitude** edit boxes. The **Magnitude** edit box available at the left in the dialog box is used to specify the magnitude of the thrust load whereas, the **Magnitude** edit box at the right in the dialog box is used to specify the magnitude of the radial load. Note that in case of thrust load, the direction vector of the load being applied will automatically be selected in the axial direction of the surfaces selected. Therefore, the **X**, **Y**, and **Z** edit boxes available in the left side in the dialog box for specifying the direction vector for the thrust load will not be enabled. However, you can reverse the default direction of the thrust direction vector by choosing the **Toogle Vector Direction** button.

To specify the direction vector for the radial load, select the **X**, **Y**, or **Z** radio button. You can also choose the **Custom** radio button to specify the radial direction vector other than the global X, Y, and Z directions by entering the value of x, y, and z vectors manually in their respective edit boxes. These edit boxes will be enabled on choosing the **Custom** radio button. Note that the radial direction vector should not be parallel to the thrust direction vector.

You can also specify the load case/load curve to be followed for the bearing load (thrust/radial load) being applied by using the **Load Case/Load Curve** spinner of the dialog box. After specifying all the parameters for applying the bearing load (thrust/radial load), choose the **OK** button; the bearing load will be applied to the selected cylindrical surface.

Figure 8-69 shows the thrust load applied on one half of the cylindrical surface in the FEA Editor environment and Figure 8-70 shows the same model in the Result environment after performing the analysis. You will learn more about Result environment in the later chapters.

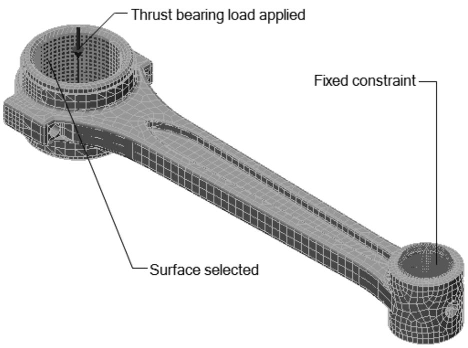


Figure 8-69 *Thrust load applied on a surface of the model in the FEA Editor environment*

Figure 8-70 *The model in the Result environment after applying the thrust load*

Figure 8-71 shows the radial load applied on one half of the cylindrical surface in the FEA Editor environment and Figure 8-72 shows the same model in the Result environment after performing the analysis.

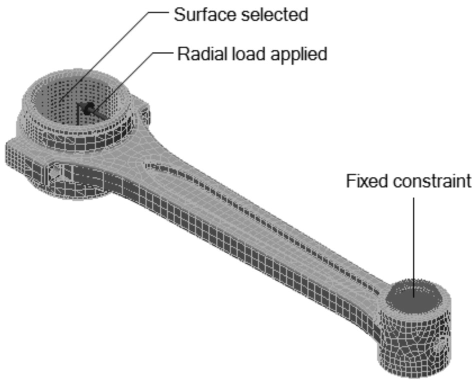


Figure 8-71 *Radial load applied on a surface of the model in FEA Editor environment*

Figure 8-72 *The model in the Result environment after applying the radial load*

Gravity

Ribbon: Setup > Loads > Gravity

Gravity is defined as a force per unit mass. It applies constant acceleration along the direction to a part that has mass. In Autodesk Simulation Mechanical, you can simulate the force of gravity by defining the acceleration occurred due to the body force. To define gravity or acceleration load for a model, choose the **Gravity** tool from the **Loads** panel of the **Setup** tab in the **Ribbon**; the **Analysis Parameters** dialog box will be displayed with **Gravity/Acceleration** tab chosen, refer to Figure 8-73.

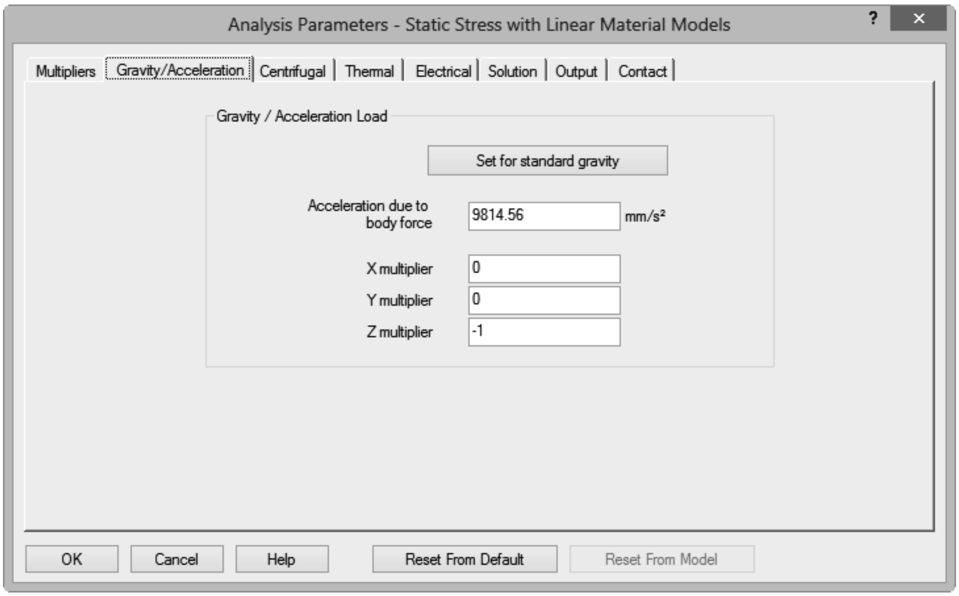


Figure 8-73 *The* ***Analysis Parameters*** *dialog box with the* ***Gravity/Acceleration*** *tab chosen*

To specify the standard acceleration due to gravity on earth, choose the **Set for standard gravity** button from the **Gravity/Acceleration** tab. As soon as you choose this button, the standard acceleration due to the gravity on earth as per the units defined for the model will be entered automatically in the **Acceleration due to body force** edit box of the **Gravity/Acceleration** tab in the dialog box. You can also enter a different acceleration magnitude as per the requirement in the **Acceleration due to body force** edit box.

After specifying the acceleration, you need to define the direction along which the acceleration will be applied. The **X multiplier**, **Y multiplier**, and **Z multiplier** edit boxes of the **Gravity/Acceleration** tab in the dialog box are used to define the vector along which the acceleration will be applied. To specify the direction of acceleration towards the positive direction of X axis, specify a positive value in the **X multiplier** edit box of the dialog box. To specify the direction of acceleration towards the negative direction of X axis, specify a negative value in the **X multiplier** edit box. Similarly, you can specify the direction of acceleration towards the positive or negative direction of Y and Z axes. In addition to specifying the acceleration direction along the X, Y, or Z axes, you can also specify the acceleration along an arbitrary direction by specifying the value in more than one edit box. For example, if you enter 1 in the X multiplier and 1 in the Y multiplier, the direction of acceleration will bisecting the X and Y axes.

Note

*The value entered in the **Acceleration due to body force** edit box will be multiplied by the values entered in the **X multiplier**, **Y multiplier**, and **Z multiplier** edit boxes before it is applied to the model in the direction of acceleration of gravity.*

The **Reset From Default** button of the dialog box is used to reset the values to default.

After defining the gravity and its direction, choose the **OK** button from the dialog box; the gravity will be applied and the direction arrow will be displayed in the model.

Note

*You may need to change the display style of the model to view the direction arrow of the applied gravity to either Mesh or Edges display style. To do so, click on the **View** tab in the **Ribbon** and then choose the **Mesh** or **Edges** option from the **Visual Style** drop-down list in the **Appearance** panel of the **View** tab.*

Figure 8-74 shows a model with an arrow pointing in the downward direction, representing the acceleration direction of the gravity applied.

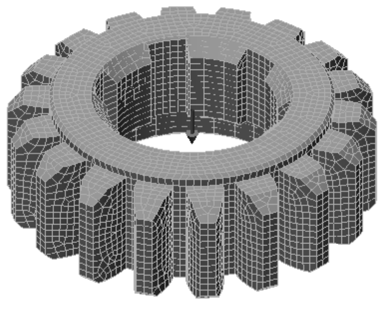


Figure 8-74 *A meshed model with direction arrow of the gravity applied*

TUTORIALS

Tutorial 1

In this tutorial, you will open the meshed model of Tutorial 1 of Chapter 5, refer to Figure 8-75. You will then assign Stainless Steel (AISI 405) material to the model. After assigning the material, you will apply the boundary conditions such as force, pin constraint, and frictionless constraint to the model, refer to the Figure 8-76. While applying the Pin constraint, you need to fix the radial and axial movement.

(**Expected time: 30 min**)

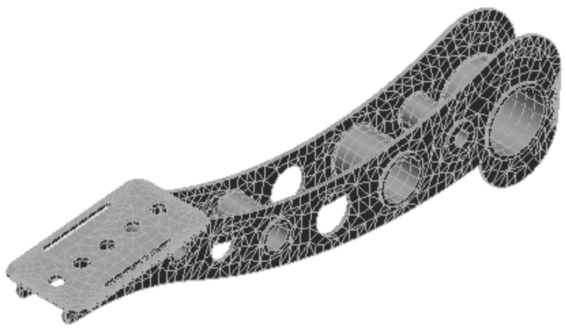


Figure 8-75 *The meshed model for Tutorial 1*

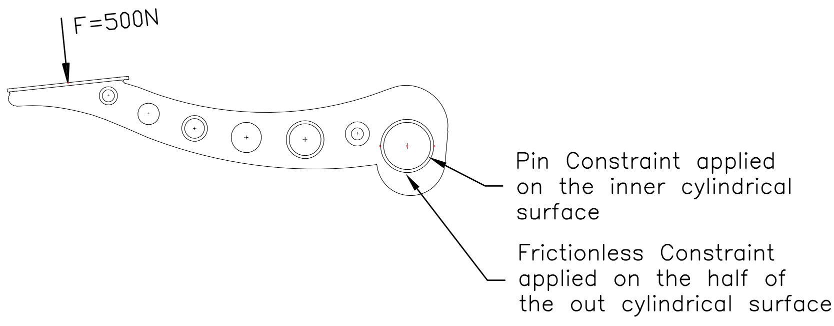


Figure 8-76 *Required boundary conditions of the model*

The following steps are required to complete this tutorial:

a. Start Autodesk Simulation Mechanical.
b. Open Tutorial 1 of Chapter 5 in Autodesk Simulation Mechanical.
c. Save the Tutorial 1 of Chapter 5 model in the *c08* folder with the name *c08_tut01*.
d. Assign material to the model.
e. Apply pin Constraint.
f. Apply frictionless Constraint.
g. Apply force.
h. Save and close the model.

Opening Tutorial 1 of Chapter 5

As the required input file is saved in the *c05* folder, you need to browse to this folder and then open the *c05_tut01.fem* file in Autodesk Simulation Mechanical.

1. Start Autodesk Simulation Mechanical and then choose the **Open** button from the **Quick Access Toolbar**; the **Open** dialog box is displayed.

2. Browse to the *C:\Autodesk Simulation Mechanical\c05\Tut01* folder.

3. Select the *c05_tut01.fem* file and then choose the **Open** button from the dialog box; the selected *fem* file is opened in the Autodesk Simulation Mechanical.

Saving the .fem File in the c08 Folder

When you open a file created in some other chapter, it is recommended to first save the file with a different name in the folder of the current chapter (document), before modifying it. On saving the file in the current chapter folder, the original file of the other chapter will not get modified.

1. Choose the **Save As** button from the **Application Menu**; the **Save As** dialog box is displayed.

2. Browse to the *\Autodesk Simulation Mechanical* folder. Next, create a new folder with the name *c08* by using the **New Folder** button. Create one more folder with the name *Tut01* inside the *c08* folder. Make the newly created *Tut01* folder the current folder by double-clicking on it.

3. Enter **c08_tut01** as the new name of the file in the **File name** edit box and then choose the **Save** button to save the file.

   The file is saved with the new name and is now opened in the graphic area of Autodesk Simulation Mechanical.

Assigning Material

Now, you need to assign material to the model.

1. Expand the **Part** node by clicking on the **+** sign available on its left in the **Tree View**. Next, right-click on the **Material <Unnamed>** option available under the **Part** node in the **Tree View** and then select the **Edit Material** option from the shortcut menu displayed; the **Element Material Selection** dialog box will be displayed, refer to Figure 8-77.

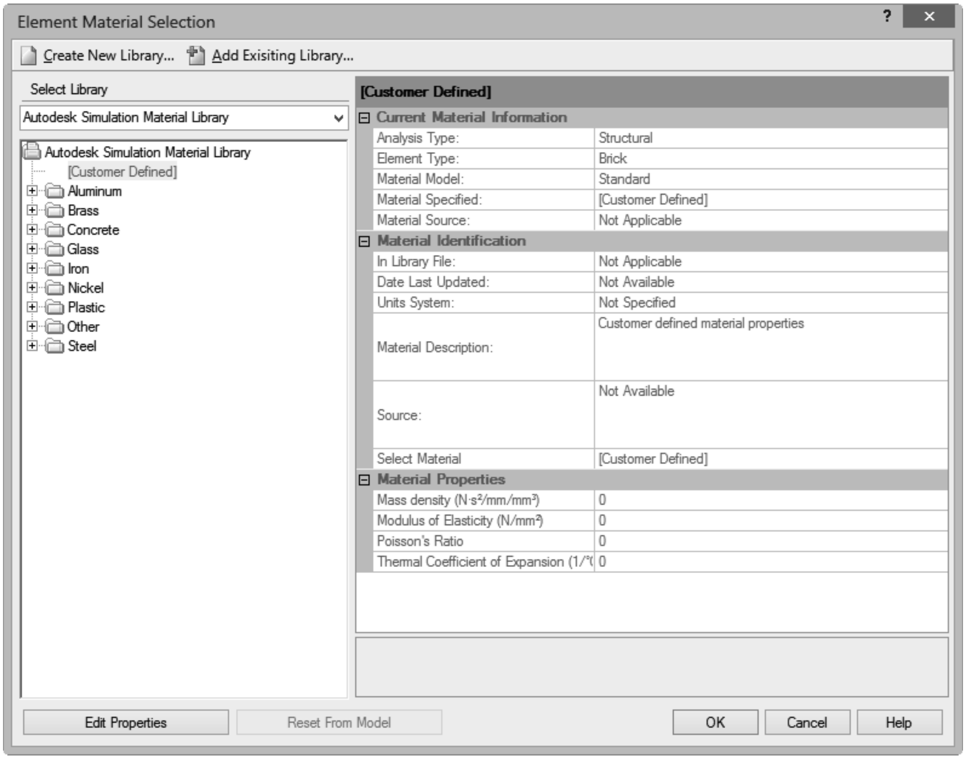


Figure 8-77 *The* ***Element Material Selection*** *dialog box*

2. Make sure **Autodesk Simulation Material Library** is selected in the **Select Library** drop-down list of the dialog box. Next, expand the **Steel** material family node by clicking on the + sign available on the left of the **Steel** node.

3. Expand the **Stainless** sub-node from the **Steel** material family and then select the **Stainless Steel (AISI 405)** material; all the default properties of the Stainless Steel (AISI 405) material are displayed on the right in the dialog box, as shown in Figure 8-78.

4. Choose the **OK** button from the dialog box; all the properties of the selected material are assigned to the model.

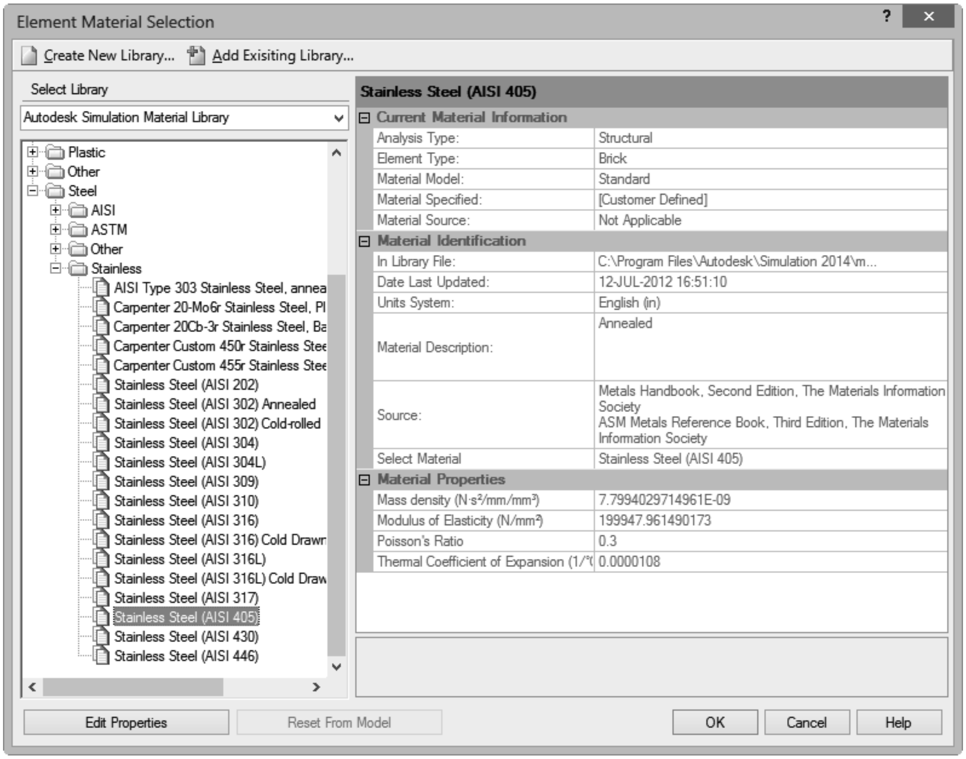


*Figure 8-78 The **Element Material Selection** dialog box with the Stainless Steel (AISI 405) material selected*

Applying Pin Constraint

Now, you need to apply the pin constraint to the inner cylindrical surface of the model, refer to Figure 8-76.

1. Change the current orientation of the model similar to the one shown in Figure 8-79 by using the **ViewCube**.

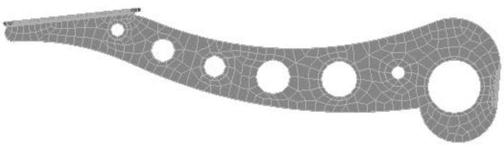


Figure 8-79 Changed orientation of the model

2. Click on the **Selection** tab of the **Ribbon** and then choose the **Circle** tool from the **Shape** panel of the **Selection** tab. Next, choose the **Surfaces** tool from the **Select** panel from the **Selection** tab in the **Ribbon**.

3. Draw a circle around the large hole of the model, refer to Figure 8-80. All the surfaces that are completely enclosed inside the circle are selected, refer to Figure 8-81.

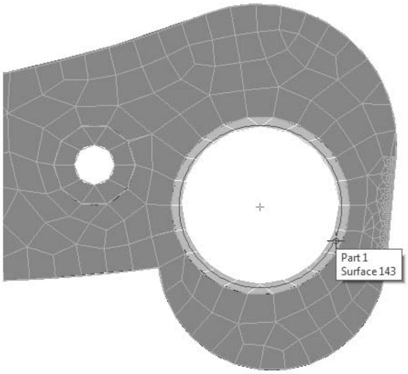


Figure 8-80 The circle being drawn around the large hole of the model

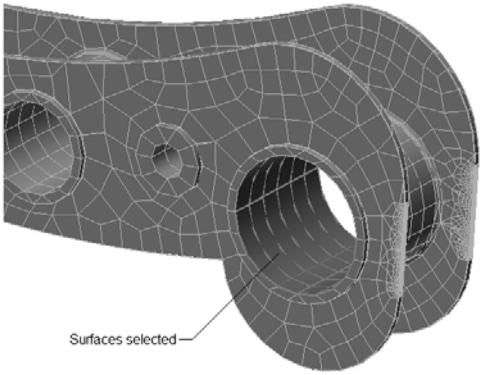


Figure 8-81 Surfaces that are completely enclosed inside the circle drawn are selected

4. Choose the **Setup** tab of the **Ribbon** and then choose the **Pin Constraint** tool from the **Constraints** panel of the **Setup** tab in the **Ribbon**; the **Creating Pin Constraint Objects** dialog box is displayed, as shown in Figure 8-82.

5. Select the **Fix Radial** and **Fix Axial** check boxes from the **Constraints** area of the dialog box. Next, choose the **OK** button; the pin constraint is applied. Figure 8-83 shows the rotated view of the model with the pin constraint applied.

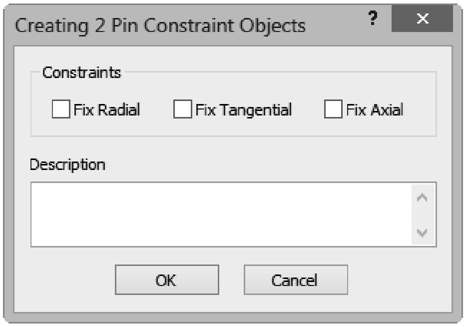


Figure 8-82 *The* ***Creating Pin Constraint Objects*** *dialog box*

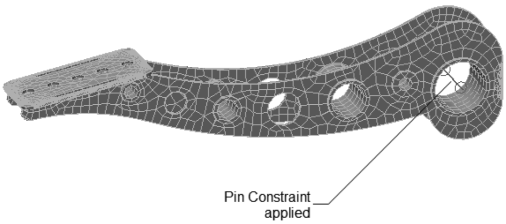


Figure 8-83 The rotated view of the model with pin constraint applied

Applying Frictionless Constraint

Now you need to apply frictionless constraint to one half of the larger outer cylindrical surface of the model, refer Figure 8-76.

1. Change the current orientation of the model similar to the one shown in Figure 8-84 by using the **ViewCube**.

2. Click on the **Selection** tab of the **Ribbon** and then choose the **Point or rectangle** tool from the **Shape** panel of the **Selection** tab. Next, choose the **Surfaces** tool from the **Select** panel of the **Selection** tab in the **Ribbon**.

3. Select the bottom surface of the outer cylinder by clicking the left mouse button, refer to Figure 8-85.

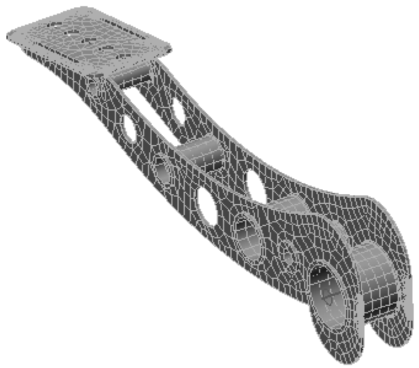


Figure 8-84 *Rotated view of the model*

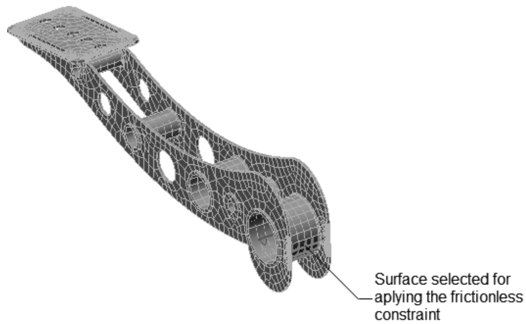


Figure 8-85 *Surface selected for applying the frictionless constraint*

4. Choose the **Setup** tab of the **Ribbon**. Next, expand the **Constraints** panel of the **Setup** tab in the **Ribbon** by clicking on the down arrow available on the **Constraints** panel bar.

5. Choose the **Frictionless** tool from the expanded **Constraints** panel, refer to Figure 8-86; the **Creating Frictionless Constraint Object** dialog box is displayed, as shown in Figure 8-87.

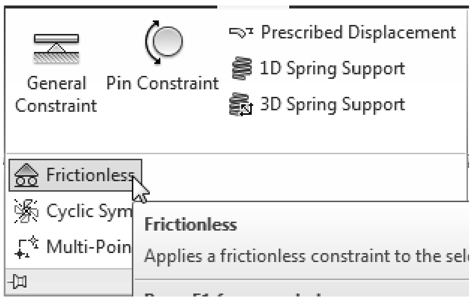


Figure 8-86 *The expanded* ***Constraints*** *panel of the* ***Setup*** *tab*

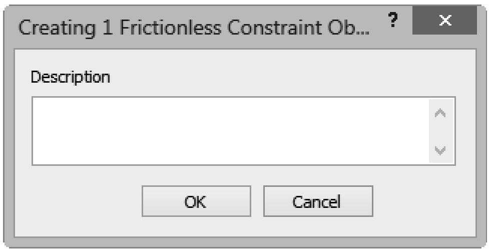


Figure 8-87 *The* ***Creating Frictionless Constraint Object*** *dialog box*

6. Choose the **OK** button from the **Creating Frictionless Constraint Object** dialog box; the frictionless constraint is applied to the selected surface of the model, refer to Figure 8-88.

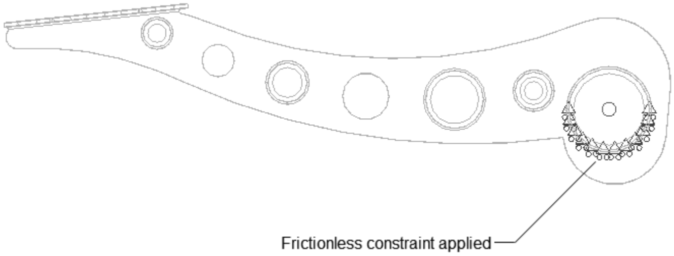


Figure 8-88 *The frictionless constraint applied on the surface*

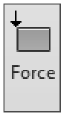
Note

*In Figure 8-88, the visual style of the model has been changed to edges visual style for clarity. You can do so choosing the **Edges** tool from the **Visual Style** drop-down of the **Appearance** panel of the **View** tab in the **Ribbon**.*

Applying Force

Now you need to apply force of 500 N on the top planar surface of the model, refer to Figure 8-76.

1. Select the top planar surface of the model for applying the force of 500 N by clicking the left mouse button, refer to Figure 8-89.



2. Choose the **Setup** tab of the **Ribbon** and then choose the **Force** tool from the **Loads** panel; the **Creating Surface Force Object** dialog box is displayed.

3. Enter **500** in the **Magnitude** edit box of the dialog box. Next, make sure that the **Normal** radio button is selected in the **Direction** area of the dialog box.

4. Choose the **OK** button from the dialog box; the force of magnitude 500 N is applied to the selected surface and the arrows representing the force are displayed in the graphic area, as shown in Figure 8-90.

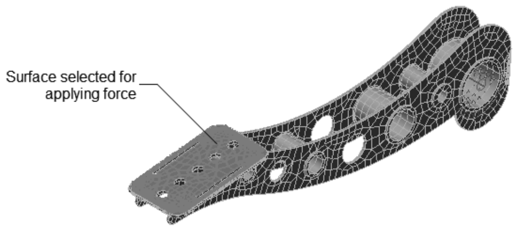


Figure 8-89 *Surface selected for applying force*

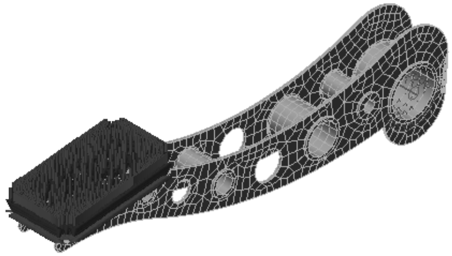


Figure 8-90 *The arrows representation of force applied*

Saving the Model

1. Choose the **Save** button from the **Quick Access Toolbar** to save the model.

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

Tutorial 2

In this tutorial, you will open model that was meshed in Tutorial 1 of Chapter 7, refer to Figure 8-91. You will then assign Aluminum Alloy 6061-O material to all the parts of the model. After assigning the material, define the cross-section area for the truss element used in pin joint part of the model to 2 square millimeter. Next, apply the boundary conditions such as force of magnitude 500 N and 1D spring constraint with stiffness value 1000 N/mm, refer to Figure 8-92. (**Expected time: 30 min**)

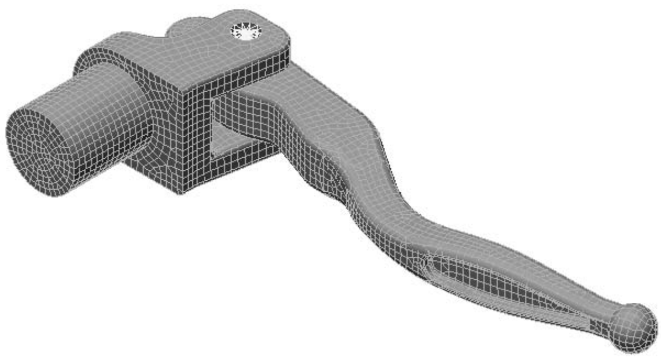


Figure 8-91 *The model for Tutorial 2*

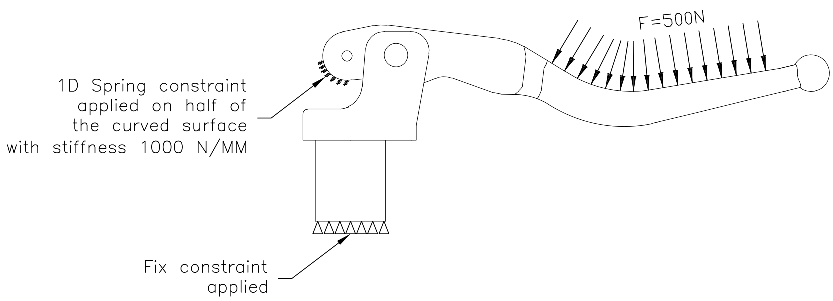


Figure 8-92 *Required boundary conditions of the model*

The following steps are required to complete this tutorial:

a. Start Autodesk Simulation Mechanical.
b. Open Tutorial 1 of Chapter 7 in Autodesk Simulation Mechanical.
c. Save the opened model in the *c08* folder with the name *c08_tut02*.
d. Assign material to the model.
e. Define cross-section for truss element in pin joint.
f. Apply 1D spring support constraint.
g. Apply fixed constraint.
h. Apply force.
i. Save and close the model.

Opening Tutorial 1 of Chapter 7

As the required input file is saved in the *c07* folder, you need to browse to this folder and then open the *c07_tut01.fem* file in Autodesk Simulation Mechanical.

1. Start Autodesk Simulation Mechanical, if not started already. Next, choose the **Open** button from the **Quick Access Toolbar**; the **Open** dialog box is displayed.

2. Browse to the *C:\Autodesk Simulation Mechanical\c07\Tut01* folder.

3. Select the *c07_tut01.fem* file and then choose the **Open** button from the dialog box; the selected *fem* file is opened in the Autodesk Simulation Mechanical.

Saving the .fem File in the c08 Folder

When you open a file created in some other chapter, it is recommended that you first save the file with a different name in the folder of the current chapter (document), before modifying it. This is because if you save the file in the folder of the current chapter, the original file will not get modified.

1. Choose the **Save As** button from the **Application Menu**; the **Save As** dialog box is displayed.

2. Browse to the *Autodesk Simulation Mechanical* folder. Next, create a new folder with the name *c08* by using the **New Folder** button, if not created in Tutorial 1 of this chapter. Next, create a folder with the name *Tut02* inside the *c08* folder. Make the newly created *Tut02* folder the current folder by double-clicking on it.

3. Enter **c08_tut02** as the new name of the file in the **File name** edit box and then choose the **Save** button to save the file.

   The file is saved with the new name and is now opened in the graphic area of Autodesk Simulation Mechanical.

Assigning Material

Now, you need to assign material to all the parts of the model.

1. Expand the **Part 1** and **Part 2** nodes in the **Tree View** and then select the **Material <Unnamed>** options of both the parts (**Part 1** and **Part 2**) by pressing the CTRL key.

2. Right-click to display a shortcut menu and select the **Edit Material** option from the shortcut menu displayed; the **Element Material Selection** dialog box is displayed.

3. Expand the **Aluminum** material family node by clicking on the **+** sign available on the left of the **Aluminum** node; the materials available in this family are displayed.

4. Select the **Aluminum Alloy 6061-O** material from the list of **Aluminum** material family; all the properties of the selected material are displayed on the right side of the dialog box.

5. Choose the **OK** button from the dialog box; all the properties of the **Aluminum Alloy 6061-O** material are assigned to the selected parts of the model.

6. Similarly, assign the **Aluminum Alloy 6061-O** material to the **Part 3** of the model.

Defining Cross-section for Truss Element in Pin Joint

Now, you need to define cross-section for the truss elements used from creating the pin joint in the model.

1. Expand the **Part 3** nodes in the **Tree View**, if not expanded already. Next, select the **Element Definition** option and right-click to display a shortcut menu.

2. Select the **Edit Element Definition** option from the shortcut menu; the **Element Definition - Truss** dialog box is displayed, as shown in Figure 8-93.

3. Enter **2** in the **Cross-sectional area** edit box of the dialog box. Next, choose **OK**; the cross-section of 2 square millimeter is defined for the truss elements.

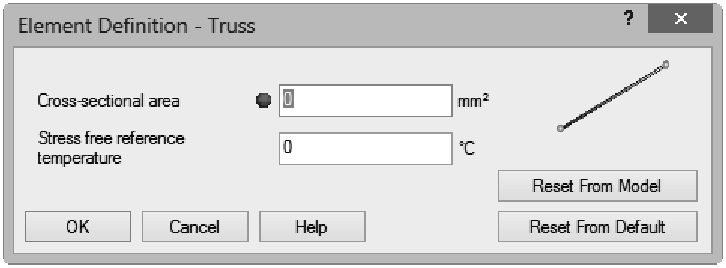


*Figure 8-93 The **Element Definition - Truss** dialog box*

Applying 1D Spring Support Constraint

Now, you will apply 1D spring support constraint.

1. Change the current orientation of the model similar to the one shown in Figure 8-94 by using the **ViewCube**.

2. Click on the **Selection** tab of the **Ribbon** and then choose the **Point or Rectangle** tool from the **Shape** panel of the **Selection** tab. Next, choose the **Surfaces** tool from the **Select** panel from the **Selection** tab in the **Ribbon**.

3. Select the curved surface by clicking the left mouse button, refer to Figure 8-95.

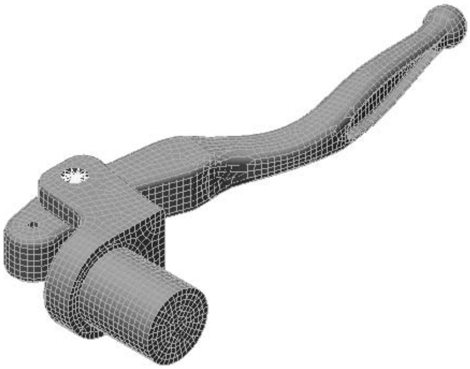


Figure 8-94 Changed orientation of the model

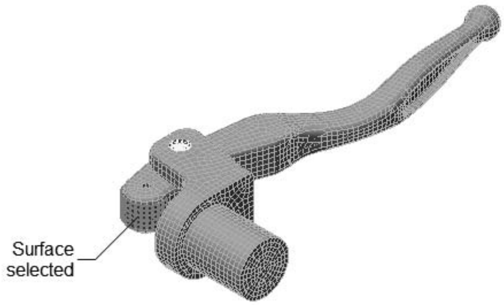


Figure 8-95 Surface selected for applying the 1D spring support constraint

4. Choose the **1D Spring Support** tool from the **Constraints** panel of the **Setup** tab in the **Ribbon**; the **Creating Surface 1D Spring Support Element** dialog box is displayed, as shown in Figure 8-96.

5. Make sure that the **Translation** radio button is selected in the **Type** area and the **Custom** radio button is selected in the **Direction** area of the dialog box, refer to Figure 8-96.

6. Enter **1** in both the **X** and **Y** edit boxes of the **Direction** area in the dialog box.

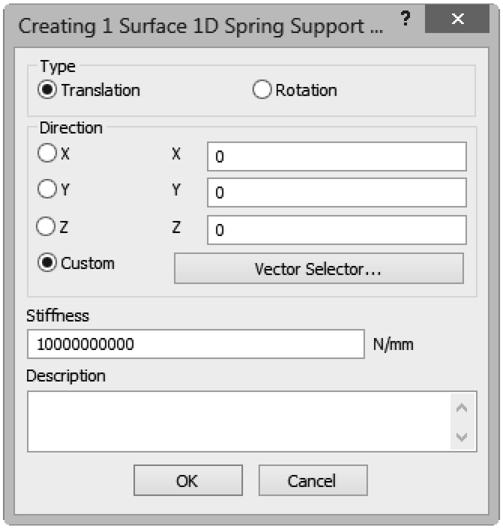


*Figure 8-96 The **Creating Surface 1D Spring Support Element** dialog box*

7. Enter **1000** in the **Stiffness** edit box of the dialog box. Next, choose the **OK** button; the 1D spring support is applied to the selected surface of the model, refer to Figure 8-97.

Applying Fixed Constraint

Now, you need to apply fixed constraint to the surface of the model, refer to Figure 8-92.

1. Select the surface of the model for applying the fixed constraint, refer to Figure 8-98.

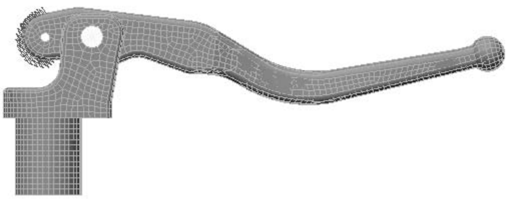


Figure 8-97 The 1D spring support applied to the selected surface of the model

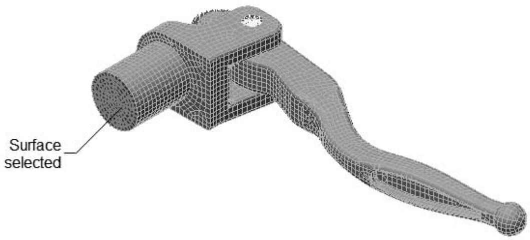


Figure 8-98 Surface selected for applying the fixed constraint

2. Select the **General Constraint** tool from the **Constraints** panel of the **Setup** tab in the **Ribbon**; the **Creating Surface General Constraint Object** dialog box is displayed, as shown in Figure 8-99.

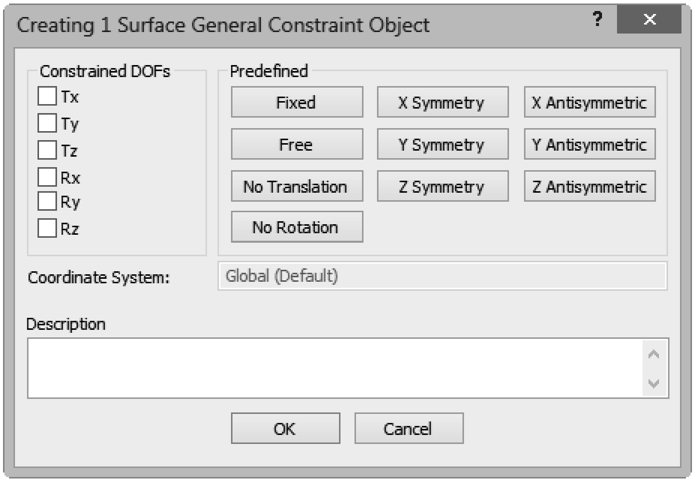


Figure 8-99 *The* ***Creating Surface General Constraint Object*** *dialog box*

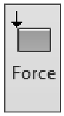
3. Choose the **Fixed** button from the **Predefined** area of the dialog box; all the check boxes available in the **Constrained DOFs** area of the dialog box are selected.

4. Choose the **OK** button; the fixed constraint is applied to the selected surface.

Applying Force

Now, you need to apply force of 500 N on the top planar surface of the model, refer to Figure 8-92.

1. Select the top planar surfaces of the model for applying the force of 500 N by pressing the CTRL key, refer to Figure 8-100.



2. Choose the **Force** tool from the **Loads** panel of the **Setup** tab in the **Ribbon**; the **Creating Surface Force Objects** dialog box is displayed.

3. Enter **500** in the **Magnitude** edit box of the dialog box. Next, make sure that the **Normal** radio button is selected in the **Direction** area of the dialog box.

4. Choose the **OK** button from the dialog box; the force of magnitude 500 N is applied to the selected surfaces and the arrows representing the force are displayed in the graphic area, as shown in Figure 8-101.

Saving the Model

1. Choose the **Save** button from the **Quick Access Toolbar** to save the model.

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

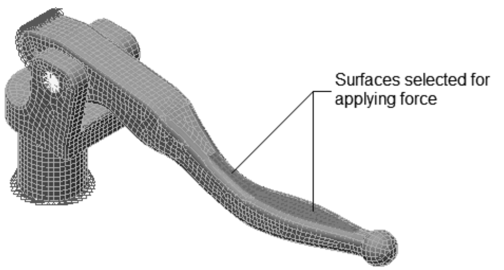


Figure 8-100 *Surfaces selected for applying force*

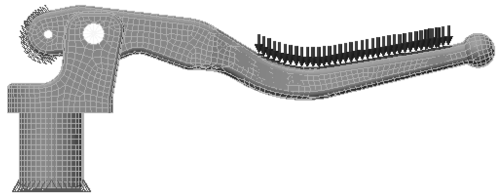


Figure 8-101 *The arrows representation of force applied*

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, the default properties of a material cannot be modified. (T/F)

2. In Autodesk Simulation Mechanical, the material library file is saved in the .mlb file format. (T/F)

3. You can create a copy of an existing material library to create a new material library. (T/F)

4. The **General Constraint** tool is used to constraint the degree of freedom of nodes. (T/F)

5. In Autodesk Simulation Mechanical, by default, you are only provided with Autodesk Simulation Material Library. (T/F)

6. The general constraint applied on a node is called __________.

7. The __________ tool is used to constraint the radial, tangential, and axial degree of freedom of the selected cylindrical surface.

8. On choosing the **Remote Loads** tool, the __________ dialog box will be displayed.

9. In Autodesk Simulation Mechanical, you can apply __________ or __________ pressure to the selected surface by using the **Creating Surface Pressure/Traction Object** dialog box.

10. On choosing the __________ button from the **Predefined** area of the **Creating Nodal General Constraint Object** dialog box, the translation degree of freedom along the X, Y, and Z axes of the selected node will be constrained.

Review Questions

Answer the following questions:

1. In Autodesk Simulation Mechanical, in addition to the default libraries, you can create a new material library. (T/F)

2. In Autodesk Simulation Mechanical, you cannot delete a material from the user-defined material libraries. (T/F)

3. You can apply pin constraint to the cylindrical and planar surfaces of the model. (T/F)

4. Which of the following dialog boxes is displayed on choosing the **Variable Pressure** tool?

   (a) **Variable Pressure** (b) **Creating Variable Pressure**
   (c) **Creating Surface Variable Pressure** (d) None of these

5. Which of the following dialog boxes is displayed on choosing the **Gravity** tool?

   (a) **Analysis Gravity** (b) **Analysis Parameters**
   (c) **Gravity** (d) None of these

6. Which of the following dialog boxes is displayed on choosing the **Edit Properties** button from the **Element Material Selection** dialog box?

   (a) **Element Material Specification** (b) **Edit Specification**
   (c) **Edit Material Specification** (d) None of these

7. Which of the following radio buttons of the **Creating Surface Pressure/Traction Object** dialog box is used to apply a pressure that is oriented in a specific directions?

   (a) **Pressure** (b) **Traction**
   (c) **Pressure/Traction** (d) None of these

8. On choosing the __________ button from the **Creating Surface Pressure/Traction Object** dialog box, the **Multiplier Table Editor** dialog box will be displayed.

9. On choosing the **Import CSV** button from the **Multiplier Table Editor** dialog box, you can import the data of an existing load curve is saved in the __________ file format.

EXERCISES

Exercise 1

In this exercise, you will open the model created in Tutorial 3 of Chapter 7, refer to Figure 8-102. You will then assign Stainless Steel (AISI 202) material to all parts of the model. After assigning the material, apply the boundary conditions such as pressure of magnitude 1000 N inside the model and the fixed constraint at the base end of the model, refer to Figure 8-103.

(**Expected time: 30 min**)

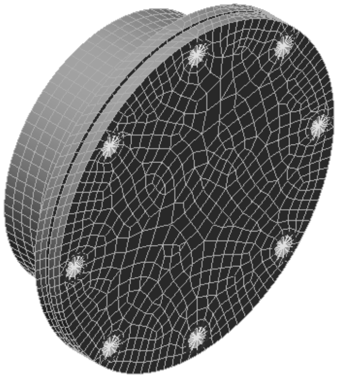


Figure 8-102 *The model for Exercise 1*

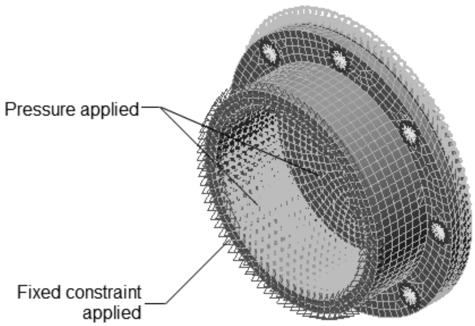


Figure 8-103 *The model after applying boundary conditions*

Exercise 2

In this exercise, you will open the model created in Tutorial 2 of Chapter 7, refer to Figure 8-104. You will then assign Stainless Steel (AISI 446) material to all parts of the model. After assigning the material, define the cross-section area for the truss elements used in pin joint parts of the model to 1 square millimeter. Next, apply the boundary conditions such as force of magnitude 650 N at the top planar face of the model towards down and fixed constraint at the base of the model, refer to Figure 8-105.

(**Expected time: 30 min**)

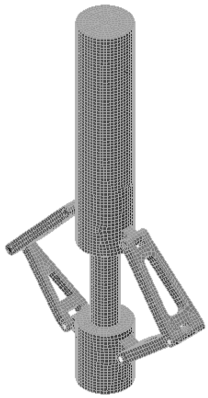


Figure 8-104 *The model for Exercise 2*

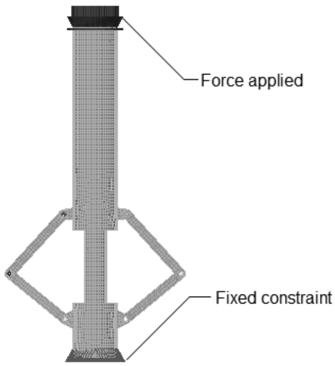


Figure 8-105 *The model after applying boundary conditions*

Answers to Self-Evaluation Test

1. F, **2.** T, **3.** T, **4.** T, **5.** F, **6.** nodal general constraint, **7. Pin Constraint**, **8. Creating Surface Remote Force**, **9.** pressure and traction, **10. No Translation**