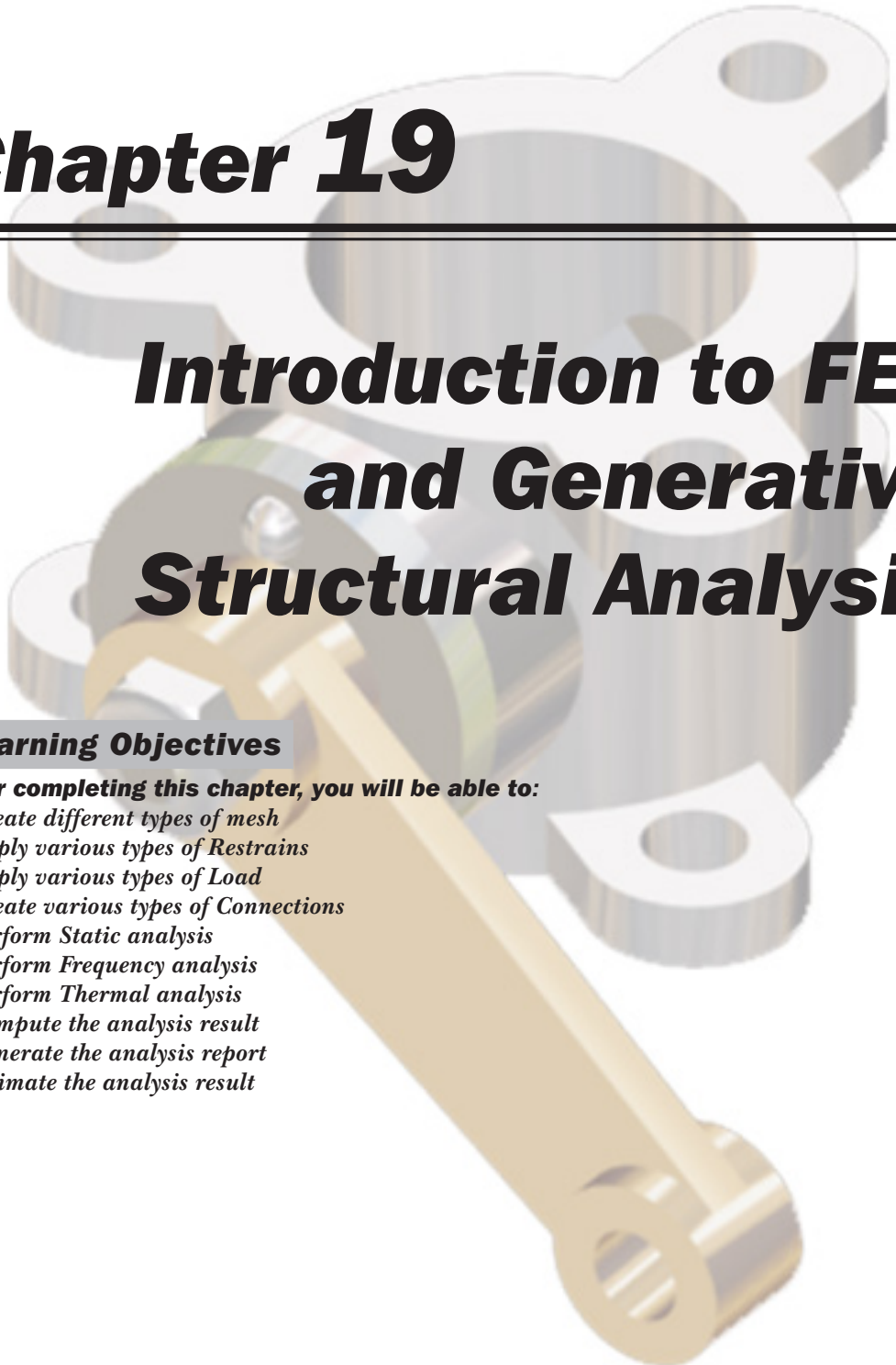
Chapter 19

Introduction to FEA and Generative Structural Analysis



Learning Objectives

After completing this chapter, you will be able to:

- *Create different types of mesh*
- *Apply various types of Restrains*
- *Apply various types of Load*
- *Create various types of Connections*
- *Perform Static analysis*
- *Perform Frequency analysis*
- *Perform Thermal analysis*
- *Compute the analysis result*
- *Generate the analysis report*
- *Animate the analysis result*

INTRODUCTION TO FEA

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

The concept of FEA can be explained with a small example of measuring the perimeter of a circle. To measure the perimeter of a circle without using the conventional formula, divide the circle into equal segments, as shown in Figure 19-1. Next, join the start point and the endpoint of each of these segments with a straight line. Now, you can measure the length of straight line very easily, and thus, the perimeter of the circle by adding the length of these straight lines.

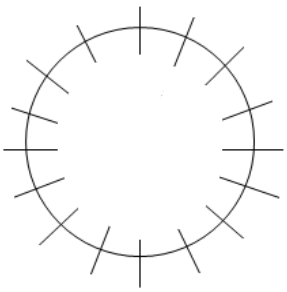


Figure 19-1 *The circle divided into small equal segments*

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

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

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

TYPES OF ENGINEERING ANALYSIS

The following types of analysis can be performed by using the FEA software:

1. Structural analysis
2. Thermal analysis
3. Fluid flow analysis
4. Electromagnetic field analysis
5. Coupled field analysis

Structural Analysis

In structural analysis, first the nodal degrees of freedom (displacement) are calculated and then the stress, strains, and reaction forces are calculated from nodal displacements. The classification of structural analysis is shown in Figure 19-2. The types of structural analysis are discussed next.

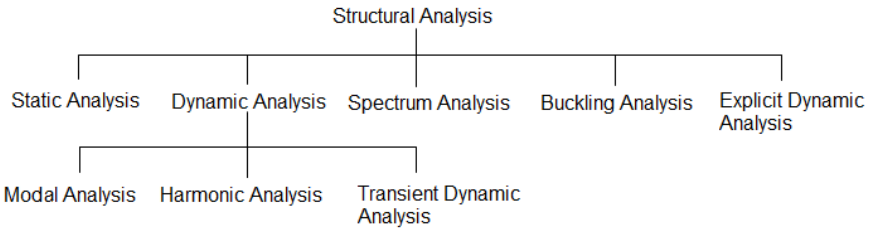


Figure 19-2 *Types of structural analysis*

Static Analysis

In static analysis, the load or field conditions do not vary with respect to time. Therefore, it is assumed that these conditions are applied gradually, not suddenly. The system under this analysis can be linear or nonlinear. The inertia and damping effects are ignored in structural analysis. In structural analysis, the following matrices are solved:

$$[K] \times [X] = [F]$$

Where,

K = Stiffness Matrix
X = Displacement Matrix
F = Load Matrix

The above equation is called the force balance equation for the linear system. If the elements of matrix [K] are the function of [X], the system is known as the nonlinear system. Nonlinear systems include large deformation, plasticity, creep, and so on. The loadings that can be applied in a static analysis include:

1. Externally applied forces and pressures
2. Steady-state inertial forces (such as gravity or rotational velocity)
3. Imposed (non-zero) displacements
4. Temperatures (for thermal strain)
5. Fluences (for nuclear swelling)

Following can be the outputs of analysis performed by FEA software:

1. Displacements
2. Strains
3. Stresses
4. Reaction forces

Dynamic Analysis

In dynamic analysis, the load or field conditions do vary with time. Therefore, it is assumed that these conditions are applied suddenly. The system can be linear or nonlinear. The dynamic load includes oscillating loads, impacts, collisions, and random loads. The three main categories of the dynamic analysis are discussed next.

Modal Analysis

It is used to calculate the natural frequency and mode shape of a structure.

Harmonic Analysis

It is used to calculate the response of a structure to the loads that are varying with time harmonically.

Transient Dynamic Analysis

It is used to calculate the response of a structure to arbitrary time varying loads.

Spectrum Analysis

This is an extension of the modal analysis and is used to calculate stress and strain due to the response of the spectrum (random vibrations). For example, you can use it to analyze how well a structure will perform and survive in an earthquake.

Buckling Analysis

This type of analysis is used to calculate the buckling load and the buckling mode shape. Slender structures (thin and long) when loaded in the axial direction, buckle under relatively small loads. For such structures, the buckling load becomes a critical design factor.

Explicit Dynamic Analysis

This type of structural analysis is used to get fast solutions for large deformation dynamics and complex contact problems, for example, explosions, aircraft crash worthiness, and so on.

Thermal Analysis

The thermal analysis is used to determine the temperature distribution and related thermal properties such as Thermal distribution, Amount of heat loss or gain, Thermal gradients and Thermal fluxes.

The types of Thermal analysis are:
1. Steady state thermal analysis
2. Transient thermal analysis

Steady State Thermal Analysis
In this analysis, the system is studied under steady thermal loads with respect to time.

Transient Thermal Analysis
In this analysis, the system is studied under varying thermal loads with respect to time.

Fluid Flow Analysis
This analysis is used to determine the flow distribution and temperature of a fluid. The outputs that are expected from the fluid flow analysis are velocities, pressure, temperature, and film coefficients.

Electromagnetic Field Analysis
This type of analysis is conducted to determine the magnetic fields in electromagnetic devices. The types of electromagnetic analyses are:

1. Static analysis
2. Harmonic analysis
3. Transient analysis

Coupled Field Analysis
This type of analysis considers the mutual interaction between multiple fields. It is impossible to solve fields separately because they are interdependent. Therefore, you need a program that can solve both the physical problems by combining them.

For example, if a component is bent in different shapes using one of the metal forming processes and then subjected to heating, the thermal characteristics of the component will depend on the new shape of the component. Therefore, first the shape of the component has to be determined through structural simulations. This is known as coupled field analysis.

GENERAL PROCEDURE TO CONDUCT FINITE ELEMENT ANALYSIS
The following steps are used to conduct the Finite Element Analysis:

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

FEA through Software

The Finite Element Analysis process can be carried out in three main phases using software: preprocessor, solution, and postprocessor, refer to Figure 19-3.

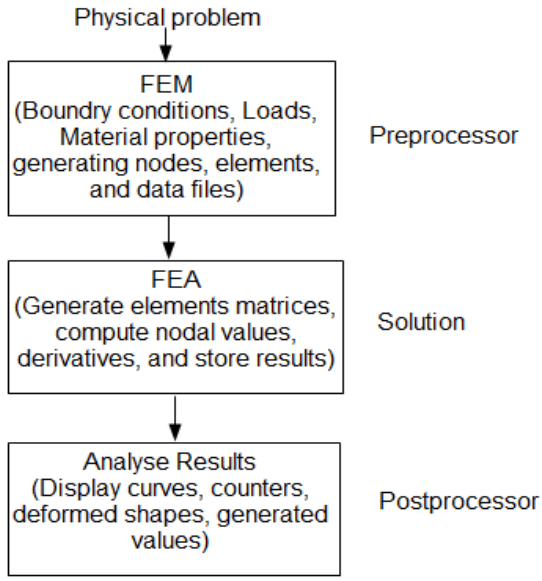


Figure 19-3 *The general process of FEA*

Preprocessor

The preprocessor is a phase that processes the input data to produce the results, which are used as input in the subsequent phase (solution). The following are the input data that need to be given to the preprocessor:

1. Type of analysis
2. Geometric Model
3. Meshing
4. Material properties
5. Loadings and boundary conditions

The input data are preprocessed for the output data. These data files are used in the subsequent phase (solution), refer to Figure 19-3.

Solution

The solution phase is completely automatic. The FEA software generates element matrices, computes nodal values and derivatives, and stores the result data in files. These files are further used in the subsequent phase (postprocessor) to review and analyze the results through the graphic display and tabular listings, refer to Figure 19-3.

Postprocessor

The output from the solution phase (result data files) is in numerical form and consists of nodal values of the field variable and its derivatives. For example, in structural analysis, the output of

the postprocessor is nodal displacement and stress in elements. The postprocessor processes the result data and displays them in graphical form to check or analyze the result. The graphical output gives the detailed information about the required result data. The postprocessor phase is automatic and generates graphical output in the specified form, refer to Figure 19-3.

IMPORTANT TERMS AND DEFINITIONS

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

Strength

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

Load

The external force acting on a body is called load.

Stress

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

$$p = F/A$$

Where,

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

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

Tensile Stress

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

Compressive Stress

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

Shear Stress

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

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

Strain

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

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

Elastic Limit

The maximum stress that can be applied to a material without producing the permanent deformation is known as the elastic limit of the material. If the stress is within the elastic limit, the material returns to its original shape and dimension on removing the external force. The following laws are used to define the response of elastic limit.

Hooke's Law

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

Stress / Strain = Constant (within the elastic limit)

Young's Modulus or Modulus of Elasticity

In case of axial loading, the ratio of intensity of the tensile or compressive stress to the corresponding strain is constant. This ratio is called Young's modulus, and is denoted by E.

E = p/e

Shear Modulus or Modulus of Rigidity

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

Ultimate Strength

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

Yield Strength

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

Factor of Safety

The ratio of the ultimate strength to the estimated maximum stress in ordinary use (design stress) is known as factor of safety. It is necessary that the design stress is within the elastic limit and to achieve this condition, the ultimate stress should be divided by a 'factor of safety'.

Lateral Strain

If a cylindrical rod is subjected to an axial tensile load, the length (*l*) of the rod will increase (*dl*) and the diameter (*0*) of the rod will decrease (*d0*). In short, the longitudinal stress will not

only produce a strain in its own direction, but will also produce a lateral strain. The ratio *dl/l* is called the longitudinal strain or the linear strain, and the ratio $d\,0/0$ is called the lateral strain.

Poisson's Ratio

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

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

Bulk Modulus

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

Stress Concentration

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

Bending

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

Bending Stress

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

Creep

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

Degrees of Freedom (DOF)

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

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

MESHING

The Finite Element Method (FEM) is a numerical approximation method which investigates the behavior of complex structures by breaking it down into smaller and simpler pieces. These pieces are called finite elements which are connected to each other through nodes. These elements and nodes are also called meshes and the process of generating mesh is known a meshing. The types of mesh elements are discussed next.

Type of Mesh Elements

In meshing, instead of specifying an element shape, you need to specify the type of meshing to be used for a particular type of element. Different types of meshes are used for different types of parts. These meshes type can be described as follow:

One Dimensional Elements

One dimensional mesh element type is used when one dimension of a part is very large as compared to the third dimension. The shape of the 1D element is a line. Example of one dimensional elements are rod, truss (bar), and beam elements.

Two Dimensional Elements

Two dimensional mesh element type is used when two dimensions of a part are larger compared to the other dimension. The shape of the 2D element are triangle and the quadrilateral. Two dimensional elements are planer elements and are also known as surface elements.

Three Dimensional Elements

Three dimensional mesh element type is used when all the three dimensions of a part are comparable to each other. The various shape of the 3D element are tetrahedron, quadrilateral pyramid, triangular prism, and hexahedron. They all have triangular and quadrilateral faces. Three dimensional elements are also known as solid elements.

The Figure 19-4 shows various types of mesh elements with number of nodes.

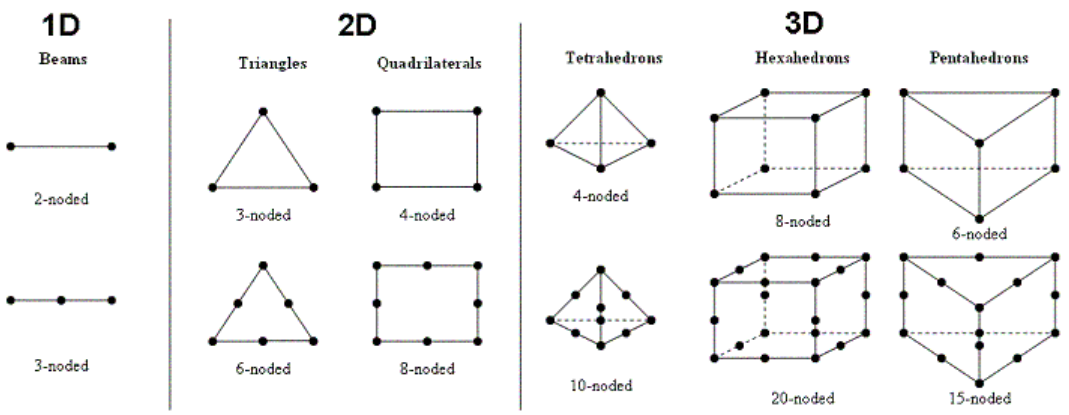


Figure 19-4 *Different type of mesh elements*

ADVANCED MESHING TOOLS WORKBENCH

The Advanced Meshing Tools workbench provides you tools to quickly generate a finite element model for the complex parts with advanced control on mesh specifications whether they are surface parts or solid parts.

Staring the Advanced Meshing Tools Workbench

Start a new session of CATIA V5. Choose Start > Analysis and Simulation > Advanced Meshing Tools from the menu bar to start a new file in the **Advanced Meshing Tools** workbench. You can also invoke the **Advanced Meshing Tools** workbench by choosing **File> New** from the menubar. On doing so, the **New** dialog box is displayed. In this dialog box, select the **Analysis** option and choose **OK**. To import a part into this workbench, make sure that part file is already opened in the background before invoking this workbench.



Note

*The workbench that will be invoked on selecting the **Analysis** option from the **New** dialog box depends on the last workbench that was invoked from the **Start > Analysis & Simulation**. Else, the **Advanced Meshing Tools** workbench will be invoked by default.*

On invoking the **Advanced Meshing Tools** workbench, a new environment gets started. The screen display of the CATIA V5 after invoking the **Advanced Meshing Tools** workbench is shown in Figure 19-5. You will notice that the toolbars related to this workbench will also be displayed. The tools in these toolbars are discussed later in this chapter.

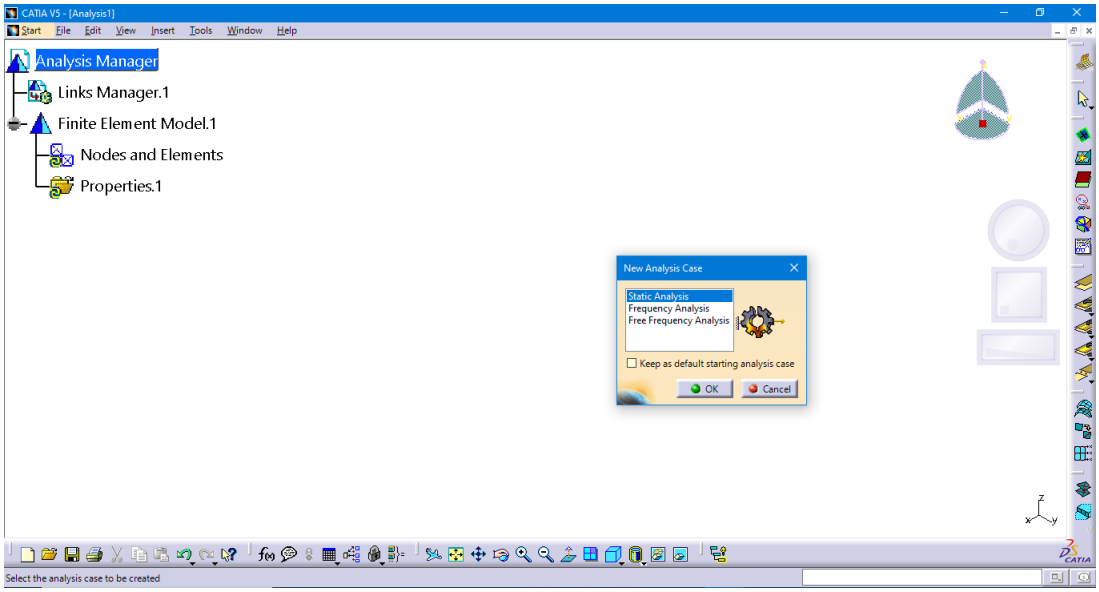


Figure 19-5 *Screen displayed after starting a new environment in the **Advanced Meshing Tools** workbench*

You will also notice that the **New Analysis Case** dialog box is displayed, as shown in Figure 19-6. This dialog box displays a selection area with three different types of analysis which are **Static Analysis**, **Frequency Analysis**, and **Free Frequency Analysis**. Choose the required analysis type from this selection box. If you want to keep the selected analysis case as default while launching the workbench again then select the **Keep as default starting analysis case** check box. By default, this check box is cleared.

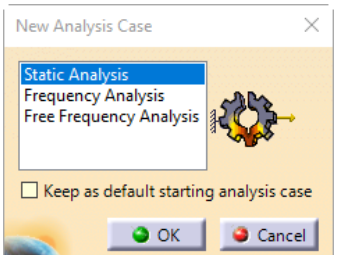


***Figure 19-6** The **New Analysis Case** dialog box*

CREATING MESH ON PARTS

The basic idea of FEA is to make calculations at only finite numbers of points and then interpolate the result for the entire domain. Any continuous object has infinite degree of freedom and it is just not possible to solve the problem in that format. Finite element method reduces the degree of freedom from infinite with the help of discretization or mesh. The different types of meshing tools are discussed next.

Beam Mesher

Toolbar: Meshing Methods > Beam Mesher



The **Beam Mesher** tool is used to Mesh the 1D parts which are also known as line bodies. In this workbench, you cannot mesh a sketch directly and therefore the sketch must be converted into a line body before meshing it. Otherwise, the body to be meshed must be a wireframe geometry.

To apply 1D mesh to a line body, choose the **Beam Mesher** tool from the **Meshing Methods** toolbar; the **Beam Meshing** dialog box will be displayed, as shown in Figure 19-7. Select the required element type button from the **Element type** area of this dialog box; you will be prompted to select the required line body or wireframe geometry to be meshed. Select the required line body and specify the required size of element in the **Element Size** edit box. You can also use the **Scale** button next to this edit box to measure the element size from the graphics window.

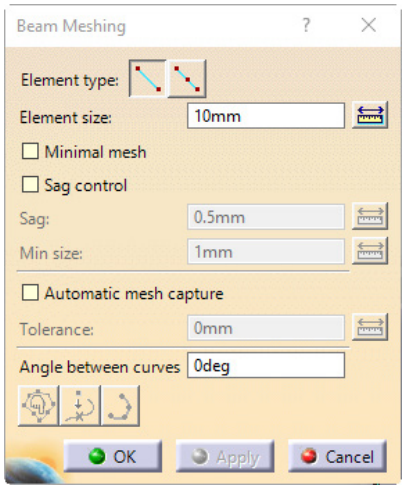


***Figure 19-7** The **Beam Meshing** dialog box*

Select the **Minimal mesh** check box to generate only one element between support vertices for the specified mesh size. Select the **Sag control** check box to control the sag of the element. This check box will be inactive if the **Minimal mesh** check box is selected. On selecting the **Sag Control** check box the **Sag** and **Min size** edit boxes will get activated.

You can specify the distance between the mesh elements and the geometry in the **Sag** edit box and specify the minimum element size in the **Min size** edit box. Select the **Automatic mesh capture** check box to capture the external mesh with a given tolerance; the **Tolerance** edit box

under this check box gets activated and the **Mesh Part Selector** button will be available on the right side of this check box. On choosing this button, the **Mesh Part Selector** selection box will be displayed. Select the required mesh part and choose **OK**.

Surface Mesher

Toolbar: Meshing Methods > Surface Meshing Methods > Surface Mesher



The **Surface Mesher** tool is used to Mesh the 2D parts which are also known as surface bodies. This tool creates a triangular or quadrangular mesh element on the selected surface. This type of mesh is also known as 2D mesh.

To generate surface mesh or 2D mesh on a surface body, choose the **Surface Mesher** tool from the **Surface Meshing Method** sub-toolbar; you will be prompted to select the required surface geometry. Select the required surface; the **Global Parameters** dialog box will be displayed, as shown in Figure 19-8. By default, the **Mesh** tab is chosen in this dialog box.

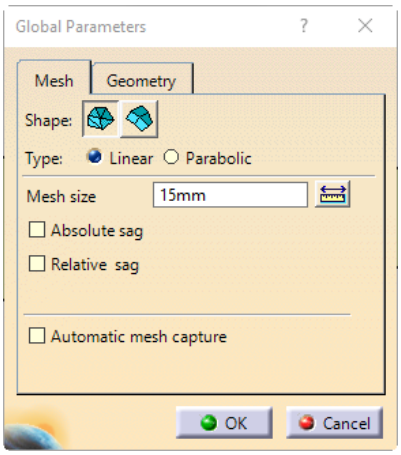


Figure 19-8 *The* ***Global Parameters*** *dialog box*

Choose the required shape (triangle or quadrangle) of mesh element by choosing the required button from the **Shape** area of this dialog box. Select the required radio button from the **Type** area to specify the element type as linear or parabolic. You can specify the required size of the mesh element in the **Mesh size** edit box.

If you choose the **Set frontal triangle method** button from the **Shape** area, the **Absolute sag** and **Relative sag** check boxes will be displayed under the **Mesh size** edit box, refer to Figure 19-8. On selecting the **Absolute sag** check box, the **Absolute sag** and **Min size** edit boxes will be displayed. If you select the **Relative sag** check box, the **Relative sag** edit box will be displayed on its right. Now, you can specify the required values in the required edit boxes where the Absolute sag is the maximum gap between the mesh and the geometry and the Relative sag is the ratio between the local absolute sag and the local mesh edge length.

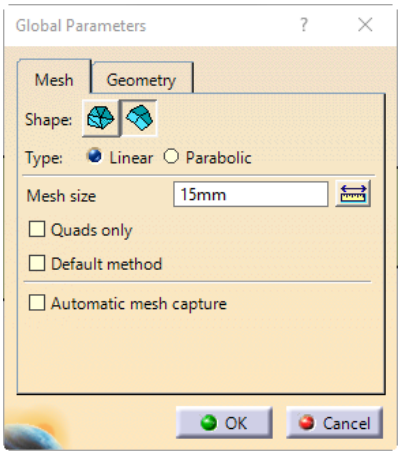


Figure 19-9 *The* ***Global Parameters*** *dialog box with* ***Set frontal quadrangle method*** *button selected*

If you choose the **Set frontal quadrangle method** button, the **Quads only** and **Default method** check boxes will be displayed, as shown in Figure 19-9. On selecting the **Quads only** check box the mesh will be automatically generated with quadrilateral elements and the **Default method** check box will get deactivated. If you select the **Default method** check box, a drop-down list will be displayed on the right side of this check box. You can select the required option from this drop-down list. Figure 19-10 shows the Local absolute sag and relative sag graph.

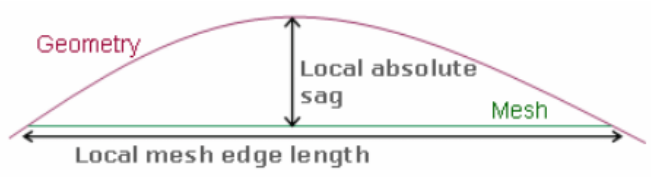


Figure 19-10 *The Local absolute sag and relative sag*

If you choose the **Geometry** tab from the **Global Parameters** dialog box, the options under this tab will be changed, refer to Figure 19-11.

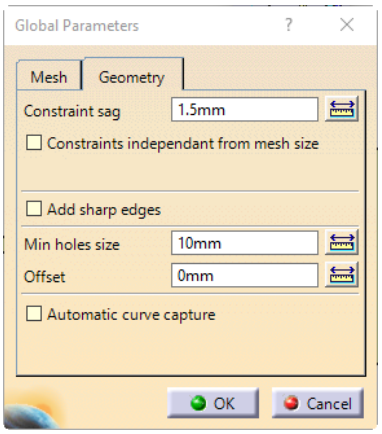


Figure 19-11 *The **Global Parameters** dialog box with **Geometry** tab selected*

Specify the constraint sag value in the **Constraint sag** edit box. The constraint is a boundary created along the edge of a face to avoid creating distorted elements across the edge. Figure 19-12 and Figure 19-13 shows the constraint created across the edge with too high and too low constraint sag value, respectively. On selecting the **Constraints independent from mesh size** check box, the **Constraint ref** size edit box will be displayed under this check box. You can specify the required value in this edit box. On selecting the **Add sharp edges** check box, the **Add sharp edges** edit box will be displayed on its right side. You can specify the value of the angle computed between two tangents on a mesh contour in this edit box. Specify the required values in the **Min hole size** edit box and meshing offset in the **Offset** edit box and choose **OK**.

To capture the mesh dynamically for all the constraints, select the **Automatic curve capture** check box; the **Tolerance** edit box is displayed under this check box. You can specify the maximum distance for curve capture in this edit box.

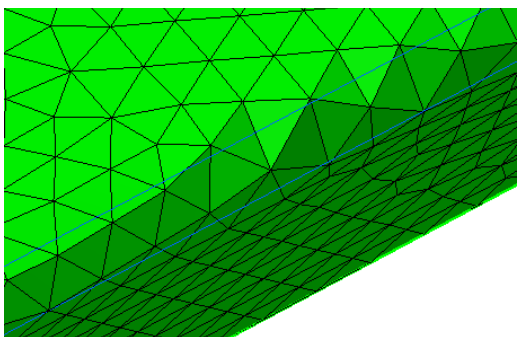


Figure 19-12 *The constraint created across the edge with high constraint sag value*

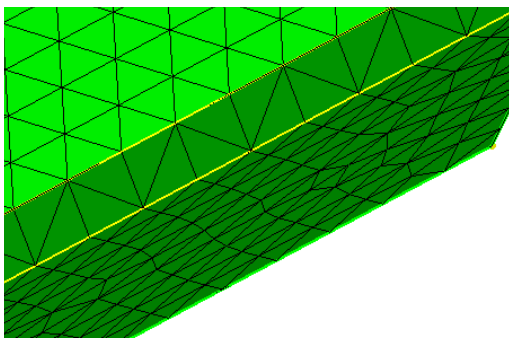


Figure 19-13 *The constraint created across the edge with low constraint sag value*

Advanced Surface Mesher

Toolbar: Meshing Methods > Surface Meshing Methods > Advanced Surface Mesher



The **Advanced Surface Mesher** tool is used to mesh the 2D parts or surface bodies. This tool creates the triangular or quadrangular mesh elements on the required surface. Meshing generated using such elements is also termed as 2D mesh.

To generate the advanced surface mesh on a surface body, choose the **Advance Surface Mesher** tool from the **Surface Meshing Method** sub-toolbar. On doing so, you will be prompted to select the required surface geometry. Select the required surface; the **Global Parameters** dialog box will be displayed, as shown in Figure 19-14. By default, the **Mesh** tab is chosen in this dialog box.

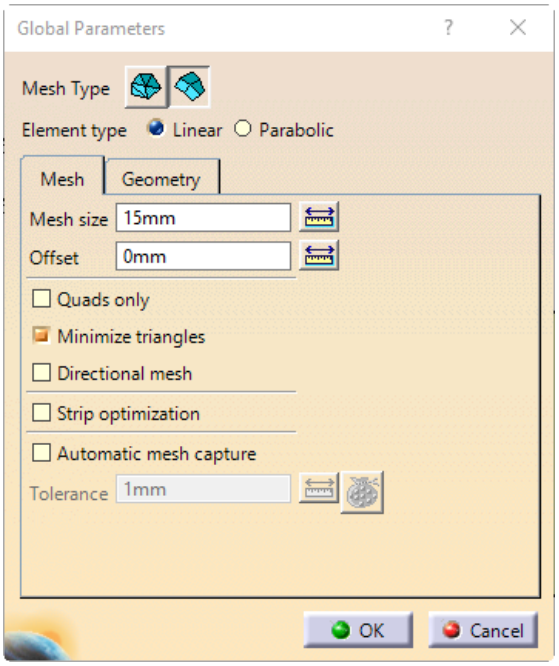


Figure 19-14 *The* ***Global Parameters*** *dialog box displayed after choosing the* ***Advanced Surface Mesher*** *tool*

Most of the options of this dialog box has already been discussed in the previous topic. To generate minimum number of triangular mesh elements, select the **Minimize triangles** check box and to generate a mesh based on a direction for all elements, select the **Directional mesh** check box. Figure 19-15 and Figure 19-16 shows the mesh created with the **Directional mesh** check box cleared and selected, respectively. If there are strips in the surface model then select the **Strip optimization** check box to generate a regular mesh along that strip. Note that the **Minimize triangles**, **Directional mesh** and **Strip optimization** check boxes will remain inactive if the **Quads only** check box is selected.

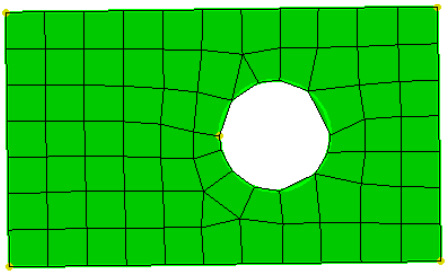


Figure 19-15 The mesh created with ***Directional mesh*** *check box cleared*

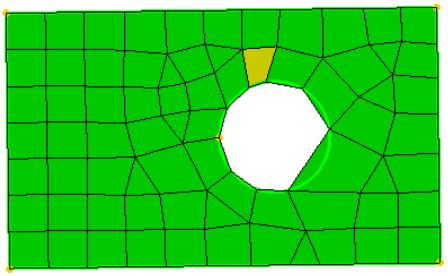


Figure 19-16 The mesh created with ***Directional mesh*** *check box selected*

You can specify the value of angle between two corresponding faces in the **Angle between faces** edit box under the **Geometry** tab. You can also specify the angle between two tangents of the contour of mesh in the **Angle between curves** edit box. Select the **Merge during simplification** check box to improve the quality of the mesh elements in the critical zones by optimizing the position of their nodes. On selecting this check box, the **Min size** edit box under this check box gets activated and you can specify the minimum size of an element in this edit box.

Tip

After choosing ***OK*** *from the* ***Surface mesher*** *and* ***Advance Surface mesher*** *tool, the Execution environment will be invoked. You can also mesh and unmesh the part using the* ***Mesh the Part*** *and* ***Remove Mesh*** *tools, respectively from the* ***Mesh/Unmesh*** *sub-toolbar of the* ***Execution*** *toolbar. You can exit the Execution environment using the* ***Exit*** *tool from the* ***Execution*** *toolbar.*

Octree Triangle Mesh

Toolbar: Meshing Methods > Surface Meshing Methods > Octree Triangle Mesher

The **Octree Triangle Mesher** tool is used to mesh the 2D parts or surface bodies. This tool creates only the triangular mesh elements on the selected surface.

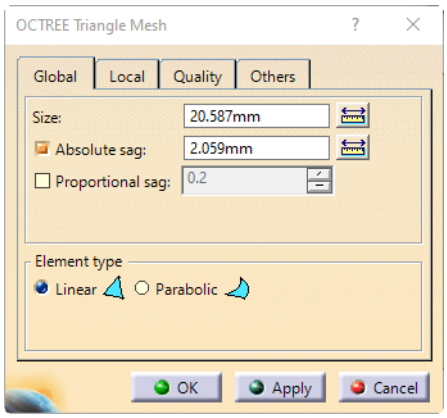


Figure 19-17 The ***OCTREE Triangle Mesh*** *dialog box*

To generate a triangular mesh on a surface body, choose the **Octree Triangle Mesher** tool from the **Surface Meshing Methods** sub-toolbar; you will be prompted to select the geometry. Select the required geometry; the **OCTREE Triangle Mesh** dialog box will be displayed, as shown in Figure 19-17. Also, you will notice a mesh symbol will be displayed at the center of the part in the graphics area. By default, the **Global** tab is chosen in this dialog box. Specify the size, the absolute sag value, and the proportional sag value of a mesh element in the **Size**, **Absolute sag** and **Proportional sag** edit boxes, respectively. The proportional sag is the ratio between the local absolute sag and the mesh edge length. Select the required radio button from the **Element Type** area to specify the element type as linear or parabolic triangle.

To add local meshing parameters to the part, choose the **Local** tab from this dialog box. Select the required parameters from **Available specs** area; the **Add** button under this area gets activated. Choose the **Add** button; a new dialog box will be displayed.

If you select the **Local size** parameter from the **Available specs** area and then choose the **Add** button; the **Local Mesh Size** dialog box will be displayed, as shown in Figure 19-18. Now, specify the support and the mesh size value and choose **OK**; the **OCTREE Triangle Mesh** dialog box will be displayed again.

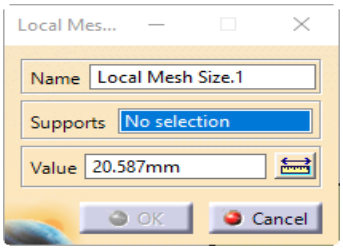


Figure 19-18 *The* ***Local Mesh Size*** *dialog box*

To optimize the quality of mesh for a part, choose the **Quality** tab from this dialog box. Choose the required criteria from the **Criteria** drop-down. The **Intermediate nodes parameters** area will be active only if you choose the **Parabolic** radio button from the **Element type** area of the **Global** tab of this dialog box. Specify the required Jacobian and warp values in the **Jacobian** and **Warp** spinners, respectively to set the position of intermediate nodes of the parabolic triangle.

You can specify the maximum size of the element to be ignored during the process of meshing in the **Geometry size limit** edit box in the **Details simplification** area under the **Others** tab. If the **Mesh edges suppression** check box is not selected by default then select this check box to remove all the small edges during meshing and specify the required limit in the spinner next to it. This spinner remains inactive if the **Mesh edges suppression** check box is cleared. Figure 19-19 and Figure 19-20 show the mesh created on a surface using the **Mesh edges suppression** check box cleared and selected respectively.

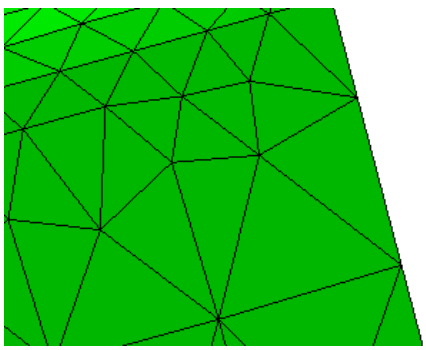


Figure 19-19 *Mesh created with the* ***Mesh edges suppression*** *check box cleared*

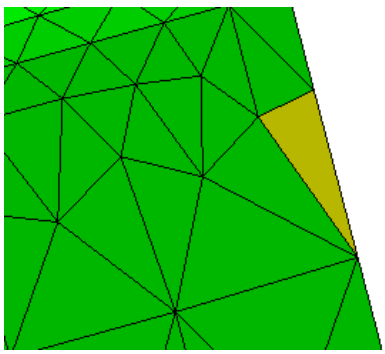


Figure 19-20 *Mesh created with the* ***Mesh edges suppression*** *check box selected*



Note

If all the edges of a surface are smaller than the value entered in the ***Geometry size limit*** *edit box then this surface will be ignored by default during the meshing process.*

If you select the **Global split** check box, you can generate the mesh with at least two element layers and also mesh edge on a curve connecting two nodes or vertices or a mesh edge on a face connecting two nodes or curves will be split automatically. The **Global interior size** check box remains inactive in case of octree triangle mesh type. You can specify the value of the minimum

size of the mesh to be refined due to sag specifications in the **Min. size for sag specs** edit box. Specify the maximum number of attempts needed to succeed in meshing in case of a complex geometry in the **Max. number of attempts** spinner. Choose **OK** to generate the mesh.

Tetrahedron Filler

Toolbar: Meshing Methods > Solid Meshing Methods > Tetrahedron Filler



The **Tetrahedron Filler** tool is used to generate tetrahedron mesh from the surface mesh. It creates a volume mesh from the surface mesh elements. Here the quality of 3D mesh depends on the quality of the 2D mesh.



Note

To generate a solid mesh, make sure the mesh surface is closed in terms of connectivity and the surface meshes has no intersection between them.

To generate 3D mesh on a surface body, choose the **Tetrahedron Filler** tool from the **Solid Meshing Method** sub-toolbar; the **Tetrahedron Filler** dialog box will be displayed, as shown in Figure 19-21; you are prompted to select a 2D mesh part to be filled. Select the required closed 2D mesh part; the selected meshed part will be displayed in the **MeshParts** area. You can select any type of 2D meshed parts, before that make sure the surface mesh is already generated on that 2D part. Select the **Linear** radio button if you want to generate linear meshed elements without intermediate nodes which is useful to fill a part with straight edges otherwise select the **Parabolic** radio button to generate the parabolic mesh elements with intermediate nodes which is useful to fill a part with curves from the **Element type** area of this dialog box. Specify the size progression factor in the **Size progression** edit box. By default 2 is specified is this edit box. Choose **OK** to generate the volume mesh.

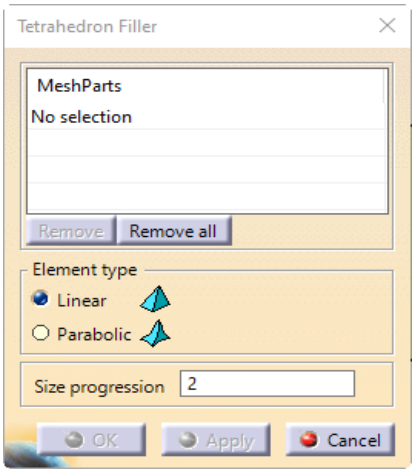


Figure 19-21 *The* ***Tetrahedron Filler*** *dialog box*

Figure 19-22 shows a meshed part created using the surface meshed feature and 2D surface mesh generated on it. Figure 19-23 shows the same part with volume mesh generated on it using the **Tetrahedron Filler** tool. Note that the parts shown in these figures are cutting views of the actual parts as it is mandatory to have closed surface to generate volume mesh using this tool.



Note

If the mesh is not visible once you have applied it then you need to update the mesh. To do so, expand the ***Nodes and Elements*** *sub node of the* ***Finite Element Model*** *node and right-click on the mesh that is to be updated. Choose* ***Update Mesh*** *or* ***Update All Mesh*** *from the shortcut menu displayed according to the requirement.*



Tip

To view the cutting view of an element, select the ***Cutting Plane*** *tool from the* ***Mesh Visualization Tools*** *toolbar.*

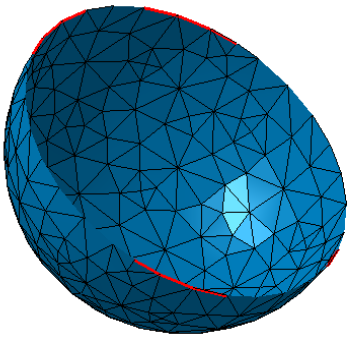


Figure 19-22 Sphere with 2D mesh

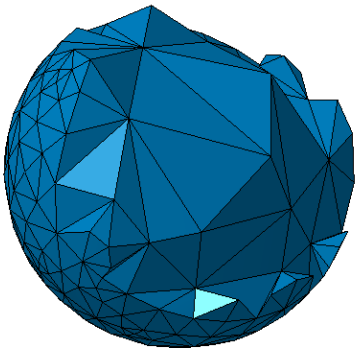


Figure 19-23 Sphere with volume mesh created using the ***Tetrahedron Filler***

Octree Tetrahedron Mesher

Toolbar: Meshing Methods > Solid Meshing Methods > Octree Tetrahedron Mesher



The **Octree Tetrahedron Mesher** tool is used to generate octree tetrahedron mesh elements on 3D parts. A tetrahedron mesh element have 4 vertices, 6 edges, and is bounded by triangular faces. In some cases a tetrahedral volume mesh can be generated automatically. Figure 19-24 shows an example of a tetrahedron mesh element in wireframe.

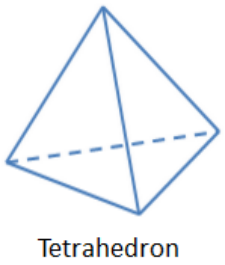


Figure 19-24 A Tetrahedron mesh element

To generate Octree Tetrahedron Mesh on a 3D part, choose the **Octree Tetrahedron Mesher** tool from the **Solid Meshing Methods** sub-toolbar; the **OCTREE Tetrahedron Mesh** dialog box will be displayed, as shown in Figure 19-25. Also, you will notice a mesh symbol displayed at the center of the part in the graphics area. The options in this dialog box has been discussed earlier.

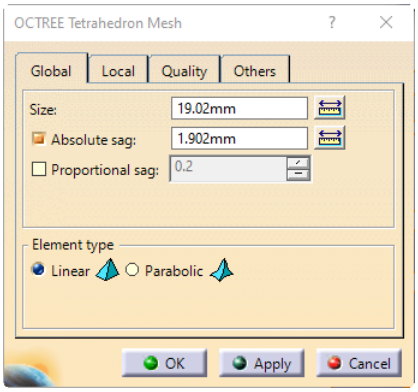


Figure 19-25 The ***OCTREE Tetrahedron Mesh*** *dialog box*

Sweep 3D

Toolbar: Meshing Methods > Solid Meshing Methods > Sweep 3D

The **Sweep 3D** tool is used to mesh the parts by generating hexahedron and wedge or prism elements. It generates a volume mesh from a predefined surface mesh.

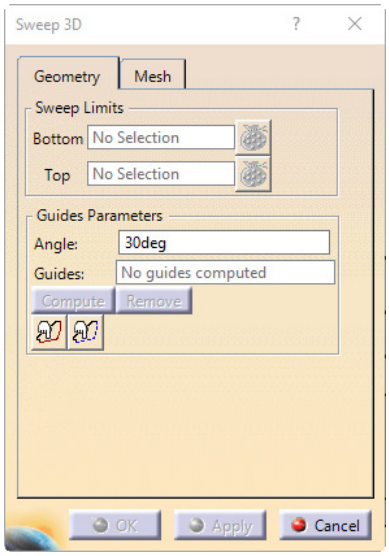


Figure 19-26 *The **Sweep 3D** dialog box*

To generate hexahedron or prism mesh on a 3D part, choose the **Sweep 3D** tool from the **Solid Meshing Methods** sub-toolbar; the **Sweep 3D** dialog box will be displayed, as shown in Figure 19-26. In this dialog box, the **Geometry** tab is chosen by default. Select the 3D part on which you want to generate the sweep 3D mesh but before that make sure a surface mesh is already generated on that 3D part; you are prompted to select the limits for the volume mesh. Select the bottom face of the part in the **Bottom** selection box and the top face in the **Top** selection box, refer to Figure 19-27 and Figure 19-28, respectively. These faces will work as the limits for the 3D elements. Now, specify the angle between two curves to compute the guides. The **Guides** display box displays the total number of guides computed. You can compute the guide curves by choosing the **Compute** button from the **Guides Parameters** area. You can also remove the selection of the guide curve by choosing the **Remove** button. On choosing the **Impose Guide** button from the **Guide parameters** area; the **Impose Guide** dialog box will be displayed and you can select a particular edge that you want to include into the guide computation in the **Support** selection box. Choose the **Exclude Guide** button; the **Exclude Guide** dialog box will be displayed. Select the required curve to exclude in the **Support** selection box.

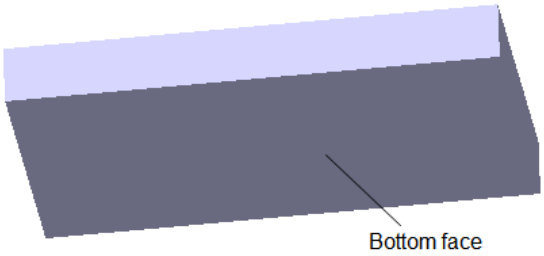


Figure 19-27 *Bottom face to be selected*

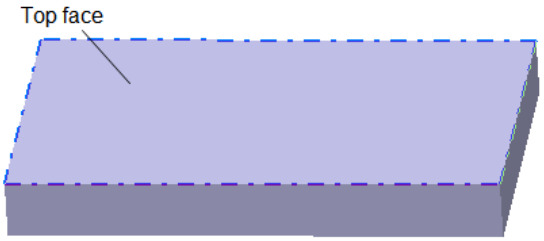


Figure 19-28 *Top face to be selected*

Choose the **Mesh** tab to specify mesh properties. Choose the **Linear** or **Parabolic** radio button from the **Element Type** area to generate linear or parabolic elements, respectively. Note that you cannot generate the parabolic mesh elements if the surface part mesh is built with linear elements. You can increase the mesh quality, if required, by selecting the **Internal and top mesh** check box from the **Smoothing** area.

You can select the distribution type from the **Type** drop-down list available in the **Distribution** area. If you select the **Uniform** option, the distance between all the distributed nodes of the elements will become same. If you select the **Arithmetic** and **Geometric** option, the distance between the nodes will be defined by arithmetical and geometrical distribution, respectively. On selecting these options, the **Size ratio** edit box and the **Symmetry** check box will also be activated. You can initialize the layers number field by choosing button next to this edit box. You can specify the value of common difference for an arithmetic distribution or the common ratio value for a geometric distribution in the **Size ratio** edit box. Select the **Symmetry** check box if you want a symmetric distribution. You can specify the required number of layers in the **Layers number** edit box.

You can capture the updated bottom face, top face, and lateral meshes to create the solid mesh with the help of the **Capture** area. Specify the tolerance value of the capture in the **Tolerance** edit box, which is the maximum distance for the mesh capture. Choose the **Initialize** button to automatically initialize the tolerance value of the capture. Select the **Preview** button to visualize the captured mesh. Choose **OK** to generate the mesh. Figure 19-29 shows mesh generated with the help of the **Sweep 3D** tool.

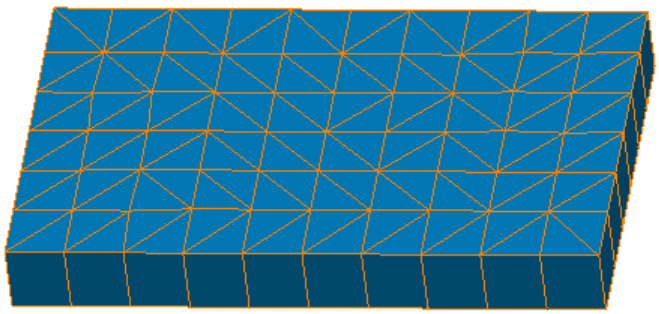


Figure 19-29 *Mesh created using the* ***Sweep 3D*** *tool*

MESH TRANSFORMATIONS

The **Advanced Meshing Tools** environment allows you to generate mesh on a 2D surface or on a 3D part as well as to transform it in the required direction with the help of some meshing tools provided. These tools works in case of 1D, 2D and 3D mesh parts only. By using these tools, if you want to create a 1D or 2D transformed mesh, a surface mesh is requires and for a 3D transformed mesh, a 3D mesh is required. The mesh transformation tools are discussed next.

Translation Mesher

Toolbar: Mesh Transformation > Transformation sub-toolbar> Translation Mesher



The **Translation Mesher** tool is used to create a translation mesh part from an already defined mesh part.

Choose the **Translation Mesher** tool from the **Transformation** sub-toolbar; the **Translation** dialog box will be displayed, as shown in Figure 19-30. Also, you will be prompted to select the parameters of the translation. Select the required part with surface mesh; the selected mesh part will be displayed in the **MeshParts** display box. Select the direction of translation in the **Direction** selection box, refer to Figure 19-31. Specify the distance of the direction in the **Distance** edit box.

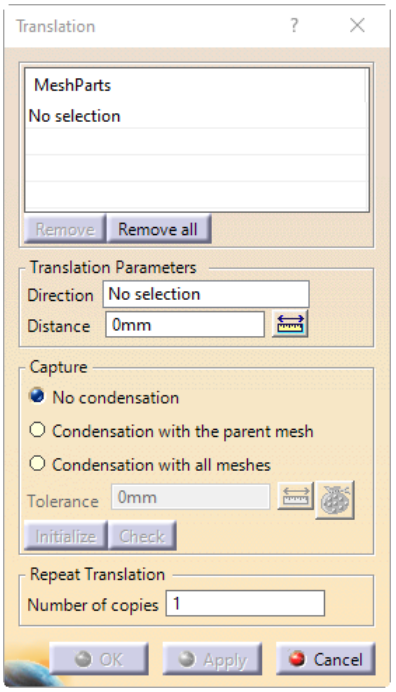


Figure 19-30 *The* ***Translation*** *dialog box*

The **Capture** area enables you to capture the updated mesh parts to condense modes. Select the **No condensation** radio button if you do not want to condense the nodes of the transformed mesh and the parent mesh. Select the **Condensation with the parent mesh** radio button to condense the nodes of the transformed mesh and the parent mesh. Select the **Condensation with all meshes** radio button to condense the nodes of the transformed mesh part with the updated mesh part.

Choose the **Initialize** button to automatically initialize the tolerance value of the condensation. Note that this button remains inactive when the **No condensation** radio button is selected. Choose the **Check** button to check the tolerance value if there are intersections or interferences between the transformed mesh. Note that this button gets activated on choosing the **Apply** button from this dialog box. Specify the required number of repeat translation copies in the **Number of copies** edit box of the **Repeat Translation** area and choose the **OK** button. Figure 19-32 shows the mesh after translation.

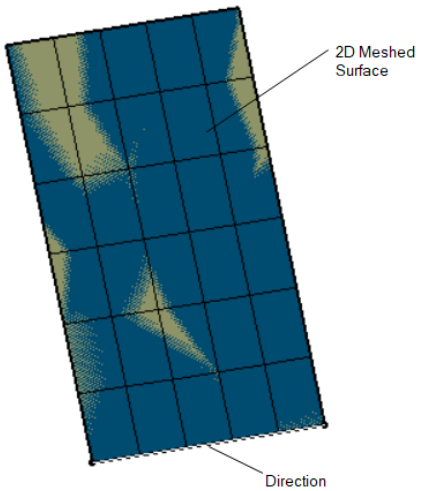


Figure 19-31 *Mesh part and direction to be translated*

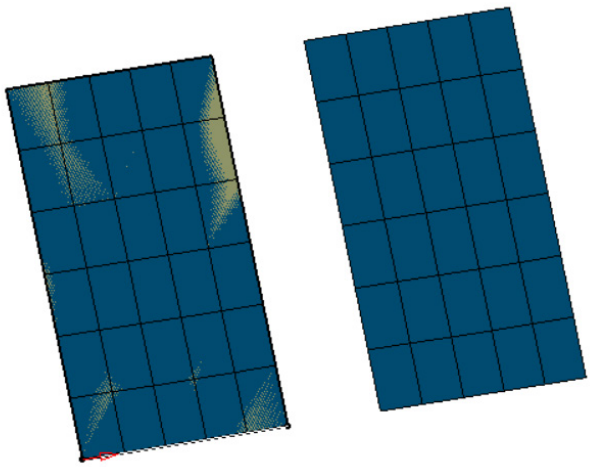


Figure 19-32 *Mesh after translation*

Rotation Mesher

Toolbar: Mesh Transformation > Transformation > Rotation Mesher



The **Rotation Mesher** tool is used to create a rotated mesh part from an already defined mesh part.

To create a rotated mesh part, choose the **Rotation Mesher** tool from the **Transformation** sub-toolbar; the **Rotation** dialog box will be displayed as shown in Figure 19-33. Also, you will be prompted to select the parameters. Select the required part with surface mesh; the selected mesh part will be displayed in the **MeshParts** display box. Select the axis of rotation in the **Axis** selection box and specify the angle of rotation in the **Angle** edit box in the **Rotation Parameters** area, refer to Figure 19-34. Specify the required parameters and choose **OK**. The resultant mesh after rotation is shown in Figure 19-35.



Note

This transformation follows the rules of associativity implying that if a load is applied to the parent mesh part, the same load will be applied automatically to the transformed mesh part.

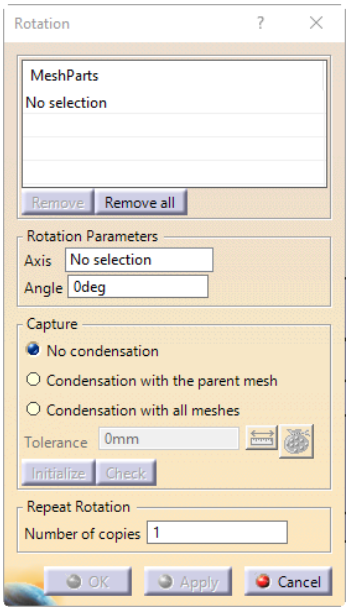


Figure 19-33 *The* ***Rotation*** *dialog box*

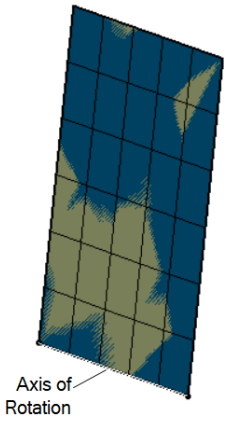


Figure 19-34 *Mesh part and axis of rotation*

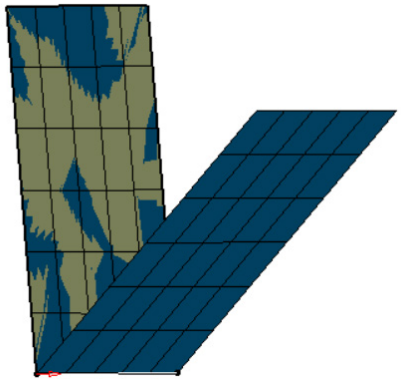


Figure 19-35 *Mesh after rotation*

Symmetry Mesher

Toolbar: Mesh Transformation > Transformation > Symmetry Mesher



The **Symmetry Mesher** tool is used to create a symmetry mesh part from an already defined parent mesh part.

To do so, choose the **Symmetry Mesher** tool from the **Transformation** sub-toolbar; the **Symmetry** dialog box will be displayed, as shown in Figure 19-36. Also, you will be prompted to select the required parameters. Select the part with surface mesh; the selected mesh part will be displayed in the **MeshParts** display box. Select the plane of symmetry in the **Plane** selection box in the **Symmetry Parameters** area, refer to Figure 19-37. Specify the required parameters and choose **OK**. The resultant mesh after symmetry is shown in Figure 19-38. You will notice that the symmetric mesh part will be added under the **Nodes and Elements** node of the specification tree.

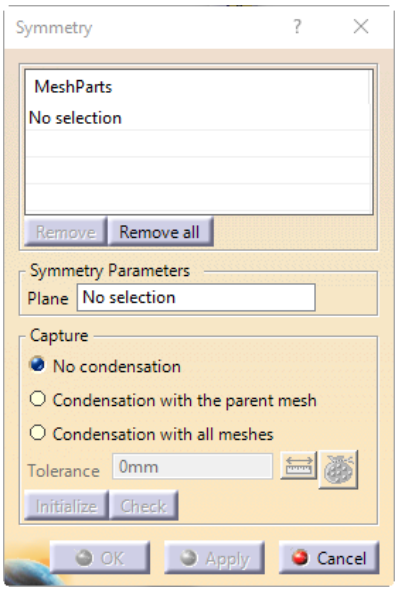


Figure 19-36 *The* ***Symmetry*** *dialog box*

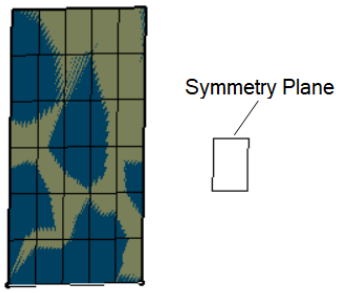


Figure 19-37 *Mesh part to be selected*

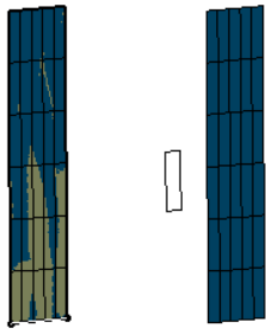


Figure 19-38 *Mesh after Symmetry*

Extrude Mesher with Translation

Toolbar: Mesh Transformation >Extrude Transformation > Extrude Mesher with Translation



The **Extrude Mesher with Translation** tool is used to extrude a surface mesh with translation. To do so, choose this tool from the **Extrude Transformation** sub-toolbar; the **Extrusion with Translation** dialog box will be displayed, as shown in Figure 19-39. Select

the mesh part. Select the axis of translation in the **Axis** selection box and specify the start and end limits of extrusion in the **Start** and **End** edit boxes, respectively.

Choose the node distribution type from the **Type** drop-down list. If you select the **Uniform** option then the distance between all the distributed nodes will be the same, if you select the **Arithmetic** option then the distance between the distributed nodes will be defined by an arithmetical distribution, and if you select the **Geometric** option, the distance between the distributed nodes will be defined by a geometrical distribution. You can specify the common difference value for an arithmetic distribution or the common ratio value for a geometric distribution in the **Size ratio** edit box. This edit box is activated only if you select the **Arithmetic** or **Geometric** as distribution type from the **Type** drop-down list. Specify the number of layers in the **Layers number** edit box. If you want to create a symmetrical distribution then you can select the **Symmetry** check box and choose **OK**. Figure 19-40 shows the mesh part and axis to be selected and Figure 19-41 shows resultant mesh extrusion with start limit as 20 mm and end limit as 100 mm.

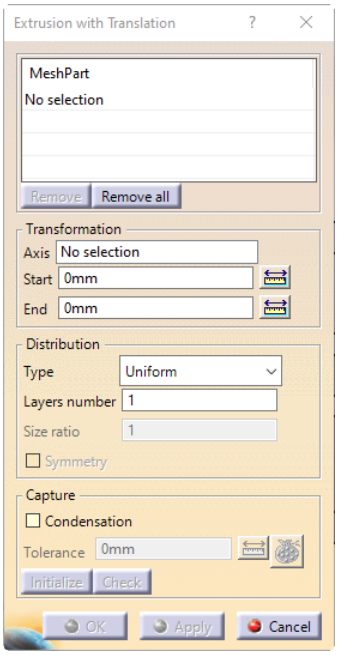


Figure 19-39 *The* ***Extrusion with Translation*** *dialog box*

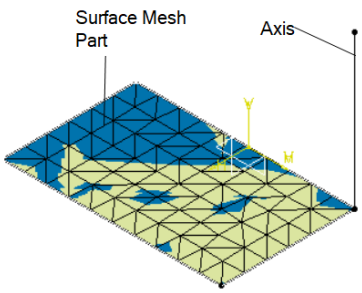


Figure 19-40 *Mesh part and axis of extrusion*

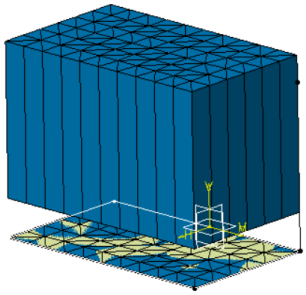


Figure 19-41 *Mesh after Extrusion*

Extrude Mesher with Rotation

Toolbar: Mesh Transformation >Extrude Transformation > Extrude Mesher with Rotation



The **Extrude Mesher with Rotation** tool is used to extrude a surface mesh with respect to a rotation axis. To do so, choose this tool from the **Extrude Transformation** sub-toolbar; the **Extrusion with Rotation** dialog box will be displayed, as shown in Figure 19-42. Select the surface mesh part. Select the axis of rotation in the **Axis** selection box and specify the start and end angles of extrusion in the **Start** and **End** edit boxes, respectively. Also, specify the required options and values from the **Distribution** area and choose **OK**. Figure 19-43 shows the mesh part and axis to be selected and Figure 19-44 shows resultant mesh rotation with start limit as 10 deg and end limit as 50 deg.

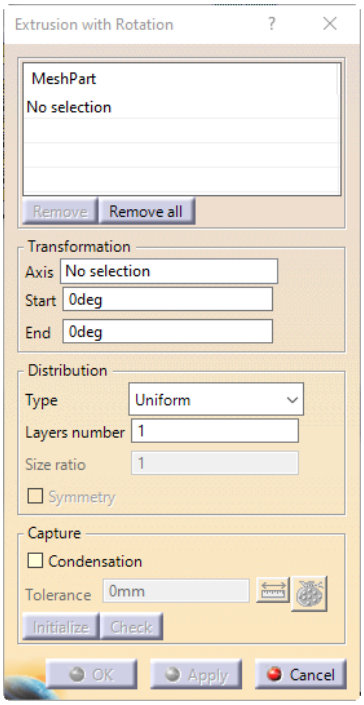


Figure 19-42 *The* ***Extrusion with Rotation*** *dialog box*

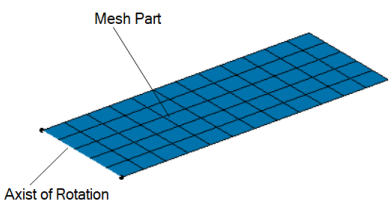


Figure 19-43 *Mesh part and axis of rotation*

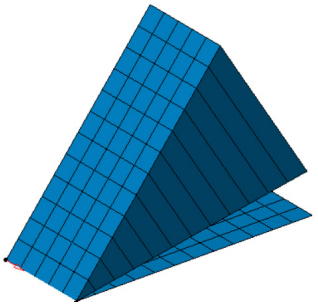


Figure 19-44 *Mesh after rotation*

Extrude Mesher with Symmetry

Toolbar: Mesh Transformation > Extrude Transformation > Extrude Mesher with Symmetry



The **Extrude Mesher with Symmetry** tool is used to extrude a surface mesh with respect to a plane of symmetry. To do so, choose the **Extrude Mesher with Symmetry** tool from the **Extrude Transformation** sub-toolbar; the **Extrusion with Symmetry** dialog box will be displayed, as shown in Figure 19-45. Select the surface mesh part. Select the plane of symmetry in the **Plane** selection box and choose the required options and then choose **OK**. Figure 19-46 shows the mesh part and plane to be selected and Figure 19-47 shows resultant mesh symmetry extrusion with uniform distribution and 13 layers.

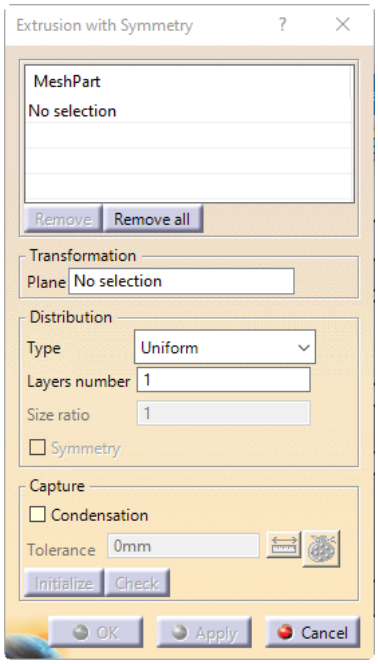


Figure 19-45 *The* ***Extrusion with Symmetry*** *dialog box*

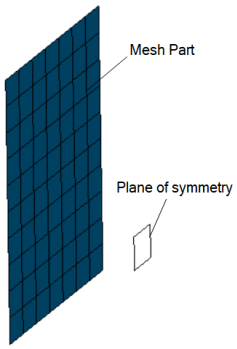


Figure 19-46 *Mesh part and plane of symmetry*

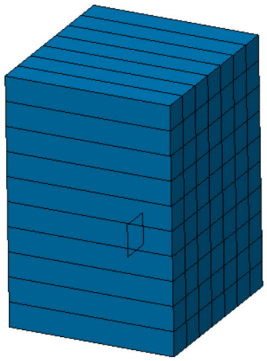


Figure 19-47 *Mesh after symmetry extrusion*

Extrude Mesher along Spine

Toolbar: Mesh Transformation > Extrude Transformation > Extrude Mesher along Spine



The **Extrude Mesher along Spine** tool is used to extrude a mesh along a specific spine curve. To do so, choose this tool from **Extrude Transformation** sub-toolbar; the **Extrusion along Spine** dialog box will be displayed, as shown in Figure 19-48. Select the mesh part and select the spine in the **Axis** selection box. Specify the start and end limits in the **Start** and **End** edit boxes, respectively. On selecting the **Anchor point** check box, the display box under it gets activated and you can specify the reference point of the spine for the extrusion in this display box. Next, choose **OK**. Figure 19-49 shows the 1D mesh part and spine to be selected and Figure 19-50 shows resultant mesh along the spine.

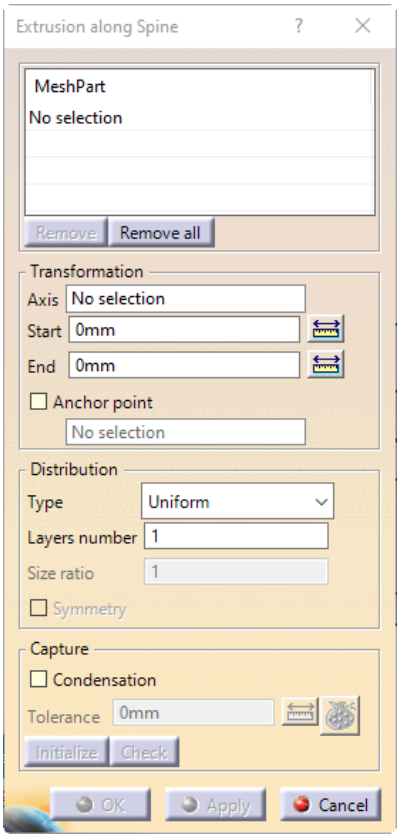


Figure 19-48 *The* ***Extrusion along Spine*** *dialog box*

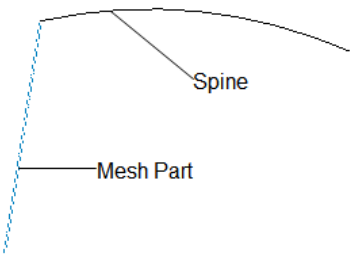


Figure 19-49 *Mesh part and spine to be selected*

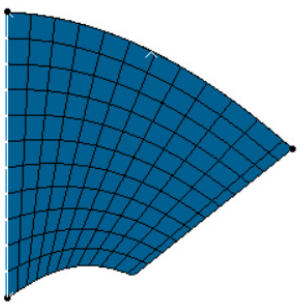


Figure 19-50 *Resultant mesh along spine*

Mesh Intersection and Interference

Toolbar: Mesh Analysis Tools > Intersections / Interferences



The **Intersections / Interferences** tool is used to check the intersections and the interferences between the parent mesh and transformed mesh.

To check this interference and intersection choose the **Intersections/ Interferences** tool from the **Mesh Analysis Tools** toolbar; the **Interference Check** dialog box will be displayed as shown in Figure 19-51. Specify the required clearance value in the **Clearance** edit box. Choose the

required option from the **Mesh Parts** drop-down If you want to perform dynamic check, select the **Dynamic check** check box and choose the **Apply** button; the elements in intersection and interference will be displayed in the **Number of elements** area. On the basis of this result, you can edit the mesh by double clicking on it from the specification tree. Choose the **Close** button to close the dialog box.

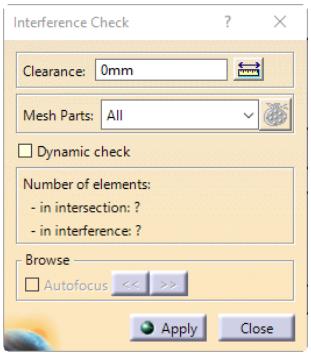


Figure 19-51 *The* ***Interference Check*** *dialog box*

MESH OPERATORS

In Advanced Meshing Tools environment, you can perform various operation on the mesh created with the help of certain tools provided. These tools are discussed below.

Mesh Offset

Toolbar: Mesh Operators > Mesh Offset



The **Mesh Offset** tool is used to offset a 2D mesh at a required offset value. To do so, choose this tool from the **Mesh Operators** toolbar; you will be prompted to select a meshed part. Select the meshed part, if not selected by default; the **Mesh Offset** dialog box will be displayed, as shown in Figure 19-52. Select the 2D mesh, the mesh offset direction will be displayed, as shown in Figure 19-53. You can reverse the direction of offset by clicking on the arrows displayed on the selected mesh part, refer to Figure 19-53. Specify the offset value in the **Offset** edit box and choose **OK**.

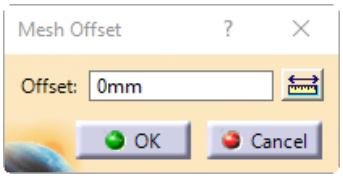


Figure 19-52 *The* ***Mesh Offset*** *dialog box*

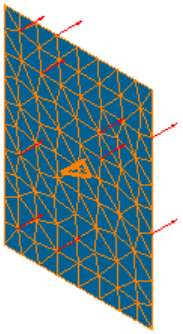


Figure 19-53 *The mesh selected for offset*



Note

Sometimes, for complex reference surface mesh, the offset surface mesh may not be created. In such cases, you need to reduce the mesh offset value or modify the initial geometry.

Split Quads

Toolbar: Mesh Operators > Split Quads



The **Split Quads** tool is used to split quadrangle mesh elements into triangle mesh elements. You can only split the linear quadrangle elements using this tool.

To do so, choose this tool from the **Mesh Operators** toolbar; you will be prompted to select the mesh part to be split. Select the mesh part to be split; the **Split Quads** dialog box will be displayed, as shown in Figure 19-54. Select the **Shortest diagonal** radio button to choose the shortest distance between both diagonals as split. Select the **Best quality** radio button to choose the diagonal with the best quality elements as split. Select the required criteria from this dialog box and choose **OK**. Figure 19-55 displays a quad mesh selected from the **Split Quads** dialog box, splited with the shortest diagonal option.

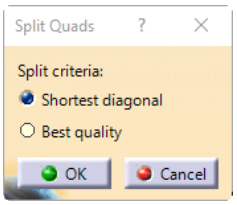


Figure 19-54 *The* ***Split Quads*** *dialog box*

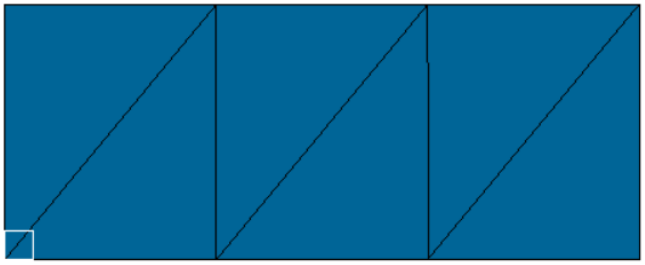


Figure 19-55 *Quads splitted with shortest diagonal option*

GENERATIVE STRUCTURAL ANALYSIS WORKBENCH

The Generative Part Structural Analysis and the Generative Assembly Structural Analysis allow you to quickly perform mechanical analysis for 3D parts and assembly and to accurately calculate the displacements and stresses within the part under a variety of loading conditions. They also allow the vibration characteristics of parts to be assessed by calculating the natural frequencies and the associated mode shapes. This workbench also allows the user to perform an analysis on a volume part, any surface part, and any wireframe geometry.

Working with Generative Structural Analysis Workbench

There are two methods to invoke the **Generative Structural Analysis** workbench of CATIA V5. The primary method is by choosing **Start > Analysis and Simulation > Generative Structural Analysis** from the menu bar. The other method of invoking the **Generative Structural Analysis** workbench is by choosing **File > New** from the menubar. On doing so, the **New** dialog box is displayed. In this dialog box, select **Analysis** and choose **OK**.

On invoking the **Generative Structural Analysis** workbench, a new analysis environment gets started in it. The screen display of the CATIA V5 after invoking the **Generative Structural Analysis** workbench is shown in Figure 19-56. You will notice that the toolbars related to this workbench will also be displayed. The tools in these toolbars are discussed later in this chapter.

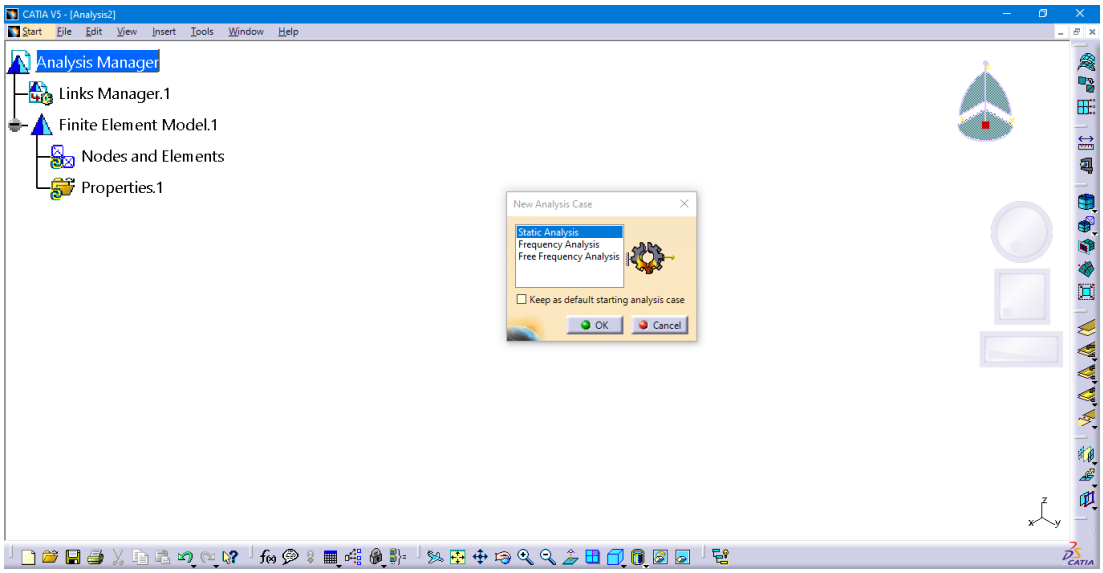


Figure 19-56 *Screen displayed after starting a new file in the Generative Structural Analysis workbench*

APPLYING PHYSICAL PROPERTIES

You can add or change the physical properties of a shape design by selecting a meshed wireframe geometry. There are three types of physical properties in model manager: 1D property, 2D property, and 3D property.

Adding 1D Physical Properties

Toolbar: Model Manager > 1D property > 1D property



The **1D Property** tool allows you to add 1D physical properties to a shape designed by selecting a meshed wireframe geometry.

To add 1D physical property, choose the **1D Property** tool from the **Model Manager** toolbar; the **1D Property** dialog box will be displayed, as shown in Figure 19-57. Also, you will be prompted to select a mesh part or a 1D geometry. Select the required geometry; its name will be displayed in the **Supports** selection box. The **Material** selection box displays the material applied to the part. Select the **User-defined material** check box if you want to select an isotropic material defined by you. Select the section type from the **Type** drop down list. If you choose the **Component edition** button; the **Beam Definition** dialog box will be displayed. You can specify the required parameters of the beam and can change the orientation of a geometry in this dialog box. You can select a sketched line, a point, or an edge in the **Orientation geometry** selection box as directional reference.

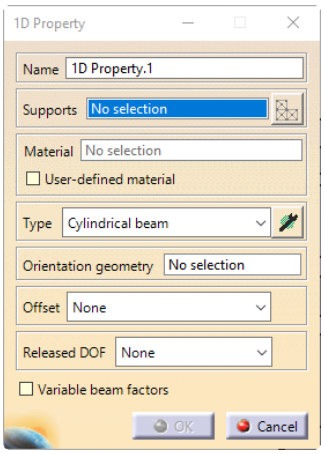


Figure 19-57 *The **1D Property** dialog box*

The **Offset** drop-down list lets you specify the offset value at each end of the beam. By default, the **None** option is selected in this drop down, hence no offset will be defined. If you choose the **Customized** option, the **Component edition** button will be displayed next to the option. On choosing this button; the **Offset Definition** dialog box will be displayed. You can specify the required parameters of the beam in this dialog box for both the ends of the beam.

The **Released DOF** drop-down list enables you to release or detain the degree of freedom. By default, the **None** option is selected, hence there will be no change in degree of freedom. If you choose the **Customized** option, the **Component edition** button will be displayed next to the option. On choosing this button, the **Released Degrees of Freedom** dialog box will be displayed. You can select the required degree of freedom for start and end points of the beam using this dialog box.

The **Variable beam factors** check box allows you to create a linear approximation of variable cross section beams. On selecting this check box, the **Multiplication Factors on Extremities** area will be displayed under this check box. By using options in this area, you can define the scaling factor on each side of the beam. After specifying the required parameters, choose **OK**. Figure 19-58 displays a line body converted into a cylindrical section beam with the **1D Property** tool.

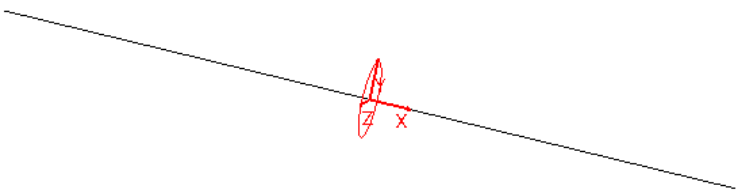


Figure 19-58 *A cylindrical beam with 1D property*



Note

You can select the meshed element only from the specification tree and not from the graphics window.

Adding 2D Physical Properties

Toolbar: Model Manager > 2D property > 2D property



The **2D Property** tool allows you to add 2D physical properties to a shape design geometry.

To add 2D physical properties to a shape design, choose the **2D Property** tool from the **Model Manager** toolbar; the **2D Property** dialog box will be displayed, as shown in Figure 19-59. Also, you will be prompted to select a 2D meshed surface. Select the surface and specify the required thickness in the **Thickness** edit box. After specifying the thickness, specify the offset value in the **Offset** edit box and choose **OK**. Figure 19-60 displays a surface body on applying the 2D property with an offset of 4mm.

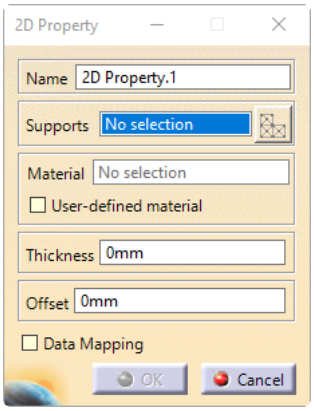


*Figure 19-59 The **2D Property** dialog box*

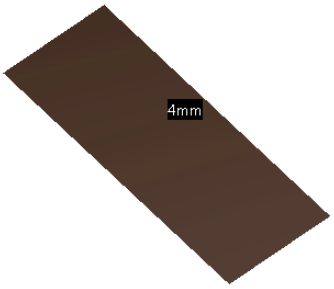


Figure 19-60 A surface with 2D property and 4mm Offset

Adding 3D Physical Properties

Toolbar: Model Manager > 3D property



The **3D Property** tool allows you to add 3D physical properties to a shape design geometry.

To add 3D physical properties to a shape design, choose the **3D Property** tool from the **Model Manager** toolbar; the **3D Property** dialog box will be displayed, as shown in Figure 19-61. Also, you are prompted to select a part or mesh part. Select the part; the name of the selected part will be displayed in the **Supports** selection box. You can define the orientation by using the **Orientation** drop-down list. By default, the **None** option is selected in this drop-down list.

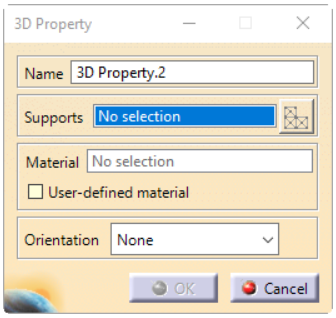


*Figure 19-61 The **3D Property** dialog box*

If you choose the **By Axis** option, the **Component edition** button will be displayed next to the option. On choosing this button, the **Orientation Definition** dialog box will be displayed. Choose the required axis system and choose **OK** from this dialog box. If you choose the **By Surface** option, the **Component editor** button will be displayed next to the option. On choosing this button; the **Orientation Definition** dialog box will be displayed. Select the required surface for orientation in the **Surface** selection box. Also, you can select a line or any edge in the **Direction** selection box for the direction of orientation and choose **OK** from this dialog box. After specifying the required options, choose **OK** from the **3D property** dialog box.

CREATING VIRTUAL PARTS

Virtual parts represent bodies for which no geometry model is available but which play an important role in the structural analysis of a single part or an assembly system. Virtual parts are used to transmit action at a certain distance.

Applying Smooth Virtual Part

Toolbar: Virtual Parts > Virtual Parts > Smooth Virtual Part



A smooth virtual part is a rigid body which connects a specified point to a specified part geometry. It behaves as a mass-less rigid object which will softly transmit masses, restraints, or loads applied at the geometry.

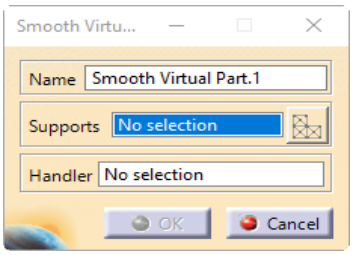


*Figure 19-62 The **Smooth Virtual Part** dialog box*

To apply smooth virtual part, choose the **Smooth Virtual Part** tool from the **Virtual Parts** sub-toolbar; the **Smooth Virtual Part** dialog box will be displayed, as shown in Figure 19-62. Select a face or a particular entity as support; its name will be displayed in the **Supports** selection box. Select the required handler point in the **Handler** selection box and choose **OK**. Note that a handler point is that point upto which the virtual part will be created. Figure 19-63 displays a part with smooth virtual part applied on a part.

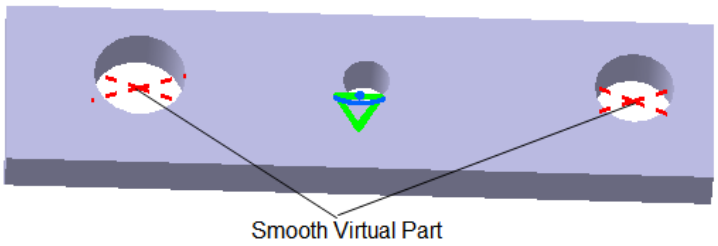


Figure 19-63 The smooth virtual part applied on a part

Applying Contact Virtual Part

Toolbar: Virtual Parts > Virtual Parts > Contact Virtual Part



A contact virtual part is also a rigid body which connects a specific point to specific part geometry. It behaves as a mass-less rigid object which will softly transmit masses, restraints, or loads applied at the geometry.

To apply contact virtual part, choose the **Contact Virtual Part** tool from the **Virtual Parts** sub-toolbar; the **Contact Virtual Part** dialog box will be displayed, as shown in Figure 19-64. Select a face or a particular entity as support; its name will be displayed in the **Supports** selection box. Select the required handler point in the **Handler** selection box. Specify the required clearance value in the **Clearance** edit box and choose **OK**. Figure 19-65 displays a part with contact virtual part applied on a part.

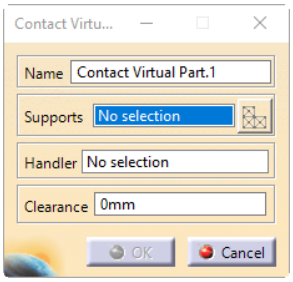


*Figure 19-64 The **Contact Virtual Part** dialog box*

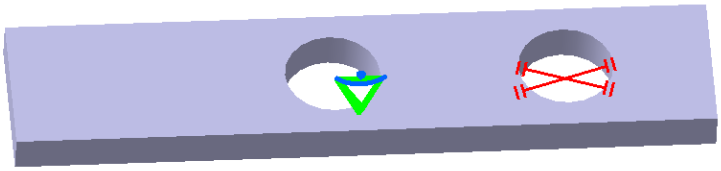


Figure 19-65 *The contact virtual part applied on a part*

Note

When several virtual parts share same handler point then only one finite element node will be generated.

Applying Rigid Virtual Part

Toolbar: Virtual Parts > Virtual Parts > Rigid Virtual Part

A rigid virtual part is a rigid body which connects a specific point to some specific part geometry by stiffening the deformable part to which it is attached. It behaves as a mass-less rigid object which will softly transmit masses, restraints, or loads applied at the geometry.

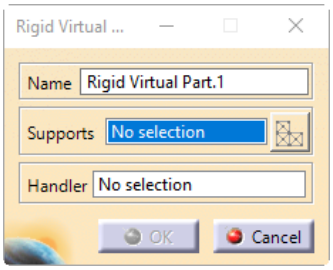


Figure 19-66 *The* ***Rigid Virtual Part*** *dialog box*

To apply rigid virtual part, choose the **Rigid Virtual Part** tool from the **Virtual Parts** sub-toolbar; the **Rigid Virtual Part** dialog box will be displayed, as shown in Figure 19-66. Select a face or a particular entity as support; its name will be displayed in the **Supports** selection box. Select the required handler point in the **Handler** selection box and choose **OK**. Figure 19-67 displays a part with rigid virtual part applied on a part.

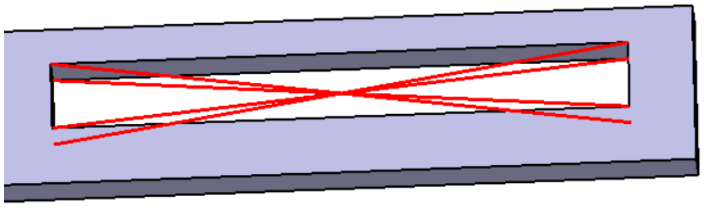


Figure 19-67 *The rigid virtual part applied on a part*

Applying Rigid Spring Virtual Part

Toolbar: Virtual Parts > Virtual Parts > Rigid Spring Virtual Part

A rigid spring virtual part is an elastic body which connects a specific point to a specific geometry by stiffening the deformable part to which it is attached, It behaves as a six degree of freedom spring in series with a mass-less rigid body which will stiffly transmit masses, restraints, or loads applied at the handle point.

To apply rigid spring virtual part, choose the **Rigid Spring Virtual Part** tool from the **Virtual Parts** sub-toolbar; the **Rigid Spring Virtual Part** dialog box will be displayed, as shown in Figure 19-68. Select a face or a particular entity as support; its name will be displayed in the **Supports** selection box. Select the required handler point in the **Handler** selection box.

You can select the required type of axis system from the **Type** drop-down list in the **Axis System** area of this dialog box. By default, **Global** is selected. As a result, the resultant force vectors will be converted as relative to the global coordinate system. If you select the **User** axis system, the components of the resultant force vector will be converted as relative to the specified coordinate system. Also, on selecting this option from the **Type** drop-down list; the **Current axis** selection box will be displayed. Select the required existing axis in this selection box.

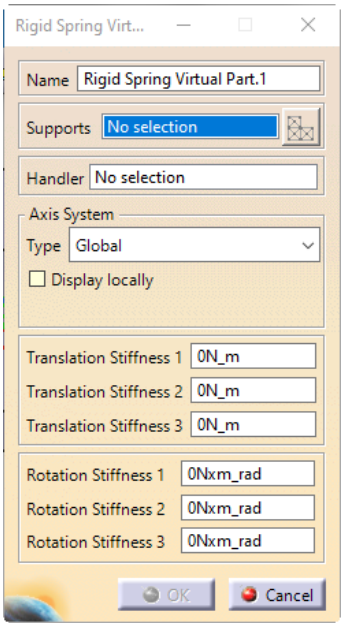


Figure 19-68 *The* ***Rigid Spring Virtual Part*** *dialog box*

Specify values for the six degrees of freedom spring constants in the required **Translation Stiffness** and **Rotation Stiffness** edit boxes and choose **OK**. Figure 19-69 displays a part with rigid spring virtual part applied on a part.

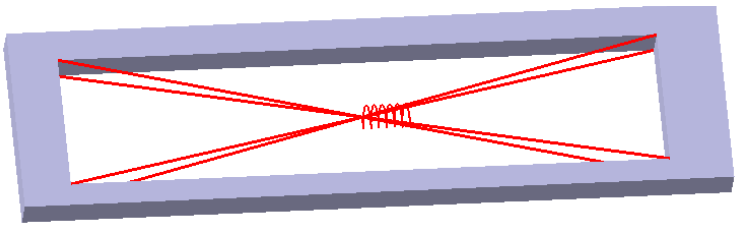


Figure 19-69 *The rigid spring virtual part applied on a part*

Applying Smooth Spring Virtual Part

Toolbar: Virtual Parts > Virtual Parts > Smooth Spring Virtual Part



A smooth spring virtual part is an elastic body which connects a specific point to a specific geometry by stiffening the deformable part to which it is attached. It also behaves as a six degree of freedom spring in series with a mass-less rigid body which will softly transmit masses, restraints, or loads applied at the handle point.

To apply smooth spring virtual part, choose the **Smooth Spring Virtual Part** tool from the **Virtual Parts** sub-toolbar; the **Smooth Spring Virtual Part** dialog box will be displayed, as shown in Figure 19-70. Select a face or a particular entity as support; its name will be displayed in the **Supports** selection box. Select the required handler point in the **Handler** selection box. Specify the required axis system and values of degrees of freedom and choose **OK**. Figure 19-71 displays a part with smooth spring virtual part applied on a part.

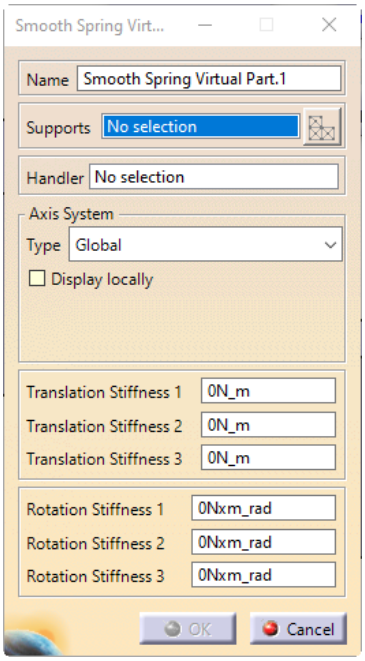


Figure 19-70 *The* ***Smooth Spring Virtual Part*** *dialog box*

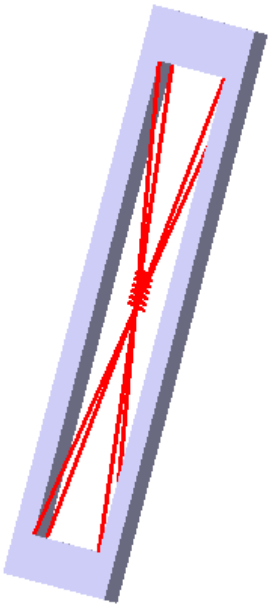


Figure 19-71 *The smooth spring virtual part applied on a part*

Applying Periodicity Conditions

Toolbar: Virtual Parts > Periodicity Condition



The periodicity conditions enable you to perform analysis on a solid section of a periodic part. This solid section represents a cyclic period of the entire part.

To apply this condition, choose the **Periodicity Condition** tool from the **Virtual Parts** toolbar; the **Periodicity Condition** dialog box will be displayed, as shown in Figure 19-72. Choose the faces of the part to be used for generating periodicity of the part section, refer to Figure 19-73 and Figure 19-74. Choose **OK** to add periodicity conditions.

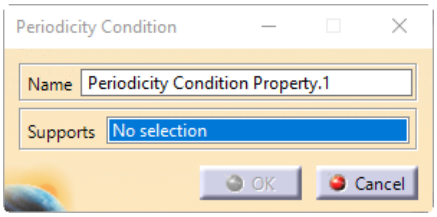


Figure 19-72 *The* ***Periodicity Condition*** *dialog box*

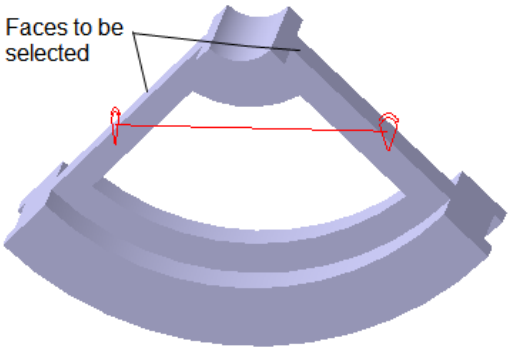


Figure 19-73 *The periodicity condition applied on a cyclic part*

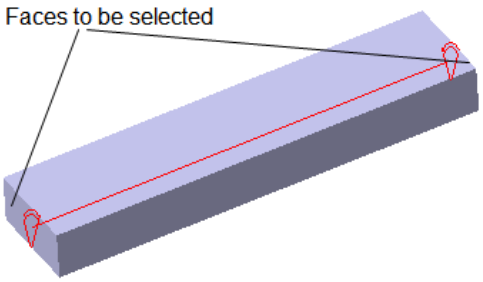


Figure 19-74 *The periodicity condition applied on a regular part*

Note

To use periodicity conditions, make sure the geometry is continuous as well as the applied restraints and loads are periodic. Also, the geometry needs to be regular at the place the section is created.

APPLYING RESTRAINS

The **Generative Structural Analysis** workbench of CATIA V5 allows you to restrict the degrees of freedom of a component or a group of components. This workbench provides various types of restraints to restrict the degree of freedom of a component. These restraints are discussed next.

Applying Clamp

Toolbar: Restraints > Clamp

In the **Generative Structural Analysis** workbench, the **Clamp** tool is used to restrict the degree of freedom of a part with respect to the surface or curve geometries on which clamp is applied.

To apply clamp to a part or geometry, choose the **Clamp** tool from the **Restraints** toolbar; the **Clamp** dialog box will be displayed, as shown in Figure 19-75. Select a face of the part; its name will be displayed in the Supports selection box. Next specify the name of the clamp in the **Name** edit box of this dialog box and choose **OK**. The clamp restrain gets added, as shown in Figure 19-76.

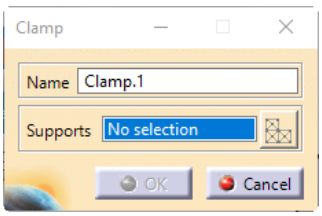


Figure 19-75 *The* ***Clamp*** *dialog box*

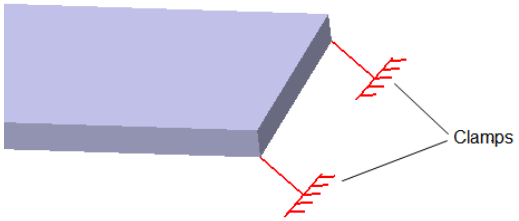


Figure 19-76 *The* ***Clamp*** *added*

On selecting the required geometry to apply clamp, the **Select Mesh Parts** button next to the **Supports** selection box gets activated. This button allows you to choose one mesh part from all the activated or deactivated mesh parts associated to this support. On selecting this button; the **Mesh Parts** dialog box is displayed. This dialog box displays the mesh parts selected by you.

Applying Surface Slider

Toolbar: Restraints > Mechanical Restraints > Surface Slider



The **Surface Slider** tool is used to add a slider constraint joint between two surfaces. Out of which one of the surfaces is a rigid surface. Hence, one direction of the surface is restrained. It can only slide along the coinciding rigid surface.

To apply surface slider restraint to a part or geometry, choose the **Surface Slider** tool from the **Mechanical Restraints** sub-toolbar; the **Surface Slider** dialog box will be displayed, as shown in Figure 19-77. Select a face of the part; its name will be displayed in the **Supports** selection box. Next specify the name of the clamp in the **Name** edit box of this dialog box and choose **OK**. The surface slider restraint gets added as shown in Figure 19-78. Also, you will notice that the **Surface Slider** sub-node gets added under the **Restraints** node of the **Analysis Manager** tree.

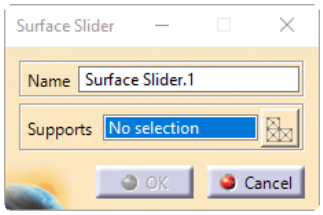


Figure 19-77 *The* ***Surface Slider*** *dialog box*

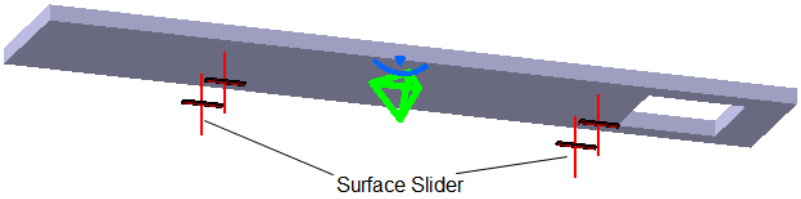


Figure 19-78 *The* ***surface slider*** *restraint*

Applying Slider

Toolbar: Restraints > Mechanical Restraints > Slider



The **Slider** tool is used to add a slider restraint to the handle points of virtual parts. It forces them to slide along a given axis so that one of the two handle points and the sliding axis are fixed.

To apply slider restrain to a part or geometry, first you need to create a virtual part as this tool handles only the points of the virtual parts. The creation of virtual parts is later discussed in this chapter. To apply Slider restrain, choose the **Slider** tool from the **Mechanical Restraints** sub-toolbar; the **Slider** dialog box is displayed as shown in the Figure 19-79. Select the created virtual part as a support in the **Support** selection box. On doing so, a small cuboid gets attached to the virtual part indicating a slider restraint. Select the type of axis system with respect to which the slider restraint will be applied as **Global** or **User** from the **Type** drop-down list in the **Axis System** area of this dialog box.

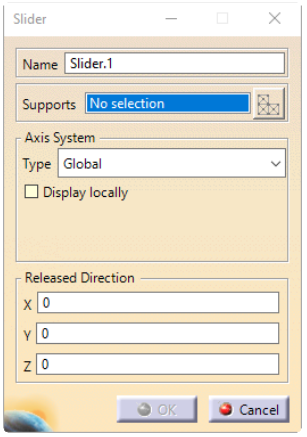


Figure 19-79 *The* ***Slider*** *dialog box*

On selecting the **User** option from the **Type** drop-down list, the **Current axis** selection box and the **Local orientation** drop-down list gets displayed in the **Axis System** area. You can select the axis of the reference axis system in the **Current axis** selection box. Note that this reference axis system must be created in the part document. You can select the orientation of the user axis system as **Cartesian**, **Cylindrical** or **Spherical** from the **Local orientation** drop-down list. You can enter the values of the sliding direction relative to the axis system in the **X**, **Y** and **Z** edit boxes of the **Released Direction** area. You will notice that a **Slider** sub-node gets added under the **Restraints** node of the **Analysis Manager** tree. Figure 19-80 shows a master rod with slider restraint and a smooth virtual part.

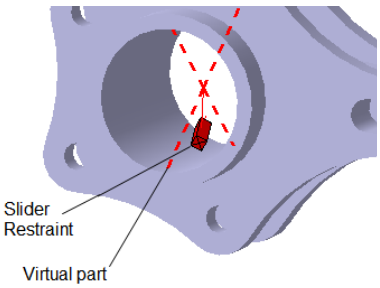


Figure 19-80 *The slider restraint*

Applying Sliding Pivot

Toolbar: Restraints > Mechanical Restraints > Slider Pivot



The **Slider Pivot** tool is used to add slider restraints to a cylindrical part to the handle points of virtual parts. It forces them to constrain along a given axis so that they translate and rotate along that axis.

To apply the sliding pivot restraint to a part or geometry, choose the **Sliding Pivot** tool from the **Mechanical Restraints** sub-toolbar; the **Sliding Pivot** dialog box will be displayed, as shown in Figure 19-81. Select the required virtual part with respect to which you want to apply this restrain in the **Supports** selection box. Next specify the name of the restraint in the **Name** edit box of this dialog box and choose **OK**. On doing so, a small red cylindrical object gets attached to the virtual part indicating slider pivot restraint, as shown in Figure 19-82. Also, you will notice that a **Sliding Pivot** sub-node gets added under the **Restraints** node of the **Analysis Manager** tree. The options in the **Axis System** and **Released Direction** areas have already been discussed.

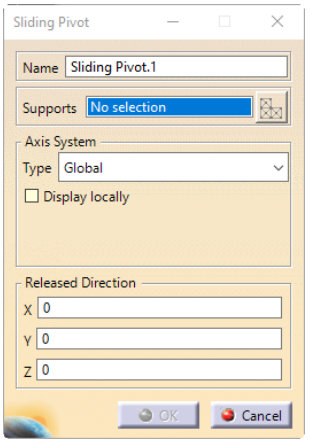


Figure 19-81 *The* ***Sliding Pivot*** *dialog box*

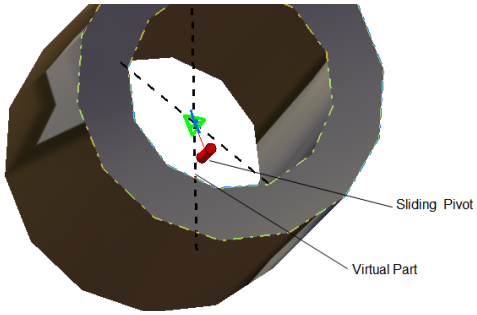


Figure 19-82 *The sliding pivot restrain*

Applying Ball Joint

Toolbar: Restraints > Mechanical Restraints > Ball Joint

The **Ball Joint** tool adds a spherical joint restraint to the points of virtual parts constraining the part to rotate around a fixed point.

To apply ball joint restraint to a part or geometry, choose the **Ball Joint** tool from the **Mechanical Restraints** sub-toolbar; the **Ball Joint** dialog box will be displayed, as shown in Figure 19-83. Select the virtual part with respect to which you want to apply this restraint in the **Supports** selection box and specify the name of the restraint in the **Name** edit box of this dialog box and choose **OK**. On doing so, a small ball joint icon gets attached to the virtual part indicating the ball joint restraint, as shown in Figure 19-84.

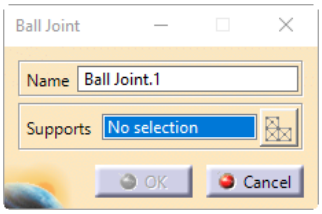


Figure 19-83 *The* ***Ball Joint*** *dialog box*

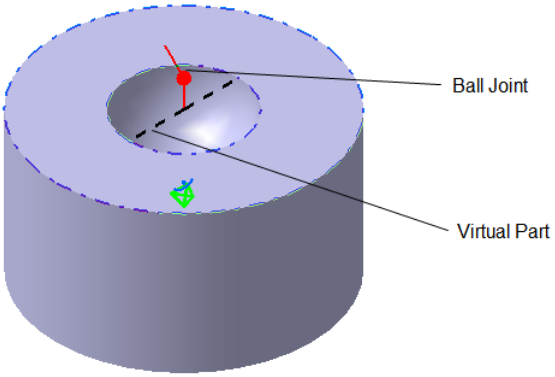


Figure 19-84 *The ball joint restraint*

Applying Pivot

Toolbar: Restraints > Mechanical Restraints > Pivot

The **Pivot** tool adds a conical join restraint to the points of virtual parts. It constraints the point to rotate around a given fixed axis. This joint is also known as hinge joint which allows a relative rotation between two points.

To apply the pivot restraint to a part or geometry, choose the **Pivot** tool from the **Mechanical Restraints** sub-toolbar; the **Pivot** dialog box will be displayed, as shown in Figure 19-85. Select the virtual part with respect to which you want to apply this restraint in the **Supports** selection box and specify the name of the restraint in the **Name** edit box of this dialog box and choose **OK**. On doing so, a pivot icon gets attached to the virtual part which shows the pivot restraint, as shown in Figure 19-86.

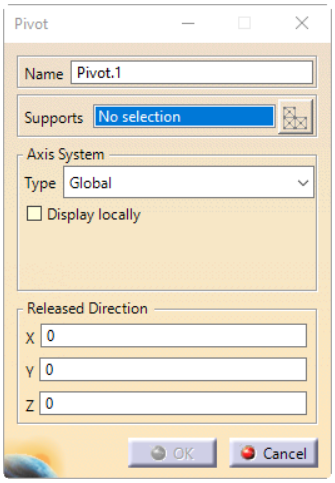


Figure 19-85 *The **Pivot** dialog box*

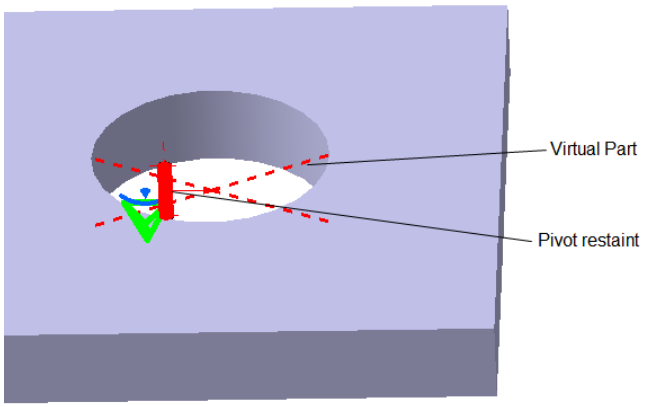


Figure 19-86 *The pivot restraint*

Applying User-defined Restraint

Toolbar: Restraints > Advance Restraints > User-defined Restraint



The user-defined restraints are generic restraints which allow you to fix any combination of nodal degrees of freedom which is available on an arbitrary geometry.

To apply the user-defined restraint to a part or geometry, choose the **User-defined Restraint** tool from the **Advance Restraints** sub-toolbar of the **Restraints** toolbar; the **User-defined Restraint** dialog box will be displayed, as shown in Figure 19-87. Select the virtual part with respect to which you want to apply this restraint in the **Supports** selection box and specify the name of the restraint in the **Name** edit box of this dialog box. Next, choose **OK**; a small user-defined restraint icon will get attached to the virtual part which shows the user-defined restraint, as shown in Figure 19-88.

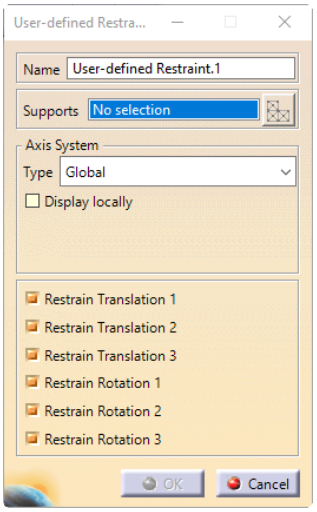


Figure 19-87 *The **User-defined Restraint** dialog box*

You can also select the desired type of axis system from the **Type** drop-down available in the **Axis System** area of this dialog box. By default, the **Global** axis system is selected in this drop-down list. As a result, the degree of freedom directions will translate as relative to the fixed global coordinate system. Select the **Implicit** axis system from the **Type**

drop-down to translate the degree of freedom directions as relative to a local variable coordinate system whose type will depend on the support geometry. On selecting the **User** axis system from this drop-down list, the direction of the degree of freedom will be relative to the specified axis system. Their translation will further depend on the axis type and orientations that can be selected from the **Local orientation** drop-down and will be available only on selecting this option.

Also, you can restrain the degree of freedom by checking and clearing the **Restrain Translation** and **Restrain Rotation** check boxes from the **User-defined Restraint** dialog box.

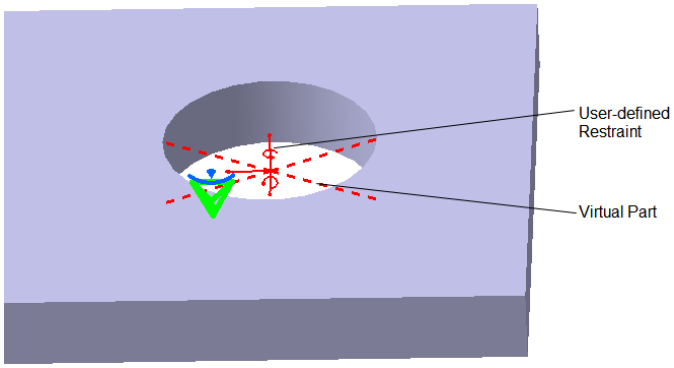


Figure 19-88 *The user-defined restraint applied to a part*

Applying Isostatic Restraint

Toolbar: Restraints > Advance Restraints > Isostatic Restraint



The **Isostatic Restraints** are statically definite restraints, that allow you to simply restrain a body to a particular location.

To apply the isostatic restraints to a part or geometry, choose the **Isostatic Restraint** tool from the **Advance Restraints** sub-toolbar; the **Isostatic Restraint** dialog box will be displayed, as shown in Figure 19-89.

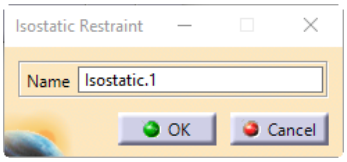


Figure 19-89 *The* ***Isostatic Restraint*** *dialog box*

Select the part that is to be restrained statically and specify the name of restraint in the **Name** edit box of the **Isostatic Restraint** dialog box. Next, choose **OK**; the **Isostatic Restraints** will be added to the specification tree with the name specified in the **Isostatic Restraint** dialog box.

APPLYING LOADS

The main goal of Generative Structural Analysis is to examine how a structure or component will respond to certain loading conditions. Specifying the proper loading conditions is, therefore, a key step in the analysis. The Generative Structural Analysis environment of CATIA allows you to apply loads on the model in different ways.

Applying Pressure

Toolbar: Loads > Pressure



The pressure is the force applied per unit area in a direction perpendicular to the surface of the model. They are intensive loads that represent scalar pressure fields applied to the surface of a geometry.

To apply the pressure load on a surface, choose the **Pressure** tool from the **Loads** toolbar; the **Pressure** dialog box will be displayed, as shown in Figure 19-90 and you will be prompted to select a face. Select the face from the geometry area; the name of selected face will be displayed in the **Supports** selection box. Specify the pressure value in the **Pressure** edit box. By choosing the **Data Mapping** check box, you can use the data mapping functionality in which you can import load descriptions in the form of text or Excel files. Choose **OK** to apply pressure. Figure 19-91 shows the surface to be selected to apply pressure and Figure 19-92 shows the part with pressure applied to the selected surface.

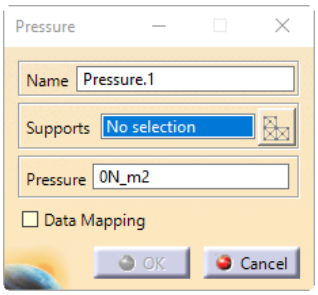


Figure 19-90 *The* ***Pressure*** *dialog box*

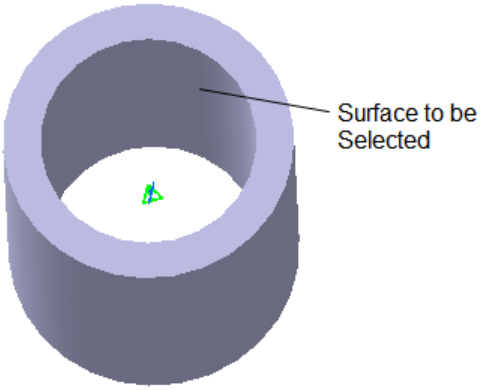


Figure 19-91 *Surface to be selected to apply pressure*

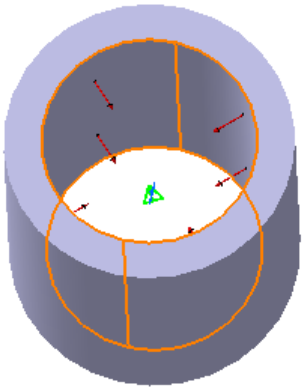


Figure 19-92 *Pressure applied to the selected surface*

Applying Distributed Force

Toolbar: Loads > Forces > Distributed Force



The distributed forces are those forces which are statically equivalent to a given resultant force at a given point or area. It is not aimed to uniformly distribute this force on the selected supports.

To apply this force on a face or surface, choose the **Distributed Force** tool from the **Forces** sub-toolbar; the **Distributed Force** dialog box will be displayed, as shown in Figure 19-93. Also, you are prompted to select a support as a vertex, a face, or a virtual part to apply this force. Select the required support from the geometry area and specify the value of the distributed force, as shown in Figure 19-94.

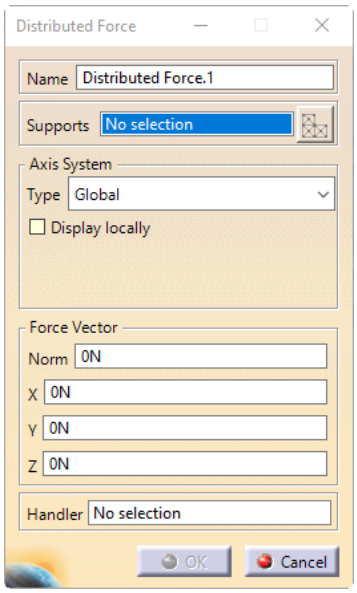


Figure 19-93 *The* ***Distributed Force*** *dialog box*

You can select the required type of axis system form the **Type** drop-down list in the **Axis System** area of this dialog box. By default, the **Global** axis system is selected. As a result, the resultant force vectors will be converted as relative to the global coordinate system. If you select the **User** axis system, the components of the resultant force vector will be converted as relative to the specified coordinate system. Also, on selecting the **User** option from the **Type** drop-down list, the **Current axis** display box and the **Orientation** drop-down list will be displayed in the **Axis System** area of this dialog box.

You can define the force vector by specifying the force magnitude in the **Norm** edit box. Also, you can specify the force vectors in the **X**, **Y** and **Z** edit boxes in the **Force Vector** area. Select the required handler point in the **Handler** selection box and choose **OK**.

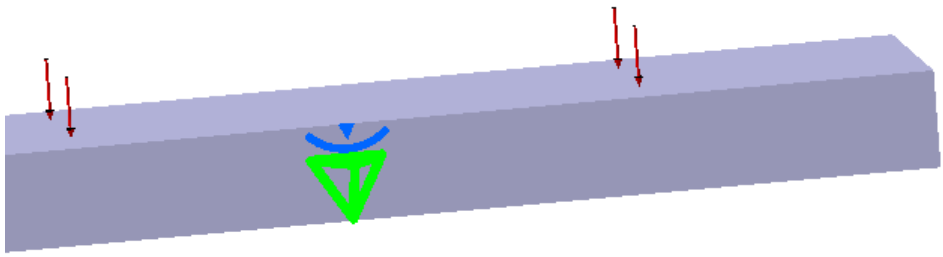


Figure 19-94 *The Distributed force applied to a selected surface*

Note

*To select the **User** axis system, you must activate it by selecting it from the specification tree. On activating, its name will be displayed in the **Current axis** selection box.*

Applying Moment

Toolbar: Loads > Forces > Moment

The moment load tends to overturn or bend the axis of rotation of a model angularly. It is the ability of the force to twist or turn the object about an axis.

To apply moment, choose the **Moment** tool from the **Forces** sub-toolbar; the **Moment** dialog box will be displayed, as shown in Figure 19-95. Select the required surface and specify the other required options and choose **OK**. Figure 19-96 shows the moment applied to a part.

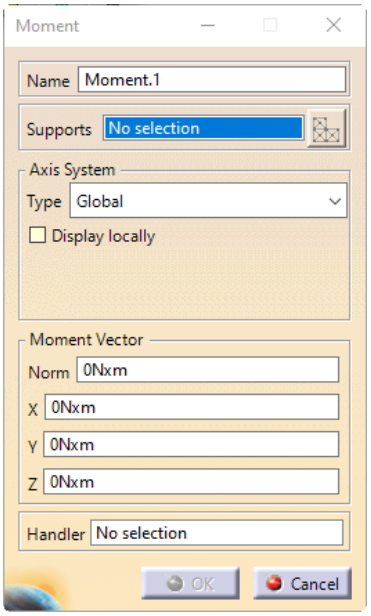


*Figure 19-95 The **Moment** dialog box*

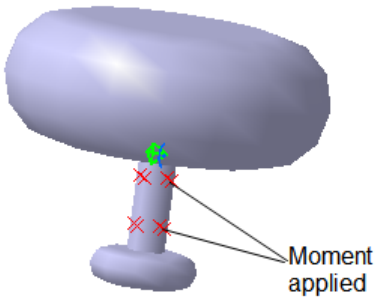


Figure 19-96 Moment applied to a part

Applying Bearing Load

Toolbar: Loads > Forces > Bearing Load

A bearing load is a simulated contact load applied to a cylindrical surface. To apply a bearing load to a surface, choose the **Bearing Load** tool from the **Forces** sub-toolbar; the **Bearing Load** dialog box will be displayed, as shown in Figure 19-97. Select the support surface from the geometry area; the name of support surface will be displayed, in the **Supports** selection box.

You can specify the angle on which the forces will be distributed in the **Angle** edit box. The **Orientation** drop-down allows you to distribute forces in two ways: radial and parallel. If you select the **Radial** option then all the force vectors at the mesh nodes will orient normal to surface. If

you choose the **Parallel** option then all the force vectors will orient parallel to the surface. Choose the profile type from the **Type** drop-down in the **Profile** area. Specify the force distribution as **Inward** and **Outward** from the **Distribution** drop-down and choose **OK**. Figure 19-98 shows a part with bearing load applied on it.

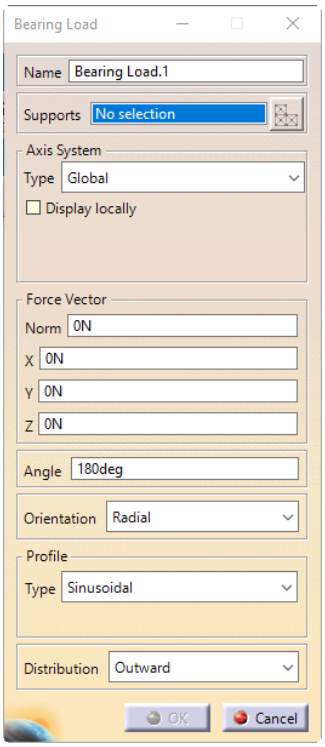


*Figure 19-97 The **Bearing Load** dialog box*

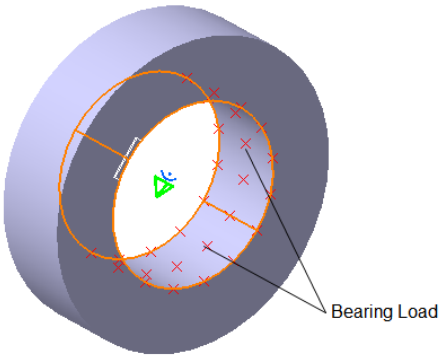


Figure 19-98 Bearing Load applied to a part

Applying Acceleration

Toolbar: Loads > Body Mass Forces > Acceleration



The acceleration are the intensive loads which represents mass body force fields of uniform magnitude applied on the parts. It is also known as the net result of any or all forces that acts on a part. To apply acceleration to a part, choose the **Acceleration** tool from the **Body Mass Forces** sub-toolbar; the **Acceleration** dialog box will be displayed, as shown in Figure 19-99.

Select the support surface from the geometry area; the name of the surface will be displayed in the **Supports** selection box. Next, specify the required options and choose **OK**. Figure 19-100 shows a part with acceleration applied on it.



Tip

You can select several supports in a sequence to apply the acceleration to all the supports simultaneously.

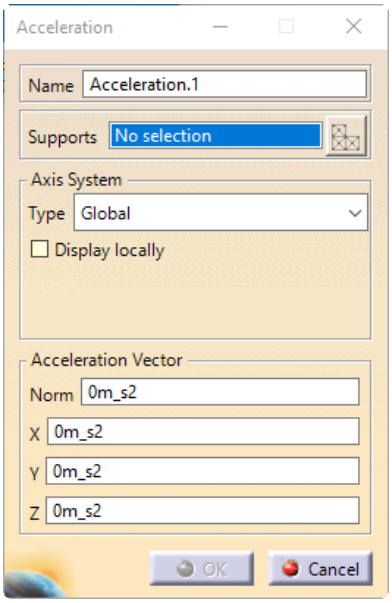


Figure 19-99 *The* ***Acceleration*** *dialog box*

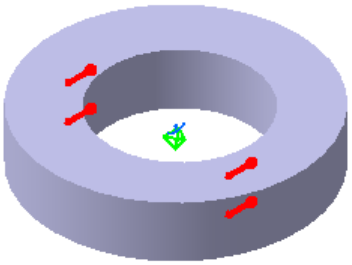


Figure 19-100 *Acceleration applied to a part*

Applying Rotation Forces

Toolbar: Loads > Body Mass Forces > Rotation Force



The rotation forces are accelerated loads which represent mass body force fields generated by the rotational motion applied to a part. To apply a rotational force, to a part, choose the **Rotation Force** tool from the **Body Mass Forces** sub-toolbar; the **Rotation Force** dialog box will be displayed, as shown in Figure 19-101.

Select the support surface from the geometry area; the name of the support surface will be displayed in the **Supports** selection box. Click once in the **Rotation Axis** selection box and choose the rotation axis. Specify the value of the angular velocity and angular acceleration in the **Angular Velocity** and **Angular Acceleration** edit boxes, respectively and choose **OK**. Figure 19-102 displays a part with rotational force applied to it.

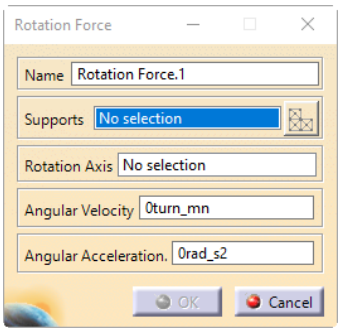


Figure 19-101 *The* ***Rotation Force*** *dialog box*

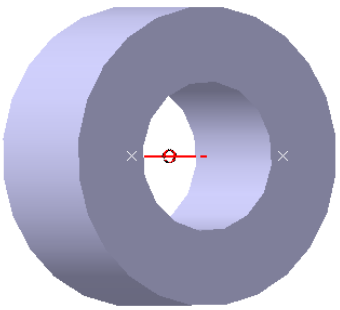


Figure 19-102 *Rotation force applied to a part*

Applying Line Force Density

Toolbar: Loads > Force Densities > Line Force Density



The line force densities are the intensive loads which represent the line resistance fields of uniform magnitude applied to various geometries.

To apply the line force density to a part, choose the **Line Force Density** tool from the **Force Densities** sub-toolbar; the **Line Force Density** dialog box will be displayed, as shown in Figure 19-103. Select a line body or an edge of a part as support and specify the required options from the dialog box and choose **OK**. Figure 19-104 shows a part with its edge selected as support to apply line force density.

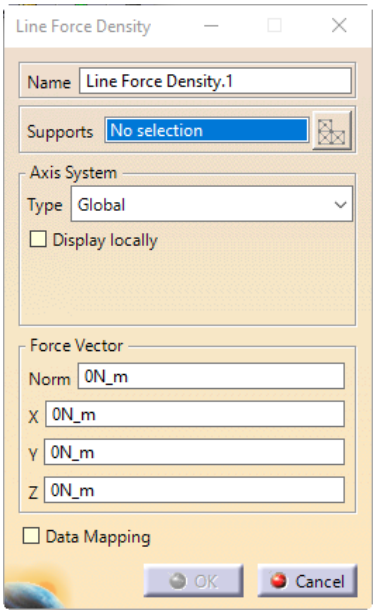


Figure 19-103 *The* ***Line Force Density*** *dialog box*

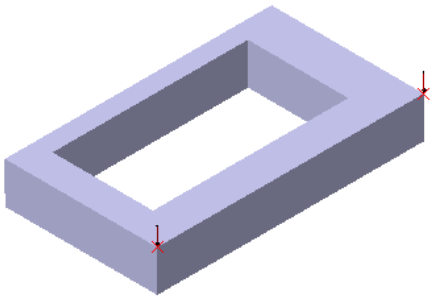


Figure 19-104 *Line force density applied to an edge of a part*

Applying Surface Force Density

Toolbar: Loads > Force Densities > Surface Force Density



The surface force densities are the intensive loads which represent the surface resistance fields of uniform magnitude applied to the curve geometries.

To apply the surface force density on a part, choose the **Surface Force Density** tool from the **Force Densities** sub-toolbar; the **Surface Force Density** dialog box will be displayed, as shown in Figure 19-105. Select a surface body or a face of a part as support and specify the required options from the dialog box and choose **OK**. Figure 19-106 shows a part with its upper face selected as support to apply surface force density.

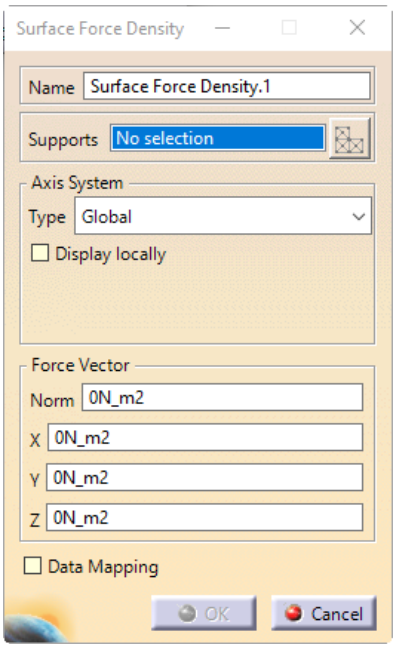


Figure 19-105 *The* ***Surface Force Density*** *dialog box*

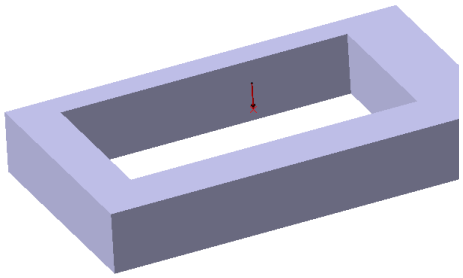


Figure 19-106 *Surface force density applied to a face of a part*

Applying Volume Force Density

Toolbar: Loads > Force Densities > Volume Force Density



The volume force densities are intensive loads which represent the volume body force fields of uniform magnitude applied to the parts.

To apply the volume force density to a part, choose the **Volume Force Density** tool from the **Force Densities** sub-toolbar; the **Volume Force Density** dialog box will be displayed, as shown in Figure 19-107. Select a part as support and specify the required options from the dialog box and choose **OK**. Figure 19-108 shows a part to be selected as support to apply volume force density.

Applying Force Density

Toolbar: Loads > Force Densities > Force Density



The **Force Density** tool is used to create the equivalent of the existing force, surface force, and volume force densities. To do so, choose the **Force Density** tool from the **Force Densities** sub-toolbar; the **Force Density Defined by Force Vector** dialog box will be displayed, as shown in Figure 19-109. Select the required part and specify the required parameters in this dialog box and choose **OK**.

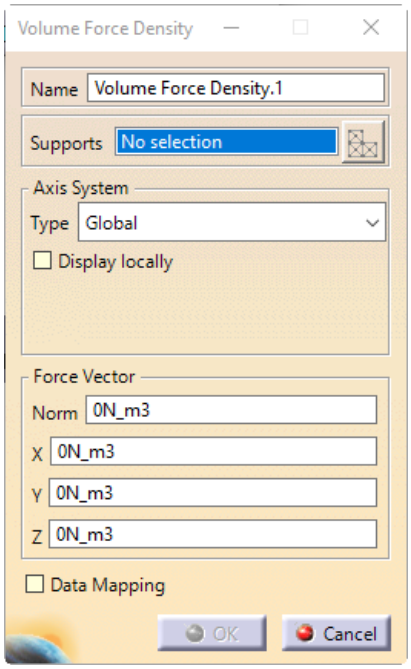


Figure 19-107 *The* ***Volume Force Density*** *dialog box*

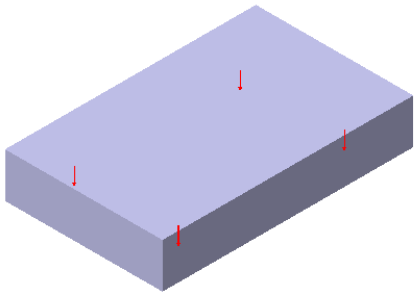


Figure 19-108 *Volume force density applied to a part*

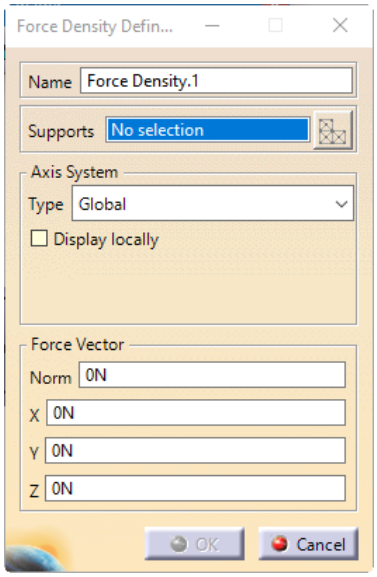


Figure 19-109 *The* ***Force Density Defined by Force Vector*** *dialog box*

Applying Enforced Displacement

Toolbar: Loads > Enforced Displacement



The enforced displacement loads are those loads that are applied to support geometries. As a result, the subsequent analysis will assign a non-zero value to the displacement in previously restrained direction.

To apply the enforced displacement, choose the **Enforced Displacement** tool from the **Loads** toolbar; the **Enforced Displacement** dialog box will be displayed, as shown in Figure 19-110. Select the required restraint in the **Restraint** selection box and specify the translation and rotation values in the **Translation** and **Rotation** edit boxes. Choose **OK**. The enforce displacement will be attached to the selected restraint, refer to Figure 19-1111.



Note

Make sure you have entered non-zero values only for those degrees of freedom which have been fixed by the associated restraint object. Non-zero values for any other degree of freedom will be ignored by the program.

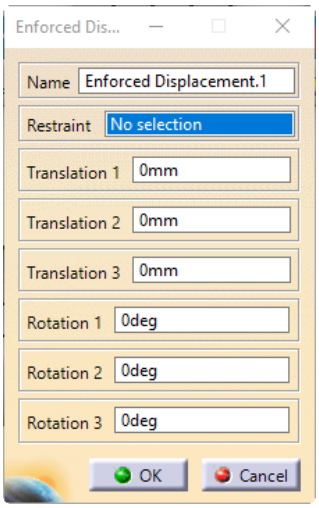


Figure 19-110 *The* ***Enforced Displacement*** *dialog box*

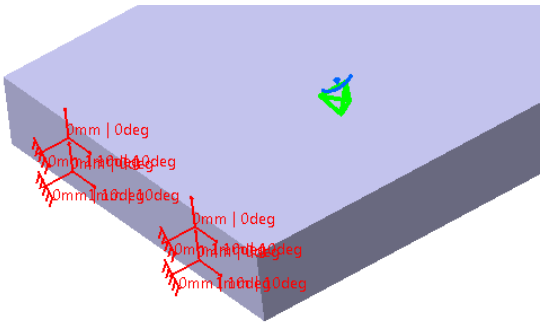


Figure 19-111 *Enforced displacement applied to a part*

Applying Temperature Field

Toolbar: Loads > Temperature Field > Temperature Field



The **Temperature Field** tool is used to apply temperature constant to a part. To do so, choose the **Temperature Field** tool from the **Temperature Field** sub-toolbar; the **Temperature Field** dialog box will be displayed, as shown in Figure 19-112. Select the required part to apply the temperature field; you will notice a **T** sign gets attached to the selected part. Specify the temperature in the **Temperature** edit box and choose **OK**. Figure 19-113 displays a part with temperature field applied on it.

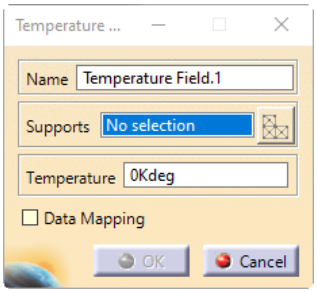


Figure 19-112 *The* ***Temperature Field*** *dialog box*

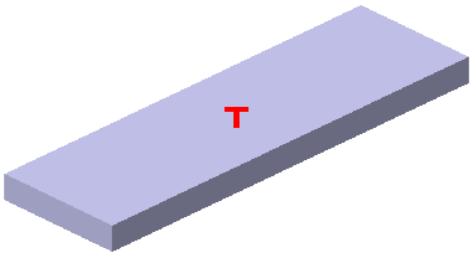


Figure 19-113 *Temperature field applied to a part*

COMPUTING THE ANALYSIS RESULT

Toolbar: Compute > Compute

The **Compute** tool is used to compute the result for the analysis. To do so, choose the **Compute** tool from the **Compute** sub-toolbar; the **Compute** dialog box will be displayed, as shown in Figure 19-114. Choose the required operation to compute from the drop-down available in this dialog box and choose **OK**. After choosing the **OK** button, the **Computing** process window will be displayed, refer to Figure 19-115. Once the computing process completes, the **Computation Resources Estimation** message box will be displayed, refer to Figure 19-116. To continue the computation, choose the **Yes** button from this dialog box. Note that you can view the required analysis results with the help of tools available in the **Image** toolbar.

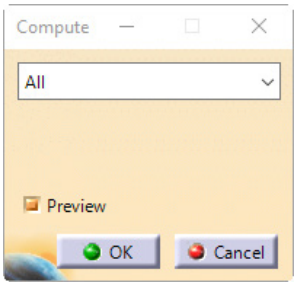


Figure 19-114 *The* ***Compute*** *dialog box*

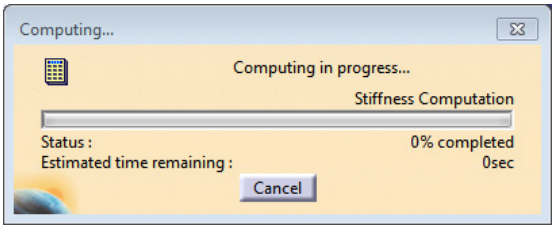


Figure 19-115 *The* ***Computing*** *process window*

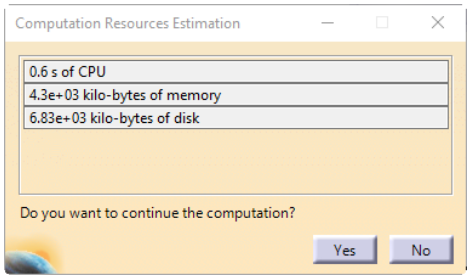


Figure 19-116 *The* ***Computation Resources Estimation*** *message box*

Note

*After choosing **OK** from the **Compute** dialog box if a warning message box is displayed with several warnings related to the conditions you have applied, choose **OK** from this dialog box to continue.*

GENERATING THE ANALYSIS REPORT

Toolbar: Analysis Results > Report > Generate Report

The **Generate Report** tool is used to generate the complete report of the preprocessor, solver, and postprocessor data of the problem after completing the solution. To generate the complete report, choose the **Generate Report** tool from the **Report** sub-toolbar of the **Analysis Result** toolbar; the **Report Generation** dialog box will be displayed, as shown in Figure 19-117. Specify the required output directory path in the **Output directory** edit box with the help of the **Browse** button next to it and choose **OK**. On doing so, the reporting process window will be displayed. After completing the process, stress analysis report will be displayed on the internet browser.

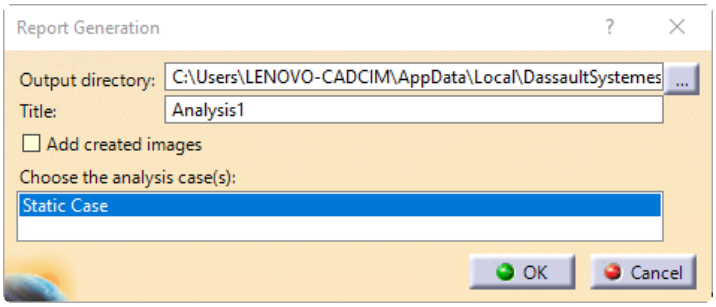


***Figure 19-117** The **Report Generation** dialog box*

ANIMATING RESULTS

Toolbar: Analysis Tools > Animate

The **Animate** tool is used to generate the animation view of the results generated in the postprocessing phase. This tool remains active only when the valid solution is available. On choosing the **Animate** tool from the **Analysis Tools** toolbar; the **Animation** dialog box will be displayed, as shown in Figure 19-118. The options in this dialog box are used to play, stop, and set the speed of animation. Specify the required parameters and choose **OK**. The model will start animating on the basis of the selected parameters.

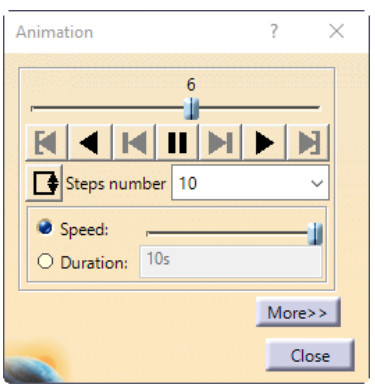


***Figure 19-118** The **Animation** dialog box*

TUTORIALS

Tutorial 1

In this tutorial, you will analyze the effect of loads on the spring shown in Figure 19-119. The spring is fixed at one end and the load is applied on the other end. Under the given load and constraints you will perform static analysis on this spring. The part file of this tutorial can be downloaded from www.cadcim.com. The boundary conditions are given within the tutorial steps only. **(Expected time: 1hr)**

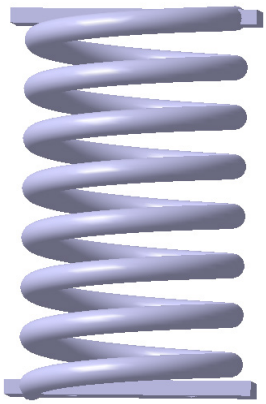


Figure 19-119 *Model of the spring*

The following steps are required to complete the tutorial.

a. Download and import the input file to the **Generative Structural Analysis** workbench.
b. Generate the mesh.
c. Apply constraints and loads.
d. Compute the analysis.
e. Analyze the results.
f. Generate the analysis report.
g. Save the analysis.

Downloading the Part File

Before starting the tutorial, you need to download the spring part file for this tutorial from the CADCIM website.

Download the file *c19_catia_tut1.zip* file from *www.cadcim.com*. The path of the file is as follows: *Textbook > CAD/CAM > CATIA > CATIA V5-6R2022 for Designers > Input files.*

Starting the Static Analysis

After extracting the input file, you need to open the spring part file in CATIA.

1. After opening the file in CATIA, start a new file in the **Generative Structural Analysis** workbench by choosing **Start > Analysis & Simulation > Generative Structural Analysis**. The **New Analysis Case** dialog box will be displayed along with the imported spring part in the **Generative Structural Analysis** workbench.

2. Choose the **Static Analysis** option from the **New Analysis Case** dialog box and choose **OK**.

The **Generative Structural Analysis** workbench along with various analysis tools will be displayed. Also, you will notice that a mesh icon will be displayed at the center of the spring.



Note

*On invoking this workbench if a warning message box is displayed informing that the material is not properly defined on the part, choose **OK** from this warning message box to continue.*

Applying the Material

Next, you need to apply the material to the spring.

1. Click once on the spring from the graphics area and invoke the **Library (Read only)** dialog box by choosing the **Material on Analysis Connection** tool from the **Material on Analysis Connection** toolbar.

2. Make sure the **Metal** tab is selected in this dialog box.

3. Select the **Brass** metal to apply on the imported spring and choose the **Apply Material** button from this dialog box to apply the material to the spring and choose **OK**.

Generating the Mesh

Next, you need to generate the mesh on the spring.

1. Invoke the **OCTREE Tetrahedron Mesh** dialog box by double clicking on the **OCTREE Tetrahedron Mesh** sub-node of the **Nodes and Elements** node.

2. Enter **5 mm** in the **Size** edit box and specify **1 mm** in the **Absolute sag** edit box of the **Global** tab, and keeping the other values as default, choose **OK**.

Applying Restraints and Loads to the Spring

Next, you need to apply boundary constraints and loads on the spring.

1. Choose the **Clamp** tool from the **Restraints** toolbar; the **Clamp** dialog box is displayed.

2. Choose the inner face of the holes at the bottom face of the spring base, as shown in Figure 19-120 and then choose the **OK** button from the **Clamp** dialog box; the clamp restraint is applied, refer to Figure 19-121.

After applying the material and restraints to the spring, you need to apply load on the holes at the top face of the spring.

1. Choose the **Distributed Force** tool from the **Force** sub-toolbar of the **Loads** toolbar; the **Distributed Force** dialog box is displayed.

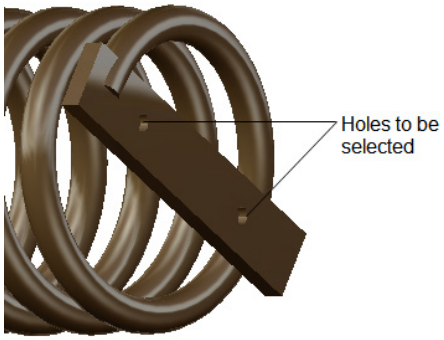


Figure 19-120 The holes to be selected

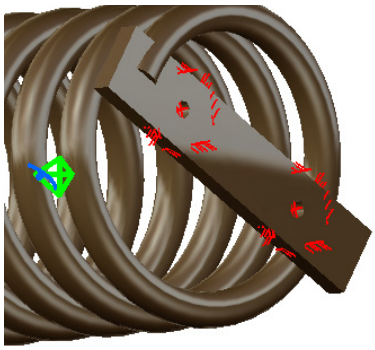


Figure 19-121 The clamp restraint applied to the base holes

2. Select the inner face of the holes on the top face of the spring from the graphics area, refer to Figure 19-122. The name of the selected faces of holes are displayed in the **Supports** selection box

3. Enter **-100 N** in the **Z** edit box in the **Force Vector** area and choose **OK**. Figure 19-123 shows the spring after applying the load.

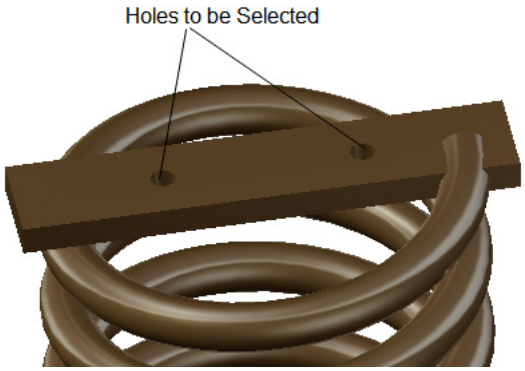


Figure 19-122 The holes to be selected

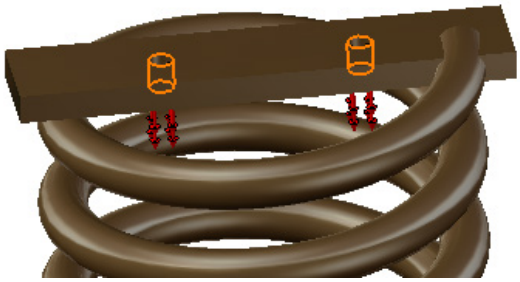


Figure 19-123 Load applied on the holes of the upper face of the spring

Computing the Analysis

After applying all the restraints, loads, and mesh, you need to solve the analysis for the spring.

1. Choose the **Compute** tool from the **Compute** sub-toolbar; the **Compute** dialog box is displayed. Select the **All** option from the drop-down list available in this dialog box if not selected by default and choose **OK** from this dialog box. The **Computing** process window and the **Computation Status** message box are displayed.

After the computing process is over, the **Computation Resources Estimation** message box is displayed.

2. Choose **Yes** from this message box. The **Computing** process window and **Computation Status** message box are displayed again.

Once the processing is over; the result is computed and you can view the results using various tools available in the **Image** toolbar.

Analyzing the Result

Now, after the computation process is completed, you will analyze the result.

1. Choose the **Deformation** tool from the **Image** toolbar; the deformation generated in the model is displayed in the graphics area, refer to Figure 19-124.

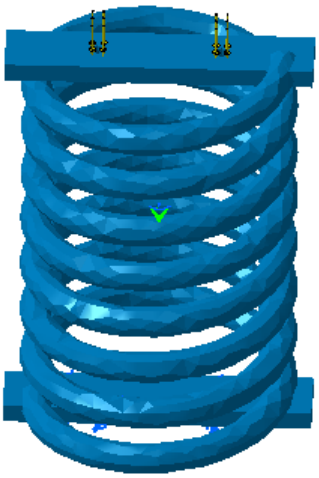


Figure 19-124 *The deformation in the spring*

2. Choose the **Von Mises Stress** tool from the **Image** toolbar; the Von Mises Stress generated in the model is displayed in the graphics area, refer to Figure 19-125.

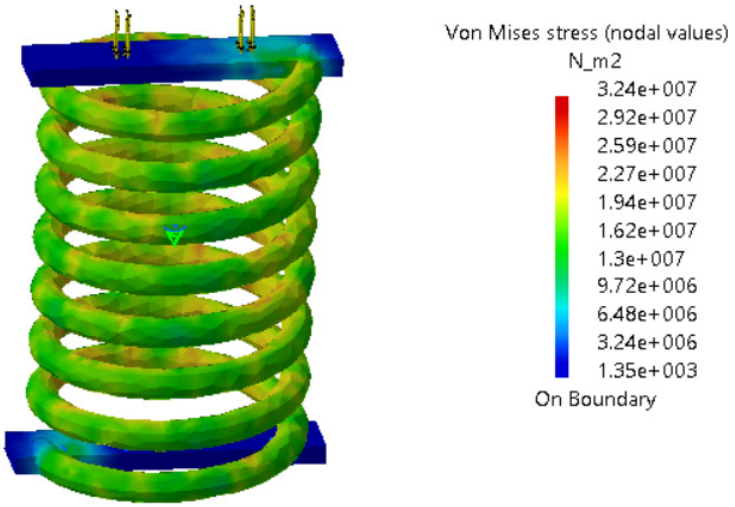


Figure 19-125 *The Von Mises Stress generated in the spring*

Generating the Analysis Report

Next, you will generate the analysis report of the result.

1. Choose the **Generate Report** tool from the **Report** sub-toolbar of the **Analysis Report** toolbar; the **Report Generation** dialog box is displayed. Specify the location to save the report and choose **OK**. The analysis report is generated in *.html* format and is displayed in your default internet browser.

After completing the study procedure and generating the analysis report, you need to save the model.

Saving and Closing the File

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box.

2. Enter **c19tut1** in the **File name** edit box and choose the **Save** button. The file gets saved at *C:\CATIA\c19*.

3. Close the part file by choosing **File > Close** from the menubar.

Tutorial 2

In this tutorial, you will analyze the buckling case static analysis with enforce displacement on the variable radius shaft shown in Figure 19-126. It is fixed at one end and the load is applied on the other end. The part file of this tutorial can be downloaded from www.cadcim.com. Under the given load and constraints, you will perform the static analysis on this shaft and then the buckling analysis with the help of the solution of the static analysis. The boundary conditions are given within the tutorial steps only.

(Expected time: 1.5hr)

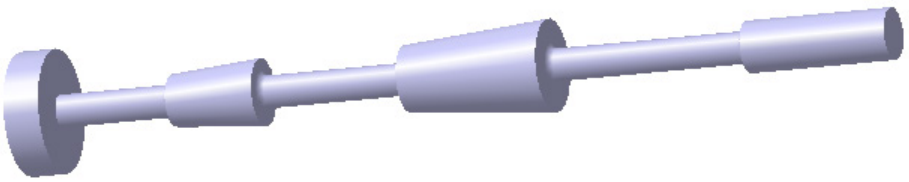


Figure 19-126 *Model of the variable radius shaft*

The following steps are required to complete the tutorial.

a. Download and import the input file to the **Generative Structural Analysis** workbench.
b. Generate the mesh.
c. Apply constraints and loads.
d. Compute the static analysis.
e. Insert and compute buckling case.
f. Analyze the results.
g. Generate the buckling case analysis report.
h. Save the analysis.

Downloading the Part File

Before starting the tutorial, you need to download the part file for this tutorial from the CADCIM website.

Download the file *c19_catia_tut2.zip* file from *www.cadcim.com*. The path of the file is as follows: *Textbook > CAD/CAM > CATIA > CATIA V5-6R2022 for Designers > Input file*.

Starting the Static Analysis

After extracting the input file, you need to open the variable radius shaft part file in CATIA.

1. Start a new file in the **Generative Structural Analysis** workbench by choosing **Start > Analysis & Simulation > Generative Structural Analysis**. The **New Analysis Case** dialog box gets displayed along with the imported variable radius shaft part in the **Generative Structural Analysis** workbench.

2. Choose the **Static Analysis** option from the **New Analysis Case** dialog box and choose **OK**.

The **Generative Structural Analysis** workbench will be displayed along with various analysis tools. Also, you will notice that a mesh icon at the center of the shaft.



Note

*On invoking this workbench if a warning message box is displayed informing that the material is not properly defined on the part, choose **OK** from this warning message box to continue.*

Applying the Material

Next, you need to apply the material to the shaft.

1. Click once on the shaft from the graphics area and invoke the **Library (Read only)** dialog box by choosing the **Material on Analysis Connection** tool from the **Material on Analysis Connection** toolbar.

2. Make sure the **Metal** tab is selected in this dialog box.

3. Select the **Iron** metal to apply it on the imported shaft and choose the **Apply Material** button from this dialog box to apply the material to the shaft and choose **OK**.

Generating the Mesh

Next, you need generate the mesh on the shaft.

1. Invoke the **OCTREE Tetrahedron Mesh** dialog box by double clicking on the **OCTREE Tetrahedron Mesh** sub-node of the **Nodes and Elements** node.

2. Enter **10** mm in the **Size** edit box of the **Global** tab, **2** mm in the **Absolute sag** edit box, and keeping the other values as default, choose **OK**.

Applying the Restraints and Loads to the Spring

Next, you need to apply the boundary constraints and loads on the shaft.

1. Choose the **Clamp** tool from the **Restraints** toolbar; the **Clamp** dialog box is displayed.

2. Choose the bottom face of the shaft, as shown in Figure 19-127 and then choose the **OK** button from the **Clamp** dialog box; the clamp restraint is applied, refer to Figure 19-128.

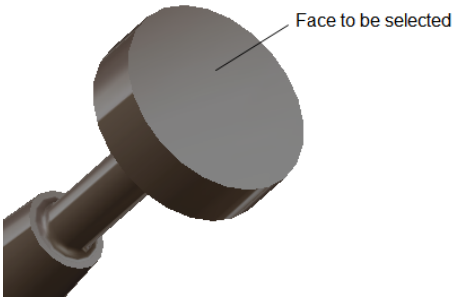


Figure 19-127 *The bottom face to be selected*

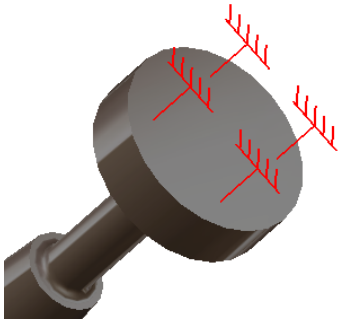


Figure 19-128 *The clamp restraint applied to the selected face*

After applying the material and restraints to the shaft, you need to apply enforced displacement on the top face of the shaft. To apply the enforced displacement, first you need to create a user-defined restraint.

3. Choose the **User-defined Restraint** tool from the **Advance Restraint** sub-toolbar of the **Restraints** toolbar; the **User-defined Restraint** dialog box is displayed.

4. Select the top face of the shaft, as shown in Figure 19-129. The selected face gets displayed in the **Supports** selection box. Next clear all the restrain translation and rotation check boxes except the **Restrain Translation 2** check box. Choose **OK**; the user defined restraint is applied, refer to Figure 19-130.



Figure 19-129 *The top face to be selected*



Figure 19-130 *The* ***User-defined Restraint*** *restraint applied to the selected face*

Next, you will apply the enforced displacement to the user-defined restraint applied on the shaft.

5. Choose the **Enforced Displacement** tool from the **Loads** toolbar; the **Enforced Displacement** dialog box is displayed.

6. Choose the **User-defined Restraint** applied on the shaft from the graphics area; its name gets displayed in the **Restraint** selection box of the **Enforced Displacement** dialog box.

7. Enter **-2** in the **Translation 2** edit box and choose **OK**.

Computing the Analysis

After applying all the restraints, loads, and mesh, you need to solve the static analysis for the shaft.

1. Choose the **Compute** tool from the **Compute** sub-toolbar; the **Compute** dialog box is displayed. Select the **All** option from the drop-down list available in this dialog box if not chosen by default and choose **OK** from this dialog box; the **Computing** process window and the **Computation Status** message box are displayed.

After the computing process is over, the **Computation Resources Estimation** message box is displayed.

2. Choose **Yes** from this message box. The **Computing** process window and the **Computation Status** display box are displayed again.

Once the processing is over, the result is computed which you can view using the tools in the **Image** toolbar. The Von Mises Stress generated is shown in Figure 19-131.

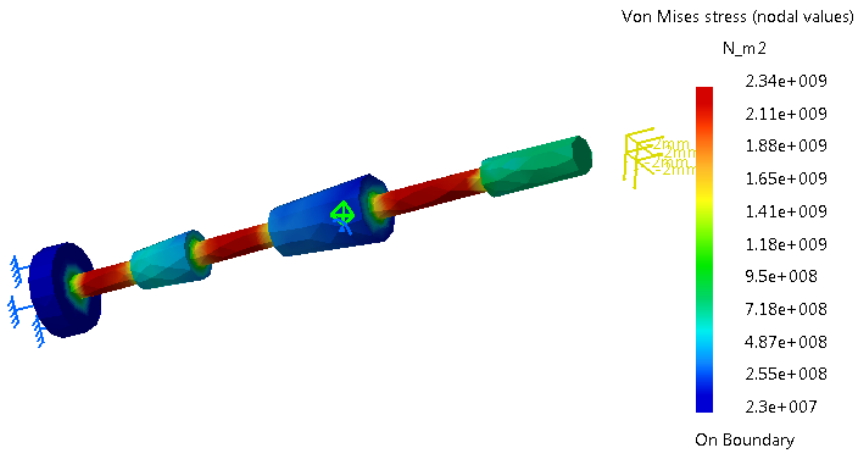


Figure 19-131 *The Von Mises Stress generated in the shaft*

Inserting the Buckling Case

After analyzing the static case, you need to insert the buckling case to analyze the buckling in the Shaft.

1. Choose the **Insert > Buckling Case** option from the menubar; the **Buckling Case** dialog box will be displayed.

2. Select the static case analysis solution from the specification tree; its name will be displayed in the **Reference** selection box of the **Buckling Case** dialog box. Next choose OK.

You will notice that a **Buckling Case** gets added as a new node to the , refer to Figure 19-132.

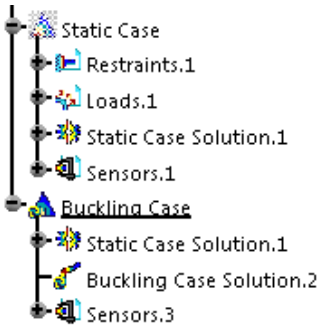


Figure 19-132 *The **Buckling Case** node added to specification tree*

Computing the Buckling Case Analysis

Next, you need to compute the result for the buckling case analysis.

1. Choose the **Compute** tool from the **Compute** sub-toolbar; the **Compute** dialog box is displayed. Again, choose the **All** option from the drop-down list available in this dialog box and choose **OK** from this dialog box. The **Computing process** window and the **Computation Status** display box are displayed.

After the computing process is over, the **Computation Resources Estimation** message box is displayed.

2. Choose **Yes** from this message box. The **Computing** process window and the **Computation Status** message box are displayed again.

Once the processing is over, the result is computed which you can view using the tools in the **Image** toolbar.

Analyzing the Result for the Buckling Case Analysis

After the computation process is completed, you can analyze the result.

1. Choose the **Deformation** tool from the **Image** toolbar; the deformation generated in the shaft is displayed in the graphics area, refer to Figure 19-133

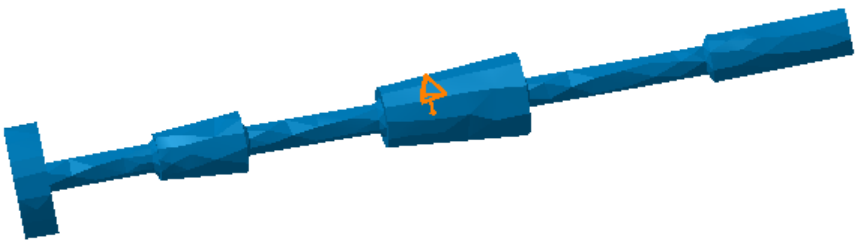


Figure 19-133 *The deformation in the shaft*

2. Double click on the **Deformed mesh** sub node, refer to Figure 19-134; the **Image Edition** dialog box gets displayed, as shown in Figure 19-135. Choose the **Occurrences** tab if not chosen by default.

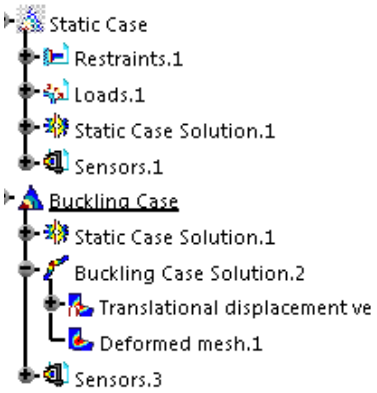


*Figure 19-134 The **Deformed mesh** sub-node*

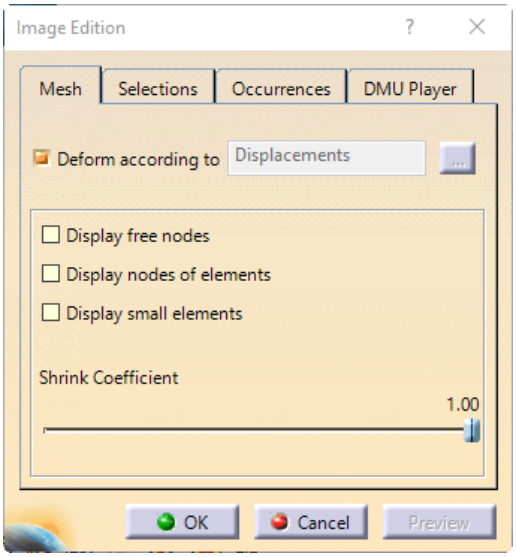


*Figure 19-135 The **Image Edition** dialog box*

3. Select the required modes from the available buckling factors under the **Occurrences** tab of this dialog box and choose **OK**. The selected deformed mode gets displayed in the graphics area. Figure 19-136 displays the deformed shaft.



Figure 19-136 The changed deformation in the shaft

Generating the Buckling Analysis Report

Next, you will generate the buckling analysis report of the result.

1. Choose the **Generate Report** tool from the **Report** sub-toolbar of the **Analysis Report** toolbar; the **Report Generation** dialog box is displayed. Specify the location to save the report. Choose the **Buckling Case** from the **Choose the Analysis case(s)** area and then choose **OK**. The analysis report is generated in *.html* format and is displayed in your default internet browser.

After completing the study procedure and generating the analysis report, you need to save the model.

Saving and Closing the File

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box.

2. Enter **c19tut2** in the **File name** edit box and choose the **Save** button. The file will be saved at *C:\CATIA\c19*.

3. Close the part file by choosing **File > Close** from the menubar.

Tutorial 3

In this tutorial, you will analyze frequency analysis results of a leaf spring shown in Figure 19-137. The part file of this tutorial can be downloaded from *www.cadcim.com*. The leaf spring is fixed at its both ends. Under the given load and restraints, you will perform the frequency analysis on this leaf spring. The boundary conditions are given within the tutorial steps only.

(Expected time: 1.5hr)



Figure 19-137 *Model for leaf spring*

The following steps are required to complete the tutorial.

a. Download and import the input file to **Generative Structural Analysis** workbench.
b. Generate the mesh.
c. Apply restraints.
d. Compute the frequency analysis.
e. Analyze the results.
f. Generate the analysis report.
g. Save the analysis.

Downloading the Part File

Before starting the tutorial, you need to download the part file for this tutorial from CADCIM website.

Download the file *c19_catia_tut3.zip* file from *www.cadcim.com*. The path of the file is as follows: *Textbook > CAD/CAM > CATIA > CATIA V5-6R2022 for Designers > Input files.*

Starting the Static Analysis

After extracting the input file, you need to open the leaf spring part file in CATIA.

1. Start a new file in the **Generative Structural Analysis** workbench by choosing **Start > Analysis & Simulation > Generative Structural Analysis**. The **New Analysis Case** dialog box will be displayed along with the imported Leaf spring part in the **Generative Structural Analysis** workbench.

2. Choose the **Frequency Analysis** option from the **New Analysis Case** dialog box and choose **OK**.

The **Generative Structural Analysis** workbench will be displayed along with various analysis tools. Also, you will notice that a mesh icon will be displayed at the center of the leaf spring.



Note

*On invoking this workbench, if a warning message box is displayed, informing that the material on the part is not displayed then choose **OK** to continue.*

Applying the Material

Next, you need to apply the material to the leaf spring.

1. Click once on the leaf spring from the graphics area and invoke the **Library (Read only)** dialog box by choosing the **Material on Analysis Connection** tool from the **Material on Analysis Connection** toolbar.

2. Make sure the **Metal** tab is selected in this dialog box if not selected.

3. Select the **Steel** metal to apply it on the imported leaf spring and choose the **Apply Material** button from this dialog box to apply the material to the leaf spring and then choose **OK**.

Generating the Mesh

Next, you need to generate the mesh on the leaf spring.

1. Invoke the **OCTREE Tetrahedron Mesh** dialog box by double clicking on the **OCTREE Tetrahedron Mesh** sub-node of the **Nodes and Elements** node.

2. Enter **1** mm and **0.1** mm in the **Size** and **Absolute sag** edit boxes of the **Global** tab respectively, keeping the other values as default, choose **OK**.

Applying the Restraints to the Spring

Next, you need to apply the boundary constraints and loads on the leaf spring.

1. Choose the **Clamp** tool from the **Restraints** toolbar; the **Clamp** dialog box is displayed.

2. Choose the inner face of the curved face of the leaf spring, as shown in Figure 19-138 and then choose the **OK** button from the **Clamp** dialog box; the clamp restraint is applied, refer to Figure 19-139.

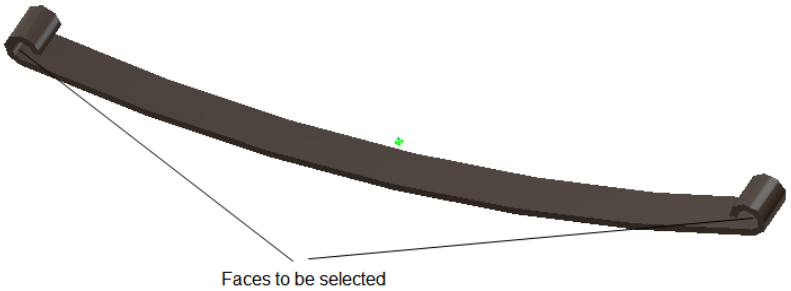


Figure 19-138 *The faces to be selected*



Figure 19-139 *The clamp restraint applied to the selected face*

Computing the Analysis

After applying all restrains, and mesh, you need to solve the analysis for the model.

1. Choose the **Compute** tool from the **Compute** sub-toolbar; the **Compute** dialog box is displayed. Select the **All** option from the drop-down list available in this dialog box, if not chosen by default and choose **OK** from this dialog box. The **Computing** process window and the **Computation Status** message box are displayed.

After the computing process is over, the **Computation Resources Estimation** message box is displayed.

2. Choose **Yes** from this message box. The **Computing** process window and the **Computation Status** display box are displayed again.

Once the processing is over, the result is computed. You can view the results using various tools available in the **Image** toolbar.

Analyzing the Result

Now, after the computation process is completed, you can analyze the result.

1. Choose the **Von Mises Stress** tool from the **Image** toolbar; the deformation generated in the leaf spring is displayed in the graphics area, refer to Figure 19-140.

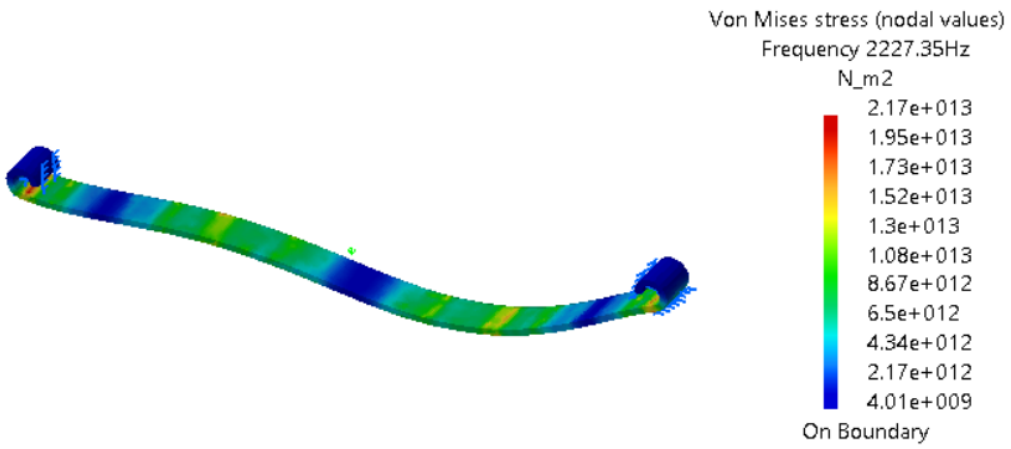


Figure 19-140 *The Von Mises Stress generated in the Leaf Spring*

Now, you can view various occurrences of frequency.

2. Double click on the **Von Mises stress** sub node, refer to Figure 19-141; the **Image Edition** dialog box gets displayed. Choose the **Occurrences** tab from this dialog box, if not chosen by default.

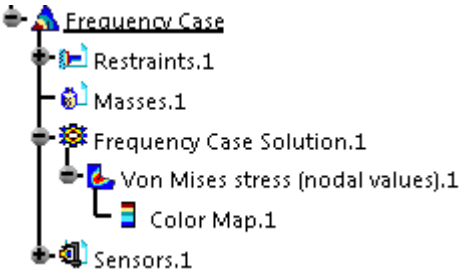


Figure 19-141 *The **Von Mises stress** sub-node*

3. Select the required mode from the available **Frequency (Hz)** list under the **Occurrences** tab at and choose **OK**.

Generating the Analysis Report

Next, you will generate the analysis report of the result.

1. Choose the **Generate Report** tool from the **Report** sub-toolbar of the **Analysis Report** toolbar; the **Report Generation** dialog box is displayed. Specify the location to save the report and choose **OK**. The analysis report is generated in *.html* format and is displayed in your default internet browser.

After completing the study procedure and generating the analysis report, you need to save the model.

Saving and Closing the File

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box.

2. Enter **c19tut3** in the **File name** edit box and choose the **Save** button. The file is saved at *C:\CATIA\c19*.

3. Close the part file by choosing **File > Close** from the menubar.

Tutorial 4

In this tutorial, you will analyze the effect of temperature on the Engine block shown in Figure 19-142. The part file of this tutorial can be downloaded from *www.cadcim.com*. Under the given load and constraints you will perform temperature analysis on this part. The boundary conditions are given within the tutorial steps only,

(Expected time: 1.5hr)

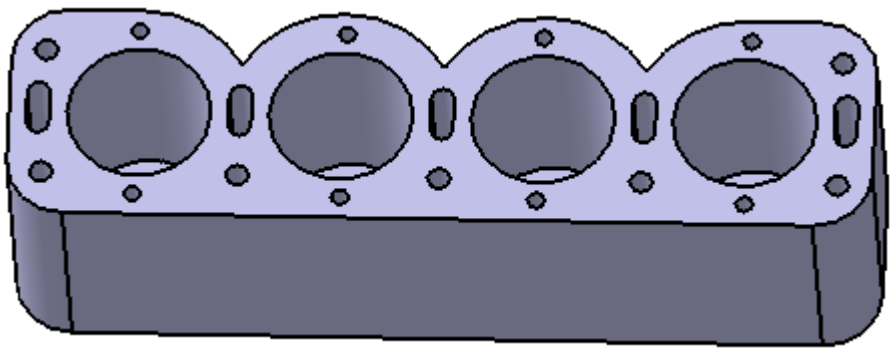


Figure 19-142 *Model for Engine block*

The following steps are required to complete this tutorial.

a. Download and import the input file to the **Generative Structural Analysis** workbench.
b. Generate the mesh.
c. Apply restraints.
d. Apply temperature field.
e. Compute the analysis.
f. Analyze the results.
g. Generate the analysis report.
h. Save the analysis.

Downloading the Part File

Before starting the tutorial, you need to download part file for this tutorial from the CADCIM website.

Download *c19_catia_tut4.zip* file from *www.cadcim.com*. The path of the file is as follows:
Textbook > CAD/CAM > CATIA > CATIA V5-6R2022 for Designers > Input files.

Starting the Static Analysis

After extracting the input file, you need to open the Engine block part file in CATIA.

1. Next, start a new file in the **Generative Structural Analysis** workbench by choosing **Start > Analysis & Simulation > Generative Structural Analysis**. The **New Analysis Case** dialog gets be displayed along with the imported Engine block part in this workbench.

2. Choose the **Static Analysis** option from the **New Analysis Case** dialog box and choose **OK**.

The **Generative Structural Analysis** workbench will be displayed along with various analysis tools. Also, you will notice a mesh icon at the center of the Engine block part.

Applying the Material

Next, you need to apply the material to the Engine block.

1. Click once on the Engine block from the graphics area and invoke the **Library (Read only)** dialog box by choosing the **Material on Analysis Connection** tool from the **Material on Analysis Connection** toolbar.

2. Make sure the **Metal** tab is selected in this dialog box.

3. Select the **Aluminium** metal to apply it on the imported Engine block and choose the **Apply Material** button from this dialog box to apply the material to the Engine block. Next, choose **OK**.

Generating the Mesh

Next, you need to generate the mesh on the Engine block.

1. Invoke the **OCTREE Tetrahedron Mesh** dialog box by double clicking on the **OCTREE Tetrahedron Mesh** sub-node of the **Nodes and Elements** node.

2. Enter **1** and **0.5** mm in the **Size** edit box and the **Absolute sag** edit box of the **Global** tab respectively, and keeping the other values as default choose **OK**.

Applying the Restraints to the Engine block

Next, you need to apply the boundary constraints and loads on the Engine block.

1. Choose the **Surface Slider** tool from the **Mechanical Restraints** sub-toolbar of the **Restraints** toolbar; the **Surface Slider** dialog box is displayed.

2. Choose all the 15 holes on the upper face of the Engine block, as shown in Figure 19-143. Apply the surface slider restraint to all the flat faces of the elongated holes. Choose the **OK** button from the **Surface Slider** dialog box; the surface slider restraint is applied to the selected holes, refer to Figure 19-144.

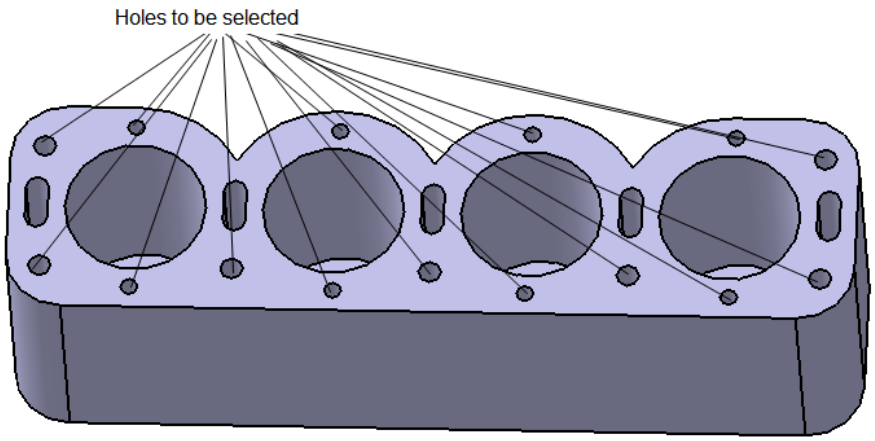


Figure 19-143 *The holes to be selected*

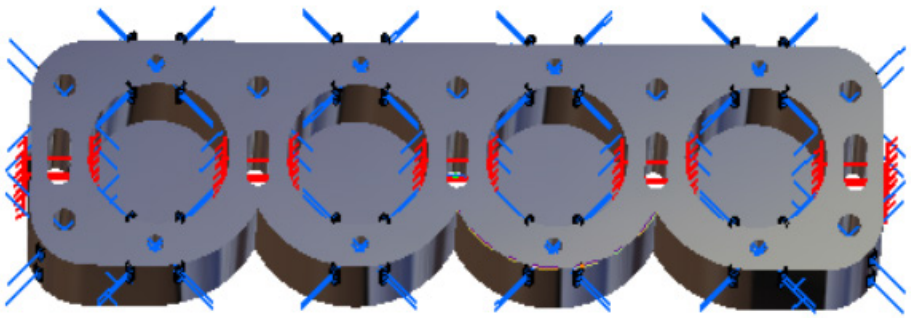


Figure 19-144 *The Surface Slider restraint applied to the selected holes*

Applying Temperature Field to the part

Next, you need to apply temperature filed on the Engine block.

1. Choose the **Temperature Field** tool from the **Temperature** sub-toolbar of the **Loads** toolbar; the **Temperature Field** dialog box is displayed.

2. Select the Engine block from the graphics area; its name gets displayed in the **Support**s selection box of this dialog box.

3. Specify the temperature value as **10** in the **Temperature** edit box and choose **OK**,

You will notice a T symbol will be displayed at the center of the Engine block.

Computing the Analysis

After applying all the restraints, meshes, and temperature fields, you need to solve the analysis for the model.

1. Choose the **Compute** tool from the **Compute** toolbar; the **Compute** dialog box is displayed. Select the **All** option from the drop-down list available in this dialog box if not chosen by default and choose **OK** from this dialog box. The **Computing** process window and the **Computation Status** message box are displayed.

After the computing process is over, the **Computation Resources Estimation** message box is displayed.

2. Choose **Yes** from this message box. The **Computing** process window and the **Computation Status** display box are displayed again.

Once the processing is over the result is computed, you can view the results using various tools available in the **Image** toolbar. Note that after the computation process if a message box displays that the rotation constraints were not taken into account as mesh part is 3D. Choose **OK** from this message box to continue and choose **Yes** from the **Computation Resources Estimation** message box.

Analyzing the Result

Now, after the computation process is completed, you can analyze the result.

1. Choose the **Von Mises Stress** tool from the **Image** toolbar; the deformation generated in the Engine block is displayed in the graphics area, refer to Figure 19-145.

2. Choose the **Principal Stress** tool from the **Image** toolbar; the principal stress generated in the Engine block model is displayed in the graphics area, refer to Figure 19-146.

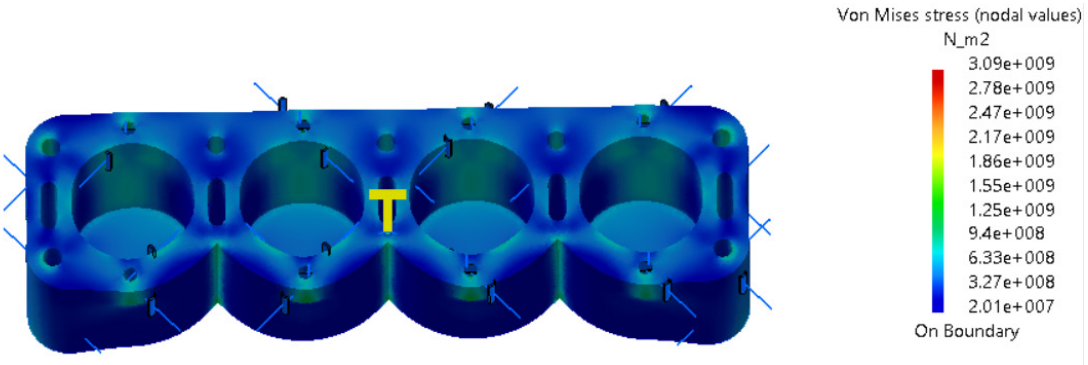


Figure 19-145 *The Von Mises Stress generated in the Engine Block*

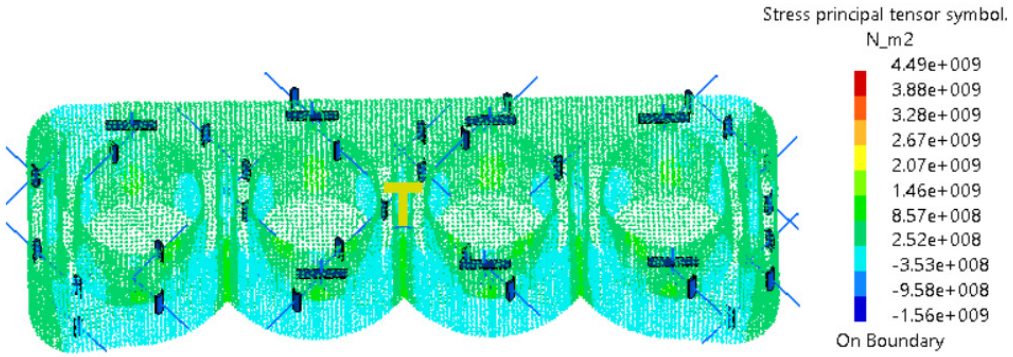


Figure 19-146 *The Principal Stress generated in the Engine Block*

Generating the Analysis Report

Next, you will generate the analysis report of the result.

1. Choose the **Generate Report** tool from the **Analysis Report** toolbar; the **Report Generation** dialog box is displayed. Specify the location to save the report and choose **OK**. The analysis report is generated in *.html* format and is displayed in your default internet browser.

After completing the study procedure and generating the analysis report, you need to save the model.

Saving and Closing the File

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box.

2. Enter **c19tut4** in the **File name** edit box and choose the **Save** button. The file saved at *C:\CATIA\c19*.

3. Close the part file by choosing **File > Close** from the menubar.

Self-Evaluation Test

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

1. The _____ tool is used to generate and save the animation view of the results generated in the postprocessing phase.

2. The maximum stress that can be applied to a material without producing permanent deformation is known as the _____ of the material.

3. The _____ tool adds a spherical joint restraint to the points of virtual parts, it constraint a part to rotate around a fixed point.

4. You can restraint the required degree of freedom by checking and clearing the required _____ and _____ check boxes from the **User-defined Restraint** dialog box.

5. The_____ tool creates a triangular or quadrangular mesh elements on the selected surface.

6. The **Beam Mesher** tool is used to mesh the _____ parts which are also known as line bodies.

7. The **3D Property** tool allows you to add 3D physical properties to a shape design by selecting a meshed wireframe geometry. (T/F)

8. The User-defined restraints are generic restraints which restrict fixing any combination of nodal degree of freedom available for any arbitrary geometry. (T/F)

9. The **Ball Joint** tool adds a spherical join restraint to the points of virtual parts. (T/F)

10. The **Translation Mesher** tool is used to create a rotation mesh part from an already defined mesh part. (T/F)

Review Questions

Answer the following questions:

1. Which of the following tools is only available with the CATIA V5 **Advanced Meshing Tools** workbench.

(a) **Extrude** (b) **Edge Fillet**
(c) **Revolve** (d) **Sweep 3D**

2. The _____are generic restraints allow you to fix any type of combination of nodal degree of freedom which is available for any arbitrary geometry.

3. The _____ tool allows you to add 3D physical properties to a shape design by selecting a meshed wireframe geometry.

4. The _____ tool is used to split quadrangle mesh elements into triangle mesh elements.

5. The_____ tool allows you to add 2D physical properties to a shape design by selecting a meshed wireframe geometry.

6. The _____tool is used to apply temperature constant to a part.

7. The **Temperature Field** tool is available in the _____ toolbar.

8. Generative Structural analysis examines the response of a structure or component to design strategies. (T/F)

9. A rigid spring virtual part is an elastic body which connects a specific point to a specific geometry. (T/F)

10. The **Ball Joint** tool adds a spherical join restraint to the points of virtual parts (T/F)

EXERCISES

EXERCISE 1

Download the file *c19_catia_exr01.zip* file from *www.cadcim.com*. The path of the file is as follows: *Textbook > CAD/CAM > CATIA > CATIA V5-6R2022 for Designers > Input file* and then extract it to save the file. Next, you will perform the static analysis of the model. The holes of the model that are fixed are shown in Figure 19-147. A load of 100 N is applied to the upper holes, refer to Figure 19-147. The model is made up of steel.

(Expected time: 45 min)

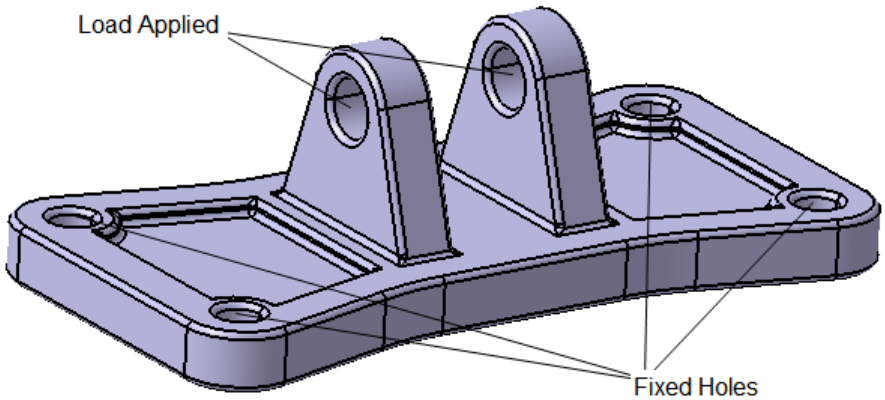


Figure 19-147 *The model for Exercise 1*

Answers to Self-Evaluation Test

1. **Animate**, **2**. elastic limit, **3**. **Ball Joint**, **4**. **Restrain Translation**, **Restrain Rotation**, **5**. **Surface Mesher**, **6**. 1D, **7**. T, **8**. F, **9**. T, **10**. T