

TABLE OF CONTENTS

Introduction	1
Generative Structural Analysis	2
Pull-down Menus	3
Insert	3
Tools	4
Generative Structural Analysis Workbench	5
Bottom Toolbar Changes	8
Preview	9
Meshing	19
Restraints	35
Clamp Restraints	35
Clamp	35
Mechanical Restraints	36
Surface Slider	36
Slider	37
Sliding Pivot	38
Ball Joint	39
Pivot	40
Advanced Restraints	41
User-Defined Restraint	41
Isostatic Restraint	42
Loads	43
Pressures	43
Pressure	43
Forces	44
Distributed Force	44
Moment	45
Bearing Load	46
Imported Force	48
Imported Moment	49
Accelerations	50
Acceleration	50
Rotation Force	51
Force Density	52
Line Force Density	52
Surface Force Density	53
Volume Force Density	54
Force Density	55
Enforced Displacement	56
Temperature Field	57
Temperature Field from Thermal Solution	58

Restraints and Loads Exercise	61
Results	67
Computing Results	67
Viewing Results	72
Imaging Tools	72
Visualization Tools	80
Creating Sensors	86
Global Sensors	86
Local Sensors	88
Resultant Sensors	91
Adaptivity	97
Managing Results	100
Results Validation	113
Color Bands	114
Virtual Parts	121
Rigid Virtual Parts	121
Smooth Virtual Parts	122
Contact Virtual Part	123
Rigid Spring Virtual Parts	124
Smooth Spring Virtual Part	126
Periodicity Conditions	128
Virtual Parts Exercise	129
Frequency Analysis	137
Distributed Mass	139
Mass Density	141
Line Mass Density	141
Surface Mass Density	142
Distributed Mass and Inertia	142
Combined Masses	143
Assembled Masses	144
Generative Assembly Structural Analysis	147
Analysis Connections	147
General Analysis Connection	147
Point Analysis Connection	148
Point Analysis Connection within one Part	149
Line Analysis Connection	150
Line Analysis Connection within one Part	151
Surface Analysis Connection	152
Surface Analysis Connection within one Part	153
Points to Points Analysis Connection	154
Point Analysis Interface	155

Connection Properties	156
Face/Face Connections	156
Distant Connections	162
Welding Connections	168
GAS Exercise	173
Analysis Connections and Connection Properties	173
Restraints and Loads	194
Computing and Viewing Results	198
Sensors	205
Saving	208
Advanced Meshing Tools	209
Advanced Meshing Tools Workbench	210
Advanced Meshing Tools	213
Meshing Methods	213
Beam Mesher	213
Surface Mesher	215
Advanced Surface Mesher	217
Octree Triangle Mesher	219
Octree Tetrahedron Mesher	222
Global Parameters	225
Local Specifications	227
Global Specifications	235
Execution Tools	236
Mesh Edition	239
Clean Holes	239
Edit Simplification	239
Merge	240
Imposed Elements	241
Remesh Domain	241
Remove Mesh by Domain	242
Lock Domain	242
Edit Mesh	242
Split Quadrangles	244
Mesh Analysis Tools	251
Mesh Operators and Mesh Transformations	264
Extrude Mesher with Translation	274
Extrude Mesher with Rotation	275
Extrude Mesher with Symmetry	276
Extrude Mesher along a Spline	277
FMS Continued Exercises	280
Tetrahedron Filler	283
Coating 1D Mesh	286
Coating 2D Mesh	288
Move Mesh Nodes	292
Sweep 3D	296

Welding Meshing Methods	300
Spot Welding Connections	300
Seam Welding Connections	301
Surface Welding Connections	302
Nodes to Nodes Connection Mesh	303
Node Interface Mesh	304
Import/Export Mesh	305
FMS Exercise	307
2D-3D Meshing Exercise	339
Composite Panel Exercise	349
Product Engineering Optimizer	373
Miscellaneous	383
Data Mapping	383
Periodic Conditions	386
Grouping	388
Thermo-Mechanical Loads	392
Visualization Transferred onto Mesh	393
Self-balancing on Load set	398
Practice Problems	399
Problem #1	399
Problem #2	401
Problem #3	403
Problem #4	405
Problem #5	407
Problem #6	408
Problem #7	409
Problem #8	410
Appendix A - Options	411
General - Parameters and Measure - Knowledge	411
Analysis & Simulation - External Storage	412
Analysis and Simulation - General	413
Analysis and Simulation - Graphics	414
Analysis and Simulation - Post Processing	415
Analysis and Simulation - Quality	416
Analysis and Simulation - Reporting	417

Introduction

CATIA Version 5 Generative Structural Analysis

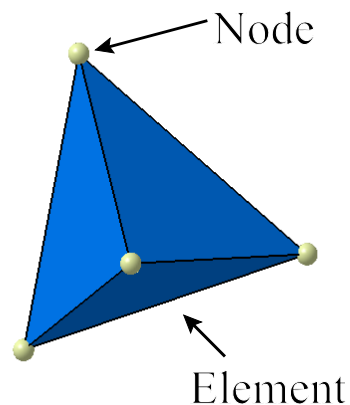
Upon completion of this course the student should have a full understanding of the following topics:

- Applying a mesh
- Defining restraints
- Defining loads
- Defining virtual parts
- Applying an isotropic material
- Defining groups
- Applying each analysis type
- Managing results
- Refining results

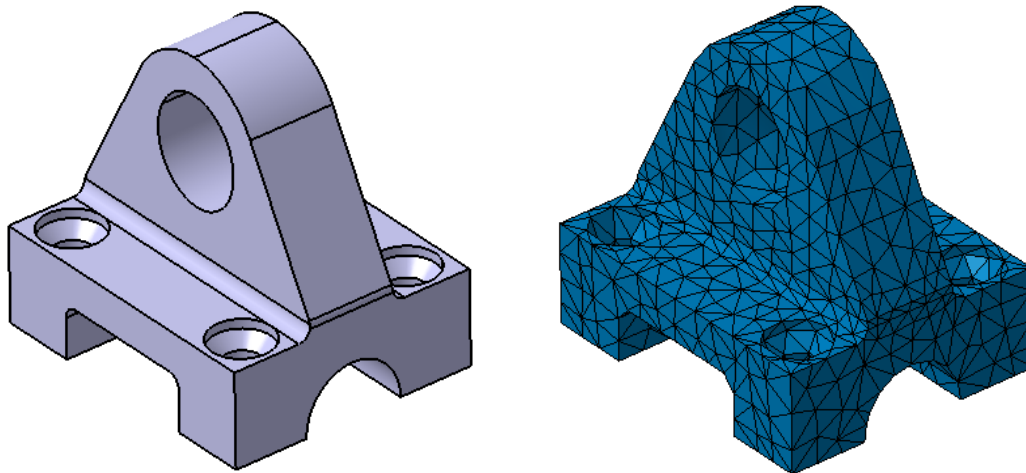
Generative Structural Analysis

Generative structural analysis is useful to acquire the various structural characteristics of your parts and products in a 3D environment. Using these tools allows you to analyze your parts or products to determine their structural qualities before they are manufactured.

The Generative Structural Analysis workbenches utilize the Finite Element Method of numerical approximation. This method works by approximating the model by breaking it down into smaller, more simplified pieces. These broken down pieces are referred to as elements. Elements are connected together at what are commonly known as nodes. The illustration below provides greater clarity.



Below is an original model and its finite element model representation. The representation will vary based on the size and shape of the elements. This allows the user to customize analysis. Based on the simplicity and size of the elements, the analysis can be very simple or very complex based on the requirements of the analysis.




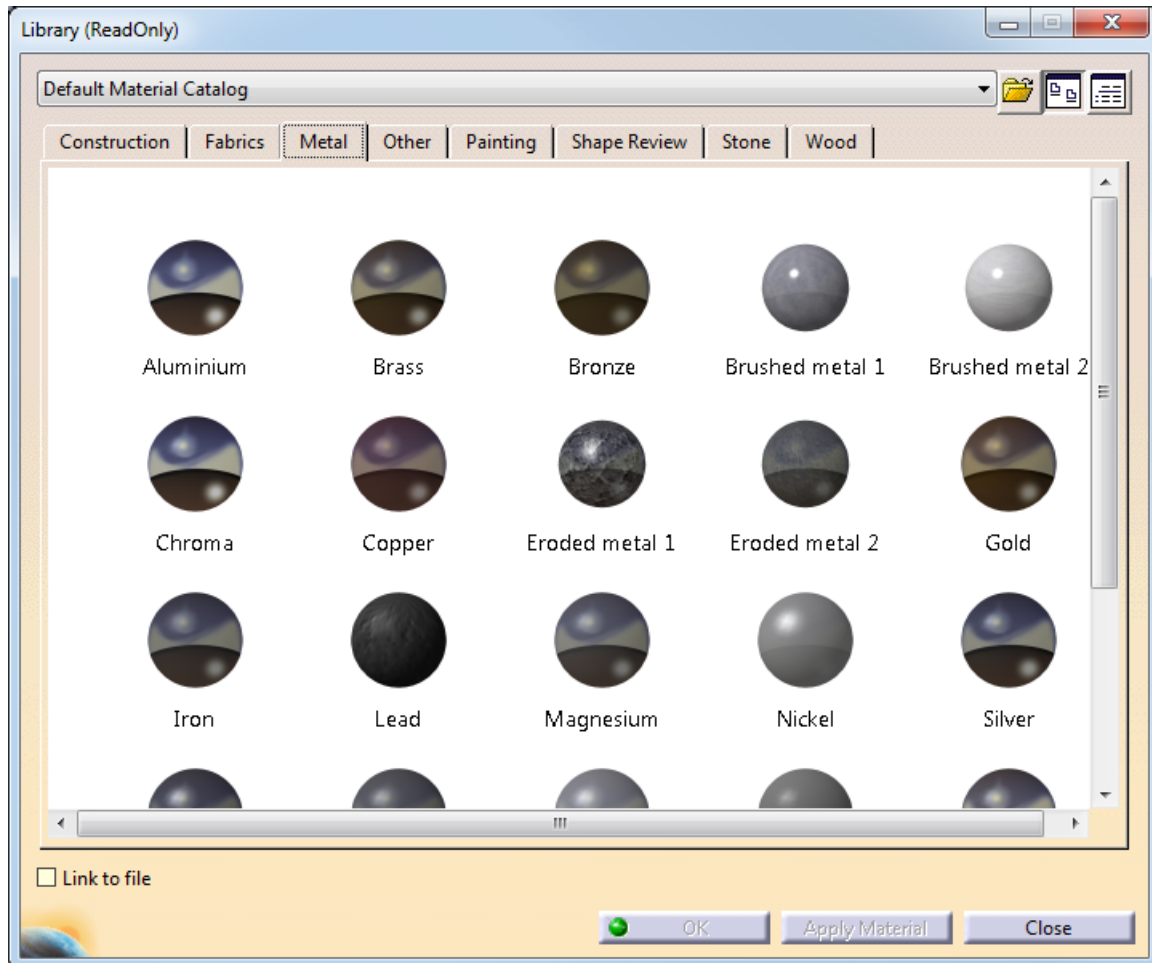
It is important to understand that to fully utilize the tools in this course you should be familiar with the fundamentals of the Finite Element Method. It is not the intention of this course to teach you Finite Element Analysis. However, it is not a requirement of this course that you fully understand the Finite Element Method since utilizing the tools in this course do not require it.

Preview

This section will give you a brief overview of what the ensuing sections will cover in detail.

Open the Basic document. This is a basic part. You must have a material defined for any part that you wish to create an analysis on. Therefore, the first step will be to apply a material to the part.

Select the Apply Material icon in the bottom toolbar.  The *Library* window appears.

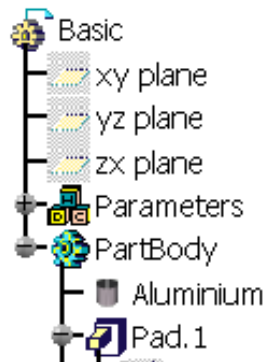


From this library, you may select any type of material that is listed and apply it to your part.

Select the *Metal* tab. The library changes to display all of the metal materials.

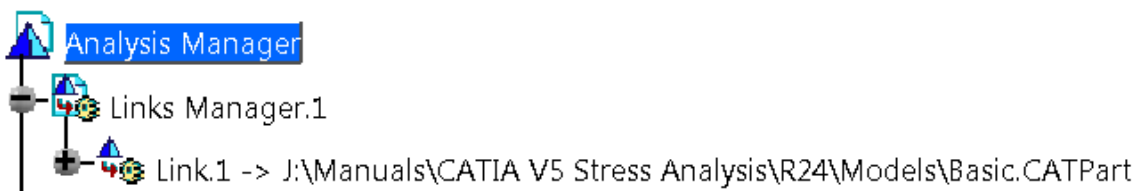
Select *Aluminium* from the list and select the *Partbody* from the specification tree. It is a good idea to apply the material to the partbody and not the part model itself.

Select **OK** to apply the material. It should appear in the tree as shown.

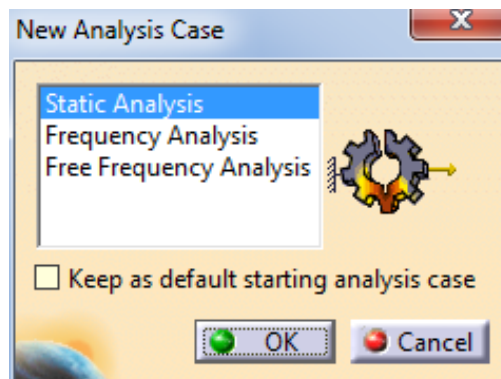


Now that the part has a material applied to it, an analysis may be created.

Switch to the Generative Structural Analysis workbench. It is located in the *Start* menu under *Analysis and Simulation*. This will create an analysis of the part. The CATAnalysis will be linked back to the original part as shown below.

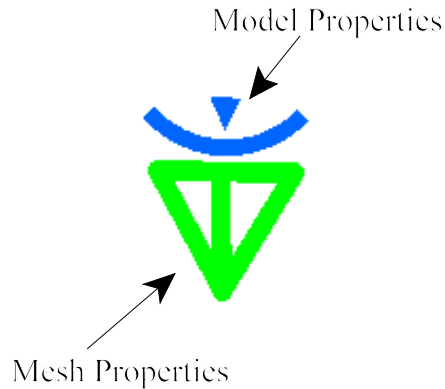



The *New Analysis Case* window appears. You have to define what type of analysis you would like to do.

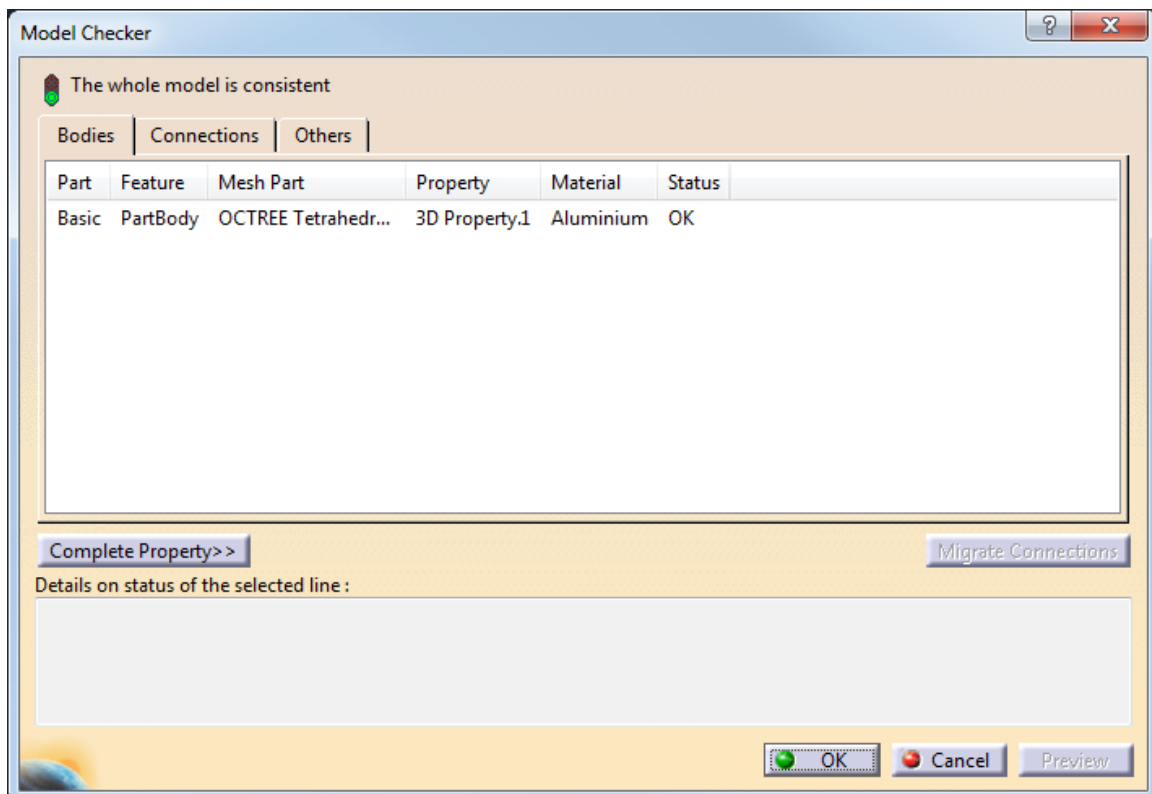


Select the *Static Analysis* case and select **OK**. This creates a *Static Case* analysis document. You will see the *Static Case* branch in the specification tree. You actually have a new document up at this point.

By default, a mesh and some model properties are applied to each body in the part when the analysis is created. For now, we will work with the default mesh and properties. Later in the course, we will experiment with adjusting the mesh and properties in order to refine the results. The mesh and model properties are represented by the following symbols in the 3D environment.



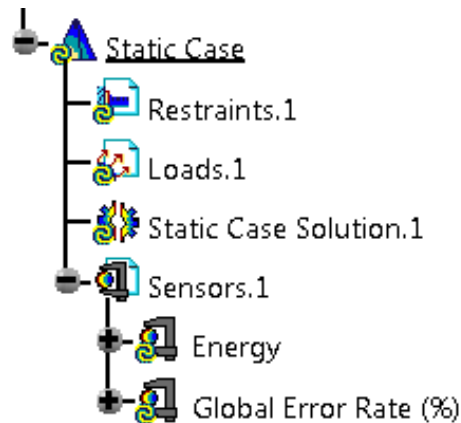
Select the **Model Checker** icon.  The *Model Checker* window appears.



The model checker will show you all of the specifications of the model and determine if the model is okay to use or not. If the model is okay, the *Status* will read *OK*. If there is something wrong, the *Status* will read *KO*. This will play a more important part in the steps later on when there are more details defined.

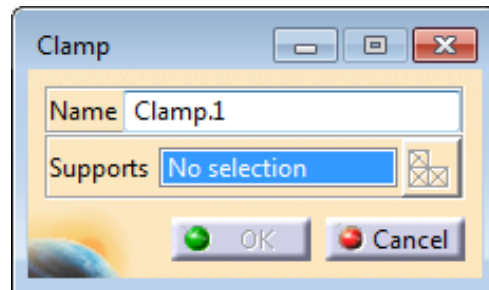
Select **OK**. The model is ready for an analysis.

Take a look at the *Static Case* analysis that was inserted into the specification tree. Notice that there are several branches underneath the *Static Case*.

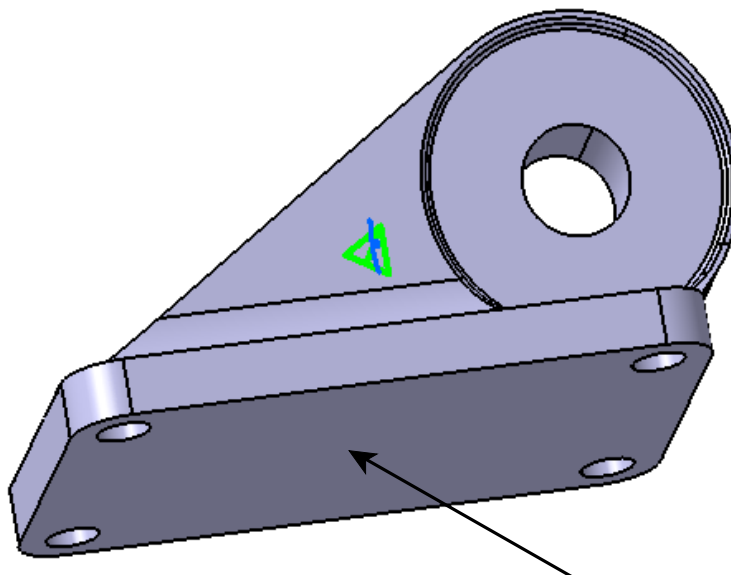


There are no restraints or loads applied to the model, and therefore, there are no solutions to the static case. Defining the restraints and loads on the model are the only remaining steps to get some results from your analysis.

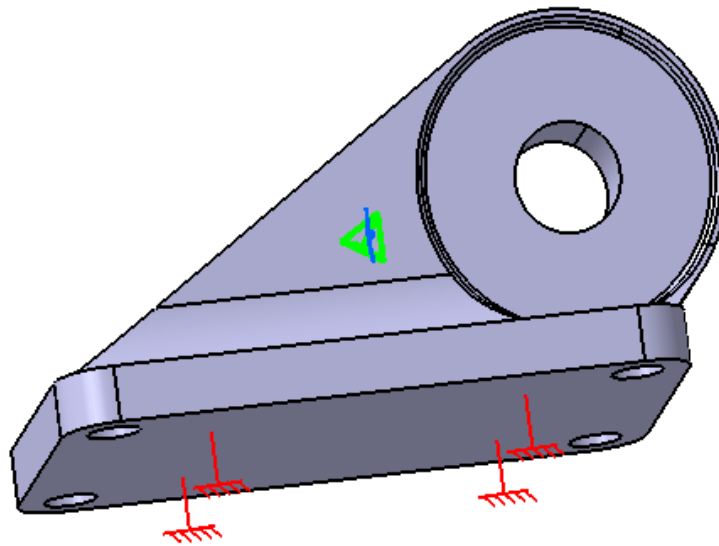
Select the **Clamp** icon.  The *Clamp* window appears.



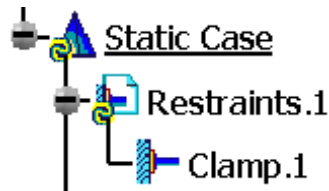
Select the **bottom face of the part as shown**. This will define the support for the clamp.



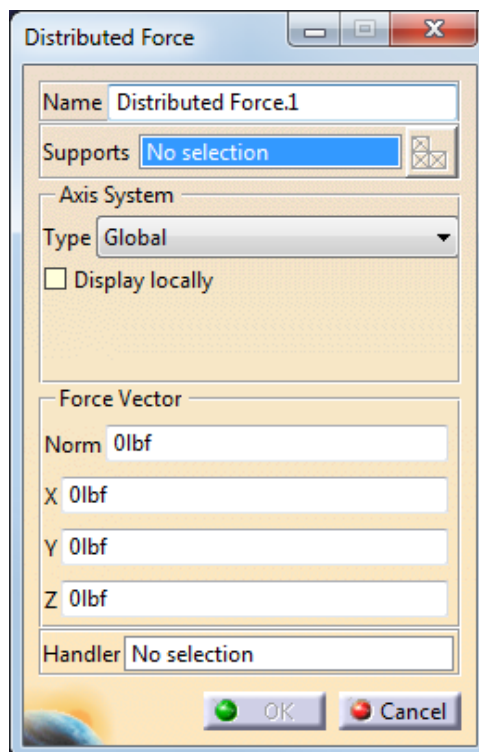
Select **OK**. The clamp should appear on the model as shown.



The clamp appears under the *Restraints.1* branch of the specification tree.

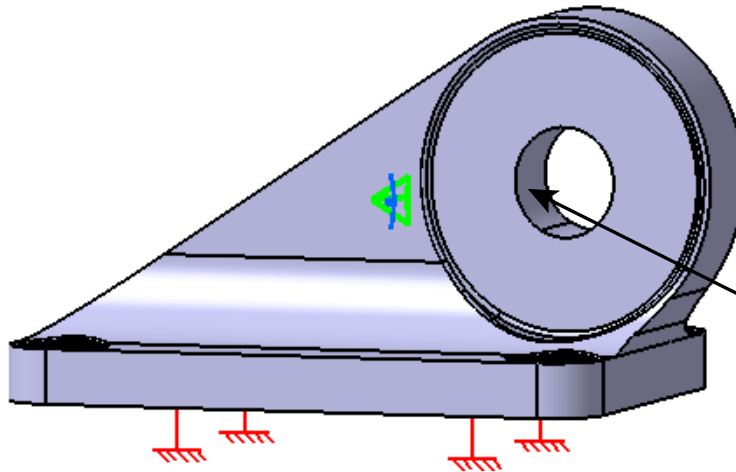


Select the **Distributed Force** icon.  The *Distributed Force* window appears.



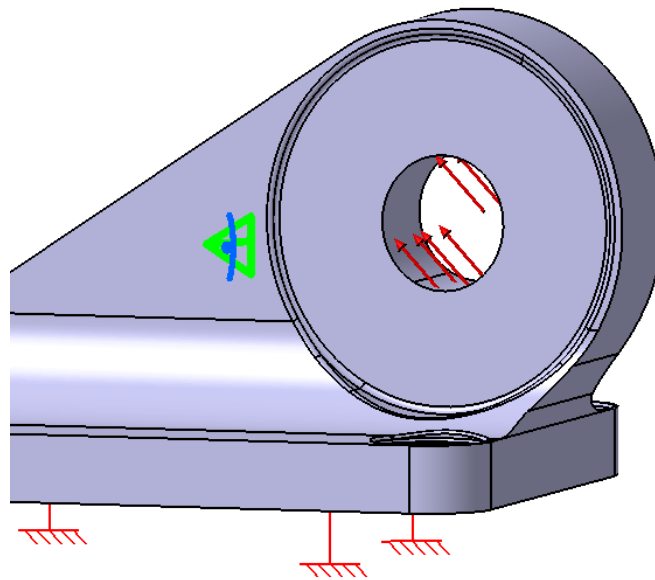
<i>Supports</i>	Defines the supports that the force acts on
<i>Axis System</i>	
<i>Type</i>	Defines the axis system that the force will be based upon
<i>Display locally</i>	Displays a local axis to help show orientation with respect to the part's local axis
<i>Force Vector</i>	
<i>Norm</i>	Defines the force in the normal direction
<i>X</i>	Defines the force in the X-direction
<i>Y</i>	Defines the force in the Y-direction
<i>Z</i>	Defines the force in the Z-direction
<i>Handler</i>	Defines the point where the forces are applied. By default this point is at the centroid of the selection.

Select the hole as shown for the support.

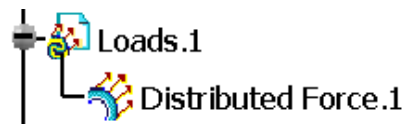


Set the X, Y and Z values to be 50.0, 50.0 and 0.0 respectively. This will yield a *Norm* vector of 70.711 lbf.


Select **OK**. The force is applied as shown.



The distributed force appears under the *Loads.1* branch in the specification tree.



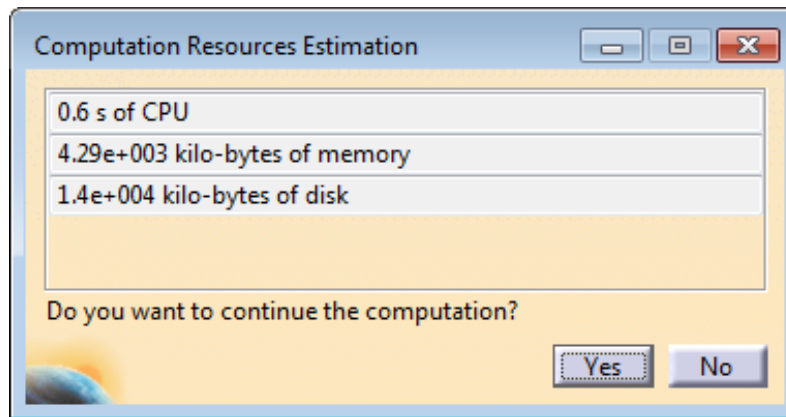
This completes the creation of the necessary elements required for the analysis model. Now the analysis is ready to be computed. It is recommended that the model be saved before you compute the analysis.

Select the **Compute** icon.  The *Compute* window appears.




- | | |
|---|--|
| <i>All</i> | Computes everything |
| <i>Mesh Only</i> | Computes the mesh only |
| <i>Analysis Case Solution Selection</i> | Allows you to select a specific case solution to compute |
| <i>Selection by Restraint</i> | Computes based off of an individual constraint |

Select **All** and select **OK**. The *Computation Resources Estimation* window appears.

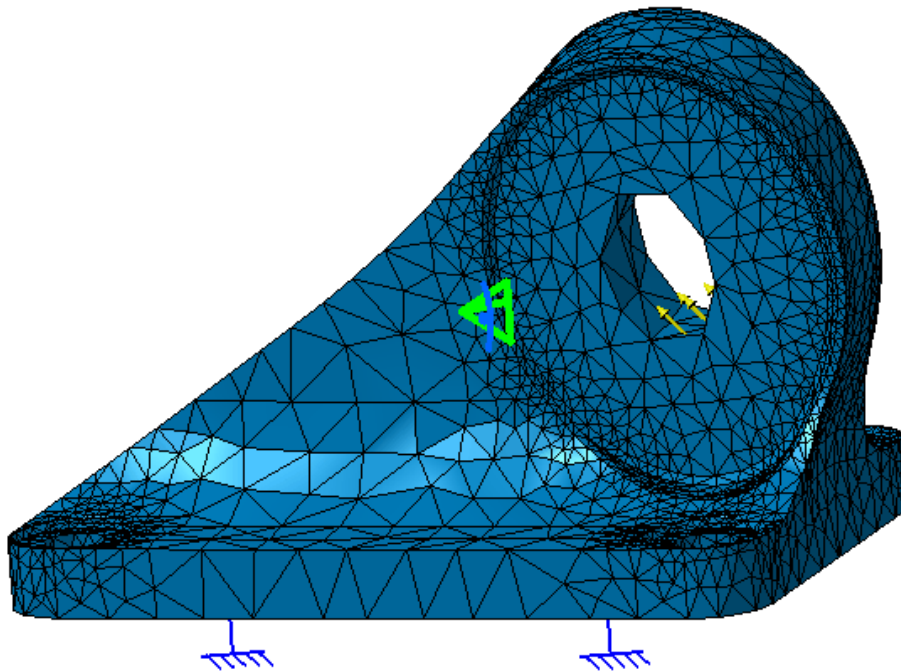


This window gives an approximation on the time the analysis will take as well as how much memory and disk space will be necessary. You may want to check to make sure that the computer has the necessary memory and disk space.

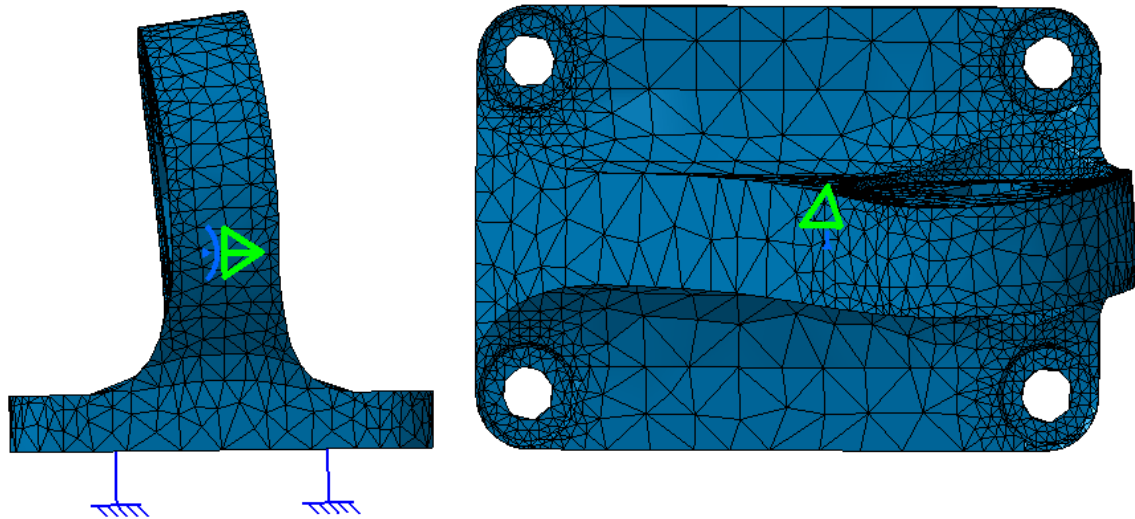
Select **Yes**. The analysis is computed.

Select the **Deformation icon**.  This will display the deformation of the model based on the applied restraints and loads.

The model should appear as shown.



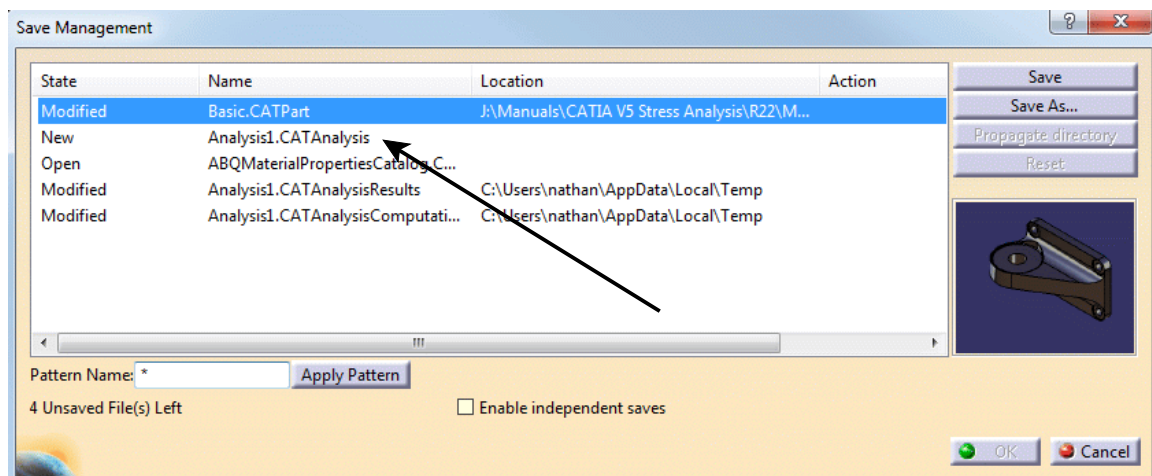
Rotate the model around to view the deformation more clearly.



Notice that the part deforms off to one side even though the restraints and loads were applied symmetrically. The reason for this is the fact that the mesh is automatically generated and, therefore, not necessarily symmetric. Additional restraints would be necessary to force the part to behave correctly.

There are many other things that could be done in order to acquire more results. However, at this time we will stop here. The next thing that needs to be done is to save the analysis.

From the *File* pull down, select *Save Management*. The *Save Management* window appears.

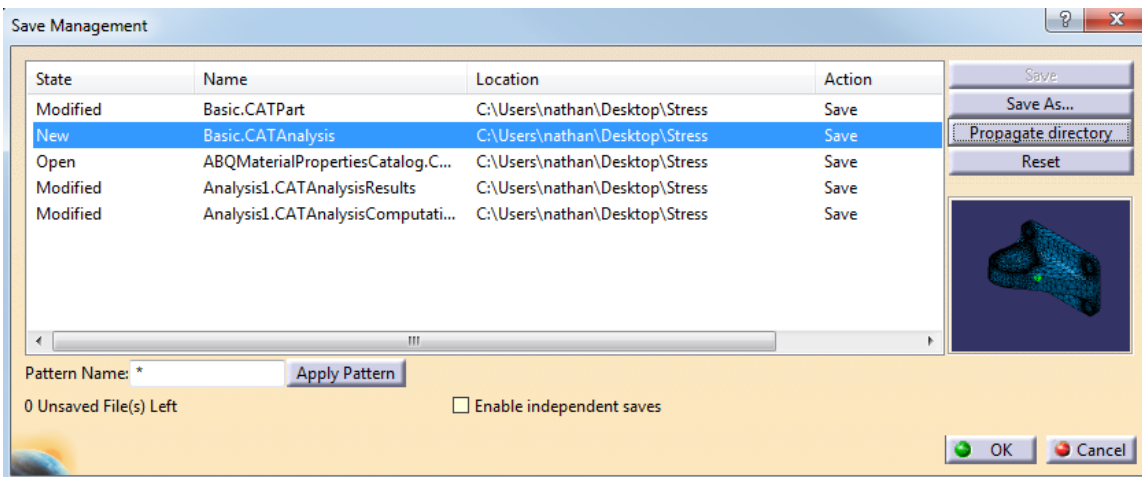


Notice the part and analysis both need to be saved. There are also two temporary files that need saved as well. The two extra files are the results and computations files.

Select *Analysis1.CATAnalysis* as shown above and select *Save As*. Define a place to save the file. It is a good idea to create a new folder and save everything that pertains to the analysis in the same folder.

Save the analysis as Basic. You should be returned to the *Save Management* window.

Select the **Propagate directory** option. This will save not only the analysis and part, but also the temporary files into the directory that you defined. This is important because, by doing so, you will not have to worry about breaking links between any of the necessary components.



Select **OK** to save the documents.

Close all documents.

Virtual Parts Exercise

This exercise will walk you through applying a couple of virtual parts to a model and then applying restraints and loads to the model as well.

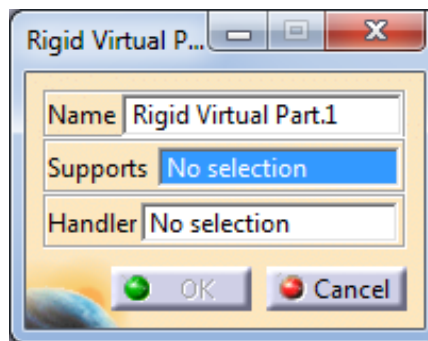
Open the Virtual Parts document. This is just a part model.

Switch to the Generative Structural Analysis workbench. The *New Analysis Case* window appears.

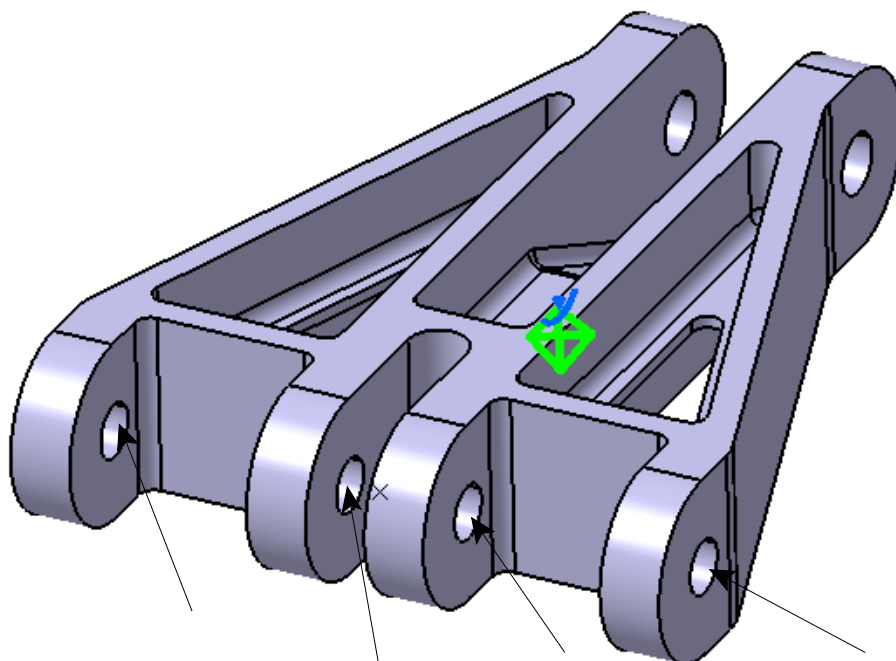
Select *Static Analysis* and select *OK*. The analysis is started.

We will create a virtual part to represent a bolt passing through the holes on the end of the model. A rigid virtual part will be used since we want to stiffen the geometry that it will be attached to.

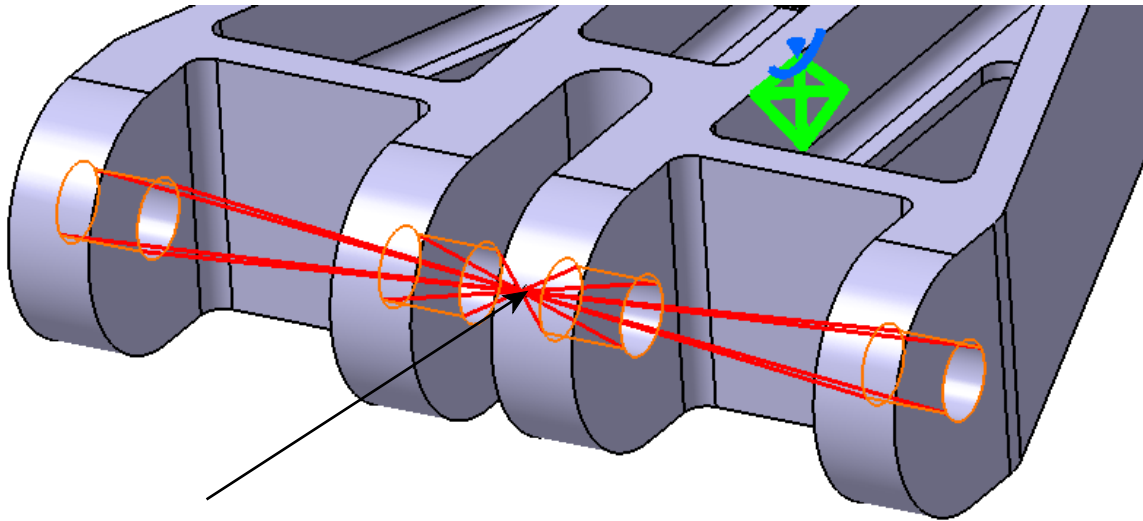
Select the Rigid Virtual Part icon.  The *Rigid Virtual Part* window appears.



Select the inside of the four holes as shown to define the *Supports* for the virtual part. Make sure that you select the inner faces and not the edges.



Notice that the virtual part appears as shown.

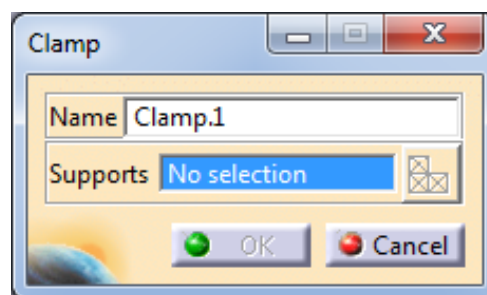


Select the point as shown above to define the *Handler* location for the virtual part. It is a part design point. In this case it was not really necessary to define the *Handler* since the centroid for the four faces that were selected is located exactly where we want the handler point anyway. If it was not, however, you would just create a point in the model and define that to be the handler point.

Select **OK**. The virtual part is created.

This virtual part will not allow the geometry supports to deform.

Select the **Clamp** icon.  The *Clamp* window appears.

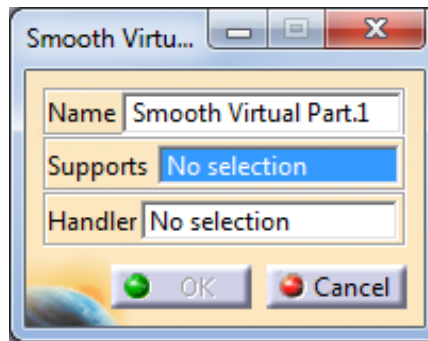


Select the **rigid virtual part** to define the *Supports* for the restraint. This will restrain the virtual part so that it cannot move at all.

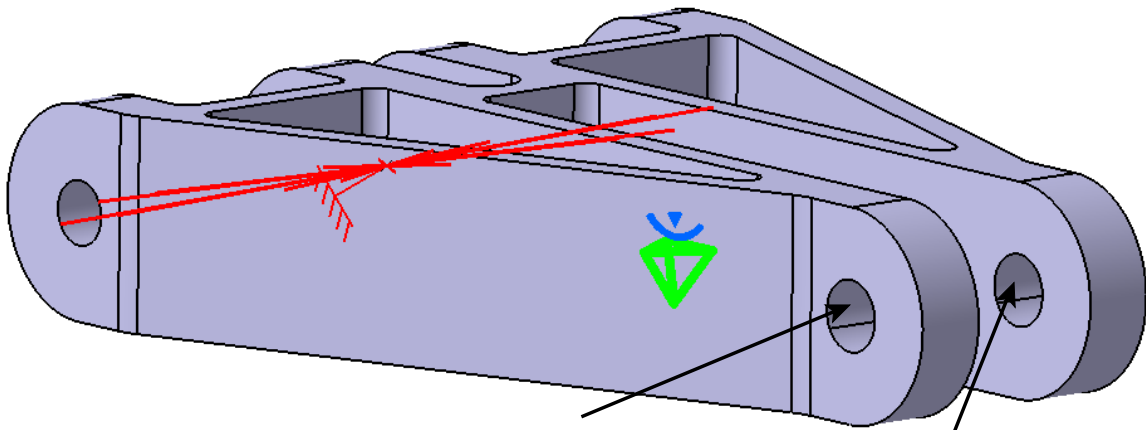
Select **OK**. The virtual part is restrained.

Now we will create another virtual part through the holes on the opposite side of the model.

Select the **Smooth Virtual Part** icon.  The *Smooth Virtual Part* window appears.



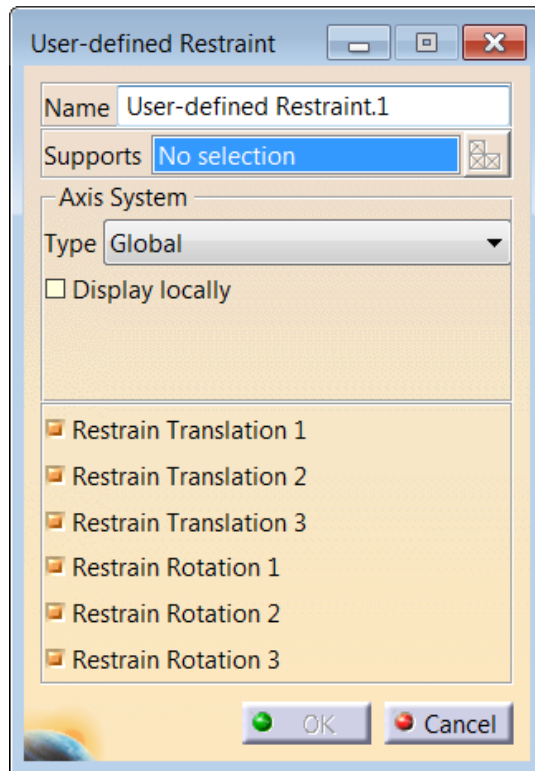
Select the inner faces of the two holes as shown to define the *Supports* for the virtual part.



Since we want the *Handler* point to be at the centroid of the two surfaces, we don't need to define the *Handler*.

Select *OK*. The virtual part is created. A smooth virtual part was chosen so that the geometry will deform based on its own properties.

Select the **User-Defined Restraint** icon.  The *User-Defined Restraint* window appears.

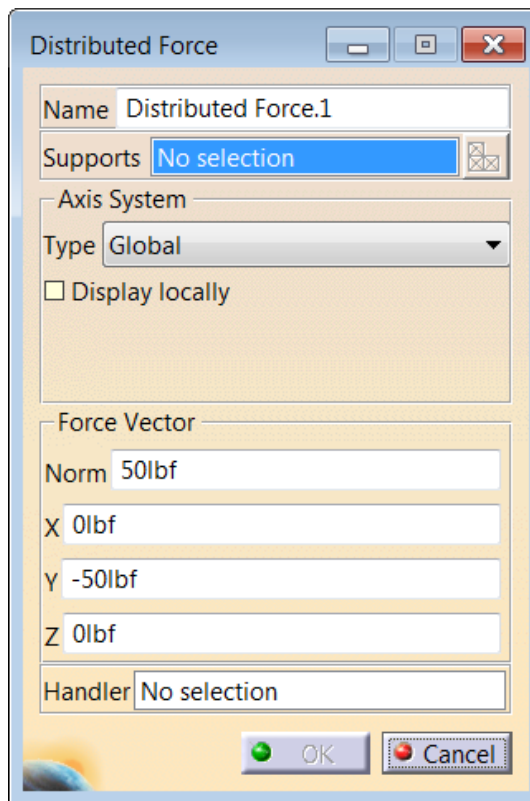


Select the smooth virtual part to define the *Supports* for the restraint.

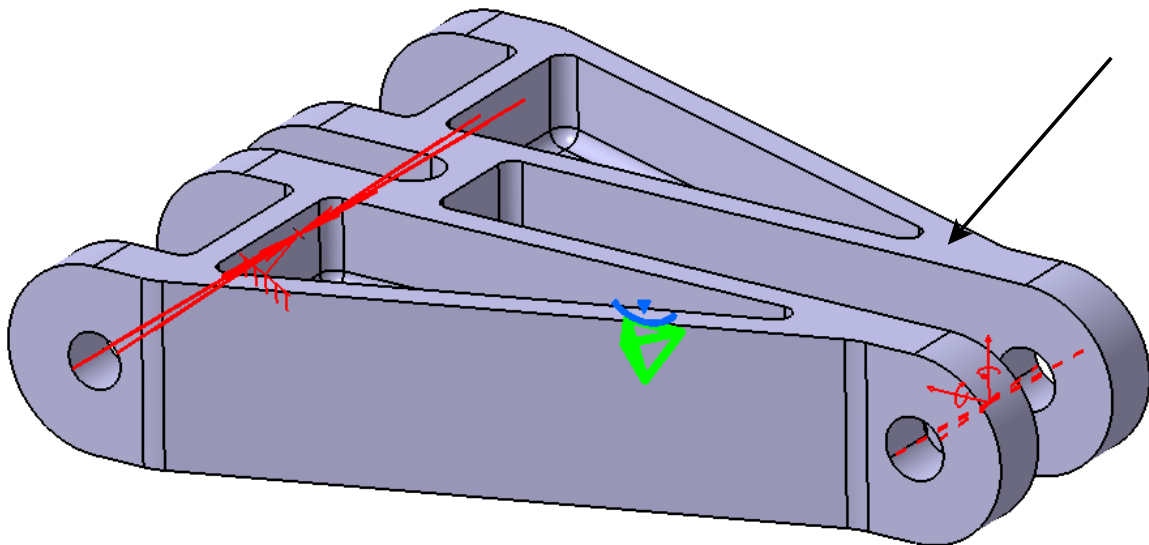
Release the translation and rotation in direction 2 and select *OK*. The restraint is created. We want the part to be able to slide and rotate about the axis of the holes.

Now we will apply loads to the model.

Select the **Distributed Force** icon.  The *Distributed Force* window appears.

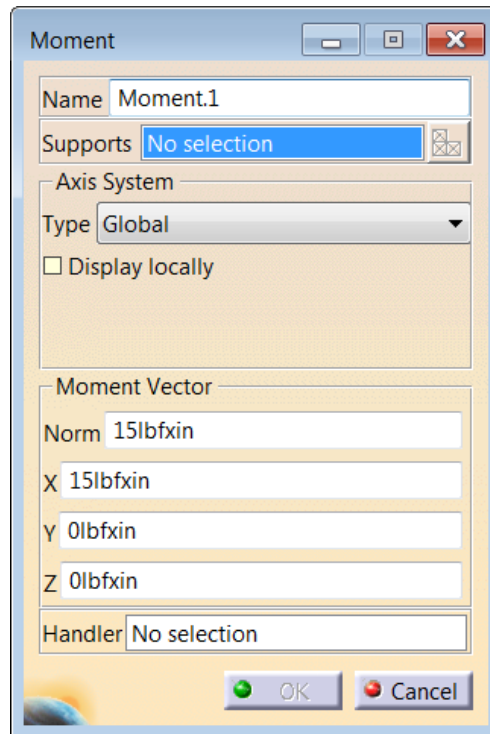


Select the face as shown to define the *Supports* for the load.



Set the force in the *Z* direction to be **-50.0** and select *OK*. The load is created.

Select the **Moment** icon.  The *Moment* window appears.

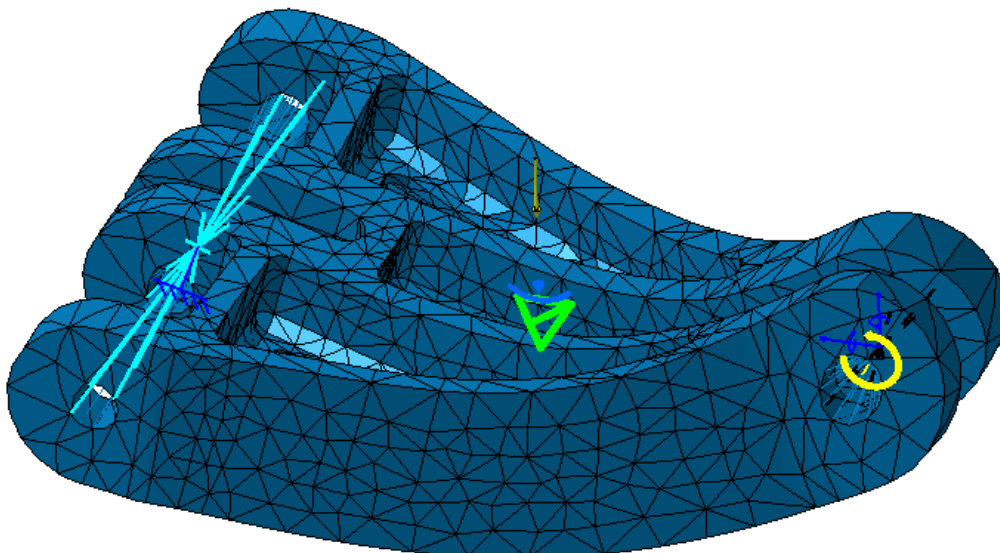


Select the smooth virtual part to define the *Supports* for the moment. The moment will act through the handler point of the virtual part.

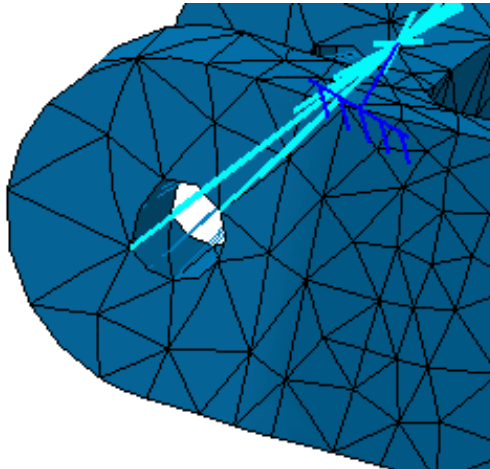
Set the *Y* direction to be 15 and select *OK*. The moment is created. Make sure that the other two values are zero.

Compute the analysis.

Select the **Deformation** icon.  The model is deformed.

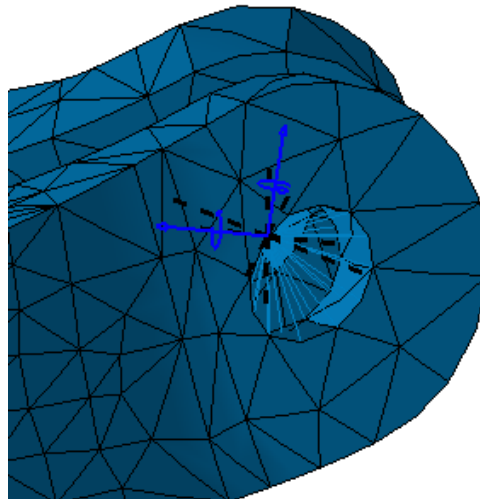


Take a look at the holes on the back of the model.



Notice that the holes are all still fairly round. These holes had a rigid virtual part attached to them which prevented them from being deformed.

Look at the holes on the front of the model.



Notice that the holes appear to be more oval now. These holes had a smooth virtual part attached to them, so they were allowed to deform based on their properties. The moment has been hidden for better viewing.

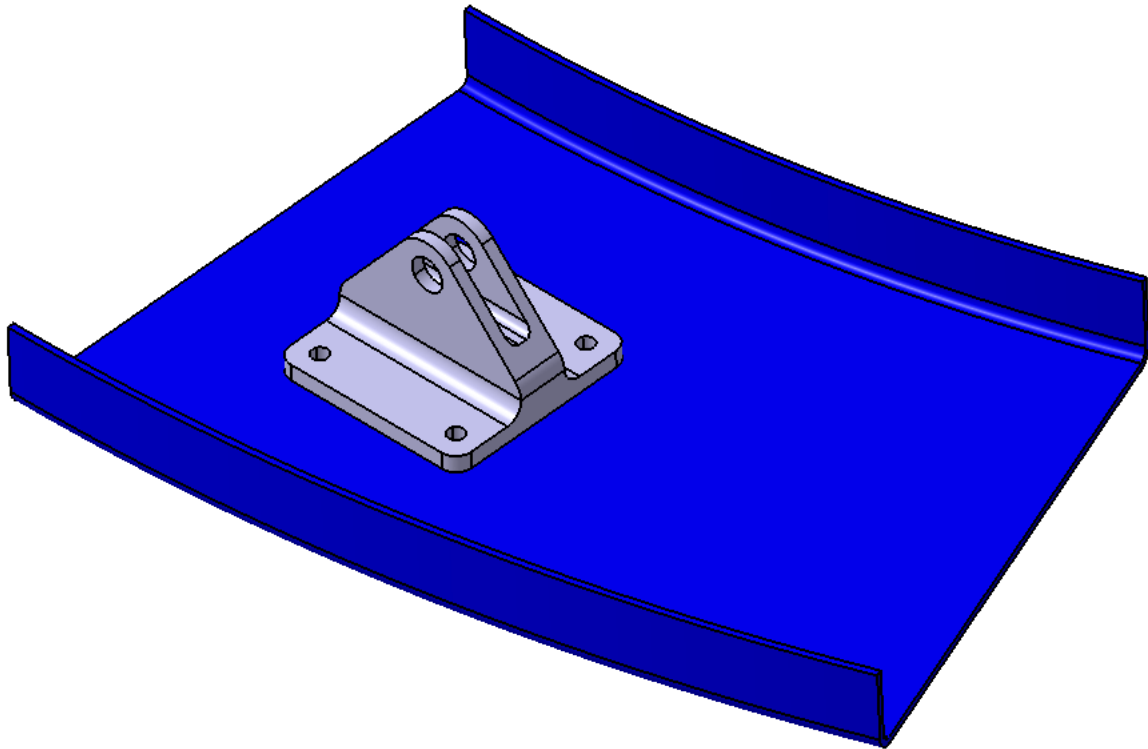
Virtual parts can be used to represent parts that have not been modeled or even to simply exert more control on how a restraint or load will be applied upon the model.

Save and close the documents.

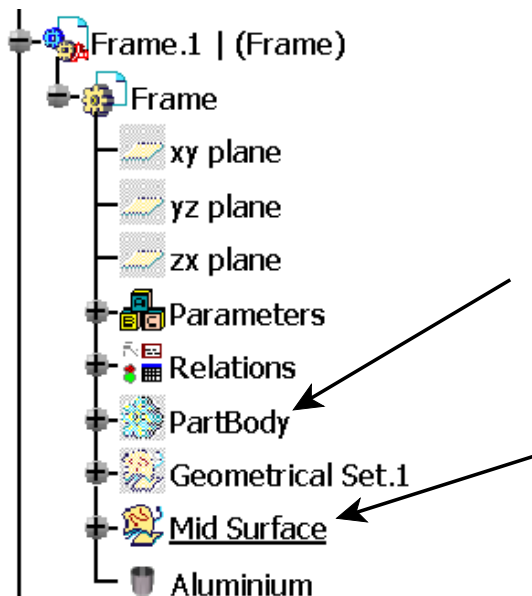
2D-3D Meshing Exercise

This exercise will involve using solid meshing and surface meshing in combination to complete an analysis.

Open the **2D-3D Meshing** document located in the *2D-3D Meshing* folder. The two parts in the assembly have already been constrained together.

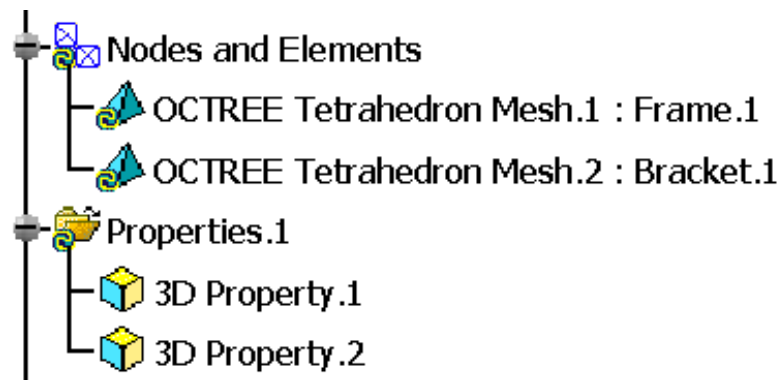


Expand the **Frame** part and hide the *PartBody* and show the *Mid Surface* geometrical set. The mid surface has already been created for you.



Switch to the Generative Structural Analysis workbench and create a Static Analysis.

Expand the *Nodes and Elements* branch and the *Properties* branch in the specification tree. It should appear as shown.




Notice a mesh and a property was automatically generated for each of the parts. Unfortunately, the mesh and property for the Frame are incorrect, because they were based off of the solid. Since you will be using a surface mesh for this part, the automatically generated mesh and property are not needed.

Delete the mesh and the property that corresponds with the Frame. To see which property is tied to the Frame, expand the Frame part in the specification tree so that you can see the *PartBody*. When you select the *Property* in the tree, it should highlight the *PartBody* that it is attached to.

A mesh must now be created for the Frame.

Switch to the Advanced Meshing Tools workbench.

Select the **Advance Surface Mesher** icon, then select the surface in the display.  The *Global Parameters* window appears.

Set the *Mesh size* to be 0.25in. Set the *Element type* to be *Linear* and turn on the *Minimize triangles* option. All other options in the *Mesh* tab should be deactivated.

In the *Geometry* tab, set the *Constraint sag* to be 0.025in and turn on the *Automatic curve capture* option with a *Tolerance* of 0.06in.

Select the **Multi Selection** icon.  The *Geometry Selector* window appears.

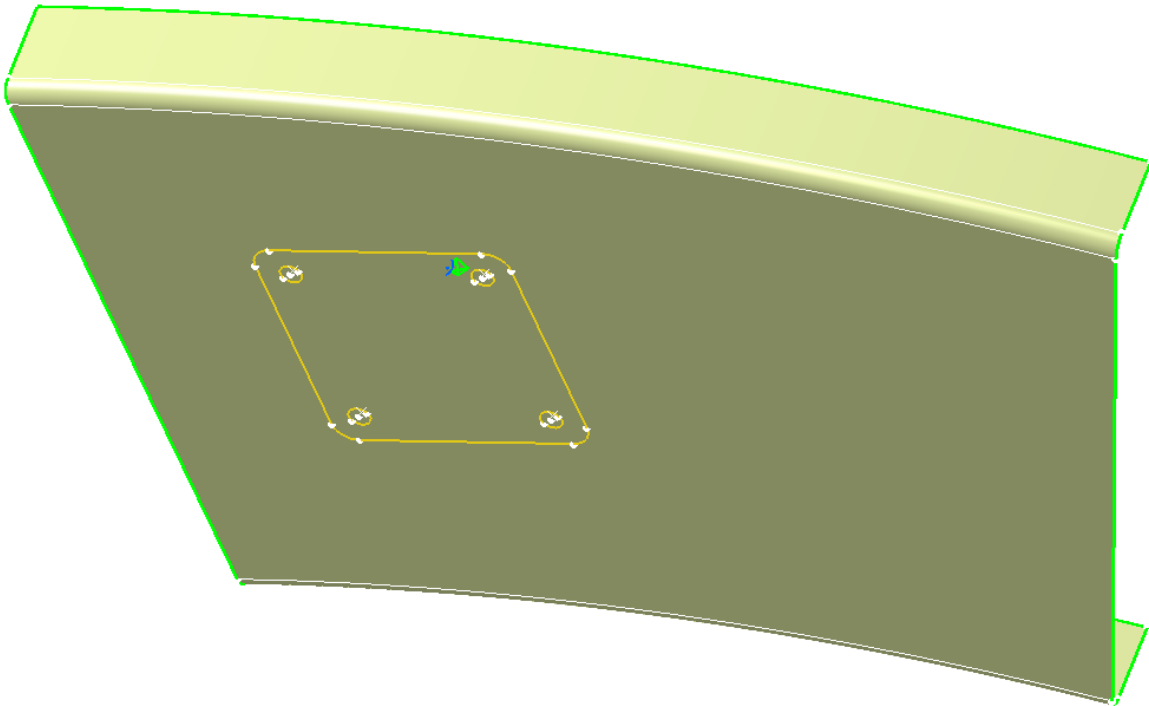
Select the bracket from the display. The entire body will be selected.

Select *OK* to the *Geometry Selector* window and the *Global Parameters* window. You are switched to the Surface Meshing workbench. You should notice yellow curves matching the outline of the Bracket's interface with the Frame.


Expand the Bracket part and show the *Fastener Locations* geometrical set. The set contains four points specifying where the Bracket will be attached to the Frame. Spot welding connections will be used to simulate the connections at those locations.

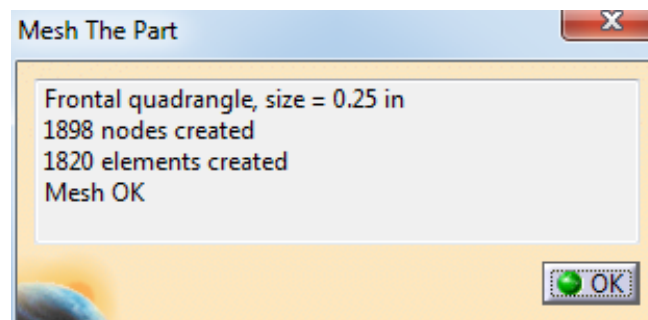
Select the *Add/Remove Constraints* icon.  The *Add/Remove Constraints* window appears.

Select the *Points* tab and select the four points from the display. It may be easier to select the points from the back side of the surface as shown.

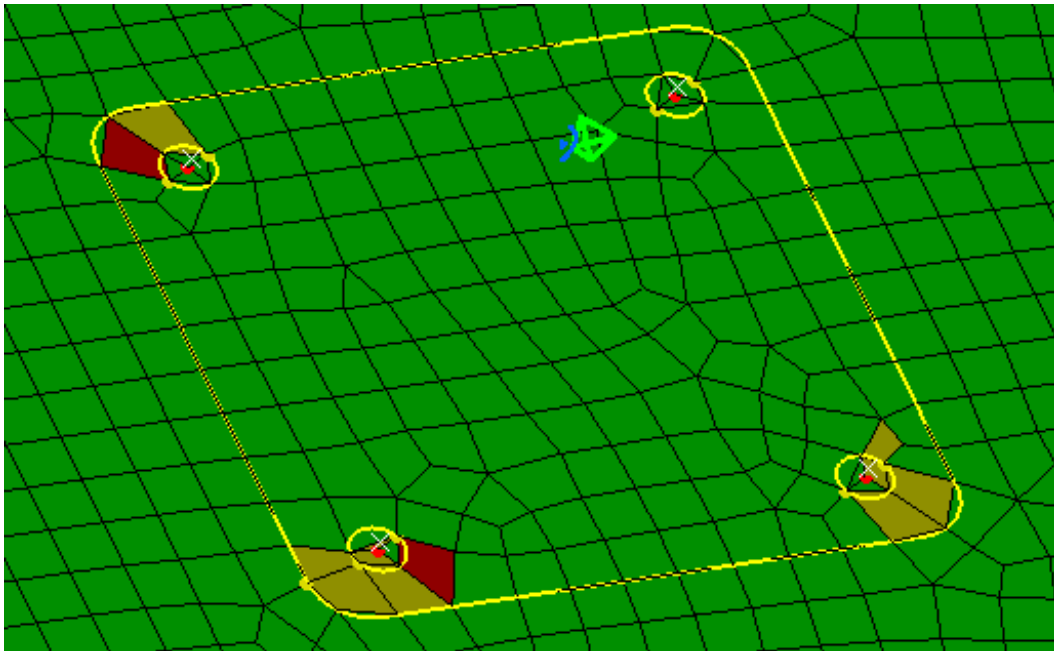


Select *OK*. This will force nodes to be created at the four point locations.

Select the *Mesh the Part* icon.  The mesh is created and the *Mesh The Part* window appears.

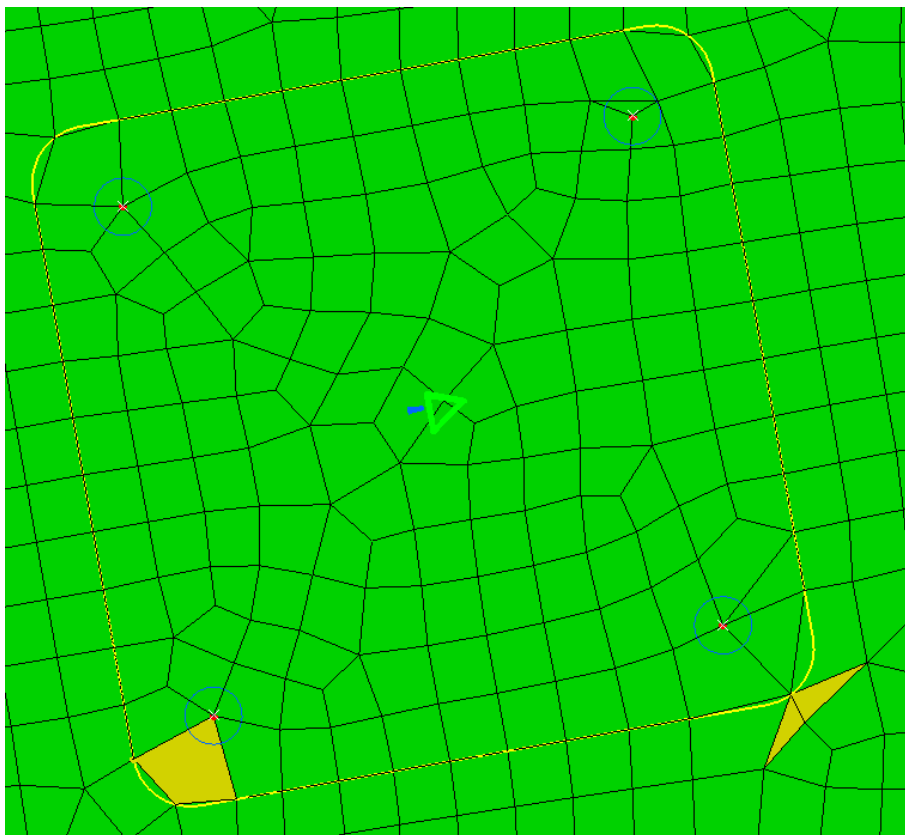



Select **OK**. Notice there are a few poor elements.

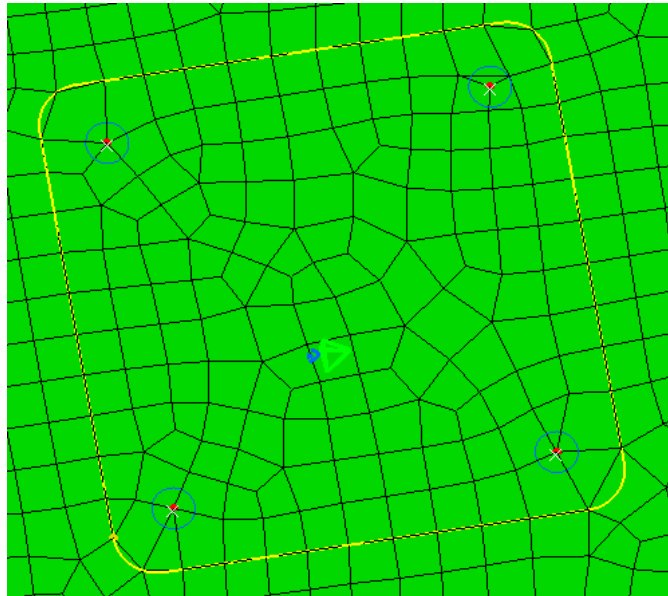


Select the **Edit Simplification** icon.  The *Edit Simplification* window appears.

Turn on both remesh options and select the eight yellow half circles that represent the holes in the Bracket. The mesh should update as shown.



Use the **Edit Mesh** icon to split the last few yellow elements.  They should appear similar to the picture below.



Select the **Exit** icon. You are returned to the Advanced Meshing Tools workbench.

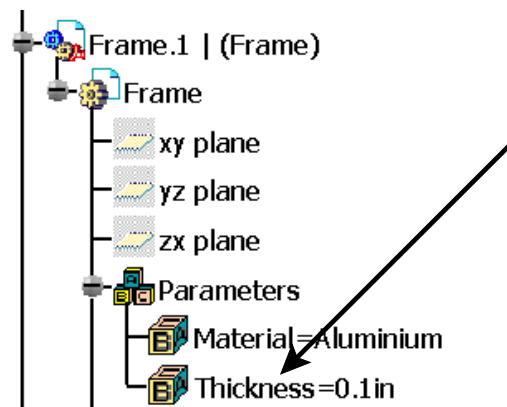
Switch to the **Generative Structural Analysis** workbench.

Select the **2D Property** icon.  The *2D Property* window appears.

Select *Advanced Surface Meshing.1* to define the *Supports* for the property.

Right select in the *Thickness* field from the *2D Property* window and select *Edit Formula* from the contextual menu. The *Formula Editor* window appears.

Select the *Thickness* parameter from the Frame model to define the formula.



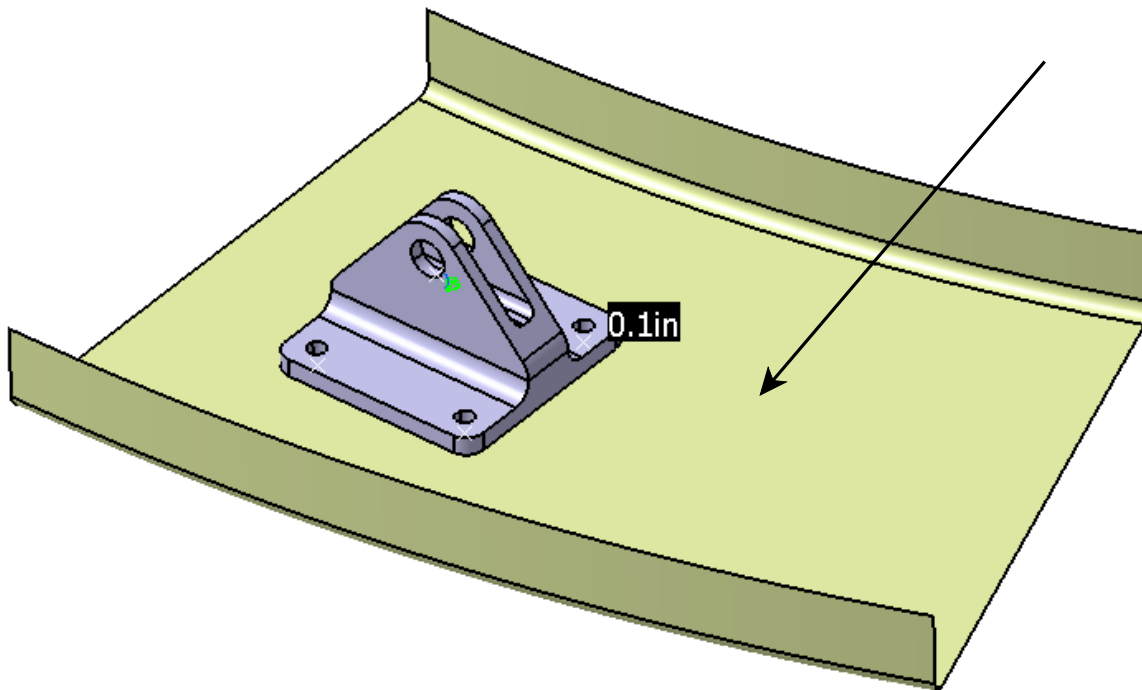
Select **OK** to the *Edit Formula* window.

Select **OK** to the *2D Property* window. This will specify that the surface mesh will behave as if it were 0.1 inches thick.

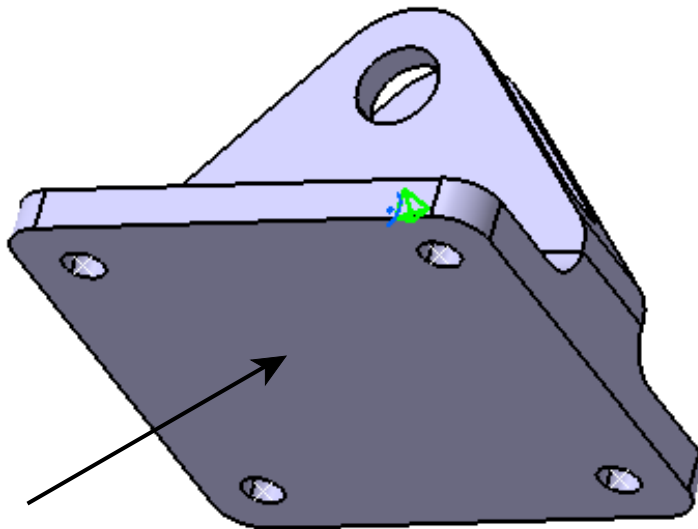
Connections need to be defined between the two parts.

Select the **General Analysis Connection** icon.  The *General Analysis Connection* window appears.

Select the face as shown to define the *First component* for the connection.



Hide the surface and select the bottom of the bracket as shown to define the *Second component* for the connection.



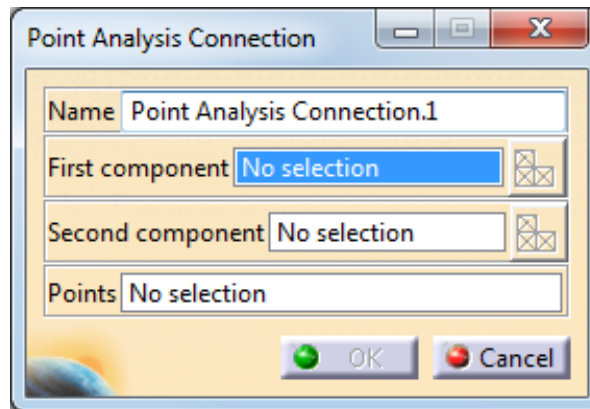
Select **OK**. The analysis connection is created.

Show the surface.

Select the **Contact Connection Property icon**.  The *Contact Connection Property* window appears.

Select the **General Analysis Connection.1** and select **OK**. The property is defined. This will specify that the two faces are connected together and cannot protrude into one another.

Select the **Point Analysis Connection icon**.  The *Point Analysis Connection* window appears.

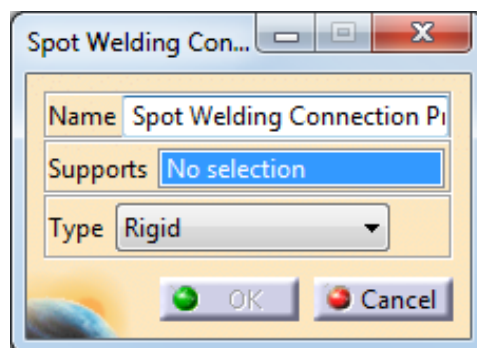


Select the surface to define the *First component* and the Bracket to define the *Second component*.

Select the *Fastener Locations* geometrical set from the Bracket part to define the *Points* selection.

Select **OK**.

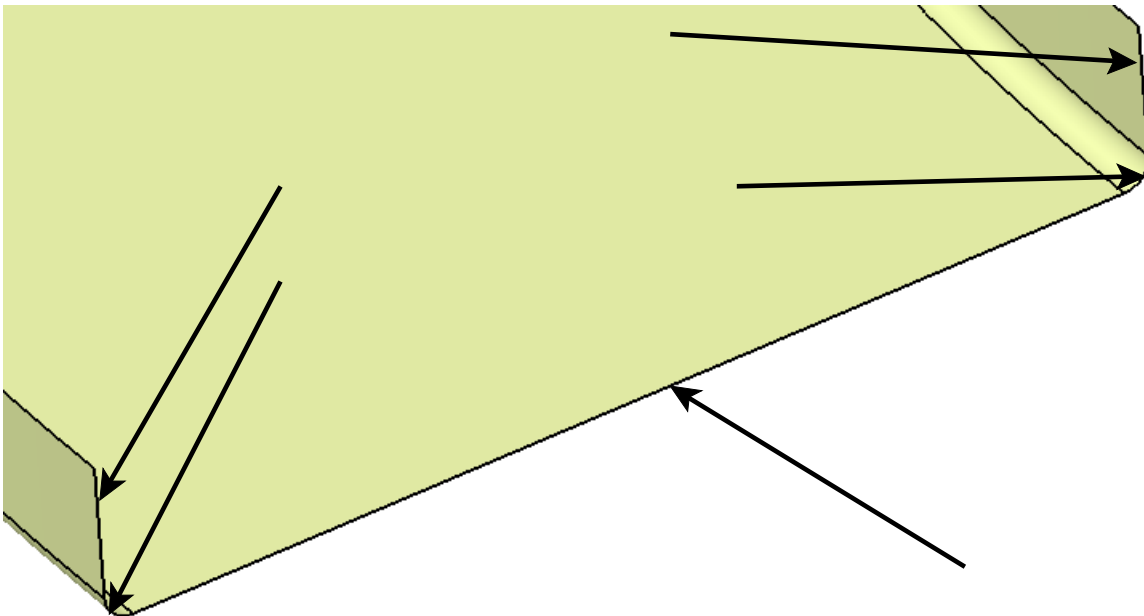
Select the **Spot Welding Connection Property icon**.  The *Spot Welding Connection Property* window appears.



Select the **Point Analysis Connection** that you just created to define the *Supports* for the property. Set the *Type* to **Rigid** and select **OK**. This creates a connection at each point in the *Fastener Locations* geometrical set that will simulate a spot weld. In this case, the actual connection would be some type of fastener, but a spot weld should approximate the connection just fine.

Select the **Clamp** icon.  The *Clamp* window appears.

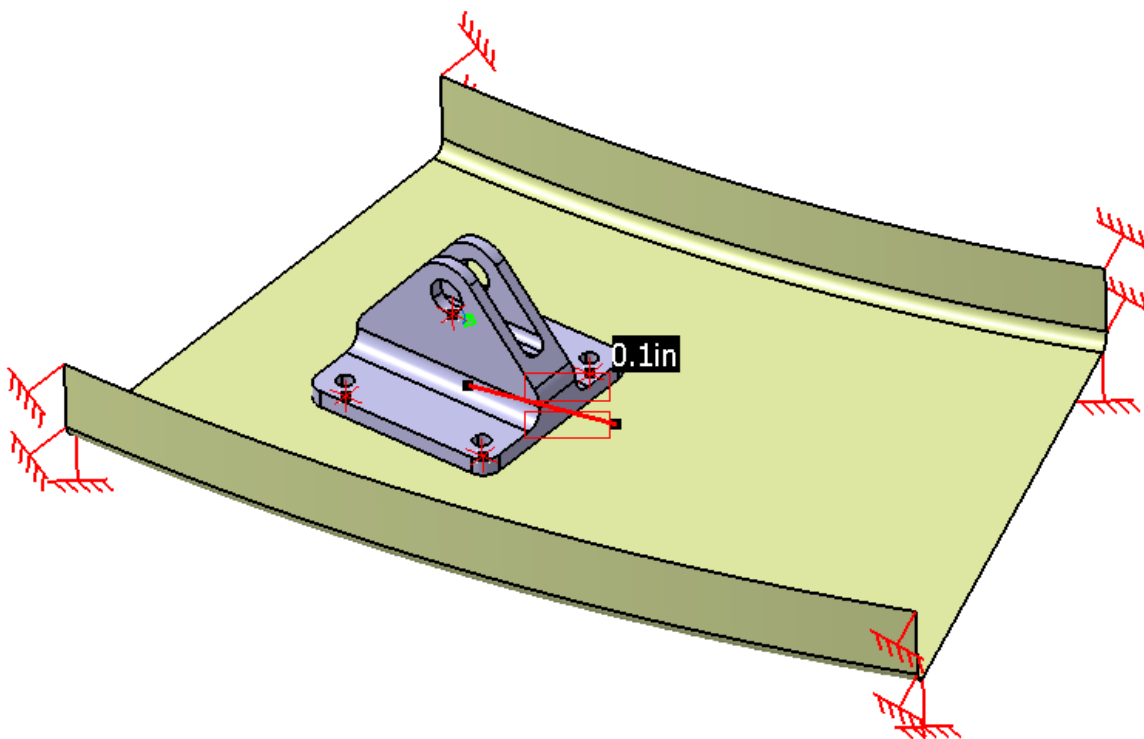
Select the edges shown below.



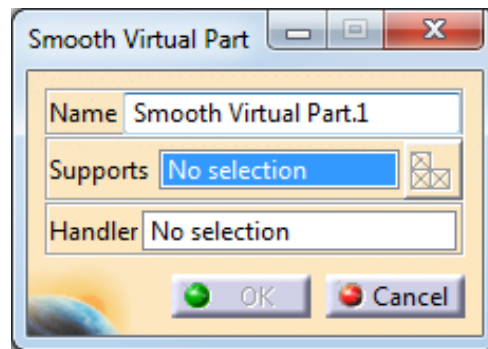
Select **OK**. The restraint is defined.

Define a second clamp restraint on the other side, selecting the corresponding 5 edges.

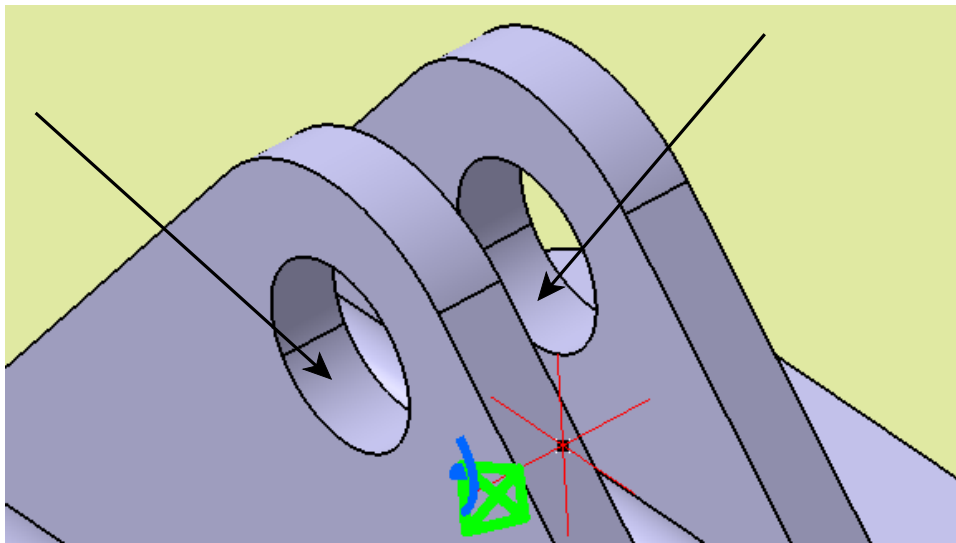
The model should appear as shown. The clamped edges will be unable to move during the analysis.



Select the **Smooth Virtual Part** icon.  The *Smooth Virtual Part* window appears.



Select the two faces shown below to define the *Supports* for the virtual part.



Select **OK**. The virtual part is created.

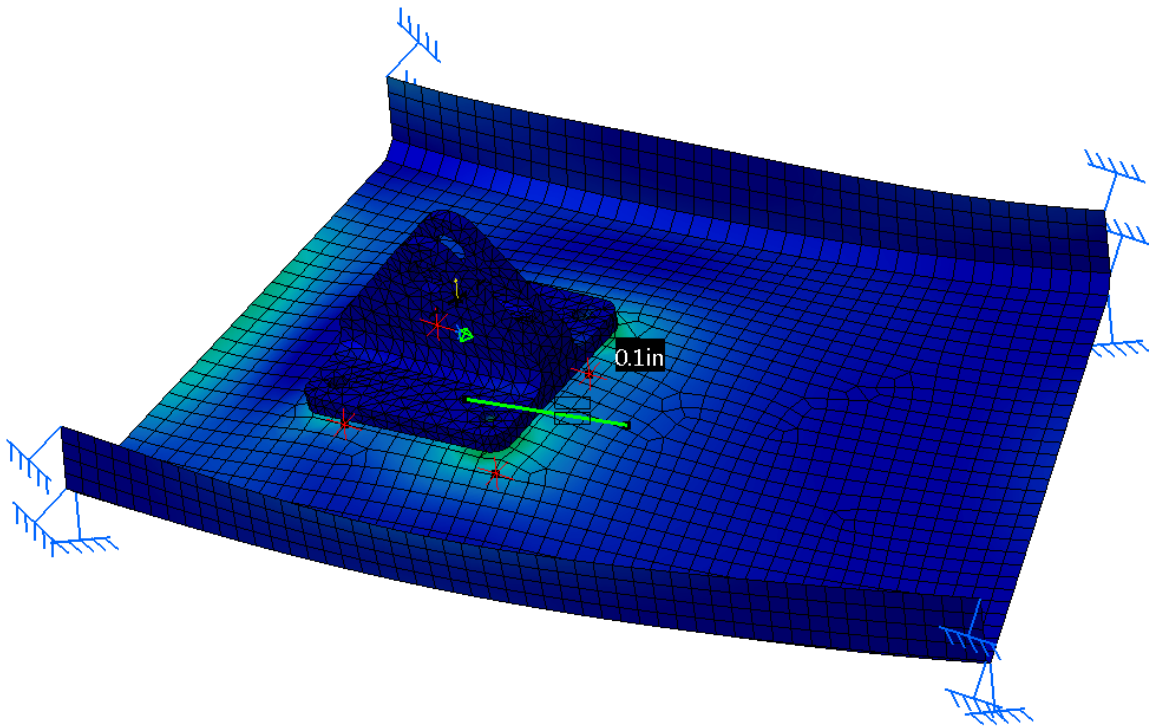
Select the **Distributed Force** icon.  The *Distributed Force* window appears.

Select the virtual part to define the *Supports* for the load.

Set the distributed force to be 150lbf in the positive X direction and select **OK**. This will define the load on the virtual part which would represent a pin or bolt through the two holes.

Compute the analysis. 

Select the **Von Mises Stress** icon.  The model should appear as shown.



By utilizing both solid and surface meshing, the analysis can be optimized so that you get accurate results while minimizing the analysis runtime.

Save and close the document.