TABLE OF CONTENTS

 ©Wichita State University

 ©Wichita State University

Introduction

CATIA Version 5 Generative Structural Analysis

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

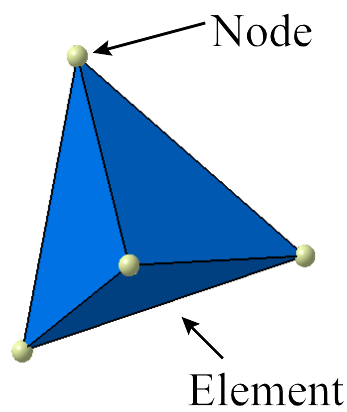
- Applying a mesh
- Defining restraints
- Defining loads
- Defining virtual parts
- Applying an isotropic material
- Defining groups
- Applying each analysis type
- Managing results
- Refining results

© Wichita State University

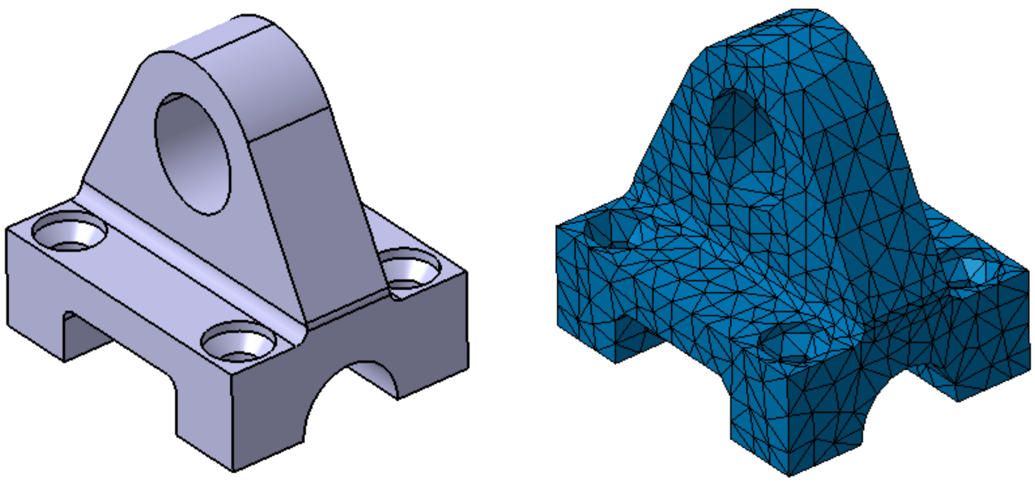
Generative Structural Analysis

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.



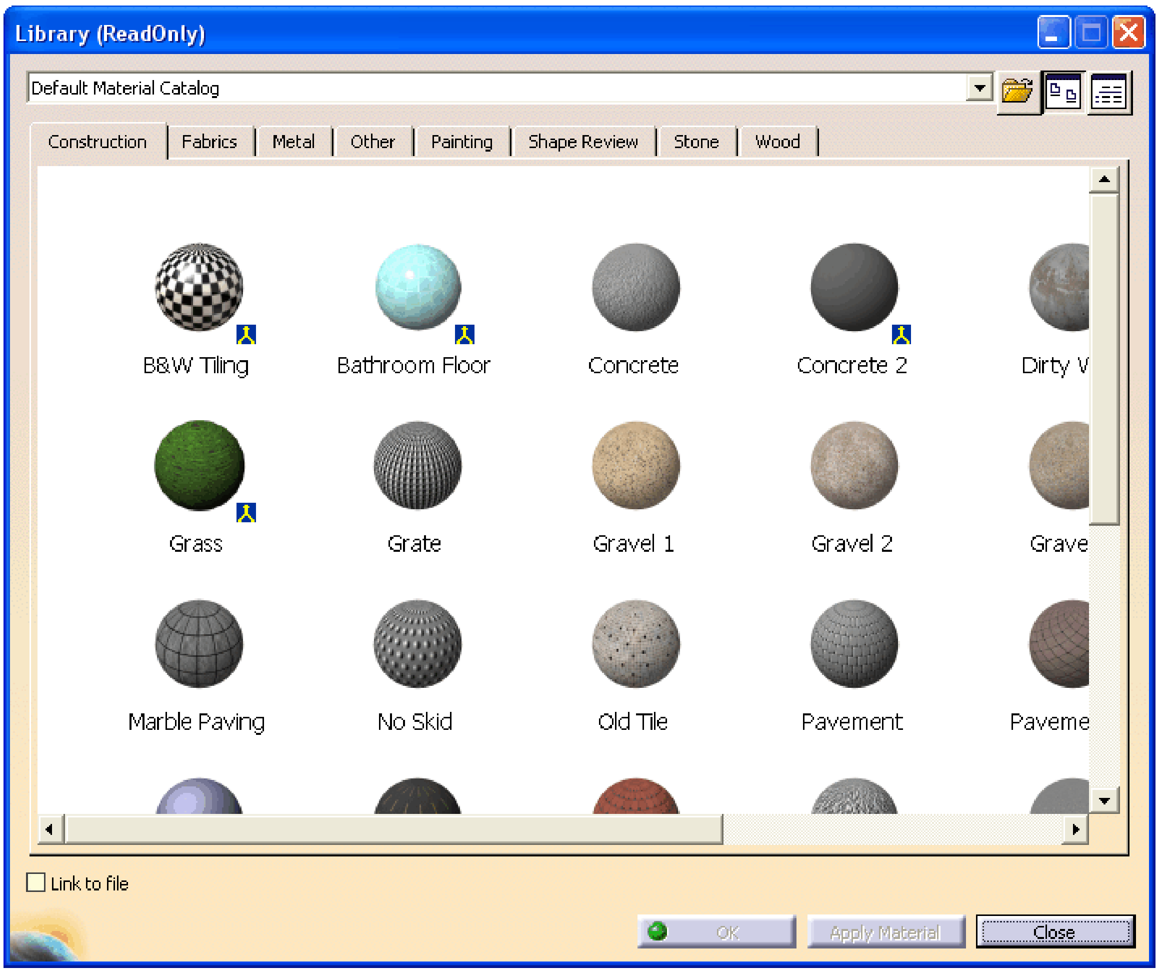
It is important to understand that to fully utilize the tools in this course you should be familiar with the fundamentals of the Finite Element Method. It is not the intention of this course to teach you Finite Element Analysis. However, it is not a requirement of this course that you fully understand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



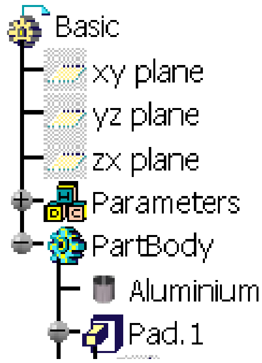
From this library, you may select any type of material that is listed and apply it to your part.

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.

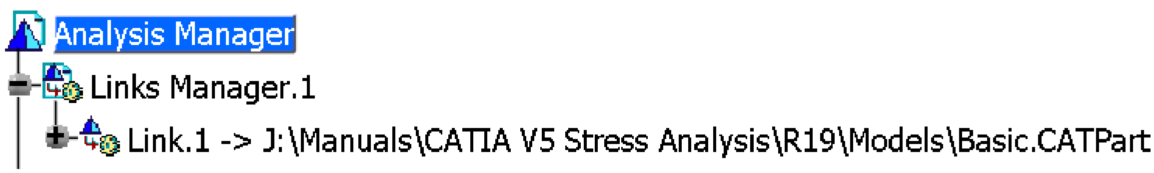
Select *OK* to apply the material. It should appear in the tree as shown

.

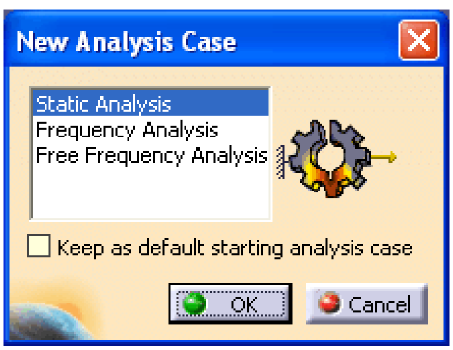


Now that the part has a material applied to it, an analysis may be created.

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.



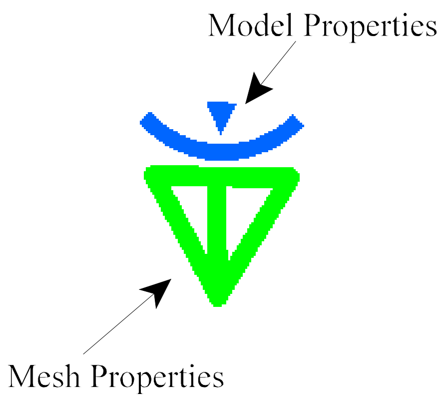
The *New Analysis Case* window appears. You have to define what type of analysis you would like to do.

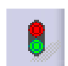


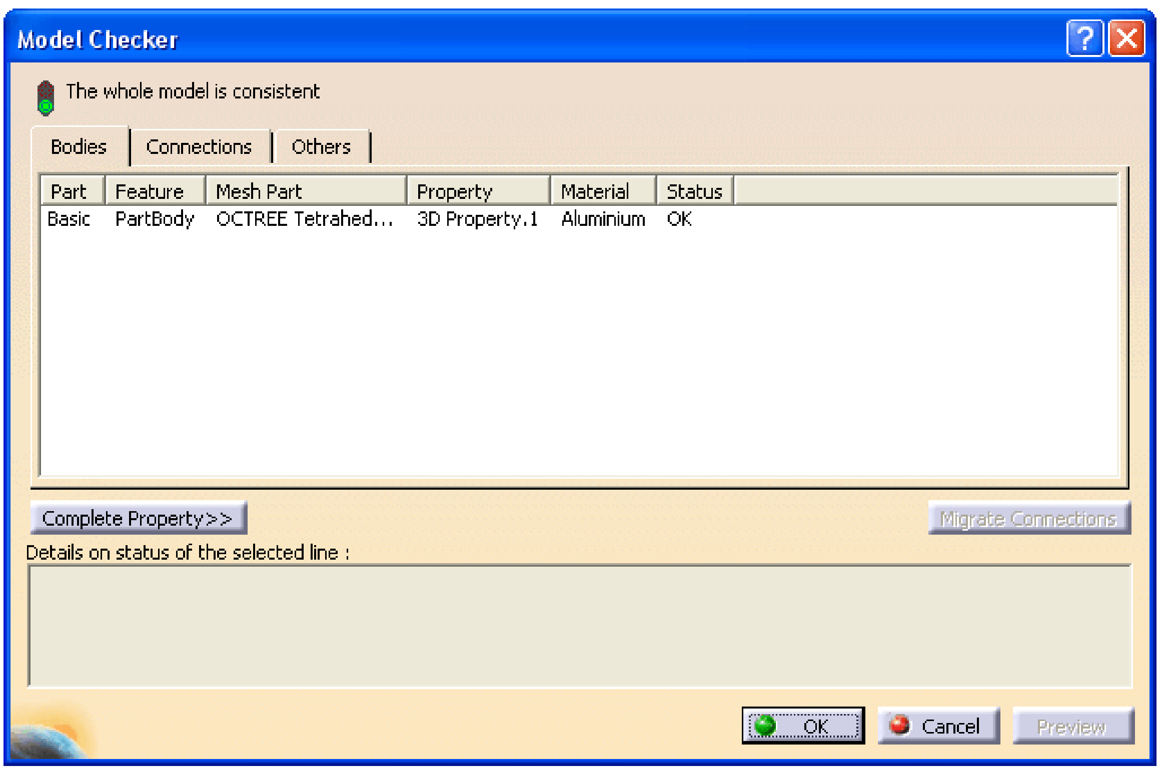
Select the *Static Analysis* case and select *OK*. This creates a *Static Case* analysis document. You will see the *Static Case* branch in the specification tree. You actually have a new document up at this point.

 ©Wichita State University

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.



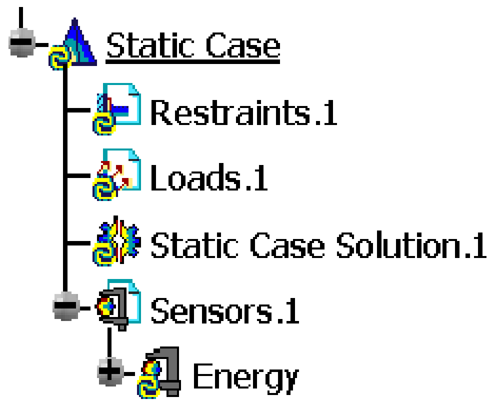
Select the Model Checker icon.  The *Model Checker* window appears.



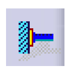
The model checker will show you all of the specifications of the model and determine if the model is okay to use or not. If the model is okay, the *Status* will read *OK*. If there is something wrong, the *Status* will read *KO*. This will play a more important part in the steps later on when there are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

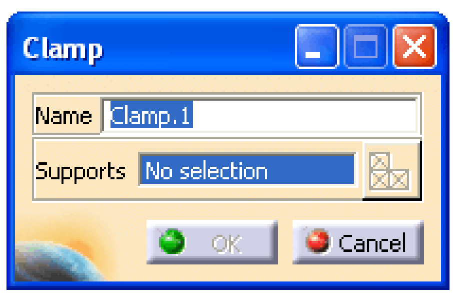
Select *OK*. The model is ready for an analysis.

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*.

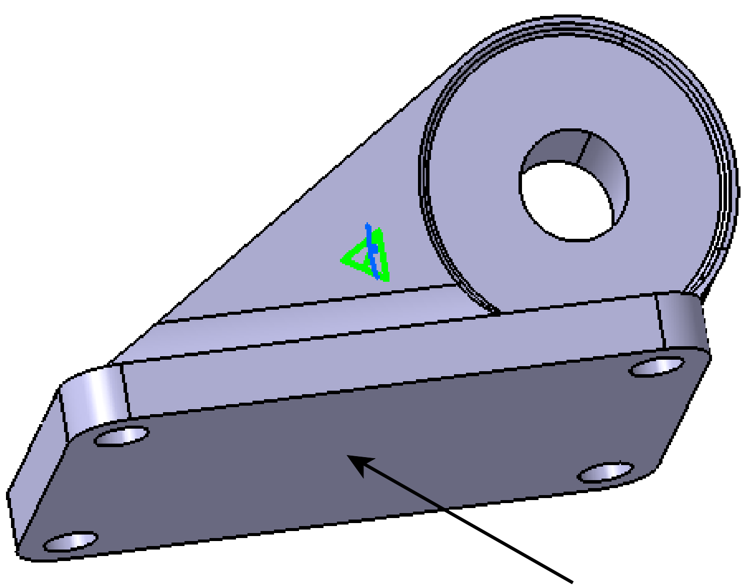


There are no restraints or loads applied to the model, and therefore, there are no solutions to the static case. Defining the restraints and loads on the model are the only remaining steps to get some results from your analysis.

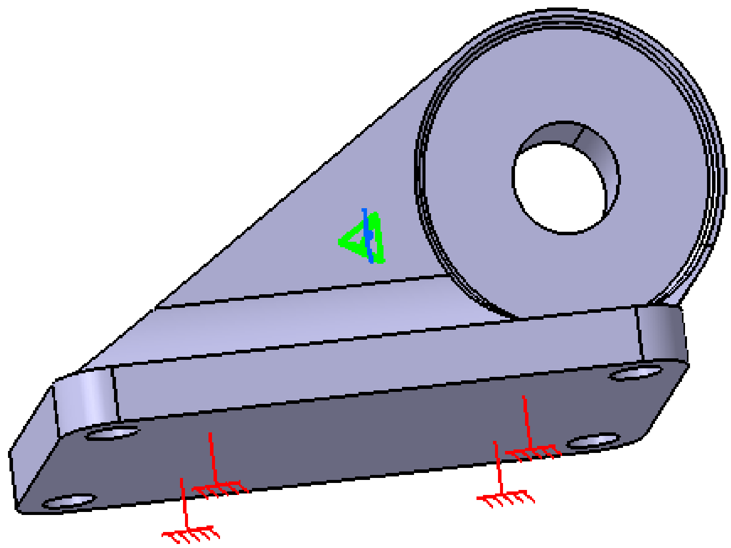
Select the Clamp icon.  The *Clamp* window appears.



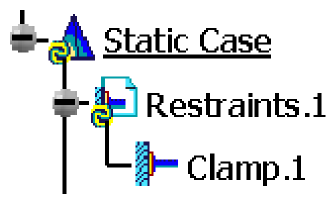
Select the bottom face of the part as shown. This will define the support for the clamp.



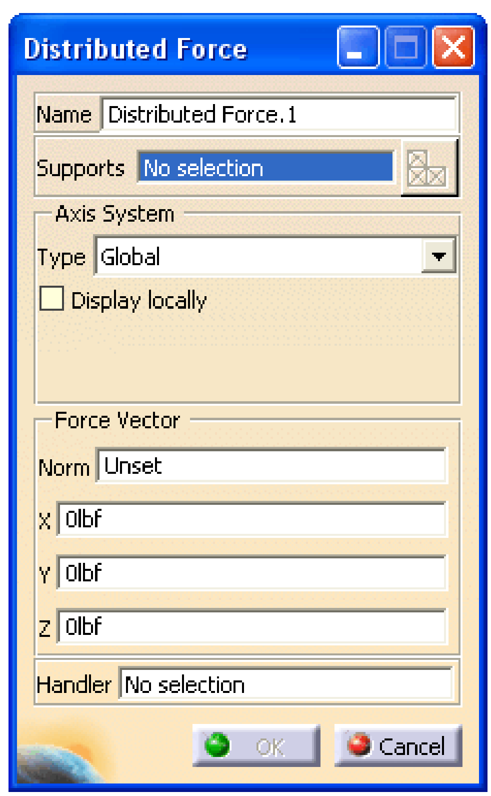
Select *OK*. The clamp should appear on the model as shown.



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

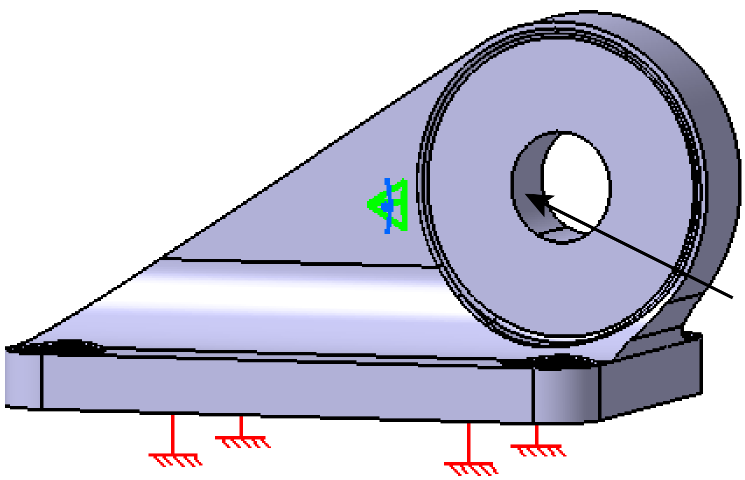


Select the Distributed Force icon.  The *Distributed Force* window appears.



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

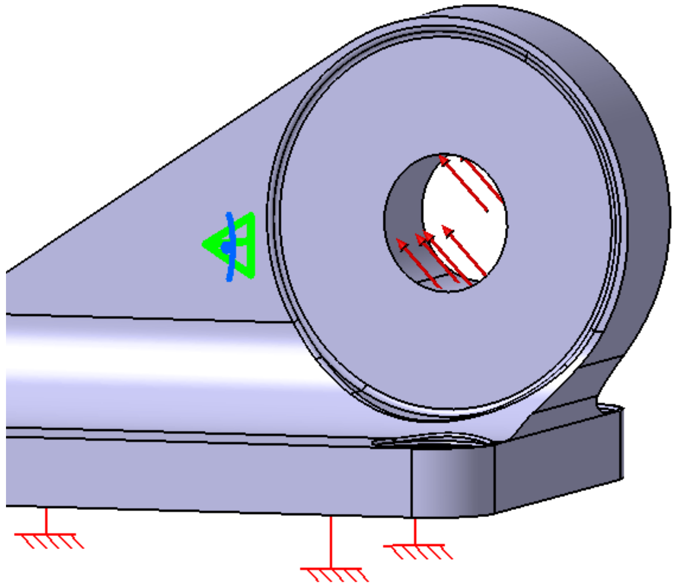
Select the hole as shown for the support.



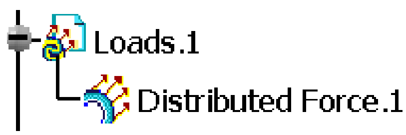
Set the *X*, *Y* and *Z* values to be 50.0, 50.0 and 0.0 respectively. This will yield a *Norm* vector of 70.711 lbf.

©Wichita State University

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

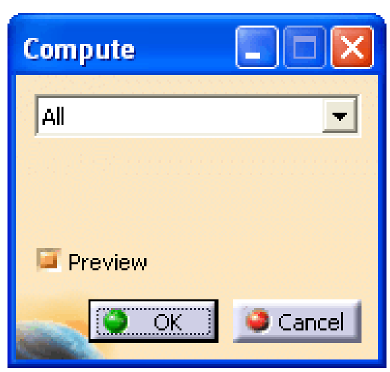


The distributed force appears under the *Loads.1* branch in the specification tree.



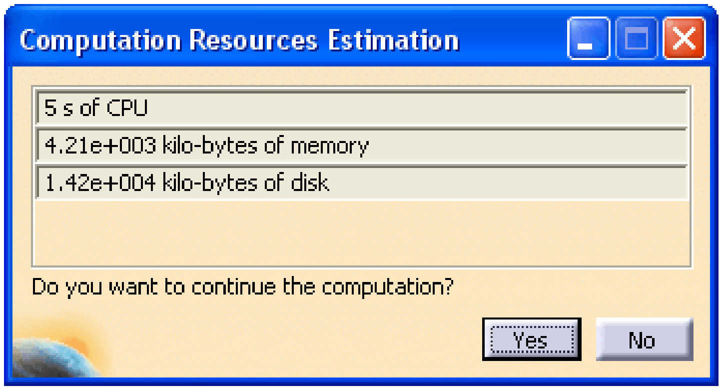
This completes the creation of the necessary elements required for the analysis model. Now the analysis is ready to be computed. It is recommended that the model be saved before you compute the analysis.

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



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

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

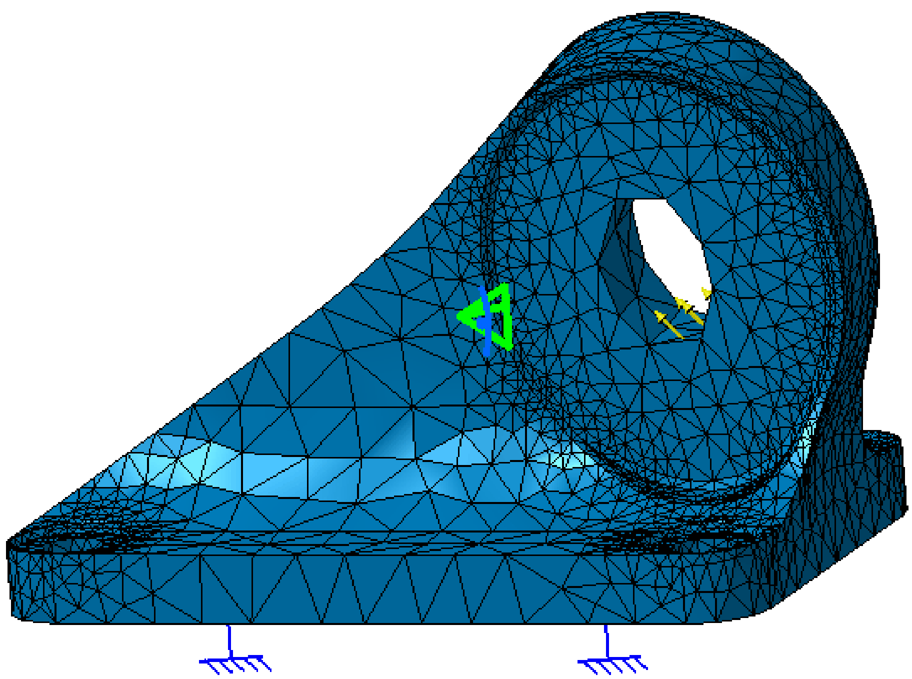


This window gives an approximation on the time the analysis will take as well as how much memory and disk space will be necessary. You may want to check to make sure that the computer has the necessary memory and disk space.

Select *Yes*. The analysis is computed.

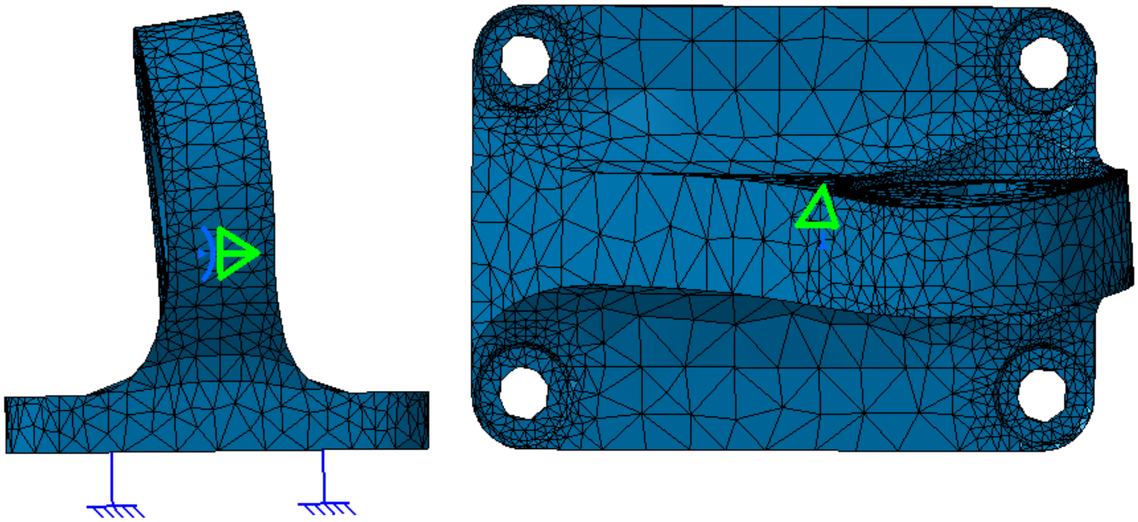
Select the Deformation icon.  This will display the deformation of the model based on the applied restraints and loads.

The model should appear as shown.



 ©Wichita State University

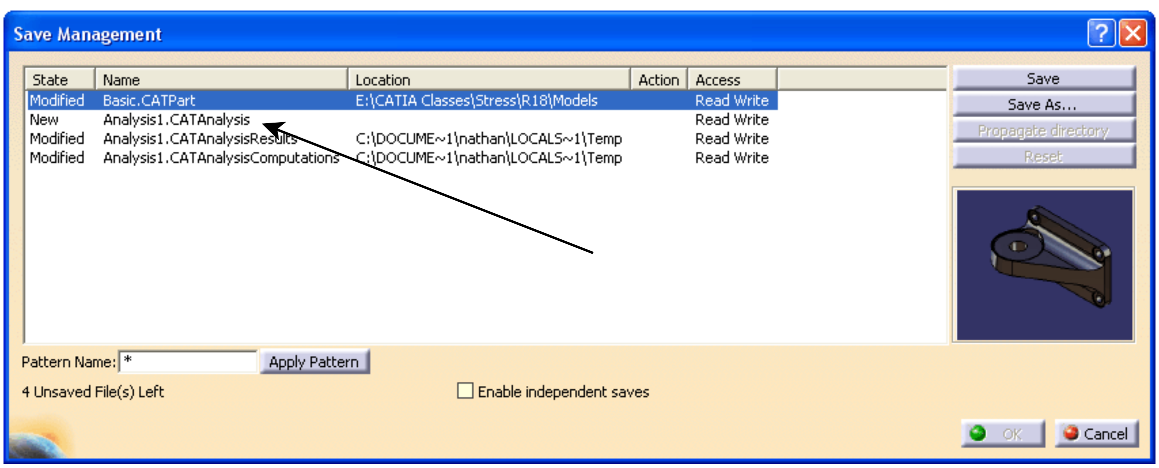
Rotate the model around to view the deformation more clearly.



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.

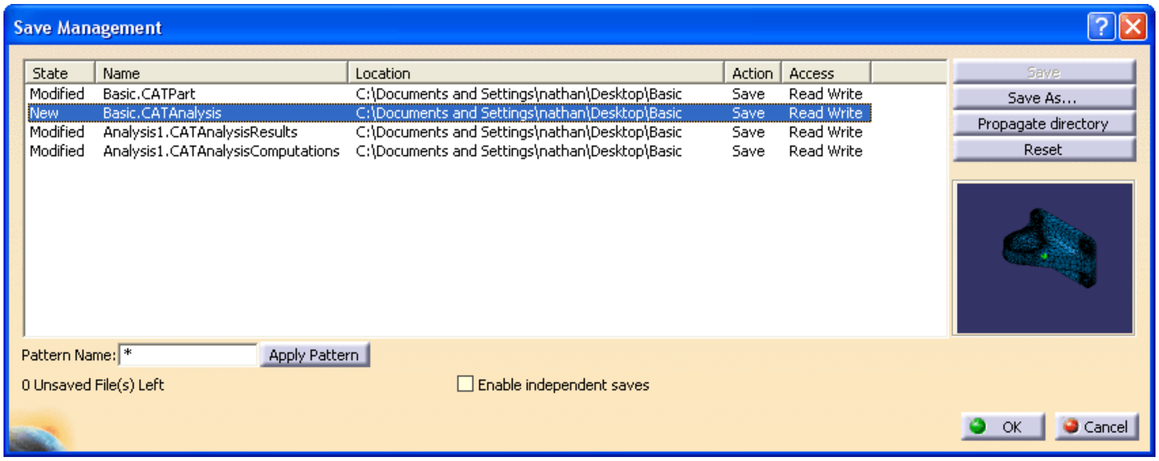


Notice that the part and analysis both need to be saved. There are also two temporary files that need saved as well. The two extra files are the results and computations files.

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 <u>Basic</u>. 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 important to do so that you don't have to worry about breaking links between any of the necessary components.



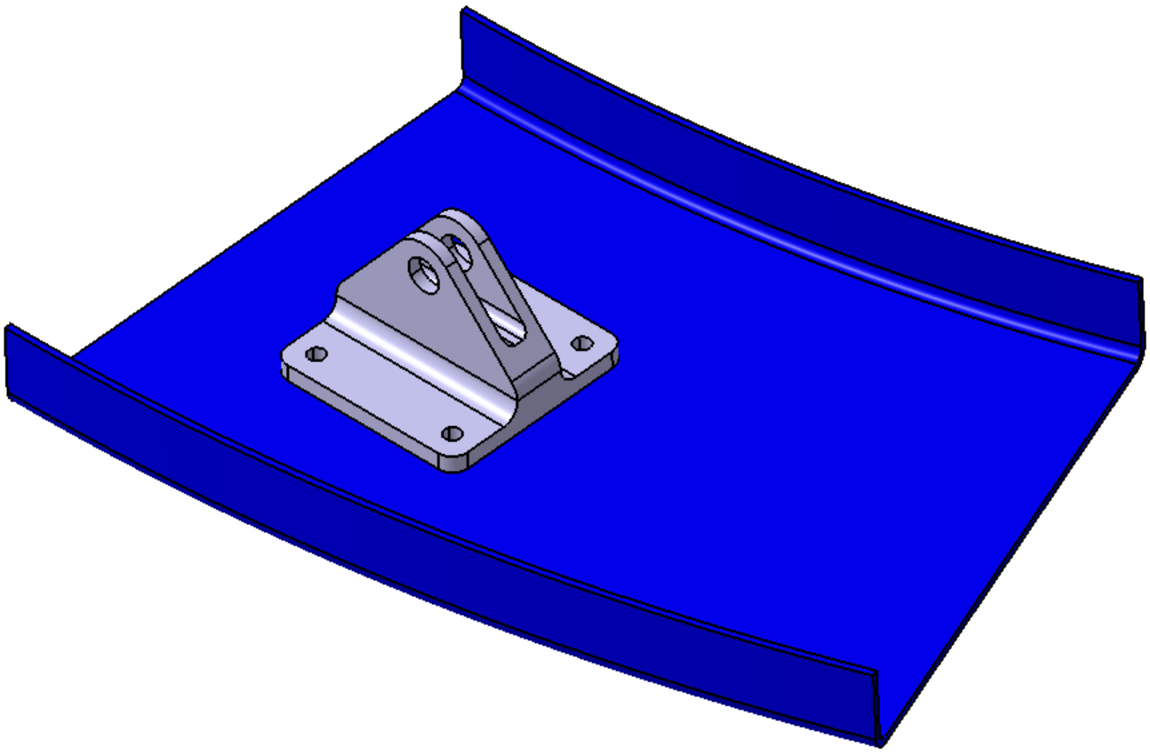
Select *OK* to save the documents.

Close all documents.

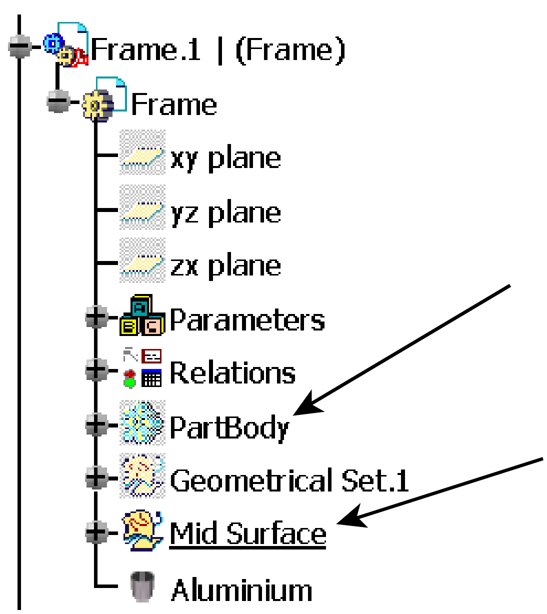
2D-3D Meshing Exercise

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.

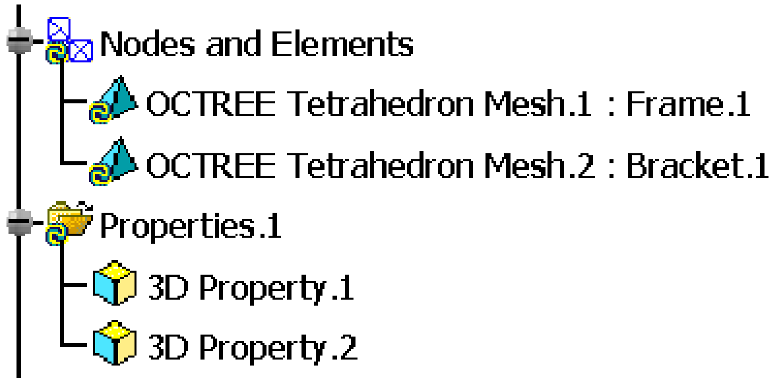


Expand the Frame part and hide the *PartBody* and show the *Mid Surface* geometrical set. The mid surface has already been created for you.



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 and select the surface from the display. 
The *Global Parameters* window appears.

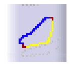
Set the *Mesh size* to be 0.25in. Set the *Element type* to be *Linear* and turn on the *Minimize triangles* option. All other options in the *Mesh* tab should be deactivated.

In the *Geometry* tab, set the *Constraint sag* to be 0.025in and turn on the *Automatic curve capture* option with a *Tolerance* of 0.02in.

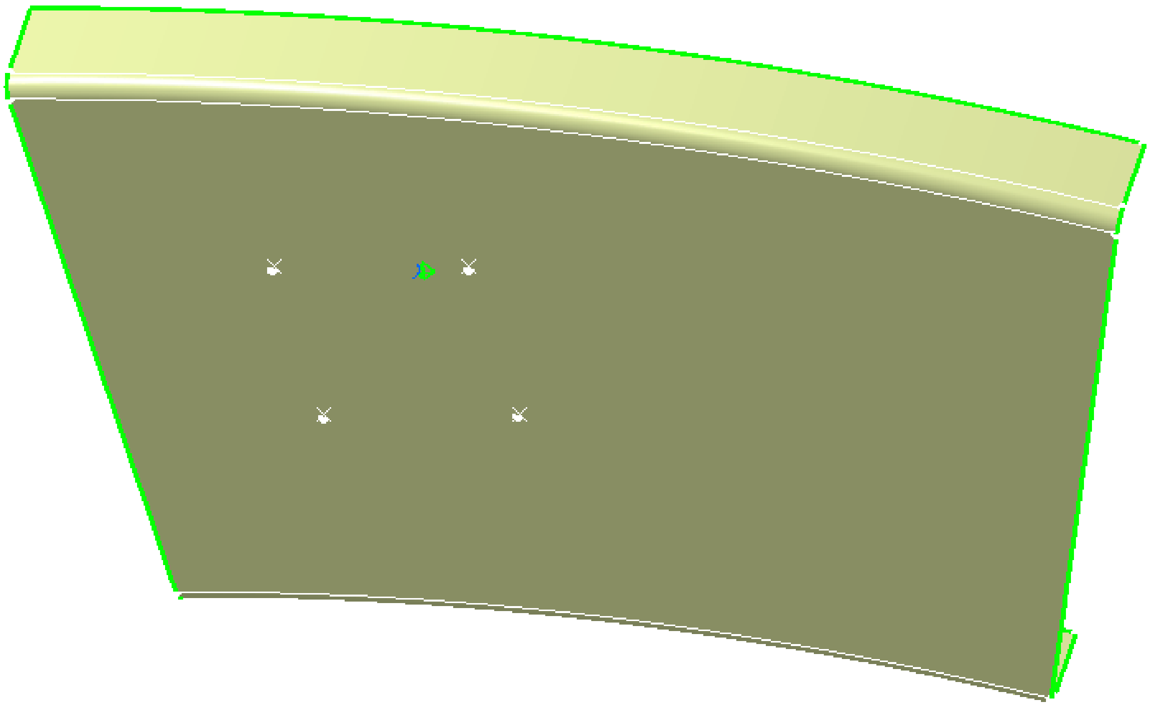
Select *OK*. You are switched to the Surface Meshing workbench.

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.

 ©Wichita State University

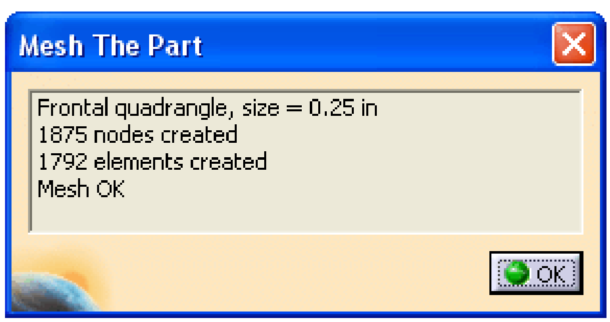
Select the Add/Remove Constraints icon.  The *Add/Remove Constraints* window appears.

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.

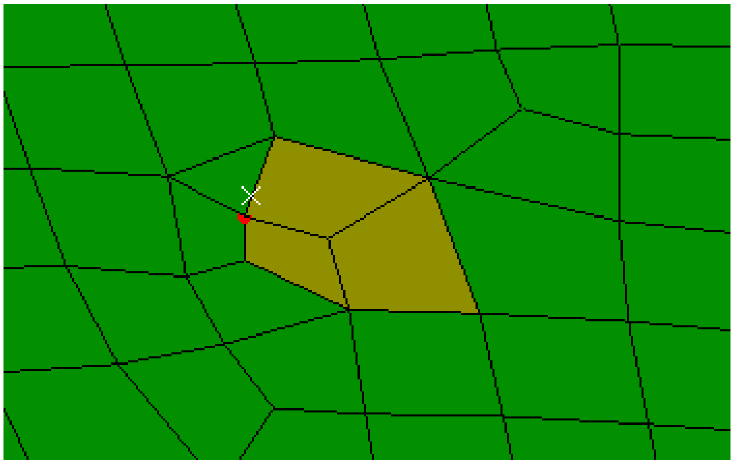


Select *OK*. This will force nodes to be created at the four point locations.

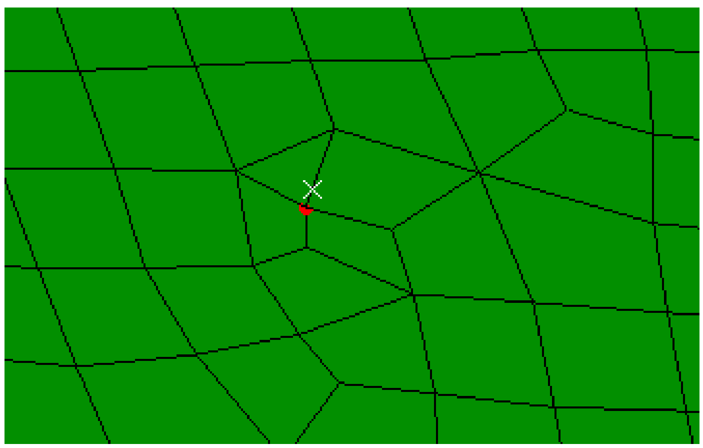
Select the Mesh the Part icon.  The mesh is created and the *Mesh the Part* window appears.



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

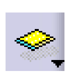


Use the Edit Mesh icon to drag the nodes around until the elements appear green. 
They should appear similar to the picture below.



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

Switch to the Generative Structural Analysis workbench.

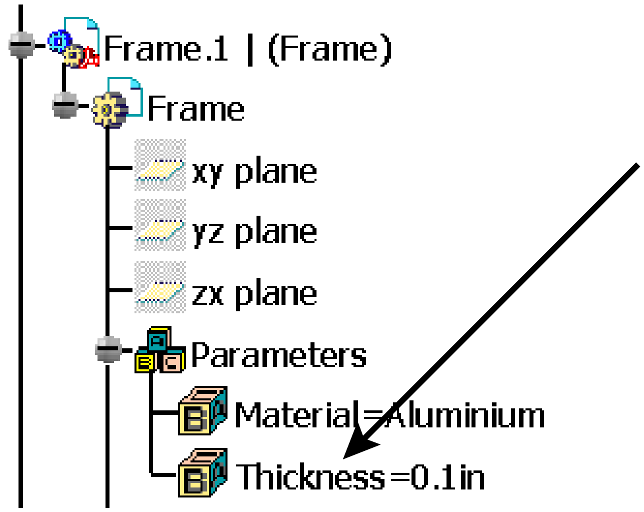
Select the 2D Property icon.  The *2D Property* window appears.

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.

 ©Wichita State University

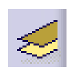
Select the *Thickness* parameter from the Frame model to define the formula.

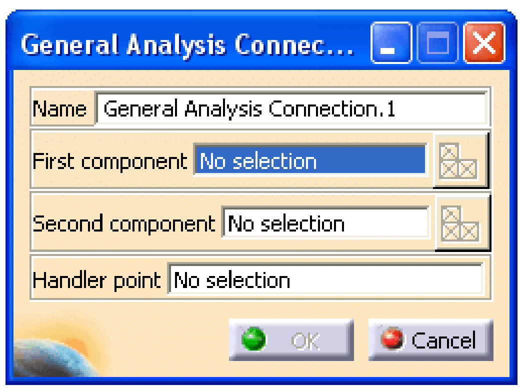


Select *OK* to the *Edit Formula* window.

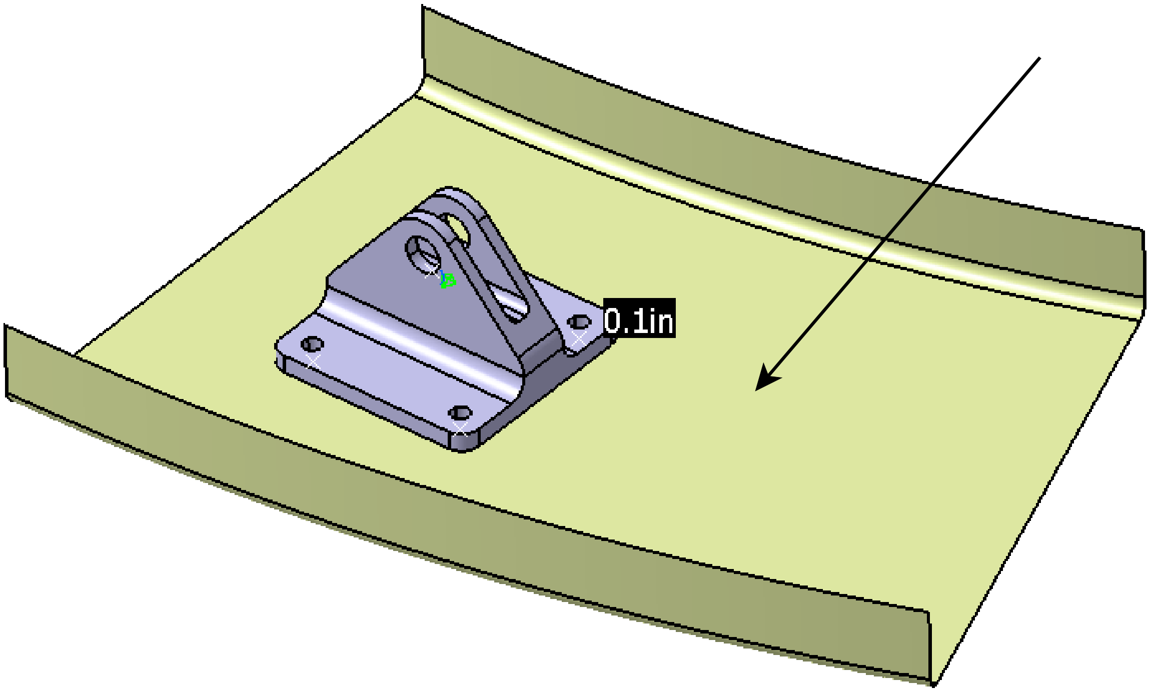
Select *OK* to the *2D Property* window. This will specify that the surface mesh will behave as if it were 0.1 inches thick.

Connections need to be defined between the two parts.

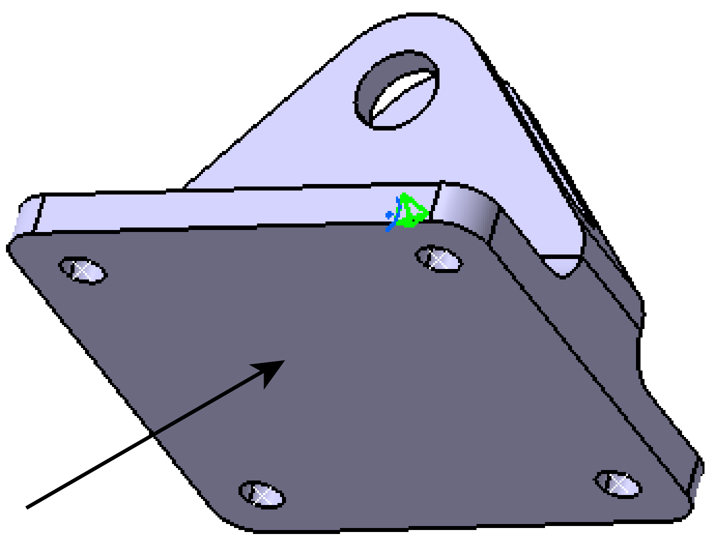
Select the General Analysis Connection icon.  The *General Analysis Connection* window appears.



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



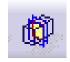
Hide the surface and select the bottom of the bracket as shown to define the *Second component* for the connection.



Select *OK*. The analysis connection is created.

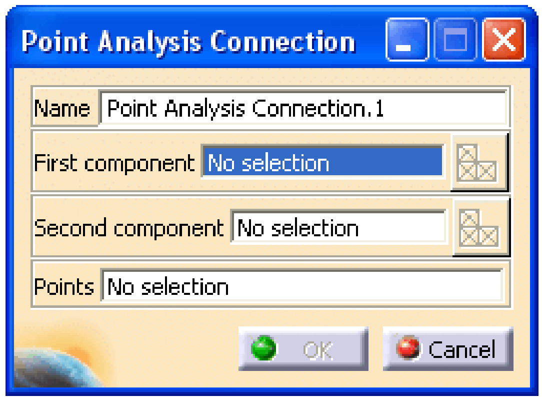
Show the surface.

 ©Wichita State University

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.

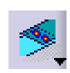
Select the Point Analysis Connection icon.  The *Point Analysis Connection* window appears.



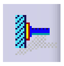
Select the surface to define the *First component* and the Bracket to define the *Second component*.

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

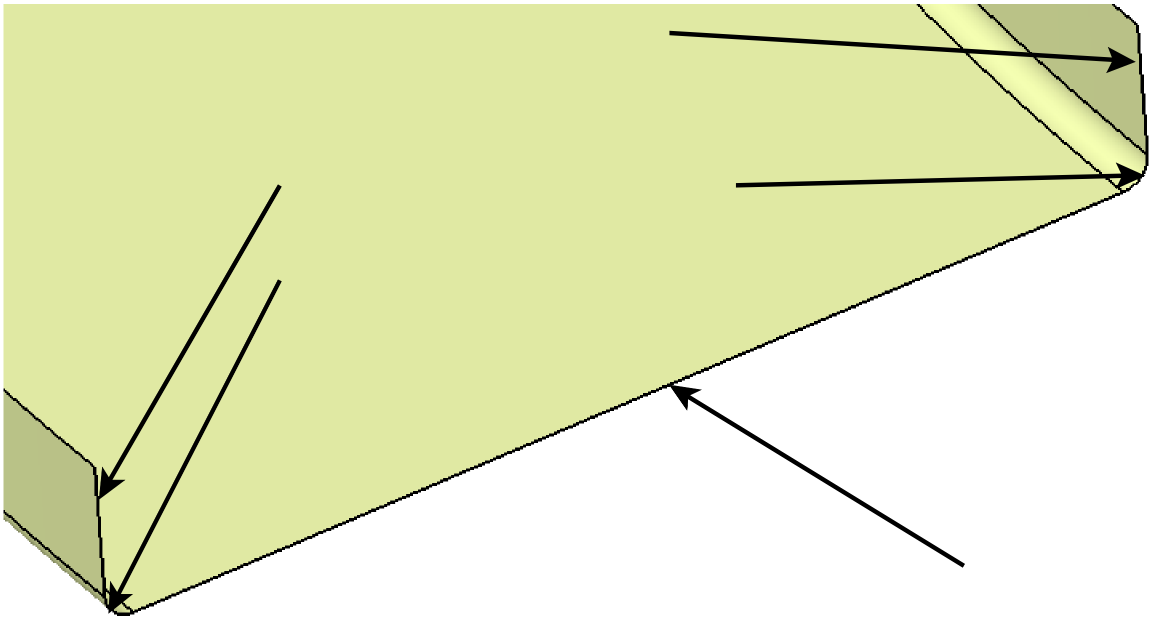
Select *OK*.

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.

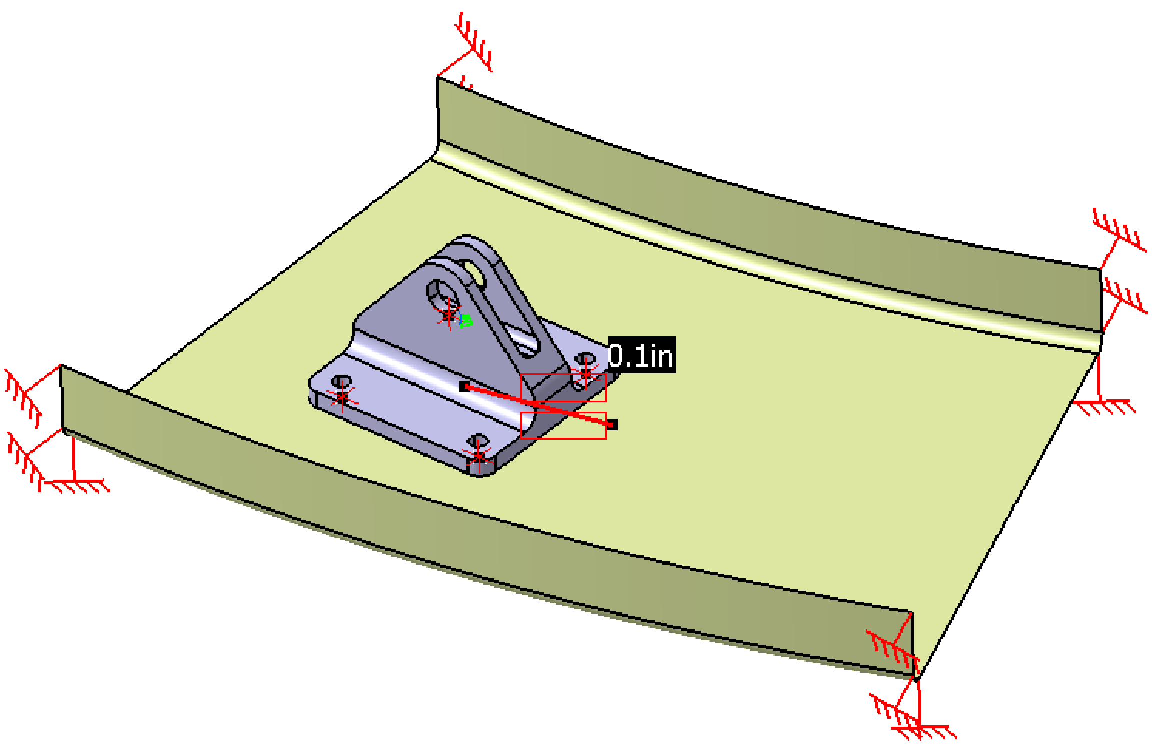
Select the edges shown below.



Select *OK*. The restraint is defined.

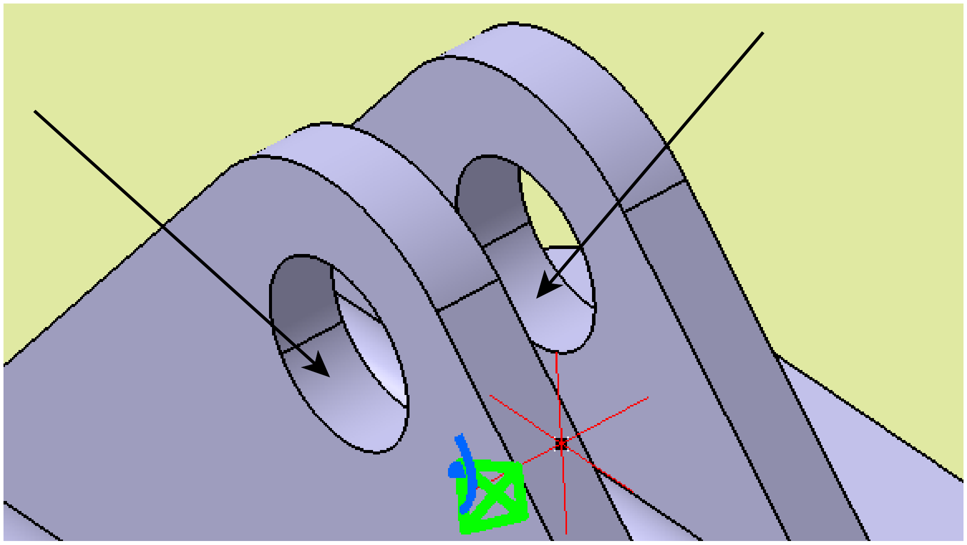
Define a second clamp restraint on the other side, selecting the corresponding 5 edges.

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

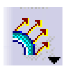


Select the Smooth Virtual Part icon.  The *Smooth Virtual Part* window appears.

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



Select *OK*. The virtual part is created.

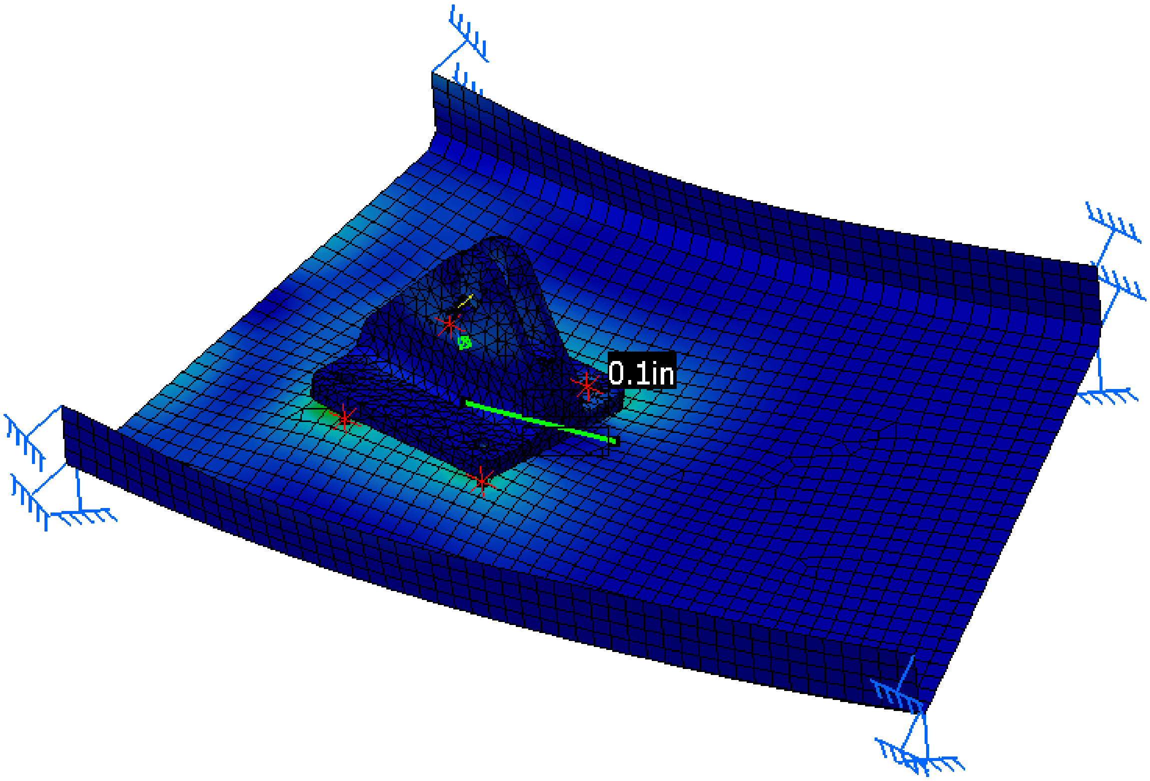
Select the Distributed Force icon.  The *Distributed Force* window appears.

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

Set the distributed force to be 150lbf in the positive Y 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.