

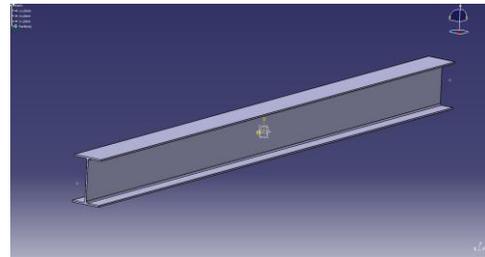
FEA BENDING, TORSION, TENSION, and SHEAR TUTORIAL in CATIA

This tutorial shows the basics of a solid bending, torsional, tension, and shear FEA (Finite Elemental Analysis) model in CATIA. Torsion - [page 7](#). Tension - [page 10](#). Shear - [page 14](#).

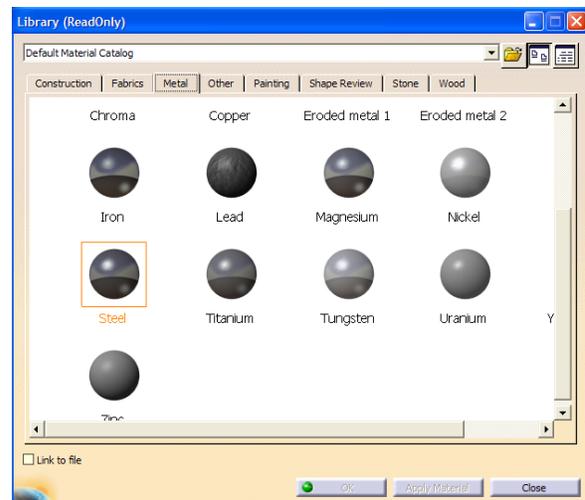
To find the following I-Beam, follow: [Beam2](#). S:\HVL\seniordesign\Catia Course\10. Spring 2011\In-Class Demonstrations\FEA\Parts\Beam2.CATPart

1) Properties

Once the part is created in CATIA, it is ready to be analyzed. This tutorial assumes the user is able to create a part in CATIA using the Part Design workbench . Once the part is created material properties need to be established by clicking on **Apply Material**  and selecting the desired material. Selecting **Steel** under the **Metal** tab in the material library box is commonly used as shown. Select the desired material, click **Apply Material**, and close the box.



If desired, to change Young's modulus and Poisson's ratio, double click on **Steel** (or whichever material you have selected) in the design tree (it may take a moment to come up). When the **Properties** box opens, select the **Analysis** tab. For a Young's modulus of 3×10^7 N-m² and Poisson's ratio of 0.3, you would enter in the appropriate boxes: **3E+7** and **0.3**, respectively.



If desired, the rendering style of the part can be changed from the "gray" appearance to a style that better represents the material chosen. Click on the **View** toolbar, **View mode**, and select **Shading with Material** .



2) Analysis Case

To begin the analysis on the part, select **Start > Analysis & Simulation > Generative Structural Analysis**  workbench. When the **New Analysis Case** box appears, keep the **Static Analysis** selection and press **OK**.

Now specify the “size” and “sag”. “Size” simply refers to the elemental size of the mesh seen on the part; and “sag” refers to the internal distance between the mesh size and the actual geometry of the part. “Sag” is unique to CATIA, and this tutorial does not go into depth.

There are two ways to change “size” and “sag”. The first is to simply double click on their respective icons in the screen as shown.

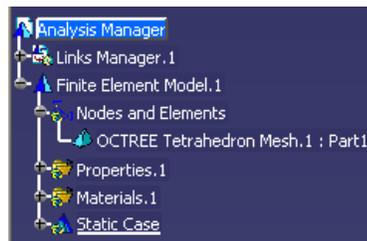
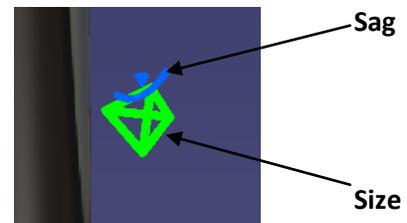
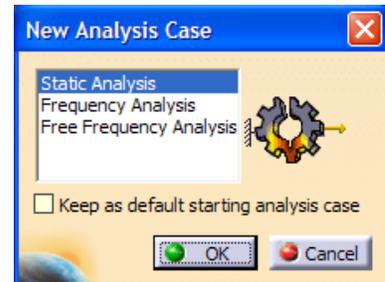
The second method is to double click on the branch **OCTREE Tetrahedron Mesh** in the design tree. Both methods will open up the **OCTREE Tetrahedron Mesh** box. Set **Size** and **Sag** and make sure **Absolute sag** is checked, which keeps the icon in the screen. You do not necessarily want **Size** to be too small.

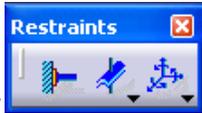
To view the mesh, right click on **Nodes and Elements** in the design tree and click **Mesh Visualization**. A warning may appear about recalculating; click **OK**. The mesh is now displayed on the part. It may be helpful to select

Shading with Edges  from the **View mode** tool bar to better see how the mesh is shaped.

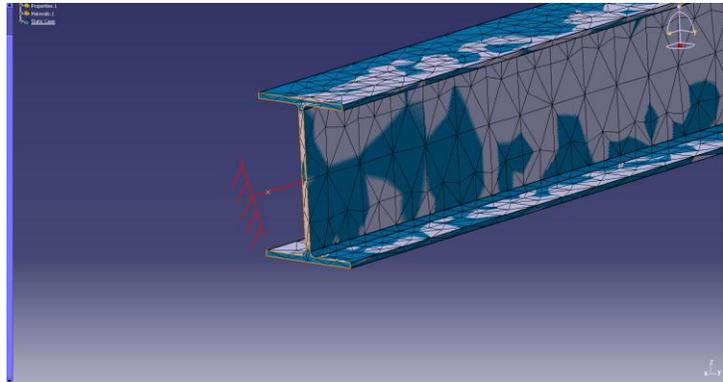
3) Restraints

To unhide the part, right click **Links Manager.1** in the design tree and select **Hide/Show** to *show* both the mesh and the part superimposed on each other. It should appear similar to the below image; this will allow you to go ahead and select the surfaces/faces desired you want restrained. If you wish to hide the mesh only, in the design tree right click **Nodes and Elements** and select **Hide/Show**. Both the mesh and the part are still activated, but this hides the mesh. If desired, you can also **Activate/Deactivate** the mesh by right clicking **Mesh** under **Nodes and Elements** in the design tree. Applying restraints to the part is done through the **Restraints**





toolbar. For this part we are going to restrain one end using the **Clamp** restraint. Be careful to select only the bottom face and not any of the edges.



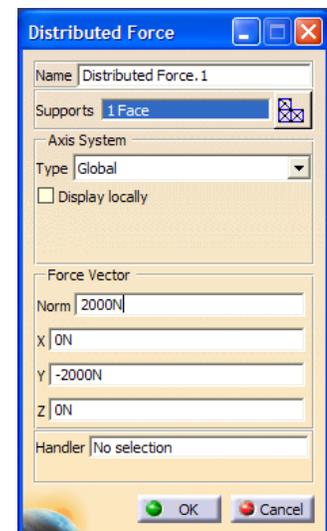
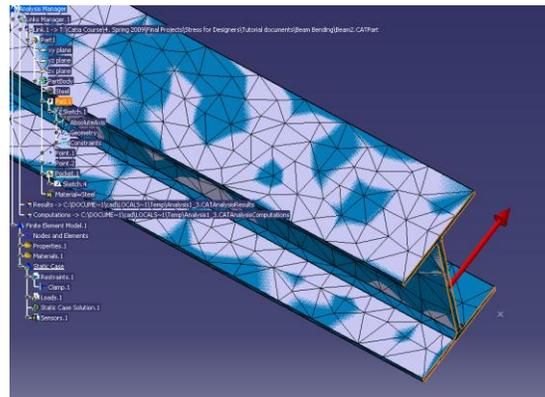
4) Loading

To apply a bending load to the above part, the **Loads**



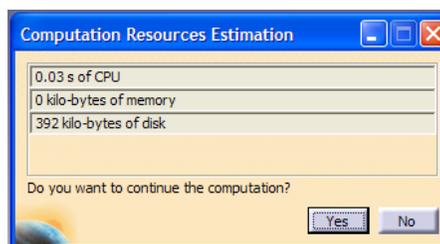
toolbar is utilized. For the purposes of this tutorial, a distributed load, or force, will be applied to the other end of the part.

Select **Distributed Force** and a box of the same name appears. For **Supports** select the entire back face. Again, like in **Restraints**, if you are unable to select the face, right click **Links Manager.1** in the Design tree and select **Hide/Show**. This will allow you to select the face(s)/edge(s) desired. Then, to apply a 2000 N force to the end, enter **2000 N** in the **Norm** box, then Tab down to the **X** direction below it, enter **"0"**, then Tab down into the next text box **Y** and enter **2000 N**, and finally Tab into the **Z**; this allows the loads to take effect. Click **OK**. The negative sign in the **Y**-direction simply points the force in the negative Y-direction, as shown. Of course, the forces and directions are up to the user's decision depending on the particular project.



The FEA analysis now needs to be performed. Click **Compute** which opens the **Compute** box, leave the selection at **All**, and click **OK**. **NOTE:** It may be necessary to use the **Compute** command after opening the analysis project again in order to perform the **Post processing** functions mentioned later in the tutorial.

CATIA then brings up the **Computation Resources Estimation** which shows the resources used to complete the analysis. Click **Yes** to continue the computation.

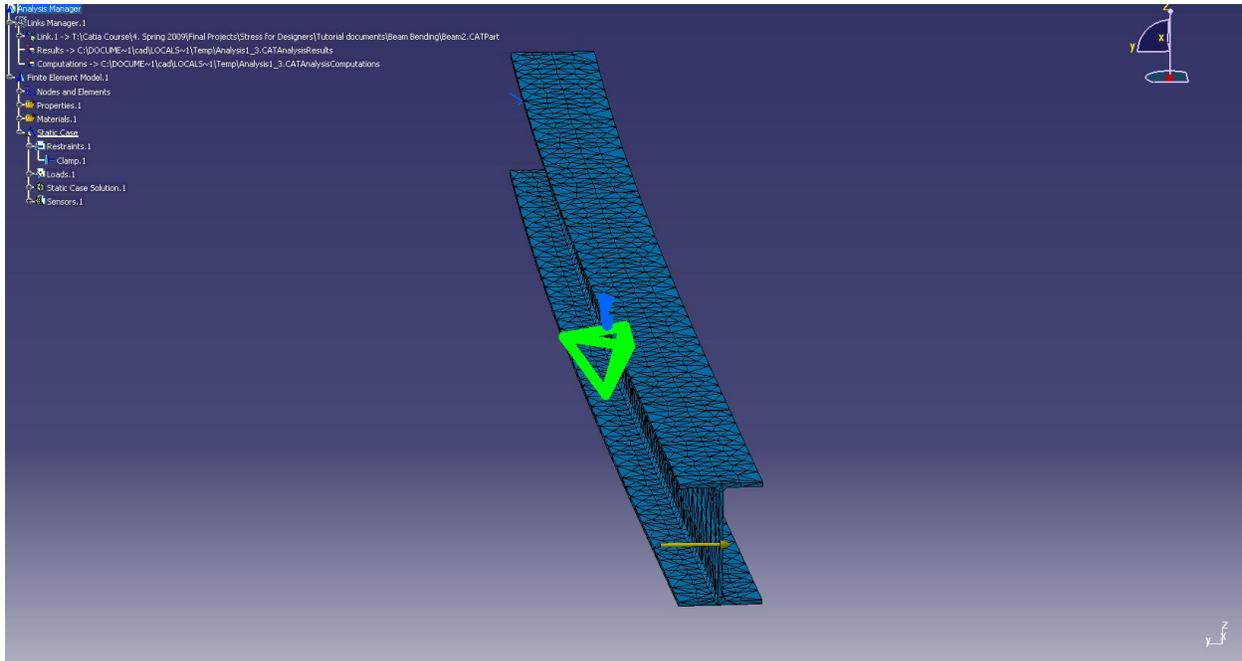


5) Post processing

To view the various post processing features, such as: Deformation, Scaling, von Mises stresses, and Displacement, use the **Image** toolbar



For example, to deform the shape, click **Deformation**  and the following appears:

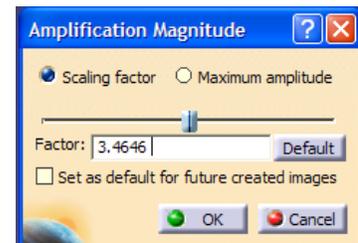


However, this displacement is scaled and not a true representation of the actual deformed shape. The scale can be changed using the **Amplification Magnitude** icon  located in the



Analysis Tools toolbar. Click the  button and the **Amplification Magnitude** box appears.

Either the **Scaling factor** or the **Maximum amplitude** can be adjusted. The default scaling factor for this analysis is 3.4646 and the coinciding maximum amplitude, or deflection length, is 73.421 mm. The bar can be manually adjusted, or a value can be inputted for the desired scaling factor or maximum amplitude.



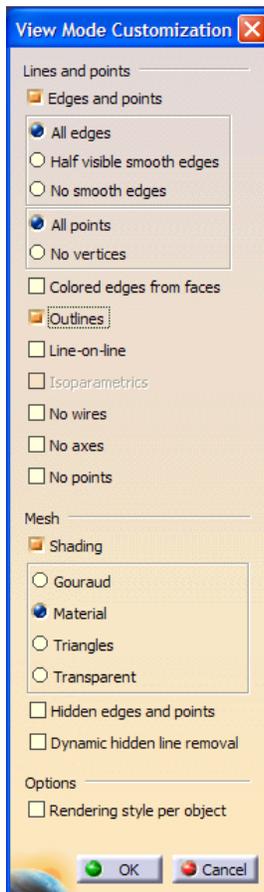
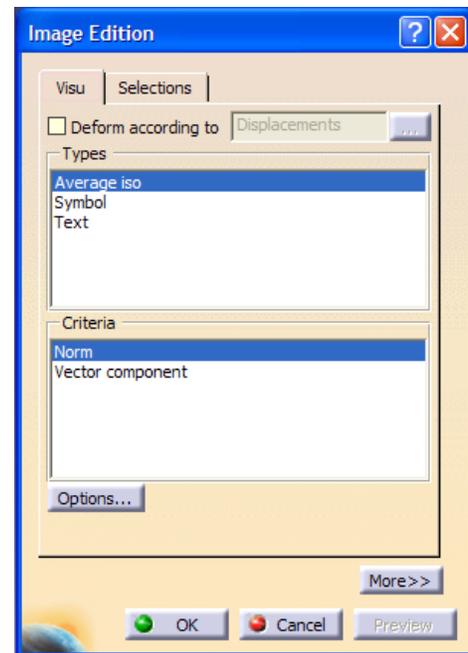
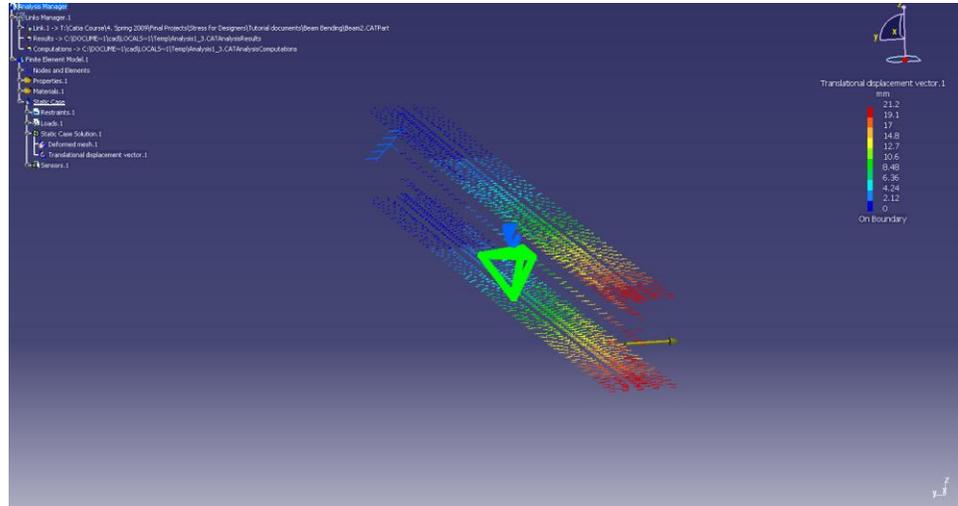
From the **Image** toolbar



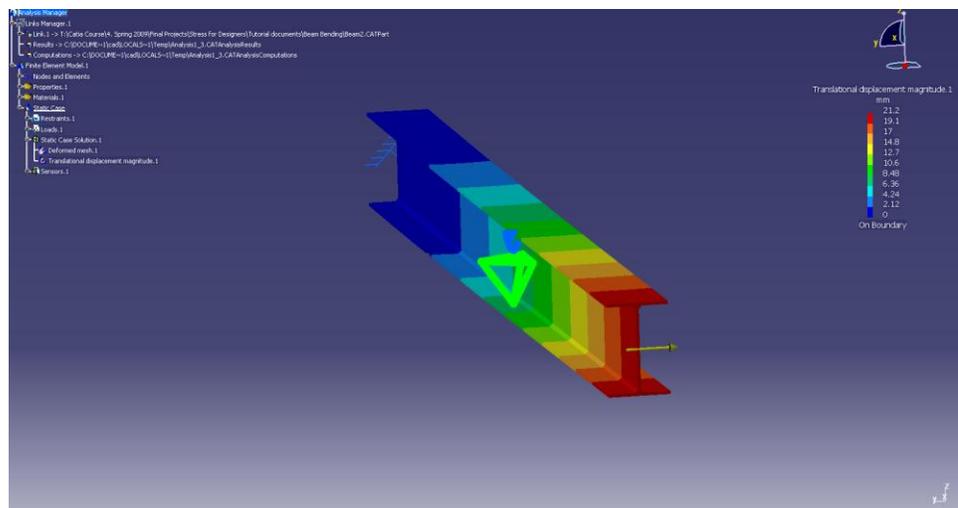
select the **Displacement** command  which shows a displacement vector field.

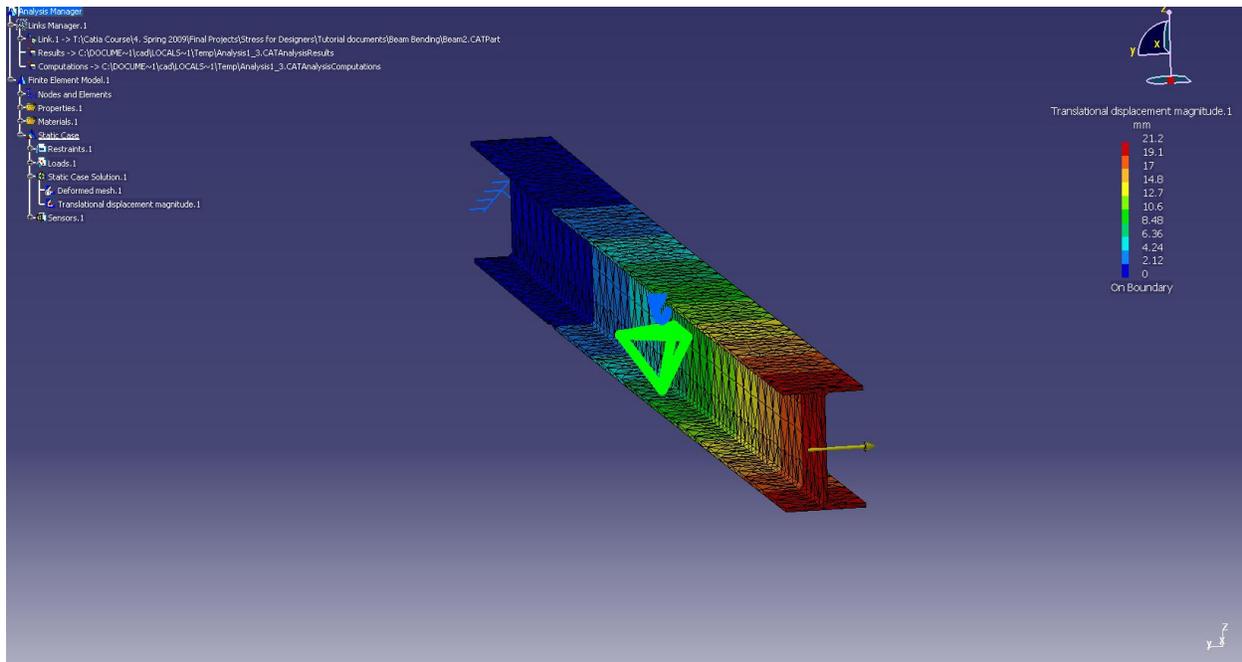
Double click on one of the arrows to bring up the **Image Edition** box, select **Average iso**, and click **OK**.

The image below is still in the **Shading with Material**  view mode. The view **Shading with edges**  shows the mesh.



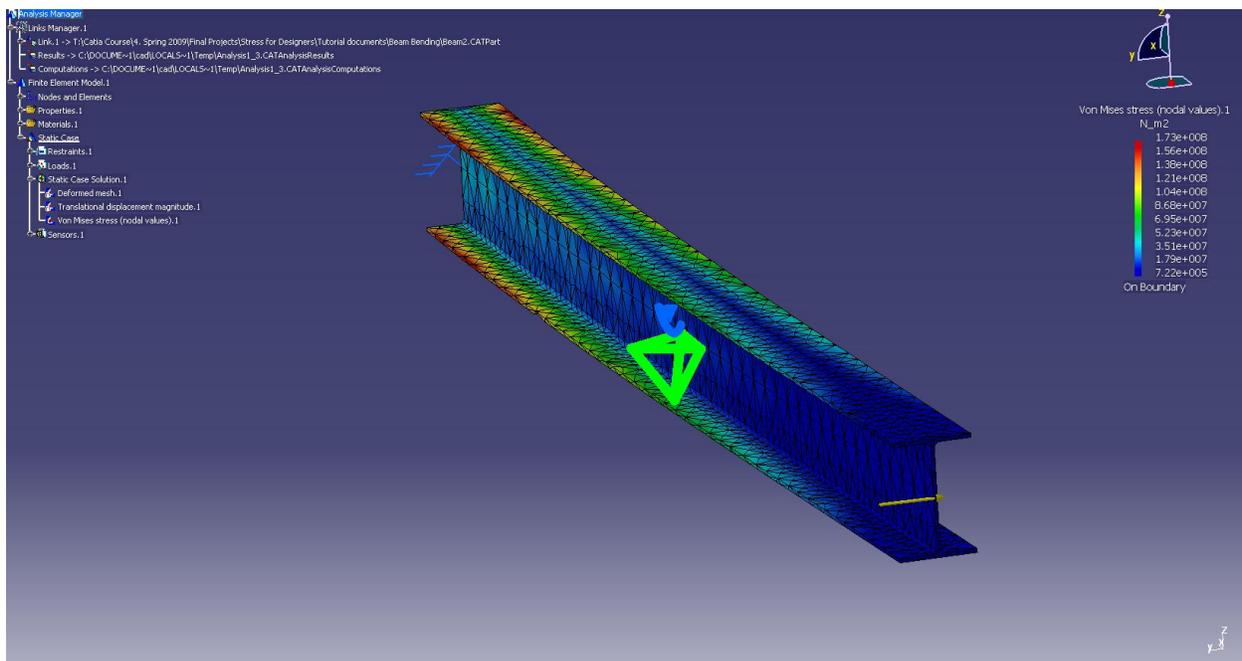
Select the **View Mode Customization** button , which opens a box of the same name, and check everything shown to the left. The result is shown on the next page.





To view the von Mises stress, click the **von Mises Stress**  command that will display the respective stresses shown below.

To view animations of the deflections or stresses, cut planes, maximum/minimums, or the features use the **Analysis Tools**  toolbar.



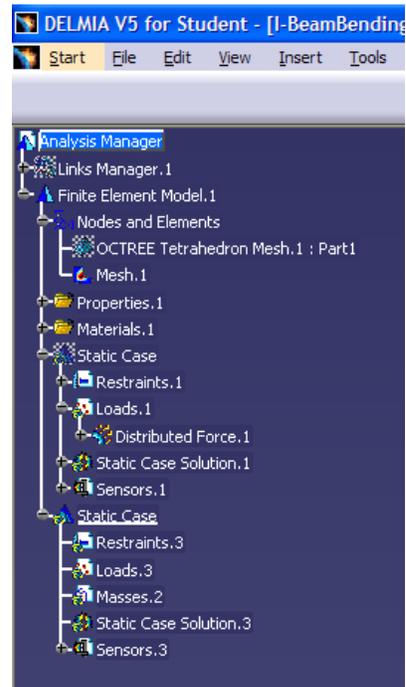
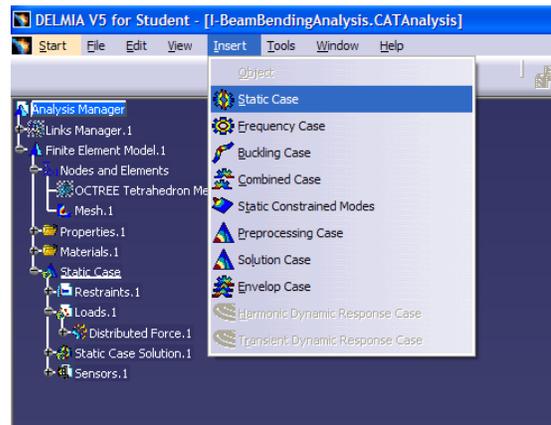
FEA TORSION TUTORIAL in CATIA

The following tutorial shows how to attach a torsion moment onto a part, assuming knowledge from previous tutorials.

If I-Beam is no longer open, follow: [Beam2](#). S:\HVLC\seniordesign\Catia Course\10. Spring 2011\In-Class Demonstrations\FEA\Parts\Beam2.CATPart

1) New Static Case

Start out by adding a new **Static Case** if the part is already in the drawing area. This allows a new load to be added to the part. Select **Static Case** under **Insert** in the **Main** menu. There are other cases, but this tutorial only shows the static case.



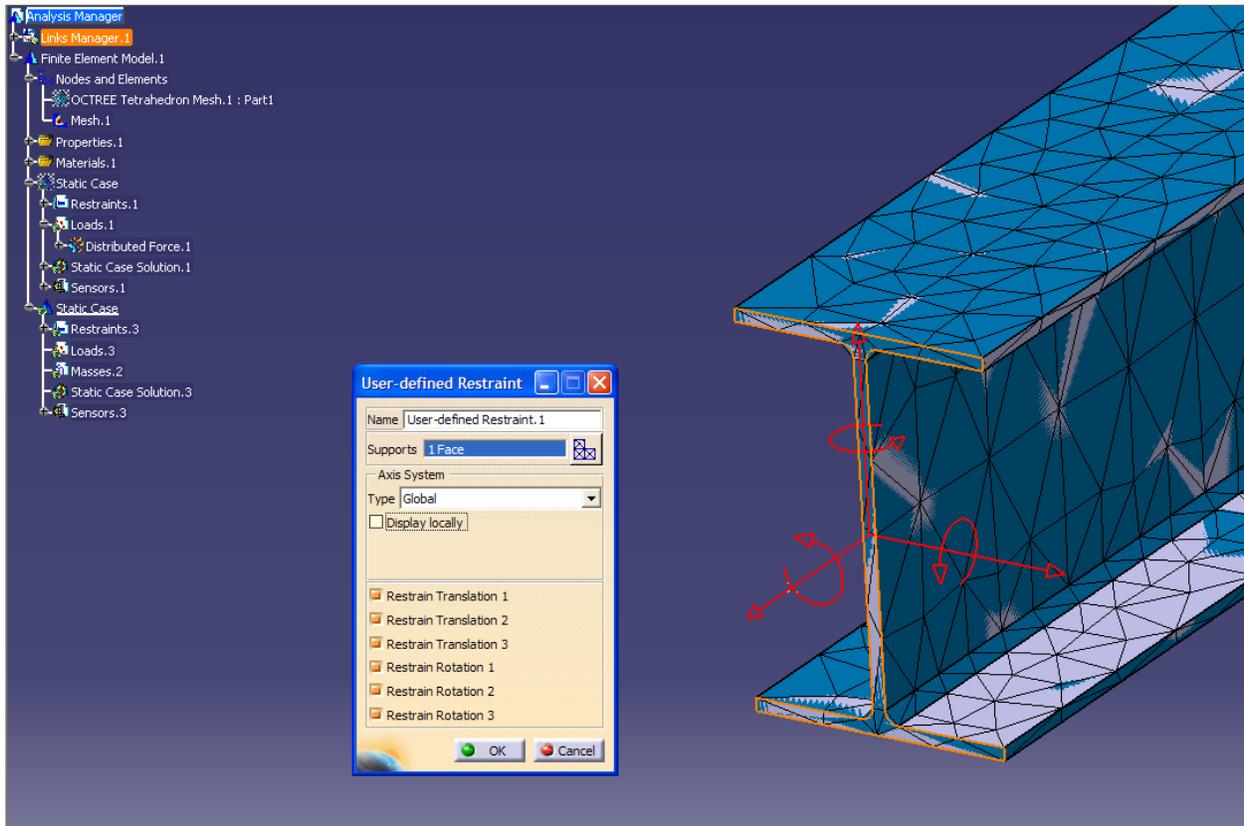
Once the **Static Case** box appears, select **OK** and you will notice that the design tree expands with another Static Case.

2) Restraints



Add **Restraints** as before in the bending tutorial. The **User-defined**

Restraint button  may be used to select where restraints are desired on the part. Once selected, a box of the same name appears that shows Translation and Rotation restraints. The end face of the I-Beam were selected here, however any other face or edge may be selected. Again, make sure the part resembled that shown by right clicking in **Links Manager.1** is selecting **Hide/Show**.



3) Loading



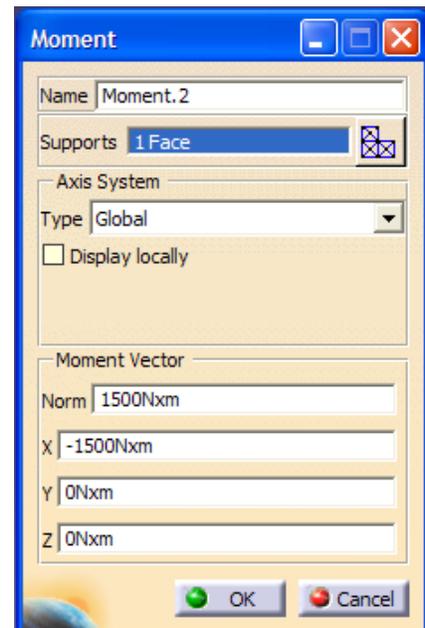
In the **Loads** toolbar

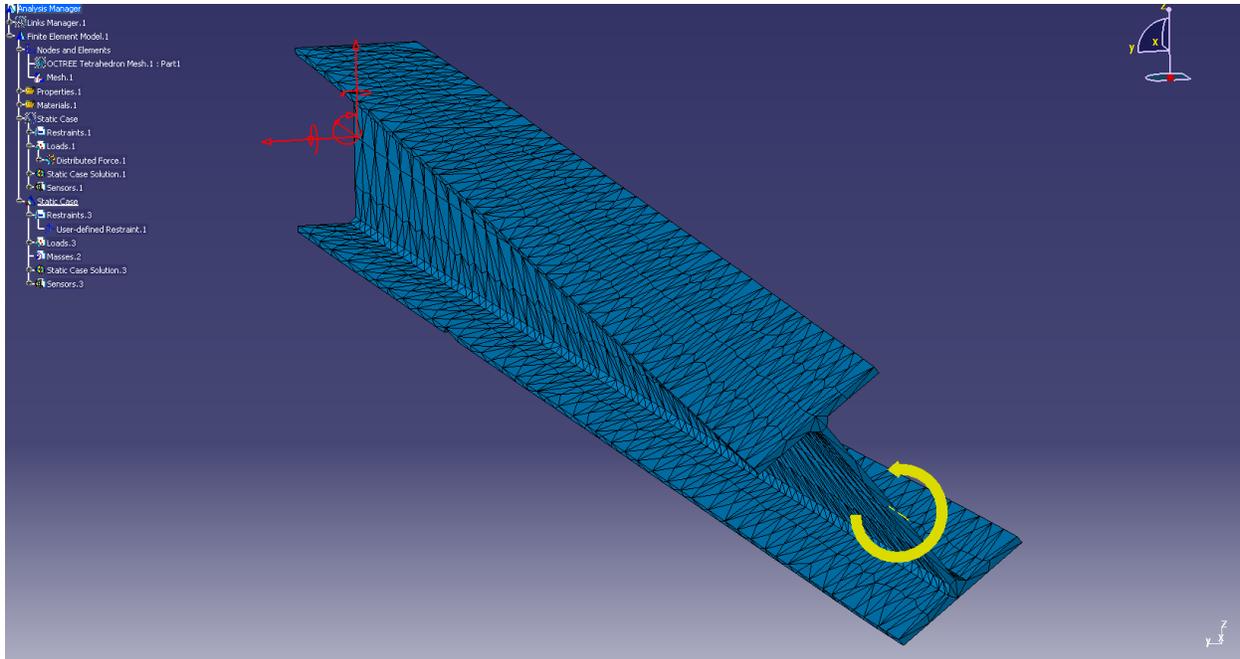
select the **Moment** command . When the box appears, specify a moment. Similar for bending select the face only for **Supports**. The units are force X length. To apply a torsion load to the beam with its current orientation, the moment has to be applied about the X-axis. To input a moment of 1500 N-m, input **1500Nxm** in **Norm**, Tab down to **X**, input **-1500Nxm**, and then Tab into the **Y**-direction. This allows the moment to take effect. Click **OK**.

You may need to check mesh and sag if not continued from bending example.

Click **Compute**  as before. A message may appear concerning restraining the part further; you may click **OK**.

Select **Deformation**  from the **Image**  toolbar to see how the beam twists.





4) Post Processing

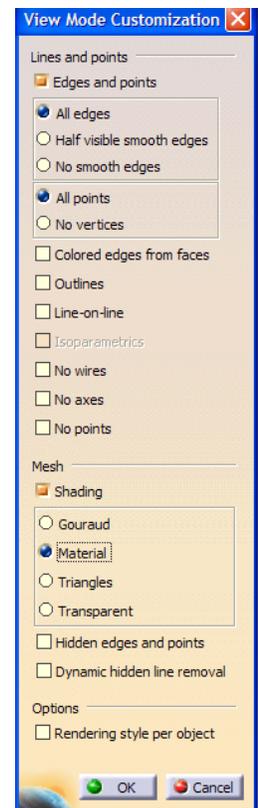
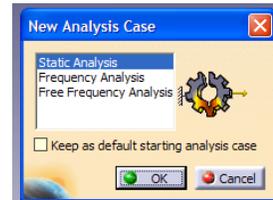
The other analyses from the **Image** toolbar may be applied  as before.

FEA TENSION TUTORIAL in CATIA

The following tutorial shows how to set up tension due to displacement onto a part, assuming knowledge from previous tutorials.

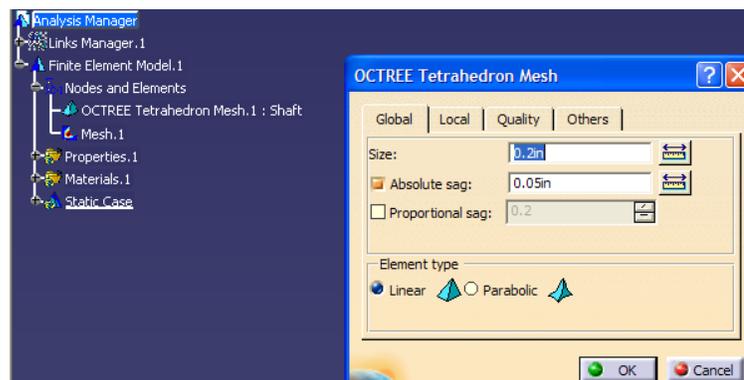
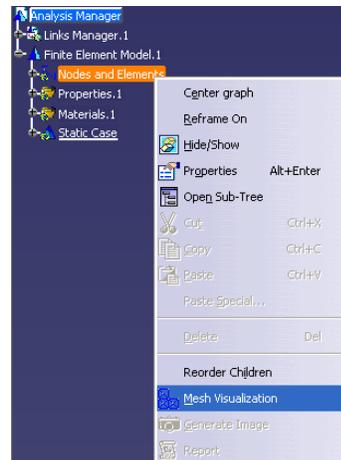
1) Setting Up:

- Open file [Shaft-tensile](#). S:\HVLC\seniordesign\Catia Course\10. Spring 2011\In-Class Demonstrations\FEA\Parts\Shaft-tensile.CATPart
- Open the Generative Structural Analysis workbench. Select Start > Analysis & Simulation > Generative Structural Analysis.
- A Warning Window will pop up saying that the material is not defined. Select ok.
- Select static case from the pop up window
- Select the apply material icon  Select the metal tab and then select steel. Click on the part in the view window and select apply.
- Select the customized view mode  and make sure the mesh shading and material are both selected.

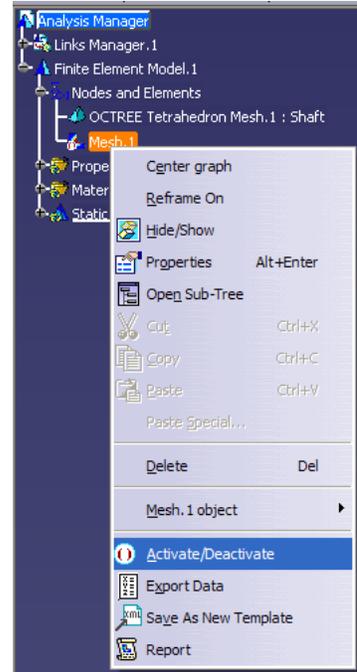


2) Mesh creation:

- Enable mesh visualization. Right click on Nodes and Elements in the design tree and then select mesh visualization. A warning will pop up saying that the mesh needs to be updated. Select ok.
- If the mesh is not an appropriate size, you can modify it. Expand Nodes and Elements under the design tree and double click on the OCTREE Tetrahedron Mesh.1. Input the values of the size and the sag as shown below. And select ok. Notice that the green and blue symbols on the part also change. The green symbol represents the size of each element and the blue symbol represents the sag.



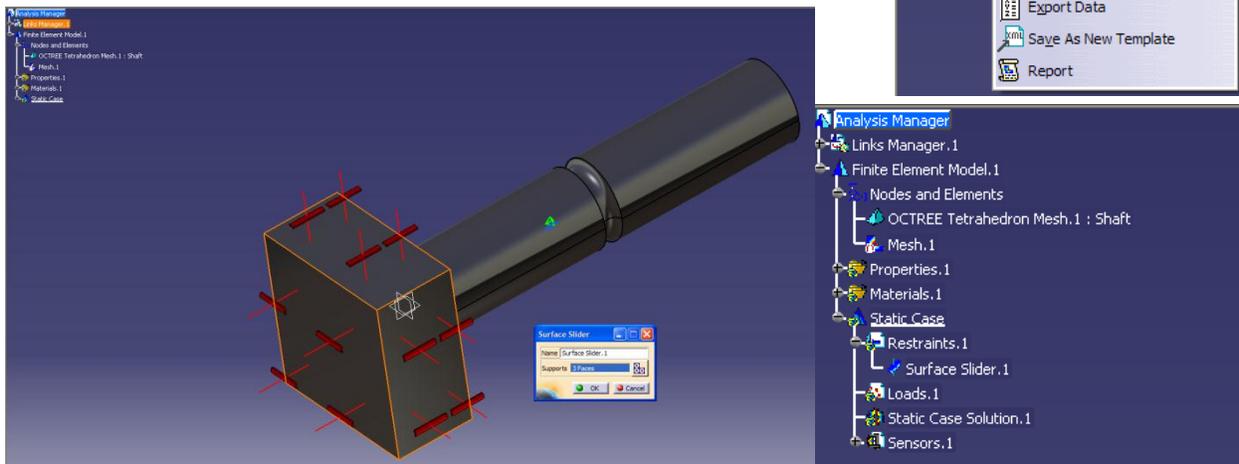
- Notice also that the mesh disappears. Right click on Nodes and Elements and select mesh visualization again to update the mesh. Smaller meshes give more accurate results but take longer to process.
- Now deactivate the created mesh. Right click on Mesh.1 under Nodes and Elements in the design tree and select Activate/Deactivate. This will hide the mesh so restraints can be applied.



3) Apply Restraints:

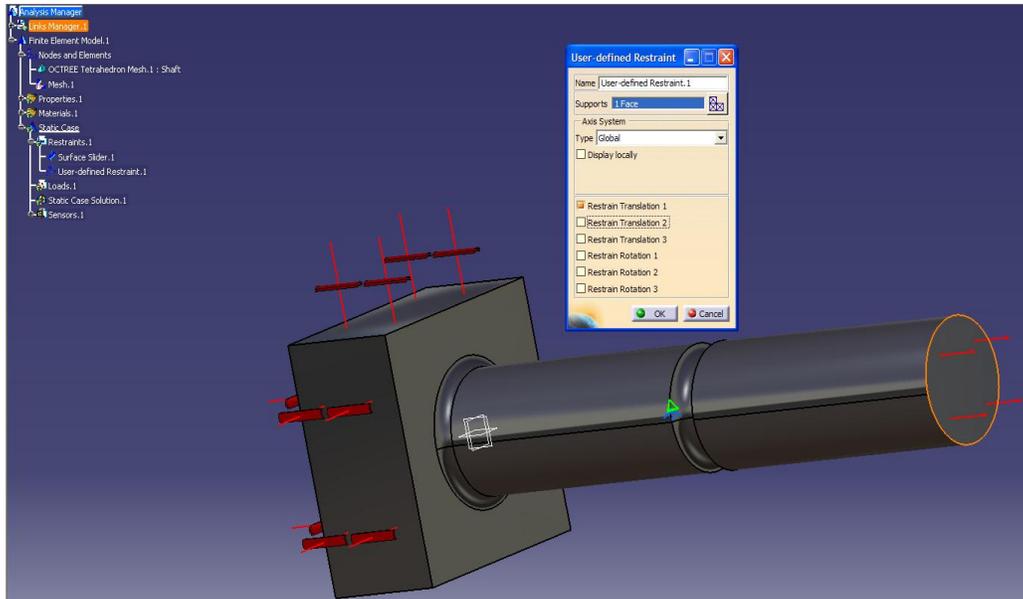


- Select surface slider and select the three faces as shown. This prevents the faces from moving in their tangential directions. Now the part is fully restrained.
- Notice how the surface sliders have been added to the design tree under restrains in the static case.

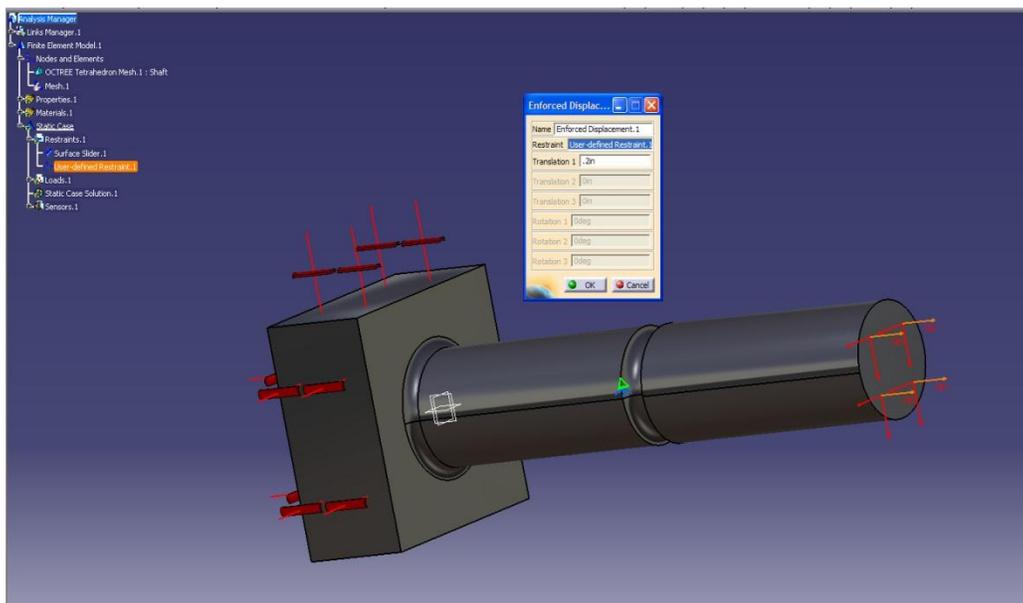


4) Apply loads:

- To apply an enforced displacement, a restraint must first be applied. Select User-defined Restraint . And select the end face as shown. Select restrain translation 1 as shown.



- Next select enforced displacement . Select the restraint that was just created either in the design tree or in the design window. Then enter the desired displacement of .2in as shown and select OK.

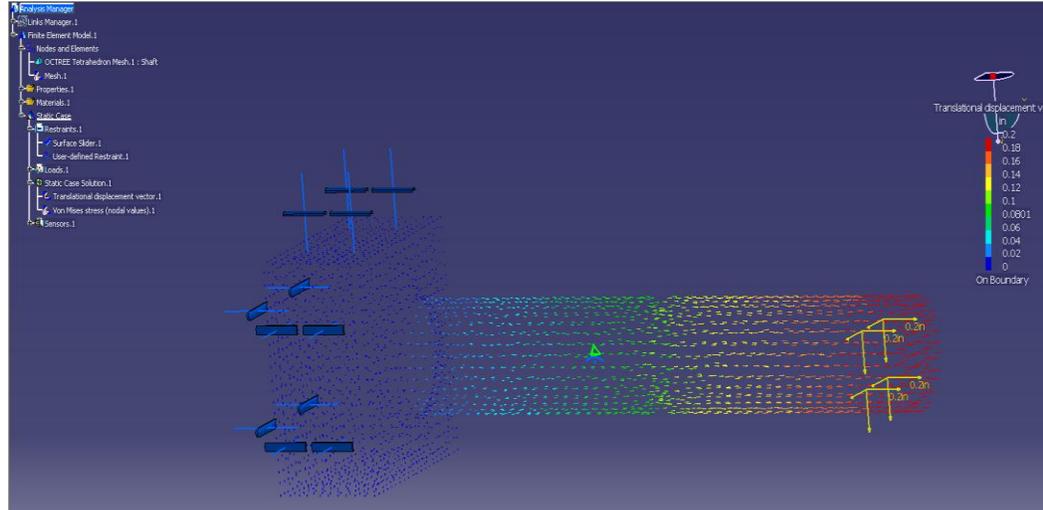


5) Compute solution:

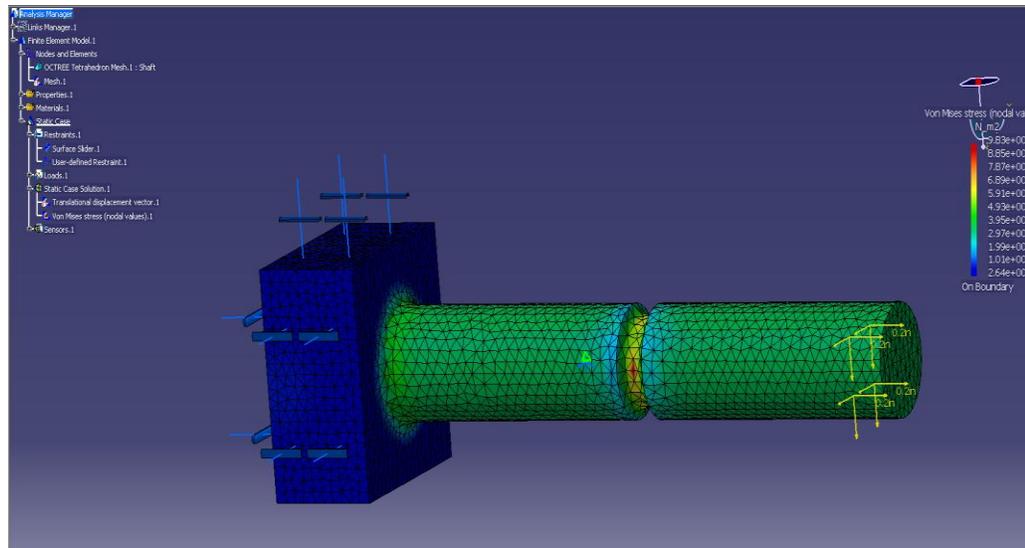
- Select Compute . A pop up box appears asking what you would like to compute. Make sure "all" is selected and then hit OK. If a warning dialogue box appears, select ok. A computation resource box then appears. Select yes and the computation will begin.

6) View Results:

- Select displacement .



- Select Von Mises Stress .



- Each of these analysis views are stored in the static case solution tab in the design tree. To select which one(s) are viewable right click on them in the design tree and select activate/deactivate.

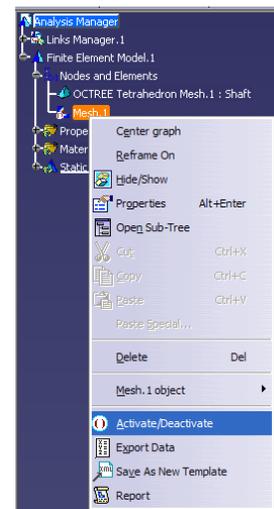
Note that there are multiple ways to go about getting a similar solution.

FEA SHEAR TUTORIAL in CATIA

The following tutorial shows how to set up shear analysis for a part, assuming knowledge from previous tutorials.

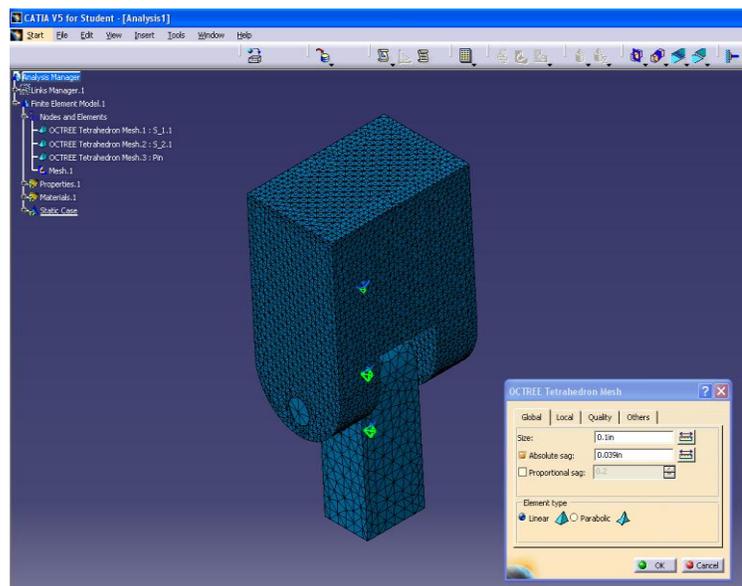
7) Setting Up:

- Open file [Shear](#). S:\HVLC\seniordesign\Catia Course\10. Spring 2011\In-Class Demonstrations\FEA\Parts\Shear.CATProduct
- Open the Generative Structural Analysis workbench. Select start-Analysis and Simulation-Generative Structural Analysis.
- Select static case from the pop up window
- On the design tree, you will see two icons under the Analysis Manager. The first one is the Links Manager. This links the Analysis to the specified part(s) or product(s). You will notice that the part has automatically been added to the links manager.
- the material has already been applied to the parts
- Select the customized view mode  and make sure the mesh shading and material are both selected.



8) Mesh creation:

- Enable mesh visualization. Right click on Nodes and Elements in the design tree and then select mesh visualization. A warning will pop up saying that the mesh needs to be updated. Select ok.
- If the mesh is not an appropriate size, you can modify it. Expand Nodes and Elements under the design tree and double click on each OCTREE Tetrahedron Mesh. Input the values of the size and the sag as shown below to each mesh and select ok. Notice that the green and blue symbols on the part also change. The green symbol represents the size of each element and the blue symbol represents the sag.
- Notice also that the mesh disappears. Right



click on Nodes and Elements and select mesh visualization again to update the mesh. Smaller meshes give more accurate results but take longer to process.

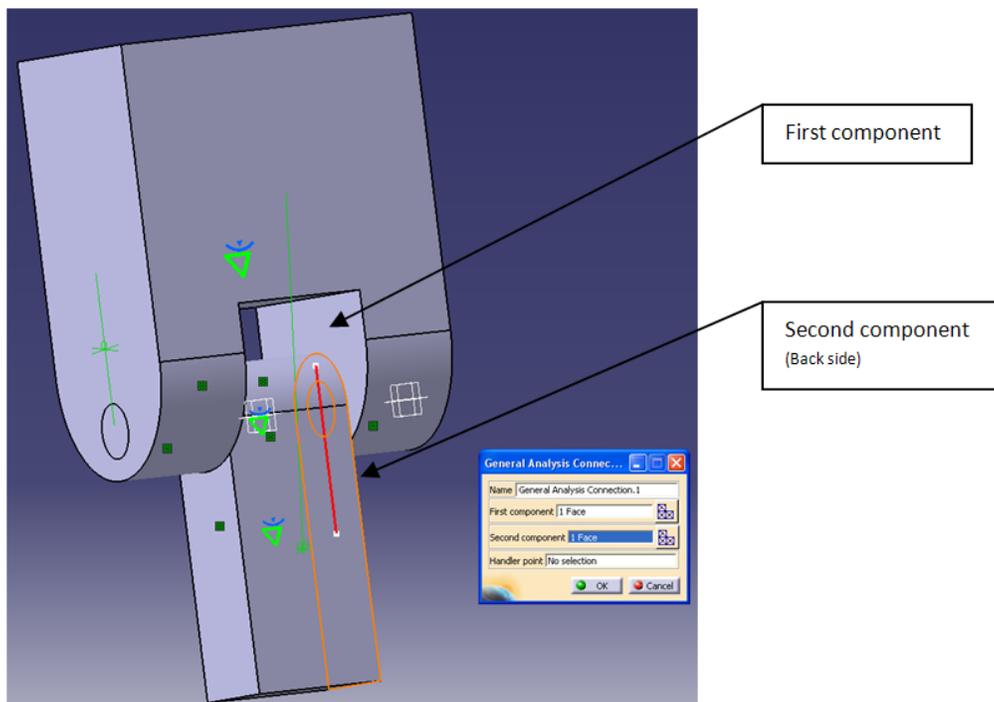
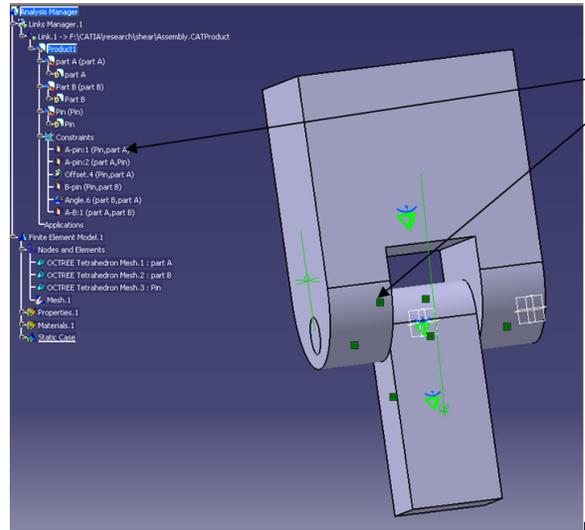
- Now deactivate the created mesh. Right click on Mesh.1 under Nodes and Elements in the design tree and select Activate/Deactivate. This will hide the mesh so restraints can be applied.

9) Create analysis supports:



- The supports for this assembly have been created in the Assembly Workbench. They are simply the restraints. They can be seen by expanding the links manager in the design tree to the constraints and can also be seen in the design window. For demonstration purposes we will add one Analysis Support. This support will constrain the contacting faces of parts A and B.

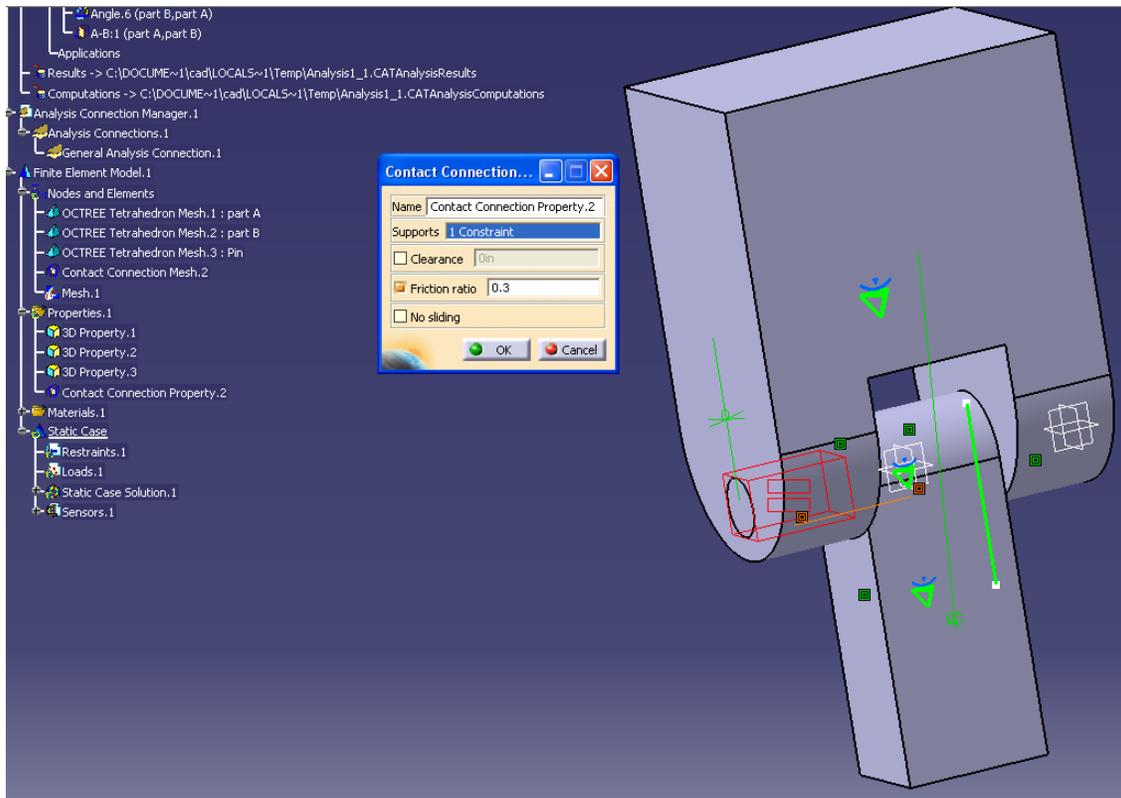
- Select the general Analysis Connection icon . For the first component and second components, select the contacting faces as shown. Select ok



10) Define connection Properties:

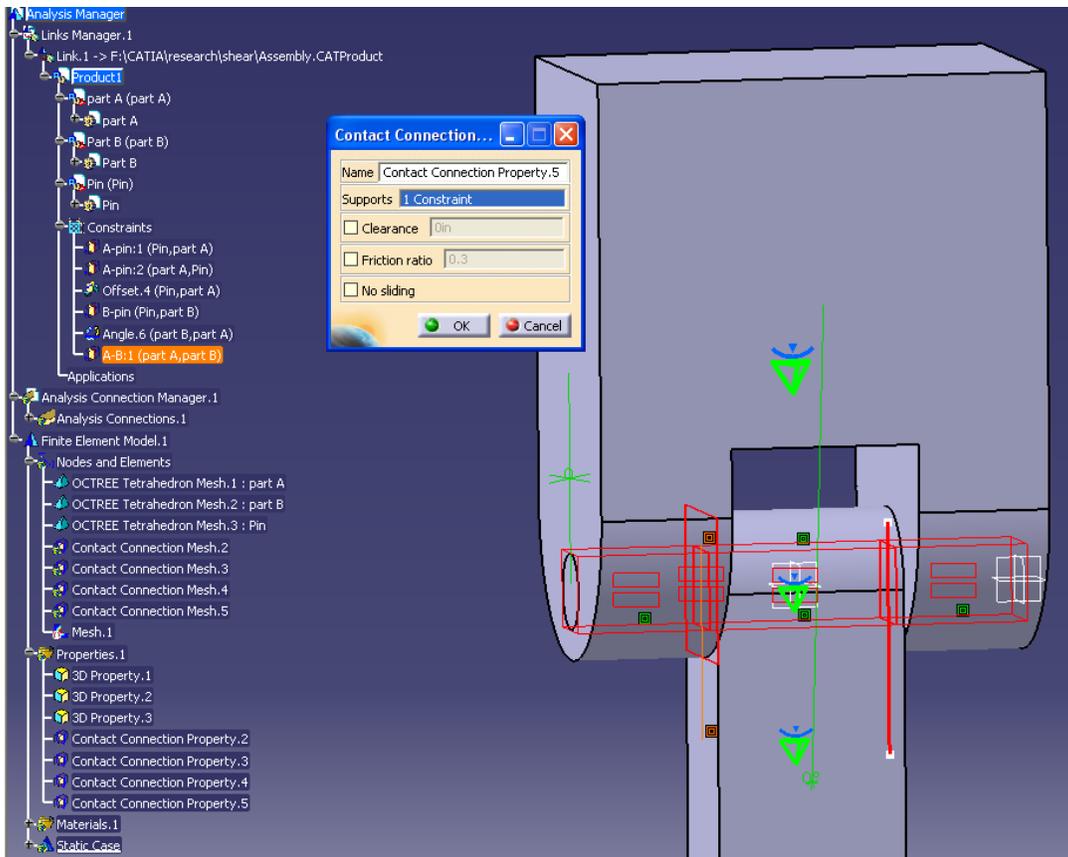


- Now properties need to be added to the connections which restrain parts A and B to the pin.
- Select Contact Connection  from the connection properties toolbar. Now expand Links Manager in the design tree so that the constraints are visible. Select A-pin:1 for the support and select the friction ratio box. This will restrain the pin from moving in the X direction. Select OK.



- Repeat the previous step for the two remaining constraints A-pin:2 and B-pin.
- Now connection properties will be added to constrain the wall contacts of part A and B. One of these connections was made in the Assembly Workbench using a contact restraint and the other is the one which was previously created.
- Select Contact Connection  and select A-B:1 in the design tree as shown. A-B:1 is a contact constraint between the walls of parts A and B. Select ok.
- Select Contact Connection again and now select either the red line in the design window which represents the Analysis connection we previously made or select it in the design tree as shown and select ok.

NOTE: FOR MORE INFORMATION ON ASSEMBLY CONNECTIONS GO [HERE](#). T:\Catia Course\10. Spring 2011\In-Class Demonstrations\FEA\TUTORIAL 2\FEA Assemblies Connections

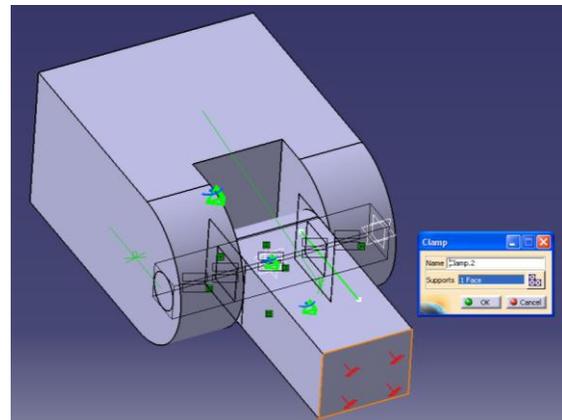


11) Apply Restraints:

- Select clamp  from the restraints toolbar and select the bottom face of part B as shown.

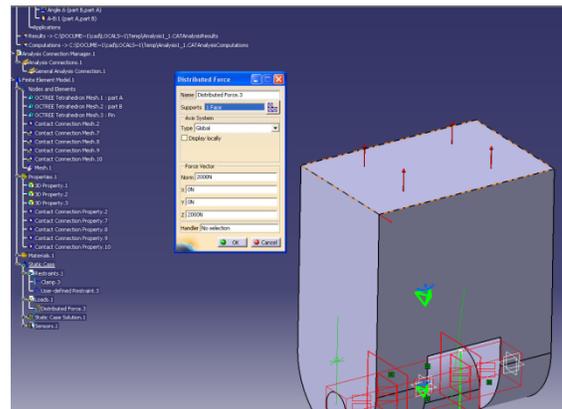
12) Apply loads:

- Select distributed force  and select the top face as shown. Enter a value of 2000N in the Z direction.

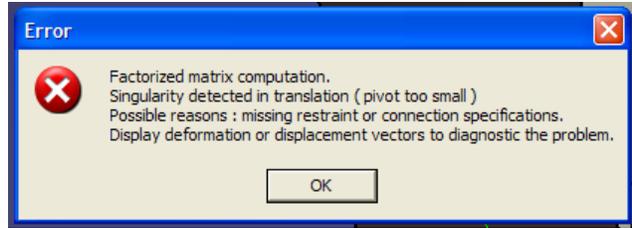


13) Compute solution:

- Select Compute . A pop up box appears asking what you would like to compute. Make sure "all" is selected and then hit OK. If a warning dialogue box appears, select ok. A computation resource box then appears. Select yes and the computation will begin.
- If an error box appears as shown. It usually means that the model is not fully restrained. Select ok and then



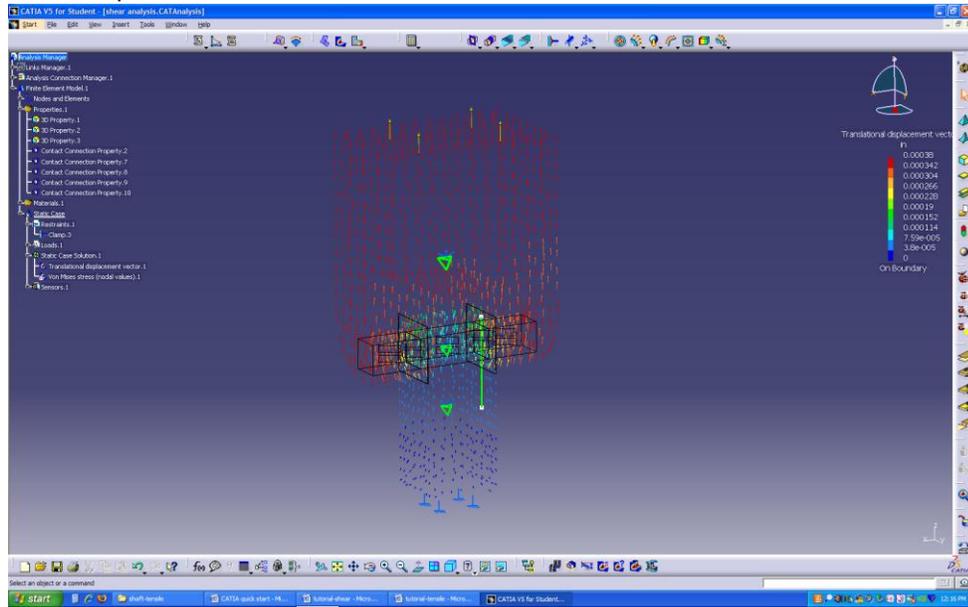
select the displacement icon  to diagnose what is wrong with your model.



14) View Results:



- Select displacement .



- Select Von Mises Stress .

