Nader G. Zamani

# FINITE ELEMENT ESSENTIALS IN 3DEXPERIENCE® 2021x

SDC PUBLICATIONS

Better Textbooks. Lower Prices.
www.SDCpublications.com

Visit the following websites to learn more about this book:

# Chapter 2

# Linear Elastic Analysis of a Notched Plate

## Introduction:

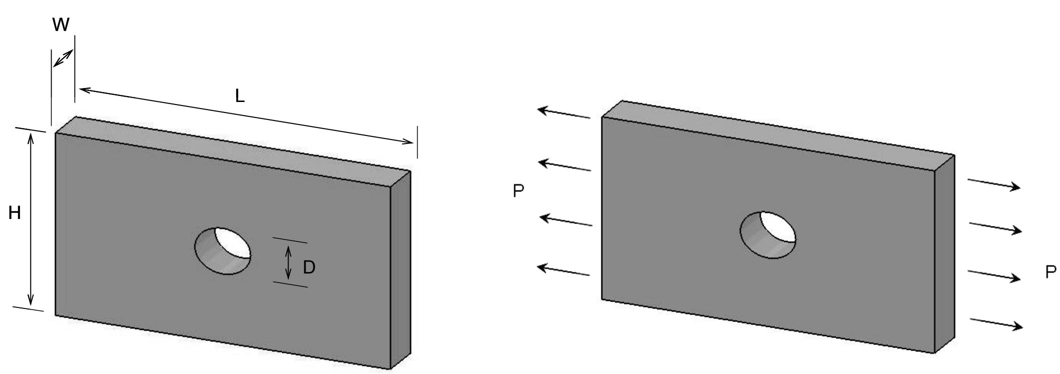
In this tutorial, a solid finite element model of a plate with central hole is created. The loading is in the plane of the plate and the deformation is small enough to warrant a linear elastic analysis.

NOTE: It is assumed that you have basic familiarity with CAD modeling in 3DEXPERIENCE allowing you to create a block with a central hole. If that is not the case, please consult the following tutorial book.

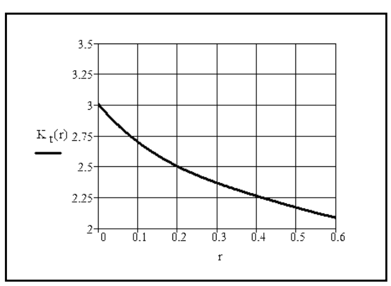
CAD Modeling Essentials in 3DEXPERIENCE, by Nader Zamani, SDC Publications, ISBN 978-1-63057-095-8.

## Problem Statement:

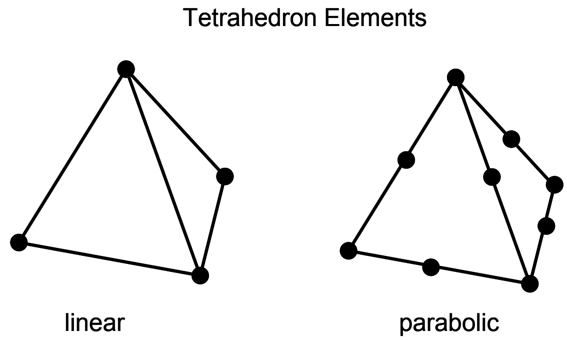
The steel plate shown below is subjected to a pressure load P at the two ends. Although the problem has three planes of symmetry, you will be modeling the full geometry. The loading is assumed to be small enough to cause a linear elastic behavior.




The dimensions of the part to be analyzed are such that the parameter $r = \dfrac{D}{H}$ is 0.25. The chart below gives a stress concentration factor of $K_t = 2.4$. This chart is based on $L >> H$ which is not true in the modeled block. However, the value of $K_t$ will be used for comparative purposes.



There are many types of solid elements available in the software; however, in this chapter, tetrahedron elements are used. They come in linear and parabolic forms. Both are referred to as tetrahedron elements and shown below.




The linear tetrahedron elements are faster computationally but less accurate. On the other hand, the parabolic elements require more computational resources but lead to more accurate results. Another important feature of parabolic elements is that they can fit curved surfaces better. In general, the analysis of bulky objects requires the use of solid elements.

In a solid continuum, the state of deformation is described by the six components of the Cauchy stress $\{\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{xz}, \tau_{yz}\}$ which vary from point to point. The von Mises stress is a combination of these according to the following expression:

$$\sigma_{VM} = \sqrt{\frac{1}{2}\left[(\sigma_x - \sigma_y)^2 + (\sigma_x - \sigma_z)^2 + (\sigma_y - \sigma_z)^2 + 6(\tau_{xy}^2 + \tau_{xz}^2 + \tau_{yz}^2)\right]}$$

For an obvious reason, this is also known as the effective stress. Note that by definition, the von Mises stress is always a positive number. In terms of principal stresses, $\sigma_{VM}$ can also be written as

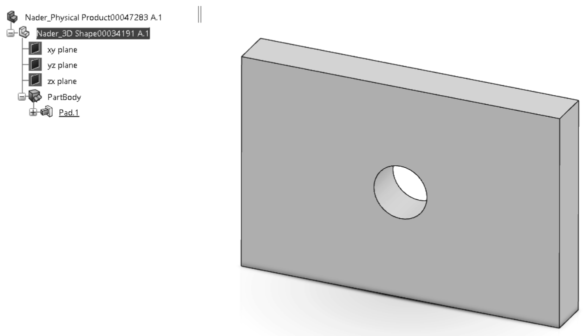
$$\sigma_{VM} = \sqrt{\frac{1}{2}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_1 - \sigma_3)^2 + (\sigma_2 - \sigma_3)^2\right]}$$

For many ductile materials, the onset of yielding (permanent plastic deformation) takes place when $\sigma_{VM} = \sigma_Y$ where $\sigma_Y$ is the yield strength of the material. For design purposes, a factor of safety “N” is introduced leading to the condition $\sigma_{VM} = \dfrac{\sigma_Y}{N}$.

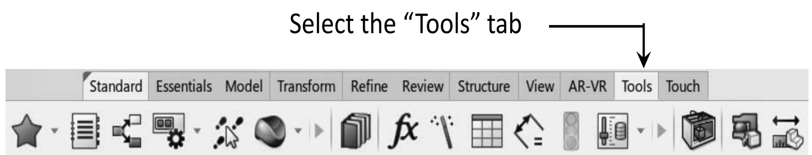
Therefore, a safe design is one where $\sigma_{VM} < \dfrac{\sigma_Y}{N}$. The von Mises stress contour plot allows you to check the above condition.

## The Model and Material Properties:

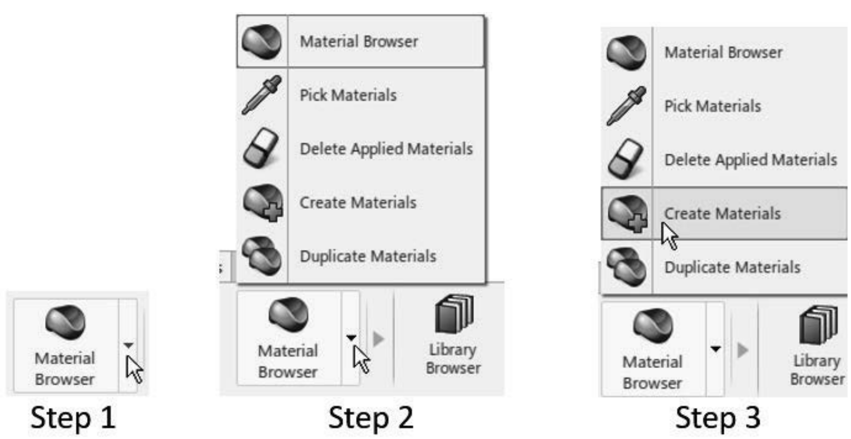
First, using the Part Design App , create a block with a central hole with the dimensions L = 0.15m, H= 0.1m, W = 0.02m, and D = 0.025m as shown below.




The first task is to apply a material property to this part. From the bottom row of icons (i.e. the action bar), select the “Tools” tab.

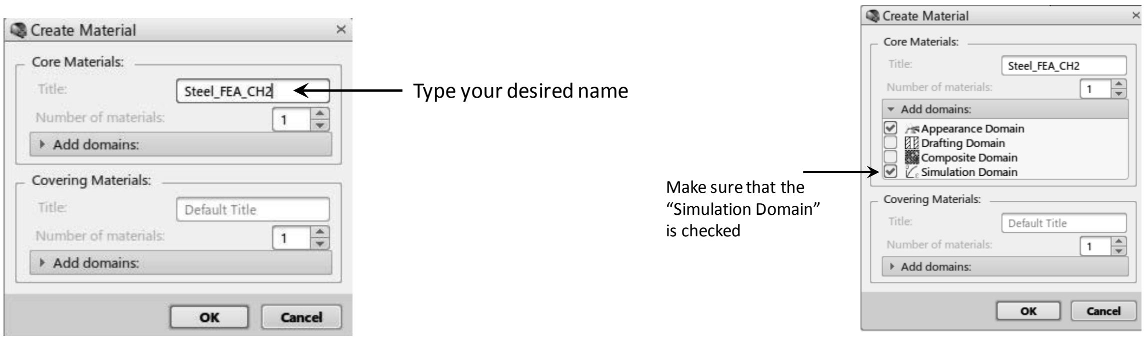



From the Tools menu, select the dropdown to the right of the “Material Browser” icon . This opens up the section menu as shown. Follow the steps outlined below to select the “Create Material” .

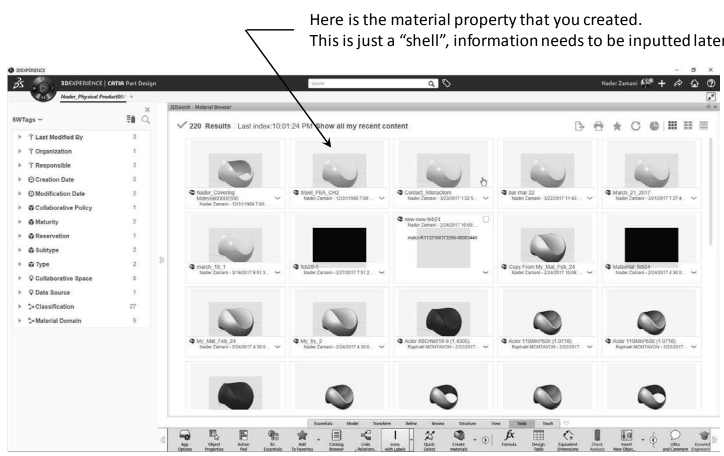



The selection of  opens up a dialogue box shown on the next page. This box allows you to supply a proper name for the material should you decide to do so. Our assumption is that you do not have a material of interest in the "cloud" database and would like to follow the steps to create it. It is a rather tedious process but will be clearly spelled out.

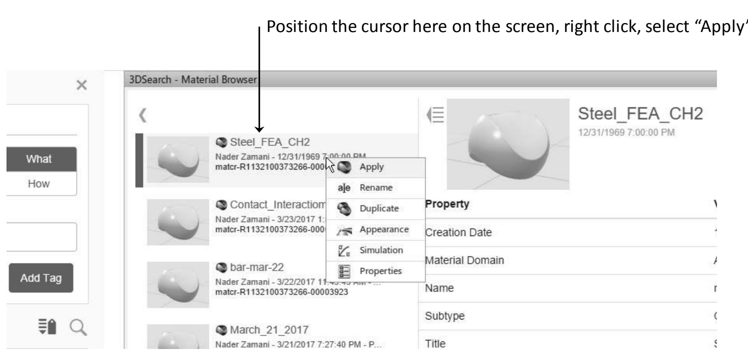
Select the "Create Material"  icon. Make sure that you check "Add domain" section of the dialogue box, and that the "Simulation Domain" is picked. Note that this creates a shell (a placeholder) and the material information needs to be supplied later.



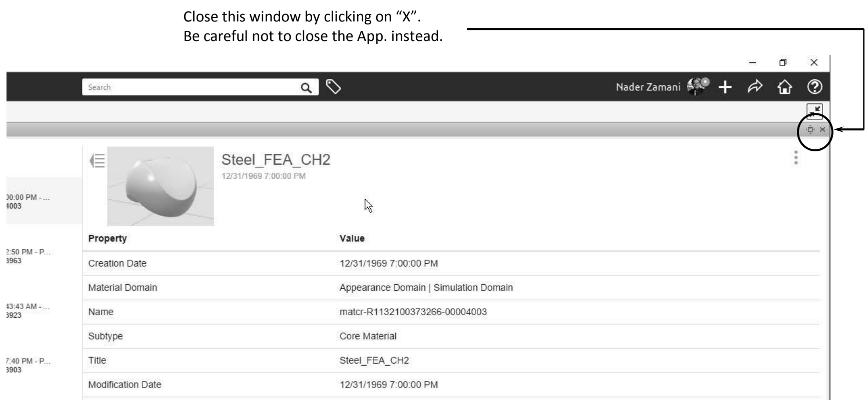

Once you close the dialogue box by clicking on "OK", you will find yourself in the material database and can identify the material that you just created, namely "Steel_FEA_CH2". The database screen is shown on the right.



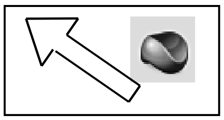

Place the cursor on your created material in the database, right click and select "Apply". You still have to return to the screen where the geometry exists and continue. This necessitates the closure of the current screen (the database screen).



Select the "X "on the top right margin of the database screen to close the window.

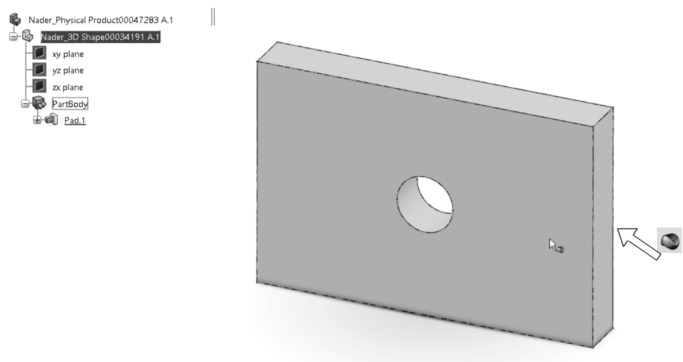


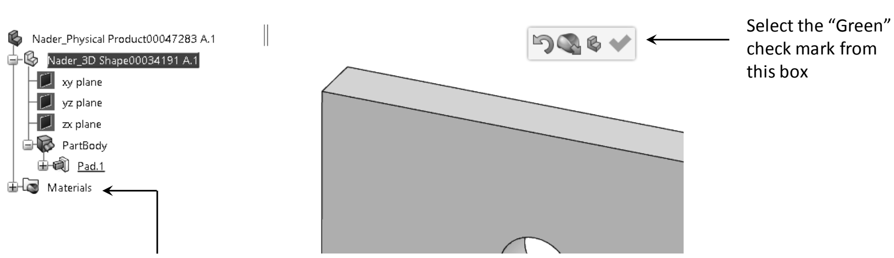

You will return to the geometry window; however, the shape of the cursor is modified as shown below.



Place the cursor on the part on the screen or on the top branch of the tree and double click.

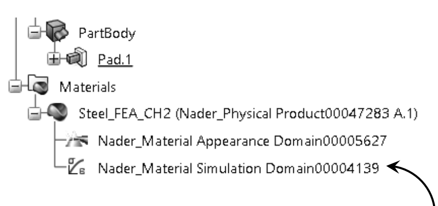
You will notice that the "Materials" branch is created at the very bottom of the tree as shown below. You can then use the cursor to select the "Green" check mark to proceed.



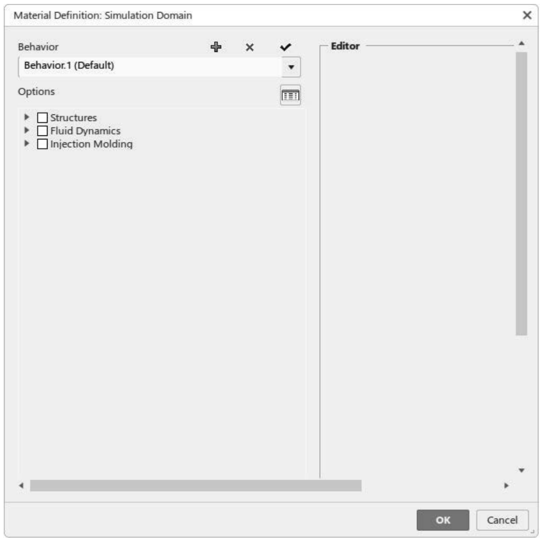

The created material has been assigned but this is just a "shell" , information needs to be inputted later.

Please note that the actual material properties are yet to be inputted. Expanding the "Materials" branch reveals two other branches. The location where the properties are inputted is the last branch "Material Simulation Domain00004139" as shown on the right.

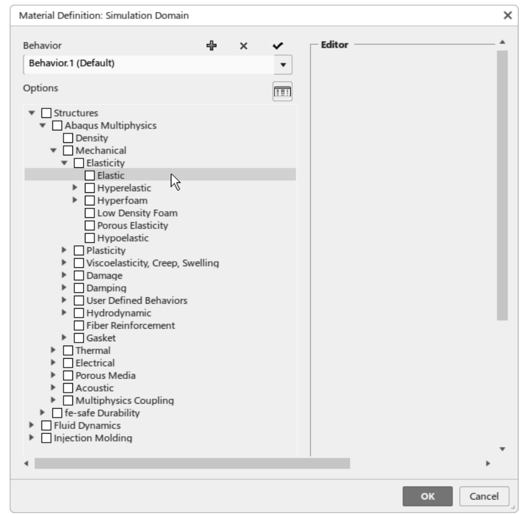



Input material properties by double clicking on this branch

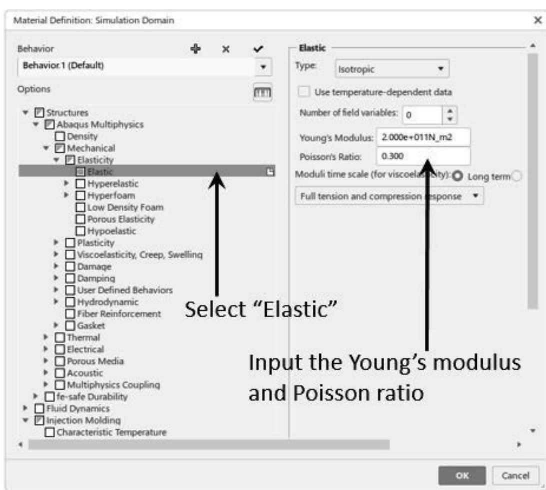
Double click on the last branch and follow the steps below.




**Step 1**




**Step 2**



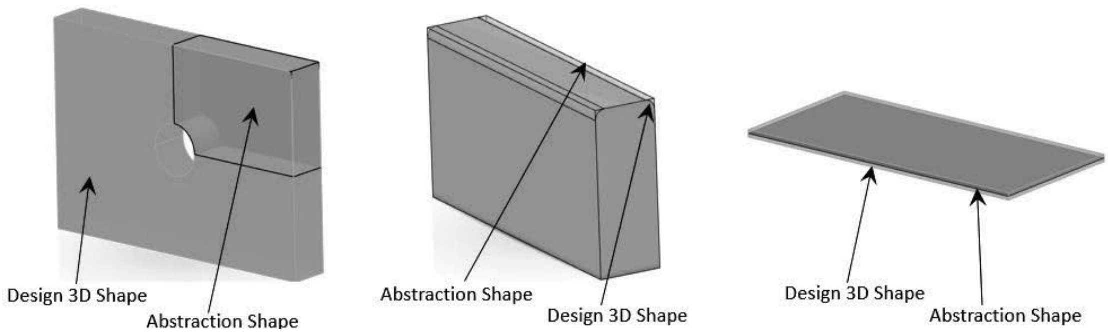

**Step 3**

**Steps:**
1. Expand the Structures Option
2. Continue Expanding until the Elastic Option is visible
3. Click on the Elastic Option & input the Young’s modulus and Poisson Ratio.

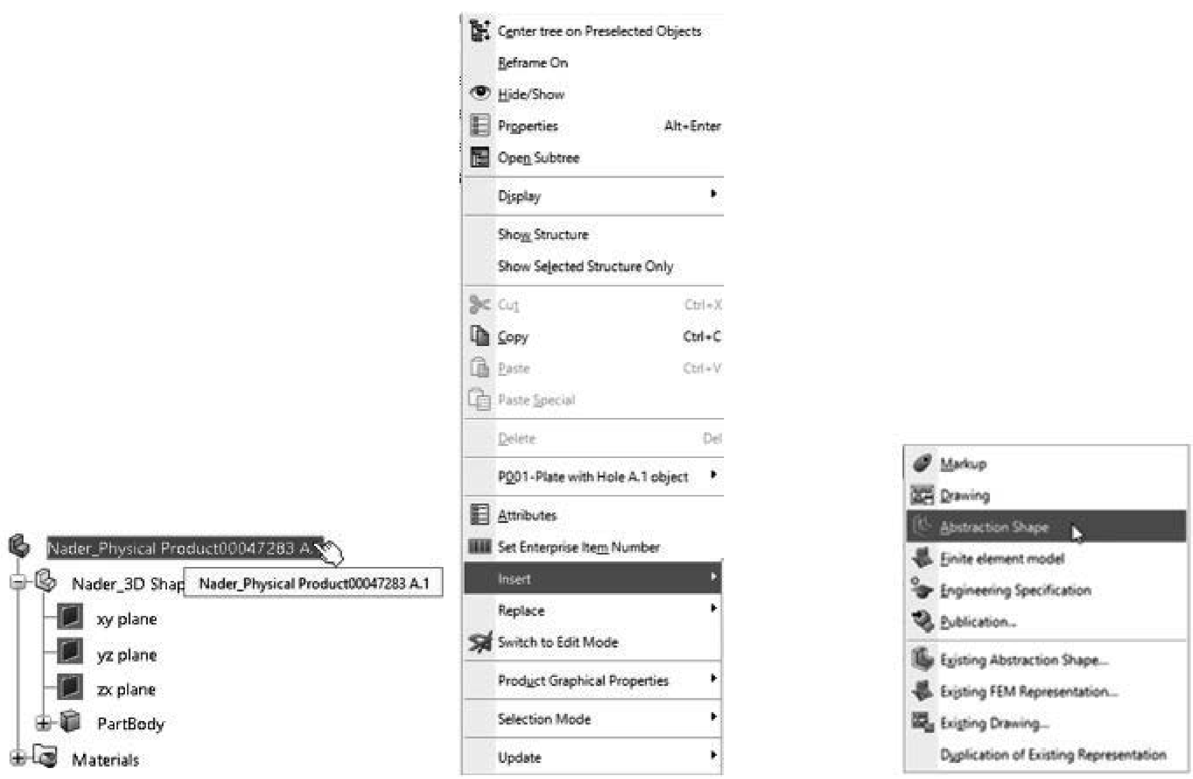
In Step 3, the Young’s modulus and Poisson ratio can be inputted. For the present problem, $E = 2 * 10^{11}\ Pa$ and $\nu = 0.3$, which are the standard values for carbon steel. Properties can be changed by simply double clicking on the last branch shown.

## Creating an Abstraction Shape:

When performing a structural analysis, one often simplifies the model. These simplifications may include modeling a symmetric portion of a component, removing small non-critical features, or even modeling a solid component as a shell. Modifying the 3D Shape used for designing and manufacturing the component could cause costly errors. Fortunately, **3D**EXPERIENCE provides a feature called an "Abstraction Shape". The Abstraction Shape provides a container with a dependent copy of the design 3D Shape.



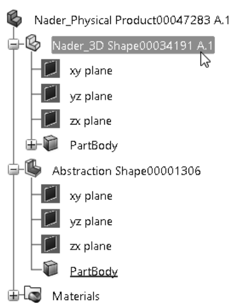

Inserting an Abstraction Shape is a straightforward process. As shown below, right click on the part, select Insert, and then select Abstraction Shape.



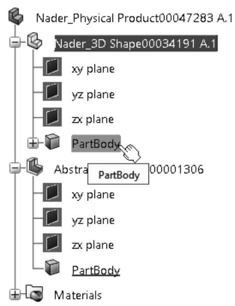

**Step 1** **Step 2** **Step 3**

Creating a dependent copy of the 3D Shape, used for the design and manufacturing, and placing it in the Abstraction Shape requires a bit more effort. To accomplish this task:

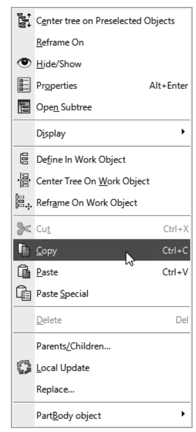
1. Double click on the 3D Shape to activate it
2. Right click on the PartBody associated with the 3D Shape
3. Select Copy
4. Double click on the Abstraction Shape to activate it
5. Right click on the PartBody associated with the Abstraction Shape
6. Select Paste Special
7. Select "As Result With Link" and then click on OK.



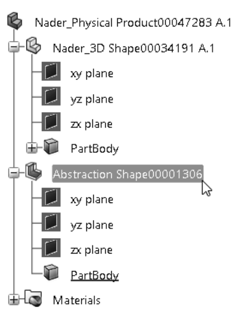
Step 1



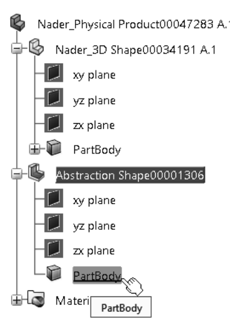
Step 2



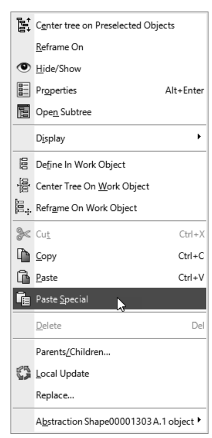
Step 3



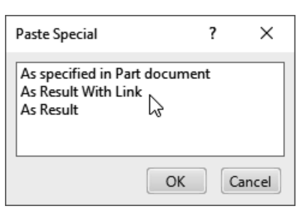
Step 4



Step 5



Step 6

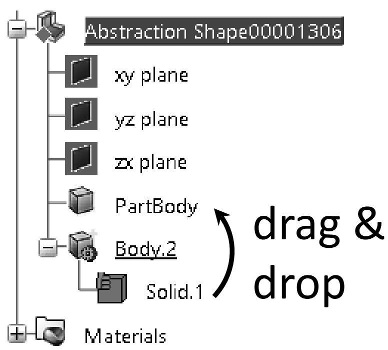



Step 7

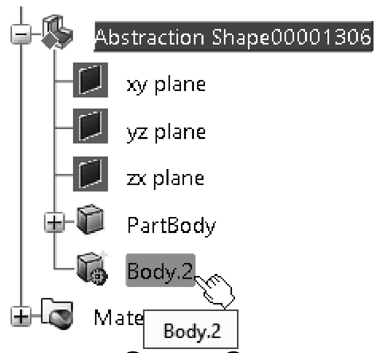
For most simulations, we could leave the model as it is. But some of the functionality used by the simulation apps requires the solid to be located under the PartBody. Therefore, we perform the following three steps to clean up the model.

1. Drag and drop the solid to the PartBody
2. Right click on the now empty body container
3. Select delete

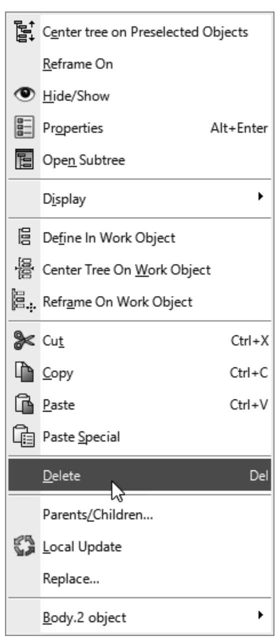
The steps to clean up the model and the final model tree are depicted below.



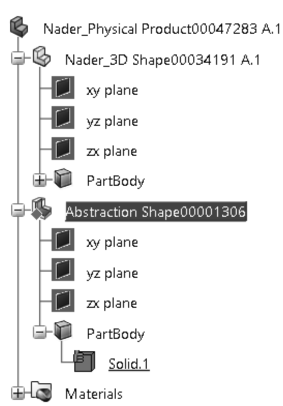

Step 1




Step 2

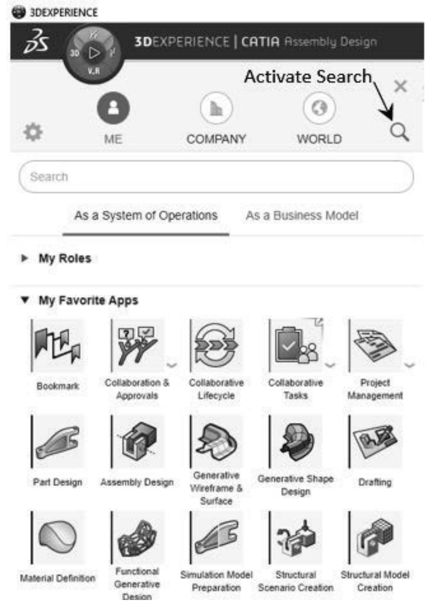


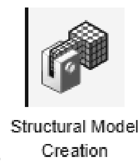

Step 3




Final

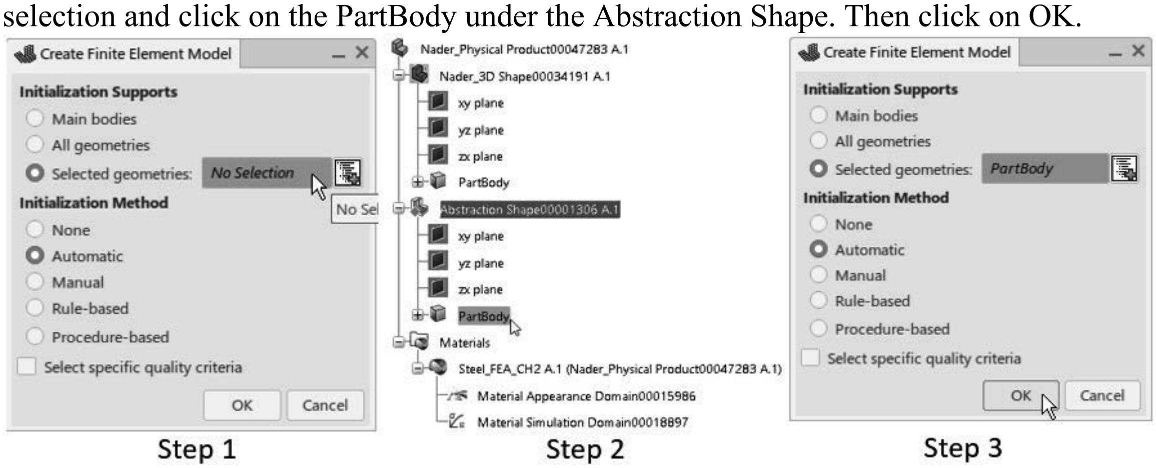
## Creating the Finite Element Model:



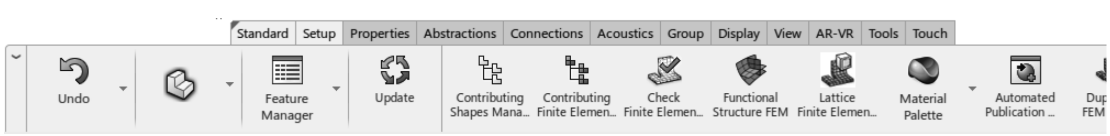

Click on the compass at the top left corner of the screen. Be sure that the "ME" and "As a System of Operations" choices are active as depicted in the figure to the right. If you have difficulty locating any app, you can activate the search capability by clicking on the magnifying glass. You can also drag any app into your "My Favorite Apps" for easy access in the future. Scroll through the applications and select the "Structural Model Creation" App 

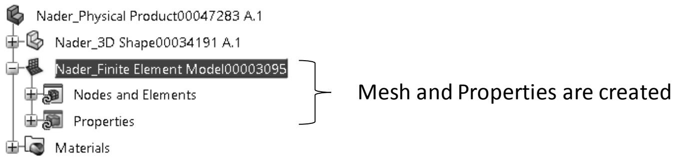
.

The "Create Finite Element Model" dialogue box appears. For now, select the "Automatic" option for the Initialization Method. The other initialization methods provide a user with additional control over the meshing process. The selection of "Automatic" creates parabolic tetrahedral elements. Next, activate the geometry selection and click on the PartBody under the Abstraction Shape. Then click on OK.




The row of icons at the bottom of your screen (action bar) changes and will appear similar to the one displayed below.

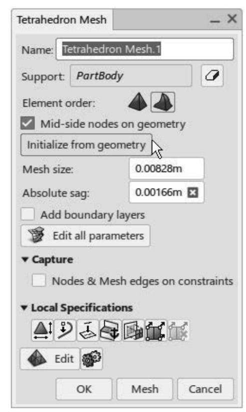




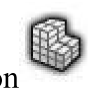

The tree indicates that a mesh and the corresponding solid section has been created.

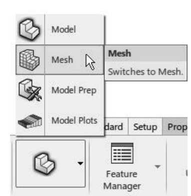

Expanding the branch "Finite Element Model0003095" further indicates that the elements are of "Tetrahedron" type and the property is "Solid Section" as expected.

Upon double clicking on the "Tetrahedron Mesh.1" branch, the corresponding dialogue box shown on the right pops up. Here, one can change the type (linear or quadratic), the size and sag, and certain other parameters. For example, local mesh refinement can be accomplished through the "Local Specifications". Click on "Initialize from geometry" and then click on OK.

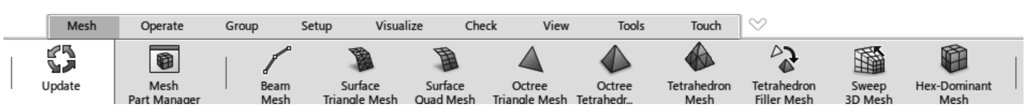



There are different methods for displaying the mesh. The instructions given below are one method of achieving that objective.

From the bottom row of icons, select the "Mesh" icon  .



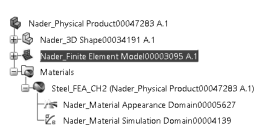

The bottom row's appearance now looks similar to below.

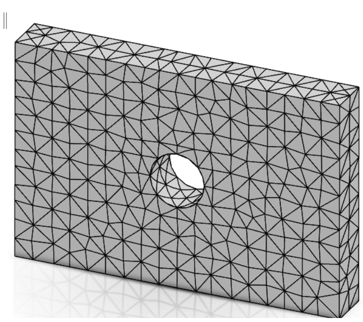



Select the "Update" icon

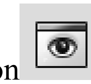

.

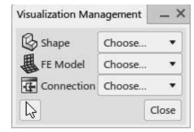
Upon updating, the mesh appears on the screen as shown on the right.



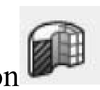



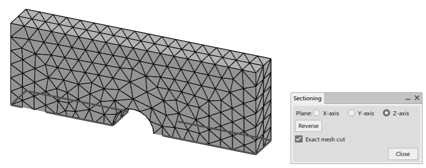
There are also different ways of hiding the mesh. For example, first select the “Display” tab from the bottom row of icons.

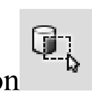

you can select the “Visualization Management” icon  from the choices. The resulting dialogue box shown on the right appears. You can then use the dropdowns to show or hide the various features.

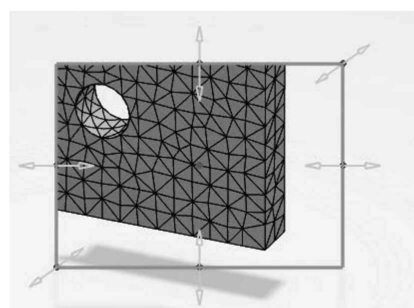



There are a few other icons that are worth mentioning here.

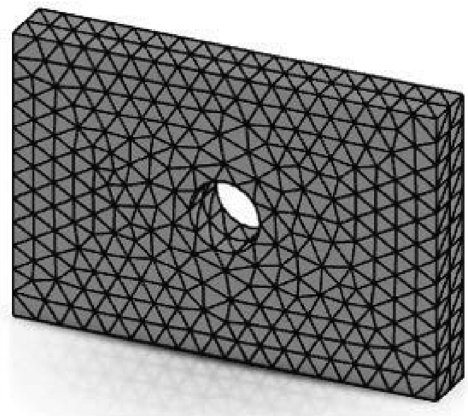
Selecting the “Sectioning” icon  cuts the mesh with the standard xy, xz, and yz planes as shown.

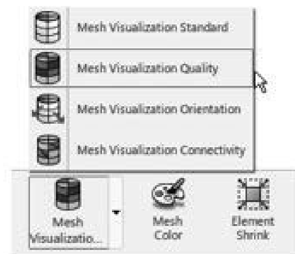


The “Clipping Box” icon  enables you to select a region of your mesh.



There are several other mesh visualization tools available including changing element color and shrinking the elements. We do not discuss those further but will highlight the mesh “Mesh Visualization Quality” plot. This allows one to quickly identify poor quality elements that could produce erroneous results.

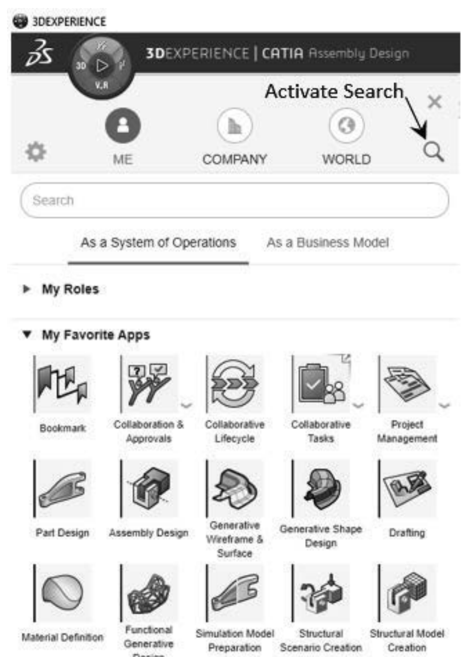




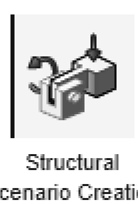

**Creating a Scenario:**

The nature of the analysis, namely Static, Dynamic, Buckling, etc., is set in the "Structural Scenario Creation" App. Furthermore, the loads, restraints, and interaction are also defined in this application.

It is also important to point out that that one could have created the Scenario before the "Finite Element Model Creation" step. In fact, the latter can be created from within the "Structural Scenario Creation" App.

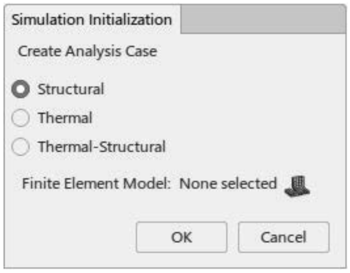


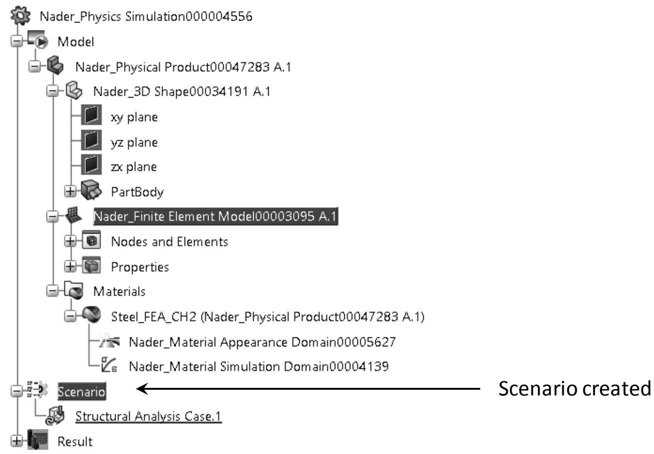

Click on the compass in the top left corner of the screen as shown here. Scroll through the applications and select the "Structural Scenario



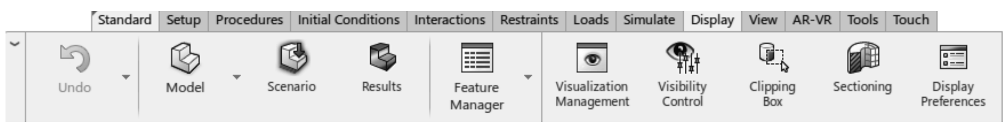

Creation" App.

The pop-up window "Simulation Initialization" shown below appears on the screen. Since this is strictly a structural problem, the radio button "Structural" should be selected. A quick glance of the tree confirms that a "Scenario" has been created.

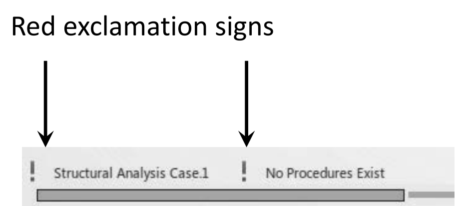


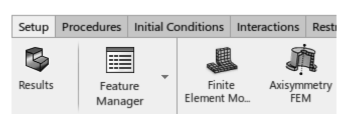



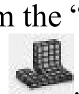

The row of icons at the bottom of your screen changes and will resemble the one displayed below.



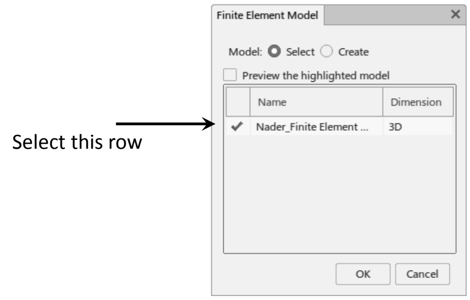

Checking the middle bottom section of the screen reveals that there are two red exclamation signs.
These pertain to "Structural Analysis Case.1" and "No Procedures Exist".



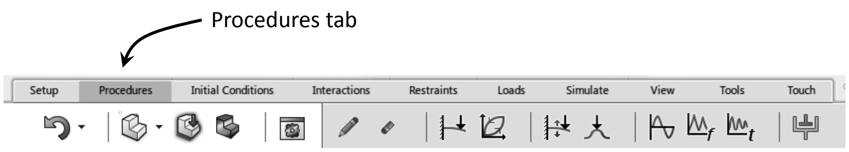



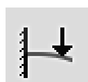

From the "Setup" tab, click the "Finite Element Model" icon . The following pop-up window displayed on the next page appears.

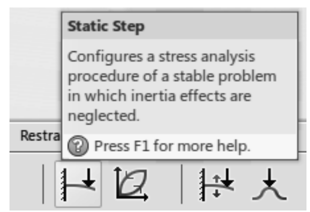
Since there is already a finite element model created, it appears in the list. Be sure that you select that row. Notice that there is also an option to create a finite element model. As mentioned earlier, the FE model can also be created within the Scenario, and this is where you would complete that task. In this case, it does not apply because an FE model was already created and selected.



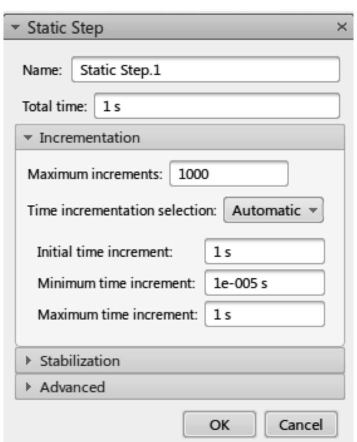

Select the "Procedures" tab from the action bar (bottom row).

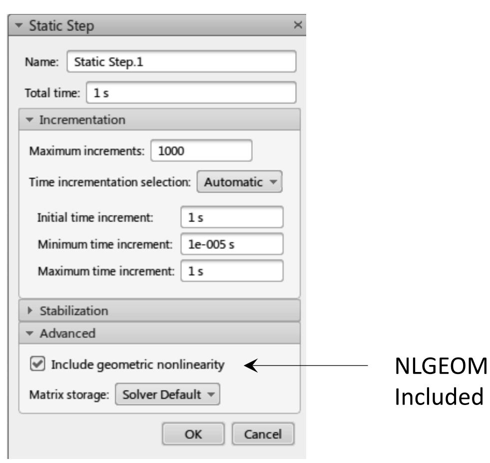



Select the "Static Step" icon  from the bottom row.

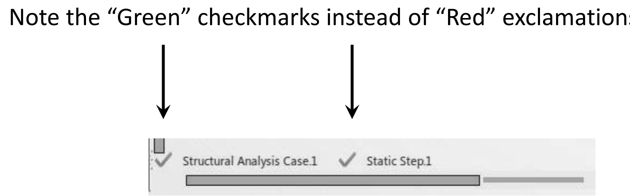



The "Static Step" dialogue box pops up. Accept all the defaults. Note that if the "Advanced" pulldown list is selected, it becomes clear that this is the point in the software where "Geometric Nonlinearities" are included, or excluded.

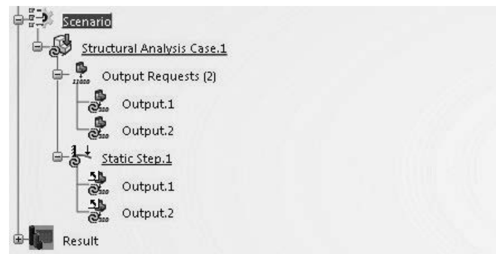


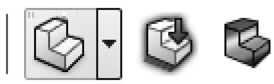
A quick glance at the bottom middle section of the screen reveals "Green" checkmarks instead of "Red" exclamation marks.




The tree indicates that "Static Step.1" has been created. There are default basic output entities that are requested upon the creation of a Step.

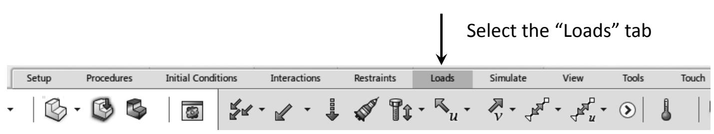



**Accessing the Model, Scenario and Results Quickly:**

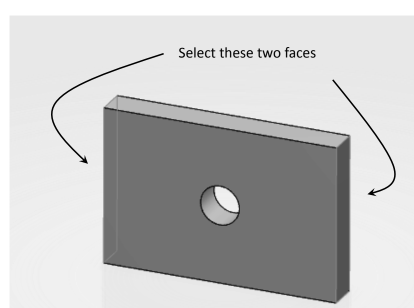
Clearly this can be done by double clicking on the corresponding branches of the tree. However, it can also be done efficiently by selecting the appropriate icon among these three: . The first one on the left is "Model and Mesh", the middle one is "Scenario" and the one on the far right is "Results"; that is the postprocessor.

**Applying the Pressure Load:**

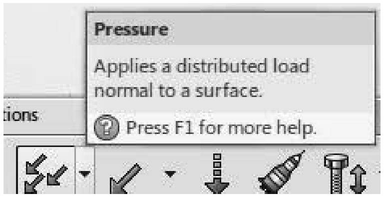
A pressure load of -100000 Pa is to be applied on the end faces. Note that positive pressure by convention is a compressive load. Since the notched block is under tension, pressure value is inputted as a negative number.




Select the two end faces of the block; keep in mind the multiple selection is done with holding the Ctrl key down.

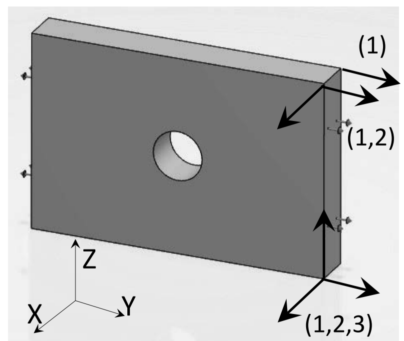



Select the "Pressure" icon  from the bottom row of icons and input the pressure value.




## Applying the Restraints:

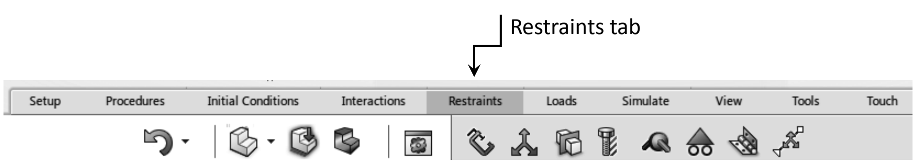
The problem under consideration is in static equilibrium without imposing any restraints. However, due to the geometrical and loading symmetries, it possesses three planes of symmetry. Therefore, in principle, one needs to model 1/8$^{th}$ of the geometry only.



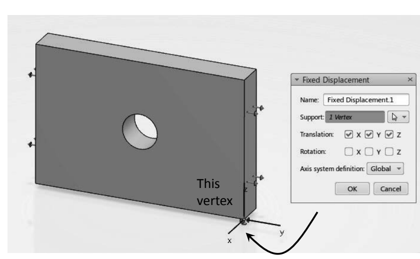

In the present model, no symmetry considerations will be made. Since the structure can move as a rigid body and still remain under equilibrium, the so called {1,2,3} rule will be imposed. This is illustrated in the figure on the right. The arrows represent a zero displacement in the shown direction.

The (1,2,3) restraint representing (x,y,z) prevents the block from flying away. The (1,2) restraint representing (x,y) prevents rotation about the x and y axis. Finally, the (3) restraint representing (z) eliminates the rotation about the z axis.

Select the “Restraints” tab from the bottom toolbar (the action bar).

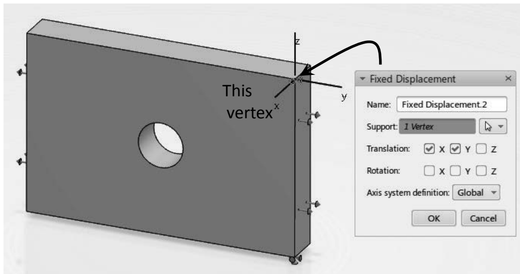


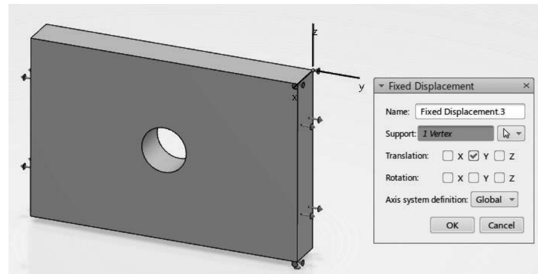

Choose the “Fixed Displacement” icon  followed by the bottom vertex as shown.

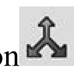



In the dialogue box, check all three “Translations”.

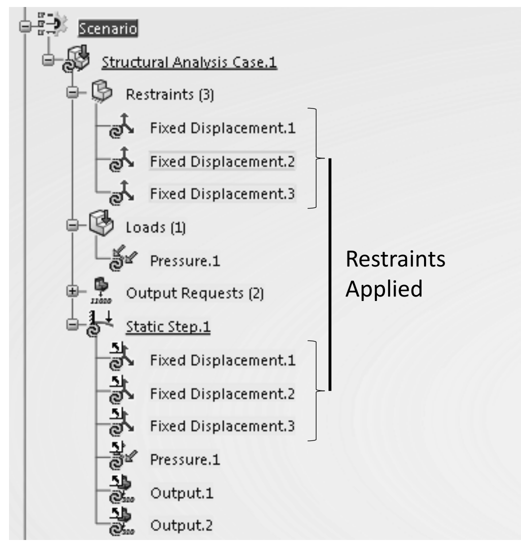
Repeat the same process for the other two vertices and choose the appropriate translations as shown.



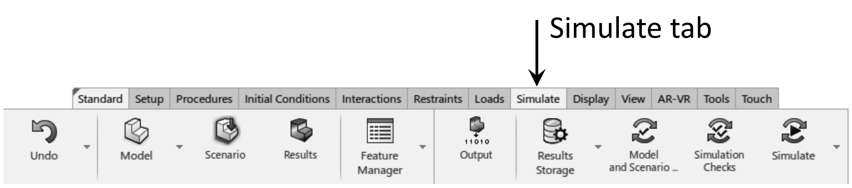

As a side comment, instead of using the "Fixed Displacement" icon  and checking all three x,y,z translations, one could have used the "Clamp" icon .

Note that the imposed restraints are now reflected in the tree.



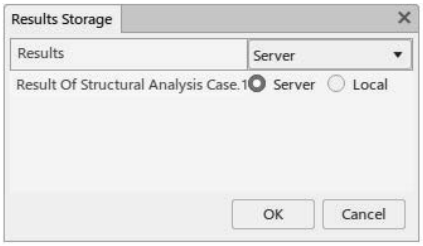

**Consistency, Model Check, and Simulation:**

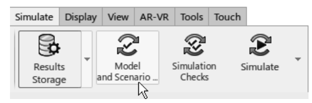
Select the "Simulation" tab from the bottom row of icons on your screen



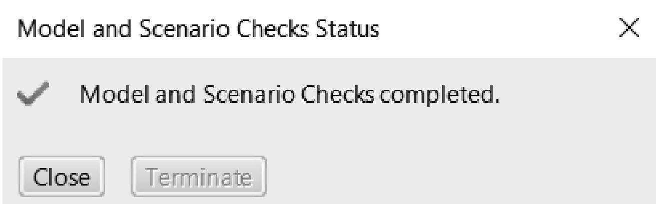

Select the "Results Storage" icon  from the bottom row to define where the simulation results will be stored. While model and scenario data is always stored on the server, simulation results may either be stored on the server or on the local machine. If the "Local" option is selected, the results will be stored in C:\Users*WinUsername*\AppData\Local\DassaultSystemes\PLM_LocalResult.

However, the data is masked, encrypted, and only accessible by the owner of the data. Students should check with their instructor to determine the proper location for storing their simulation results. For this case, we select the “Server” radio button.

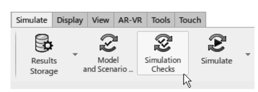


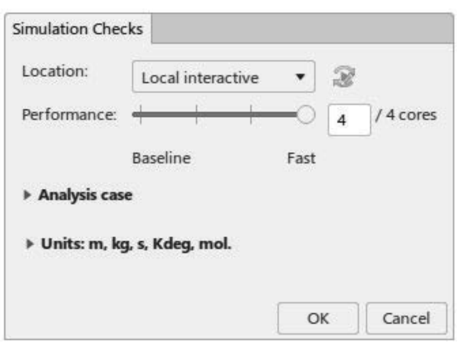




It is a good practice to perform the consistency and consistency check before submitting the work for the final run.
Select the “Model and Scenario Check” icon from the bottom row .
The software goes through a check phase and if there are no issues, a message with a “Green” check mark is returned.

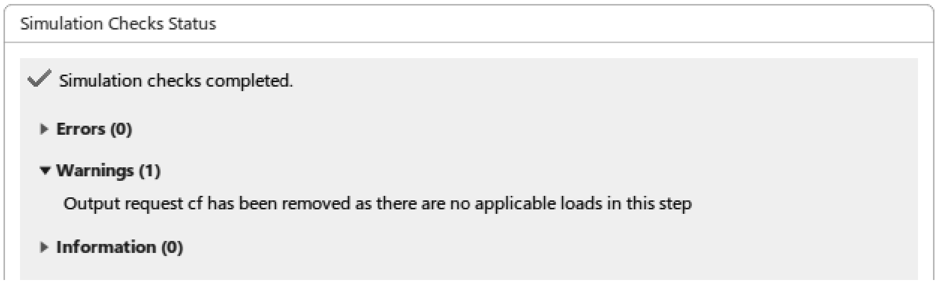



Next select the “Simulation Checks” icon  from the bottom row of icons. Accept the number of “Cores” in the pop up box below.



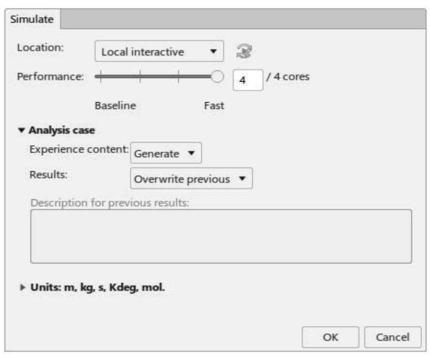



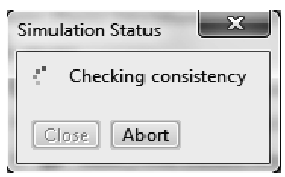
Upon the completion of the "Simulation Check", any errors or warning messages will be available in the pop up box below.




Assuming that there are no serious issues (i.e. no error messages), you are ready to submit the job for "Simulation".

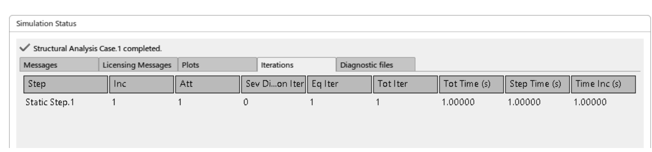
Select the "Simulation" icon  from the bottom row.
Accept the number of "cores" in the pop up box, and wait for the simulation to complete.



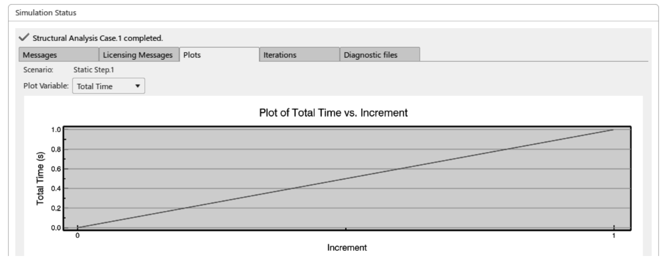




During this phase "Simulation Status", important messages such as "Licenses", "Plots", and "Iterations" are recorded in the main pop-up window. These can be viewed by selecting the appropriate tab.

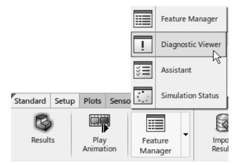

In the present run, if you select the "Iterations" tab you will see a single iteration as the problem in linear.



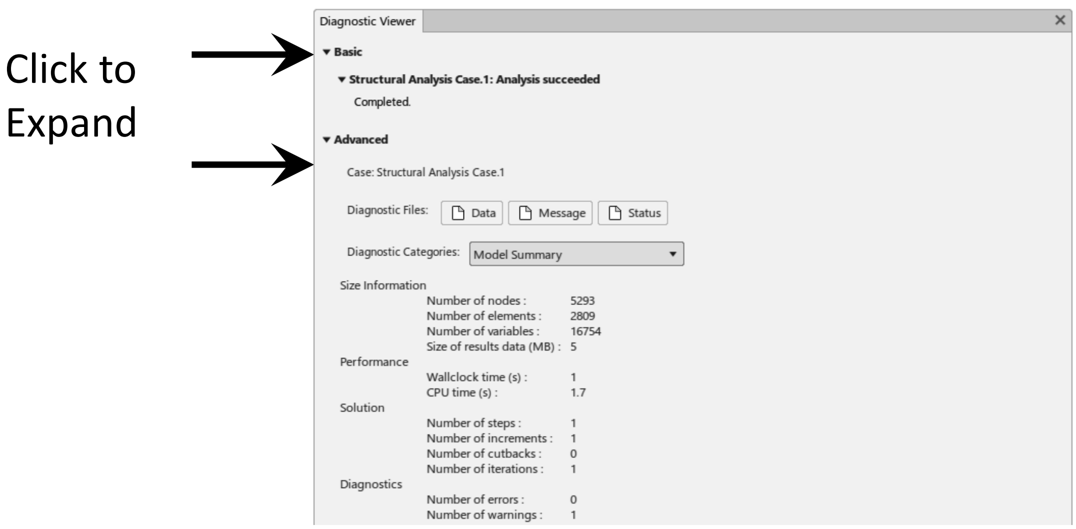
The “Plots” tab reveals nothing interesting. The problem being linear, there is only one iteration to get the solution.



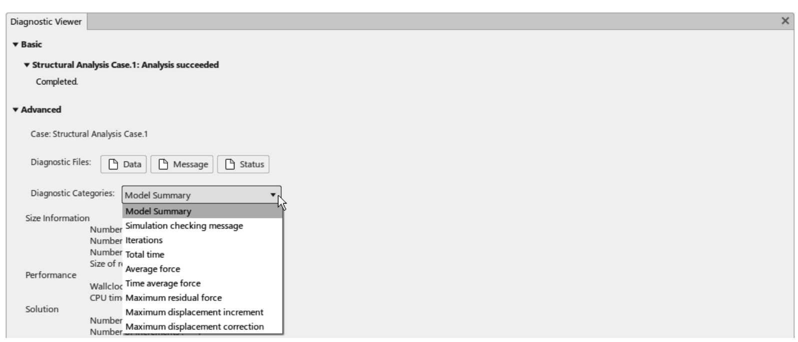

From the “Feature Manager” dropdown, select the “Diagnostic Viewer”  from the menu. This can also be launched by typing Alt+v.



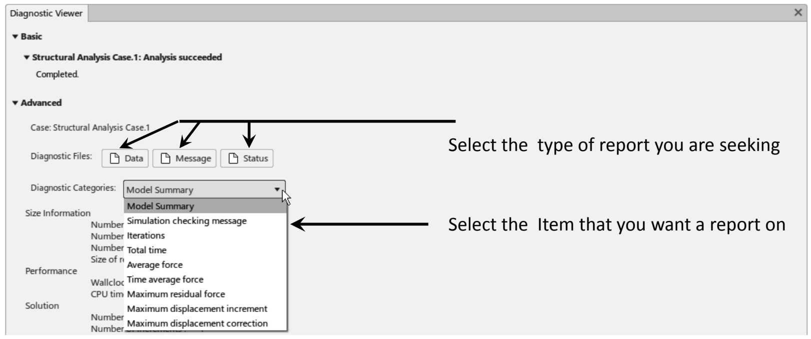

Once the window pops up, click on the arrows to expand the records.

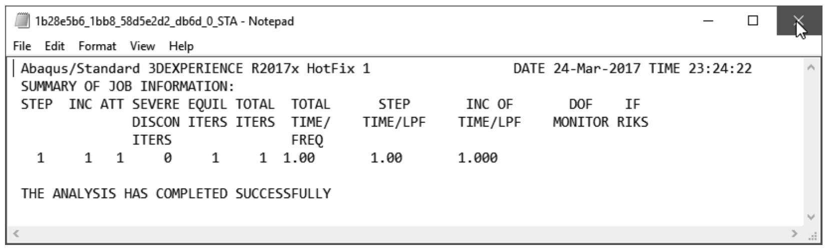



Under the “Advanced” section, use the pulldown menu to expand the choices. Here, you can select an item of interest to get information about.



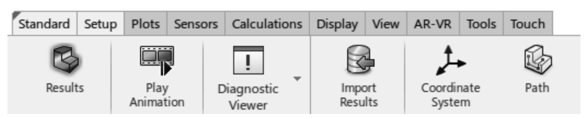

For example, if you are seeking information about “Iterations”, make that selection and click on “Status”. A text file (Notepad) is generated which pertains to the requested item.

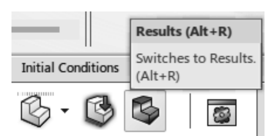


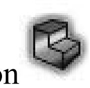




## Results (Post processing):

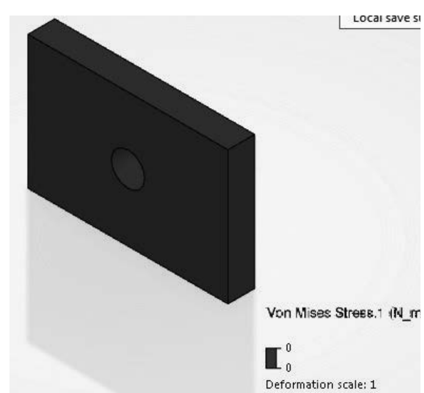
Once you close (or move) the obstructing dialogue boxes, you must be in the “Results” section and the bottom row should appear as shown on the right.



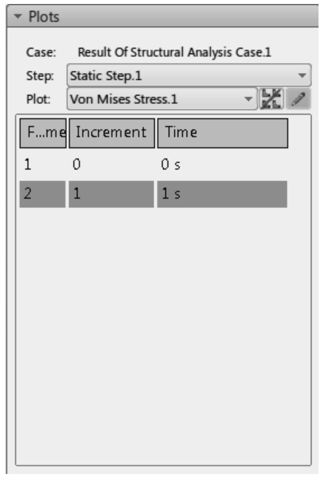

If not, click on the “Results” icon  .

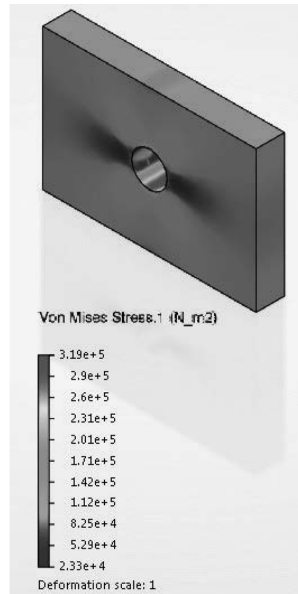
In the background, you should see the “Plots” dialogue box which shows the results of Frame1, and the initial results and the results after the first iteration. If the first of the “Plots” dialogue box is highlighted, the value of the von Mises stress is zero as shown on the right.

This is not surprising as the first row is before any incremental load is applied. If the load is zero, the displacement and stress are both zero. Use the cursor to select the second row of the "Plots" dialogue box. One can then see the von Mises stress distribution in the part.

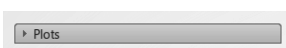
The stress concentration graph given on page 2-1 indicates a factor of 2.4 for the given dimensions. Since the magnitude of the applied pressure was 1.00E+5 Pa, a maximum von Mises stress of 3.19E+5 Pa is in reasonable agreement with theory. Keep in mind that this is a coarse grid.

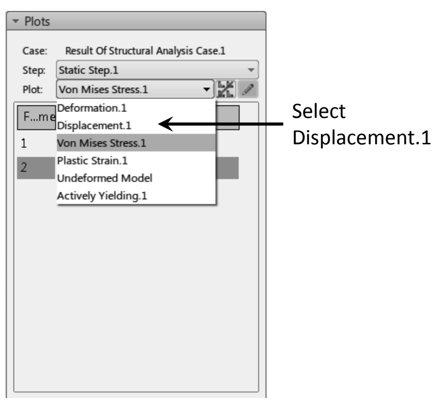


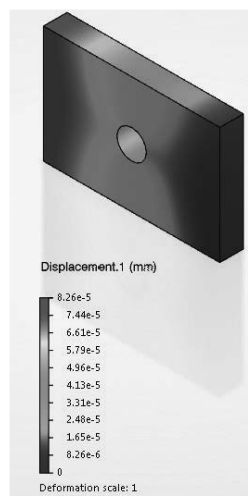




In order to display the "Displacement.1", use the "Plot" pulldown menu in this window and select "Displacement.1" as shown below.

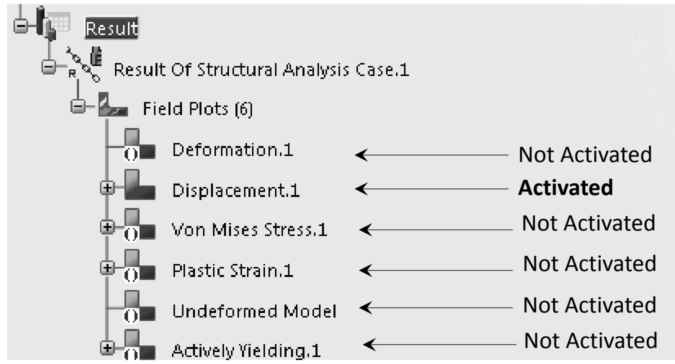
The "Plots" dialogue box can be closed to make room on the screen if needed. Use the cursor and select the arrow on the top left corner of the window; this will collapse the box as seen next.



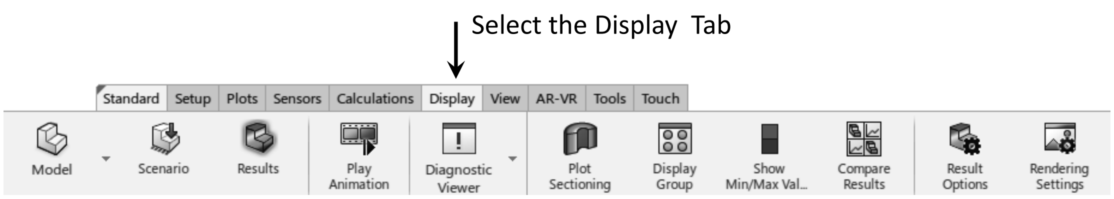



Every time a new plot is to be generated, the previous one is deactivated. Checking the status of the tree on the right indicates that the "Displacement" is "Active", whereas the other 5 (including von Mises) are "Not Active".



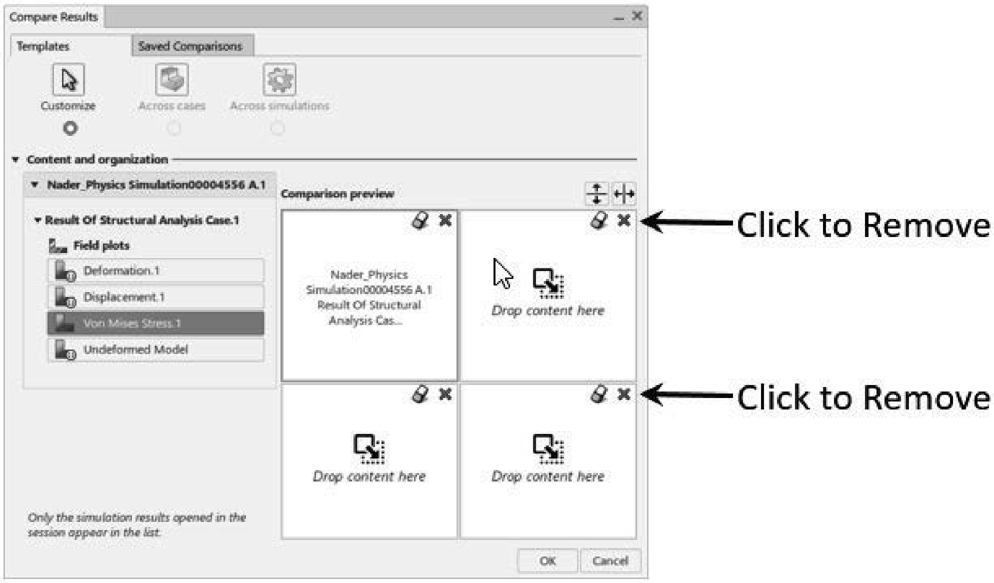

Plots which are "Not Active" have a marker  next to them. There may be good reasons that you want several plots on the screen side by side. Suppose that you want the contour plots of the displacement and the von Mises stress side by side.

Select the "Display" tab from the bottom row of icons.

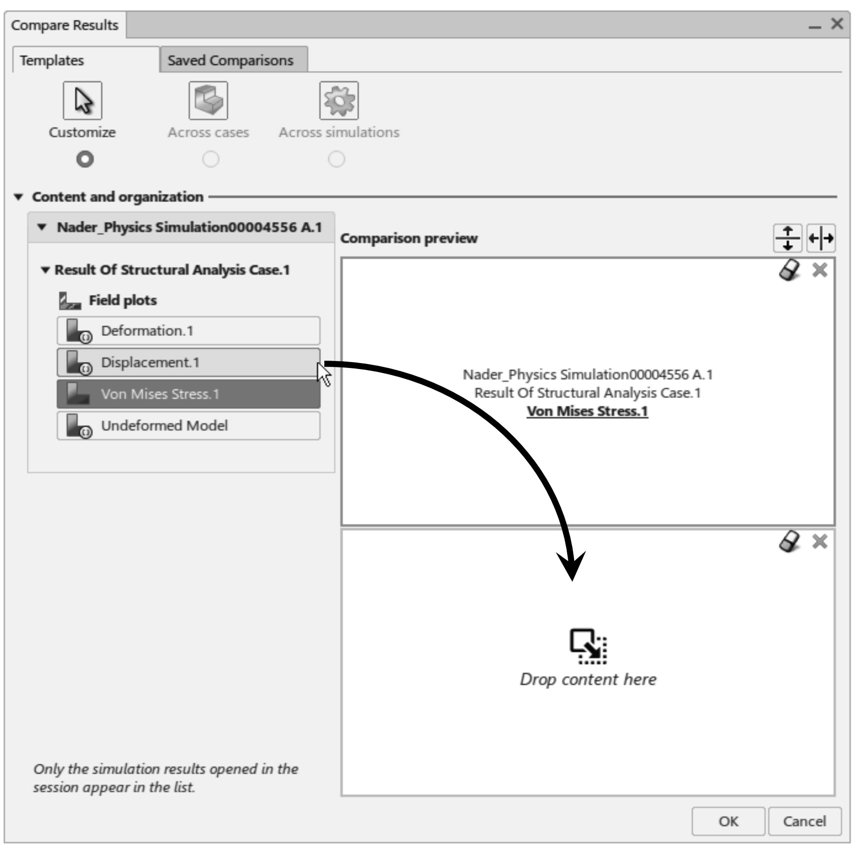



Select the "Compare Results" icon  from the bottom row.

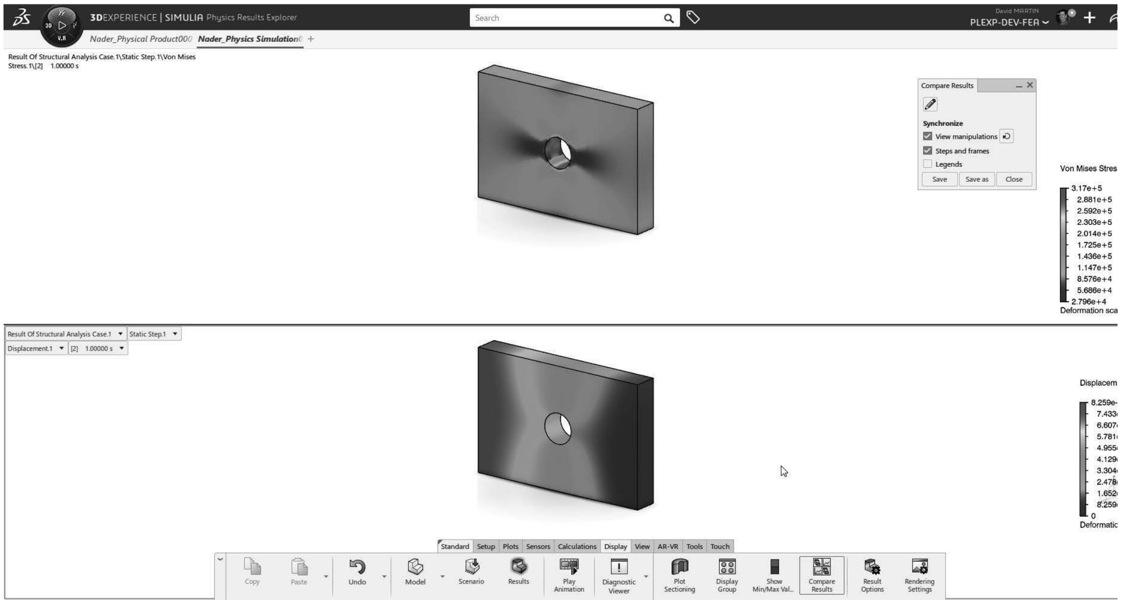
The "Compare Results" dialog will appear as shown below. Click on the X's to remove the two right panes.



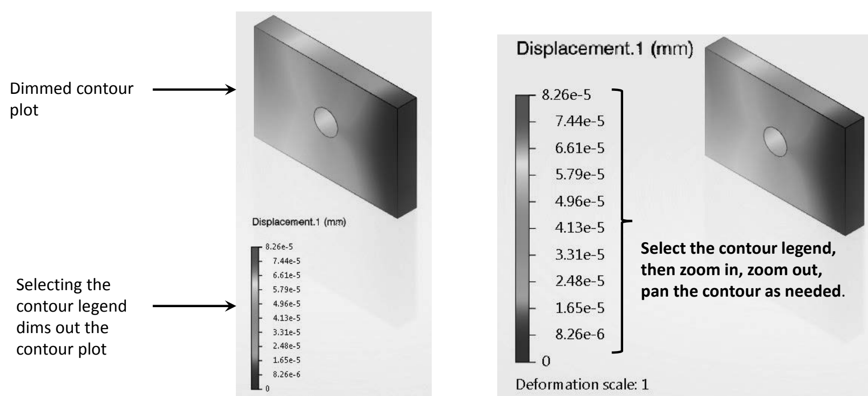

Then, drag and drop the displacement field plot into the remaining lower pane.

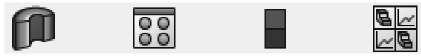


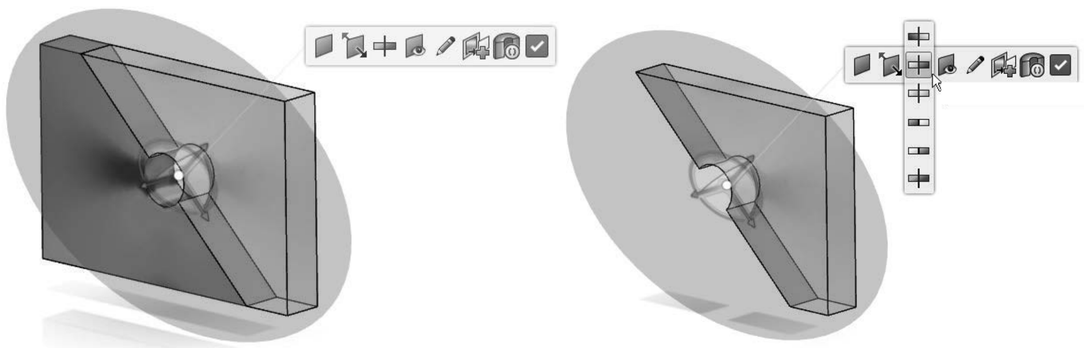

You will see that the von Mises Stress is displayed in the top sector of the screen and the displacement is displayed in the bottom sector of the screen as shown below.



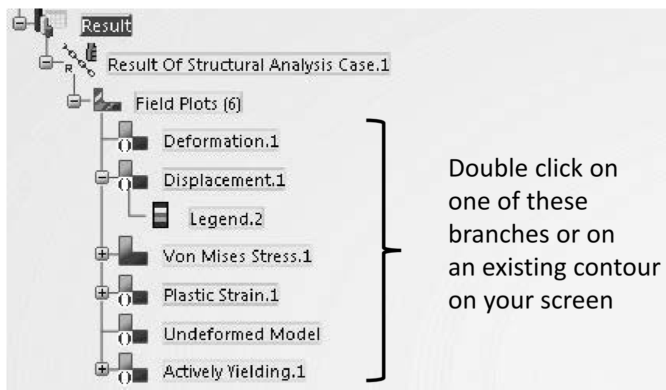

An important skill in post processing is the ability to pan, zoom in and zoom out of the contour legend. In order to explain the process, plot the contour of "Displacement.1". Point the cursor to the contour legend and select it (left click). You will see that the contour plot (on the block) becomes dim. This is an indication that the contour plot is "Not Active", whereas the contour legend is. Now, selecting the legend with the middle mouse button down, the legend can be panned. Selecting the contour legend, a single click of the middle mouse button, and forward/backward motion of the mouse enables you to zoom in and zoom out of the legend. Once you are done, select the contour legend again which will activate the contour plot.



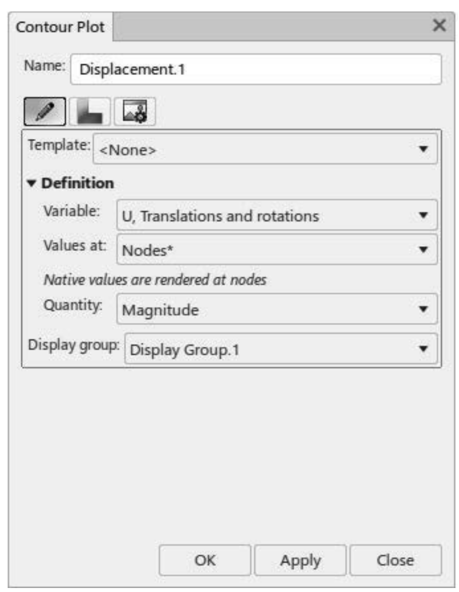
The group of icons  on the bottom of screen are very useful in post processing. The compare results icon  has already been discussed. We will now review the plot sectioning icon . Once this icon is selected, the part can be cut at an arbitrary orientation by rotating/translating the “Robot”; two random cuts are shown below.

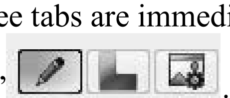


Manipulating the contour plots as far as formatting and setting the parameters is very important. To explore such features, double click on an existing contour on the screen (or simply double click on a branch of the tree which corresponds to a contour in the “Results” section).

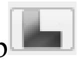


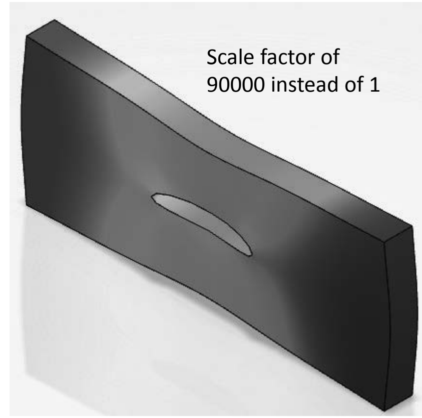

Once the contour appears on the screen, double click on it again. This action leads to the “Contour Plot” pop up window shown on the right. This window has three tabs and many pulldown menus which enables you to customize the plot.



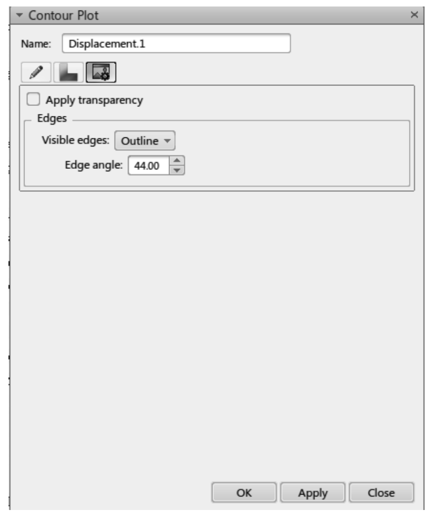

The three tabs are immediately below the “Name” .

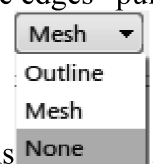
In the first tab, one can select the different variables to be plotted.

Selecting the middle tab  leads to the screen which enables you to change the "Scale factor". By default, the scale factor is "1". Change the "Scale factor" to "90000" and press the "Apply" button in the pop up window.



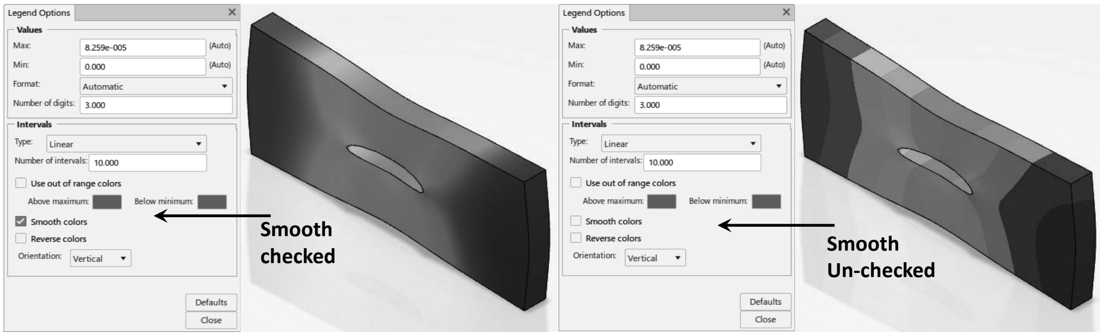

Next select the third tab  .




This tab has an important section on "Edges". The "Visible edges" pull down list gives you three options .

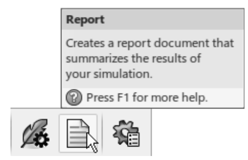

The effects are shown in the contour plots below.

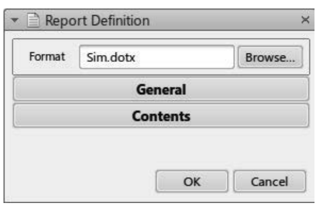

Looking at the above contour plots one notices that there are no distinct color separation borders. It could be that for certain reasons, having a smooth transition between colors is not desirable. This can easily be changed by double clicking on the contour legends which will open the "Legend Options" dialogue box. The box is shown on the next page and gives you many other options regarding the contour.




The number of colors, the maximum and minimum range for the variable can also be set in this window. Note that the setting which allows you to orient the legend “Horizontal” or “Vertical” is at the bottom of the window.

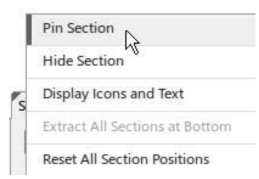
Select the “Create Report” icon  from the bottom of the screen. This will lead to the dialogue box shown below.



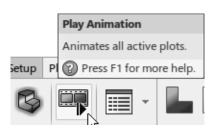




A professional looking report in “Word” or “PowerPoint” formats can be generated. Personalized information such as company/university logo can be automatically inserted.

The results animation tool is located on the “Standard” tab. It is good practice to pin the “Standard” tab to the Action Bar. To do this, right click on the tab and select “Pin Section” as shown on the right.



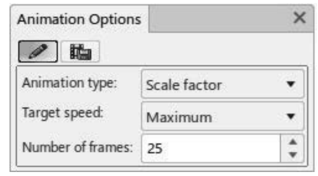

Select the “Play Animation” icon  from the bottom row.



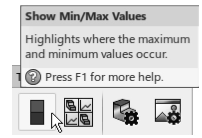

This loads the animation player as shown below.

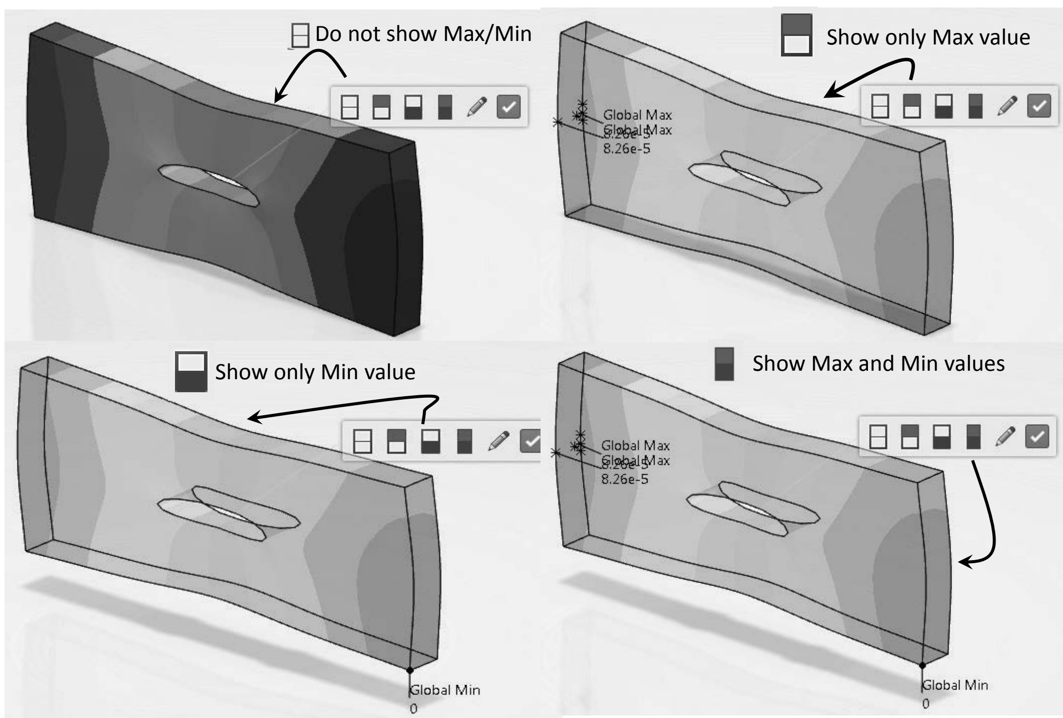
You can change the animation properties and save the animation by clicking on the “Animation Options” icon .

This loads the “Animation Options” dialog as shown on the right. Finally, looping control is achieved by clicking on the icon. You can click through the options: loop, bounce, and one shot.



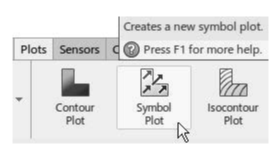

A very useful post-processing feature is the “Show Min/Max Values” icon from the bottom row of icons.



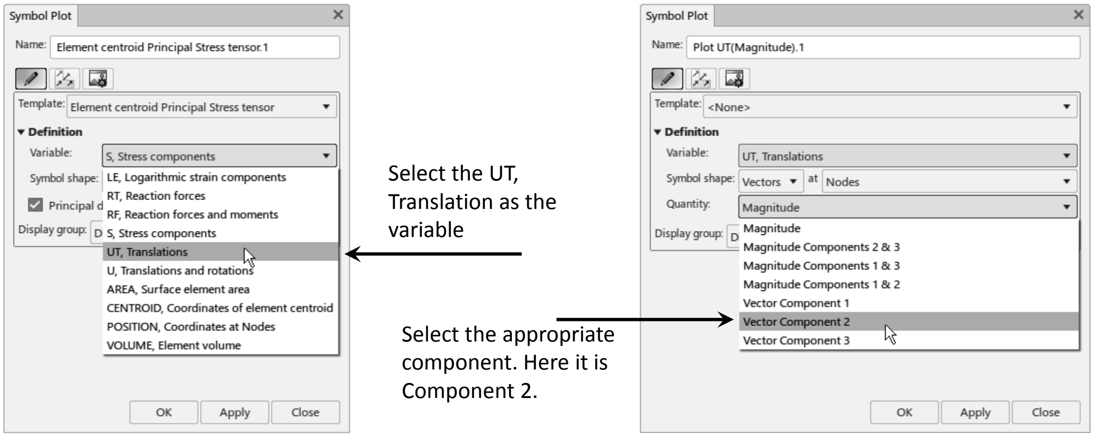




Up until this point, the post processing dealt with contour plot fully rendered. The next activity deals with generating plots which are “symbol” based; for example, the y-component of the displacement vector (or symbol) but plotted as a vector. In this example, y is the direction where the part is loaded.

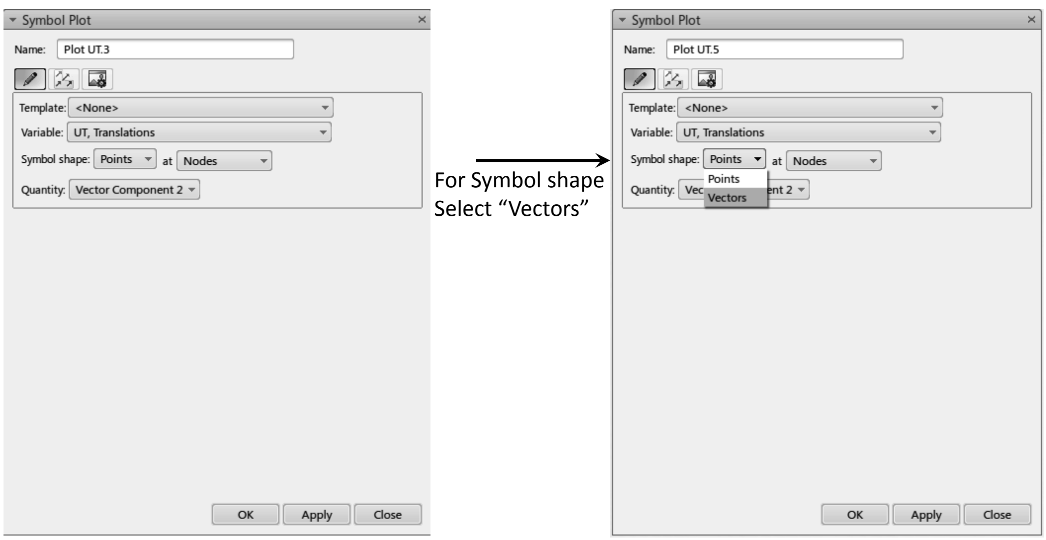
Select the “Symbol Plot” icon from the “Plots” tab at the bottom of the screen. Note that the icon “Isocontour Plot” is essentially the same as “Create Contour Plot”, except that there is no rendering.



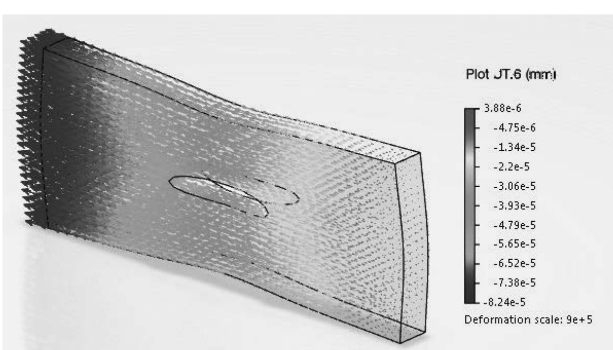
Follow the instructions below to select "UT, Translation" with "Vector Component 2".

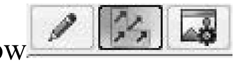


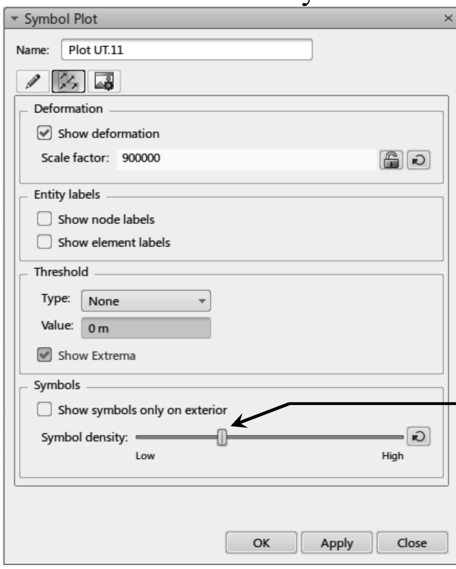

For Symbol shape, select "Vectors"

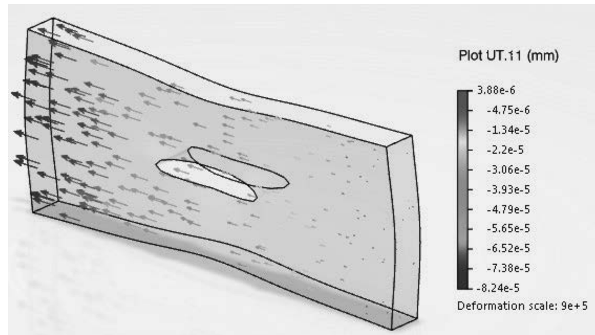



Finally press "Apply". The symbol plot (vector plot) shown below appears. The issue with this plot is that there are too many symbols (arrows).



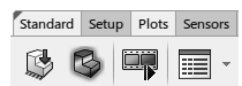
Follow the instructions given below in the middle tab of this window  to reduce the arrow density.

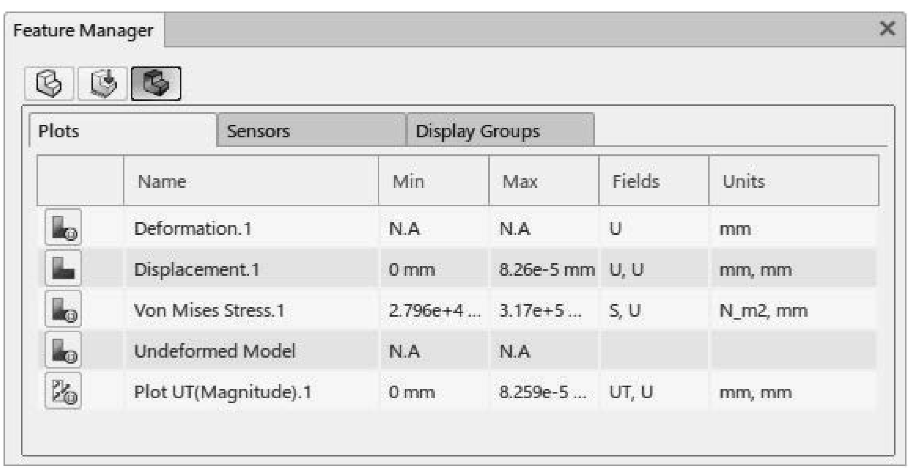



Slide the bar to reduce symbol (vector) density
When satisfied, press "Apply"

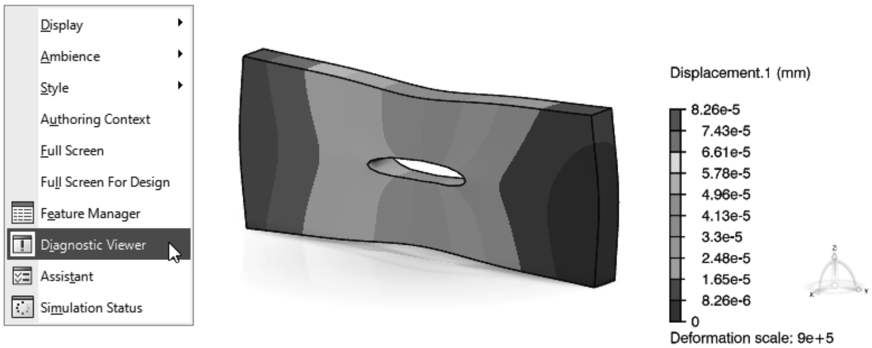
The final activity is to explore the "Feature Manager"  in the context of the simulation results. This selection leads to the "Feature Manager" dialogue box below which contains valuable information.




| | Name | Min | Max | Fields | Units |
|---|---|---|---|---|---|
| | Deformation.1 | N.A | N.A | U | mm |
| | Displacement.1 | 0 mm | 8.26e-5 mm | U, U | mm, mm |
| | Von Mises Stress.1 | 2.796e+4 ... | 3.17e+5 ... | S, U | N_m2, mm |
| | Undeformed Model | N.A | N.A | | |
| | Plot UT(Magnitude).1 | 0 mm | 8.259e-5 ... | UT, U | mm, mm |

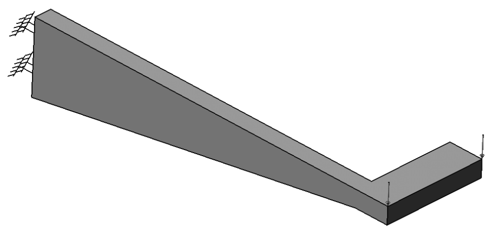
It is worth mentioning that dialogue boxes such as "Feature Manager" and the "Diagnostic Viewer"  can be accessed any time by right clicking on the screen as shown below.

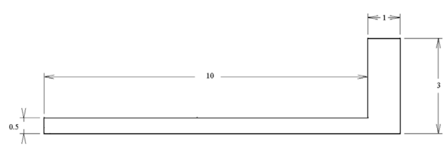


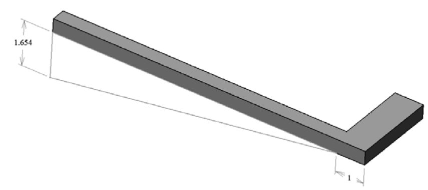

There are many, many other important features that were not touched upon in this chapter but will be explored in later ones.

## Exercise 1: Analysis of a Foot Pedal

The foot pedal shown below is made of steel with Young's modulus 30E+6 psi and Poisson ratio 0.3. The pedal is loaded with a normal force of 100 lb along the edge shown. The other end of the pedal is clamped. The geometrical dimensions are provided at the bottom of the page where all the dimensions are in inches.






Try running the model with the two different element "size" and "sag" with both the linear and parabolic type of elements. Record the results in terms of the maximum displacement and the maximum von Mises stress in a table and comment on the results. The run time of the parabolic elements with element size of 0.1 could be substantial depending on the type of processor used.

Partial Answer:

| **Size = .3, sag = .05** | | |
|---|---|---|
| Element Type | Linear | Parabolic |
| Maximum Displacement | .0158 in. | .0227 in. |
| Maximum von Mises Stress | 5.32E+3 psi | 1.1E+4 psi |

| Size = .1, sag = .05 | | |
|---|---|---|
| Element Type | Linear | Parabolic |
| Maximum Displacement | .021 in. | .0229 in. |
| Maximum von Mises Stress | 9.43E+4 psi | 1.6E+4 psi |

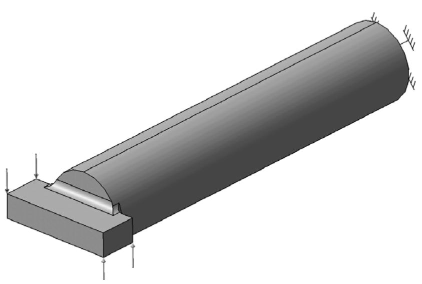
The above tables reveal an extremely important fact about finite element analysis. Making a single run and accepting the results at face value is a serious mistake. Note that for linear elements as the mesh is refined, there is a significant change in both displacement and von Mises stress. The user should not accept either value as being correct and must refine the mesh further. The refinement should reach a point at which the difference with the previous mesh is not deemed to be significant to the user. This process is referred to as a mesh convergence study.

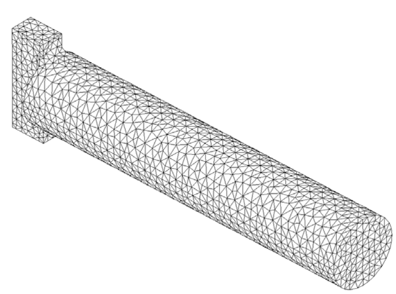
Keep in mind that the refinement need not be uniform throughout the part. One should perform the refinement in the critical areas only. It is clear that parabolic elements are superior in accuracy to linear elements. Furthermore, note that although the displacement seems to have stabilized, the von Mises is still unreliable. It is well known that the displacements in FEA are more accurate than stresses. The reason is that the stresses are obtained by differentiating the displacement, a process which magnifies the error.

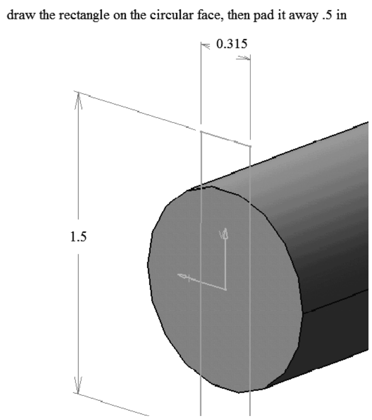
**Exercise 2: Analysis of a Cylindrical Bar under Torsion**

The cylindrical bar shown below has a clamped end. The other end is subjected to a couple caused by opposite forces on magnitude 1000 lbf separated by 1.5 in. This is equivalent to a torque of 1500 lbf.in applied to the cylinder. The material is steel with Young's modulus 30E+6 and Poisson ratio of 0.3.

The diameter of the cylinder is 1 in. and the dimensions of the loaded end are shown below. Although not showing, the length of the padded cylinder is 5 in. and the length of the padded rectangle is 0.5 in. All sharp corners at the loaded end have surface fillet of radius 0.1 in.








Model the part with linear solid elements with size = 0.1 and sag = 0.025 which results in the mesh shown in the previous page. Compare the hoop stress (The hoop stress is the largest principal stress "C11") with the theoretical solution from strength of materials.

Partial Answer:

The strength of materials solution is based on $\tau = \frac{Tr}{J}$ where T is the applied torque, r is the radius of the cylinder and J is the polar moment of inertia. In terms of the diameter, $r = \frac{D}{2}$, and $J = \frac{\pi D^4}{32}$. The hoop stress "C1" which numerically equals $\tau$ is calculated from $\frac{16T}{\pi D^3}$. For the present problem, $T = 1500$ lb.in and $D = 1$ in. Based on these parameters, a value of 7643 psi for the hoop stress is predicted.

**Notes:**