Nader G. Zamani

FINITE ELEMENT ESSENTIALS IN
3DEXPERIENCE® 2017x
USING SIMULIA®/CATIA® APPLICATIONS

Visit the following websites to learn more about this book:

SDC Publications · amazon.com · Google books · BARNES&NOBLE

# 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.
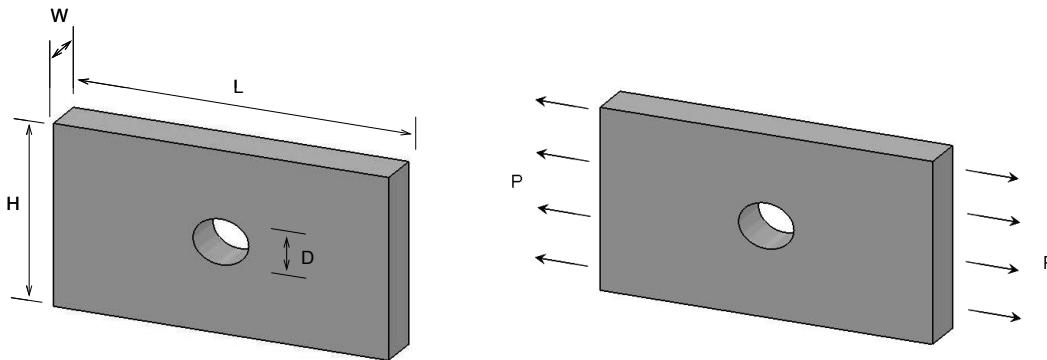
The dimensions of the part to be analyzed are such that the parameter $r = \dfrac{D}{H}$ is 0.25. The chart below gives a stress concentration factor of $K_t = 2.4$. This chart is based on $L \gg H$ which is not true in the modeled block. However, the value of K_t will be used for comparative purposes.
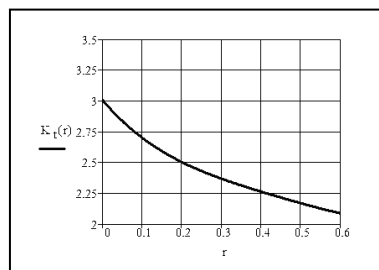
There are many types of solid elements available in the software; however, in this chapter, tetrahedron elements are used. They come in linear and parabolic forms. Both are referred to as tetrahedron elements as shown below.

Tetrahedron Elements
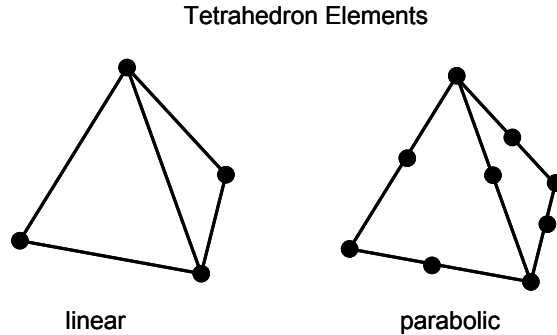
linear   parabolic

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, σ_{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 σ_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 considered to be 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 ![Part Design icon], 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.
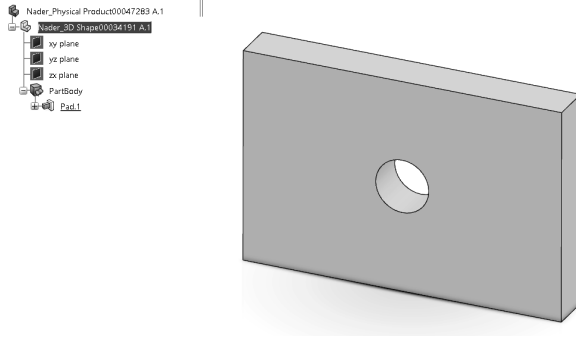
The first task is to apply a material property to this part. From the bottom row of icons (i.e., the action bar), select the "Tools" tab.

Select the "Tool" tab

From the menu, select "Material Browser" icon ![icon]. This opens up the section menu as shown. Follow the steps outlined below to select the "Create Material" ![icon].

| Step 1 | Step 2 | Step 3 | Step 4 |
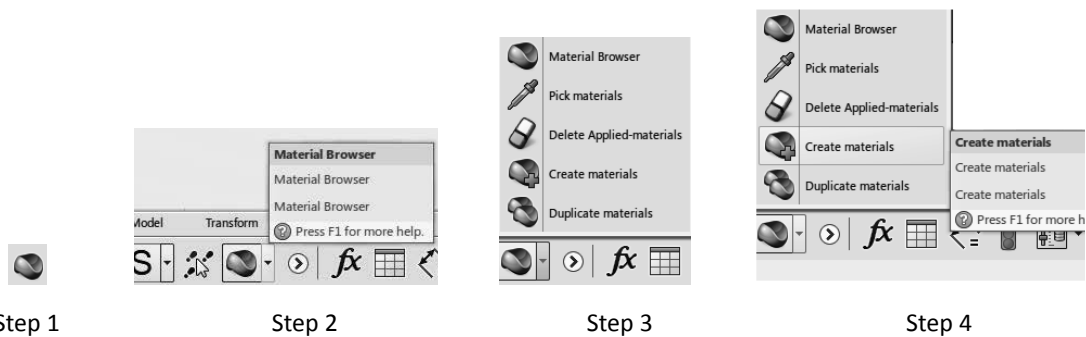
The selection of ![icon] opens up a dialogue box shown on the next page. This box allows you to supply a proper name for the material should you decide to do so. Our assumption is that you do not have a material of interest in the "cloud" database and would like to follow the steps to create it. It is a rather tedious process but will be clearly spelled out.

Select the "Create Material" [icon] icon. Make sure that you check the "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.


Type your desired name

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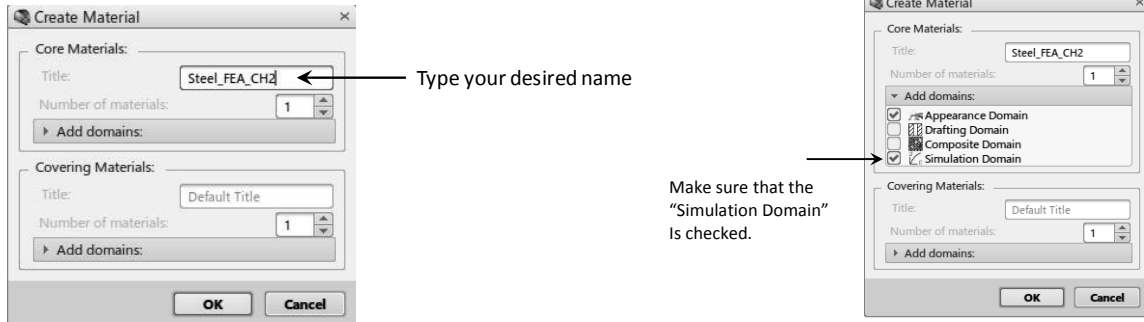
Once you close the dialogue box by clicking on "OK," you will find yourself in the material database and can identify the material that you just created, namely "Steel_FEA_CH2." The database screen is shown on the right.

Here is the material property that you created. This is just a "shell"; information needs to be inputted later.
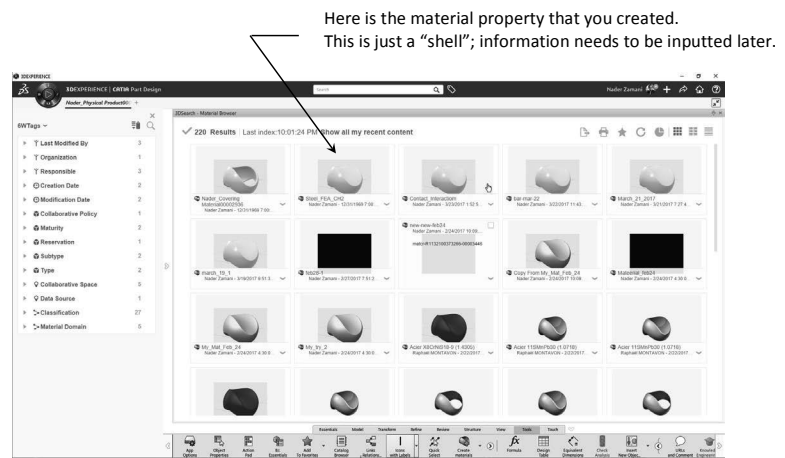
Double click on this "shell" and wait for the window to rearrange.

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).

Position the cursor here on the screen, right click, select "Apply"
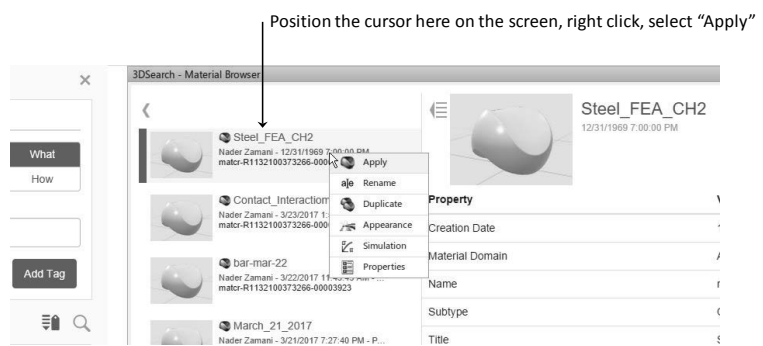
Select the "X" on the top right margin of the database screen to close the window.

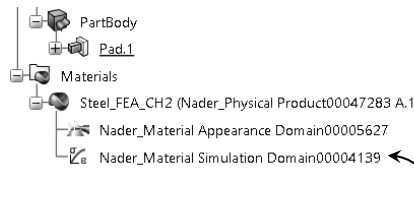You will return to the geometry window; however, the shape of the cursor is modified as shown below.

Steel_FEA_CH2
12/31/1969 7:00:00 PM

Property	Value
Creation Date	12/31/1969 7:00:00 PM
Material Domain	Appearance Domain	Simulation Domain
Name	matcr-R1132100373266-00004003
Subtype	Core Material
Title	Steel_FEA_CH2
Modification Date	12/31/1969 7:00:00 PM
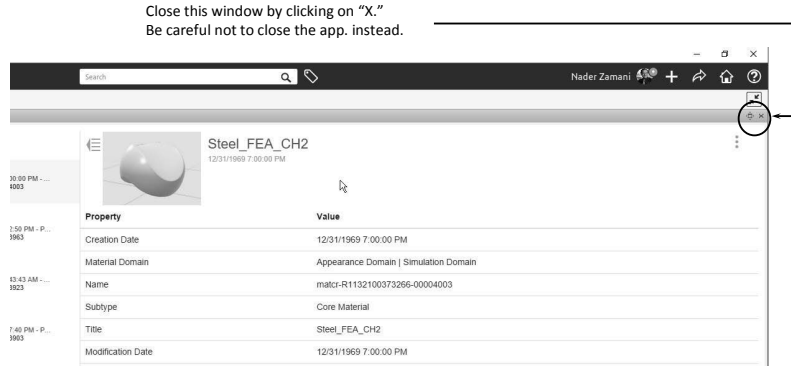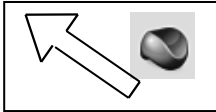
Place the cursor on the part on the screen or on the top branch of the tree and double click.

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.

Select the "Green" check mark from this box
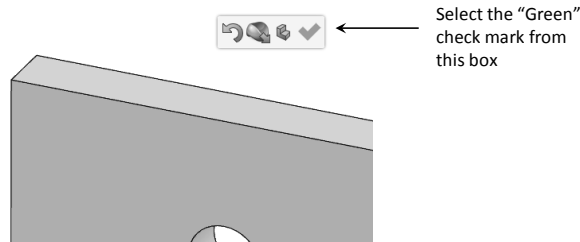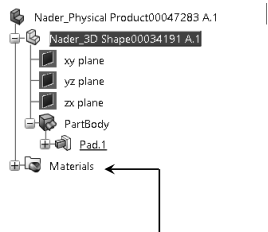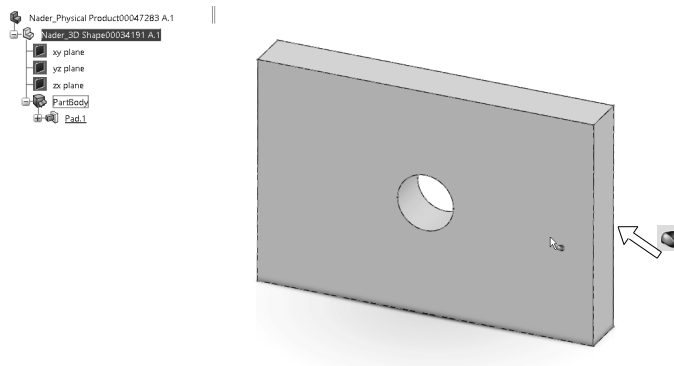
The created material has been assigned, but this is just a "shell"; information needs to be inputted later.

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 "Nader_Material Simulation Domain00004139" as shown on the right.

PartBody
Pad.1
Materials
Steel_FEA_CH2 (Nader_Physical Product00047283 A.1)
Nader_Material Appearance Domain00005627
Nader_Material Simulation Domain00004139

Input material properties by double clicking on this branch

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

Step 1

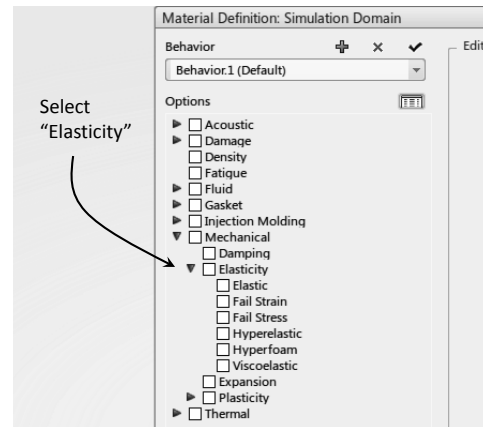Select "Mechanical"

Step 2

Input the Young's modulus and Poisson ratio

Select "Elastic"

Step 4, final step


Select "Elasticity"
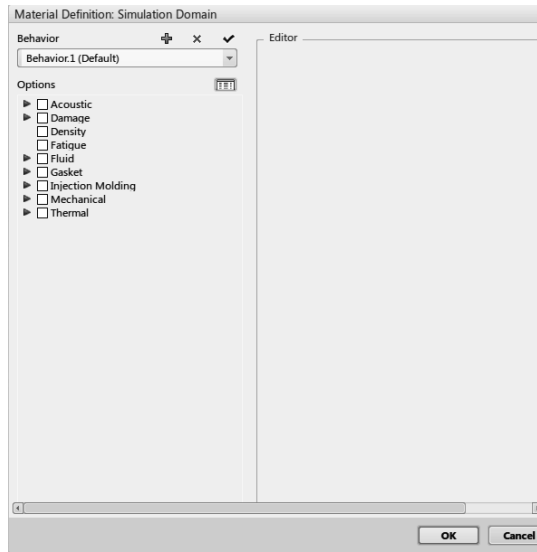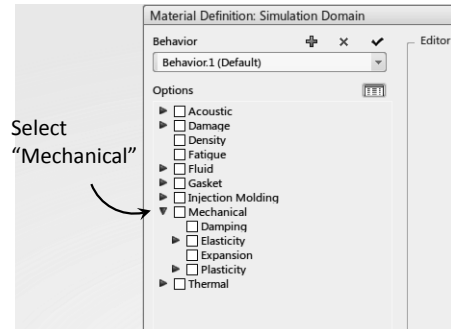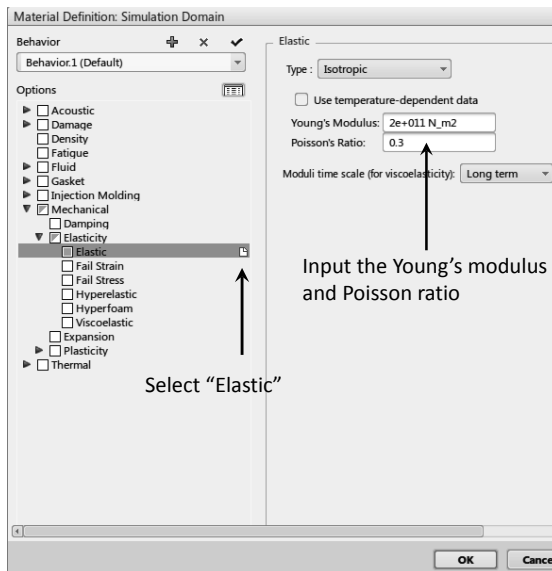
Step 3

In Step 4, 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 the Finite Element Representation (FemRep):

Locate the compass on the top left corner of the screen, and select the South sector (i.e., V+R sector) as shown on the right. Scroll through the applications and select the Structural Model App.
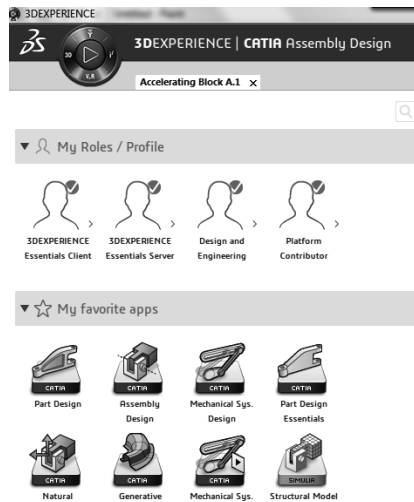
The row of icons on the bottom of your screen (action bar) changes and will appear as displayed below.
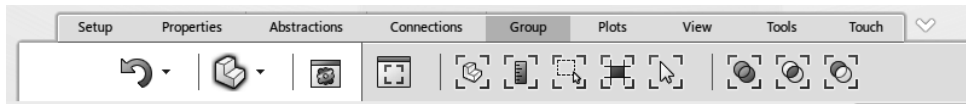
The dialogue box shown on the right, "FEM Controls," appears. For now, use the "Automatic" radio button. Other options are for user control of the meshing process. In the case of "Automatic," tetrahedral parabolic elements are created.
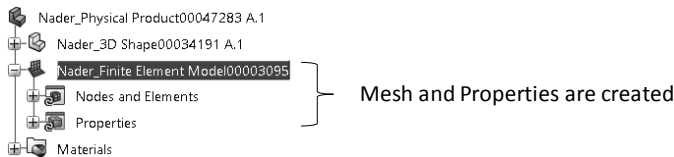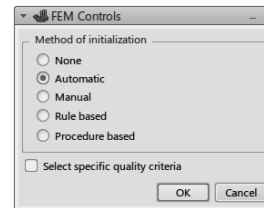
The tree indicates that a mesh and the corresponding solid section has been created.

Mesh and Properties are created

Expanding the branch "Nader_Finite Element Model0003095" further indicates that the elements are of "Tetrahedron" type and the property is "Solid Section" as expected.
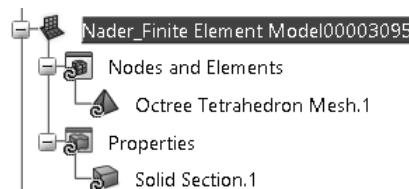
Upon double clicking on the "Octree Tetrahedron Mesh.1" branch, the corresponding dialogue box shown on the right pops up. Here, one can change the type (linear or quadratic), the size and sag, and certain other parameters. For example, local mesh refinement can be accomplished through the "Local Specifications." Use the default settings.
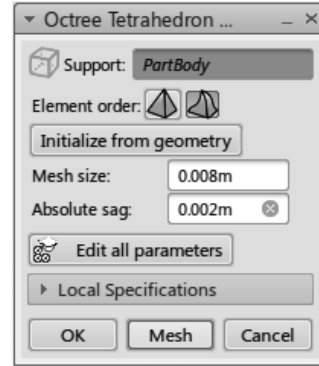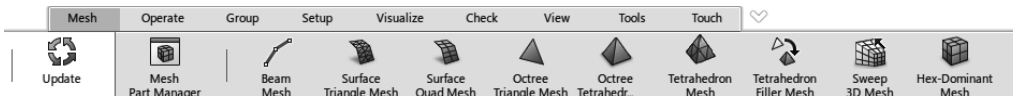
There are different methods for displaying the mesh. The instructions given below are one method of achieving that objective.
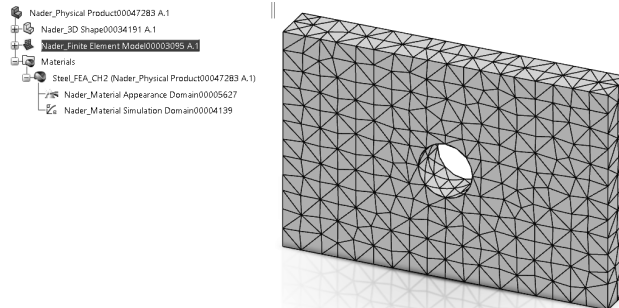
From the bottom row of icons, select the "Mesh" icon .
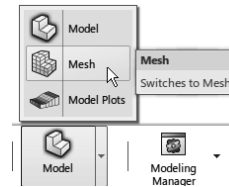
The bottom row's appearance now looks as shown below.

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 "Visualize" tab from the bottom row of icons.
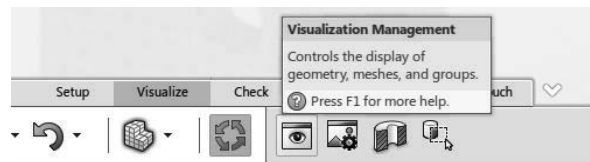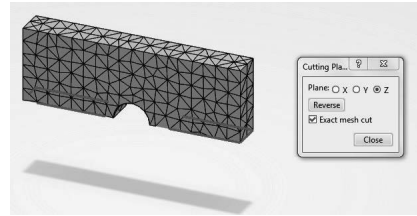
You can select the "Visualization Management" icon from the choices. The resulting dialogue box shown on the right appears.
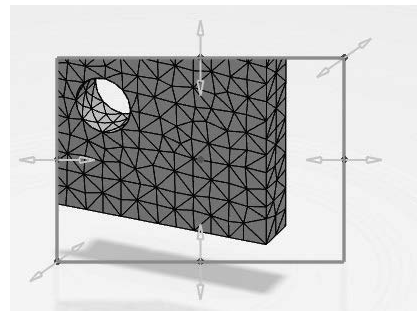
Then click on . This hides the mesh.
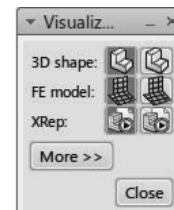
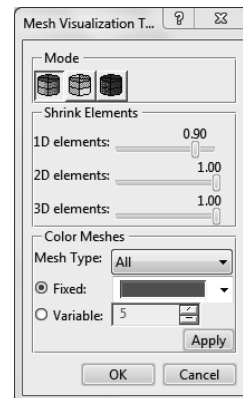There are a few other icons that are worth mentioning here.

Selecting the "Cutting Plane" 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.

The "Mesh Visualization" icon allows you to shrink elements and perform other operations that are not useful for the present problem.

Creating a Scenario:

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

It is also important to point out that one could have created the Scenario before the FemRep. In fact, FemRep can be created from within the Structural Scenario App.

Locate the compass on the top left corner of the screen, and select the South sector (i.e., V+R sector) as shown on the right. Scroll through the applications and select the Structural Scenario

App
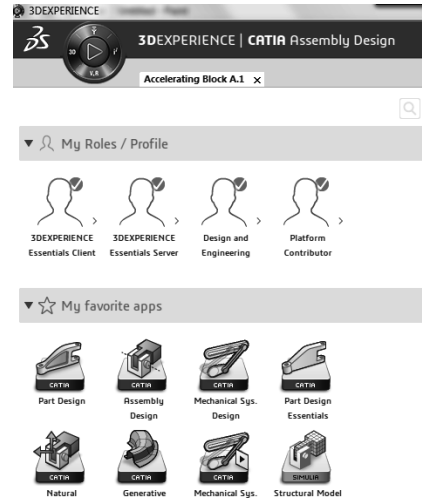
The row of icons on the bottom of your screen changes and will appear as displayed below.
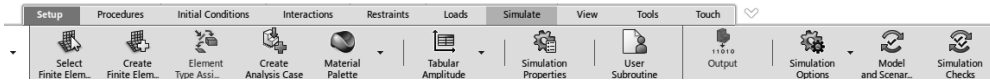
The pop-up window "Simulation Type" shown below appears on the screen. Since this is strictly a structural problem, the radio button "Structural" should be selected.
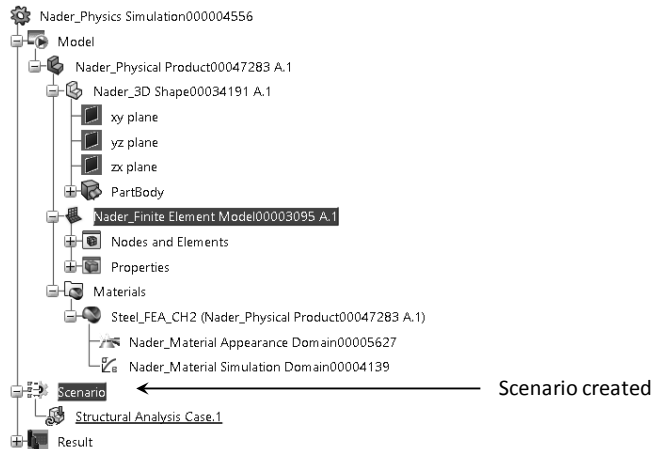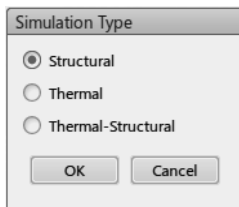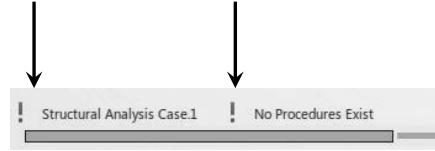
A quick glance at the tree confirms that a "Scenario" has been created.

Checking the bottom of the screen on the left side reveals that there are two red exclamation signs.
These pertain to "Structural Analysis Case.1" and "No Procedure Exist."
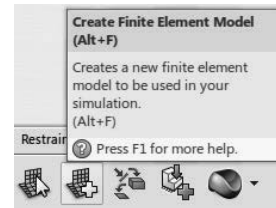
Red exclamation signs

Click the "Select the Finite Element Model" icon from the bottom row. This is referring to the FemRep which is already created. The following pop-up window appears.
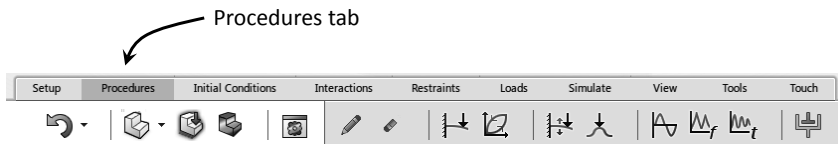
Select Finite Element Model

Selects the finite element model that will be used in your simulation.

Press F1 for more help.

Since there is already a finite element model created, it appears in the list; make sure that you select that row.
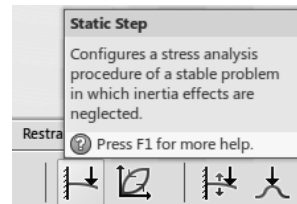
Select this row
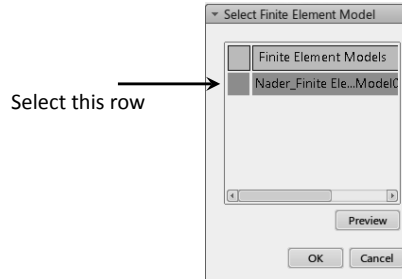
Note that there is another icon , namely the "Create Finite Element Model" from the bottom row. As was mentioned earlier, the FemReps could have also been created in Scenario, and this is the icon to achieve that task. It does not apply to you as a FemRep was already created and selected above.

Create Finite Element Model (Alt+F)

Creates a new finite element model to be used in your simulation. (Alt+F)

Press F1 for more help.

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

Procedures tab

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

Static Step

Configures a stress analysis procedure of a stable problem in which inertia effects are neglected.

Press F1 for more help.

The "Static Step" dialog 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.
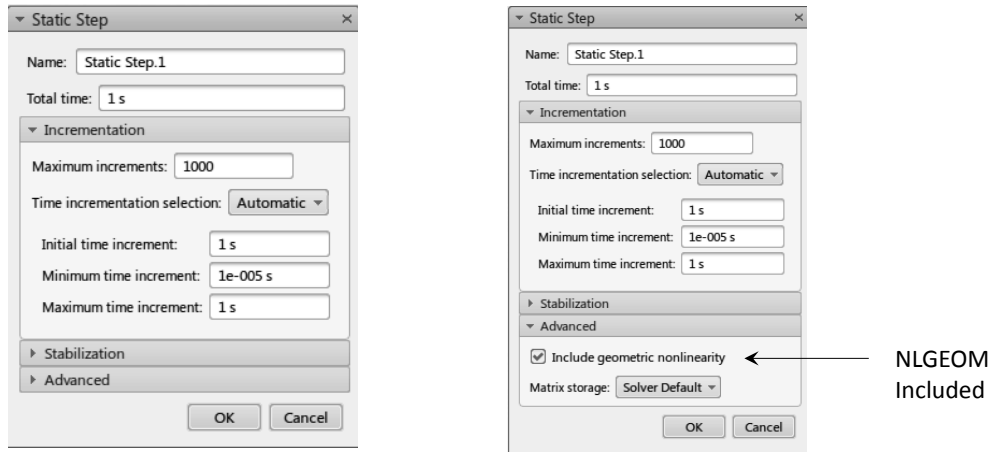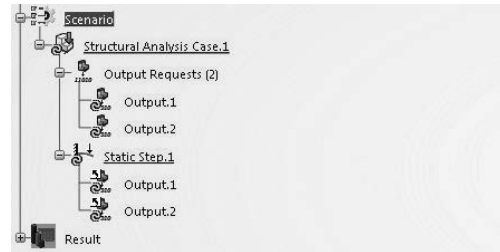

A quick glance at the bottom left bottom corner of the screen reveals "Green" checkmarks instead of "Red" red exclamation marks.

Note the "Green" checkmarks instead of "Red" exclamations

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 select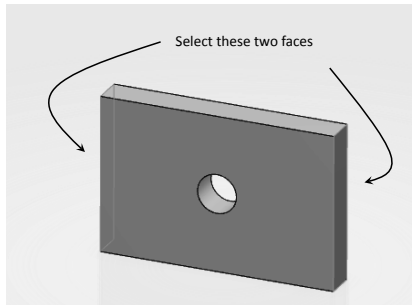ing the appropriate icon among these three: . The first one on the left is "Model and Mesh," the middle one is "Scenario" and the one on the far right is "Results"—that is the post processor.
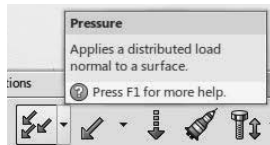
Applying the Pressure Load:

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 "Loads" tab

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

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

Select these two faces

Pressure

Applies a distributed load normal to a surface.

Press F1 for more help.

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 1/8th of the geometry only.

However, 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.
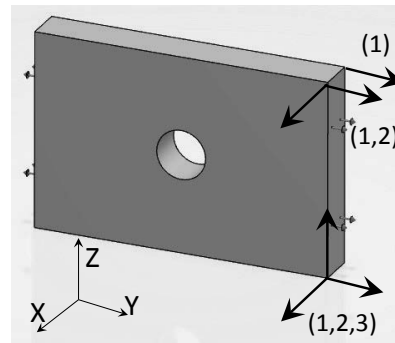

The (1,2,3) restraint representing (x,y,z) prevents the block from flying away. The (1,2) restraint representing (x,y) prevents rotation about x and y axis. Finally, the (3) restraint representing (z) eliminates the rotation about the z axis.

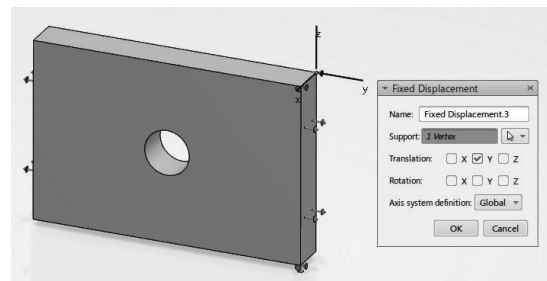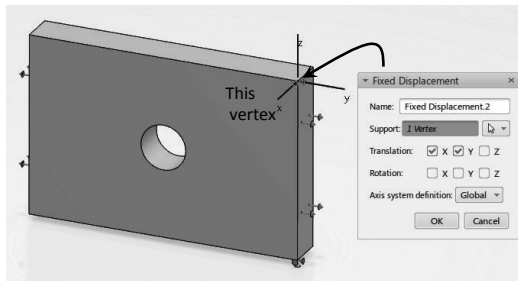Select the "Restraints" tab from the bottom toolbar (the action bar).

Restraints tab
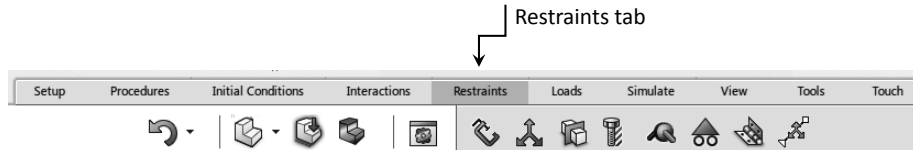
Choose the "Fixed Displacement" icon followed by the bottom vertex as shown.
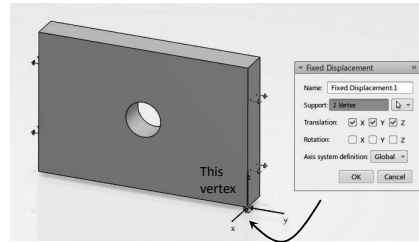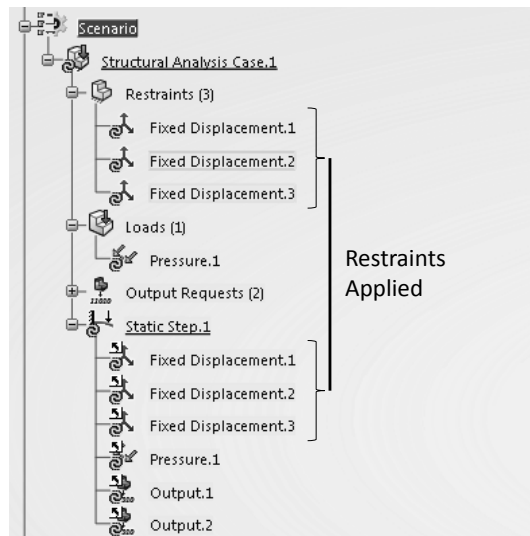
In the dialogue box, check all three "Translations."

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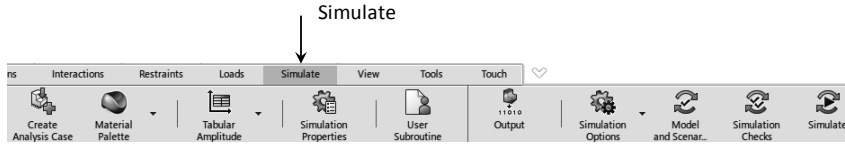

As a side comment, instead of using the "Fixed Displacement" icon and checking all three x,y,z translations, one could have used the "Clamp" icon .

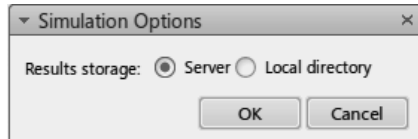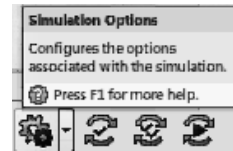Note that the imposed restraints are now reflected in the tree.

Consistency, Model Check, and Simulation:

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

Simulate

Select the "Simulation Option" icon from the bottom row.

This will give you the option of using the server or the local machine. Use the "Server" radio button.
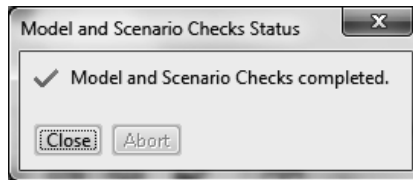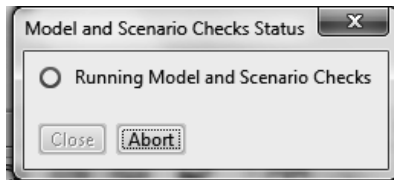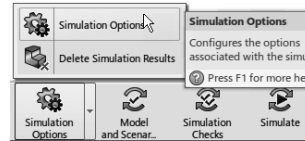

It is a good practice to perform the consistency and consistency check before submitting the work for the final run.
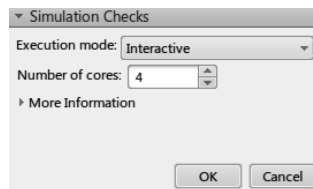Select the "Model and Scenario Check" icon from the bottom row .
The software goes through a check phase and if there are no issues, a message with a "Green" check mark is returned.

Next select the "Simulation Checks" icon from the bottom row of icons. Accept the number of "Cores" in the pop-up box below.
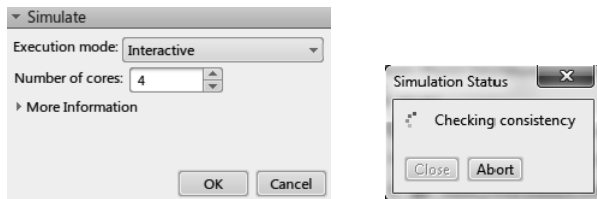
Upon the completion of the "Simulation Check," any warning messages will be available in the pop-up box below.


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.
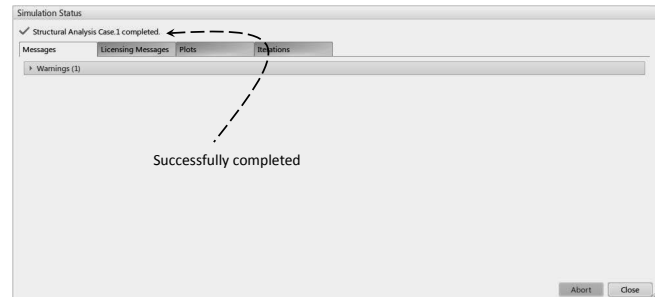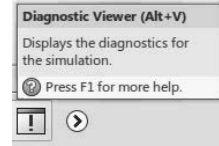

In the present run, if you select the "Iterations" tab you will see a single iteration as the problem in linear.
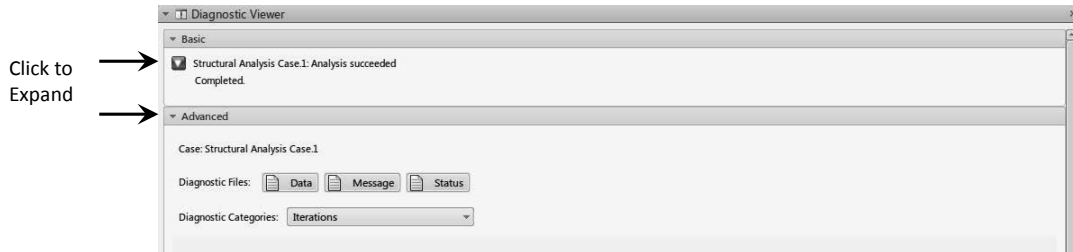

The "Plots" tab reveals nothing interesting. The problem being linear, there is only one iteration to get the solution and therefore no iteration plot generated.
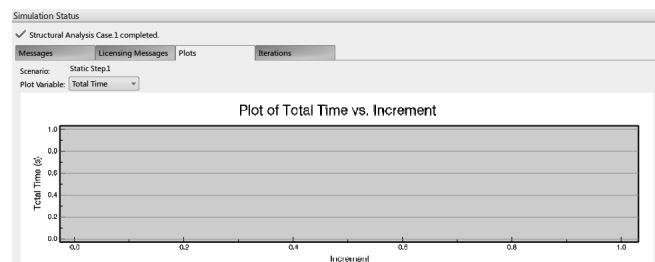
Right click and select the "Diagnostic Viewer" icon from the menu.



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.
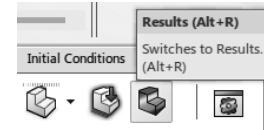

For example, if you are seeking information about "Iterations," make that selection and click on "Status." A text file (Notepad) is generated which pertains to the requested item.
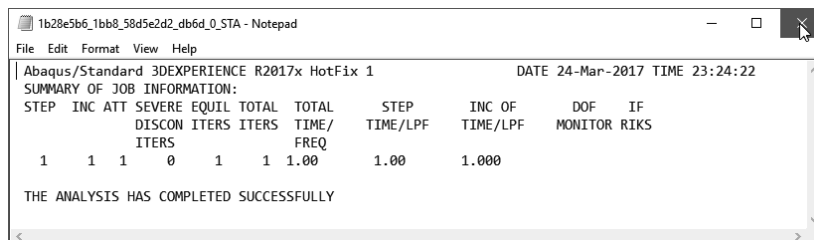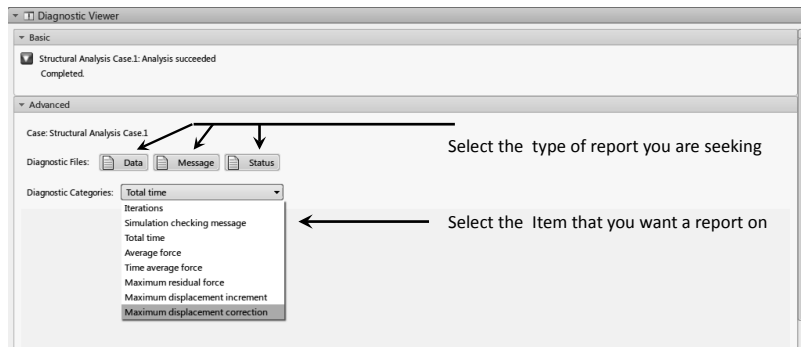
Results (Post processing):

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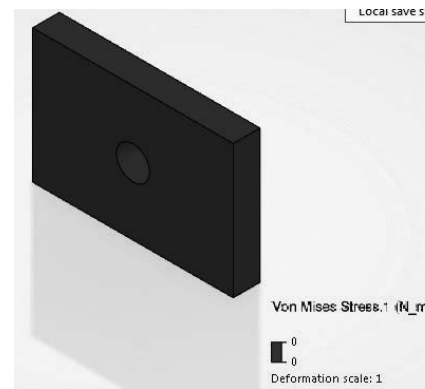
This is not surprising as the first row is before any incremental load is applied. If the load is zero, the displacement and stress are both zero. Use the cursor to select the second row of the "Plots" dialogue box. One can then see the von Mises stress distribution in the part.

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 Moses 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 on the next page.
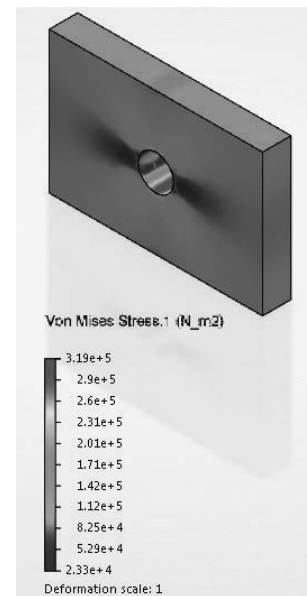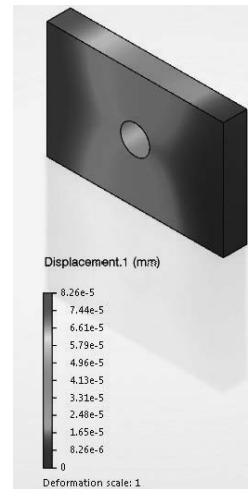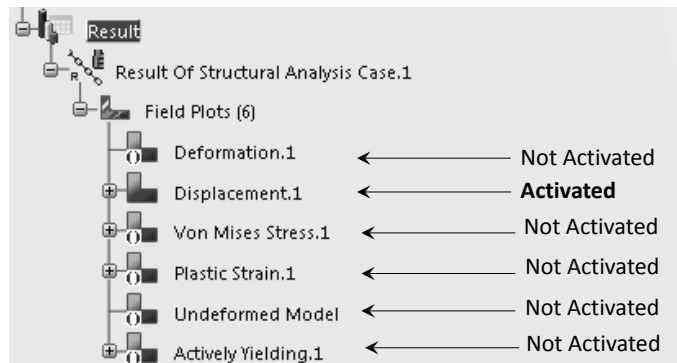
The "Plots" dialogue box can be closed to make room on the screen if needed. Use the cursor and select the arrow on the top left corner of the window; this will collapse the box as seen next.
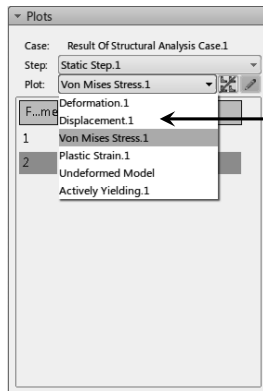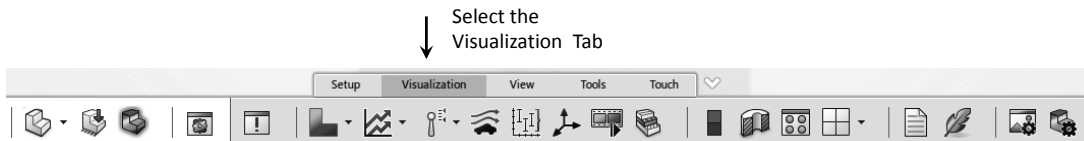
Select Displacement.1

Displacement.1 (mm)

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."

Deformation.1 ← Not Activated
Displacement.1 ← **Activated**
Von Mises Stress.1 ← Not Activated
Plastic Strain.1 ← Not Activated
Undeformed Model ← Not Activated
Actively Yielding.1 ← Not Activated

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 "Visualization" tab from the bottom row of icons.

Select the Visualization Tab

Setup   Visualization   View   Tools   Touch

Select the "Single View" icon ⬜ from the bottom row. Expand the section by choosing the "Two Rows" icon

⬜ Two Rows .

Single View
Visualize plot in single view configuration.
ⓘ Press F1 for more help.
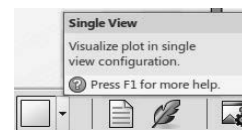
You will then see that the screen is split into two sections with the "Displacement.1" displayed in both. This is depicted below.
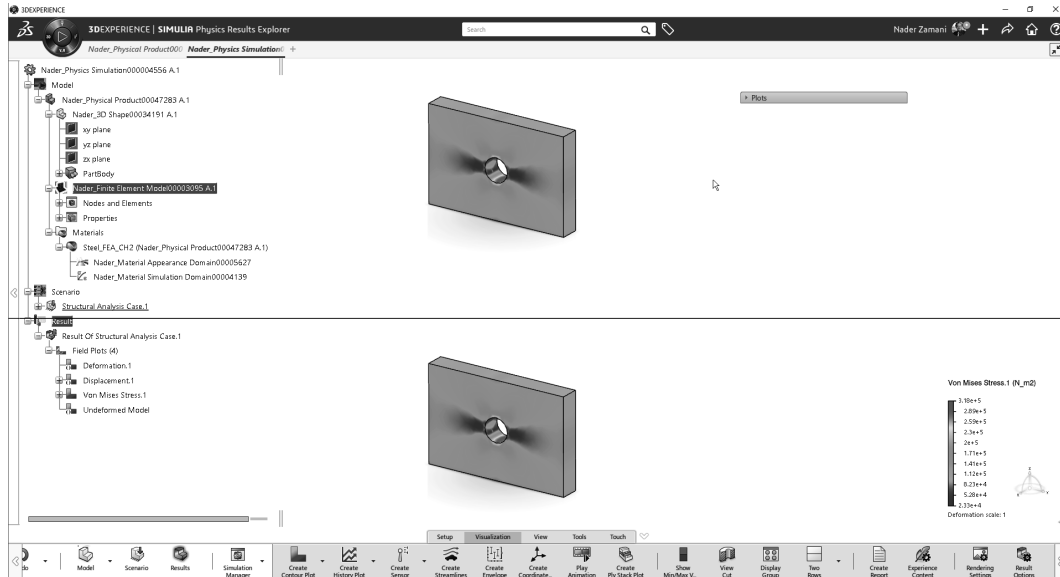
Point the cursor to the branch of the tree "von Mises Stress.1," right click and select
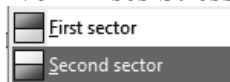
"Activate" followed by "Second Sector" .

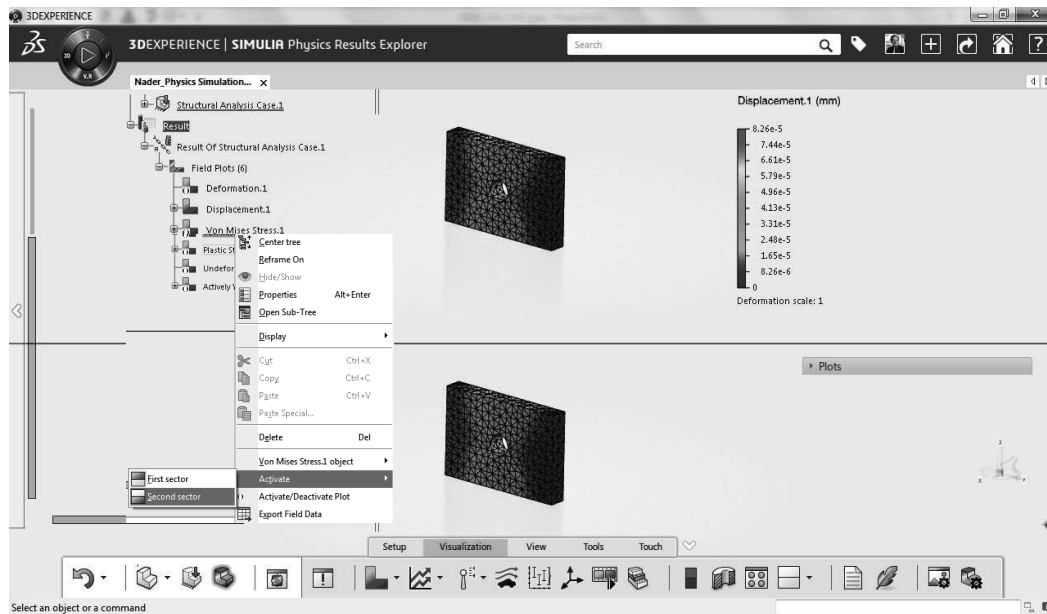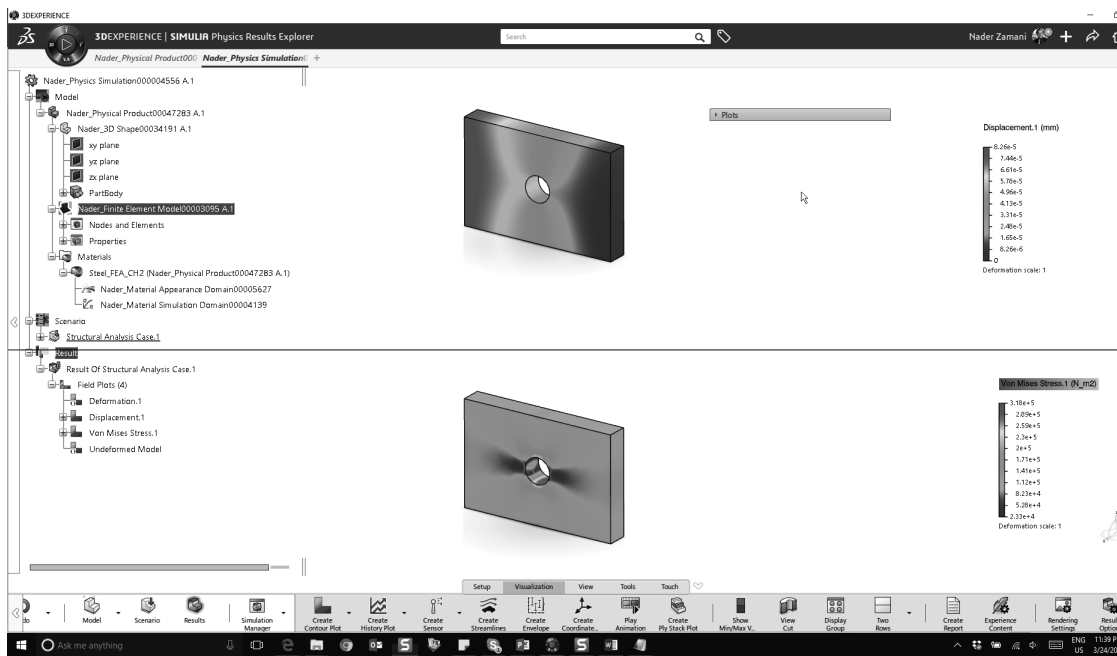You will see that the "von Mises Stress.1" contour is displayed in the bottom sector of the screen as shown below.

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 "Diplacement.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.

Dimmed contour plot

Selecting the contour legend dims out the contour plot

Select the contour legend, then zoom in, zoom out, pan the contour as needed.
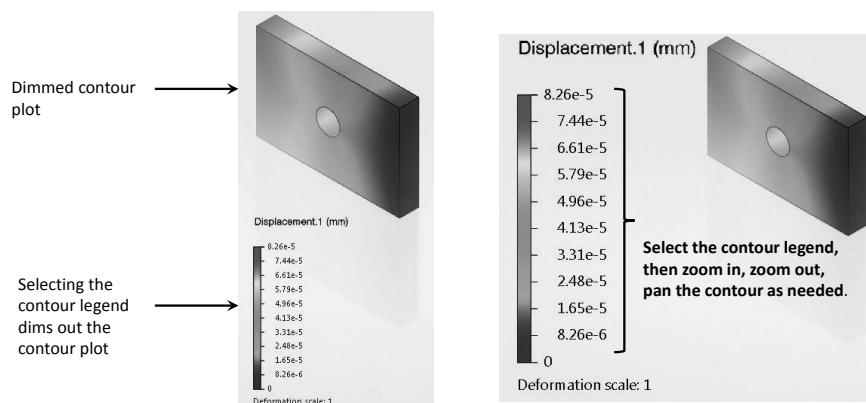
The group of icons ⬛ 📖 ⚏ ⬜ on the bottom of the screen are very useful in post processing. The multiple view icon ⬜ has already been discussed. We will now review the section cut icon 📖. Once this icon is selected, the part can be cut at an arbitrary orientation by rotating/translating the "Robot"; two random cuts are shown below.
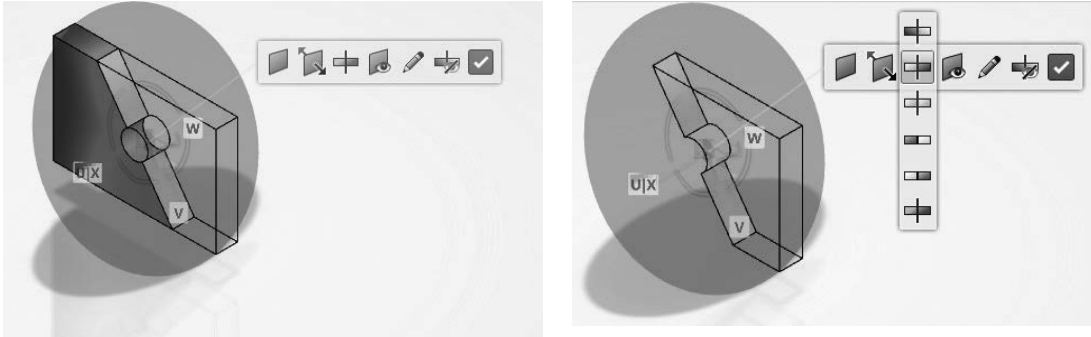
Manipulating the contour plots as far as formatting and setting the parameters is very important. To explore such features, double click on an existing contour on the screen (or simply double click on a branch of the tree which corresponds to a contour in the "Results" section).

Double click on one of these branches or on an existing contour on your screen
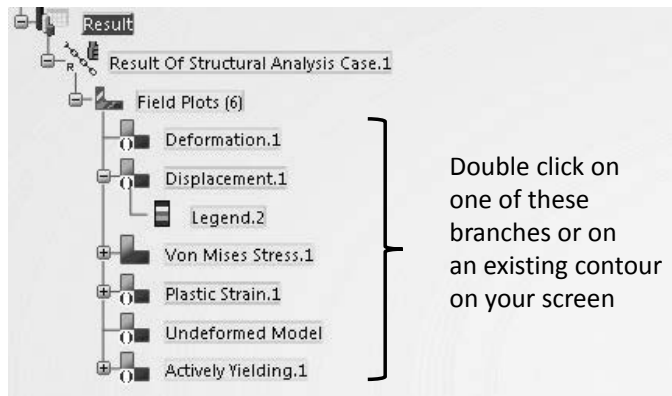
Once the contour appears on the screen, double click on it again. This action leads to the "Contour Plot" pop-up window shown on the right. This window has three tabs and many pulldown menus which enables you to customize the plot.
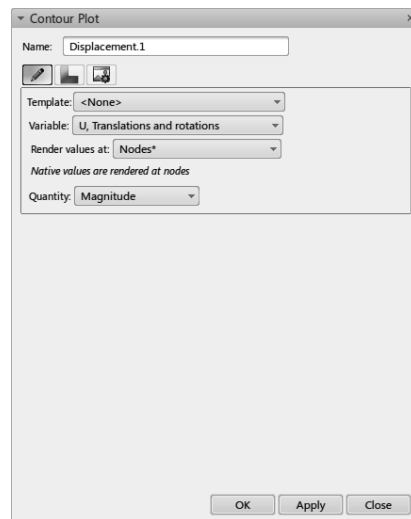
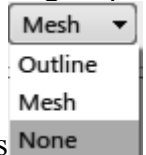The three tabs are immediately below the "Name" ✏️ 📊 🖼️.

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"
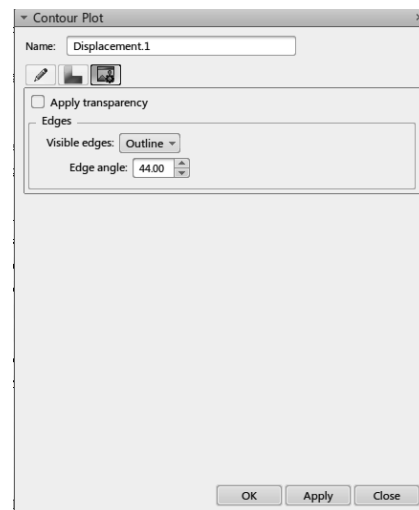button in the pop-up window.

Scale factor of
90000 instead of 1
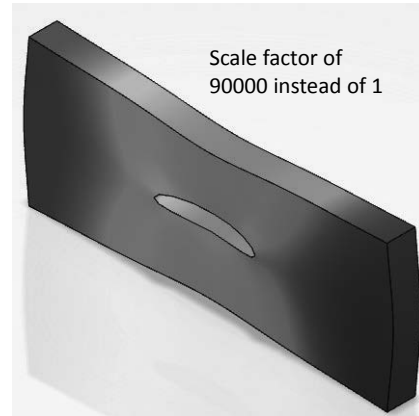
Next select the third tab 🖼️ .

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

Mesh ▾
Outline
Mesh
three options None .

The effects are shown in the contour plots
below.

Contour Plot

Name: Displacement.1

☐ Apply transparency

Edges

Visible edges: Outline ▾

Edge angle: 44.00

OK   Apply   Close

Visible edges = Mesh   **Visible edges = Outline**   **Visible edges = None**

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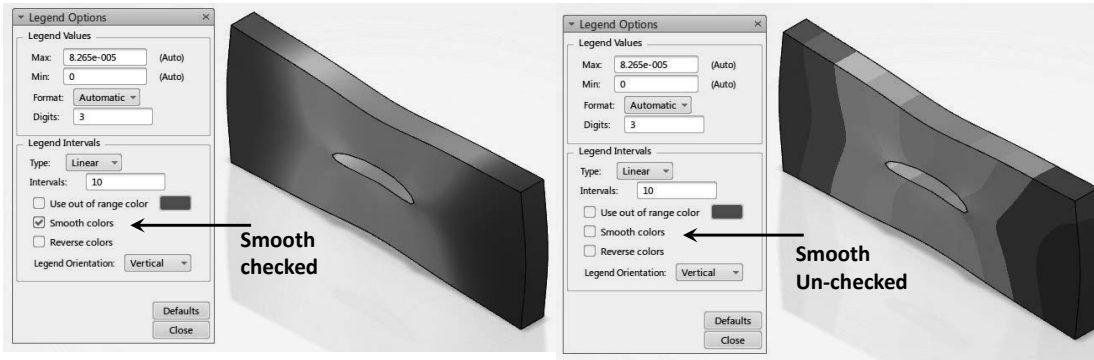
The number of colors and the maximum and minimum range for the variable can also be set in this window. Note that the setting which allows you to orient the legend "Horizontal" or "Vertical" is at the bottom of the window.

Select the "Create Report" icon from the bottom of the screen. This will lead to the dialogue box shown below.
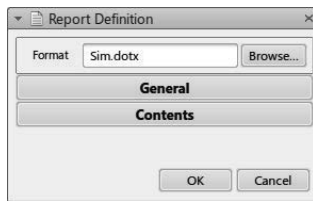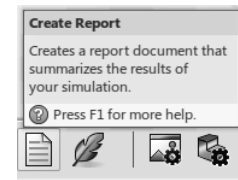
A professional looking report in "Word" or "Power Point" formats can be generated. Personalized information such as company/university logo can be automatically inserted.

You are (must be) still in the "Visualization" tab of the Results section.
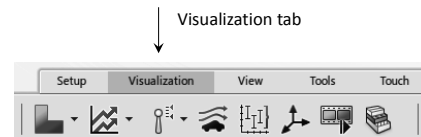
Select the "Play Animation" icon from the bottom row.
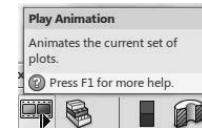
This leads to the "Play" player.

The speed of the animation can be changed via

the "Options" as shown on the right. Finally, the looping control is achieved by the

three choices .

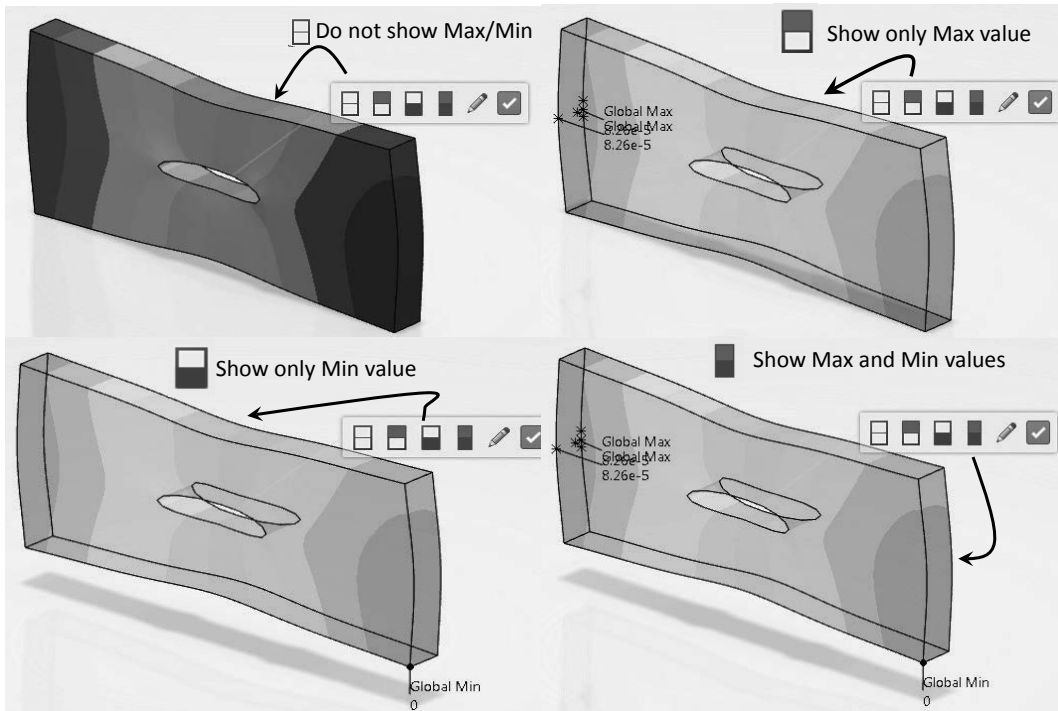A very useful post processing feature is the "Show Min/Max Values" icon ▮ from the bottom row of icons.

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 is the displacement vector (or symbol), but plotted as a vector. In this example, y is the direction where the part is loaded.

Select the "Create Contour Plot" icon ▮ from the bottom of the screen. Expand the selection to see your other options as shown below. From the available list, choose the "Create Symbol Plot" icon 🡕🡕 Create Symbol Plot . Note that the icon "Create Isocontour Plot" is essentially the same as "Create Contour Plot" ▮, except that there is no rendering.
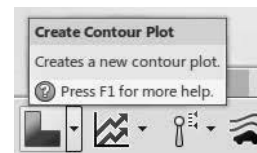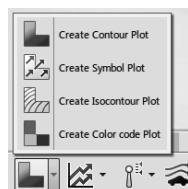
Follow the instructions below to select "UT, Translation" with "Vector Component 2."

Select the UT, Translation as the variable

Select the appropriate component. Here it is Component 2.

For Symbol shape, select "Vectors."
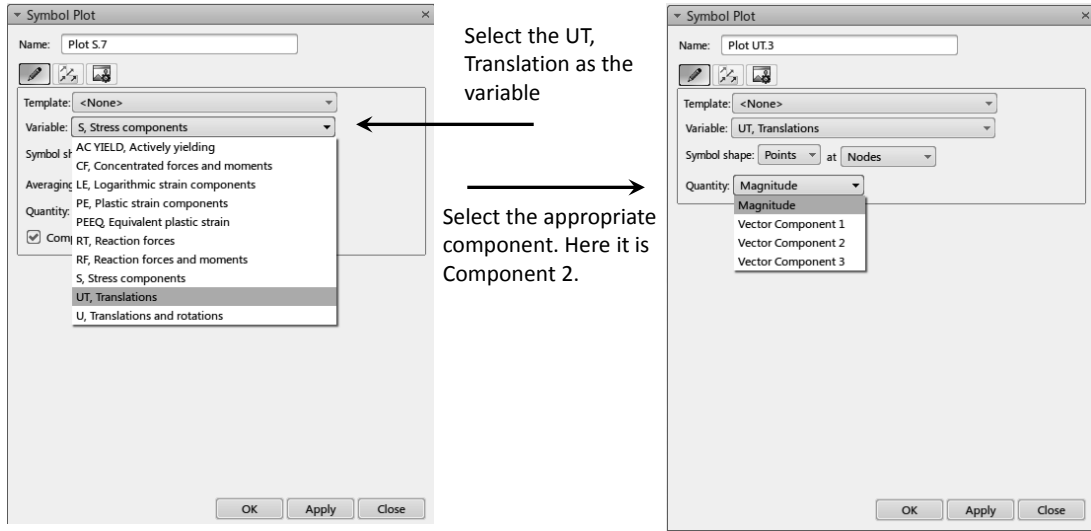
For Symbol shape Select "Vectors"
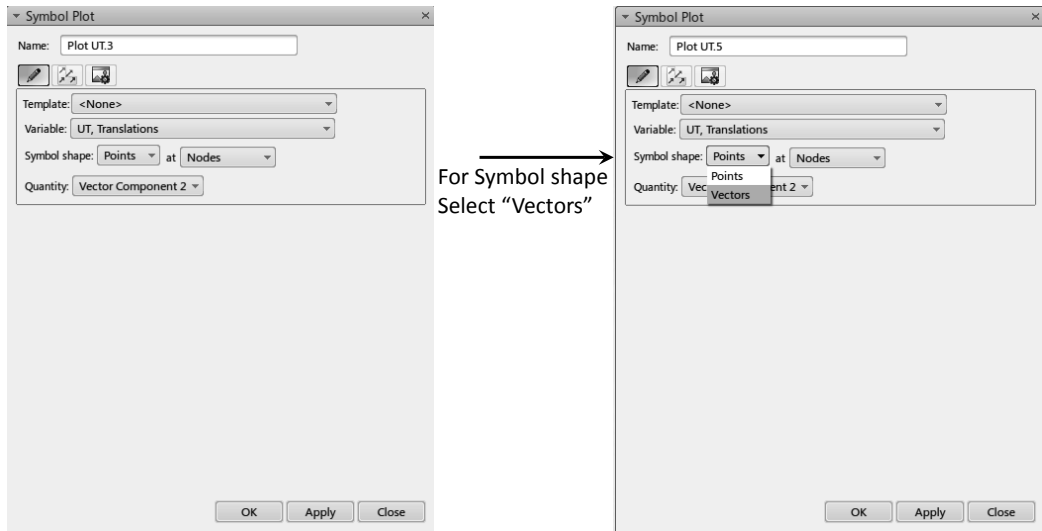
Finally press "Apply." The symbol plot (vector plot) shown below appears. The issue with this plot is that there are too many symbols (arrows).

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 "Simulation Manager" icon

which can be picked from the bottom row as shown.
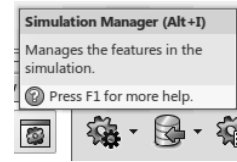
This selection leads to the "Simulation Manager" dialogue box
below which contains valuable information.

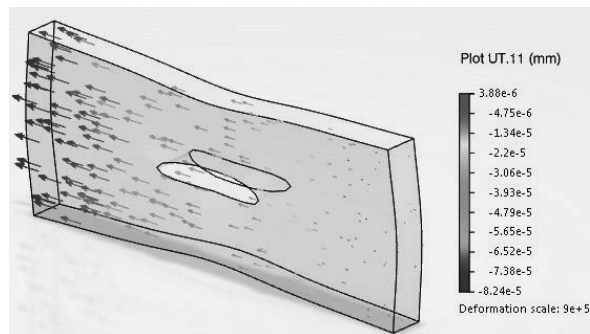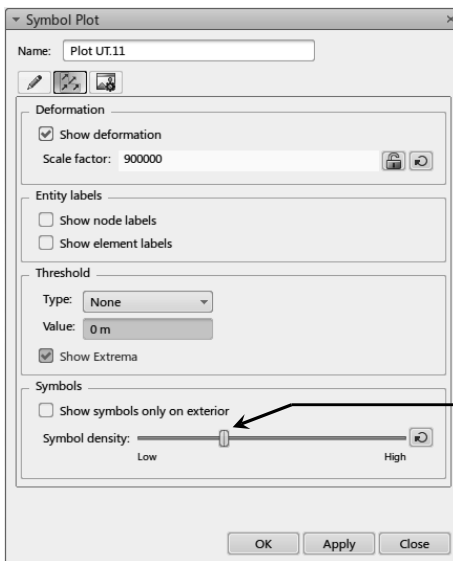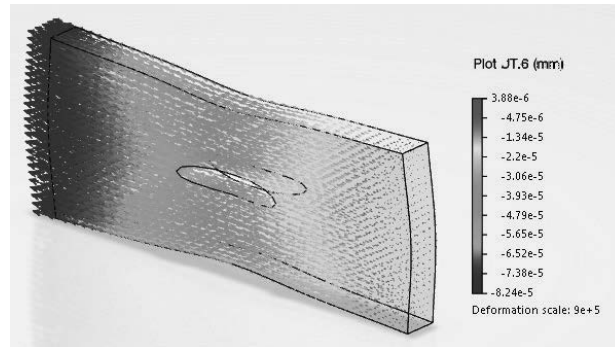| Plot Name	Min.	Max.	Fields	Units
Deformation.1	------	------	U	mm
Displacement.1	0 mm	8.26e-5 mm	U, U	mm, mm
Von Mises Stress.1	2.33e+4 N_m2	3.19e+5 N_m2	S, U	N_m2, mm
Plastic Strain.1	0	0	PEEQ, U	None, mm
Undeformed Model	------	------		
Actively Yielding.1	0	0	AC YIELD, U	None, mm

It is worth mentioning that dialogue boxes such as "Simulation Manager"   and the

"Diagnostic Viewer"   can be accessed any time by right clicking on the screen as
shown below.
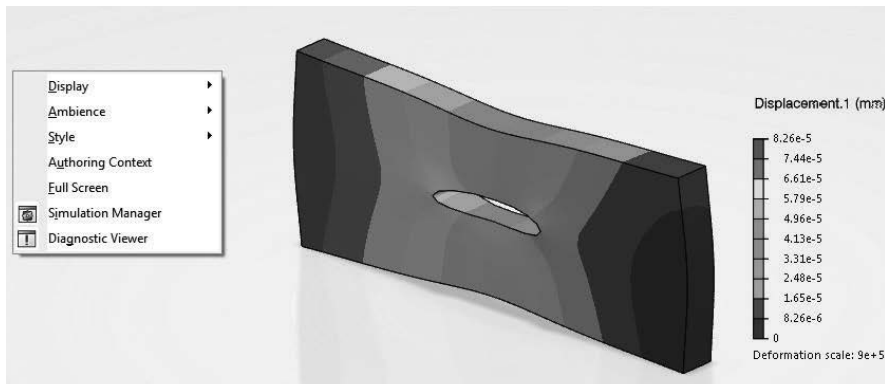
There are many, many other important features that were not touched upon in this chapter
but will be explored in later ones.

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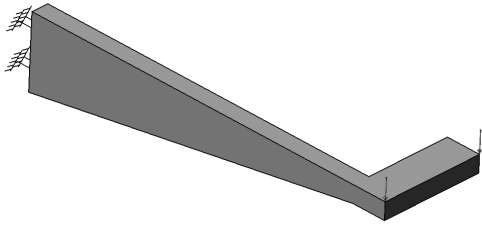
Try running the model with the two different element "size" and "sag" with both the
linear and parabolic type of elements. Record the results in terms of the maximum
displacement and the maximum von Mises stress in a table and comment on the results.
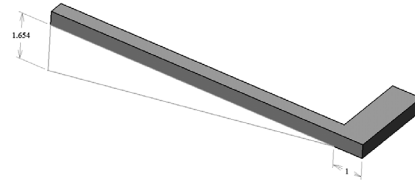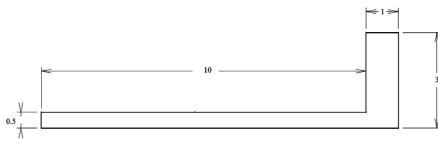The run time of the parabolic elements with element size of 0.1 could be substantial
depending on the type of processor used.

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 element. 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.

Problem 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 lb separated by 1.5 in. This is equivalent to a torque of 1500 lb.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.

Draw the rectangle on the circular face, then pad it away .5 in.
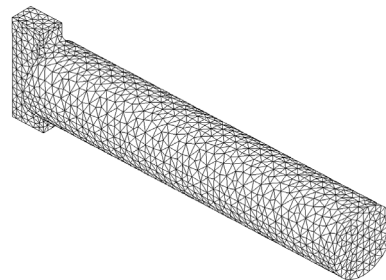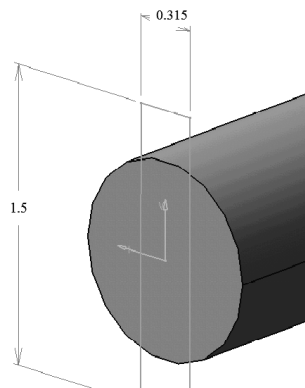
Model the part with linear solid elements with size = 0.1 and sag = 0.025 which results in the mesh shown on the previous page. Compare the hoop stress (The hoop stress is the largest principal stress "C11") with the theoretical solution from strength of materials.

Partial Answer:

The strength of materials solution is based on $\tau = \dfrac{Tr}{J}$ 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, $r = \dfrac{D}{2}$, and $J = \dfrac{\pi D^4}{32}$. The hoop stress "C1" which numerically equals τ is calculated from $\dfrac{16T}{\pi D^3}$. For the present problem, $T = 1500\,\text{lb.in}$ and $D = 1\,\text{in}$. Based on these parameters, a value of 7643 psi for the hoop stress is predicted.