Engineering Analysis with COSMOSWorks Professional
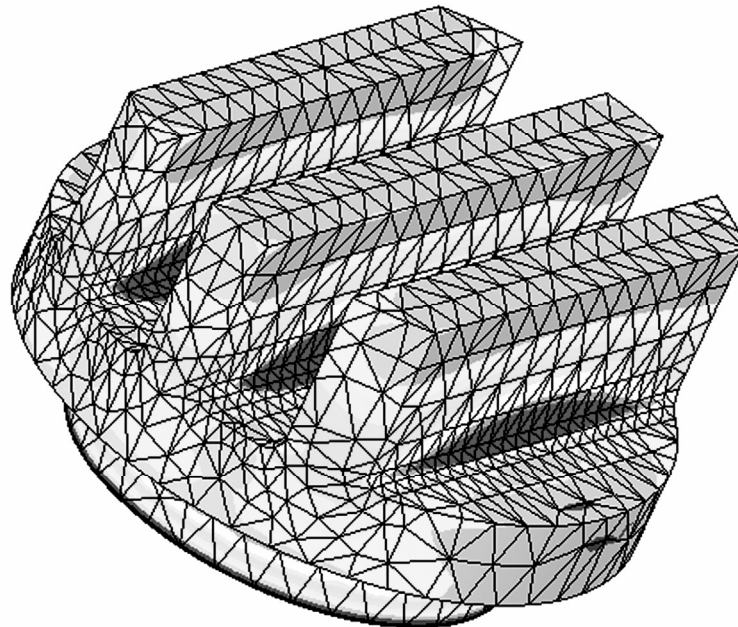
Finite Element Analysis with COSMOSWorks 2005

Paul M. Kurowski Ph.D., P.Eng.

SolidWorks

SDC
PUBLICATIONS

Design Generator, Inc.

Schroff Development Corporation

2: Static analysis of a plate

Topics covered

- ❑ Using COSMOSWorks interface
- ❑ Linear static analysis with solid elements
- ❑ The influence of mesh density on displacement and stress results
- ❑ Controlling discretization errors by the convergence process
- ❑ Presenting FEA results in desired format
- ❑ Finding reaction forces

Project description

A steel plate is supported and loaded, as shown in figure 2-1. We assume that the support is rigid (this is also called built-in support or fixed support) and that the 100,000 N tensile load is uniformly distributed along the end face, opposite to the supported face.

Fixed restraint applied to this face

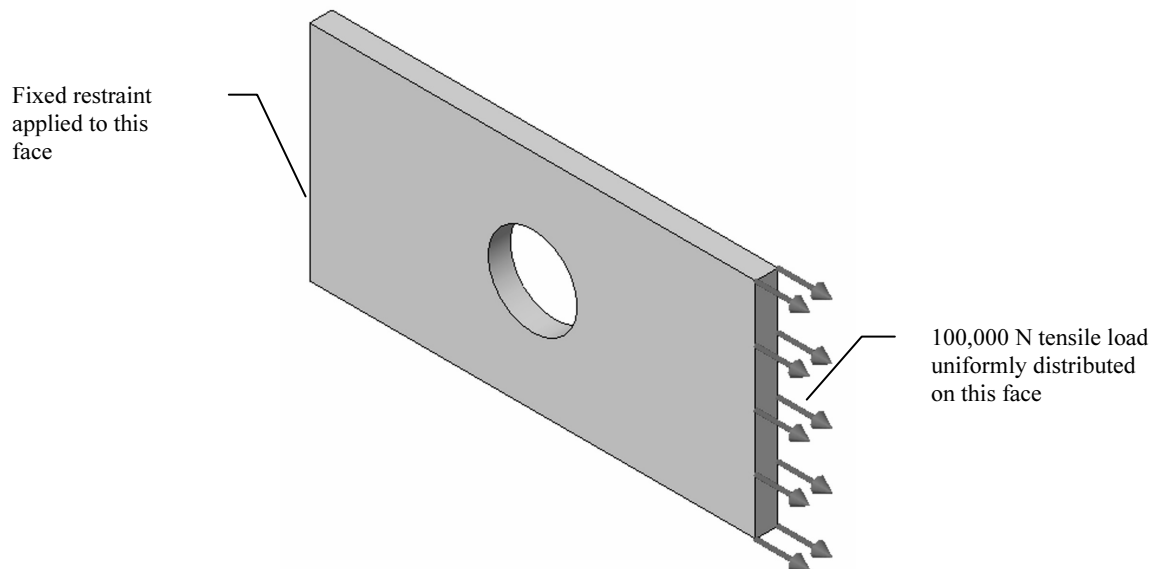100,000 N tensile load uniformly distributed on this face

Figure 2-1: SolidWorks model of a rectangular plate with a hole

We will perform displacement and stress analysis using meshes with different element sizes. Note that repetitive analysis with different meshes does *not* represent standard practice in FEA. We will repeat the analysis using different meshes only as a learning tool to gain more insight into how FEA works.

Procedure

In SolidWorks, open the model file called HOLLOW PLATE. Verify that COSMOSWorks is selected in the **Add-Ins** list. To start COSMOSWorks, select the COSMOSWorks Manager tab, as shown in figure 2-2.
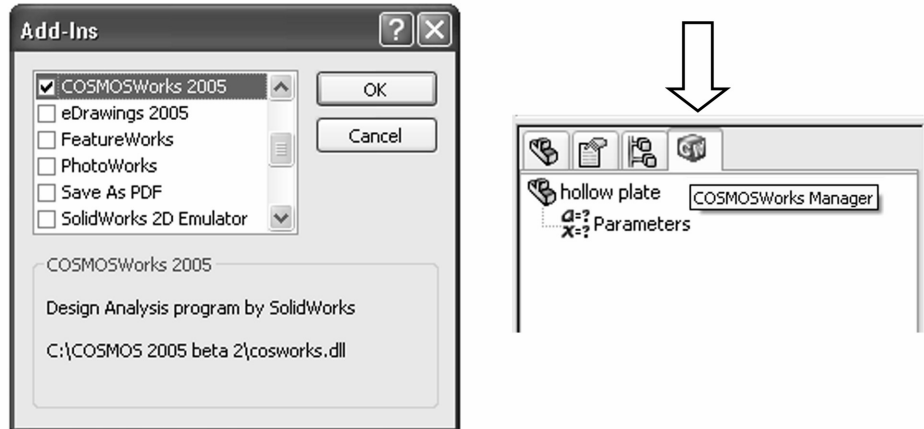

Figure 2-2: Add-Ins list and COSMOSWorks Manager tab

Verify that COSMOSWorks is selected in the list of Add-Ins (left), and then select the COSMOSWorks Manager tab (right).

To create an FEA model, solve it, and analyze the results, we will use a graphical interface. You can also do this by making the appropriate choices in the COSMOSWorks menu. To call up the menu, select COSMOSWorks from the main tool bar of SolidWorks (figure 2-3).
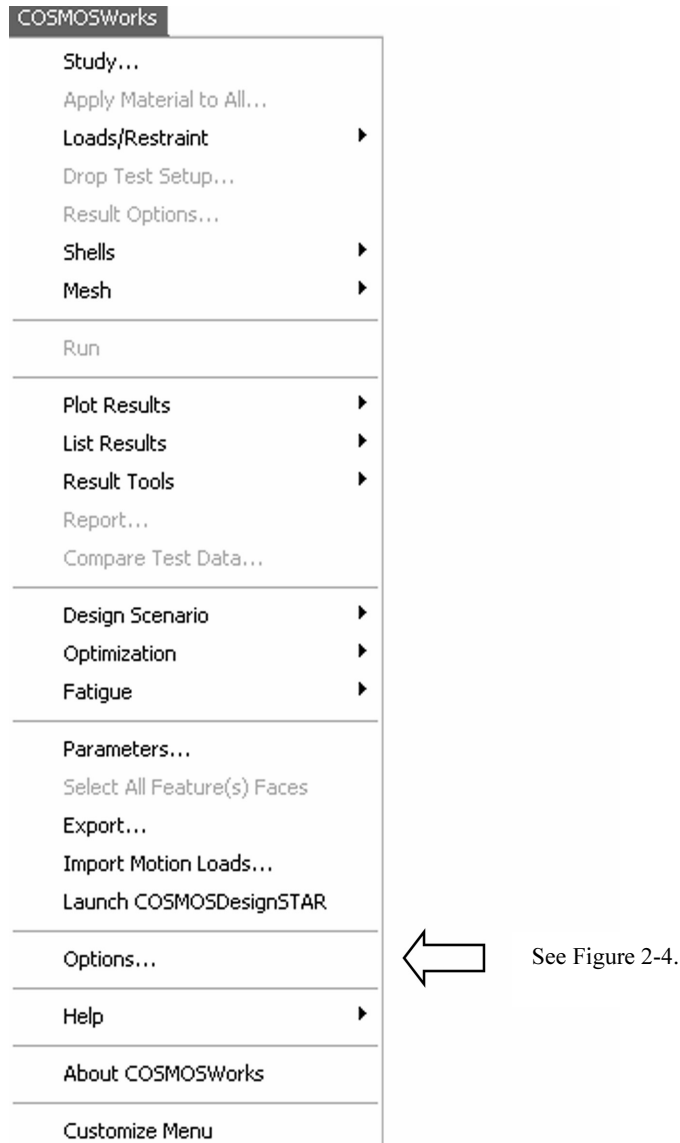
See Figure 2-4.

Figure 2-3: COSMOSWorks menu

All functions used for creating, solving, and analyzing a model can be executed either from this menu or from the graphical interface in the COSMOSWorks Manager window. We will use the second method.

Before we create the FEA model, let's review the **Options** window in COSMOSWorks (figure 2-4). This window can be accessed from the COSMOSWorks main menu (figure 2-3).
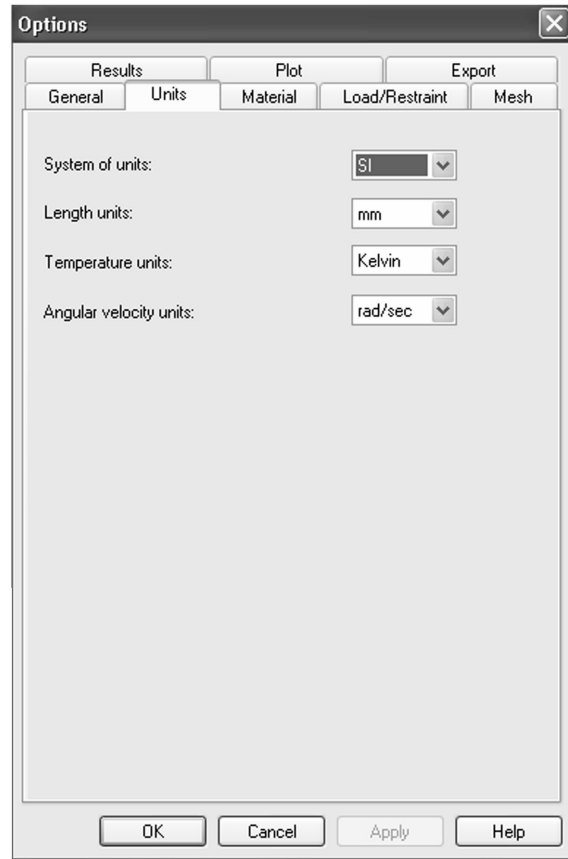
Figure 2-4: COSMOSWorks Options window; shown is Units tab

The COSMOSWorks Preferences window has several tabs. Selecting the Units tab, displayed above, allows selection of units of measurement. We will use the SI system. Please review other tabs before proceeding with the exercise.

Creation of an FEA model always starts with the definition of a study. To define a study, right-click the mouse on the *Part* icon in the COSMOSWorks Manager window and select **Study…** from the pop-up menu. In this exercise, the *Part* icon is called *hollow plate*, as it is in the SolidWorks Manager. Figure 2-5 shows the required selections in the study definition window: the analysis type is **Static**, the mesh type is **Solid mesh**. The study name is *tensile load 01*.
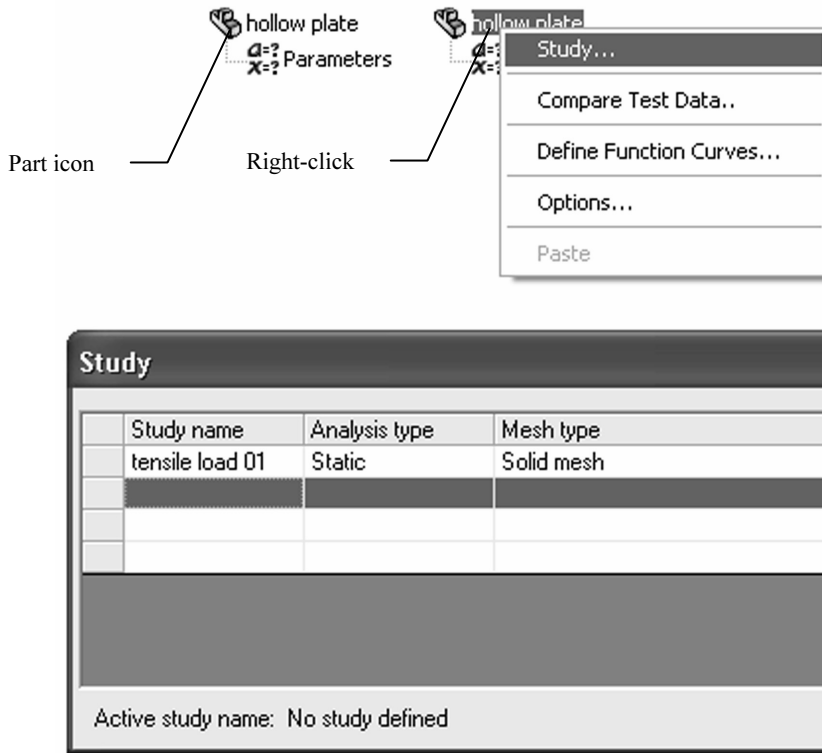
Figure 2-5: Study window

To display the Study window (bottom), right-click the Part icon in the COSMOSWorks Manager window (top left), and then from the pop-up menu (top right), select Study….

When a study is defined, COSMOSWorks automatically creates a study folder (named in this case *tensile load 01)* and places several sub-folders in it (some of these sub-folders are empty because their contents must by defined by the user), as shown in figure 2-6. Not all folders are used in each analysis. In this exercise, we will use the *Solids* folder to define and assign material properties, the *Load/Restraint* folder to define loads and restraints, and the *Mesh* folder to create the finite element mesh.
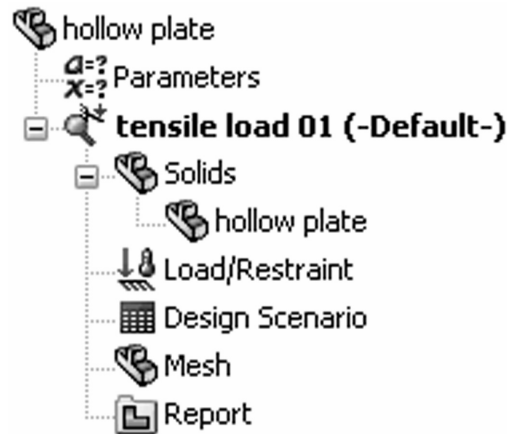
Figure 2-6: *Study* folders

COSMOSWorks automatically creates a study folder, called tensile load 01, with the following sub folders: Solids, Load/Restraint, Design Scenario, Mesh and Report folder. The Design Scenario and Report folders will not be used in this exercise, nor will the Parameters folder, which is automatically created prior to study definition.

We are now ready to define the mathematical model. This process generally consists of the following steps:

❑ Geometry preparation

❑ Material properties assignment

❑ Restraints application

❑ Load application

In this case, the model geometry does not need any preparation (it is already very simple), so we can start by assigning material properties.

You can assign material properties to the model either by:

❑ Right-clicking the mouse on the *Solids* folder, or

❑ Right-clicking the mouse on the *hollow plate* icon, which is located in the *Solids* folder.

The first method assigns the same material properties to all components in the model, while the second method assigns material properties to one particular component (in this exercise, *hollow plate*). In this example there is no difference between the two methods, because we are working with a single part and not with an assembly. Therefore, there is only one component in the *Solids* folder.

Right-click on the *Solids* folder and select **Apply Material to All**. This action opens the **Material** window shown in figure 2-7.
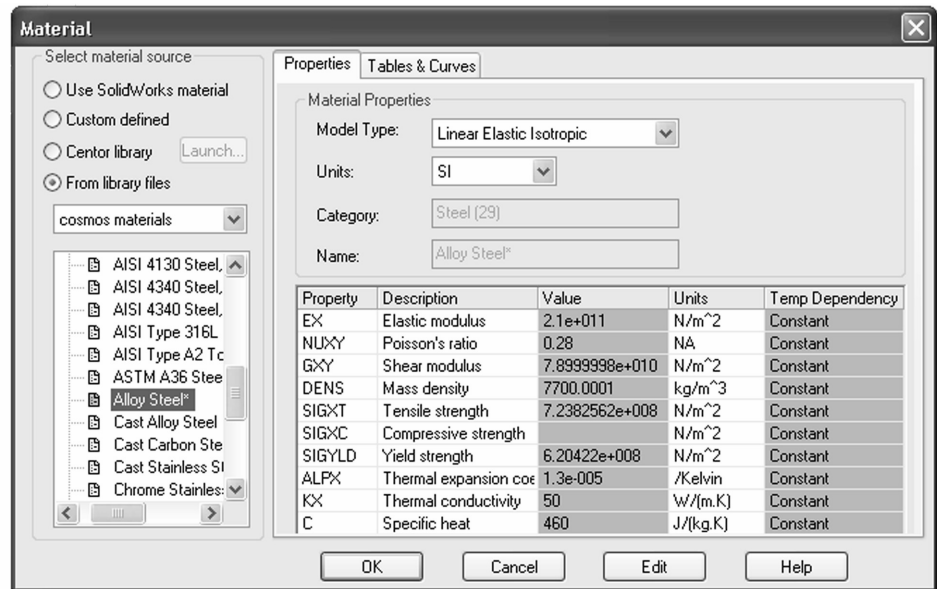

Figure 2-7: Material window

Select **From library files** in the Select material source area, then Select **Alloy Steel.** Select **SI** units under **Properties** tab (other units could be used as well). Notice that the Solids folder now shows a check mark and the name of the selected material to indicate that a material has successfully been assigned. If needed, you could define your own material by selecting **Custom Defined** material.

Note that material assignment actually consists of two steps:

❑ Material selection (or material definition if custom material is used)

❑ Material assignment to either all solids in the model or to selected components (this makes a difference only if the assembly is analyzed)

We should also notice that if a material has been defined for a SolidWorks part model, material definition is automatically transferred to the COSMOSWorks model. Assigning a material to the SolidWorks model is actually a preferred modeling technique, especially when working with an assembly consisting of parts with many different materials. We will do this in later exercises.

To display the pop-up menu that lists the options available for defining loads and restraints, right-click the *Load/Restraint* folder in *tensile load 01* study (figure 2-8).
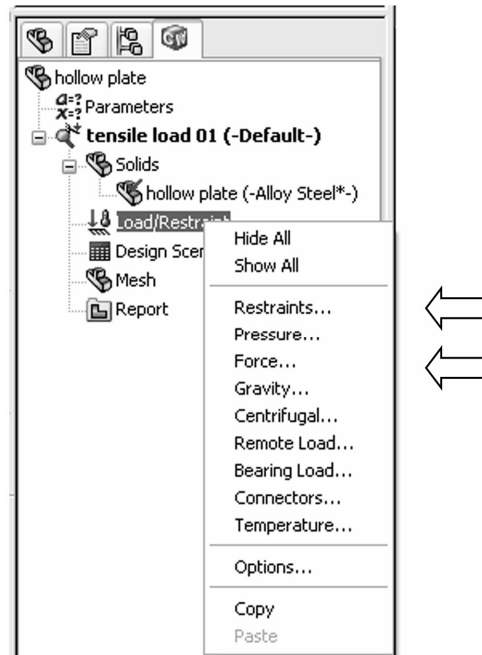

Figure 2-8: Pop-up menu for the *Load/Restraint* folder

The arrows indicate the selections used in this exercise.

To define the restraints, select **Restraints…** from the pop-up menu displayed in figure 2-8. This action opens the **Restraint** window shown in figure 2-9.
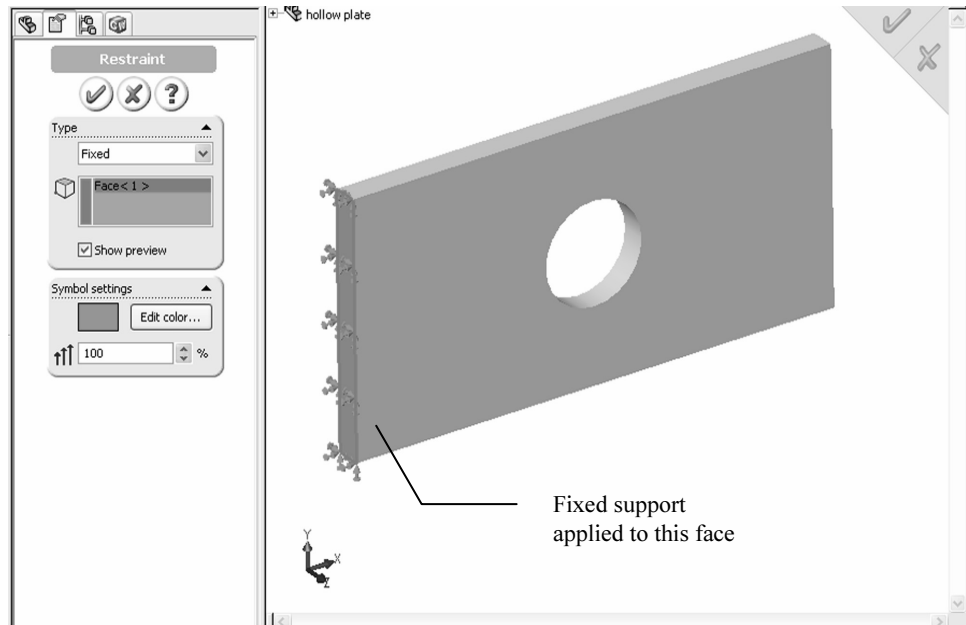
Figure 2-9: Restraint window

In Restraints, select Fixed as Restraint type and also select the model face where restraint is to be applied. In the graphic window, note the symbols of the applied restraint.

You can rotate the model in order to select the face where the restraints are applied. The Rotate, pan, zoom and all other view functions in COSMOSWorks work the same as in SolidWorks.

In the **Restraint** window, select **Fixed** as the type of restraint to apply. The following chart reviews other choices offered in the **Restraint** window:

Restraint Type	Definition
Fixed	Also called built-in or rigid support, all translational and all rotational degrees of freedom are restrained.

Note that **Fixed** restraints do not require any information on the direction along which restraints are applied. |
| **Immovable** **(No translations)** | Only translational degrees of freedom are constrained, while rotational degrees of freedom remain unconstrained.

If solid elements are used (like in this exercise), **Fixed** and **Immovable** restraints have the same effect because solid elements do not have rotational degrees of freedom. |
Reference plane or axis	This option restrains a face, edge, or vertex only in a certain direction, while leaving the other directions free to move. You can specify the desired direction of restraint in relation to the selected reference plane or reference axis.
On flat face	This option provides restraints in selected directions, which are defined by the three principal directions of the flat face where restraints are being applied.
On cylindrical face	This option is similar to **On flat face**, except that the three principal directions of a cylindrical face define the directions of restraints.
On spherical face	Similar to **On flat face** and **On cylindrical face**. The three principal directions of a spherical face define the directions of applied restraints.
Symmetry	This option is similar to **On flat face**. It applies symmetry boundary conditions automatically to a flat face. Translation in the direction normal to the face is restrained and rotations about axes aligned with the face are restrained.

When a model is fully supported (as it is in our case), we say that the model does not have any rigid body motions (term "rigid body modes" is also used), meaning it cannot move without experiencing deformation.

Note that the presence of supports in the model is manifested by both the restraint symbols (showing on the restrained face) and by the automatically created icon, *Restraint-1*, in the *Load/Restraint* folder. The display of the restraint and load symbols can be turned on and off by either:

❑ Using the **Hide All** and **Show All** commands in the pop-up menu shown in figure 2-8, or

❑ Right-clicking the restraint or load icon individually to display a pop-up menu and then selecting **Hide** from the pop-up menu.

We now define the load by selecting **Force** from the pop-up menu shown in figure 2-8. This action opens the **Force** window, (figure 2-10).
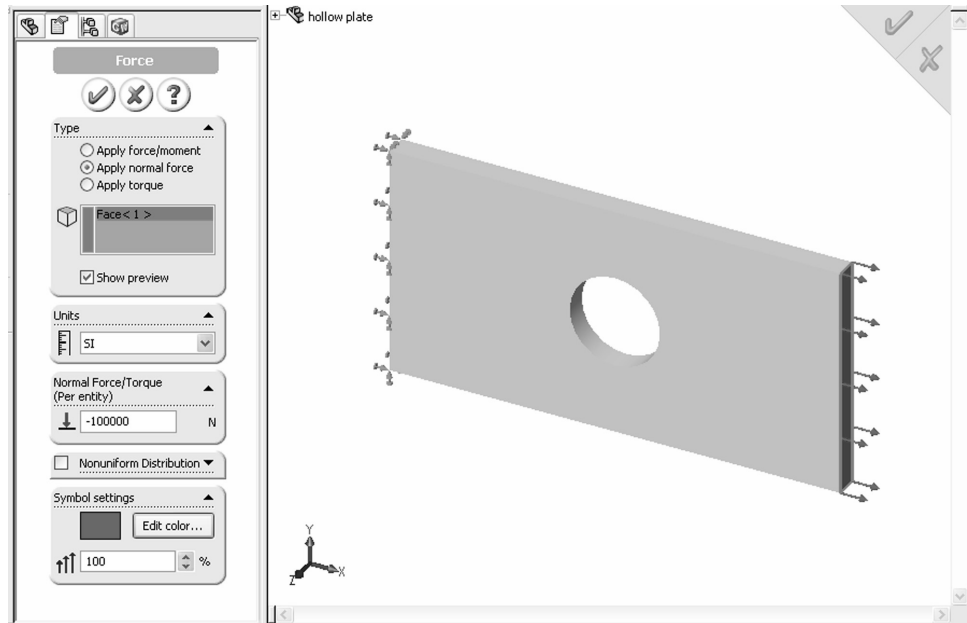

Figure 2-10: Force window

The Force window displays the selected face where tensile force is applied. This illustration also shows symbols of applied restraint and load.

In the Type area, select the **Apply normal force** button in order to load the model with 100,000 N of tensile force uniformly distributed over the end face, as shown in figure 2-10. Note that tensile force requires that the load magnitude be defined with a minus sign.

Generally, forces can be applied to faces, edges, and vertexes using different methods, which are reviewed below:

Force Type	Definition
Apply force/moment	This option applies force or moment to a face, edge, or vertex in the direction defined by selected reference geometry (plane or axis).
	Note that moment can be applied only if shell elements are used. Shell elements have all six degrees of freedom (three translations and three rotations) per node and can take moment load. Solid elements have only three degrees of freedom (translations) per node and, therefore, cannot take moment load directly.
	If you need to apply moment to solid elements, it must be represented with appropriately applied forces.
Apply normal force	Available for flat faces only, this option applies load in the direction normal to the selected face.
Apply torque	Used for cylindrical faces, this option applies torque about a reference axis using the Right-hand Rule.

The presence of load(s) is visualized by arrows symbolizing the load and by automatically created icons *Force-1* in the *Load/Restraint* folder.

Try using the click-inside technique to rename *Restraint-1* and *Force-1* icons. Note that renaming using the click-inside technique works on all icons in the COSMOSWorks Manager.

The mathematical model is now complete. Before creating the Finite Element model, let's make a few observations about defining:

❑ Geometry

❑ Material properties

❑ Loads

❑ Restraints

Geometry preparation is a well-defined step with few uncertainties. Geometry that is simplified for analysis can be checked visually by comparing it with the original CAD model.

Material properties are most often selected from the material library and do not account for local defects, surface conditions, etc. Therefore, definition of material properties usually has more uncertainties than geometry preparation.

The definition of loads is done in a few quick menu selections, but involves many assumptions. Factors such as load magnitude and distribution are often known only approximately and must be assumed. Therefore, significant idealization errors can be made when defining loads.

Defining restraints is where severe errors are most often made. For example, it is easy enough to apply a fixed restraint without giving too much thought to the fact that a fixed restraint means a rigid support – a mathematical abstract. A common error is over-constraining the model, which results in an overly stiff structure that will underestimate deformations and stresses. The relative level of uncertainties in defining geometry, material, loads and restraints is qualitatively shown in figure 2-11.
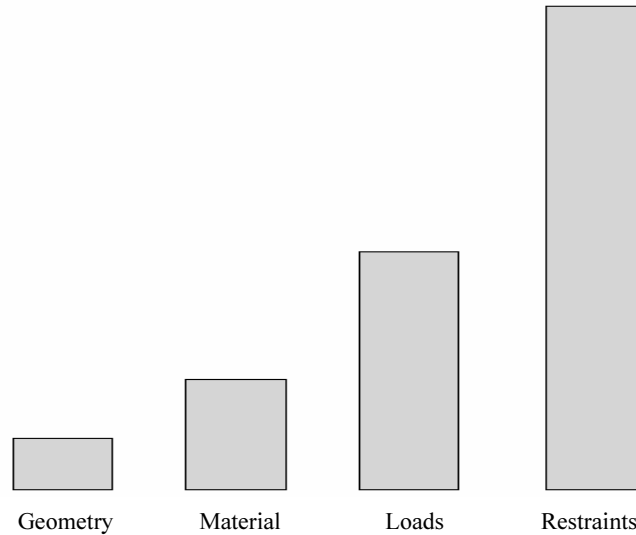
Figure 2-11: Qualitative comparison of uncertainty in defining geometry, material, loads, and restraints

The level of uncertainty (or the risk or error) has no relation to time required for each step, so the message in figure 2-11 may be counterintuitive. In fact, preparing CAD geometry for FEA may take hours, while applying restraints takes only a few mouse clicks.

In all of the examples presented in this book, we will assume that material properties, loads, and supports are known *with certainty* and that the way they are defined in the model represents an acceptable idealization of real conditions. However, we need to point out that it is the responsibility of user of the FEA software to determine if all those idealized assumptions made during the creation of the mathematical model are indeed acceptable.

We are now ready to mesh the model but first, verify under study **Options**, **Mesh** tab (figure 2-12) that **High** mesh quality is selected.
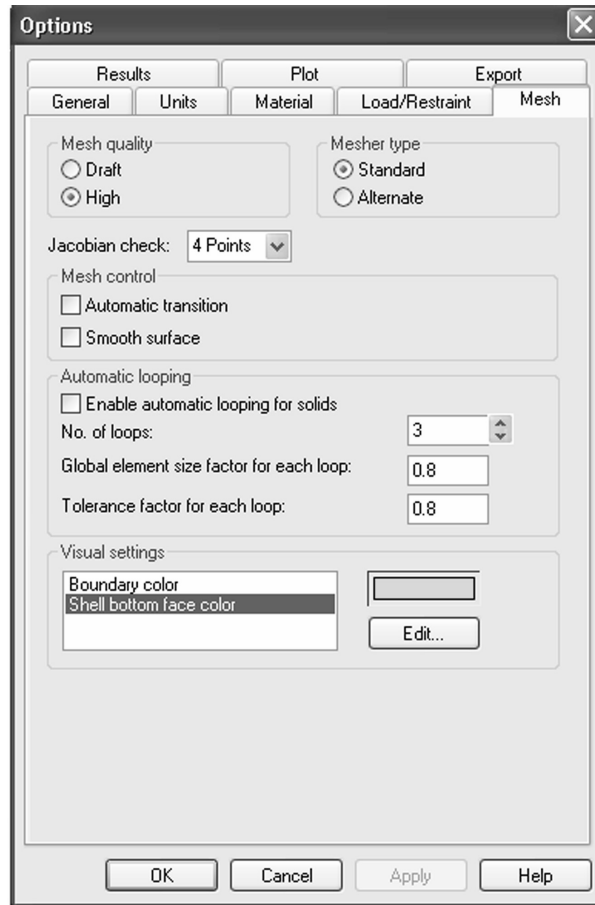
Figure 2-12: Mesh tab in the Preferences window

We use this window to verify that the choice of mesh quality is set to High.

The difference between **High** and **Draft** mesh quality is that:

❑ Draft quality mesh uses first order elements

❑ High quality mesh uses second order elements

Differences between first and second order elements were discussed in chapter 1.

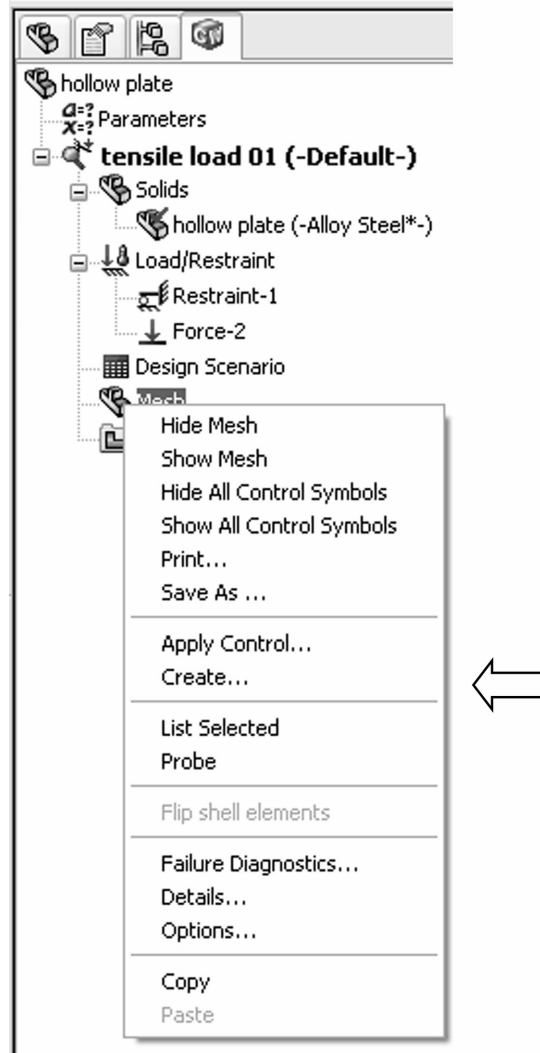Now, right-click the *Mesh* folder to display the pop-up menu (Figure 2-13).

Figure 2-13: Mesh pop-up menu

In the pop-up menu, select **Create…** to open the **Mesh** window. This window offers a choice of element size and element size tolerance. In this exercise, we will study the impact of mesh size on results.

We will solve the same problem using three different meshes: coarse, medium (default), and fine. Figure 2-14 shows the respective selection of meshing parameters to create the three meshes.
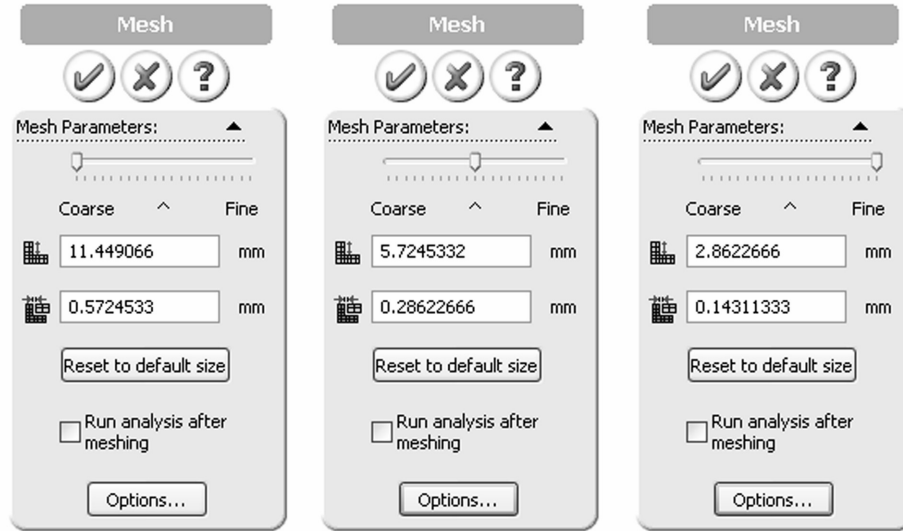
Figure 2-14: Three choices for mesh density from left to right: coarse, medium (default) and fine

The medium mesh density, shown in the middle window in figure 2-14, is the default that COSMOSWorks proposes for meshing our model. The element size of 5.72 mm and the element size tolerance of 0.286 are established automatically based on the geometric features of the SolidWorks model. The 5.72-mm size is the characteristic element size in the mesh, as explained in figure 2-15. The element size tolerance is the allowable variance of the actual element sizes in the mesh.

Mesh density has a direct impact on the accuracy of results. The smaller the elements, the lower the discretization errors, but the meshing and solving time take longer. In the majority of analyses with COSMOSWorks, the default mesh settings produce meshes that provide acceptable discretization errors, while keeping solution times reasonably short.
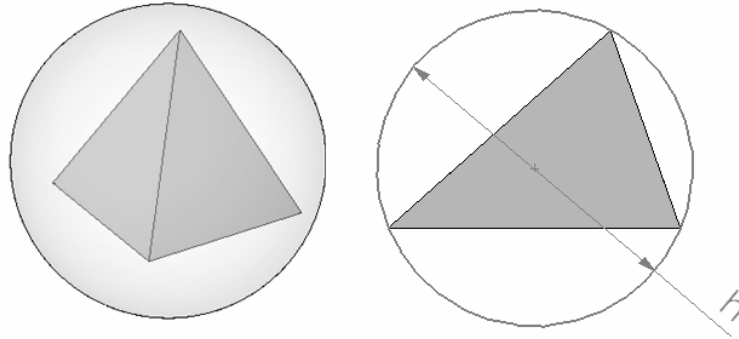

Figure 2-15: Characteristic element size for a tetrahedral element

The characteristic element size of a tetrahedral element is the diameter h of a circumscribed sphere (left). This is easier to illustrate with the 2-D analogy of a circle circumscribed on a triangle (right).

Right-click the *Mesh* icon again and select **Create…** to open **Mesh** window.

With the **Mesh** window open, set the slider all the way to the left (as illustrated in figure 2-14 left) to create a coarse mesh, and click the green check mark button. The mesh will be displayed as shown in figure 2-16.
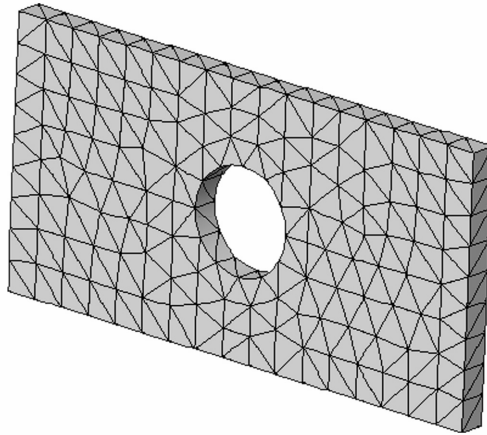

Figure 2-16: Coarse mesh created with second order, solid tetrahedral elements

*You can control the mesh visibility by selecting **Hide Mesh** or **Show Mesh** from the pop-up menu shown in figure 2-12.*

To start the solution, right-click the *tensile load 01*study folder. This action displays a pop-up menu (figure 2-17).
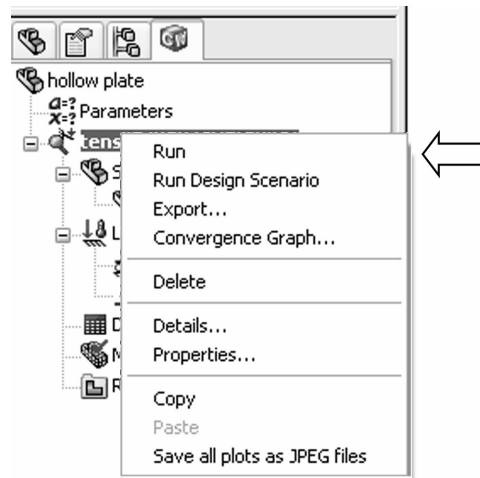
Figure 2-17: Pop-up menu for the *Study* icon

Start the solution by right-clicking the tensile load 01 icon to display a pop-up menu. Select Run to start the solution.

The solution can be executed with different properties, which will be investigated in later chapters. You can monitor the solution progress while the solution is running (figure 2-18). The exact appearance of this window depends on what solver is being used as selected in **Options** under **General** tab (figure 2-4).
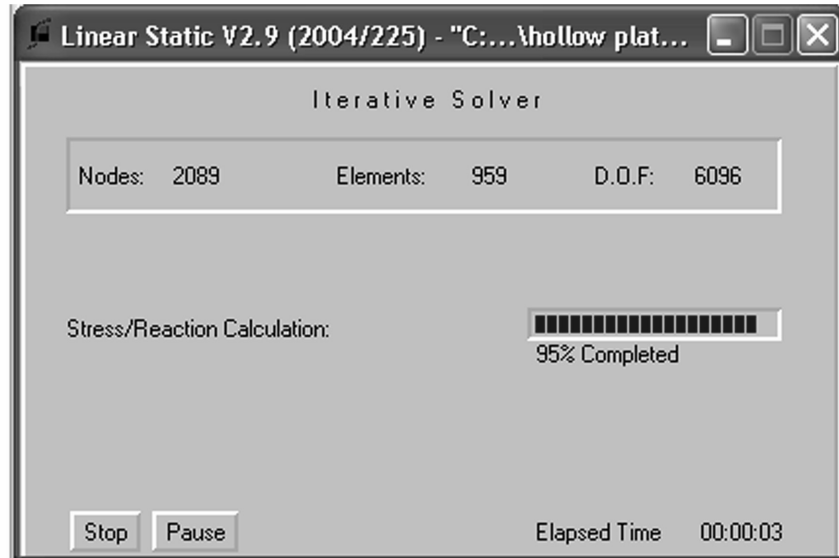

Figure 2-18: Solution Progress window

The Solver reports solution progress while the solution is running. This window is specific to FFEPlus solver.

A successful or failed solution is reported (figure 2-19) and must be acknowledged before proceeding to analysis of results.

Figure 2-19: Solution outcome: completed or failed

With the solution completed, COSMOSWorks automatically creates several new folders in the COSMOSWorks Manager window:

❑ *Stress*

❑ *Displacement*

❑ *Strain*

❑ *Deformation*

❑ *Design Check*

Each folder holds an automatically created plot with its respective type of result (figure 2-20). If desired, you can add more plots to each folder.
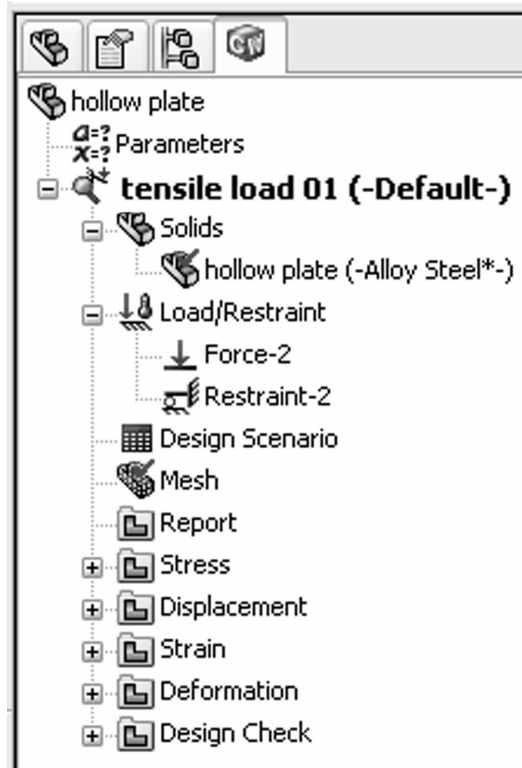

Figure 2-20: Automatically created *Results* folders

One default plot of respective results is contained in each of the automatically created Results folders: Stress, Displacement, Strain, Deformation and Design Check.

To display stress results, double-click on *Plot1* icon in the *Stress* folder or right-click it and select **Show** from a little pop-up menu. Default stress plot is shown in figure 2-21.
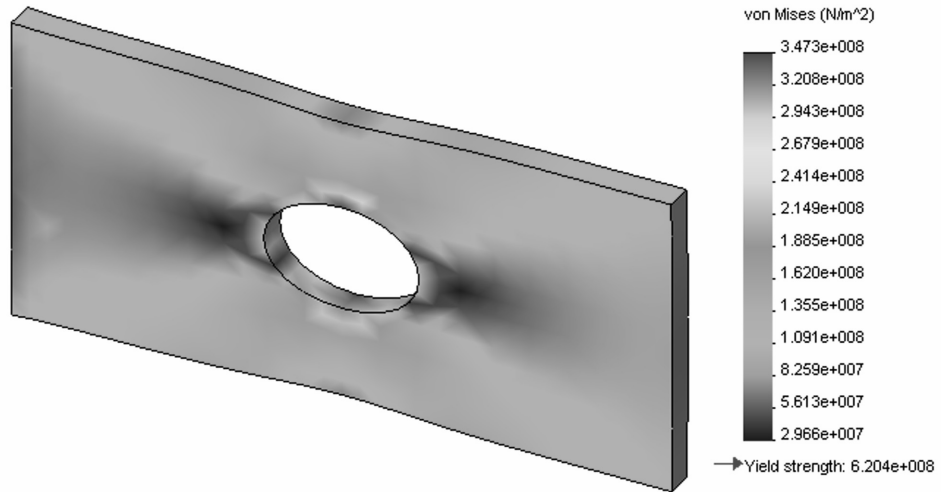
Figure 2-21: Stress plot displayed using default stress plot settings

*Von Mises stress results are shown by default in stress plot window. Notice that results are shown in (*Pa*) and the highest stress 347.3e8 Pa is below the material yield strength 620.4e8 Pa.*

Once stress plot is showing, right-click plot icon to display pop-up menu featuring different plot display options (figure 2-22).
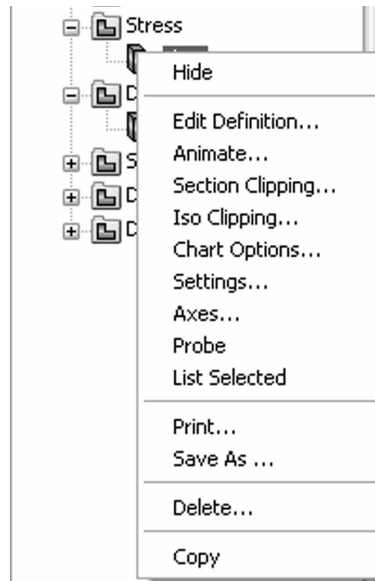

Figure 2-22: Pop-up menu with plot display options

Any type of plot can be modified using selections from this pop-up menu.

We will now examine how to modify the stress plot using **Edit Definition**, **Chart Options** and **Settings,** which are all selectable from the pop-up menu in figure 2-22.

Select **Edit Definition** to open **Stress Plot** window, change units to MPa then close the window. In the **Chart Options** window, change the range of displayed stress results from **Automatic** to **Defined**, and define the range as from 0 to 300 MPa. In **Settings,** select **Discrete** in **Fringe options** and **Mesh** in **Boundary options**. All these selections are shown in figure 2-23.
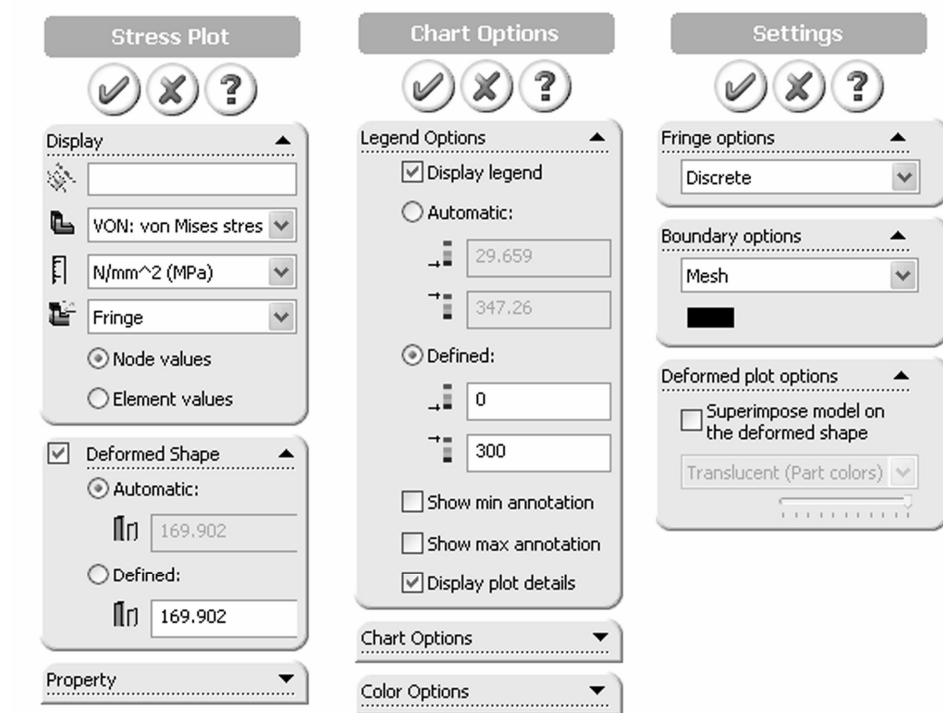

Figure 2-23: **Stress Plot** (invoked by **Edit Definition…**), **Chart Options** and **Settings** windows we use in this exercise to modify stress plot display.
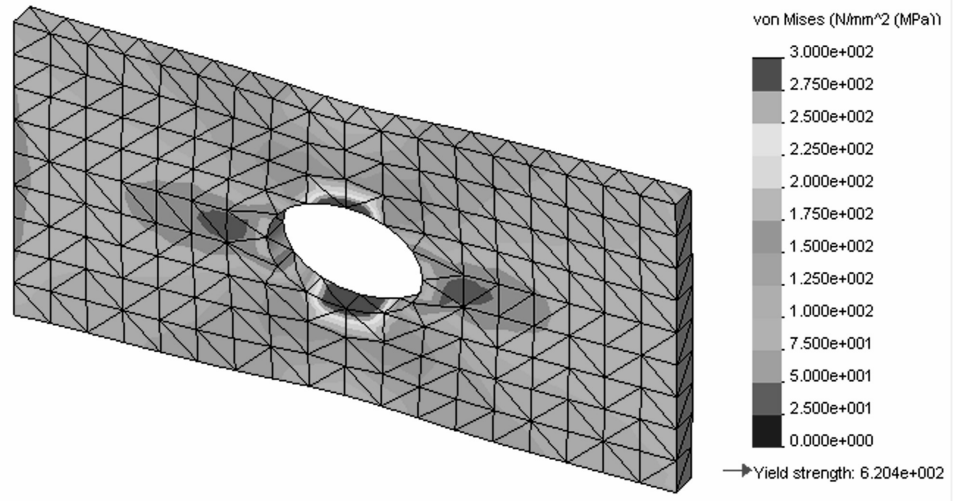
Figure 2-24: Modified stress plot results

The modified stress plot using selections shown in windows in figure 2-23.

Stress plots in figure 2-21 and 2-24 are both presented as node values, also called averaged stresses. Element values (or non-averaged stresses) can also be displayed by proper selection in the **Stress Plot** window. Node values are most often used to present stress results. See chapter 3 and the glossary of terms at the end of this book for more comments on node and element values of stress results.

Before you proceed, please investigate other selections available from the pop-up menu in figure 2-22

We will now review the displacement, strain, and deformation results. All of these plots are created and modified in the same way. Sample results are shown in:

❑ Figure 2-25 (displacement)

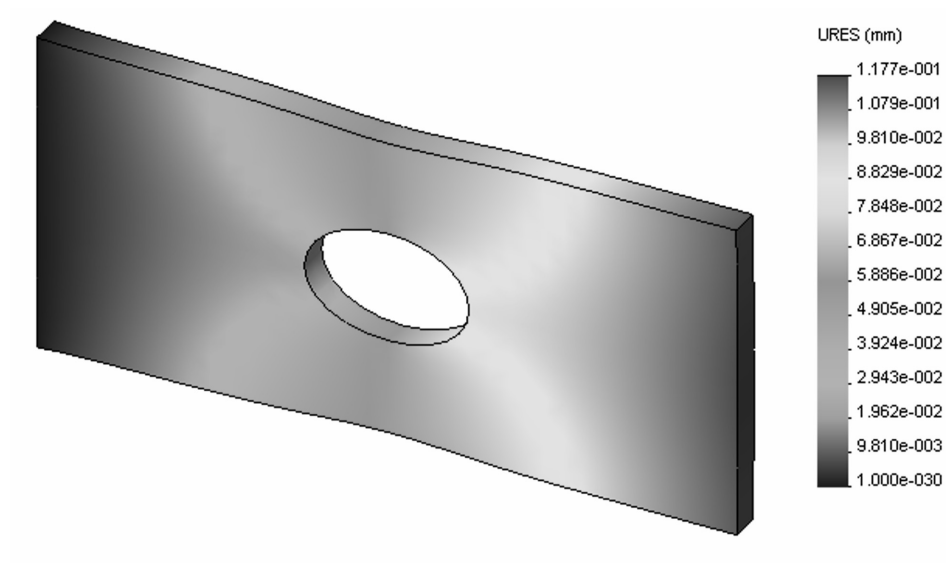❑ Figure 2-26 (strain)

❑ Figure 2-27 (deformation)

Figure 2-25: Displacement results using Continuous fringe options

Also try Discrete fringe option selectable in Settings from plot pop up menu.

This plot shows the deformed shape in an exaggerated scale. You can change the display from undeformed to deformed and modify the scale of deformation in the **Displacement Plot** window activated by right-click plot icon, **Edit Definition**.
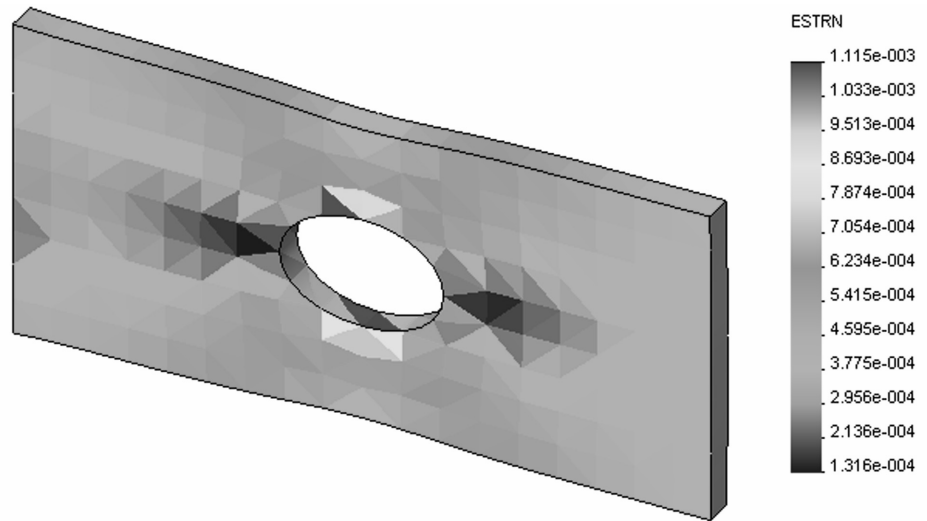
Figure 2-26: Strain results

Strain results are shown using element values. Notice that strain is dimensionless.
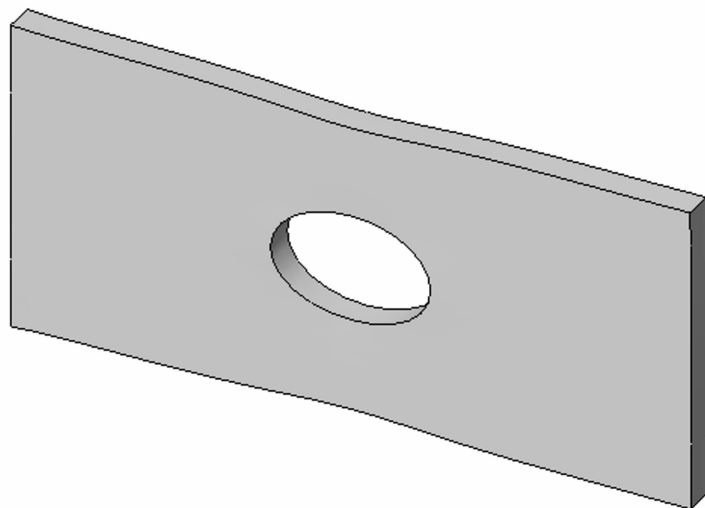

Figure 2-27: Deformation results

Note that deformed plots can also be created in all previous types of display if the deformed shape display is selected.

The last folder, called *Design Check* holds the *Plot1*, which by default displays the distribution of the factor of safety based on von Mises stress. We will modify this plot to display areas where the factor of safety falls below 2.

Right-click *Plot1* icon and select **Edit Definition…** from the pop-up menu to display the first of three **Design Check** windows. Follow step 1, step 2 and step 3 as shown in figure 2-28.
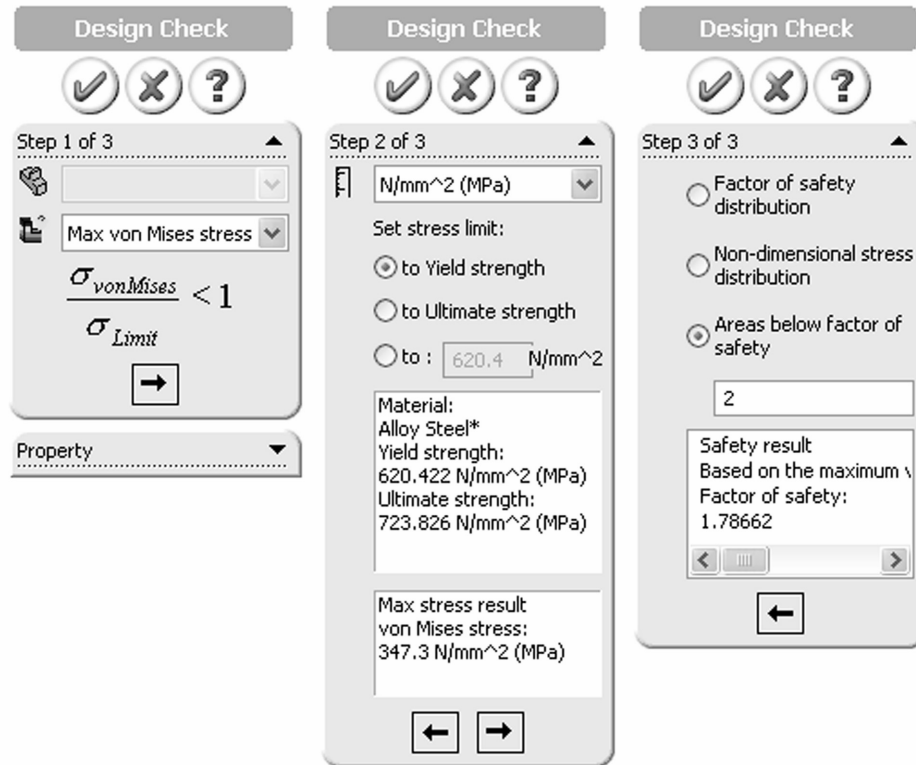

Figure 2-28: Three windows show three steps in the Design Check plot definition

Step one selects the failure criterion, step 2 selects display units and sets the stress limit, step three selects what will be displayed in the plot (here we select areas below the factor of safety 2).
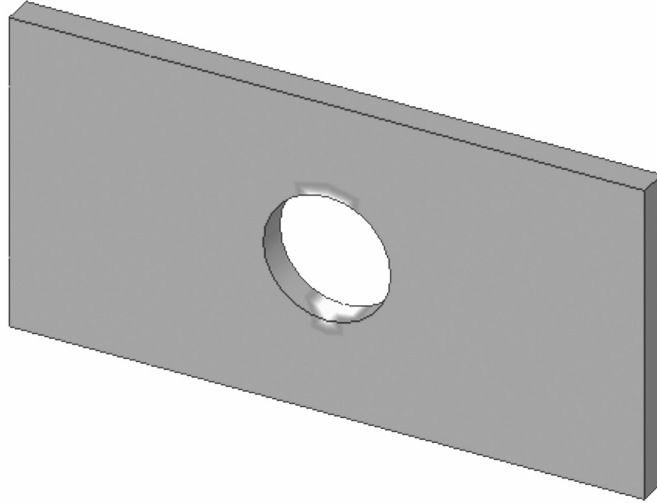
Figure 2-29: Red color (shown as light gray in this grayscale illustration) displays the areas where the factor of safety falls below 2

We have completed the analysis with a coarse mesh and now wish to see how a change in mesh density will affect the results. Therefore, we will repeat the analysis two more times using medium and fine density meshes respectively. We will use the settings shown in figure 2-13. All three meshes used in this exercise (coarse, medium, and fine) are shown in figure 2-30.
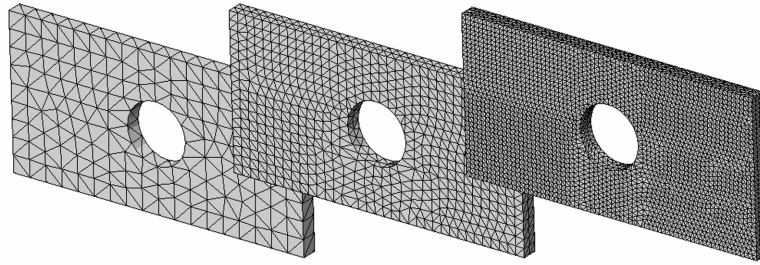
Figure 2-30: Coarse, medium, and fine meshes

We use the three meshes to study the effects of mesh density on the results.

To compare the results produced by different meshes, we need more information than is available in the plots. Along with the maximum displacement and the maximum von Mises stress, for each study we need to know:

❑   The number of nodes in the mesh

❑   The number of elements in the mesh

❑   The number of degrees of freedom in the mesh

The information on the number of nodes and number of elements can be found in **Mesh Details** (figure 2-31).
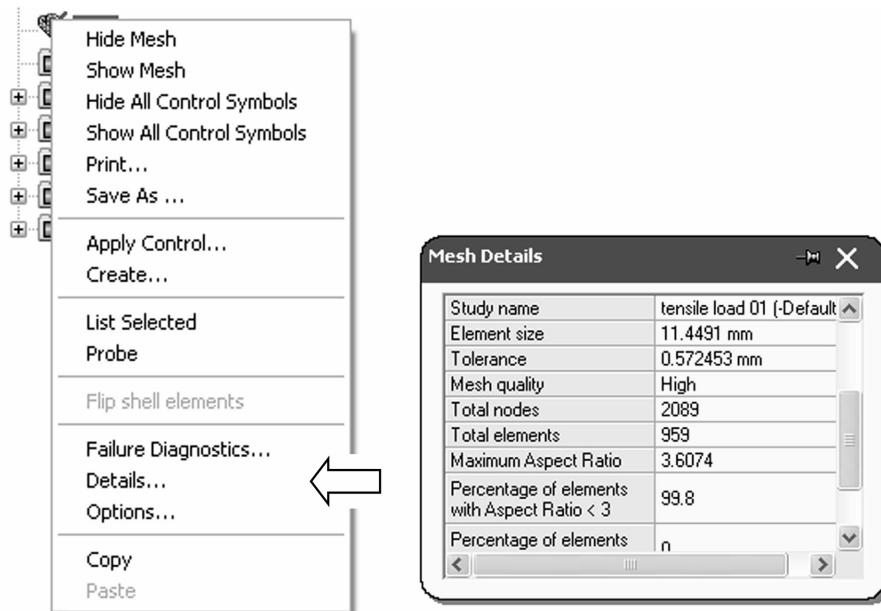

Figure 2-31: Meshing details window

Right click on Mesh, Details… to activate Mesh Details window.

The number of degrees of freedom can be found in one of COSMOS data base files located in **Work directory** folder specified in **Options** window under **Results** tab (figure 2-32).
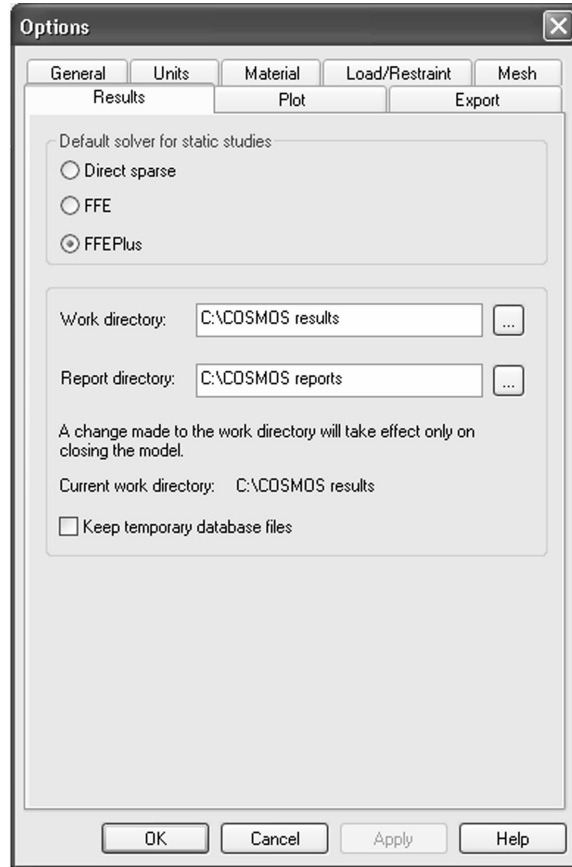

Figure 2-31: COSMOSWorks data base is located in Work directory specified under Results tab in Options window

Any folder can be used for storing COSMOSWorks results files.

Using Windows Explorer, find the file named hollow plate-tensile load 01.OUT. Open it with a text editor (e.g. Notepad) and verify that the number of degrees of freedom for *tensile load 01* study equals 6267.

Note that the OUT file is available only while the model is opened in COSMOSWorks. Upon exiting from COSMOSWorks by deselecting COSMOSWorks from the list of add-ins or by closing SolidWorks model, all data base files are compressed into one file with the extension CWR. In our case, the file is named hollow plate-tensile load 01.CWR. If more than one study has been executed, then there is one CWR file for each study. Compressing the entire data base into one file allows for a convenient backup of COSMOS results.

Now create and run two more studies: *tensile load 02* with medium (default) element size and *tensile load 03* with fine element size as shown in figure 2-13.

To create a new study we could just repeat the same steps as before but an easier way is to copy a study. To copy a study, right-click the study folder and select **Copy**. Next, right-click the model icon (hollow plate) and select **Paste** to invoke **Define Study Name** window, where the name of new study is defined in (figure 2-32).
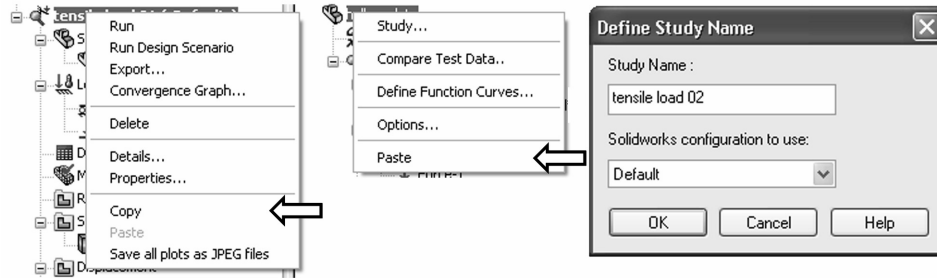

Figure 2-32: A study can be copied into another study in three steps as shown

An alternative way to copy a study is to use Copy (Ctrl-C) and Paste (Ctrl-V) technique of the Windows operating system. Yet another way is to drop study folder *tensile load 01* into *hollow plate* folder.

Note that all definitions in a study (material, restraints, loads, mesh) can also be copied individually from one study to another.

A study is copied complete with results and plot definitions. Before remeshing study *tensile load 02* with the default element size mesh, you must acknowledge the warning message shown in figure 2-33.
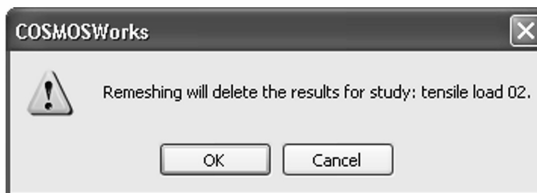

Figure 2-33: Remeshing deletes any existing results in the study

The summary of results produced by the three models is shown in figure 2-34.

mesh density	max. displ. magnitude [mm]	max. von Mises stress [MPa]	number of DOF	number of elements	number of nodes
coarse	0.1177	347	6096	959	2089
medium (default)	0.1180	368	23601	3916	8052
fine	0.1181	376	239748	52329	80419

Figure 2-34: Summary of results produced by the three meshes

Note that these results are based on the same problem. Differences in the results arise from the different mesh densities used.

Figures 2-35 and 2-36 show the maximum displacement and the maximum von Mises stress as functions of the number of degrees of freedom. The number of degrees of freedom is in turn a function of mesh density.
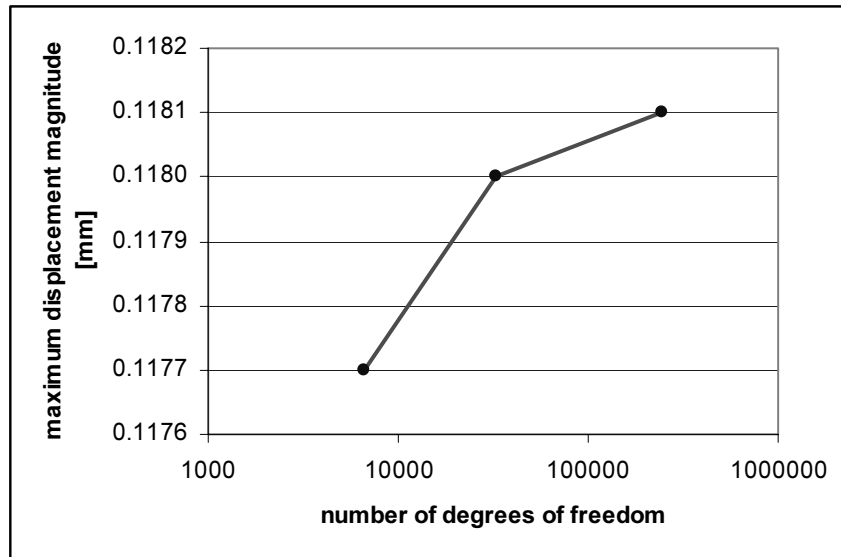

Figure 2-35: Maximum displacement magnitude

Maximum displacement magnitude is plotted as a function of the number of degrees of freedom in the mode. The three points on the curve correspond to the three models solved. Straight lines connect the three points only to visually enhance the graph.
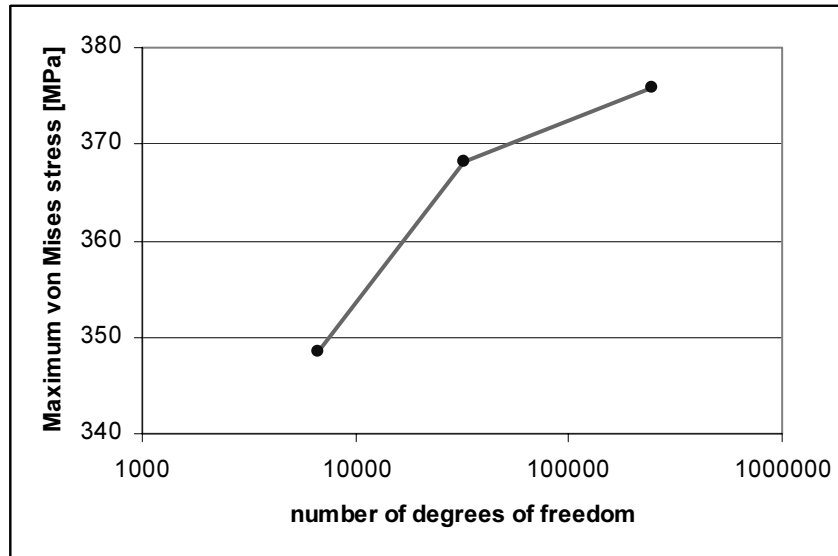
Figure 2-36: Maximum von Mises stress

Maximum von Mises stress is plotted as a function of the number of degrees of freedom in the model. The three points on the curve correspond to the three models solved. Straight lines connect the three points only to visually enhance the graph.

Having noticed that the maximum displacement increases with mesh refinement, we can conclude that the model becomes "softer" when smaller elements are used. This effect stems from the artificial constraints imposed by element definition, which become less influential with mesh refinement. In our case, because we selected second order elements, the displacement field of each element was described by the second order polynomial function. With mesh refinement, while the displacement field in each element remains a second order, and larger number of elements make it possible to approximate the real displacement and stress field more accurately. Therefore, we can say that the artificial constraints imposed by element definition become less imposing with mesh refinement.

Displacements are always the primary unknowns in FEA, and stresses are calculated based on displacement results. Therefore, stresses also increase with mesh refinement. If we continued with mesh refinement, we would see that both the displacement and stress results converge to a finite value. This would be the solution of the mathematical model. Differences between the solution of the FEA model and the mathematical model are due to discretization errors, which diminish with mesh refinement.

We will now repeat our analysis of the hollow plate by using prescribed displacements in place of a load. Rather than loading it with a 100,000 N force that has caused a 0.118 mm displacement of the loaded face, we will apply a prescribed displacement of 0.118 mm to this face to see what stresses this causes. For this exercise, we will use only one mesh with default (medium) mesh density.

Define the fourth study, called *prescribed displ*. The easiest way is to copy the definitions from the *tensile load 02* study. The definition of material properties, the built-fixed restraint to the left-side end-face and mesh are all identical to the previous design study. What we need to do is delete the load (right-click load icon and select Delete) and apply in its place prescribed displacement.

To apply the prescribed displacement to the right-side end-face, select this face and define the displacement as shown in figure 2-37. The minus sign is necessary to obtain displacement in the tensile direction.

Figure 2-37: Restraint definition window

The prescribed displacement of 0.118 mm is applied to the same face where the tensile load of 100,000 N had been applied.

Note that once prescribed displacement is defined to the end face, it overrides any load, which have been earlier applied to the same end face. While it is better to delete the load in order to keep the model clean, the load has no effect if prescribed displacement is applied to the same entity and in the same direction.

Figures 2-38 and 2-39 compare displacement and stress results for both studies.
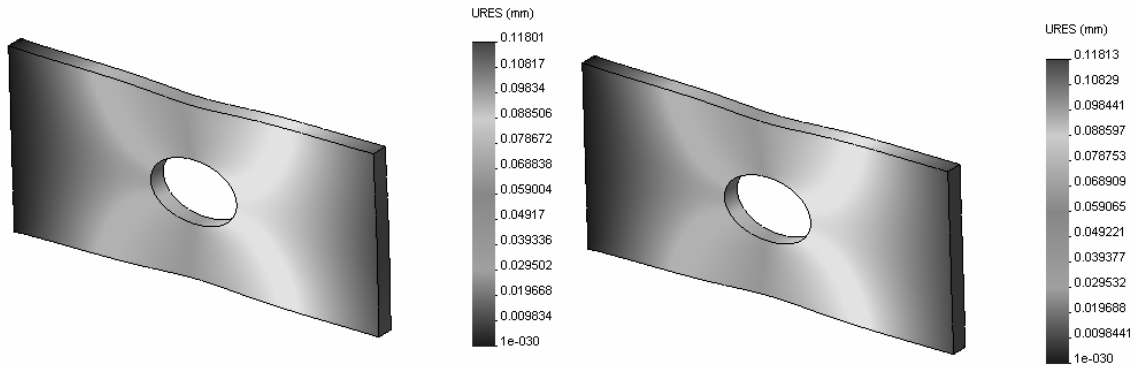

Figure 2-38: Comparison of displacement results

Displacement results in the model with the force load are displayed on the left, and displacement results in the model with the prescribed displacement load are displayed on the right.
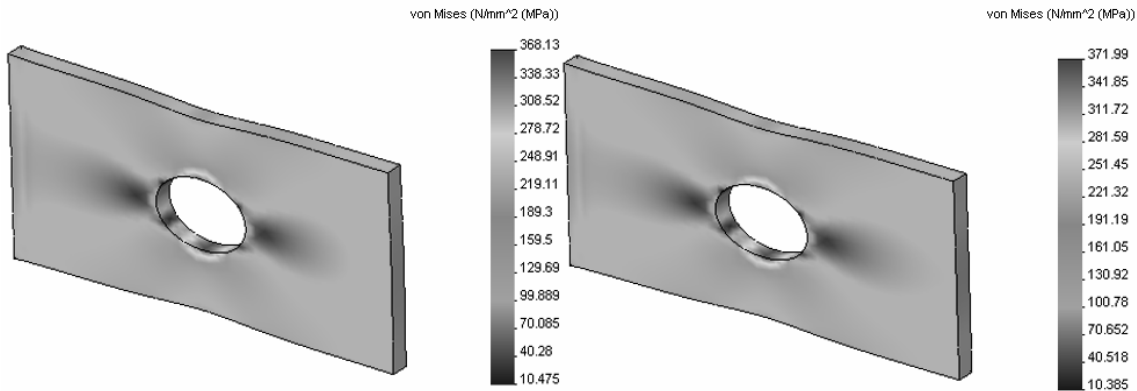

Figure 2-39: Von Mises stress results

Von Mises stress results with load applied as force are displayed on the left and Von Mises stress results with load applied as prescribed displacement are displayed on the right.

Note the numerical format of results. You can change the format in the **Chart Options** window (figure 2-23).

Results produced by applying a force load and by applying a prescribed displacement load are very similar, but not identical. The reason for this discrepancy is that in the force load model, the loaded face is allowed to deform. In the prescribed displacement model, this face remains flat, even though it experiences displacement as a whole. Also, while the prescribed displacement of 0.118 mm applies to the entire face in the prescribed displacement model, it is only seen as a maximum displacement on one point in the force load model.

We conclude our analysis of the hollow plate by examining the reaction forces. If any result plot is still displayed, hide it now (right-click the *Plot* icon and select **Hide** from pop-up window). Before accessing the reaction force results, please select the face for which we wish to obtain the reaction force results. In this case, it is the face where the fixed support was applied. Having selected the face, right-click the *Displacement* folder. A pop-up menu appears (figure 2-40). Select **Reaction Force…**.
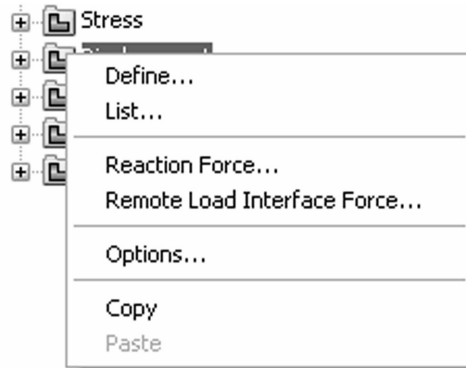

Figure 2-40: Pop-up menu associated with the *Displacement* folder

Right-click the Displacement folder to display a pop-up menu, which allows you to open the Reaction Force window.

Figure 2-41 shows the **Reaction Force** results for both studies: with force load (left) and with prescribed displacement load (right).
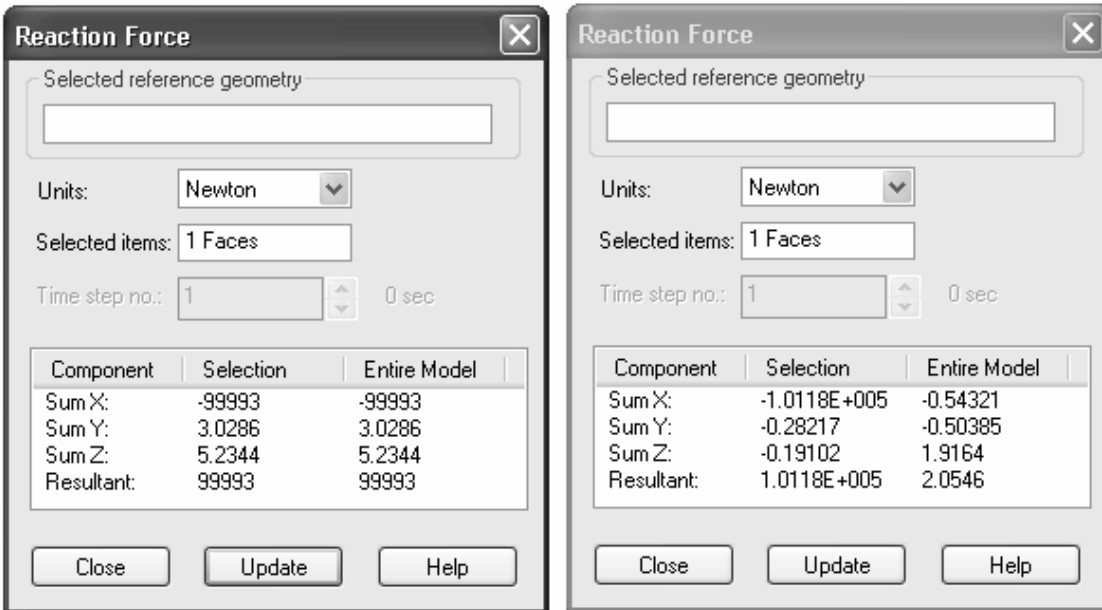

Figure 2-41: Comparison of reaction force results

Reaction forces are shown on the face where the built-in support is defined for the model with force load (left) and for the model with prescribed displacement load (right).