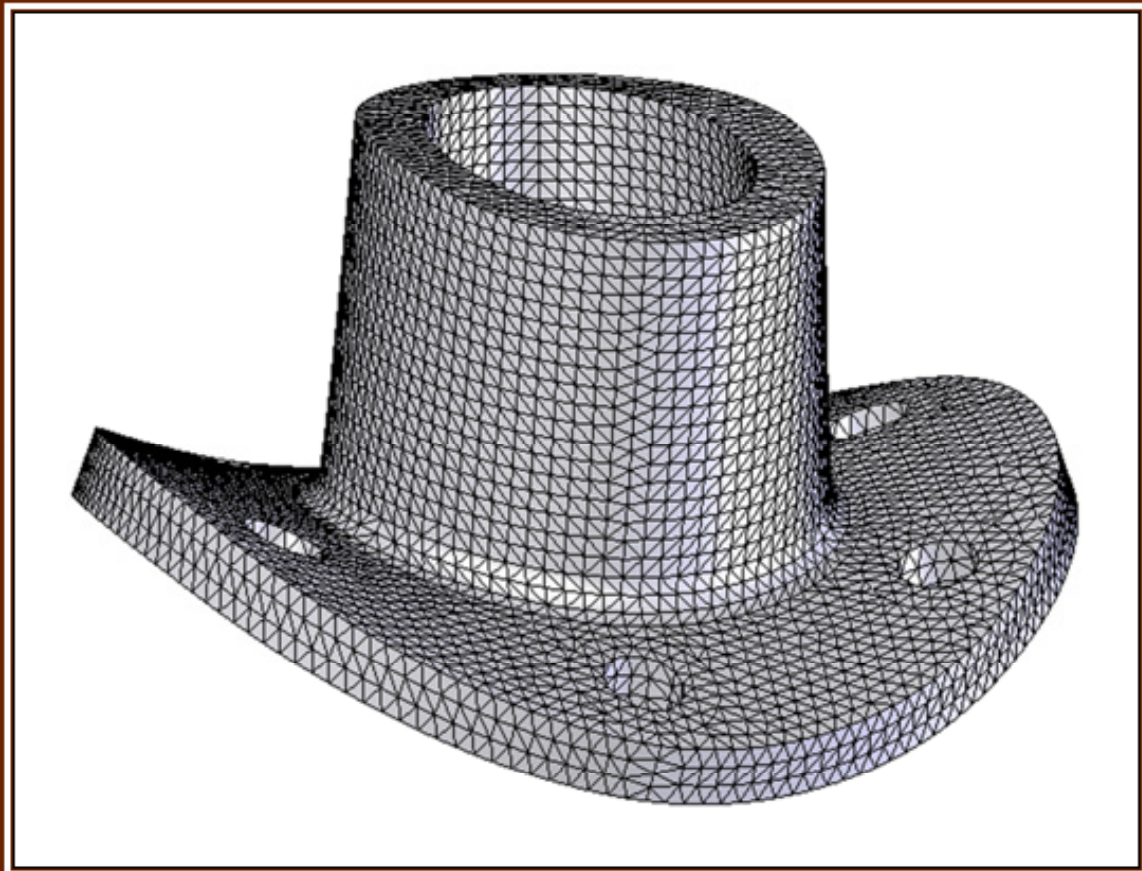# Engineering Analysis with COSMOSWorks Professional 2008

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

**SolidWorks**

**SDC**
PUBLICATIONS

**Schroff Development Corporation**

Design Generator, Inc.

# 2: Static analysis of a plate

## *Topics covered*

❑   Using COSMOSWorks interface

❑   Linear static analysis with solid elements

❑   The influence of mesh density on results

❑   Finding reaction forces

❑   Controlling discretization errors by the convergence process

❑   Presenting FEA results in desired format

## *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 100,000N tensile load is uniformly distributed along the end face, opposite to the supported face.

Fixed restraint applied to this face

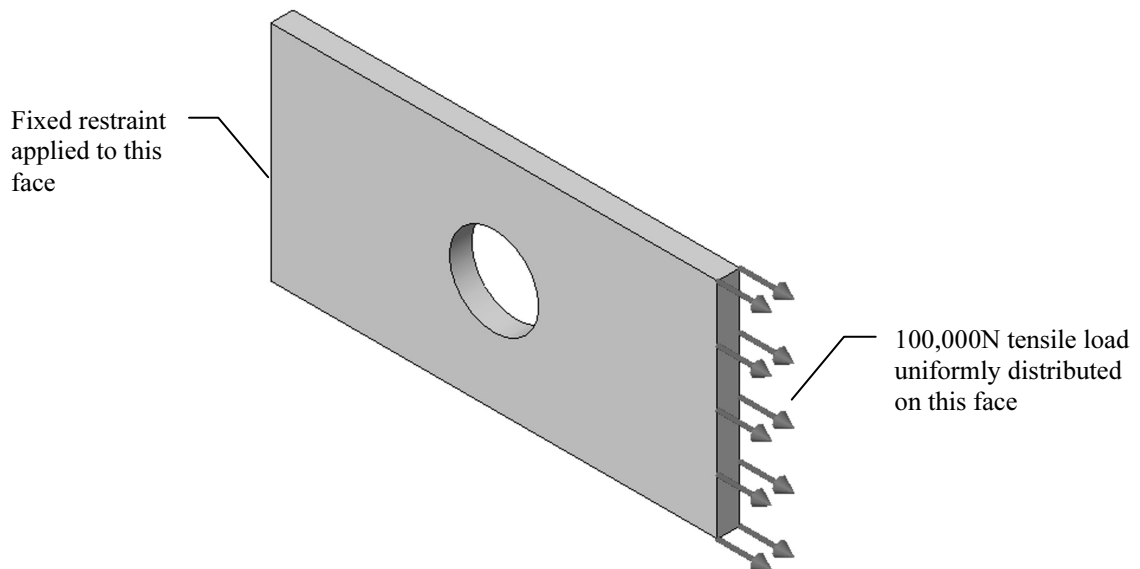100,000N tensile load uniformly distributed on this face

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.
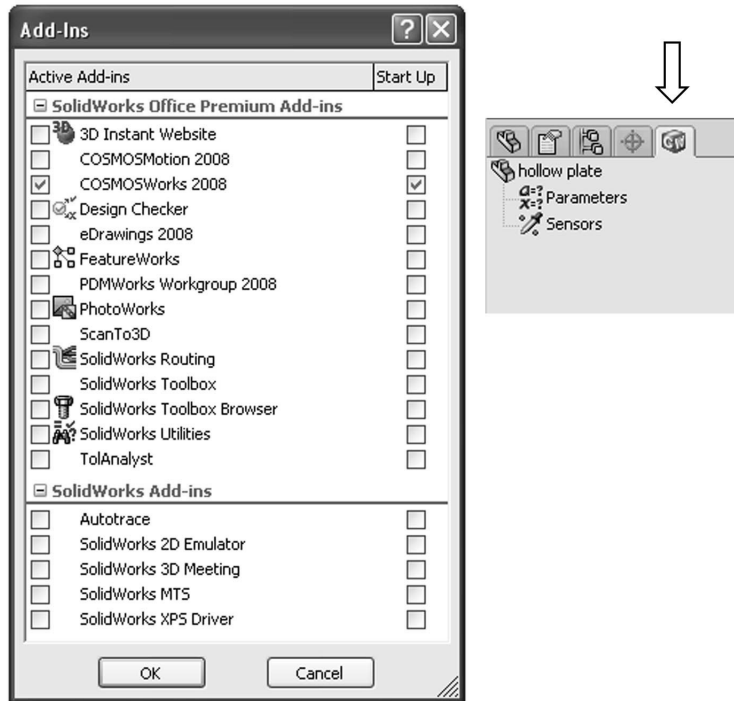


Figure 2-2: Add-Ins list and COSMOSWorks Manager tab

*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).
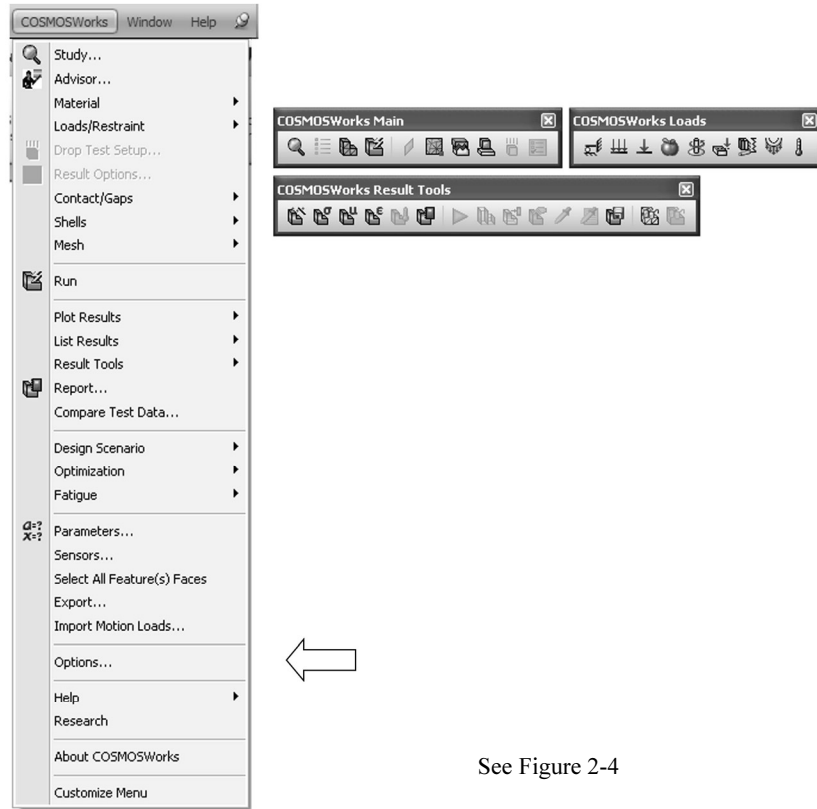
Figure 2-3: COSMOSWorks menu (left) and sample COSMOSWorks tool bars (right)

*All functions used for creating, solving, and analyzing a model can be executed either from the COSMOSWorks menu, the COSMOS tool bars, or the graphical interface in the COSMOSWorks Manager window. In this book we use the graphical interface.*

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).
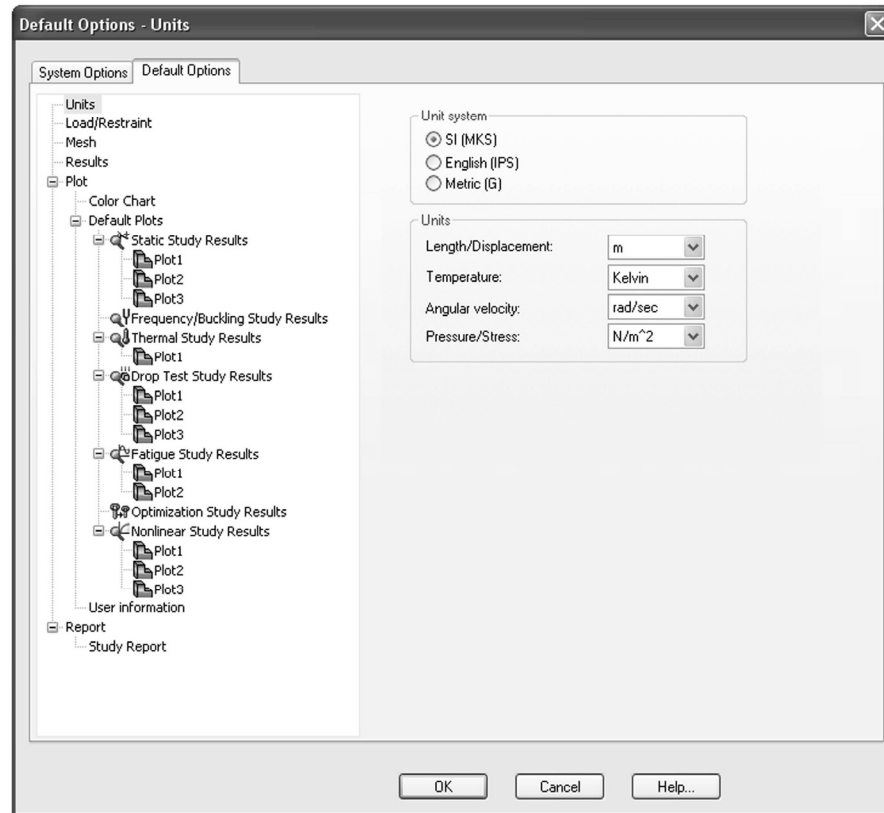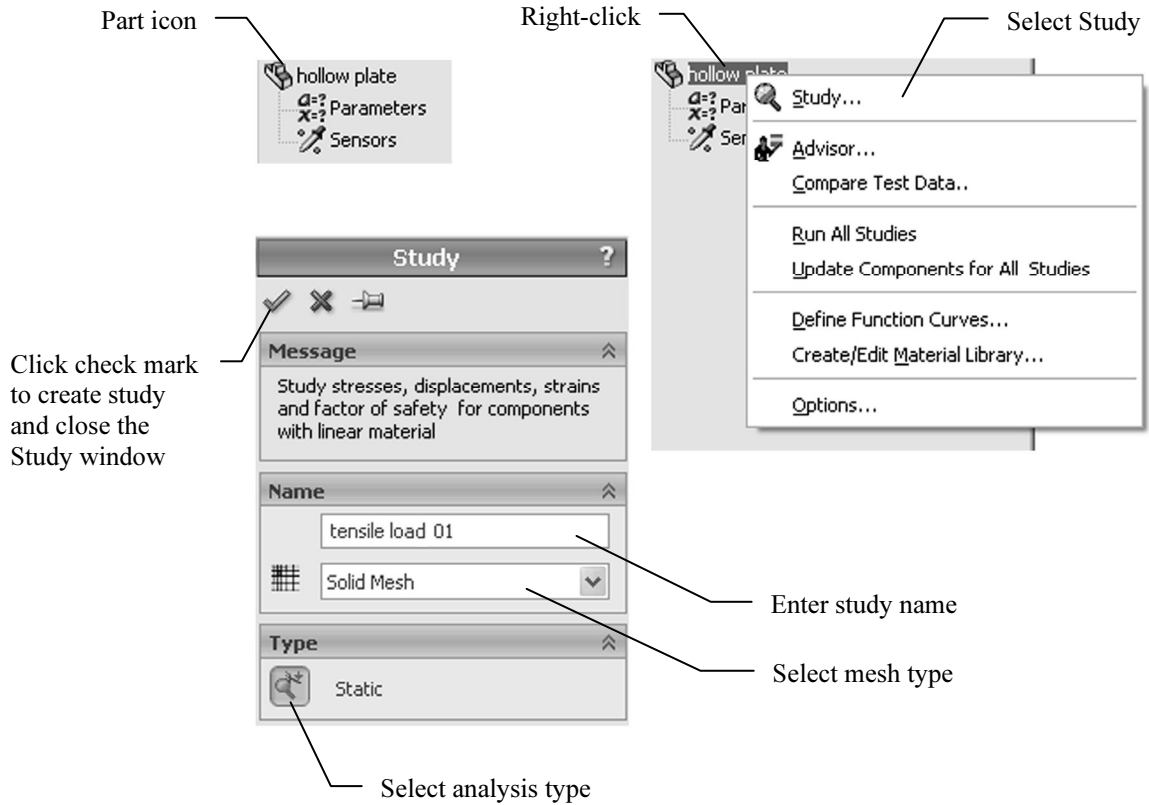


Figure 2-4: COSMOSWorks Options window; shown is Default Options tab

*The COSMOSWorks Preferences window has System Options and Default Options tabs. Review both tabs before proceeding with the exercise. Note that Default Plots can be grouped into sub-folders which are created by right-clicking on the Static Study Results folder, Thermal Study Results folder etc.*

*As shown above, we use the SI system as specified in Default Options tab.*

Creation of an FEA model always starts with the definition of a study. To define a study, right-click 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*. 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*.

Part icon

Right-click

Select Study

Click check mark
to create study
and close the
Study window

Enter study name

Select mesh type

Select analysis type

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….*

*COSMOSWorks Designer was used in this chapter, therefore the only available analysis type is Static.*

When a study is defined, COSMOSWorks automatically creates a study folder (in this case named *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.
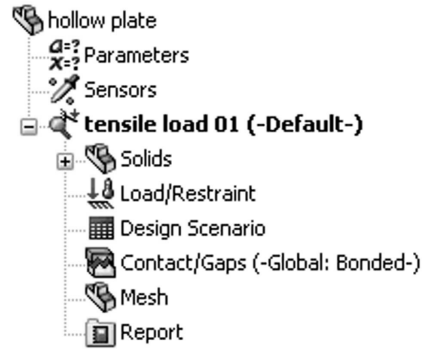
Figure 2-6: *Study* folders

*COSMOSWorks automatically creates a study folder, called tensile load 01, with the following sub folders: Solids, Load/Restraint, Design Scenario, Contact/Gaps, Mesh, and Report folder. The Design Scenario, Contact/Gaps and Report folders as well as Parameters and Sensors folders, all of which are automatically created prior to study definition will not be used in this exercise.*

We are now ready to define the analysis model. This process generally consists of the following steps:
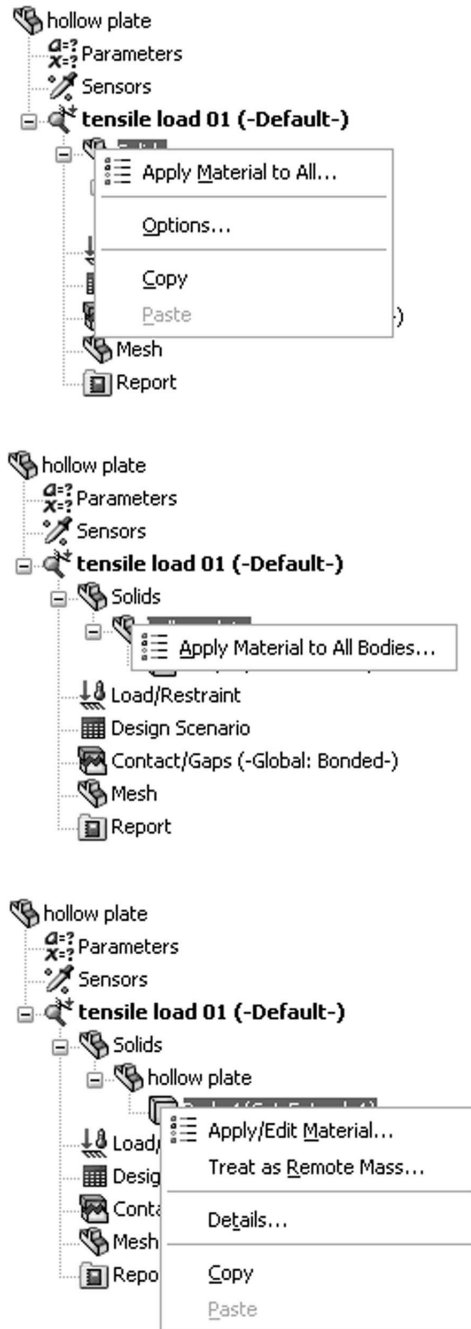
❑ Geometry preparation

❑ 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.

As illustrated in Figure 2-7, you can assign material properties to the model several ways:

❑ Right-clicking the *Solids* folder to apply material to all components in the *Solid*s folder.

❑ Right-clicking the *hollow plate* folder, which is located in the *Solids* folder to apply material to all bodies in the *hollow plate* folder.

❑ Right-clicking the *Body 1* icon, which is located in the *hollow plate* folder to apply material to *Body 1* which, in our case, is the only body in the *hollow plate* folder.

In this exercise there is only one component which consists of one body (take this opportunity to notice that COSMOSWorks supports analysis of multiple bodies). Therefore, the above three ways of applying material properties produce identical results.

hollow plate
　Parameters
　Sensors
　tensile load 01 (-Default-)

　　Apply Material to All…

　　Options…

　　Copy
　　Paste

　Mesh
　Report

hollow plate
　Parameters
　Sensors
　tensile load 01 (-Default-)
　　Solids

　　　Apply Material to All Bodies…

　Load/Restraint
　Design Scenario
　Contact/Gaps (-Global: Bonded-)
　Mesh
　Report

hollow plate
　Parameters
　Sensors
　tensile load 01 (-Default-)
　　Solids
　　　hollow plate

　　Apply/Edit Material…
　　Treat as Remote Mass…

　　Details…

　　Copy
　　Paste

Figure 2-7: *Applying material properties*

*Since the model only consists of one component with one body, the material property can be applied in any of the three ways shown above.*

33

Using the first method, **Apply Material to All**, to open the **Material** window shown in Figure 2-8.
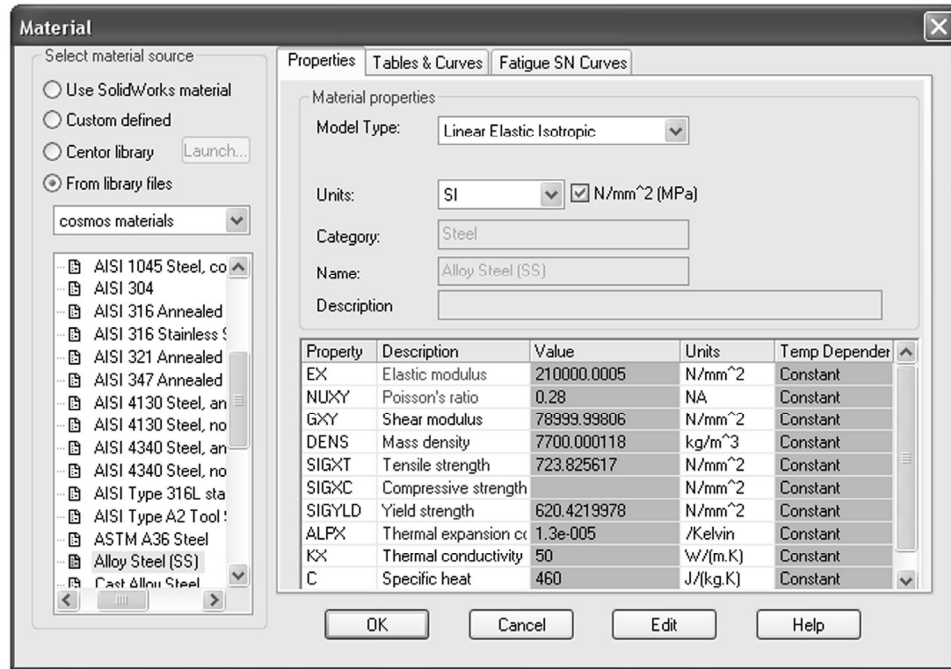


Figure 2-8: Material window

*In the Material dialog box, the properties are highlighted to indicate the mandatory and optional properties. A red description (Elastic modulus, Poisson's ratio) indicates the property that is mandatory based on the active study type and the material model. A blue description (Mass density, Tensile strength, Compressive strength, Yield strength, Thermal expansion coefficient) indicates an optional property. A black description (Thermal conductivity, Specific heat) indicates property not applicable to the current study.*

Select **From library files** in the **Select material source** area, and then select **Alloy Steel.** Select **SI** units under the **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 definition 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 bodies of a multi-body part, or to selected components of an assembly

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.

Having defined the material, we now move to defining the loads and restraints. To display the pop-up menu that lists the options available for defining loads and restraints, right-click the *Load/Restraint* folder in the *tensile load 01* study (Figure 2-9).
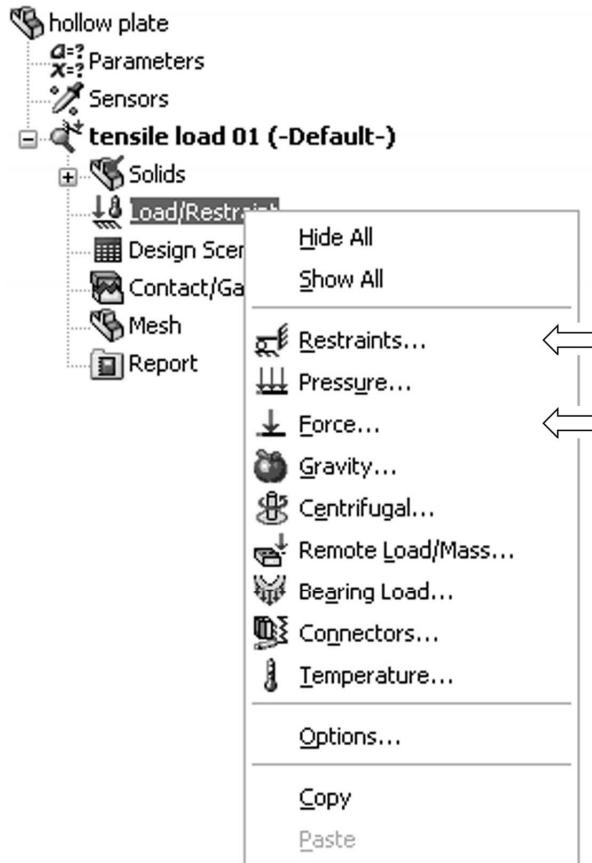


Figure 2-9: Pop-up menu for the *Load/Restraint* folder

*The arrows indicate the selections used in this exercise.*

To define the restraints, select **Restraints…** from the pop-up menu displayed in Figure 2-9. This action opens the **Restraint** window shown in Figure 2-10.

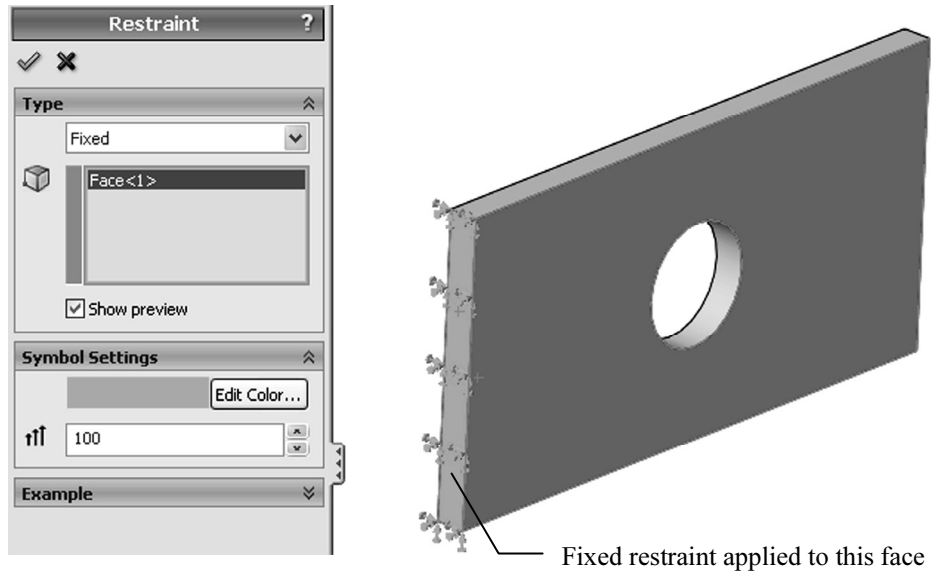Fixed restraint applied to this face

Figure 2-10: Restraint window

*In Restraints, select Fixed as Restraint type and also select the model face where the restraint is to be applied. In the graphic window, note the symbols of the applied restraint. See the table on next page for further explanation.*

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. |
| **Immovable (No translations)** | Only translational degrees of freedom are constrained, while rotational degrees of freedom remain unconstrained.<br><br>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. |
| **Symmetry** | Automatically applies symmetry boundary conditions to a flat face. Translation in the direction normal to the face is restrained and rotations about the axes aligned with the face are restrained. |
| **Roller/Sliding** | Specifies that a planar face can move freely on its plane but not in the direction normal to its plane. The face can shrink or expand under loading. |
| **Hinge** | Applies only to cylindrical face and specifies that the cylindrical face can only rotate about its own axis. The radius and the length in the axial direction of the cylindrical face remain constant under loading. This condition is similar to selecting the **On cylindrical face** restraint type and setting the radial and axial components to zero. |
| **Use reference geometry** | Restrains a face, edge, or vertex only in certain directions, while leaving the other directions free to move. You can specify the desired directions of restraint in relation to the selected reference plane or reference axis. |
| **On flat face** | 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. |
| **Cyclic symmetry** | Allows analysis of a model with circular patterns around an axis by modeling a representative segment. The segment can be a part or an assembly. The geometry, restraints, and loading conditions must be similar for all other segments making up the model. Turbines, fans, flywheels, and motor rotors can usually be analyzed using cyclic symmetry. |

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

Note that the presence of restraints 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** or **Show All** commands in the pop-up menu shown in Figure 2-9, or

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

Now define the load by selecting **Force** from the pop-up menu shown in Figure 2-9. This action opens the **Force** window, (Figure 2-11).
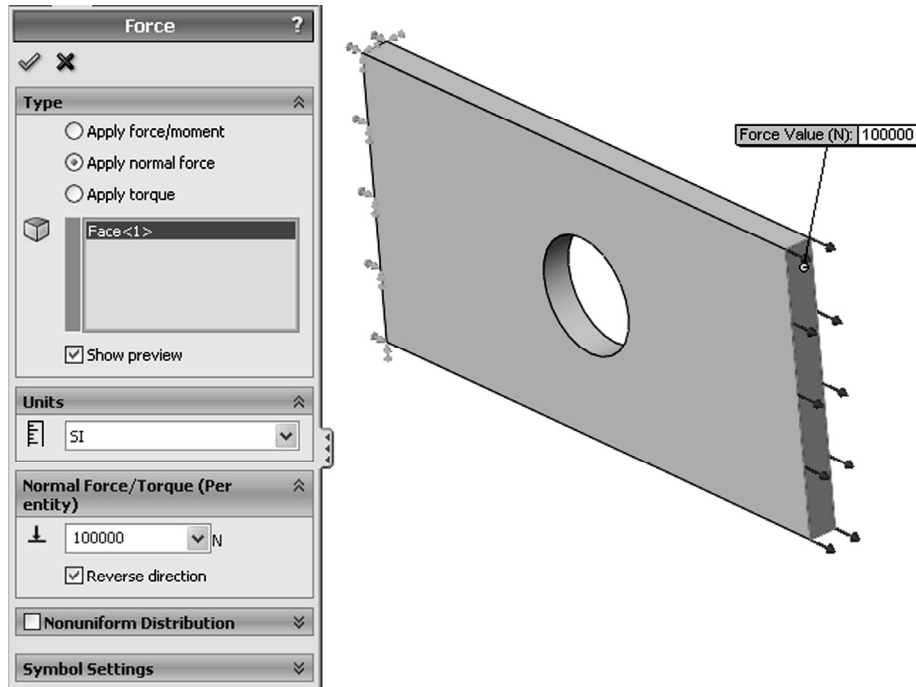


Figure 2-11: Force window

*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 a 100,000N tensile force uniformly distributed over the end face, as shown in Figure 2-11. Check **Reverse direction** option to apply a tensile load.

Generally, forces can be applied to faces, edges, and vertices 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. |
| | Note that moment can be applied only if shell elements are used. Shell elements have six degrees of freedom per node: three translations and three rotations, and can take a moment load. Solid elements only have three degrees of freedom (translations) per node and, therefore, cannot take a 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 (expressed by traction forces) 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 model is now ready for meshing. Before creating the mesh, 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 only approximately known 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 abstraction. A common error is over-constraining the model, which results in an overly stiff structure that underestimates displacements and stresses. The relative level of uncertainties in defining geometry, material, loads, and restraints is qualitatively shown in Figure 2-12.
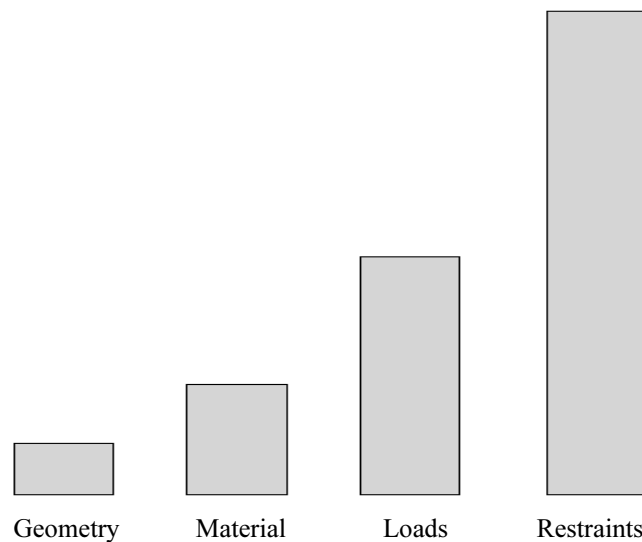
Figure 2-12: Qualitative comparison of uncertainty in defining geometry, material properties, loads, and restraints

The level of uncertainty (or the risk or error) has no relation to time required for each step, so the message in Figure 2-12 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 assume that definitions of material properties, loads, and restraints represent an acceptable idealization of real conditions. However, we need to point out that it is the responsibility of FEA user 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 **Default Options**, **Mesh** tab (Figure 2-13) that **High** mesh quality is selected. The **Options** window can be opened from COSMOSWorks main menu, as shown in Figure 2-13.
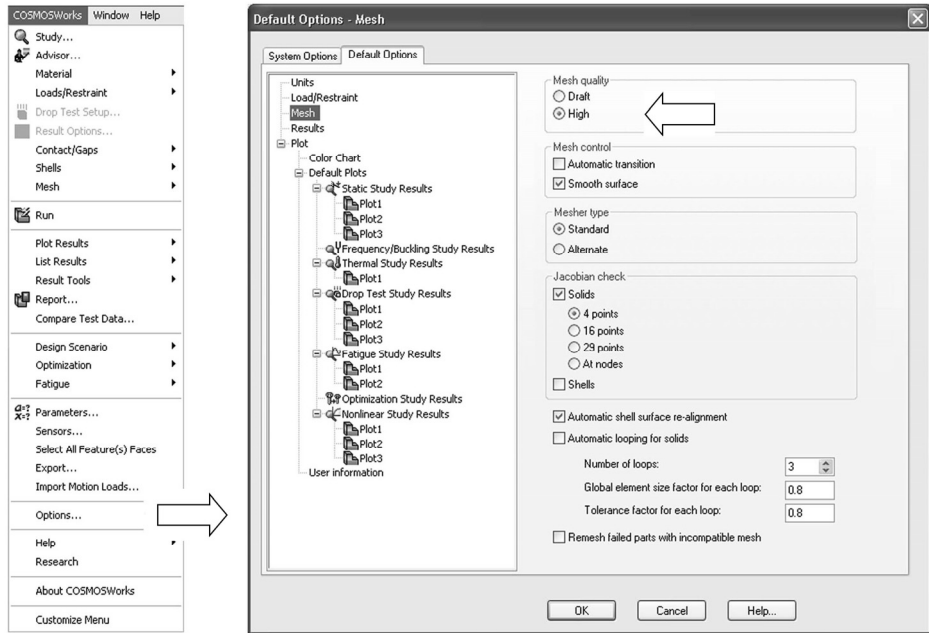


Figure 2-13: Mesh tab in the Options window

*Use this window to verify that 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 2-14).
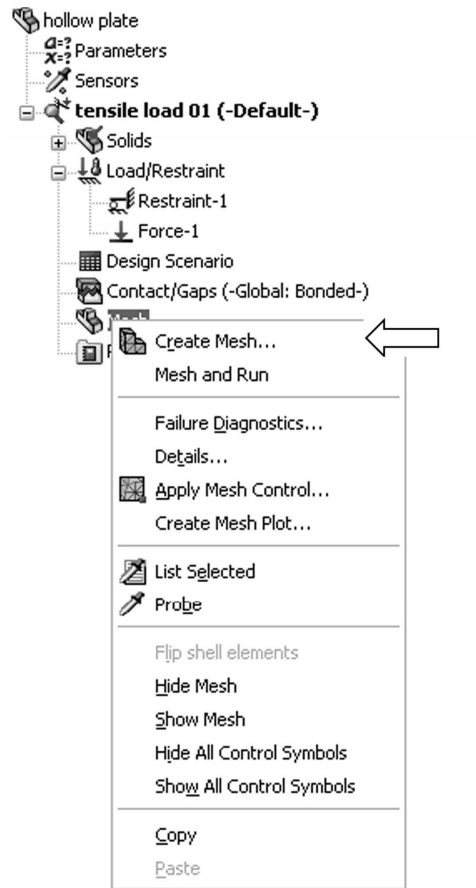


Figure 2-14: Mesh pop-up menu

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. Therefore, we will solve the same problem using three different meshes: coarse, medium (default) and fine. Figure 2-15 shows the respective selection of meshing parameters to create the three meshes.
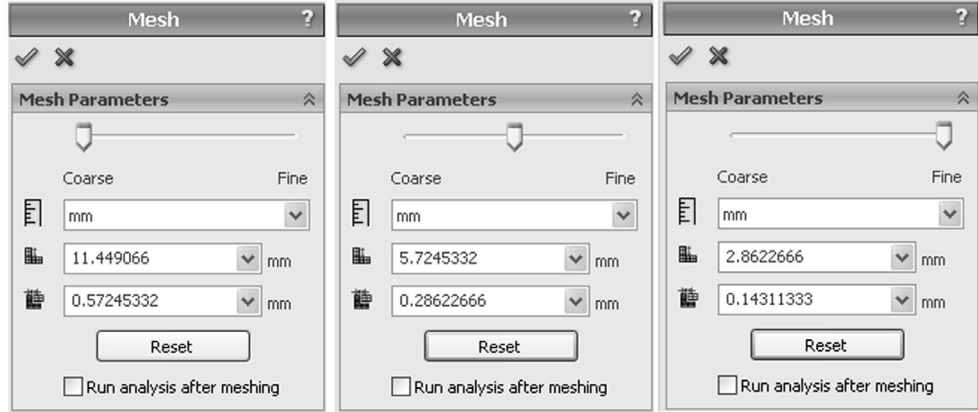
Figure 2-15: Three choices for mesh density from left to right: coarse, medium (default), and fine

The medium mesh density, shown in the middle window in Figure 2-15, is the default that COSMOSWorks proposes for meshing our model. The element size of 5.72 mm and the element size tolerance of 0.286mm 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 2-16. The default tolerance is 5% of the global element size. If the distance between two nodes is smaller than this value, the nodes are merged unless otherwise specified by contact conditions (contact conditions are not present in this model).

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 both 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.
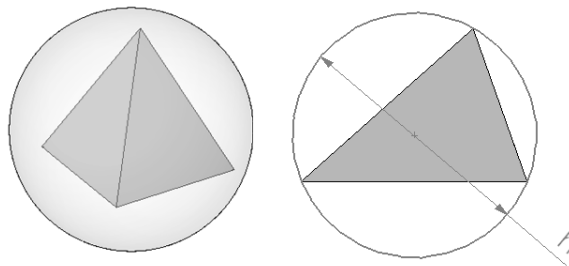


Figure 2-16: Characteristic element size for a tetrahedral element

*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 the **Mesh** window.

With the **Mesh** window open, set the slider all the way to the left (as illustrated in Figure 2-15, left) to create a coarse mesh, and click the green check mark button. The mesh will be displayed as shown in Figure 2-17.
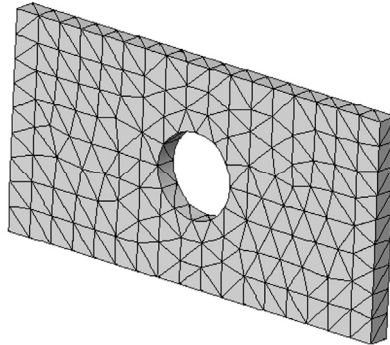


Figure 2-17: Coarse mesh created with second order, solid tetrahedral elements

*You can control the mesh visibility by selecting Hide Mesh or Show Mesh from the pop-up menu shown in Figure 2-14.*

To start the solution, right-click the *tensile load 01* study folder to display a pop-up menu (Figure 2-18). Select **Run** to start the solution.
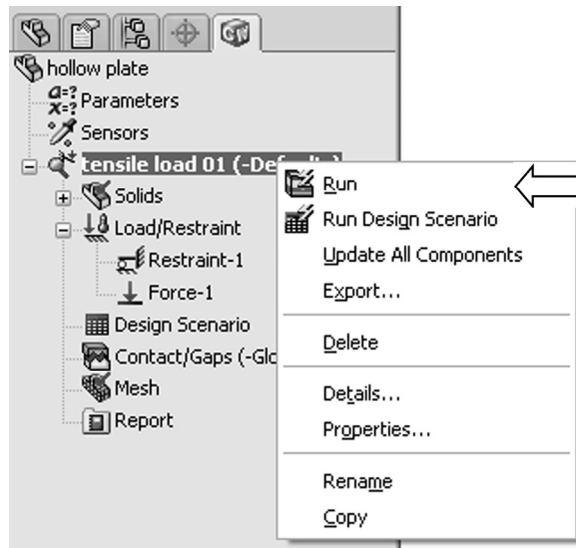


Figure 2-18: Pop-up menu for the *Study* folder

*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 2-19).
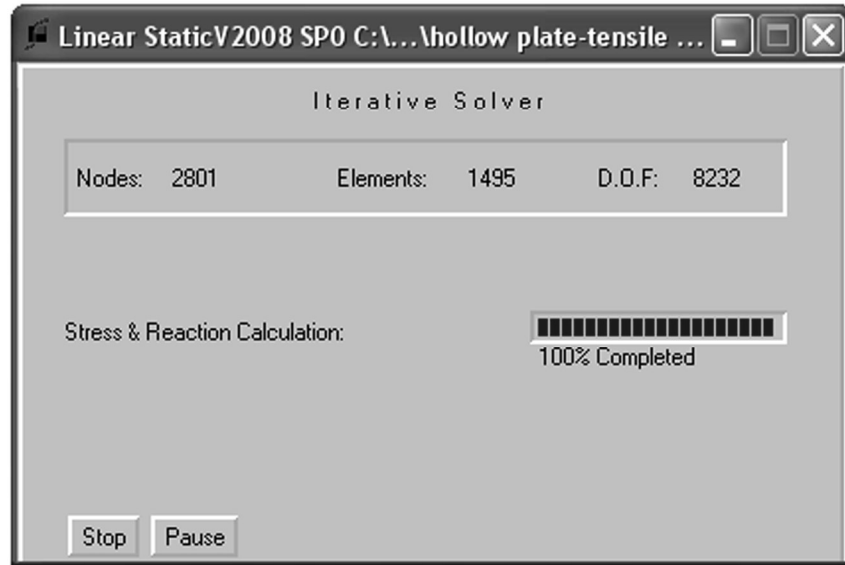


Figure 2-19: Solution Progress window

*The Solver reports solution progress while the solution is running.*

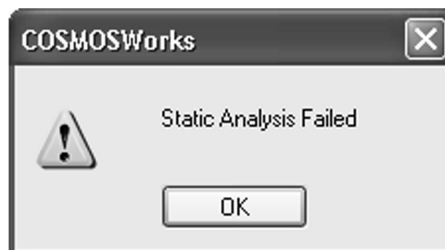If the solution fails, the failure is reported as shown in Figure 2-20.



Figure 2-20: Failed solution warning window

When the solution completes, COSMOSWorks automatically creates three plots in the *Results* folder:

❑   *Stress1*            showing von Mises stresses

❑   *Displacement1*   showing resultant displacements

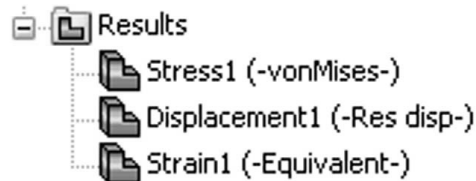❑   *Strain1*            showing equivalent strain

as shown in Figure 2-21.



Figure 2-21: Automatically created *Results* folders

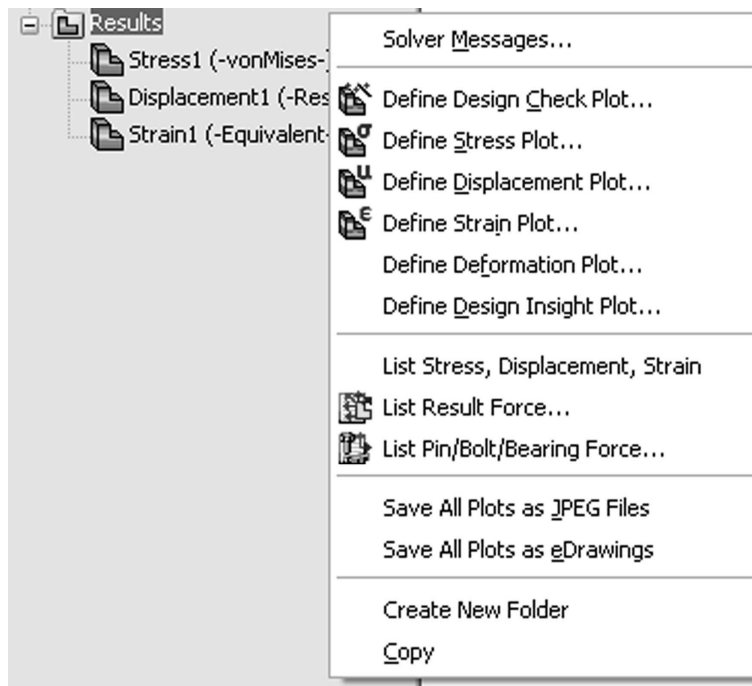If desired, you can add more plots to the *Results* folder. You can also create subfolders in the *Results* folder to group plots (Figure 2-22).



Figure 2-22: More plots and folders can be added to *Results* folder

*Right-clicking on Results activates this pop-up menu from where plots and folders may be added to the Results folder.*

To display stress results, double-click on the *Stress1* icon in the *Results* folder or right-click it and select **Show** from the pop-up menu. The default stress plot is shown in Figure 2-23.
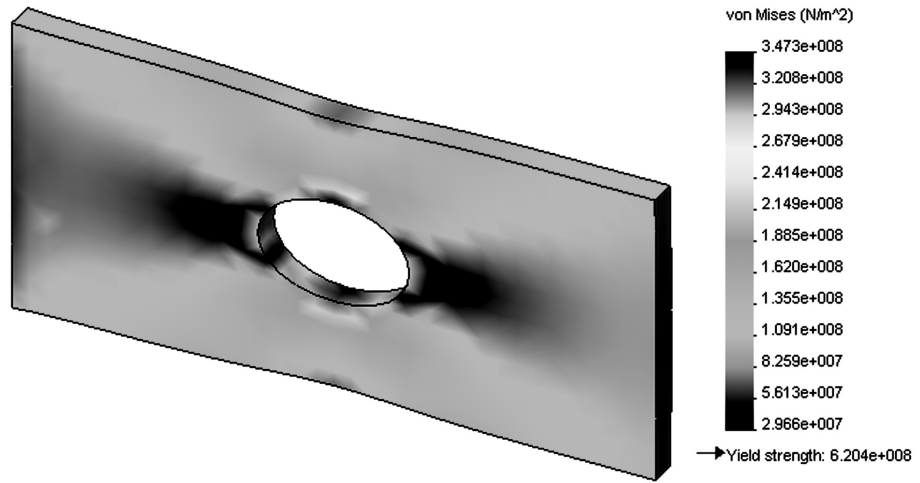


Figure 2-23: Stress plot displayed using default stress plot settings

*Von Mises stress results are shown by default in the stress plot window. Notice that results are shown in [Pa] and the highest stress 347.3x10$^8$ Pa is below the material yield strength 620.4x10$^8$ Pa. The actual numerical results may differ slightly depending on solver, software version and service pack used.*

Once the stress plot is showing, right-click the stress plot icon to display the pop-up menu featuring different plot display options (Figure 2-24).
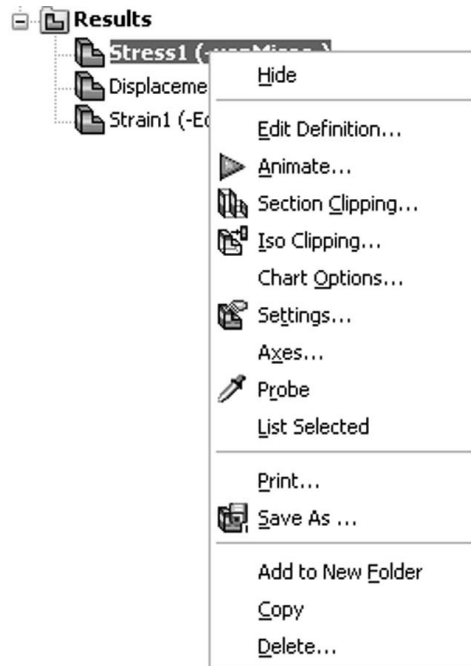
Figure 2-24: Pop-up menu with plot display options

*Any 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 2-24.

Select **Edit Definition** to open the **Stress Plot** window, change the units to MPa then close the window. In the **Chart Options** window, change the range of the 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 2-25.
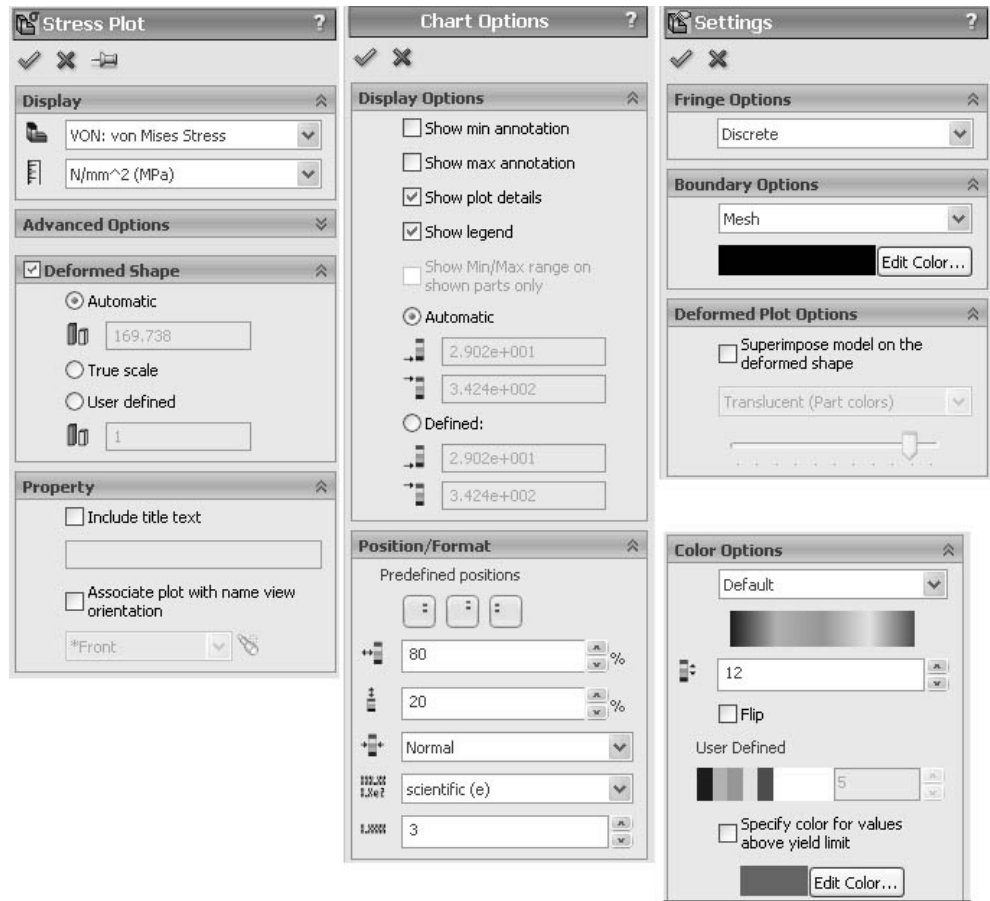
Figure 2-25: **Stress Plot** (invoked by **Edit Definition…**), **Chart Options** and **Settings** windows we use in this exercise to modify stress plot display

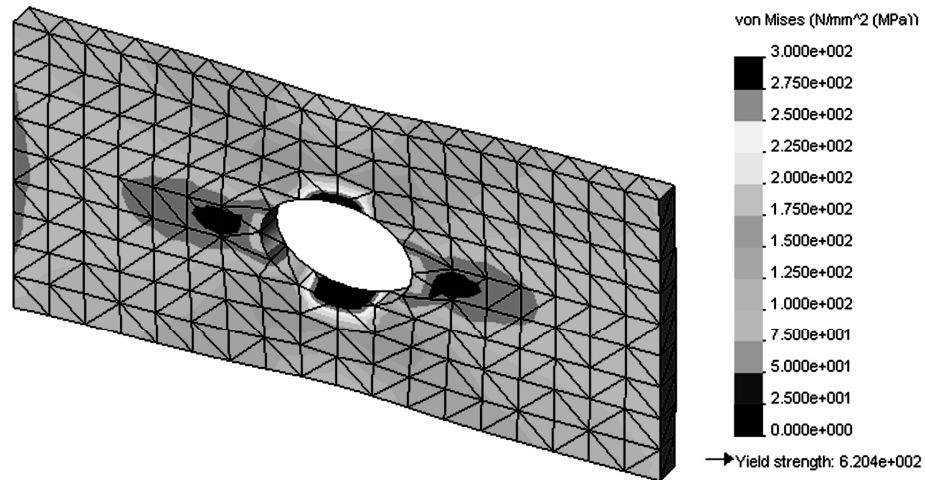*Color Options section belongs to the Chart Options window*

von Mises (N/mm^2 (MPa))

- 3.000e+002
- 2.750e+002
- 2.500e+002
- 2.250e+002
- 2.000e+002
- 1.750e+002
- 1.500e+002
- 1.250e+002
- 1.000e+002
- 7.500e+001
- 5.000e+001
- 2.500e+001
- 0.000e+000

→ Yield strength: 6.204e+002

Figure 2-26: Modified stress plot results

*The modified stress plot using selections shown in windows in Figure 2-25.*

The Stress plot in Figure 2-26 shows node values, also called averaged stresses. Element values (or non-averaged stresses) can 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 in Chapter 22 for more comments on node and element values of stress results.

Before you proceed, investigate other selections available in the windows shown in Figure 2-24 and the pop-up menu shown in Figure 2-23.

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 2-27 (displacement)

❑ Figure 2-28 (strain)

❑ Figure 2-29 (deformation)

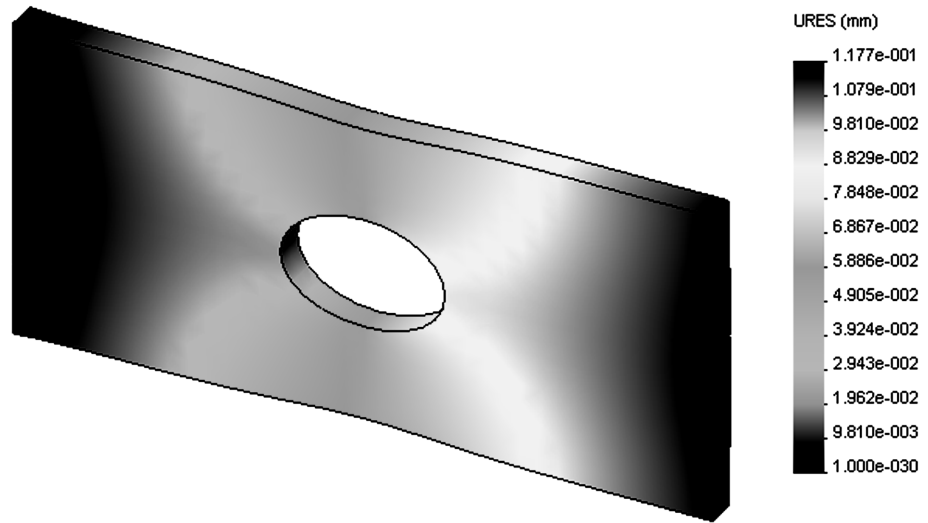Before proceeding, create a deformation plot using the pop-up menu in Figure 2-22.

Figure 2-27: Displacement results using Continuous fringe options

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

The plot in Figure 2-27 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-clicking plot icon, then selecting **Edit Definition**.
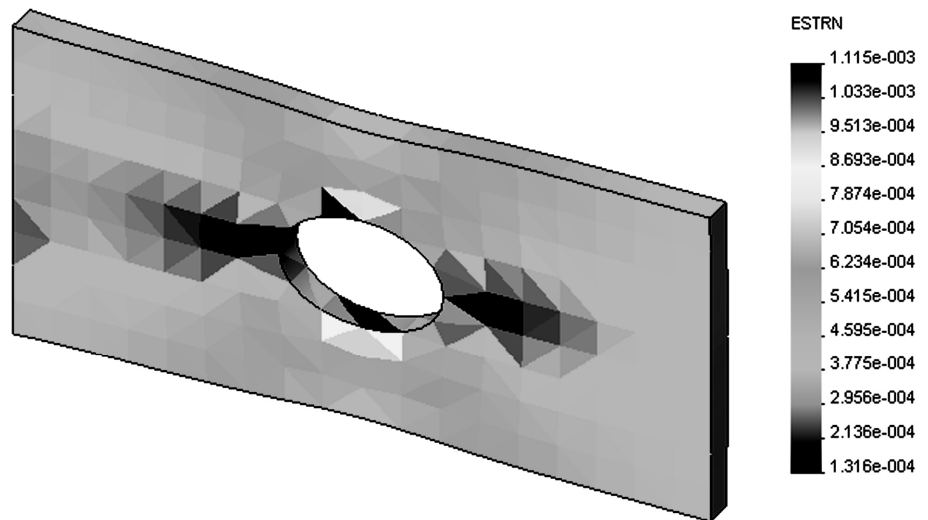


Figure 2-28: Strain results

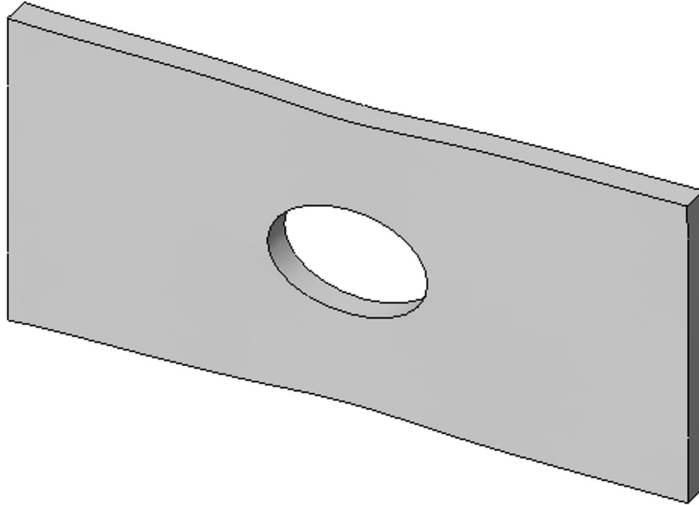*Strain results are shown here using Element values.*

Figure 2-29: Deformation results

*Note that all previous results were also showing the model in a deformed shape, because the deformed shape display was selected in the plot definition.*

Now, define **Design Check Plot** using the menu shown in Figure 2-22. The definition of Design Check Plot proceeds in three steps. Follow step 1, step 2, and step 3, as shown in Figure 2-30.
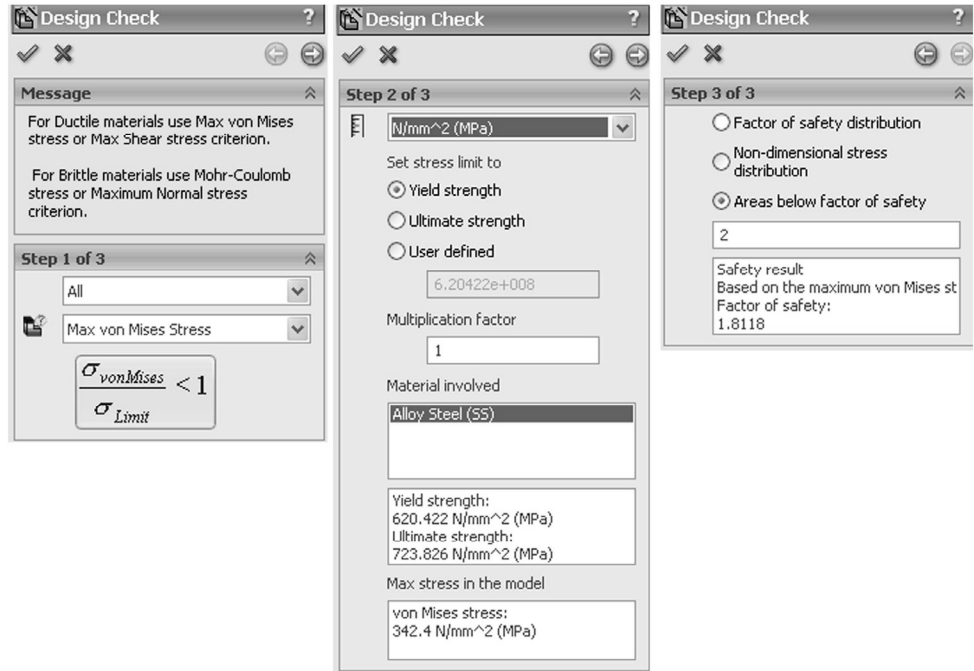
Figure 2-30: Three windows show three steps in the Design Check plot definition

*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).*
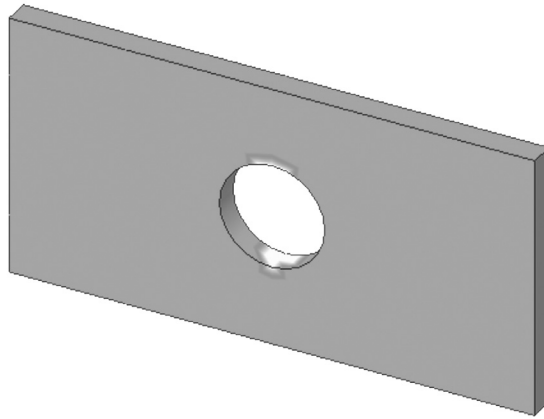
Figure 2-31: Red color (shown as white in this grayscale illustration) displays the areas where the factor of safety falls below 2

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 2-15. All three meshes used in this exercise (coarse, medium, and fine) are shown in Figure 2-32.
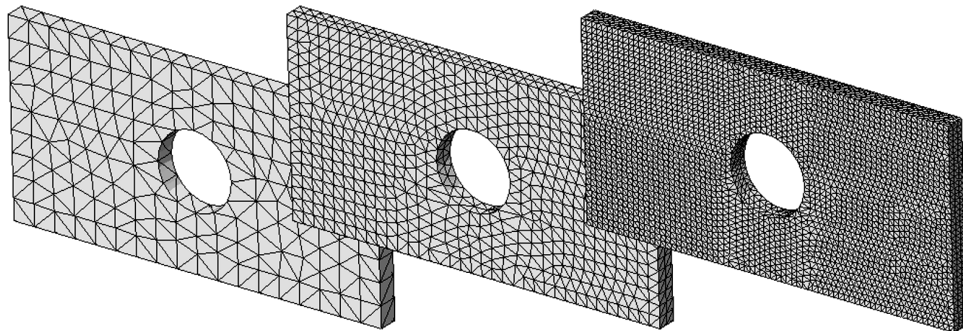


Figure 2-32: Coarse, medium, and fine meshes

*Three meshes used to study the effects of mesh density on 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 model.

The information on the number of nodes and number of elements can be found in **Mesh Details** (Figure 2-33).
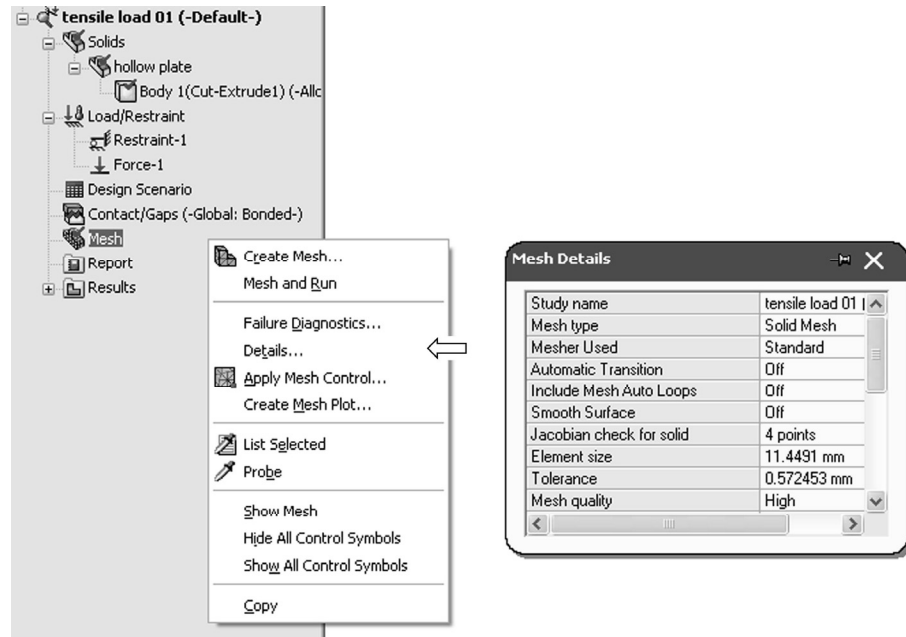


Figure 2-33: Meshing details window

*Right-click on Mesh, Details… to display the Mesh Details window.*

The number of degrees of freedom is shown in the COSMOS data base. Look for a file with the extension OUT, which is located in the **Results** folder specified in the **Options** window under **Default Options** (Figure 2-34).
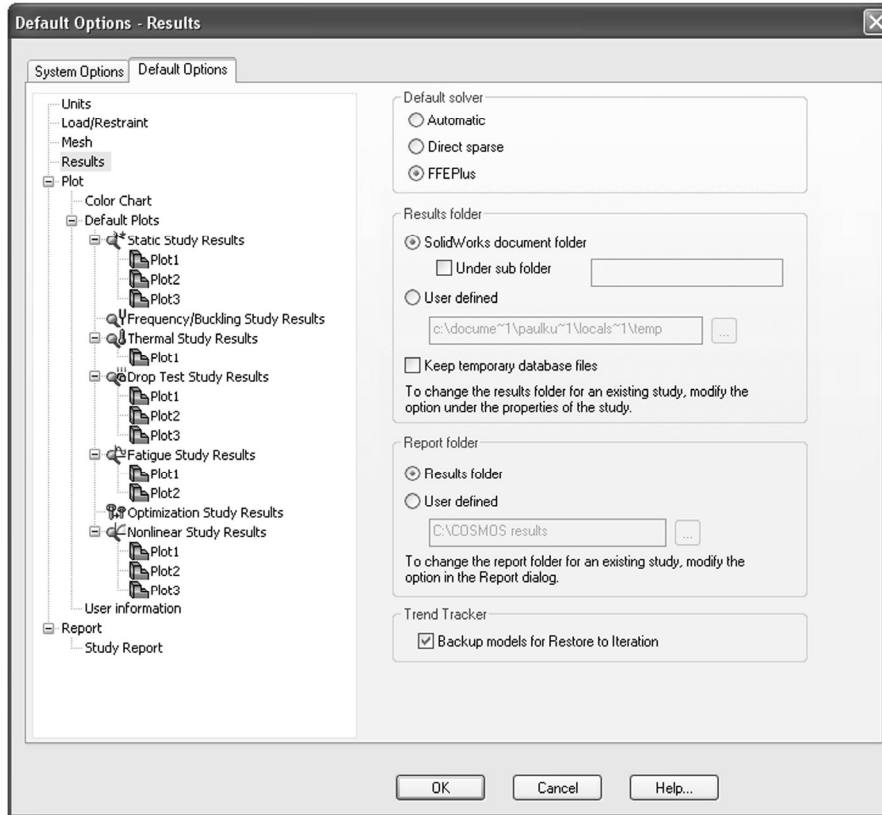
Figure 2-34: COSMOSWorks database is located in Work directory specified under Results Options in Options window

*Here, the SolidWorks document folder is used to store analysis results.*

Using Windows Explorer, find the file named hollow plate-tensile load 01.OUT. Open it with a text editor (e.g. Notepad) and find the number of degrees of freedom (referred to in the file as the number of equations).

Note that the OUT file is available only while the model is opened in COSMOSWorks. Upon exiting from COSMOSWorks (which is done by means of deselecting COSMOSWorks from the list of add-ins, or by closing the SolidWorks model), all database 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 convenient backup of COSMOSWorks results.

Yet another and most convenient way to find the number of nodes, elements, and degrees of freedom is to use the pop-up menu activated by right-clicking on the *Results* folder (Figure 2-22). Right-click on **Solver Messages** to open the **Solver Messages** window showing, among other information, the number of nodes, elements and degrees of freedom (Figure 2-35).
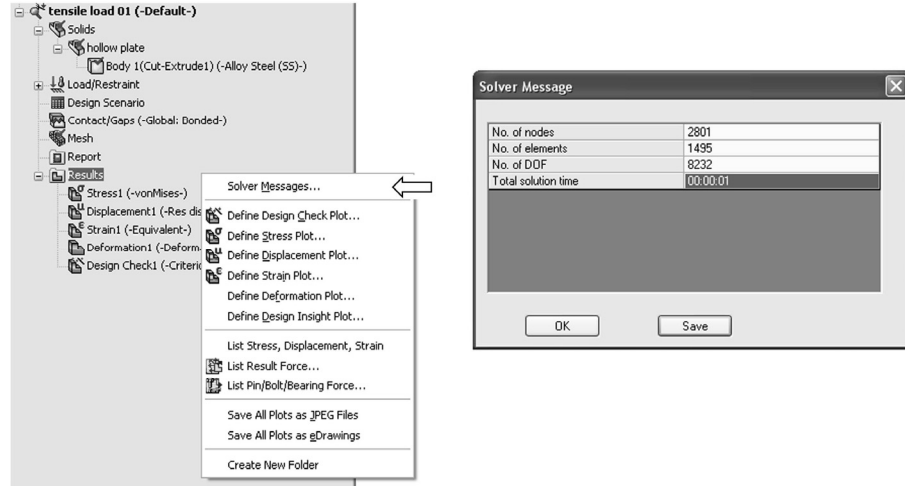
Figure 2-35: Solver Message window lists information pertaining to the solved study.

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 2-15.

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, then select **Copy**. Next, right-click the model icon (hollow plate), and select **Paste** to invoke **Define Study Name** window, where the name of a new study is defined as shown in (Figure 2-36).
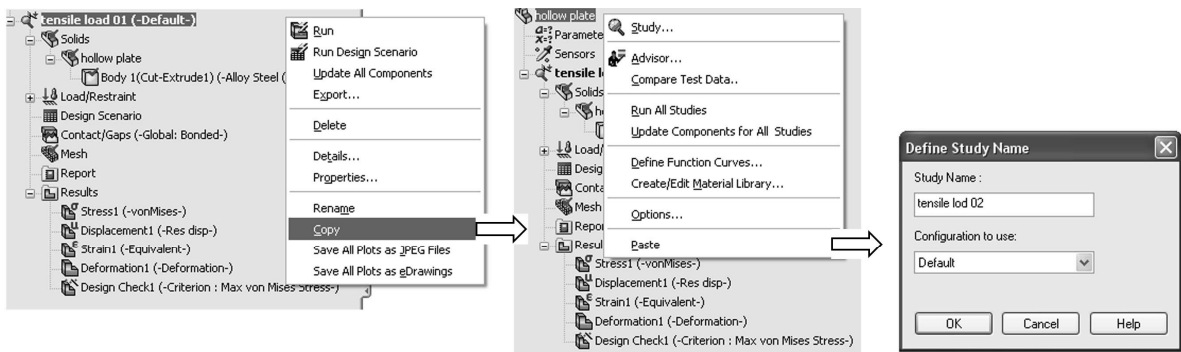


Figure 2-36: A study can be copied into another study in three steps as shown

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 the *hollow plate* icon (top of COSMOS Manager).

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

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 2-37.
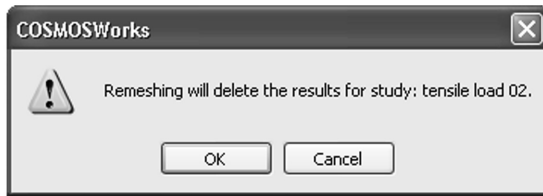


Figure 2-37: Remeshing deletes any existing results in the study

The summary of results produced by the three models is shown in Figure 2-38.

| Mesh density | Max. displ. magnitude [mm] | Max. von Mises stress [MPa] | Number of degrees of freedom | Number of elements | Number of nodes |
|---|---|---|---|---|---|
| Coarse | 0.1178 | 348 | 8313 | 1516 | 2828 |
| Medium | 0.1180 | 378 | 36285 | 7082 | 12228 |
| Fine | 0.1181 | 376 | 250473 | 54889 | 83986 |

Figure 2-38: 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 in studies tensile load 01, tensile load 02, and tensile load 03.*

*The actual number in this table may vary slightly depending on the type of solver and the release of software used for the solution. Maximum von Mises stress magnitude drops in the third study because of accumulated solution error.*

Figures 2-39 and 2-40 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.
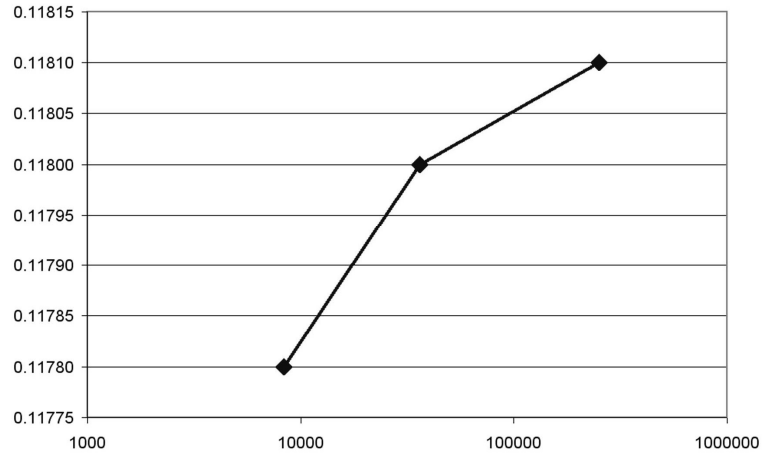
Figure 2-39: Maximum displacement magnitude [mm]

*Maximum displacement magnitude 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.*
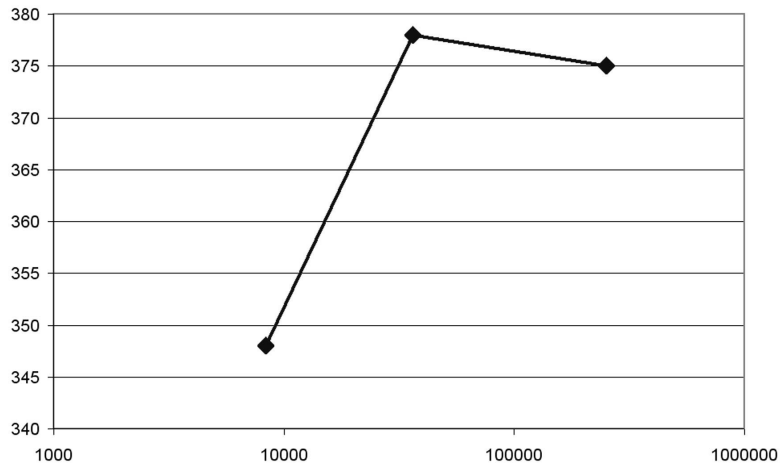


Figure 2-40: Maximum von Mises stress [MPa]

*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. With mesh refinement, a larger number of elements allows for better approximation of the real displacement and stress field. 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 structural 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 which is 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 100,000N 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 to do this is to copy the definitions from the ***tensile load 02*** study. The definition of material properties, the fixed restraint to the left-side end-face and mesh are all identical to the previous design study. We need to delete the load (right-click the load icon and select **Delete**) and apply in its place the prescribed displacement.

To apply the prescribed displacement to the right-side end-face, open the **Restraint** window, select the face and define displacement as shown in Figure 2-38. Check *Reverse direction* to obtain displacement in the tensile direction. Note that the direction of a prescribed displacement is indicated by a restraint symbol. The size and color of this symbol can be changed using **Symbol Settings** (Figure 2-41). The color and size of all load and restraint symbols is controlled the same way.
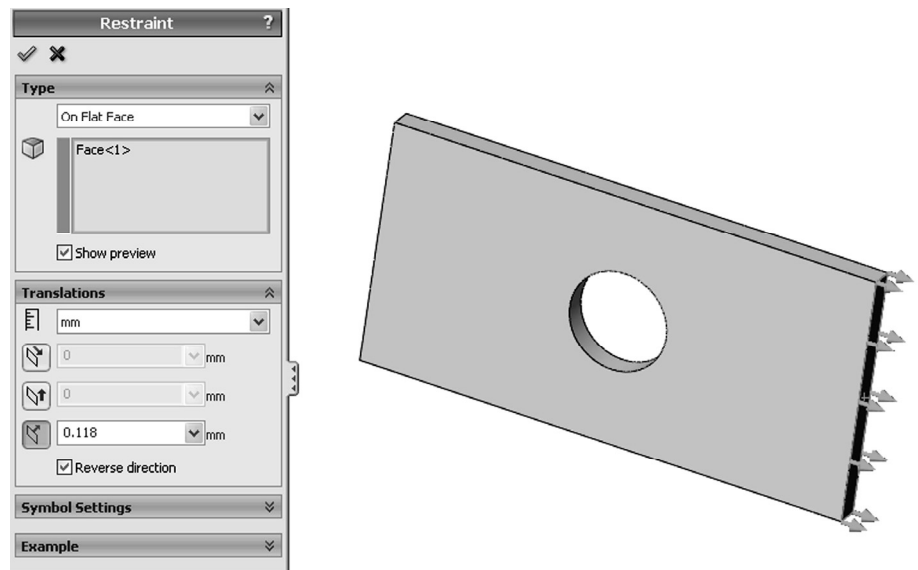


Figure 2-41: Restraint definition window

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

Once prescribed displacement is defined to the end face, it overrides any previously applied loads 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 a prescribed displacement is applied to the same entity and in the same direction.

Figures 2-42 and 2-43 compare displacement and stress results for the model loaded with force to the model loaded with prescribed displacement.
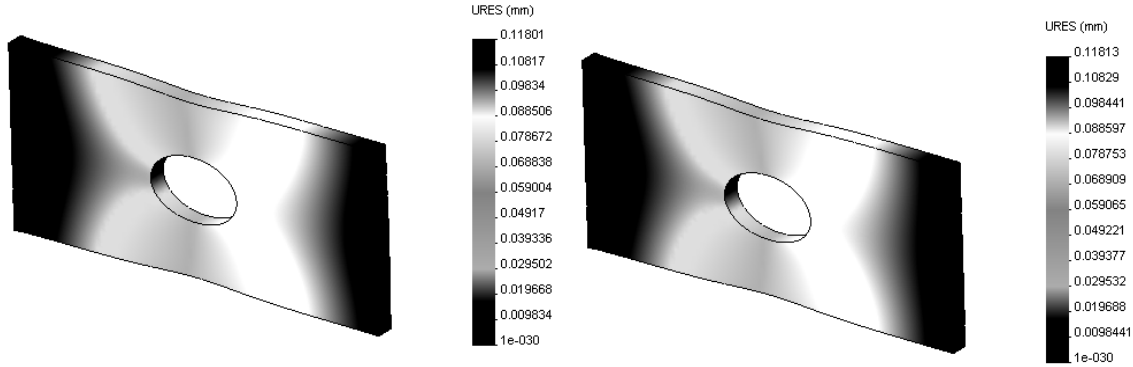


Figure 2-42: Comparison of displacement results

*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.*
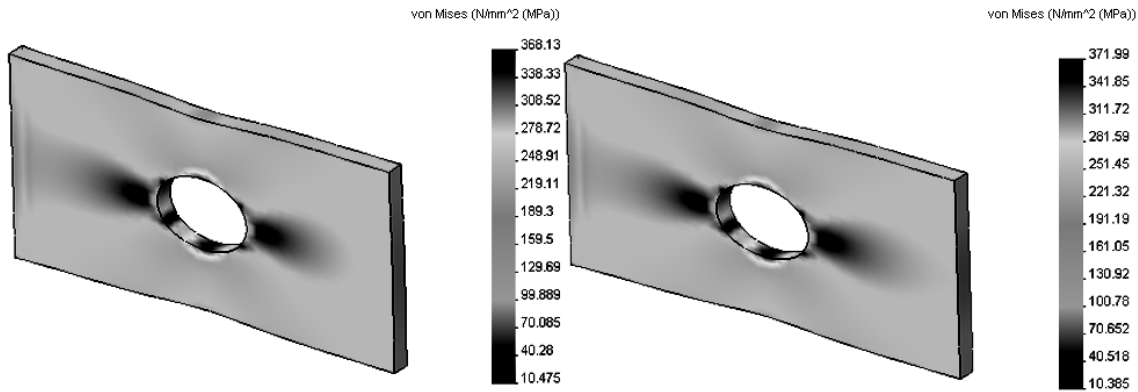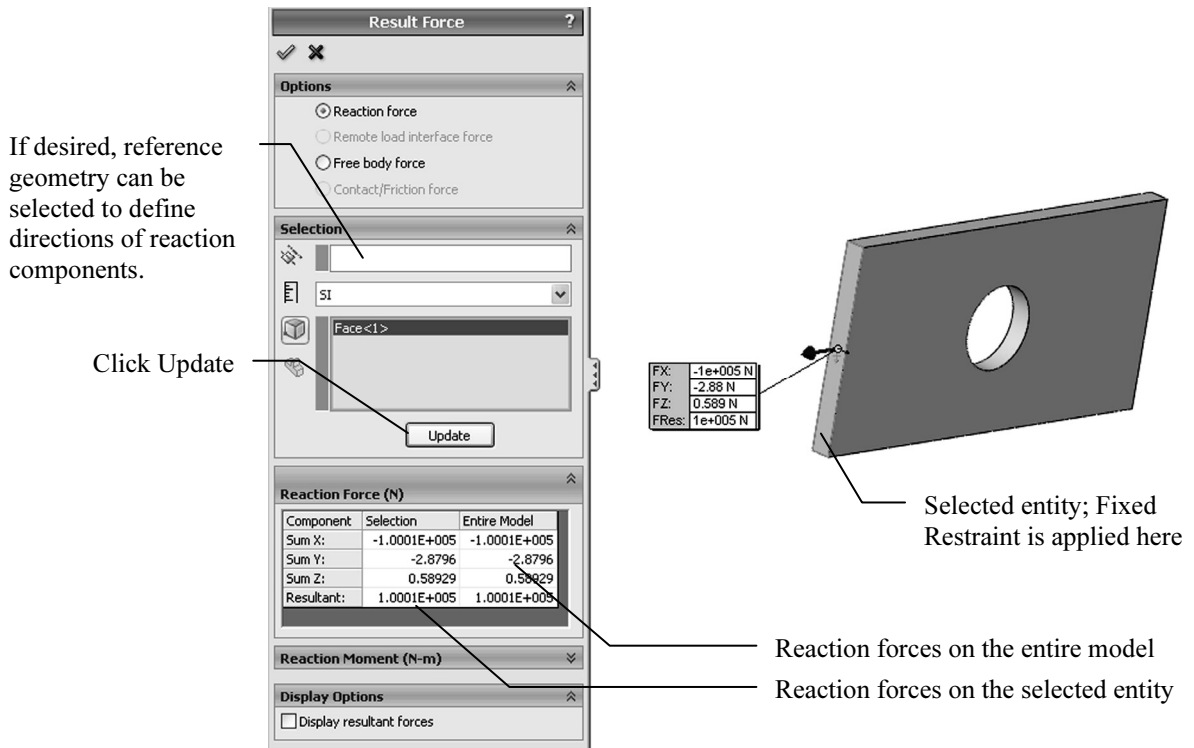


Figure 2-43: Comparison of von Mises stress results

*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 2-25).

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 in one point in the force load model.

We conclude our analysis of the hollow plate by examining the reaction forces. Let's use results of study *tensile load 02* for this analysis. Open the study *tensile load 02* folder, and right-click *Results.* From the pop-up menu select *List Reaction Force* to open the **Result Force** window. Select the face where the fixed restraint is applied and click the **Update** button. Information on reaction forces will be displayed as shown in Figure 2-44.



If desired, reference geometry can be selected to define directions of reaction components.

Click Update

Selected entity; Fixed Restraint is applied here

Reaction forces on the entire model

Reaction forces on the selected entity

Figure 2-44: Result Force window

*Reaction forces can also be displayed in components other than those defined by the global reference system. To do this, reference geometry such as plane or axis must be selected.*