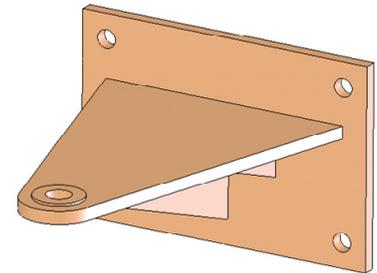


In this tutorial, we will use the COSMOSWorks finite element analysis (FEA) program to analyze the response of a component to an applied load. Finite element analysis is a powerful tool that allows engineers to quickly analyze and refine a design. It can be applied to problems involving vibrations, heat transfer, fluid flow, and many other areas. The most common use of FEA is in structural analysis, and this introductory tutorial will be limited to that use.

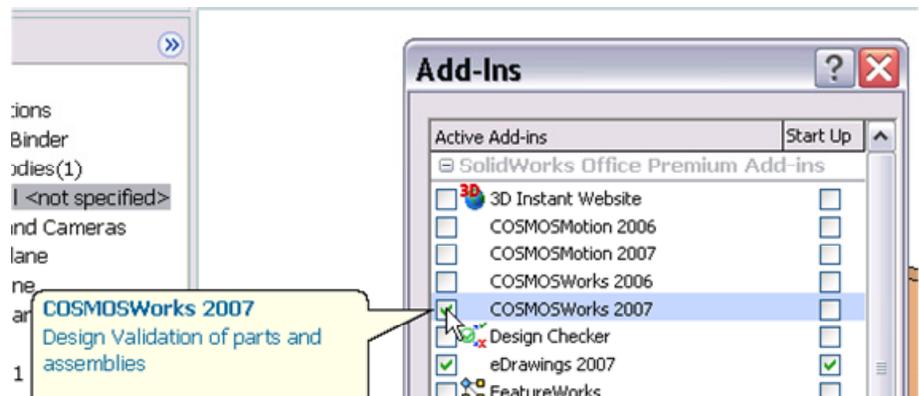
There has been much discussion during the past decade over who should be using FEA software. As the software has become easier to use, the potential for misuse has risen. An inexperienced user can quickly obtain results, but the interpretation of the results requires knowledge of the applicable engineering theories. In this tutorial, we will point out where choices and assumptions are made that could affect the accuracy of the results.

The part to be analyzed is the bracket from the tutorial of Chapter 3.

**Open the part file. From the main menu, select Tools: Add-Ins and check the COSMOSWorks 2007 box.**



If you check the box to the right of the add-in name, then that add-in will be activated whenever SolidWorks is started. Most users will prefer to activate COSMOSWorks only when it is needed for an analysis.

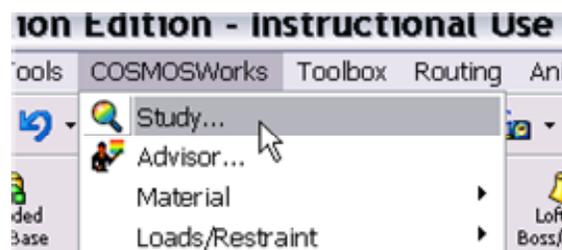


When COSMOSWorks is activated, a new menu item is created, and a tab for the analysis manager is added above the FeatureManager.



**Click the analysis tab. From the main menu, select COSMOSWorks: Study.** A study defines a specific analysis and its results. A single part file can have multiple studies associated with it.

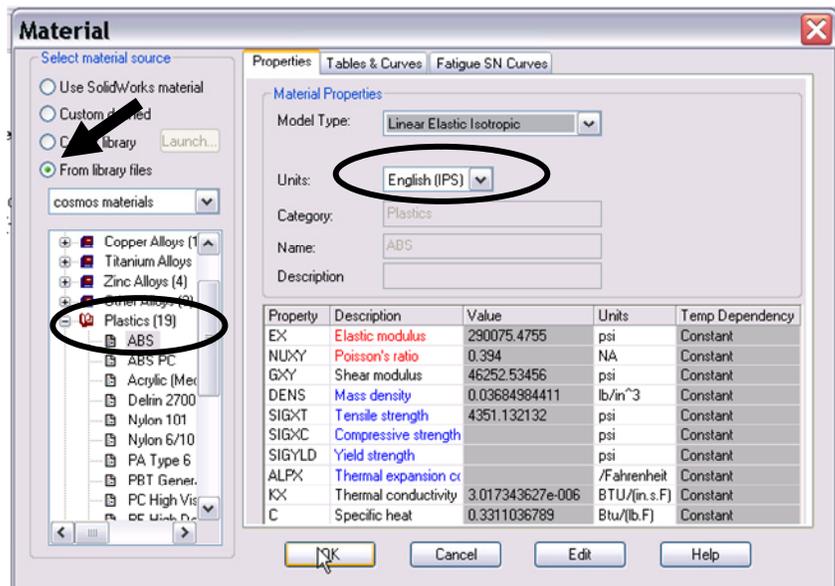
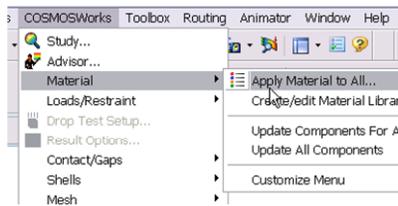
**Name the study “50 lb Load”.** Click the check mark to accept the default element type (solid mesh) and analysis type (static).



**Element Type:** There are many element types, such as plates, shells, truss members, beam elements, and solid elements. COSMOSWorks allows for solid elements to be created from solids, or shell elements to be created from either surfaces or solid mid-surfaces. Although solid elements are typically chosen when a solid model is available, solid elements are not always the best choice for many applications. Often, a few beam or shell elements will provide more accurate results than hundreds of solid elements.

**Analysis Type:** In a static analysis, we assume that loads are applied slowly. If loads are applied almost instantaneously, then dynamic effects need to be considered. A linear static analysis assumes that the response of the structure is linear – for example, a 20-lb load produces stresses and deflections that are exactly twice that of a 10-lb load. However, if the deflections are relatively large, then the stiffness of the part changes as the part deflects. In this case, a large-deflection analysis, in which the load is applied incrementally and the stiffness re-calculated at every step, may be required.

From the main menu, select **COSMOSWorks: Material: Apply Material to All**. Check “From library files” as the source of the material data, as select **ABS** from the list of plastics. (ABS stands for Acrylonitrile butadiene styrene, a common thermoplastic used in a variety of applications). Change the units to English and click OK.

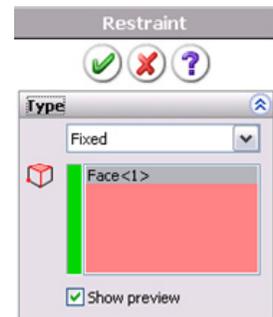
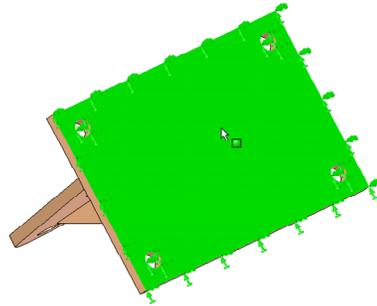
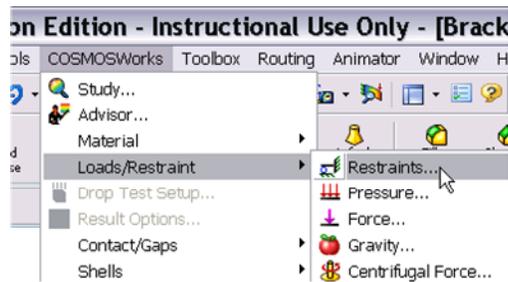


**Material:** One of the most important inputs to the model is the elastic modulus E of the material. The elastic modulus defines the stiffness (resistance to deflection) of the material. Its value is determined from material tests. A material with a high value of E will deflect less than one with a lower value of E. For comparison, steel has an E value of about 30,000,000 psi (pounds per square inch). Aluminum has an E value of 10,000,000 psi. The ABS plastic that we have chosen has a value for E of 290,000 psi, so it is about 100 times less stiff than steel.

An assumption of our model is that the material's behavior is perfectly linear, so that the deflection is exactly proportional to the load. This model is an idealization for many plastic materials, which exhibit some amount of non-linear behavior.

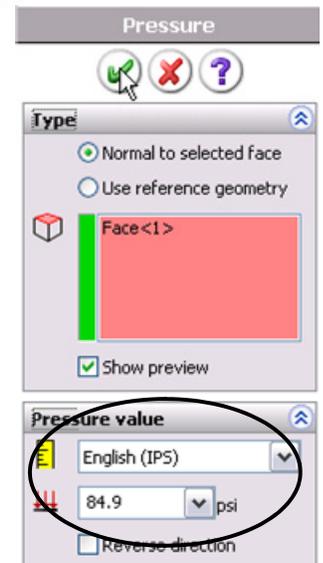
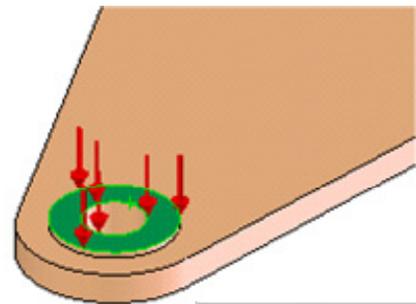
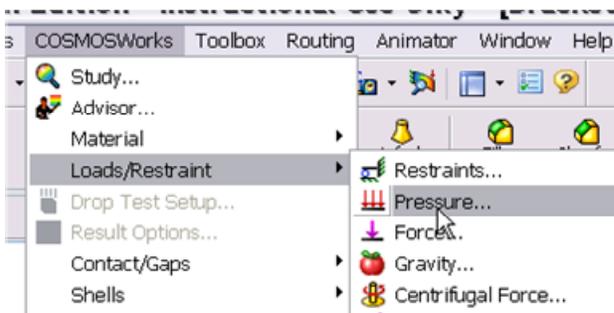
Most materials reach a point before they break at which additional loading produces much larger deflections. We say that the material has yielded at this point, and our model is not valid beyond the yield point of the material.

From the main menu, select **COSMOSWorks: Load/Restraint: Restraints**. Click on the back face of the bracket. By default, a fixed restraint will be created. Click the check mark to apply the restraint.



**Boundary Conditions:** When a component is isolated for analysis, the way in which that component is attached to another must be simulated with boundary conditions. In this case, we have chosen a fixed restraint, which means that every point on the back face of the bracket is prevented from moving in any direction. While this seems to be a reasonable assumption, it may not be entirely accurate. If screws are used to attach the bracket to a wall, then the top screws may stretch enough to allow the top of the bracket to separate from the wall. Also, the wall itself may deflect slightly. The choice of proper boundary conditions to simulate actual constraints is often one of the most important decisions to be made for an analysis.

From the main menu, select **COSMOSWorks: Load/Restraint: Pressure**. Click on the face around the 1/2-inch hole as shown here.



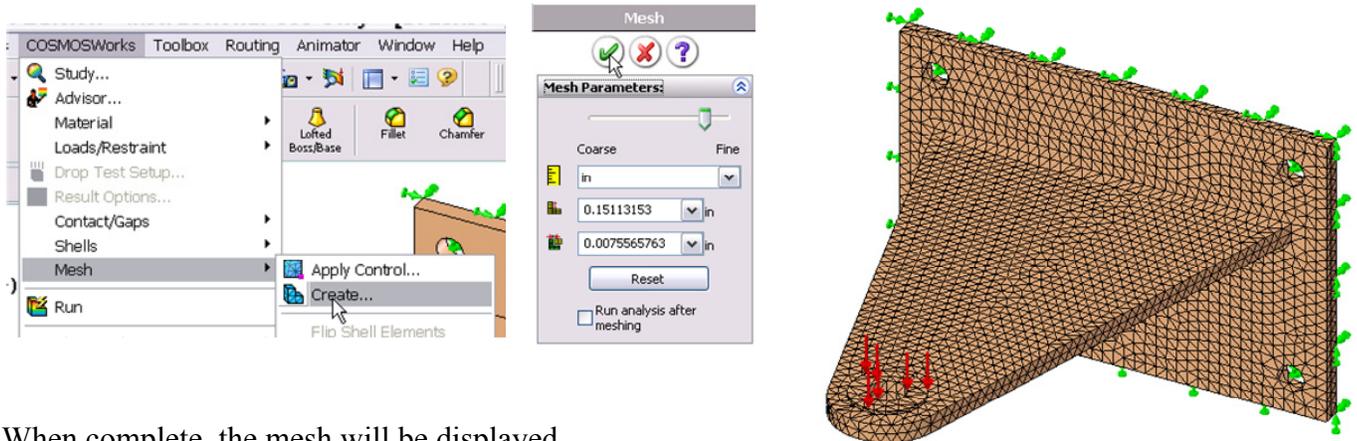
Set the pressure as **84.9 psi** (be sure to set the units to English IPS).

The pressure is calculated from the 50-lb load applied to the surface, which is one inch in diameter with a 1/2-inch hole in the center:

$$p = \frac{50 \text{ lb}}{[\pi/4 (1^2 - 0.5^2)] \text{ in}^2} = 84.9 \text{ psi}$$

Note that often a load or constraint is to be applied to only a portion of an existing face or edge. In these cases, the use of a split line can be helpful. A split line simply divides a face into multiple faces that can be selected separately. See the SolidWorks help files for information about creating split lines.

From the main menu, select **COSMOSWorks: Mesh: Create**. Move the slider bar toward the right (fine) and click the check mark.

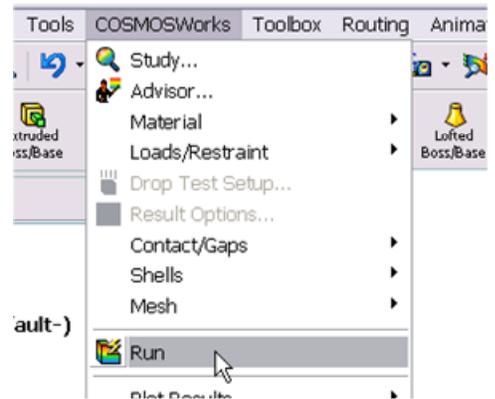


When complete, the mesh will be displayed.

**Mesh Size:** A finer mesh, with more elements, will generally produce more accurate results at the expense of longer processing time. For simple parts and a relatively fast computer, the longer processing time is not significant. However, for complex analyses (such as non-linear and time-dependent analyses), mesh size can significantly impact processing time. How many elements are needed for accuracy? Sometimes it is necessary to experiment with different meshes until the results converge to a solution. In other cases, the mesh can be refined to create more elements in a local area where stresses are greatest.

From the main menu, select **COSMOSWorks: Run**.

While the analysis is being performed, a status box will appear on the screen. For this analysis, about 8600 elements were created (your number may be more or less, depending on how far to the right you moved the mesh size slider bar). There are about 16,000 nodes, or points where the elements meet. Each node has three degrees of freedom, or possible displacement, except for those on the back face that have been constrained. Each degree of freedom has an associated equation for its displacement. While the solver is running, these equations are being formulated and solved.



This analysis should take only a few seconds on a reasonably fast computer. (A remarkable feat, considering there were about 42,000 simultaneous equations to be solved!)

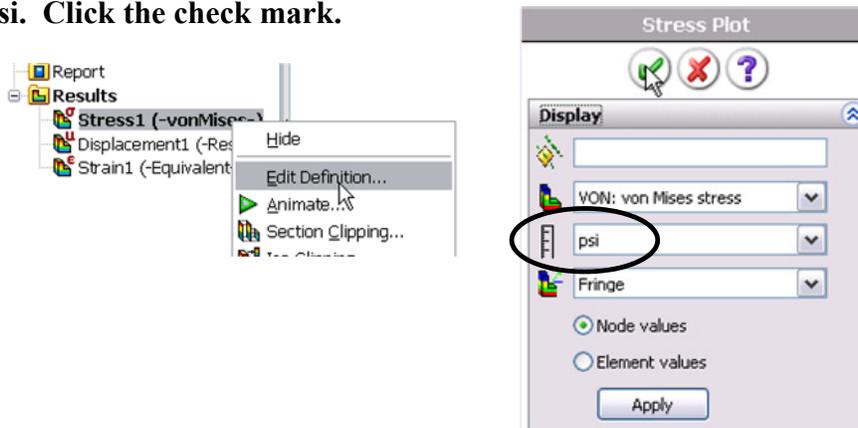
After the analysis is complete, results can be viewed in several ways.



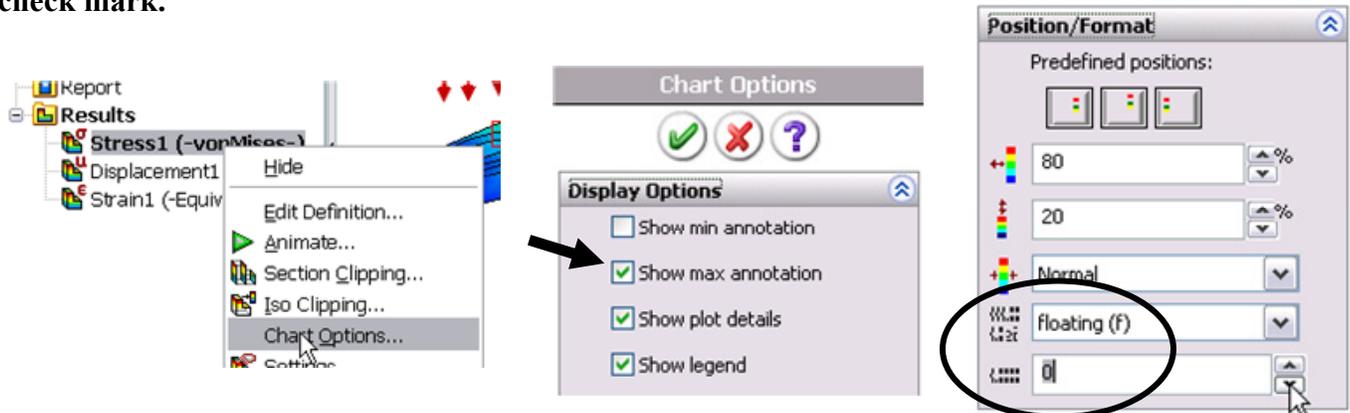
In the analysis manager, under Results, right-click on the item called Stress1 and choose Show.

A color contour plot of the stresses in the bracket is displayed. We will now change some of the default settings of the plot.

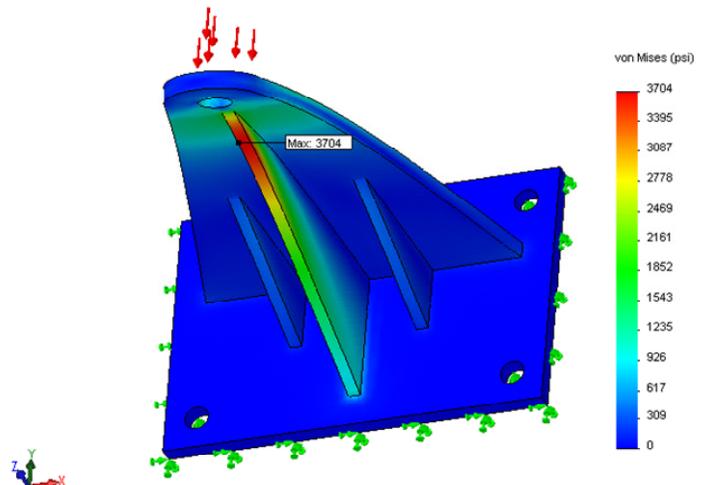
Right-click on Stress1 and choose Edit Definition. Leave the type of stress shown as von-Mises, and set the units to psi. Click the check mark.



Right-click on Stress1 again and choose Chart Options. Check the box labeled "Show max annotation". Also, change the numeric display to floating, with no decimal places shown. Click the check mark.



The resulting plot is shown here. Note the value of the maximum stress, which occurs in the center rib.

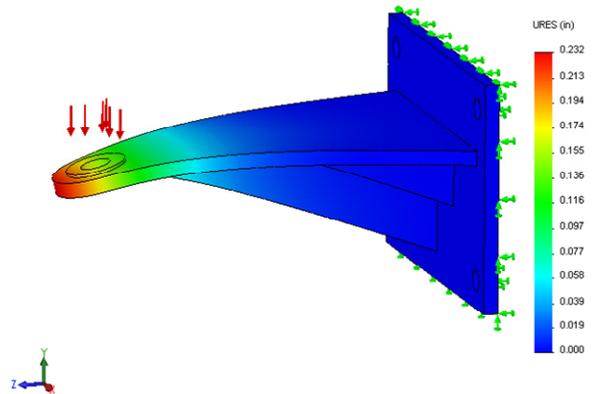


**Stress:** The simplest definition of stress is that stress is equal to force per unit area. Therefore, the units of stress are pounds per square inch or newtons per square meter (pascals). However, stress is not a single value. There are normal stresses in all three directions. Normal stresses cause a material to stretch or contract. There are also shear stresses in all three planes. Shear stresses cause a material to warp or distort. These six stress components are often combined to find principal stresses. Strength is defined as the stress at which a material will fail. Therefore, for a simple state of stress, such as a wire being stretched in one direction, we can simply compare the stress to the strength to determine if the wire will break. For a more complex state of stress, we must choose a failure theory in order to predict whether or not the part will fail. One of the most widely-used in the von-Mises, or maximum-distortion-energy theory. In our analysis, the software computed the von-Mises equivalent stress, which can be compared to the material's yield strength to predict yielding of the part. In our case, the maximum von-Mises stress is about 3700 psi. If the material's yield strength is 4350 psi, then we conclude that the part will not fail. However, the factor of safety of  $4350/3700 = 1.18$  is probably much lower than we would like to have in most applications. The factor of safety is chosen to account for all of the many uncertainties associated with the analysis (loading, material properties, environmental degradation of material, etc.) In some industries, factors of safety of 10 or more are common. In aerospace applications, where weight is critical, factors of safety of less than two are typical. When a lower factor of safety is used, extensive material testing and analysis are used to reduce uncertainty as much as is practical.

The definition of failure should also be mentioned here. Ultimate failure refers to the fracture of the material. However, we usually say that a part has failed if the material has yielded, so that additional loading produces large deflections. In some applications, excessive deflection itself may be defined as failure.

**Right-click the plot called Displacent1 and select Show. Right-click again and choose Edit Definition. Set the units to inches. Right-click a third time and select Chart Options. Set the numerical display to floating, with three decimal places.**

The maximum deflection is shown as about 0.23 inches. This value is the resultant of the deflection in all three directions. If you change the deflection type to the y-direction displacement only, you will see that the downward deflection accounts for almost the entire magnitude of the resultant.



Hopefully, this exercise has shown that finite element analysis is an incredibly useful tool to supplement engineering analysis, and that using FEA correctly requires a great deal of engineering judgment. For structural analysis, a course in mechanics of materials, usually taken at the sophomore or junior level, is a good start. In this course, you will learn about stress, strain, and deflection, and the relationships between them. Mechanical and aerospace engineers will also take more advanced courses dealing with fatigue (repeated loadings) and vibrations of machines. Civil engineers will also study vibrations for earthquake analysis. An introductory course in finite element theory is also recommended for anyone who will be responsible for conducting analysis.

Also, the tools available are constantly changing. Practicing engineers need to keep up with the newest tools through constant re-education and training.