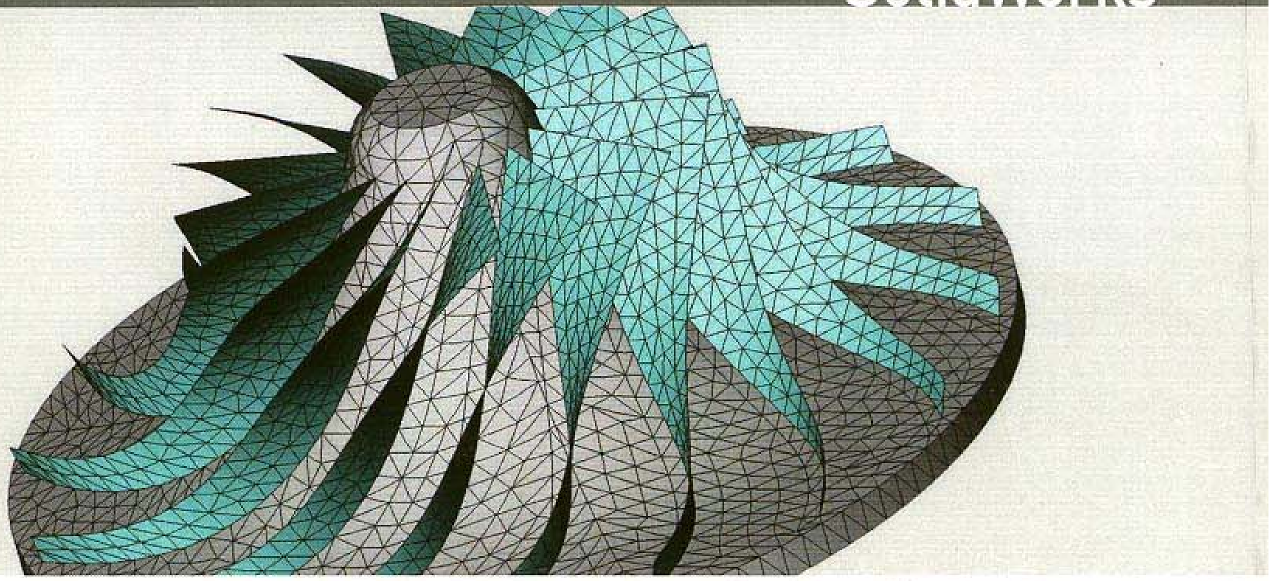
SolidWorks® 2007



COSMOS® TRAINING

COSMOSWorks Designer

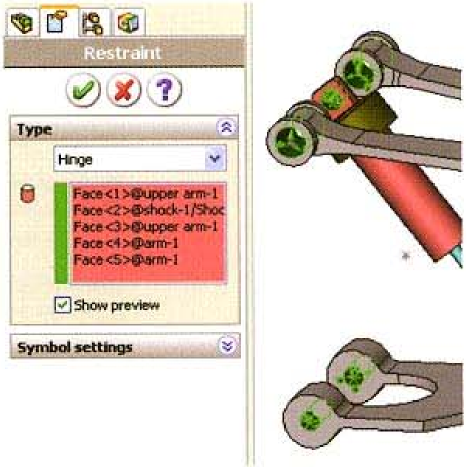
SKA renderworks

SKA Automação de Engenharias Ltda
Av. Theodomiro Porto da Fonseca, 3101,
Prédio 9
Duque de Caxias
São Leopoldo, RS 93020-080
Brasil
(51) 3591-2900

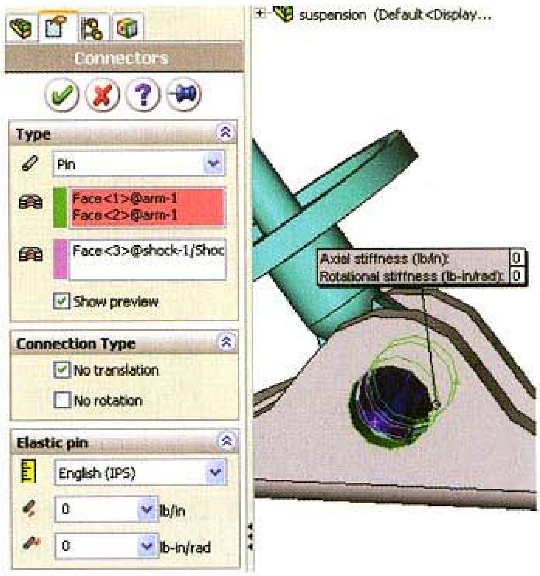
SolidWorks

6 **Assign the material.**
Specify **Alloy Steel (SS)** to all the components. You can find this material in the `cosmos materials` library.

7 **Define Hinge supports**
Define five **Hinge** supports as indicated in the figure.



8 **Connect the `shock` and the lower `arm`.**
Define a **Pin** connector between the `shock` and the lower `arm`.



9 **Connect the lower and upper `arms` to the `hub`.**
Define two **Pin** connectors between the `lower arm`, `upper arm` and the `hub`.

COSMOSWorks® 2007

COSMOSWorks Designer

Document Number: PMT0140-ENG

Table of Contents

Introduction to FEA

Introduction to FEA

About This Course

The goal of this course is to teach you how to use the COSMOSWorks Designer software to help you analyze static structural behavior of your SolidWorks part and assembly models.

The focus of this course is on the fundamental skills and concepts central to the successful use of COSMOSWorks 2007. You should view the training course manual as a supplement to, and not a replacement for, the system documentation and on-line help. Once you have developed a good foundation in basic skills, you can refer to the on-line help for information on less frequently used command options.

Prerequisites

Students attending this course are expected to have the following:

- Mechanical design experience.
- Experience with the Windows™ operating system.
- Completed the on-line SolidWorks tutorials that are available under Help. You can access the on-line tutorials by clicking **Help, Online Tutorial**.

Course Design Philosophy

This course is designed around a process- or task-based approach to training. Rather than focusing on individual features and functions, a process-based training course emphasizes processes and procedures you should follow to complete a particular task. By utilizing case studies to illustrate these processes, you learn the necessary commands, options, and menus in the context of completing a design task.

Using this Book

This training manual is intended to be used in a classroom environment under the guidance of an experienced COSMOSWorks instructor. It is not intended to be a self-paced tutorial. The examples and case studies are designed to be demonstrated "live" by the instructor.

Please note, there may be slight differences in results in certain lessons due to service pack upgrades, etc.

Laboratory Exercises

Laboratory exercises give you the opportunity to apply and practice the material covered during the lecture/demonstration portion of the course.

Training files

A complete set of the various files used throughout this course can be downloaded from the SolidWorks website, www.solidworks.com. Click on the link for Services, then Training and Certification. There you will see a link to the page where you can download the training file sets. The files are supplied as signed, self-extracting executable packages.

Windows® XP

The screen shots in this manual were made using SolidWorks 2007 and COSMOSWorks 2007 running on Windows® XP. If you are running on a different version of Windows, you may notice differences in the appearance of the menus and windows. These differences do not affect the performance of the software.

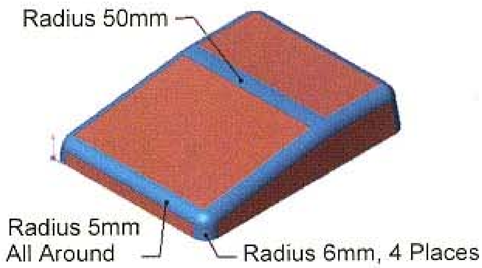
Conventions Used in this Book

This manual uses the following typographic conventions:

Convention	Meaning
Bold Sans Serif	COSMOSWorks commands and options appear in this style. For example, "Right-click `Load/Restraint` and select **Force**" means right-click the `Load/Restraint` icon in the COSMOSWorks Manager tree and select **Force** from the shortcut menu.
`Typewriter`	Feature names and file names appear in this style. For example, `Restraint-1`.
17 Do this step	Double lines precede and follow sections of the procedures. This provides separation between the steps of the procedure and large blocks of explanatory text. The steps themselves are numbered in sans serif bold.

Use of Color

The SolidWorks and COSMOSWorks user interface make extensive use of color to highlight selected geometry and to provide you with visual feedback. This greatly increases the intuitiveness and ease of use of the COSMOSWorks software. To take maximum advantage of this, the training manuals are printed in full color.



Also, in many cases, we have used additional color in the illustrations to communicate concepts, identify features, and otherwise convey important information. For example, we might show the fillet areas of a part in a different color, to highlight areas for mesh control, even though by default, the COSMOSWorks software would not display the results in that way.

What is COSMOSWorks?

COSMOSWorks is a design analysis tool based on a numerical technique called Finite Element Analysis or FEA. COSMOSWorks belongs to the family of engineering analysis software products developed by SRAC, now part of SolidWorks Corporation. Established in 1982, SRAC pioneered the implementation of FEA into desktop computing. In 1995, SRAC entered the emerging mainstream FEA software market by partnering with SolidWorks Corporation and creating COSMOSWorks software, one of the first SolidWorks Gold Products. COSMOSWorks software soon became the top-selling, add-in analysis software for SolidWorks Corporation. The commercial success of COSMOSWorks software integrated with SolidWorks CAD software resulted in the acquisition of SRAC in 2001 by Dassault Systemes, the parent company of SolidWorks Corporation. In 2003, SRAC merged with SolidWorks Corporation.

SolidWorks is a parametric, solid, feature-based CAD system. As opposed to many other CAD systems that were originally developed in a UNIX environment and only later ported to Windows, SolidWorks has, from the very beginning, been developed specifically for the Windows operating system. COSMOSWorks has also been specifically developed for the Windows operating system. Full integration between SolidWorks and COSMOSWorks is possible because both of the programs are native Windows OS applications.

COSMOSWorks comes in different "bundles", or applications, designed to best suit the needs of different users. With the exception of the COSMOSXpress bundle, which is an integral part of SolidWorks, all COSMOSWorks bundles are add-ins. A brief description of the capabilities of different bundles is as follows:

- **COSMOSXpress**
  The static analysis of parts with simple types of loads and supports.
- **COSMOSWorks Designer**
  The static analysis of parts and assemblies.
- **COSMOSWorks Professional**
  The static, thermal, buckling, frequency, drop test, optimization and fatigue analysis of parts and assemblies.
- **COSMOSWorks Advanced Professional**
  All capabilities of COSMOSWorks Professional plus nonlinear analysis; advanced dynamic analysis available in the GeoSTAR interface.

In this volume, we introduce COSMOSWorks Designer through a series of hands-on lessons intermixed with FEA fundamentals. We recommend that you study the lessons in the order presented in the text. As you go through the lessons, note that explanations and steps described in detail in earlier lessons are not repeated later.

Each subsequent lesson assumes familiarity with software functions and the FEA background discussed in previous lessons. Each lesson builds on the skills and experience gained from the previous lessons.

Before we proceed with the Lessons, let us construct a foundation for our skills in COSMOSWorks by taking a closer look at what Finite Element Analysis is and how it works.

What Is Finite Element Analysis?

In mathematical terms, FEA, also known as the Finite Element Method, is a numerical technique of solving field problems described by a set of partial differential equations. Those types of problems are commonly found in many engineering disciplines, such as machine design, acoustics, electromagnetism, soil mechanics, fluid dynamics, and others. In mechanical engineering, FEA is widely used for solving structural, vibration, and thermal problems.

FEA is not the only tool available for numerical analysis. Other numerical methods used in engineering include the Finite Difference Method, Boundary Element Method, or Finite Volumes Method. However, due to its versatility and high numerical efficiency, FEA has come to dominate the software market for engineering analysis, while other methods have been relegated to niche applications. Using FEA, we can analyze any shape, use various ways to idealize geometry and produce results with the desired accuracy. FEA theory, numerical problem formulation, and solution methods become completely transparent to users when implemented into modern commercial software, including COSMOSWorks.

A powerful tool for engineering analysis, FEA is used to solve problems ranging from very simple to very complex. Design engineers use FEA during the product development process to analyze the design-in-progress. Time constraints and limited availability of product data call for many simplifications of the analysis models. At the other end of scale, specialized analysts implement FEA to solve very advanced problems, such as vehicle crash dynamics, metal forming, or analysis of biostructures.

Regardless of the project complexity or the field of application, the fundamental steps in any FEA project are always the same, be it for example a structural, thermal, or acoustic analysis. The starting point for any analysis is the geometric model. In our case, this is a SolidWorks model of a part or an assembly. To this model, we assign material properties, and define loads and restraints. Next, as always the case when using a tool based on the method of numerical approximations, we discretize the model intended for analysis.

The discretization process, better known as meshing, splits the geometry into relatively small and simply-shaped entities, called finite elements. The elements are called "finite" to emphasize the fact that they are not infinitesimally small, but only reasonably small in comparison to the overall model size.

When working with finite elements, the FEA solver approximates the wanted solution (for example, deformations or stresses) for the entire model with the assembly of simple solutions for individual elements.

From the perspective of FEA software, each application of FEA requires three steps:

- **Preprocessing**
  The type of analysis (e.g., static, thermal, frequency), material properties, loads and restraints are defined and the model is split into finite elements.
- **Solution**
  Computing the desired results.
- **Postprocessing**
  Analyzing the results.

We follow the preceding three steps every time we use COSMOSWorks.

From the perspective of FEA methodology, we list the following FEA steps:

1. Building the mathematical model
2. Building the finite element model
3. Solving the finite element model
4. Analyzing the results

Build Mathematical Model

Analysis with COSMOSWorks starts with the geometry represented by a SolidWorks model of a part or assembly. This geometry must be meshable into a correct and reasonably small, finite element mesh. By small, we do not refer to the element size, but the number of elements in the mesh. This requirement of meshability has very important implications. We must ensure that the CAD geometry indeed meshes and that the produced mesh provides the correct solution of the data of interest, such as displacements, stresses, temperature distribution, and so on.

Often, but not always, this necessity of meshing requires modifications to the CAD geometry. Such modifications can take the form of defeaturing, idealization, and/or clean-up, described as follows:

Defeaturing

Defeaturing refers to the process of suppressing or removing geometry features deemed insignificant for analysis, such as external fillets, rounds, logos, and so on.

Idealization

Idealization presents a more aggressive exercise that may depart from solid CAD geometry as, for example, when representing thin walls with surfaces.

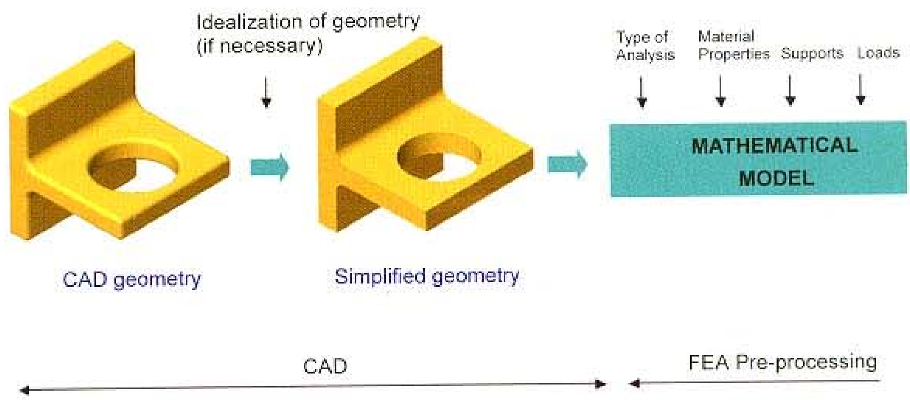
Clean-up

Clean-up is sometimes required because the meshable geometry must satisfy much higher quality requirements than those commonly followed in Solid Modeling. For clean-up, we can use CAD quality-control tools to check for problems, like sliver faces or multiple entities, that the CAD model could tolerate, but would make meshing difficult or impossible.

It is important to mention that we do not always simplify the CAD model with the sole objective of making it meshable. Often, we simplify a model that would mesh correctly "as is", but the resulting mesh would be too large and, consequently, the analysis would run too slowly. Geometry modifications allow for a simpler mesh and shorter computing time. Successful meshing depends as much on the quality of the geometry submitted for meshing as on the sophistication of the meshing tools implemented in the FEA software.

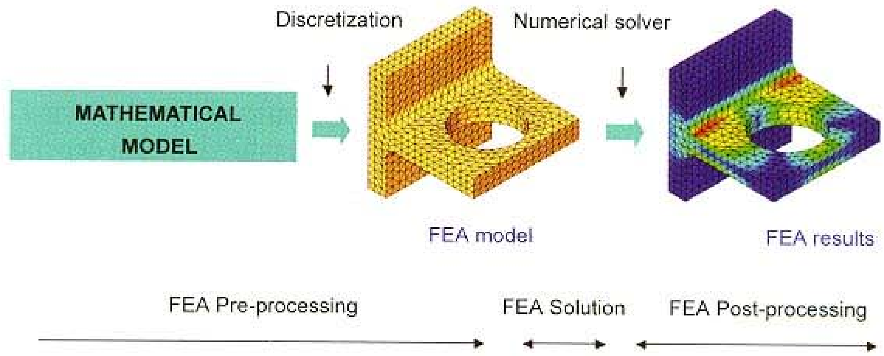
Having prepared a meshable, but not yet meshed, geometry, we define material properties, loads, supports and restraints, and provide information on the type of analysis that we wish to perform.

This procedure completes the creation of a mathematical model. Note that the process of creating the mathematical model is not FEA-specific. FEA has not yet entered the picture.



Build Finite Element Model

We now split the mathematical model into finite elements through a process of discretization, better known as meshing. Discretization visually manifests itself as the meshing of geometry. However, loads and supports are also discretized and, after the model has been meshed, the discretized loads and supports are applied to nodes of the finite element mesh



Solve Finite Element Model

After creating the finite element model, we use a solver provided in the COSMOSWorks to produce the desired data of interest.

Analyze Results

The analysis of results is often the most difficult step of all. The analysis provides very detailed results data, which can be presented in almost any format. Proper interpretation of results requires that we appreciate the assumptions, simplifications, and errors introduced in the first three steps: building the mathematical model, building the finite element model, and solving the finite element model.

Errors in FEA

The process of creating a mathematical model and discretizing it into a finite element model introduces unavoidable errors. Formulation of a mathematical model introduces modeling errors, also called idealization errors. Discretization of the mathematical model introduces discretization errors, and solution introduces numerical errors.

Of these three types of errors, only discretization errors are specific to FEA. Therefore, only discretization errors can be controlled using FEA methods. Modeling errors, affecting the mathematical model, are introduced before FEA is utilized and can only be controlled by using correct modeling techniques. Solution errors, which are round-off errors accumulated by solver, are difficult to control, but fortunately are usually very low.

Finite Elements

As we have already said, the discretization process, better known as meshing, splits continuous models into finite elements. The type of elements created in this process depends on the type of geometry meshed, the type of analysis to be executed, and sometimes on our own preferences.

COSMOSWorks features tetrahedral solid elements for meshing solid geometry, and triangular shell elements for meshing surface geometry. Why are we limited to tetrahedral, and triangular shapes? This is because the automeshers reliably mesh almost any solid or surface geometry using only those shapes of elements. Elements in other shapes, such as hexahedral (brick) elements, cannot be created reliably by the present-day automeshers. This limitation is not specific to automeshers used in COSMOSWorks. A reliable brick element automesher has not been invented yet.

Before proceeding, we need to clarify an important terminology issue. What in CAD terminology we call solid geometry, in FEA is called volumes. Solid elements are used to mesh those volumes. The term *solid* has different meanings when it is used as *solid geometry* in CAD terminology and when it is used as *solid element* in FEA terminology.

Element Types Available in COSMOSWorks

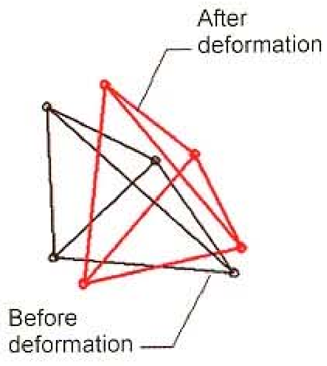
Five element types are available in COSMOSWorks: first order solid tetrahedral elements, second order solid tetrahedral elements, first order triangular shell elements, second order triangular shell elements, and two-node beam elements. The next few paragraphs describe them in this order.

COSMOSWorks terminology refers to first order elements as **Draft Quality** elements and second order elements as **High Quality** elements.

First Order Solid Tetrahedral Elements

First order (draft quality) tetrahedral elements model the first order (linear) displacements field in their volume, along faces and edges. The linear, or the first order, displacements field gives these elements their name: first order elements. If you recall from *The Mechanics of Materials*, strain is the first derivative of displacement. Therefore, strain (obtained by derivating displacement) and, consequently, stress are both constant in first order tetrahedral elements.

Each first order tetrahedral element has total of four nodes, one in each corner. Each node has three degrees of freedom, meaning that nodal displacements can be fully described by three translation components. A more detailed description of degrees of freedom follows later in this chapter.

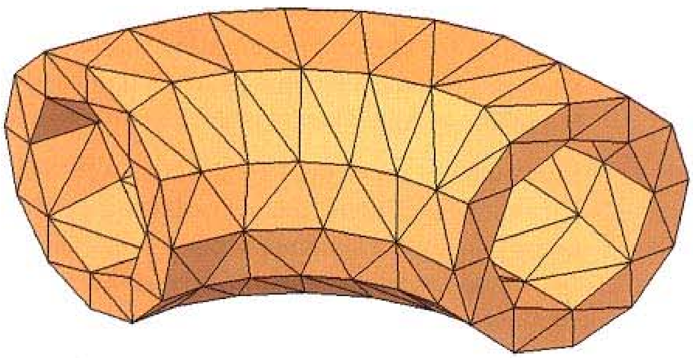


The edges of first order elements are straight and the faces are flat. These edges and faces must remain straight and flat after the elements experience deformation under an applied load.

This situation imposes a very severe limitation on the capability of a mesh constructed with first order elements to model displacements and stress fields of any real complexity. Moreover, straight edges and flat faces do not map properly to curvilinear geometry.

The failure of straight edges and flat faces to map to curvilinear geometry using first order tetrahedral elements is shown in the following diagram using an elbow geometry.

For demonstration purposes, excessively large (as compared to the model size) elements are used for this mesh. This mesh would not be sufficiently refined for any analysis.

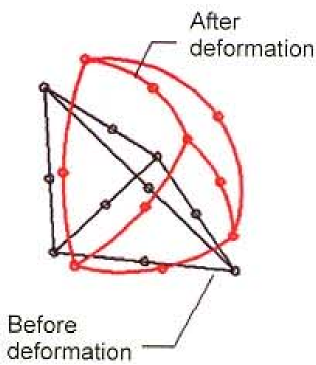


Second Order Solid Tetrahedral Elements

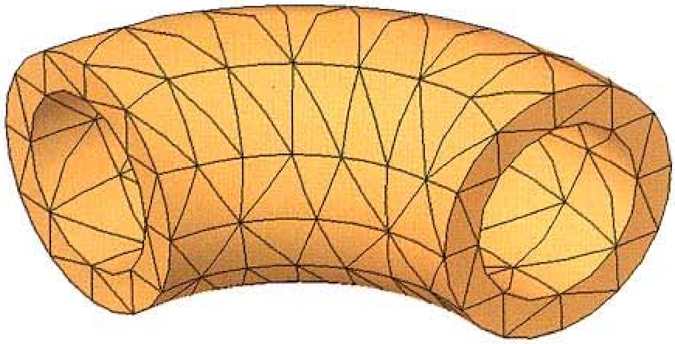
Second order (high quality) solid tetrahedral elements model the second order (parabolic) displacements field and, consequently, first order (linear) stress field (note that the derivative of a parabolic function is a linear function). The second order displacements field gives these elements their name: second order elements.

Each second order tetrahedral element has ten nodes (four corner nodes and six mid-side nodes) and each node has three degrees of freedom.

The edges and faces of second order solid elements can assume curvilinear shapes if the elements need to map to curvilinear geometry and/or during the deformation process when the elements deform under a load.



Therefore, these elements map precisely to curvilinear geometry, as illustrated by the same elbow geometry.



Again, for demonstration purposes, excessively large (as compared to the model size) elements are used for this mesh. This mesh is not sufficiently refined for analysis, even though it uses second order elements that require a significantly less-refined mesh compared to that for first order elements.

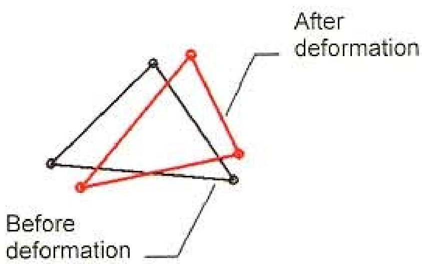
For accurate stress results, it is generally recommended to use two layers of second order elements across the wall thickness.

Because of their much better mapping capabilities and because of their ability to model the second order displacements field, second order tetrahedral elements are used for the vast majority of analyses with COSMOSWorks, even though second order elements are more computationally demanding than first order elements.

First Order Triangular Shell Elements

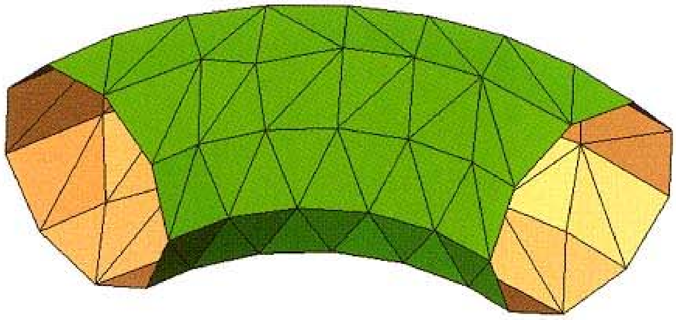
Analogous to first order solid elements, first order triangular shell elements model the linear displacements field and constant strain and stress along their faces and edges. The edges of first order shell element are straight and must remain straight while the elements deform.

Each first order shell element has three nodes (all in corners) and each node has six degrees of freedom, meaning that its displacements are fully described by three translation components and three rotation components.



If we represent the elbow with a mid-plane surface and mesh this surface with first order shell elements, note the imprecise mapping to curvilinear geometry.

This result resembles the previously demonstrated result of first order elements mapping imprecisely to curvilinear geometry.

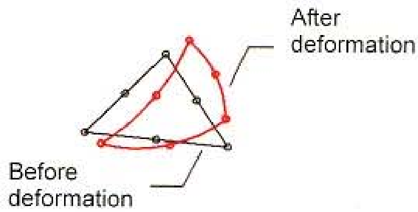


Analogous to first order solid elements shown before, these shell elements are too large for any real analysis. In the illustration, different colors are used to differentiate the element top (brown) and bottom (green). The orientation and colors are arbitrary and can be changed by "flipping" the shell elements. They do not refer, in any way, to model orientation or model geometry.

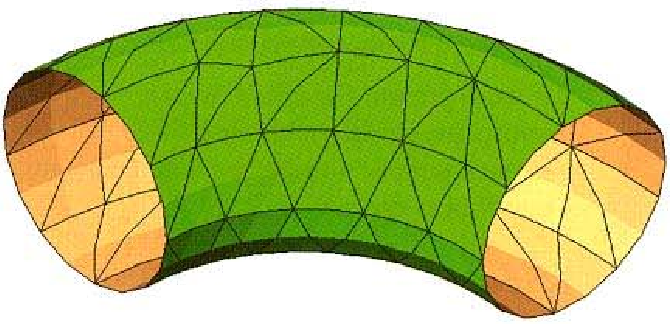
Second Order Triangular Shell Elements

Second order (high quality) triangular shell elements model the second order displacements field and the first order (linear) stress field.

Each second order shell element has six nodes: three corner nodes and three mid-side nodes. The edges and faces of second order shell elements can assume curvilinear shapes in the meshing process when the elements need to map to curvilinear geometry and/or during the deformation process when the elements deform under a load.



This shell element mesh created with second order shell elements maps precisely to curvilinear geometry as illustrated again with the elbow model.

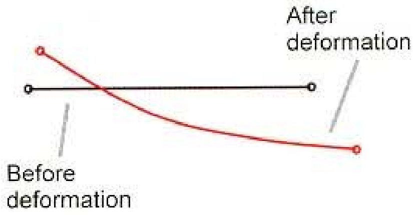


While convenient to show element mapping capabilities, the element size is too large for analysis, even though second order shell elements require less refined meshes as compared to first order shell elements.

Beam Elements

Contrary to the first order solid and shell elements, two-node beam elements model the two out-of-plane deflections as cubic functions and the axial translations and torsional rotations as linear. The shape of a two-node beam element is initially straight, but it can assume the shape of a cubic function after the deformation takes place.

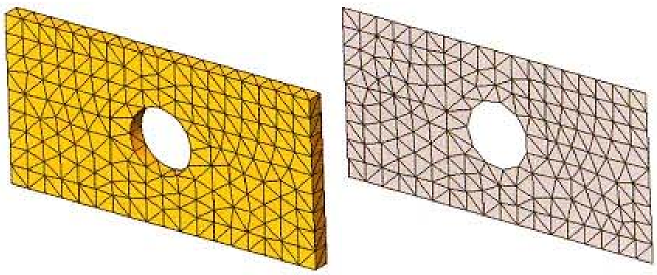
Each two-node beam element features six degrees of freedom at each end node: three translations and three rotations.



The same mesh mapping considerations that apply to the first order solid and shell elements apply to a two-node beam element as well.

Choosing Between Solid and Shell Elements

Certain classes of shapes can be modeled using either solid or shell elements, such as the elbow discussed earlier. The selection of element type: tetrahedral solid or triangular shell, used for modeling may depend on the objective of the analysis. More often, however, the nature of geometry dictates what type of element to use for meshing. For example, parts produced by casting lend themselves to be meshed with solid elements, while a sheet metal structure is best meshed with shell elements.



A hollow plate, featured in the next chapter, can be meshed with either solid elements created by meshing solid geometry or with shell elements created by meshing mid-surface.

Draft vs. High Solid and Shell Elements

First order elements, both solids and shells, should be used only for preliminary studies with specific objectives, such as verifying directions of loads or restraints, or calculating reaction forces.

The studies ready for the final computations (where the correct setup has been verified by using the Draft elements, for example) and the studies where a stress distribution is of any interest (especially in the through-thickness direction) should be modeled using High quality elements.

Degrees of Freedom

The degrees of freedom (DOF) of a node in a finite element mesh define the ability of the node to perform translation or rotation. The number of degrees of freedom that a node possesses depends on the type of element that the node belongs to. Nodes of solid elements have three degrees of freedom while nodes of shell elements have six degrees of freedom.

In order to describe transformation of a solid element from the original to the deformed shape, we need to know only three translational components of nodal displacement for each node. In the case of shell elements, we need to know, not only the translational components of nodal displacements, but also the rotational displacement components.

Consequently, built-in (or rigid) constraints applied to solid elements require only three degrees of freedom to be constrained. The same constraints applied to shell element require that all six degrees of freedom be constrained. Failure to constrain rotational degrees of freedom may result in unintentional hinge support in place of the intended rigid support.

Calculations in FEA

Each degree of freedom of each node in a finite element mesh constitutes an unknown. In structural analysis, degrees of freedom assigned to nodes can be thought of as nodal displacements. Displacements are primary unknowns and are always calculated first.

If solid elements are used, three displacement components, or three degrees of freedom (three unknowns) per node must be calculated. Using shell elements, six displacement components, or six degrees of freedom per node (six unknowns) must be calculated. All other aspects of the analysis, such as strains and stresses, are calculated based on the nodal displacements. In fact, some FEA programs offer solutions with stress calculation as an option, not a requirement.

In a thermal analysis (which determines temperatures, temperature gradients, and heat flux), the primary unknowns are nodal temperatures. Since temperature is a scalar value, unlike displacement, which is a vector, then regardless of what type of elements are used, there is only one unknown (temperature) to be found for each node in the thermal analysis model. All other results available in a thermal analysis are calculated based on those nodal temperatures.

The fact that there is only one unknown to be found for each node rather than three, or six as is the case in structural analysis, makes thermal analysis less computationally intensive than structural analysis.

Interpretation of FEA Results

The results of FEA are provided either in the form of displacements, strains and stresses for a structural analysis or in the form of temperatures, temperature gradients, and heat flux for thermal analysis. We now focus on the more intuitive structural analysis. How do we decide between a "passed" or a "failed" design?

To answer these questions, we need to establish some criteria to interpret FEA results, be they, for example, the maximum acceptable deformation, maximum stress, or the lowest acceptable natural frequency.

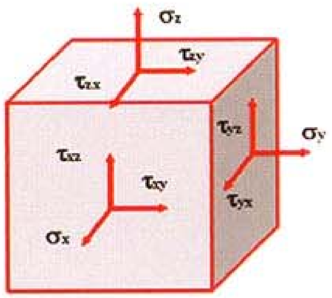
While displacement or frequency criteria are quite obvious and easy to establish, stress criteria are not.

Assume that we conduct a stress analysis in order to ensure that stresses are within an acceptable range. To assess stress results, we need to understand the mechanism of potential failure. If the part breaks, what stress component is responsible for that failure?

Discussion of various failure criteria is beyond the scope of this manual. Any book in the field of the mechanics of materials provides information on this topic. Here we limit our discussion to outlining the differences between von Mises stresses and the principal stresses, which are both common stress measures used for evaluating structural safety.

Von Mises Stress

Von Mises stress, also known as Huber stress, is a stress measure that accounts for all six stress components of a general 3D state of stress.



Two components of shear stress and one component of normal stress act on each side of an elementary cube. Due to equilibrium requirements, the general 3D state of stress is characterized by only six stress components because of equalities:

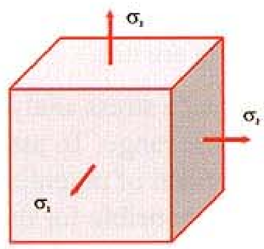
$$\tau_{xy} = \tau_{yx}, \tau_{yz} = \tau_{zy}, \tau_{xz} = \tau_{zx}$$

The von Mises stress equation can be expressed by stress components that are defined in a global coordinate system as:

$$\sigma_{eq} = \sqrt{0.5[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2] + 3(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}$$

Principal Stresses: P1, P2, and P3

The state of stress can also be described by three principal stress components: $\sigma_1, \sigma_2, \sigma_3$ whose directions are normal to faces of an elementary stress cube.



Von Mises stress is then expressed as:

$$\sigma_{eq} = \sqrt{0.5[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]}$$

Note that von Mises stress is a non-negative, scalar value. Von Mises stress is a commonly used stress measure because the structural safety of many engineering materials showing elastoplastic properties, such as steel, is well described by von Mises stress magnitude.

For those materials, the yield factor of safety or the ultimate factor of safety can be calculated by dividing von Mises stress by the yield stress (also called yield strength) or by the ultimate stress (also called ultimate strength) of the material.

In COSMOSWorks, principal stresses are denoted as P1, P2, and P3.

P1 stress which is usually tensile, is used when evaluating stress results in parts made of brittle material, whose safety is better related to P1 than to von Mises stress. P3 is used to examine compressive stresses and contact pressure.

Units of Measurement

Internally, COSMOSWorks uses the International System of Units (SI). As COSMOSWorks users, we are spared much confusion and trouble with systems of units. Data may be entered in three different systems of units: SI, Metric, and English. Similarly, results can be displayed any of those three systems of units. The available systems of units are summarized in the following table:

	International System of Units (SI)	Metric (MKS)	English (IPS)
Mass	kg	kg	lb.
Length	m	cm	in.
Time	s	s	s
Force	N	kgf	lb.
Mass density	kg/m^3	kg/cm^3	lb./in^3
Temperature	^oK	^oC	^oF

Limitations of COSMOSWorks Designer

With any FEA software, we need to take advantage of its strengths as well as work within its limitations. Analysis with COSMOSWorks Designer is conducted under the following assumptions:

- material is linear
- deformations are small
- loads are static

These assumptions are typical of the FEA software used in the design environment, and the vast majority of FEA projects are run successfully within these limitations.

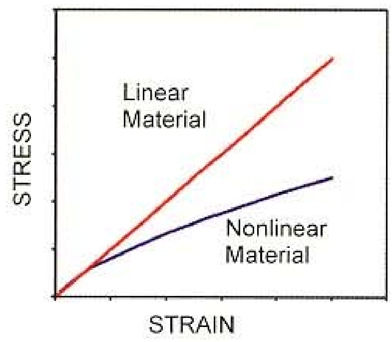
For analyses requiring nonlinear material, nonlinear geometry, or dynamic analysis, tools such as COSMOSWorks Advanced Professional can be used. Some dynamic analysis capabilities are also included in COSMOSWorks Professional, which features frequency analysis and drop test functions.

Note

COSMOSWorks also features a geometrically nonlinear solver to compute large displacement problems. However, because only a default set of the parameters for the nonlinear solver is available, the applicability of this COSMOSWorks Designer feature is limited. For full scale nonlinear problems (both the geometry and materials), COSMOSWorks Advanced Professional suite must be used.

Linear Material

In all materials used with COSMOSWorks Designer, stress is linearly proportional to strain.



Using a linear material model, the maximum stress magnitude is not limited to yield or to ultimate stress as it is in real life.

For example, in a linear model, if stress reaches 100,000 psi under a load of 1,000 lb., then stress will reach 1,000,000 psi under a load of 10,000 lb. 1,000,000 psi is, of course, a ridiculously high stress value.

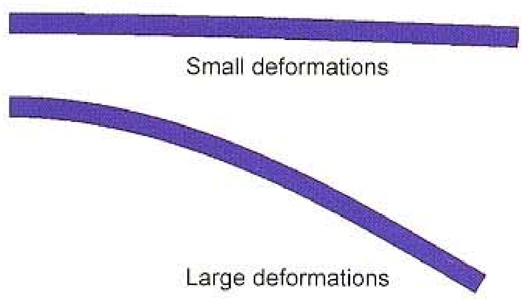
Material yielding is not modeled. Whether or not yield, in fact, takes place can only be interpreted based on the stress magnitudes reported in results.

Most analyzed structures experience stresses below yield stress, and the factor of safety is most often related to the yield stress.

Therefore, the analysis limitations imposed by linear material seldom impede COSMOSWorks Designer users.

Small Deformations

Any structure experiences deformation under load. In COSMOSWorks Designer, we assume that those deformations are small. What exactly is a small deformation? Often it is explained as a deformation that is small in relation to the overall size of the structure.



The preceding figure shows a cantilever beam in bending with small deformations and large deformations.

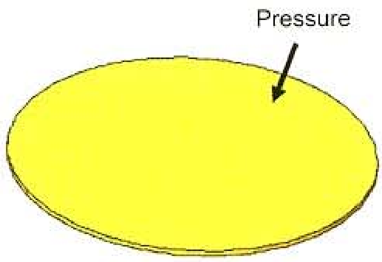
If deformations are large, then the COSMOSWorks Designer assumptions generally do not apply, even though COSMOSWorks Designer has some large displacement analysis capabilities, which we will discuss towards the end of this volume.

Other analysis tools, such as COSMOSWorks Advanced Professional must be used to analyze this structure.

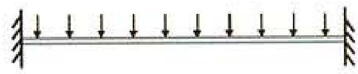
Note that the magnitude of deformation is not the deciding factor when classifying deformation as "small" or "large". What really matters is whether or not the deformation changes the structural stiffness in a significant way.

Small deformation analysis assumes that the structural stiffness remains the same throughout the deformation process. Large deformation analysis accounts for changes of stiffness caused by deformations.

While the distinction between small and large deformations is quite obvious for the beam, it is not at all obvious, for example, for a flat membrane under pressure.



For a flat membrane, initially the only mechanism resisting the pressure load is that of bending stresses.



During the deformation process, the membrane acquires membrane stiffness in addition to the original bending stiffness.

The stiffness of the membrane changes significantly during deformation. This change in stiffness requires a large deformation analysis, using tools like COSMOSWorks Advanced Professional.

Static Loads

All loads, as well as restraints, are assumed not to change with time. This limitation implies that loads are applied slowly enough to ignore inertial effects. Dynamic loading conditions cannot be analyzed with COSMOSWorks Designer.

While all loads, in reality, change with time, modeling them as static loads is most often acceptable for the purpose of design analysis. Gravity loads, centrifugal forces, pressure, bolt preloads, and so on can be successfully represented as static loads.

Dynamic analysis is generally required only for fast-changing loads. A drop test or vibration analysis definitely require that we model dynamic loads.

Summary

This short review of FEA fundamentals is not, of course, "all inclusive". It is only intended to get us started with the hands-on lessons. As we progress through lessons presented in the following chapters, we will occasionally digress from software-specific issues in order to discuss relevant FEA fundamentals.

Lesson 1
Static Analysis of a Rectangular Plate with a Hole

Objectives

Upon successful completion of this lesson, you will be able to:

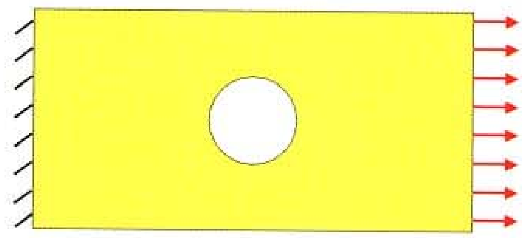
- Navigate the COSMOSWorks interface
- Execute a linear static analysis using solid elements
- Understand the influence of mesh density on displacement and stress results
- Employ various methods to present FEA results
- Manage COSMOSWorks result files
- Access available help and assistance

Project Description

The first in our series of COSMOSWorks Designer lessons is a very simple model of a rectangular hollow steel plate that is supported and loaded as shown. Working with this model familiarizes us with all the steps and the majority of the software functionality typically used in a static analysis of solid models.

In spite of its simplicity, this is probably the most important lesson in the COSMOSWorks Designer volume. This lesson walks you through all required steps; however, after the lesson is complete, we encourage you to explore other software functionality and other modeling assumptions, such as different material properties, loads, restraints, and so on.

The rectangular plate with a hole is fixed at a short-end face. A 25,000 lb load is uniformly distributed along the other end face.



In addition to learning COSMOSWorks Designer functions, our objective is to investigate the impact of different mesh densities on the results. Using FEA terminology, the objective is to investigate the effect of different discretization choices on the data of interest, in our case, on deformation and stress.

Therefore, we perform the analyses using meshes with different element sizes. Note that repetitive analysis with different meshes does not represent standard practice in FEA. We repeat the analysis using different meshes as a learning tool to gain more insight into how FEA works.

1 **Open part.**
From the SolidWorks window, open the part named `rectangular hollow plate`. Review the dimensions of the model and note down the length, width, and thickness of the part in millimeters.

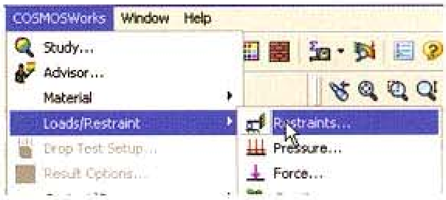
2 **Start COSMOSWorks.**
Verify that the COSMOSWorks option is selected in the add-in list.

To start COSMOSWorks, click the COSMOSWorks Manager tab.

Menu Structure

To create an FEA model, solve the model, and analyze the results, we use a graphical interface in the form of icons and folders located in the COSMOSWorks Manager window.

However, you can achieve the same results by making the appropriate choices in the COSMOSWorks menu. To open the menu, click **COSMOSWorks**, from the Main toolbar in the SolidWorks window.

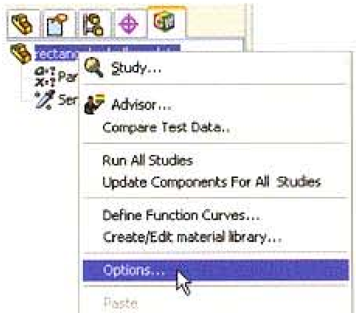


In the following text, we will use the COSMOSWorks Manager user interface to access the menu.

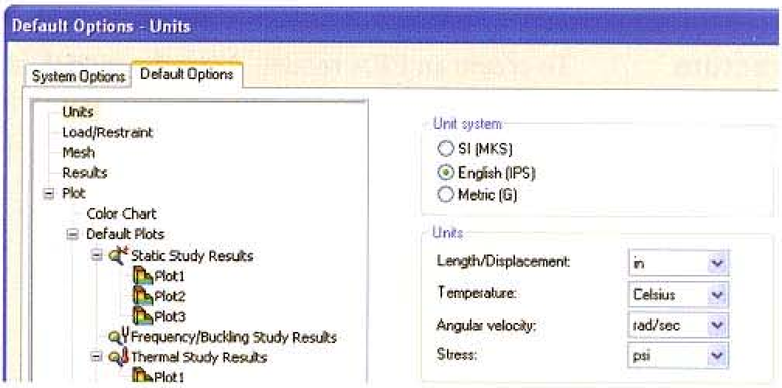
COSMOSWorks Options

Before we create the FEA model, let us review the **Options** in COSMOSWorks menu.

3 **Open Options window.**
Right-click the part icon in the COSMOSWorks Manager window and select **Options**.

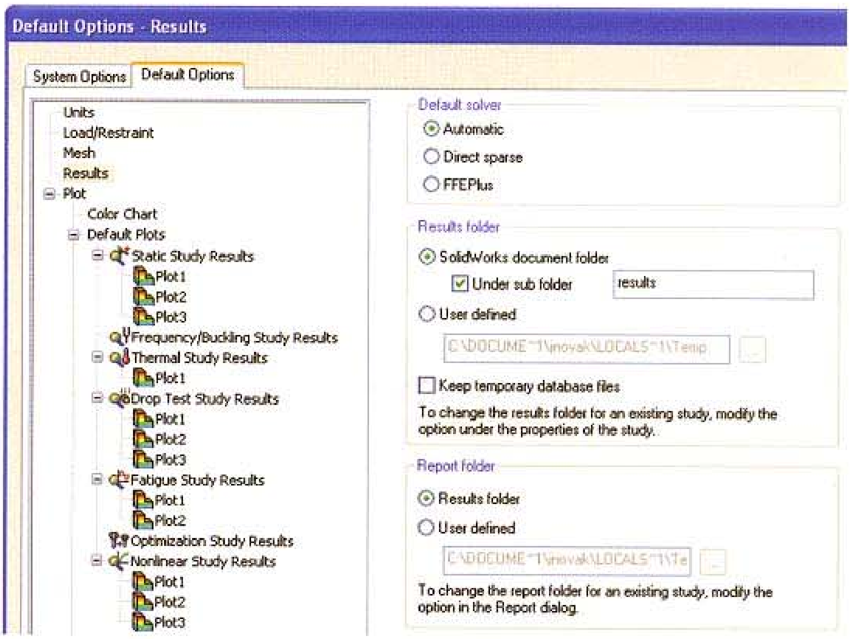


4 **Set default units for COSMOSWorks.**
Under **Default options (New Study)**, select `Units`. Make sure that the **Units** system is set to **English (IPS)** and **Length/Displacement** and **Stress** are in **inches** and **psi**, respectively.



5 Set default results.

Under **Default options**, select the `Results` folder. In this lesson, the analysis results will be created and stored in a sub-folder located in the SolidWorks document directory.



Under **Results folder**, select **SolidWorks document folder.** **SolidWorks document folder** is the folder where `rectangular hollow plate.SLDPRT` file resides in your computer.

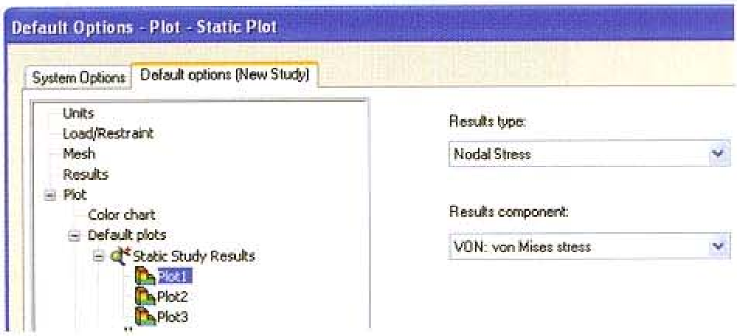
Select the **Under sub folder** check box.

In the **Under sub folder** box, enter **results**. This will automatically create a sub folder `results` to store COSMOSWorks results.

The **Report folder** (the place to store automatically created reports) is by default the same as the Result folder.

Under **Default Solver**, select **Automatic**.

6 Set default plots.



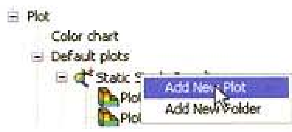
Still under the **Default options (New Study)** tab, open the `Default plots` subfolder located in the `Plot` folder. This section allows you to select default result plots to be generated after solving the analysis. As you will shortly find out, upon completion of any static analysis, COSMOSWorks automatically creates the following result plots:

Stress1, **Displacement1**, and **Strain1** (identified as `Plot1` through `Plot3` in the above dialog).

You may instruct COSMOSWorks as to which plots will be automatically created. For example, click `Plot1` under the `Static Study Results` folder to see that **VON: von Mises stress (Nodal stress)** plot is requested after each static study is run.

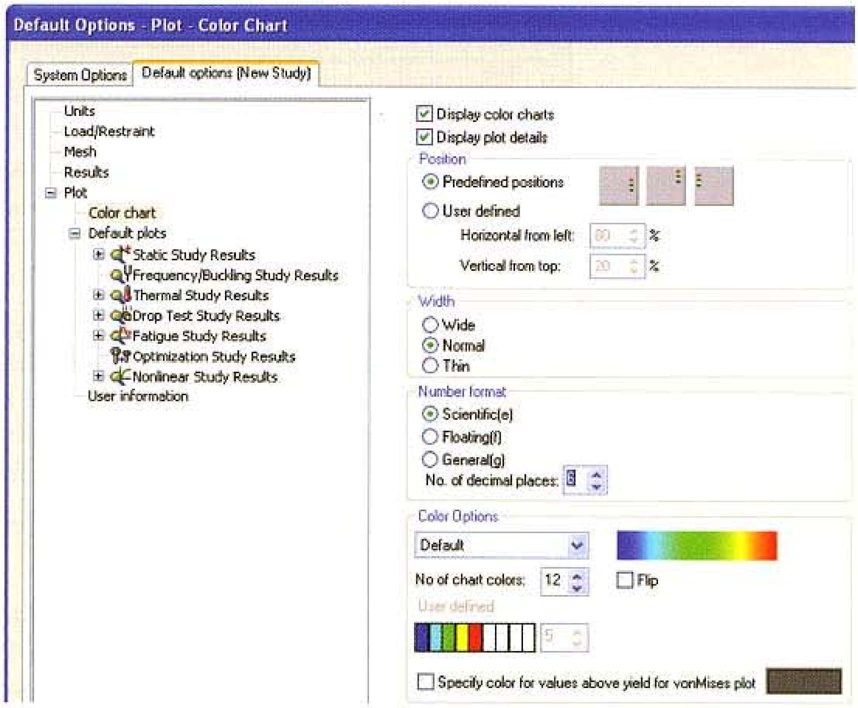
Note

To add an additional default result plot, right-click `Static Study Results` and select **Add New Plot**. If desired, each type of plot can be stored in a user-defined folder.



In this lesson we do not modify the default settings in the `Default plots` folder.

7 **Specify color chart options.**
Under the `Plot` folder, select `Color chart`.



Set **Number format** to **scientific (e)** and **No. of decimal places** to **6**.

We encourage students to explore all the chart options in this dialog.

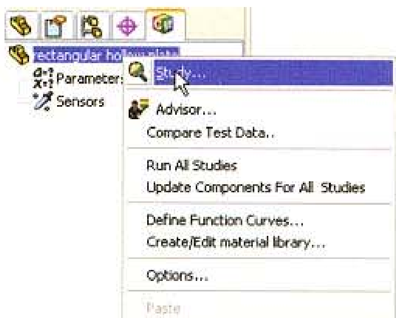
Click **OK** to close the **Options** window.

Preprocessing

8 **Create study.**
Creation of an FEA model always starts with the definition of a study.

The study definition is where we enter information in COSMOSWorks about what kind of analysis we wish to perform and what mesh we wish to use.

To define a study: Right-click the part icon in the COSMOSWorks Manager design tree, and then select **Study**.



Name the study. Any study name can be used; here we name the study `mesh1`.

Select **Solid mesh** and specify the **Type** of study as **Static**.

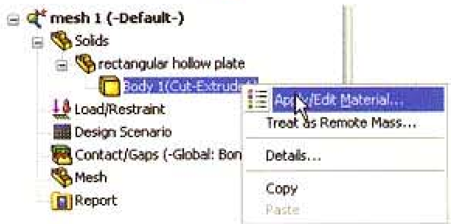
Click **OK**.

New Study

When a study is defined, COSMOSWorks automatically creates a study folder (named, in this case, `mesh1`) and places several icons in it.

Note that some of the icons are folders that contain other icons.

In this lesson, we use the `Solids` folder to define and assign material properties, the `Load/Restraint` folder to define loads and restraints, and the `Mesh` icon to create the finite element mesh.



The `Design Scenario` and `Report` folders are not used in this lesson, nor is the `Parameters` icon, which is automatically created prior to study definition.

Note that there is only one component, named `rectangular hollow plate`, in the `Solids` folder.

If an assembly (and not a part) is analyzed, then the `Solids` folder contains as many components as there are parts in the assembly.

Assigning Material Properties

We can assign material to the model in either the SolidWorks or the COSMOSWorks window.

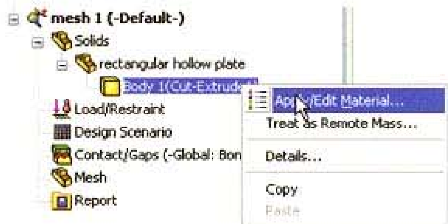
If a material was assigned in the SolidWorks window, then the material definition will be transferred automatically to COSMOSWorks.

In this lesson, we assign material to the part in the COSMOSWorks window, not because this is the preferred way, but to demonstrate this option.

9 Assign material properties.

Do one of the following to select a material and assign properties to the part:

- Right-click the `Solids` folder and select **Apply material to all**.
- Right-click the `rectangular hollow plate` icon, which is located in the `Solids` folder and select **Apply material to all bodies**.
- Right-click the `Body 1` icon, which is located in the `Solids, rectangular hollow plate` folder and select **Apply/Edit Material**.

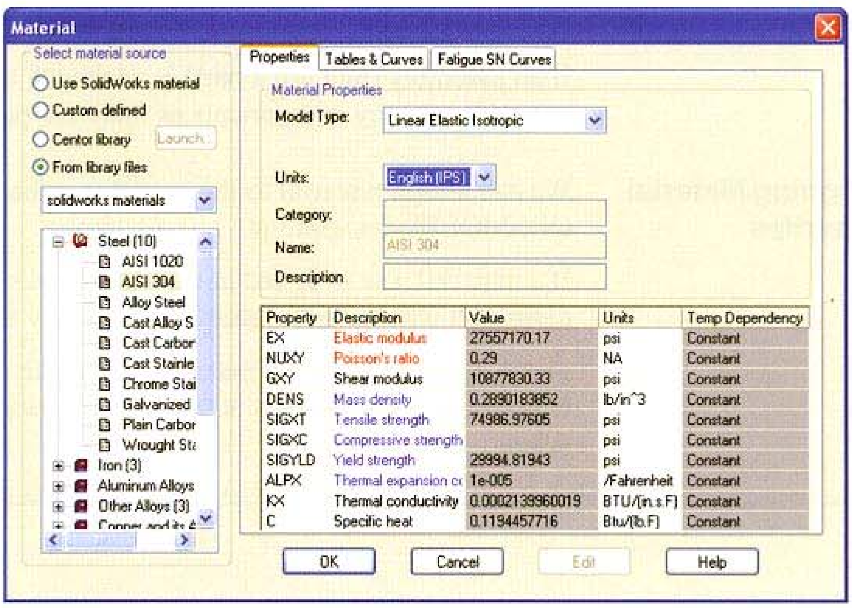


Note

The first method assigns the same material properties to all components in the model. The second method assigns material properties to one particular component and all the multi-bodies associated to the component. The third method assigns material properties to one particular body: in this lesson, the `rectangular hollow plate`. Because we are not working with an assembly but with a single part which contains only one body (i.e. this is not a multi body part) any of the above three ways of material assignment can be used.

In the **Select material source** area, select **From library files** for the material source. Note that we may choose between **SolidWorks materials** library and **Cosmos materials** library.

Select `solidworks materials` and assign **AISI 304** from the `Steel` folder.



Note that the required material constants are in red font. The constants shown in blue font may be required if specific load types are used (for example, the **Temperature** load would require the **Thermal expansion coefficient**).

Click **OK**.

Note

Note that the `rectangular hollow plate` icon in the `Solids` folder now displays a green check mark and the name of the selected material to indicate that a material has successfully been assigned.

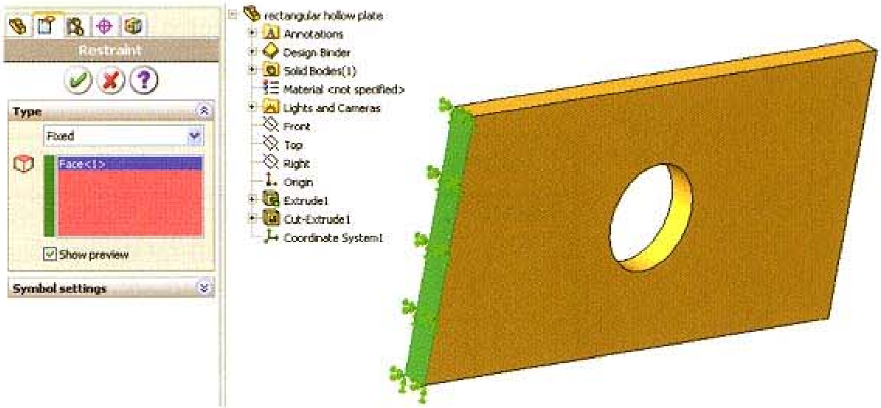
10 Define Fixed Restraints.

Right-click `Load/Restraint` and select **Restraints**.

Rotate the model and select the face to apply restraints. All view functions, including rotate, pan, and zoom, work as they do in the SolidWorks interface.

Note that the SolidWorks FeatureManager menu (we call it Flyout FeatureManager) is available in the upper left corner of the model window, but we do not use it at this time.

In the **Type** box, select **Fixed**, and then select the check box (**OK**) to close the **Restraint** PropertyManager.

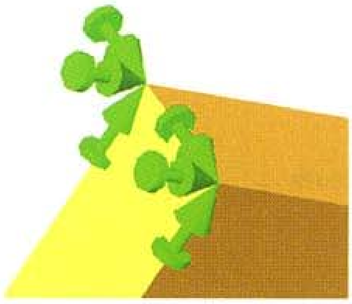


Note that each boundary condition can be renamed to help us decipher the meaning later on.

Use Window's standard click-inside technique to rename this restraint to `Fixed side`.

Restraint Types

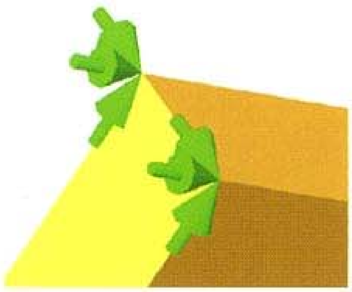
Restraint symbols are displayed on the face where they have been applied.



We select **Fixed** as the restraint type, meaning that all six degrees of freedom (three translations and three rotations) have been restrained.

The restraint symbols feature arrows to indicate translational restraints and discs to indicate rotational restraints in respective directions. In this lesson, the restraints are defined by the directions of the global coordinate system visible in the lower-left corner of the model window.

If, instead of selecting **Fixed** as the type of restraint, we selected **Immovable**, then the rotational degrees of freedom would not be constrained and the corresponding restraint symbols would feature only arrows, not discs.



Because we are using solid elements (remember that we specified **Solid mesh** in the study definition) which do not have rotational degrees of freedom, both the **Fixed** or **Immovable** options yield the same restraint condition.

Before proceeding with the remainder of the lesson, we will review the available options in the **Restraint** PropertyManager:

Restraint Type	Definition
Fixed	Also called a rigid support, all translational and all rotational degrees of freedom are constrained. **Fixed** restraints do not require any information on the direction along which restraints are applied.
Immovable (No translation)	Only translational degrees of freedom are constrained, while rotational degrees of freedom remain unconstrained. If solid elements are used, **Fixed** and **Immovable** restraints have the same effect because solid elements do not have rotational degrees of freedom and only translational degrees of freedom can be constrained.

Restraint Type	Definition
Symmetry	This option is available for use on flat face; in-plane displacements are allowed and rotation in the direction normal to the plane is allowed.
Roller/Sliding	Use the **Roller/Sliding** restraint to specify that a planar face can move freely in its plane but cannot move in the direction normal to its plane. The face can shrink or expand under loading.
Hinge	Use the hinge restraint to specify that a cylindrical face can move only about its axis. The radius and the length of the cylindrical face remain constant under loading.
Use reference geometry	This option restrains a face, edge, or vertex only in desired direction(s), while leaving the other directions free to move. You can specify the desired direction(s) of restraint in relation to the selected reference plane, axis, edge, or face. The SolidWorks Flyout FeatureManager is useful for selecting reference geometry (plane and 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 reference face define the directions in a cylindrical coordinate system; this option is very useful because you can apply a restraint that allows for rotation about the axis associated with the cylindrical face.
On spherical face	Similar to **On flat face** and **On cylindrical face**; the three principal directions of a spherical face define the directions of the applied restraints in a spherical coordinate system.
Cyclic symmetry	This option is used to restrain segments which, if periodically revolved around a specified axis of revolution, would form a rotationally symmetrical body.

Having defined restraints, we have fully fixed the model in space. Therefore, the model cannot move without elastic deformation. In FEA terminology, we say that the model does not have any rigid body motions or that it does not have any rigid body modes.

11 Display/hide restraint symbols.

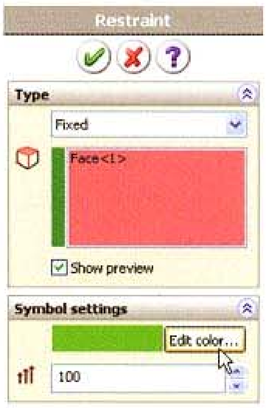
Restraint symbols can be displayed or hidden by doing one of the following actions:

- Right-click `Load/Restraint` and select **Hide All** or **Show All**.
- Right-click the restraint symbol for each restraint individually, and then select **Hide** or **Show**.
- Select or clear the **Show Preview** check box in the **Restraint** PropertyManager used to define the respective restraint.

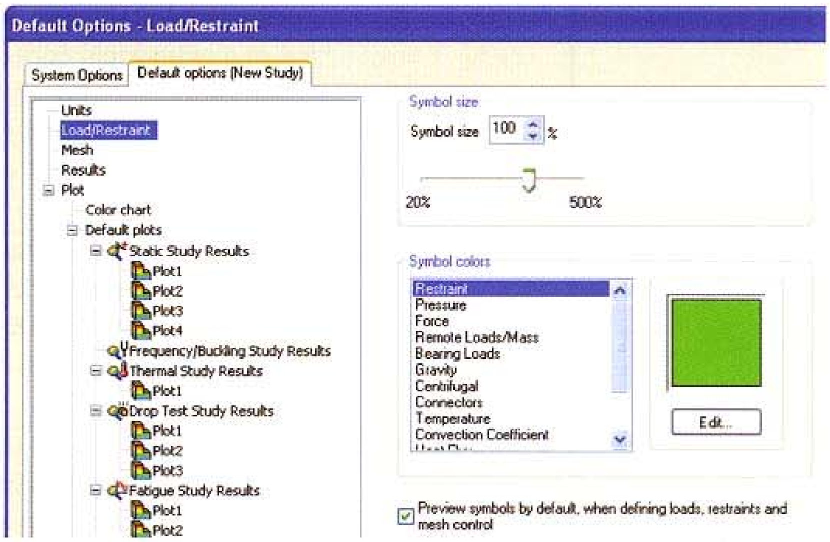
Size and color of restraint symbols

The size and color of restraint symbols can be controlled both locally and globally

The load settings of the restraint symbol is controlled from the **Symbol settings** dialog in the **Restraint** PropertyManager.



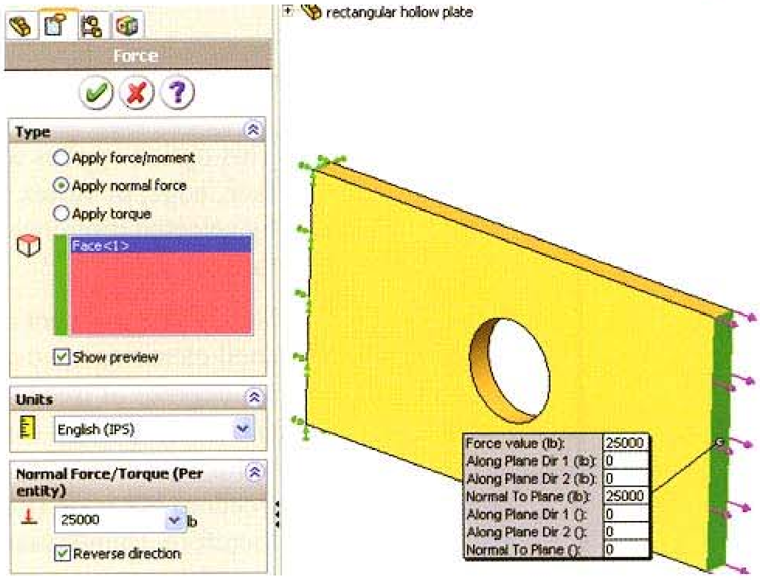
The global definitions for all the restraint symbols can be controlled under COSMOSWorks Options in the `Load/Restraint` folder.



12 Define Force.

Rotate the model to reveal the face where the 25,000 lb. tensile force is applied and select this face.

Right-click `Load/Restraint` and select **Force** to list the available options for defining loads. This action opens the **Force** PropertyManager.



In the **Type** area, select **Apply normal force**, in the **Units** dialog make sure that **English (IPS)** is selected, and in the **Normal Force/Torque** box, type **25000.**

Select the **Reverse direction** check box. This is required to define a tensile force. Clearing the **Reverse direction** check box would result in a compressive force.

When defining a normal force we do not need to use any reference geometry. Load direction is sufficiently defined by the orientation of the loaded face when **Apply normal force** is in effect.

Select the **Force** check box (**OK**) to close the **Force** PropertyManager.

Similarly, as we did in the case of the fixed restraint, rename this force definition to `Tensile force`.

13 Display/hide force symbols.

The model now shows both loads and restraints symbols, which can be displayed or hidden by doing one of the following actions:

- Right-click a particular restraint or load icon in the `Load/Restraint` folder and choose **Show**.
- Right-click the `Load/Restraint` folder to globally display or hide loads and restraints and choose **Show All**.
- Select or clear the **Show Preview** check box in the **Force** PropertyManager used to define the respective force

Force Options

Let us review the other options that are available for **Force** definition. Generally, forces can be applied to faces, edges, and vertices using various methods.

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

The presence of a load is indicated by arrows symbolizing the load and by the corresponding icon, `Tensile force`, in the `Load/Restraint` folder.

To familiarize yourself with this feature, right-click the `Fixed side` and `Tensile` icons to examine the available options.

Preprocessing Summary

Now that we have assigned the material properties, loads, and restraints, we have fully defined the mathematical model, which we intend to solve with FEA.

The mathematical model must be discretized into a finite element model. Before creating the finite element model, let us make a few observations about the following terms:

- Geometry preparation
- Material properties
- Load definition
- Restraint definition

Geometry Preparation

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

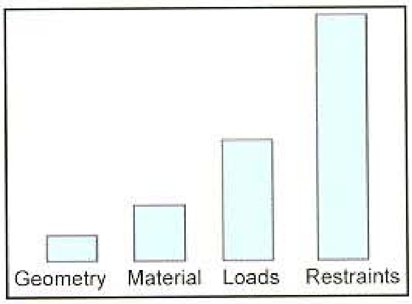
Material properties are most often selected from the material library and do not account for local defects, surface conditions, and so on. Generally, material definition has more uncertainties than geometry preparation.

Load Definition

Load definition, even though done in a few quick menu selections, involves many background assumptions because in real life, load magnitude, distribution, and time dependence are often known only approximately and must be roughly estimated in FEA with many simplifying assumptions. Therefore, significant idealization errors can be made when defining loads. Nonetheless, loads can be expressed in numbers, which makes loads easier for FEA users to relate to.

Restraint Definition

Defining restraints is where severe errors are most often made. A common error is over-constraining the model, which results in an overly stiff structure that underestimates deformations and stresses.



The relative level of uncertainties in defining geometry, material, loads, and restraints is qualitatively shown.

Idealizations and Assumptions

Geometry is the easiest to define while restraints are the most difficult, but the level of difficulty has no relation to the time required for each step, so the message in this bar graph may be counterintuitive. In fact, preparing CAD geometry for FEA may take hours, while defining material, and applying loads and restraints involves only a few mouse clicks.

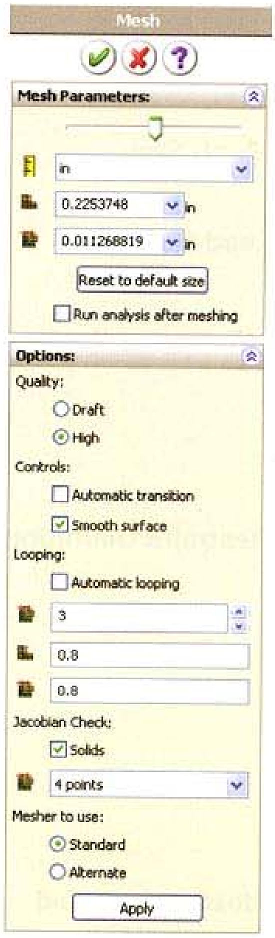
In all examples here, we assume that material properties, loads, and supports are known with certainty, and that the way they are defined in the model represents an acceptable idealization of real conditions. However, we need to emphasize that it is the responsibility of the user of the FEA software to determine if all those idealized assumptions made during the creation of the mathematical model are indeed acceptable. The best automesher and the fastest solver do not help if the mathematical model submitted for analysis with FEA is based on erroneous assumptions.

---

14 Set mesh options and generate mesh.
The model will meshed using **High** quality elements.

Right-click `Mesh` and select **Create Mesh.**

In the **Options** dialog, under **Quality**, select **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.

The differences between first and second order elements are discussed in *Element Types Available in COSMOSWorks* in the *Introduction to FEA* chapter.

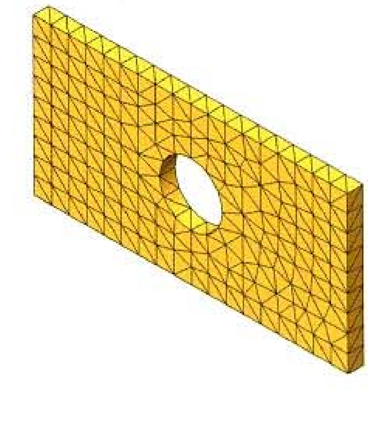
Make sure that the default **Element size** of **0.225 in** and **Tolerance** of **0.011 in** are specified.

We will review the other mesh options as we proceed with the class.

Click **OK** to generate the mesh.

The mesh appears after mesh generation is completed.

The Mesh icon in the COSMOSWorks Manager window now displays a green check mark to indicate that meshing has been successfully completed.



Meshing

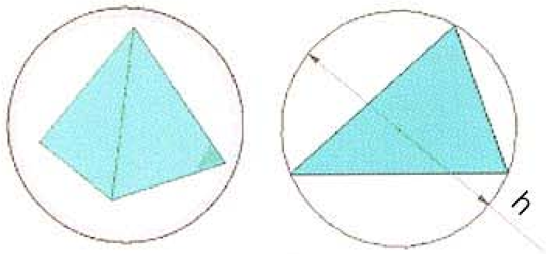
Let us review the Mesh parameters dialog (see the figure on the previous page) in more detail. This dialog offers a choice of element size and element size tolerance.

Mesh Size

The suggested medium mesh density of 0.225 in. is the default that COSMOSWorks proposes for meshing our rectangular hollow plate model.

The element size of 0.2253 inch and the element size tolerance of 0.011 inch are established automatically, based on the geometric features of the SolidWorks model. COSMOSWorks uses inches for the element size because the units of length are inches in the SolidWorks model. Remember, however, that we can enter analysis data and analyze results in any one of three unit systems: SI, Metric and English.

The 0.2253 inch size represents the characteristic element size in the mesh and is defined as the diameter of a sphere circumscribing the element (on the left in the following figure). This representation is easier to illustrate with the 2-D analogy of a circle circumscribing a triangle (on the right in the following figure).



Mesh density directly affects the accuracy of results. The smaller the elements, the lower the discretization errors, but the longer the meshing and solution times.

In the majority of analyses with COSMOSWorks, the default mesh settings produce a mesh that provides acceptable discretization errors while keeping solution times reasonably short.

Tolerance

This field sets the tolerance in the global size of finite elements. The default value is equal to 5% of the global element size. In some instances, when the mesher fails to mesh the model, increasing the tolerance may help.

---

15 Display/hide mesh.

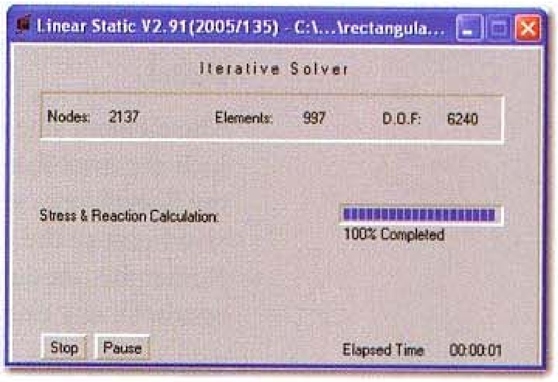
Mesh visibility can be controlled by right-clicking `Mesh`, and then doing one of the following:

- Select **Hide Mesh**.
- Select **Show Mesh**.

16 Run the analysis.

Right-click the study icon, `mesh1` and select **Run**.

You can monitor the solution progress in the solver window while the analysis is running.



Postprocessing

After the analysis is complete, COSMOSWorks automatically creates the `Results` folder with the default results plots that we specified at the beginning of the lesson: `Stress1 (-vonMises-)`, `Displacement1 (-Res disp-)`, and `Strain1 (-Equivalent-)`.

Plots

Each result plot can be displayed by doing one of the following:

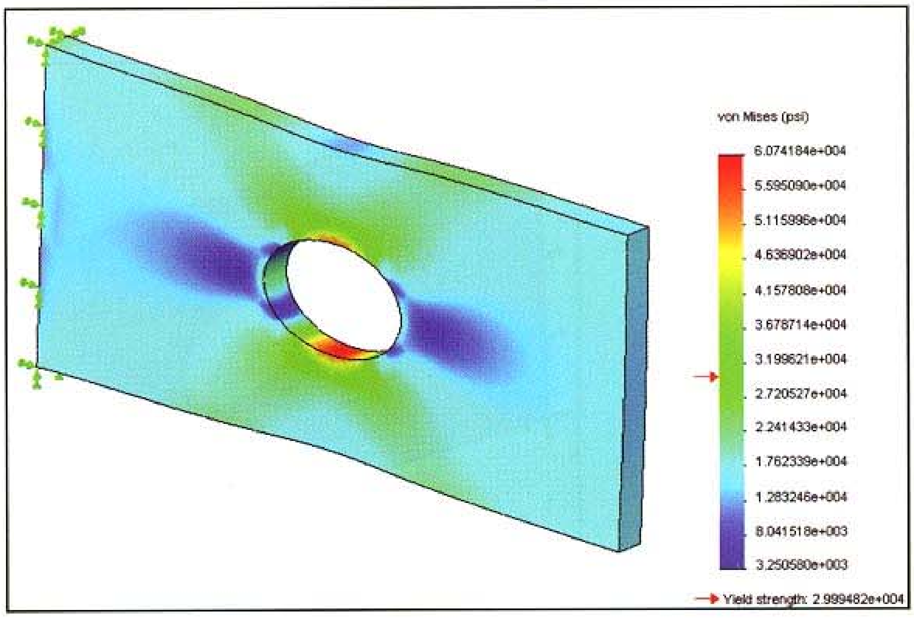
- Double-click the desired plot icon (`Stress1`, for example).
- Right-click the desired plot icon (`Stress1`, for example) and select **Show** under any folder.

While a plot is active (appears in the model window) you can right-click the plot icon again to examine the plot control options.

17 Show and Edit `Stress1 (-vonMises-)` plot.

Double-click on `Stress1 (-vonMises-)` under the `Results` folder to display the plot.

Notice that the stress plot is in psi units and the legend features scientific numbers with six digits, just as we requested in the **Options** at the beginning of the lesson.



We observe that the maximum value of Von Mises stress is 60.7 ksi, which significantly exceeds the yield stress of the material, 29.99 ksi, indicated by the red marker in following the chart.

To edit this plot, right-click on `Stress1 (-vonMises-)` and select **Edit definition**.

The **Display** dialog lets the user specify a stress component, units, and the type of plot. Also, you can choose to plot either **Node** or **Element values** (see discussion below on node and element values).

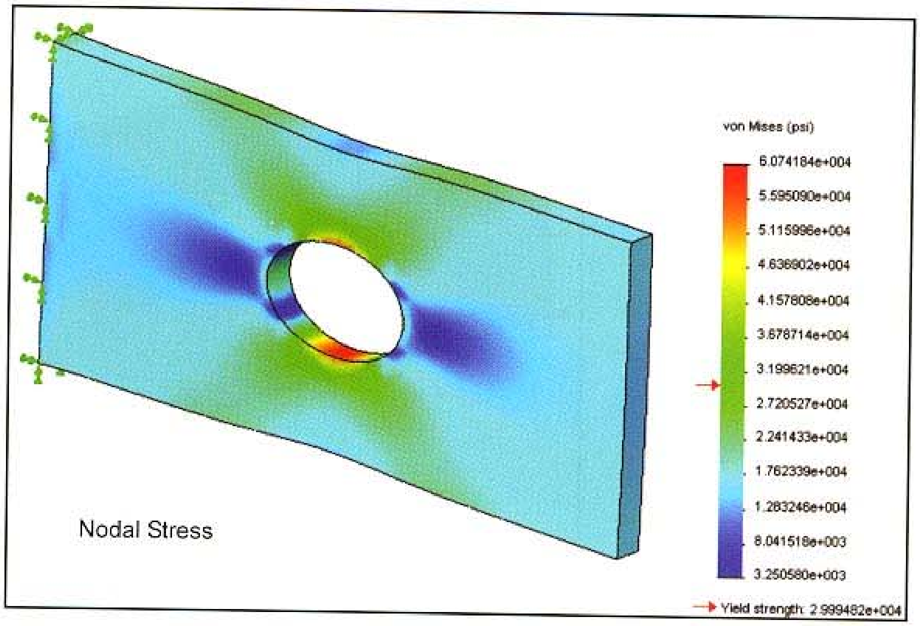
The **Deformed Shape** dialog lets the user specify the deformation scale for the plot. **Automatic** (default), **True scale**, and **Defined** (user specified) scale options are available.

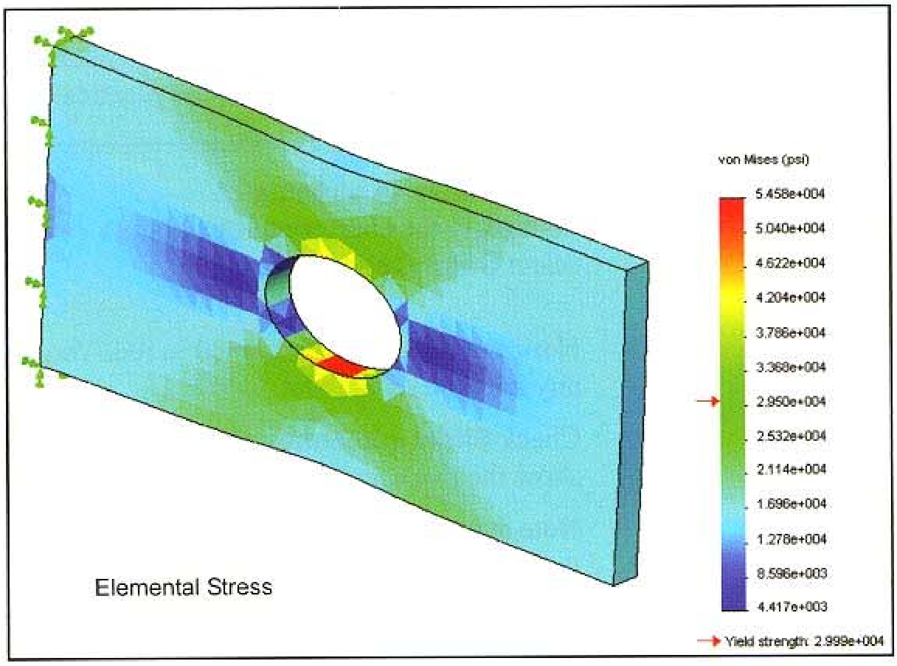
Students are encouraged to experiment with these options.

Click **OK**.

Nodal vs. Element Stresses

As mentioned above, the Edit Definition dialog lets the user choose between the nodal and elemental quantities. The following figures show the nodal and elemental values of the Von Mises stress for our model.





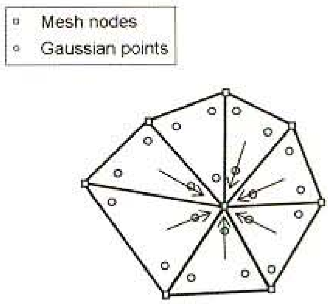
The stress plot that displays **Nodal values** appears "smooth", while the stress plot that displays **Element values** appears "rough".

To understand the reasons for these different appearances, we need to explain the differences between nodal and element stresses.

During the solution process, in each element, stress results are calculated at certain locations called Gauss points. First order tetrahedral elements (draft quality) have one Gauss point in their volume. Second order tetrahedral elements have four Gauss points. First order shell elements have one Gauss point. Second order shell elements have three Gauss points.

Nodal values

Stresses in Gauss points can be extrapolated to element nodes. Most often, one node is shared by several elements, and each element reports different stresses at the shared node. Reported values from all adjacent elements are then averaged to obtain a single value. This method of stress averaging produces averaged (or nodal) stress results.



Element values

Alternately, the stress values from all Gaussian points within each element can be averaged to report a single elemental stress. Although these stresses are averaged between Gauss points, they are called non-averaged stresses (or element stresses) because the averaging is done internally within the same element.

Element stresses and nodal stresses are always different, but too large a difference indicates that the mesh is not sufficiently refined in that location. (See Exercise 1 for the practical use of these quantities.)

---

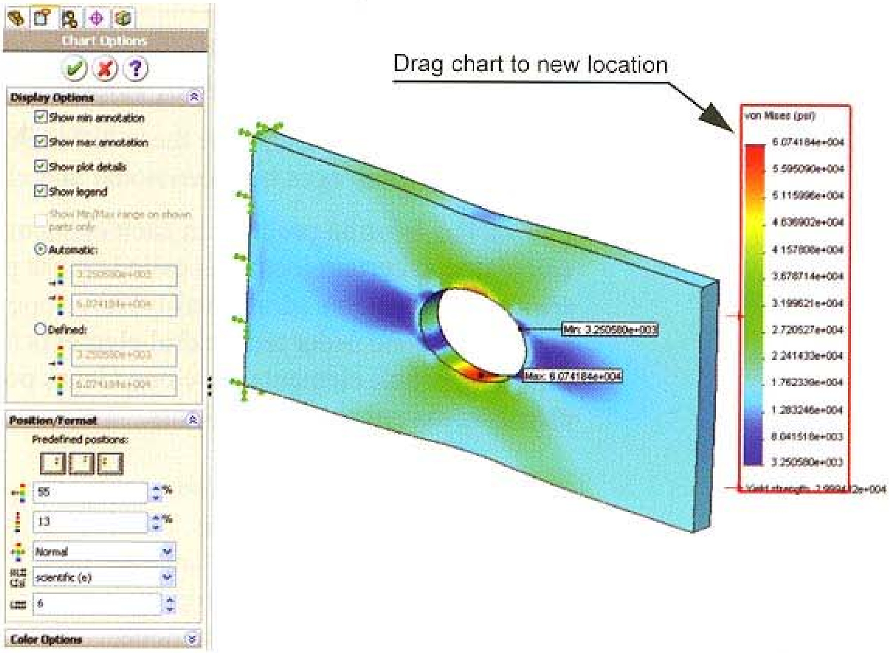
18 Modify chart in stress plot.
To modify the chart, right-click on `Stress1 (-vonMises-)` and select **Chart Options**.

Notice that the chart is framed in red. You can now drag the chart to any location on the plot.

Check **Show min annotation** and **Show max annotation** boxes to show the markers in the plot.

Note that you can also modify the limiting values in the chart, format of the numbers, and the color options.

Click **OK** to save new settings.



19 Modify settings of stress plot.
Right-click on `Stress1 (-vonMises-)` and select **Settings**.

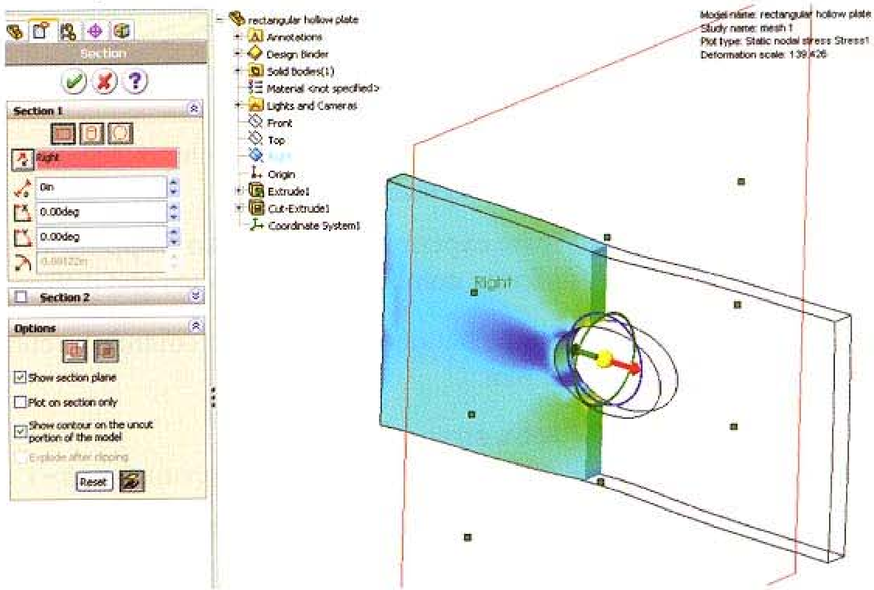
We encourage that students explore the Fringe, Boundary, and Deformed plot options in this dialog.

20 Create section plot.
In many applications it is useful to cut the model and look at the distribution of the result quantity in the through-thickness direction.

Right-click on `Stress1 (-vonMises-)` and select **Section Clipping**.

From the Solidworks fly-out menu, select `Right` plane as a **Reference entity**.

Students are encouraged to explore all the options and parameters in the **Section** dialog. Note that the user can also drag the triad to easily move the cut plane through the model.



Use the buttons and to control the cutting direction and to disable the section plot.

Click **OK** to close the **Section** dialog.

21 Create Iso plot.

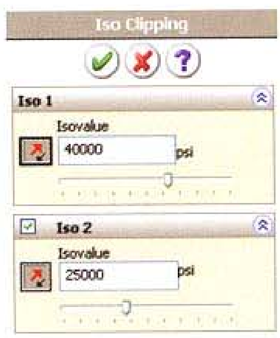
Suppose that we wish to display portions of the model where the von Mises stress is between 25,000 psi and 40,000 psi.

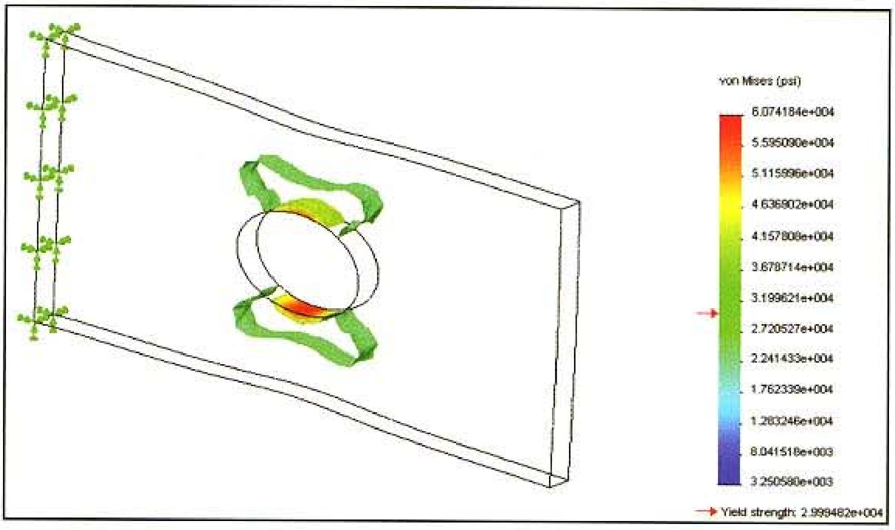
Right-click on `Stress1 (-vonMises-)` and select **Iso Clipping**. This opens the **Iso Clipping** PropertyManager.

In the **Isovalue** box, under the **Iso1** dialog, enter **40,000 psi**.

Check **Iso 2** and in the **Isovalue** box, enter **25,000 psi**.

Click **OK**.





Note that arrows on the stress legend indicate the values defined for the two iso surfaces.

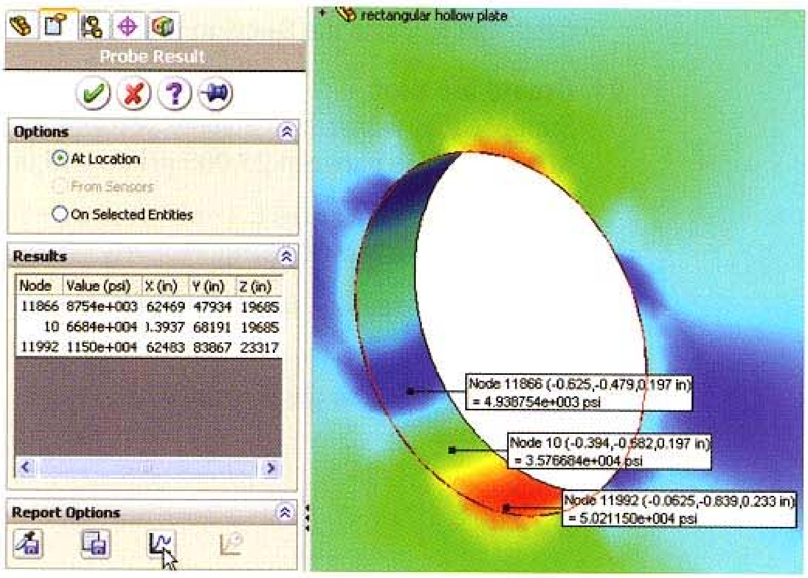
Please experiment with the **Iso Clipping** window options using different numbers of iso surfaces and different cutting directions.

Use the buttons and to control the cutting direction and to reset the plot.

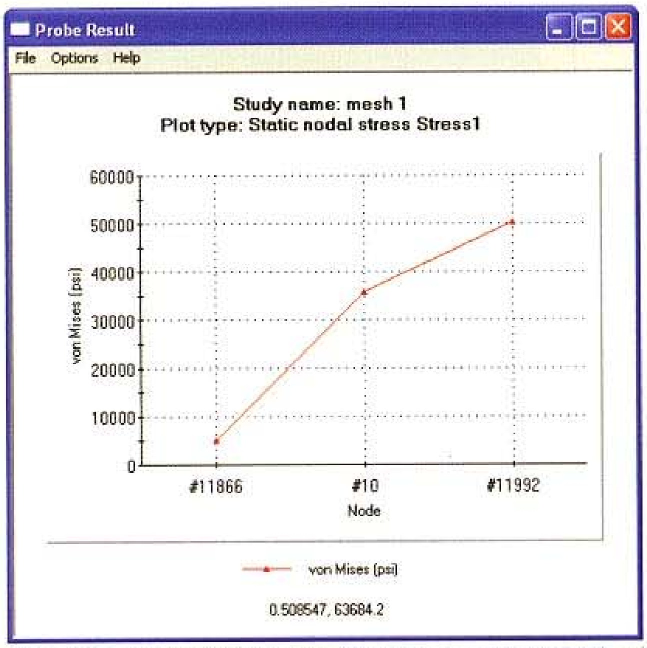
22 Probe stress results.
Right-click on `Stress1 (-vonMises-)` and select **Probe**.

Using the pointer, click the desired locations on the plot. It helps to zoom in on the area.

The stress results are listed in the **Results** dialog table and in the plot at the selected locations.



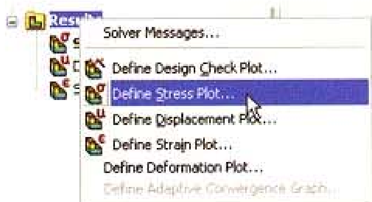
Under the **Report Option** dialog, you can save the results in a file, plot the path-graph, or save the locations as sensors. (Sensors are discussed in detail later on in the class.)



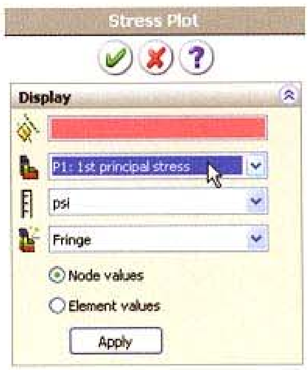
The figure above shows a Von Mises stress path plot for the selected locations.

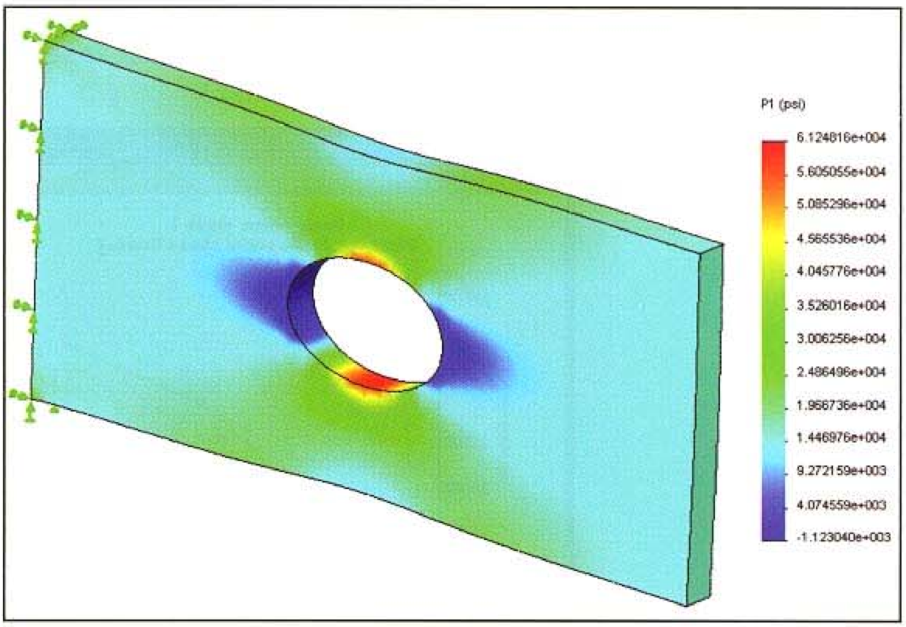
23 Define 1st principle stress plot.

To define a new stress plot, right-click on the `Results` folder and select **Define Stress Plot**.



An already familiar definition dialog appears. Select **P1: 1st principle stress** as the stress component, keep all other default options, and click **OK**.





We observe that the maximum value of the 1st principle stress, 61.2 ksi, is very close to the maximum value of the Von Mises stress, 60.7 ksi. This is because the specified `Tensile load` is the only dominant load component resulting in predominantly tensile stress along the longitudinal direction of the plate.

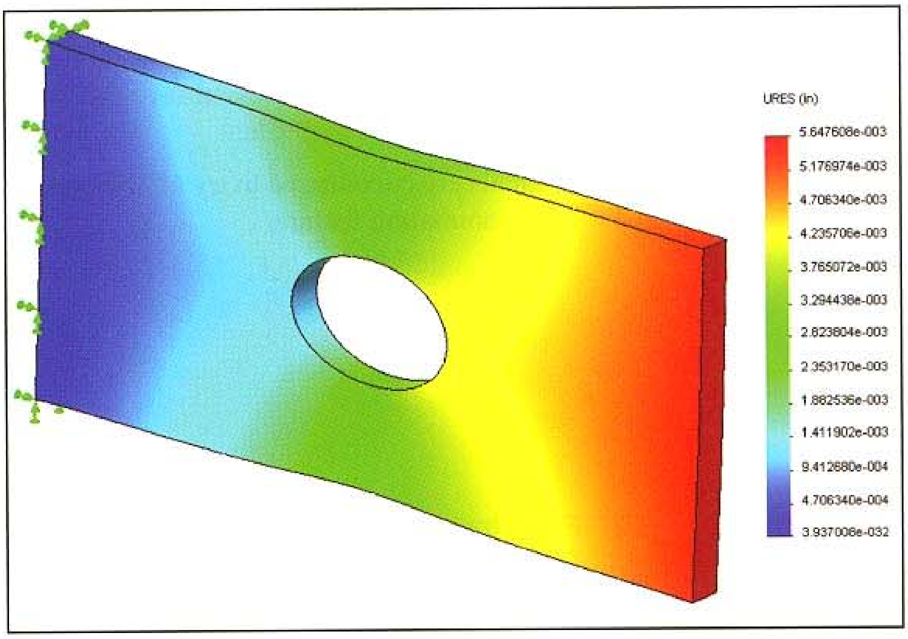
24 Define displacement plot.

Double-click the `Displacement1 (-Res disp-)` plot icon.

The post processing features that we practiced in the case of `Stress1 (-vonMises-)` are applicable to all other result quantities, such as Displacement.

Right-click `Displacement1 (-Res disp-)` and select **Edit definition**. Under **Units**, verify that **inches** are selected.

Close the window.



The displacement shows a maximum resultant displacement of 0.005710 in.

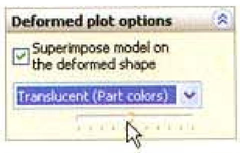
Note

We record the displacement result with 6 digits only to practice the plot options and to compare results from studies with different meshes. The uncertainties and simplifying assumptions used to create the model do not justify this accuracy.

25 Superimpose undeformed shape.

Right-click on `Displacement1 (-Res disp-)` and select **Settings**.

Activate the **Superimpose model on the deformed shape** option. You can move the slider to adjust the transparency of the undeformed image.

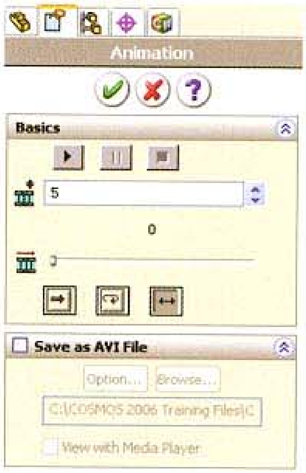


26 Animate displacement plot.

To animate the displacement plot, right-click on `Displacement1 (-Res disp-)` and select **Animate**.

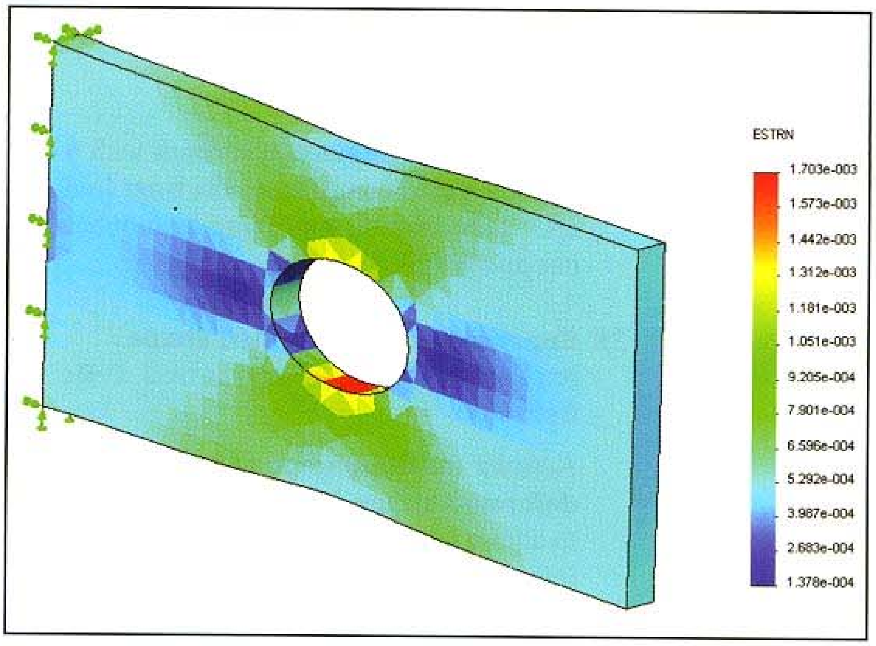
In the **Animation** PropertyManager you can start and stop the animation, set the number of frames, control the speed, and save the animation as an `.avi` file.

Students are encouraged to try all the options of the animation feature.



27 Plot strain results.

Double-click the `Strain1 (-Equivalent-)` plot icon to show the plot.



Note that strain results are dimensionless.

As opposed to stress results, which are shown as averaged (node values) by default, strain results are shown as non-averaged (element values).

Examine the strain plot showing **Element Values**.

To review the averaged strain plot, right-click on `Strain1 (-Equivalent-)` and select **Edit Definition**, and then select **Node Values**.

To examine the available chart options, right-click `Strain1 (-Equivalent-)` and select **Edit Definition**.

All post processing features that we practiced for the stress plot are available for strain plots as well.

We have completed the analysis of rectangular hollow plate with a coarse mesh and now wish to see how a change in mesh density affects the results. For this reason, we repeat the analysis one more time using fine density mesh.

Multiple Studies

To repeat the analysis with a more refined mesh, we can create a new mesh while still in the `mesh1` study, but this action would overwrite the old results.

To preserve the results of study `mesh1` that we have completed, we will create a new study, `mesh2`. Creating a new study can be done in several ways.

28 Create new study.

A new study can be created in one of two ways:

- Create a new study from scratch.
- Copy the `mesh1` study by dragging the `mesh1` study icon onto the `rectangular hollow plate` icon. (Same as right-clicking `mesh1` and selecting **Copy**; then right-clicking `rectangular hollow plate` and selecting **Paste**.)

Either way COSMOSWorks displays the **Define Study Name** window.

Using the second approach, type the new study name `mesh2`.

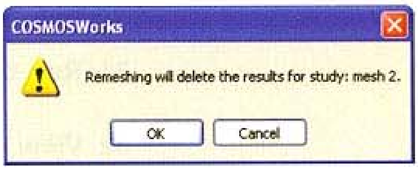
Click **OK**. The new study, `mesh2`, is identical to the study `mesh1`.

Note

The study settings, Load/Restraint, Mesh, and the study results have been copied as well.

29 Create new mesh in study `mesh2`.

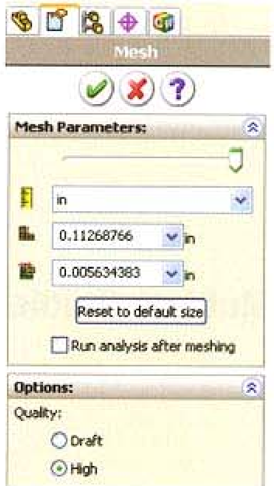
In study `mesh2`, right-click `Mesh` and select **Create**. A warning window appears.



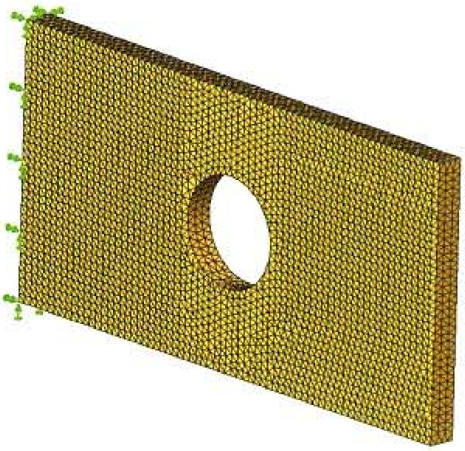
Click **OK** to open the **Mesh** window.

Move the "Mesh factor" slider all the way to the right. The **Element size** and **Tolerance** should read **0.112 in** and **0.005 in**, respectively.

Click **OK**.



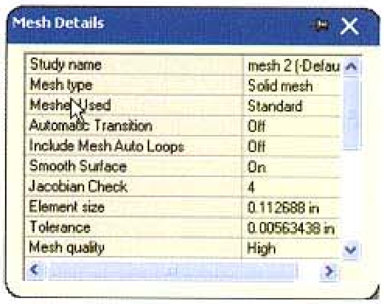
The generated mesh is displayed to the right.



30 Display mesh details.

Having created the mesh, we can access the detailed mesh information by right-clicking `Mesh` and selecting **Details**.

The same detailed information can of course be displayed for the "old" mesh in the `mesh1` study.



31 Run the analysis.

32 View displacement and stress results.

Record the maximum displacement (0.005712 in.) and the maximum von Mises stress (60,820 psi).

Note

All plot settings remain the same as in the study `mesh1` because the plot definitions are copied from the study `mesh1`.

Check Convergence and Accuracy

Now we must collect information from both studies (`mesh1` and `mesh2`) to compare the displacement and maximum von Mises stress results for the two different meshes. We can determine the maximum displacement and the maximum von Mises stress results in plots.

We must also determine the number of elements and the number of nodes in each mesh. These can be found in the **Mesh Details** window of each respective mesh.

Finally, we must determine the number of degrees of freedom (DOF) in each model. To calculate this number, we could count the number of unconstrained nodes by subtracting the number of nodes on the constrained face from the number nodes reported in mesh details. Then we could multiply this number by three because each node in a solid element mesh has 3 DOF.

33 View solver messages.
Right-click on `Results` and choose **Solver Messages**. Note the number of elements, nodes, and degrees of freedom.

Results Summary

The summary of the results produced by the two studies is shown in the following table:

Mesh density	Max. displacement [in]	Max. von Mises stress [psi]	Number of DOF	Number of elements	Number of nodes
Coarse study `mesh1`	5.710e-3	60,700	35,925	6992	12,160
Default study `mesh2`	5.712e-3	60,800	250,044	54,760	83,843

Note that all of the results of this table pertain to the same problem. The only difference is in the mesh density. You may find small differences between your own results and those presented in this table. This is due to service pack upgrades, etc. Having noted that the maximum displacement increases with mesh refinement, we can conclude that the model becomes less stiff (or softer) when the number of degrees of freedom increases. In our case, by selecting second order elements, we impose the assumption that the displacement field in each element is described by second order polynomial functions.

With mesh refinement, the displacement field in each element is still described by second order polynomial functions; however, the larger number of elements makes it possible to approximate the real displacement and stress fields more accurately.

We can say that the artificial constraints resulting from 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 both displacement and stress results converge to a finite value. This limit is the solution of the mathematical model. Differences between the solution of the FEA model and the solution of the mathematical model are due to discretization error. Discretization error diminishes with mesh refinement.

The process of consecutive mesh refinements that we have completed is called the convergence process. Its objective is to determine the impact of our discretization choices (element size) on the data of interest, which, in this lesson, are the maximum resultant displacements and the maximum von Mises stress.

Comparison With Analytical Results

An infinitely long rectangular hollow plate under a tensile load possesses the analytical solution. We compare FEA results with analytical results.

W, D and T denote plate width (100 mm), hole diameter (40 mm) and plate thickness (10 mm). P is the tensile load 25,000 lb. or 111,210 N. For comparison with analytical results, it is more convenient to use the SI system because the SolidWorks model have been defined in mm.

σ_n is the normal stress in the cross section where the hole is located, K_n is the stress concentration factor, and σ_{max} is the maximum principal stress.

$$\sigma_n = \frac{P}{(W-D) \times T} = \frac{111210}{(100-40) \times 10} = 185.35MPa$$

$$K_n = 2 + \left(1 - \frac{D}{W}\right)^3 = 2 + \left(1 - \frac{40}{100}\right)^3 = 2.216$$

$$\sigma_{max} = K_n \times \sigma_n = 185.35 \times 2.216 = 410.74MPa$$

Review the **P1: 1st principle stress** plot for study `mesh1`. The maximum value reached 61.24 ksi, which corresponds to approximately 426.9 MPa.

Therefore, the difference is:

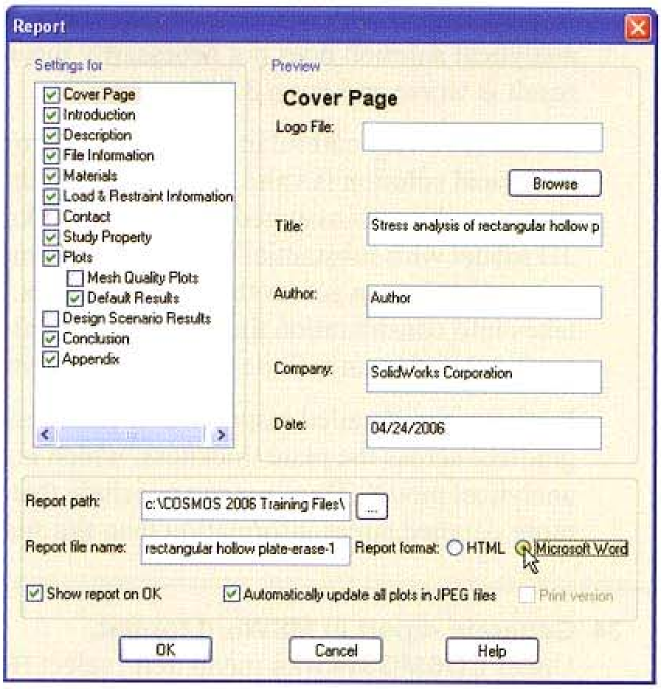
$$difference = \left[\frac{COSMOS - THEORY}{COSMOS}\right] = \left[\frac{426.9 - 410.74}{426.9}\right] = 3.78\ \%$$

The difference of 3.78% between the COSMOSWorks result and the analytical solution does not necessarily mean that the COSMOSWorks result is worse and has a 3.78% error.

We must be very careful in how we compare these results. Note that the analytical solution is valid only for a very thin plate where a plane stress condition is assumed. COSMOSWorks calculates a solution for a 3D model with substantial thickness (10 mm) and accounts for realistic stress distribution across the plate thickness. COSMOSWorks also takes into consideration the fact that the plate has a finite length (200 mm) rather that an infinite one, as the analytical solution does.

Furthermore, detailed inspection of the stress results show the stress gradient across the plate thickness, which is not accounted for in the analytical model. Thus, we can conclude that COSMOSWorks provides more detailed stress information than the analytical solution.

34 Generate report in MSWord format.
Under **COSMOSWorks** menu item, select **Report**.



Under **Settings for**, select the required report parts. (For example, deselect the option **Contacts**, as we do not have any in this analysis.)

Select **Microsoft Word** as **Report format**, enter the **Report file name**, select the option **Automatically update all plots in JPEG files**, and click **OK**.

Note that the report may also be generated in an HTML format.

35 Save and close the part.

.

Summary

We used a simple model of a hollow rectangular plate to introduce the COSMOSWorks interface and, at the same time, to go through all major steps in the FEA process.

We created multiple studies to execute a linear static analysis with two different meshes.

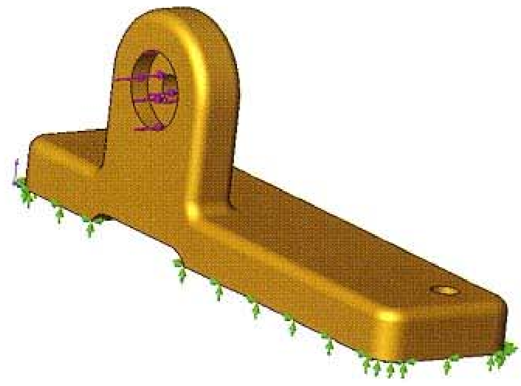
While preparing models for analysis and examining results obtained with different meshes, we introduced the concept of modeling error and discretization error.

This first lesson was intended to provide an understanding of FEA methodology and the software skills necessary to complete the lessons that follow.

Exercise 1: Static Analysis of a Part

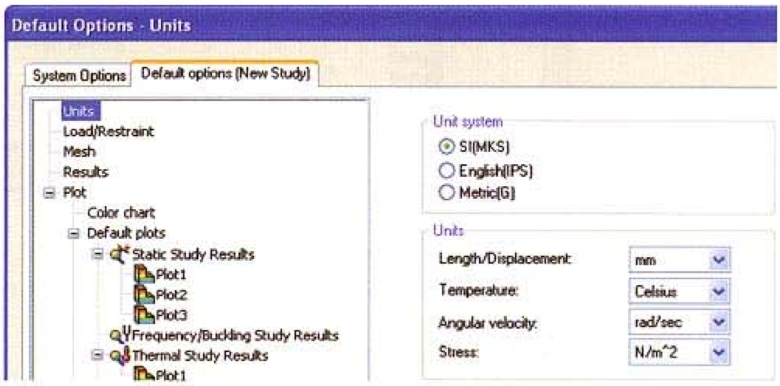
Problem Statement

The Aluminum Part of an assembly will be analyzed for its maximum stresses and displacements. The fashion in which the part is attached to the rest of the assembly can be simulated by an Immovable constraint applied to the two flat bottom surfaces. The part is then subjected to a normal force of 200 N, as indicated in the figure.



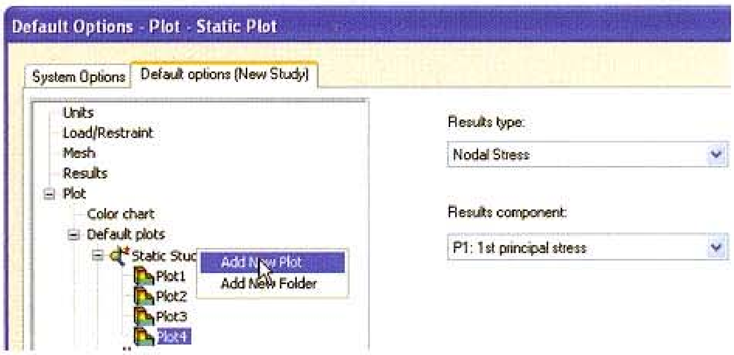
1 **Open part.**
Open the Solidworks part named `part`. The file is located in the `Exercise1` directory.

2 **Specify COSMOSWorks options.**
Under **COSMOSWorks** in the main menu, select **Options**.

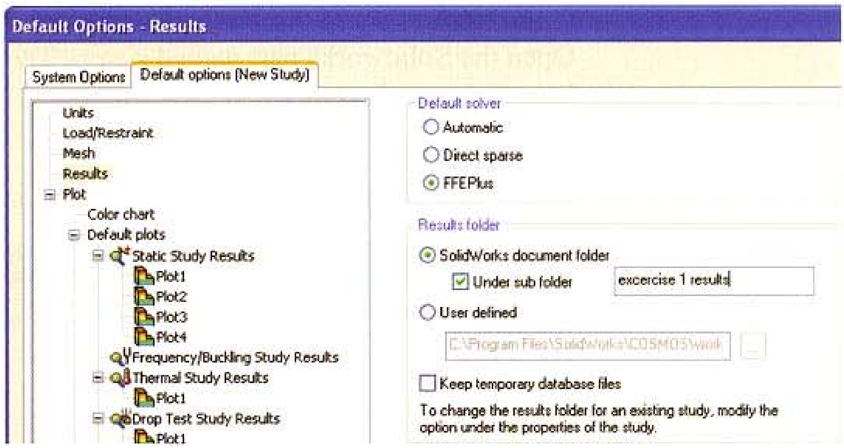


Under the **Default options (New Study)** tab, specify **SI(MKS)** as a default **Unit system** for this analysis. In the Units dialog, set the **Length/Displacement** and **Stress** fields to **mm** and **N/m^2 (Pa)**, respectively.

The following default results plots are generated after each static study is completed: Nodal von Mises stress, resultant displacement, and equivalent elemental strain.



Still in the **Default options (New Study)** tab, under `Plot`, `Default plots`, right-click on the `Static Study Results` folder and select **Add New Plot**. Request that an additional result plot for the nodal **P1: 1st principle stress** be generated as a default result plot.



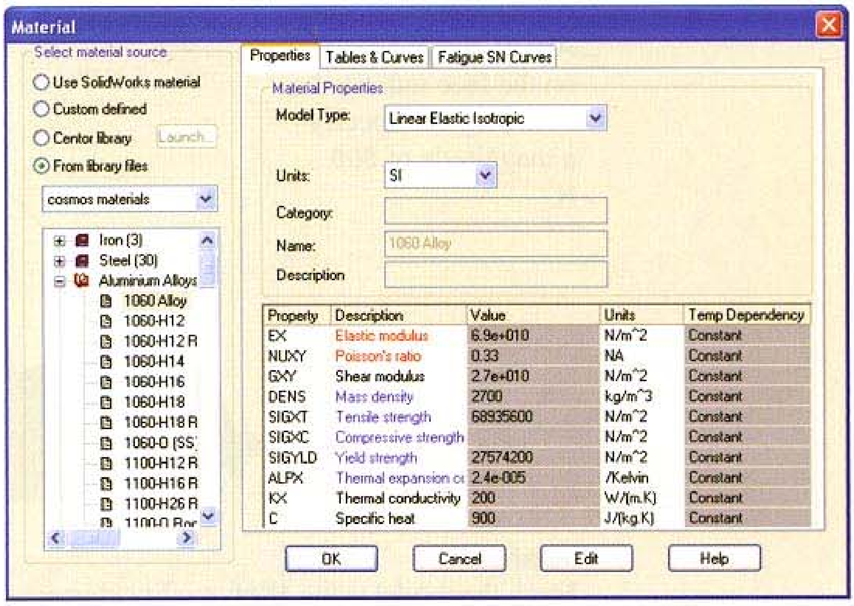
Specify the folder `exercise 1 results` in the SolidWorks document directory as a location where the results will be saved.

3 Define a static study.

Switch to the COSMOSWorks Manager and create a **Solid mesh**, **Static** study named `Static analysis`.

4 Apply material properties.

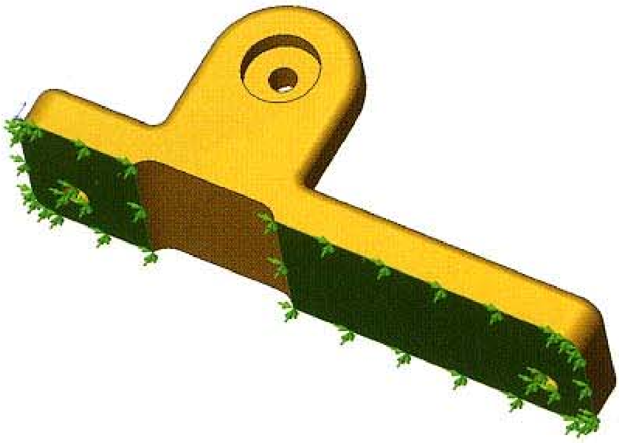
Right-click on the `Solids` folder in `COSMOSWorks Feature Manager` and select **Apply Material to All**.



Specify **Aluminum Alloy 1060** from the `cosmos materials` library.

5 Apply Immovable restraints.

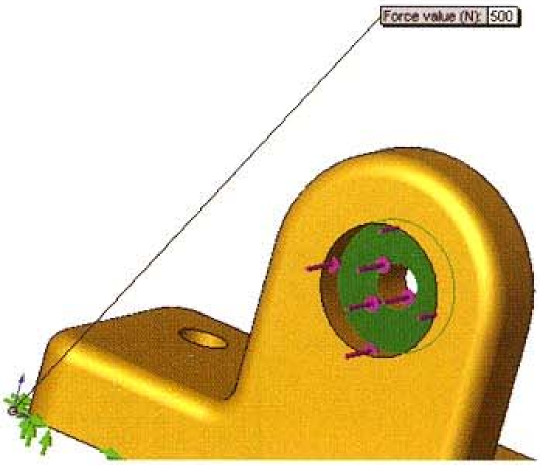
Apply **Immovable** restraints to the two bottom faces, as shown in the figure below.



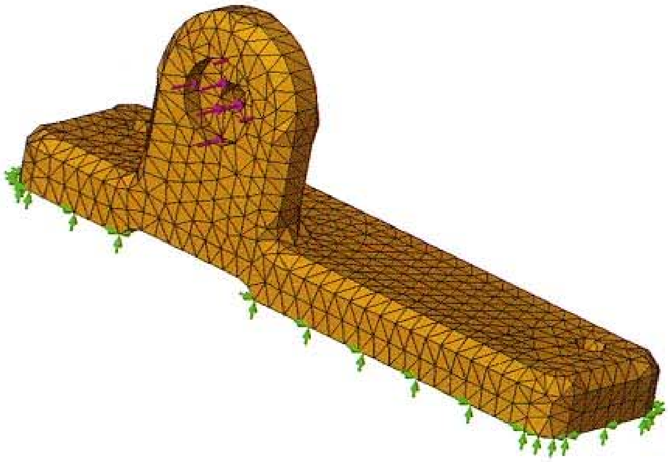
s

Note that this restraint simulates the way this part is attached to the rest of the assembly.

6 **Apply force.**
Apply normal force on the face indicated in the figure. Specify a magnitude of **500 N**.



7 **Mesh.**
Mesh the model using **High** quality elements. Use the default **Element size** and **Tolerance** of **4.39 mm** and **0.219 mm**, respectively.

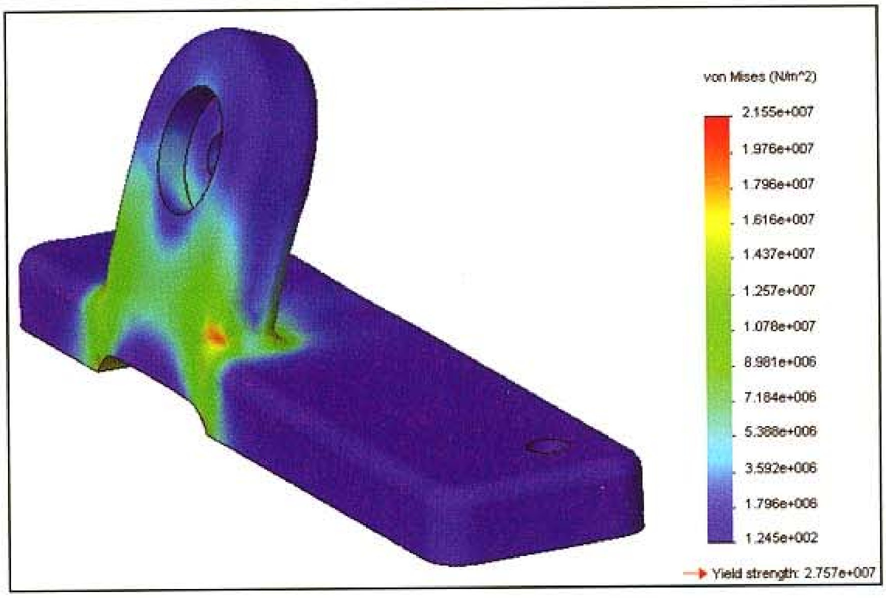


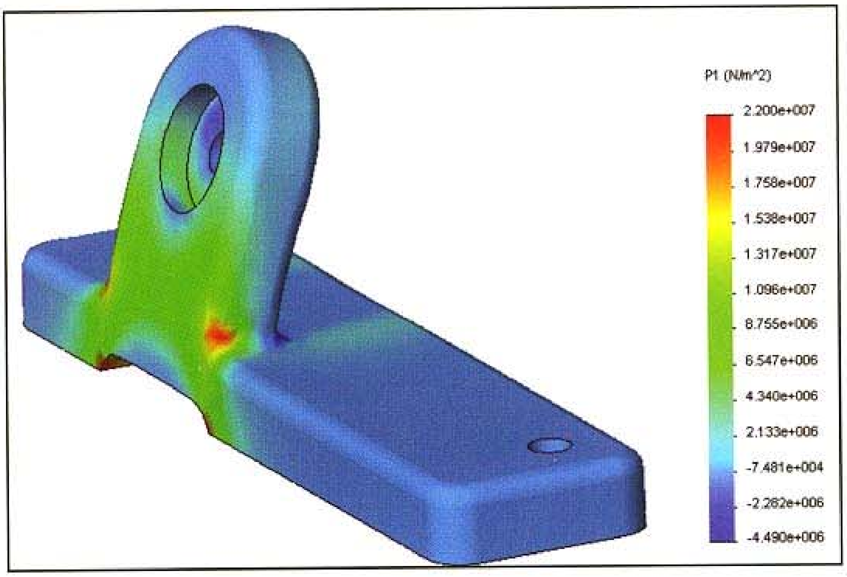
8 **Run the study.**

9 Plot stress results.

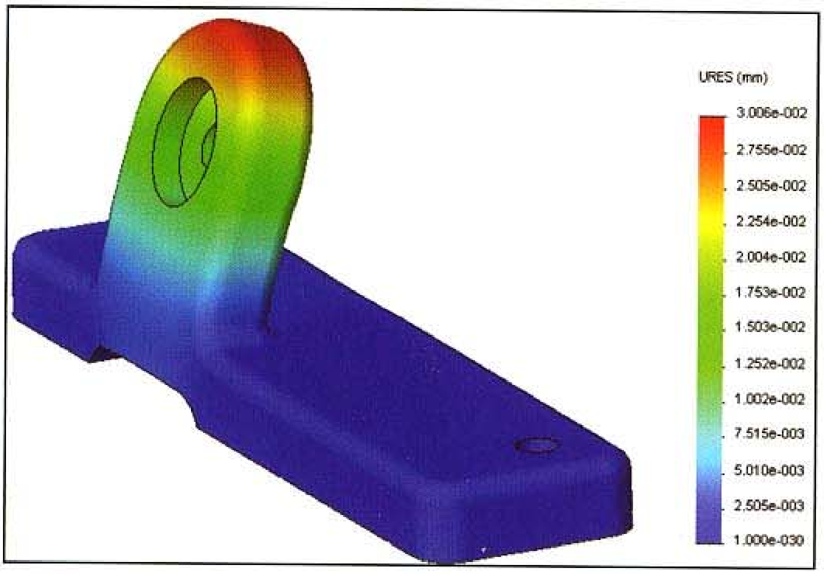
We observe that the maximum von Mises stress in the model is approximately 16.3 MPa, which is well below the yield strength of the 1060 Aluminum Alloy (27.5 MPa).

The distribution of the **P1: 1st principle stress** indicates a maximum value of approximately 22 MPa. This value corresponds to the maximum tensile stress in the part (maximum compressive stress where the value is negative).





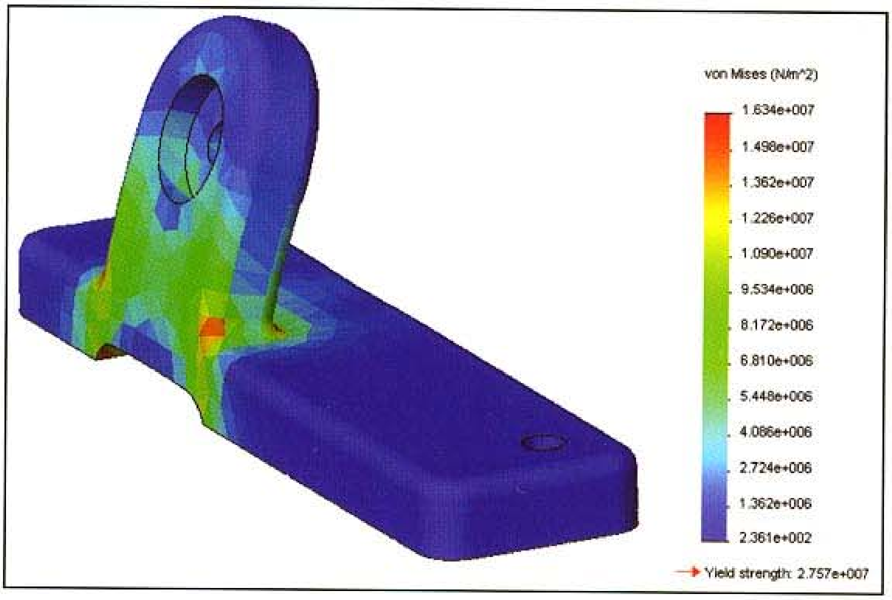
10 Plot displacement results.



We observe the maximum resultant displacement of approximately 0.03 mm.

Coarse mesh and Element stress

Are our current results accurate enough? Visual inspection of our finite element mesh suggests that it is rather coarse, especially in the regions where the fillets are present. Furthermore, inspection of the distribution of the elemental values of the von Mises stress indicates considerable stress jumps from element-to-element in the higher stress concentration areas.



To display the distribution of the elemental values of the von Mises stress, edit plot `Stress1 (-vonMises-)` and specify **Element values.**

We will repeat the analysis with finer mesh.

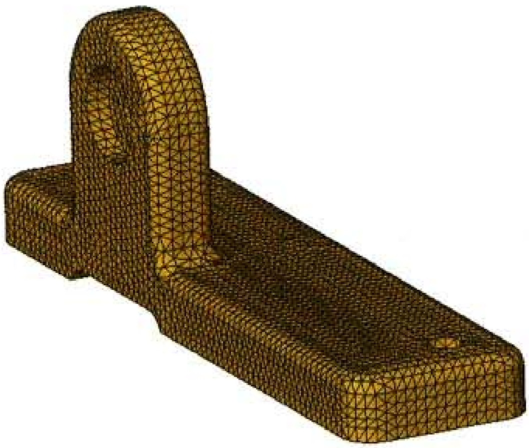
11 Create new static study.

Copy study `Static analysis` into a new study named `Static analysis - refined.`

The folders `Load/Restraint`, `Solids`, `Mesh`, and `Results` will be copied into the new study as well.

12 Create fine mesh.

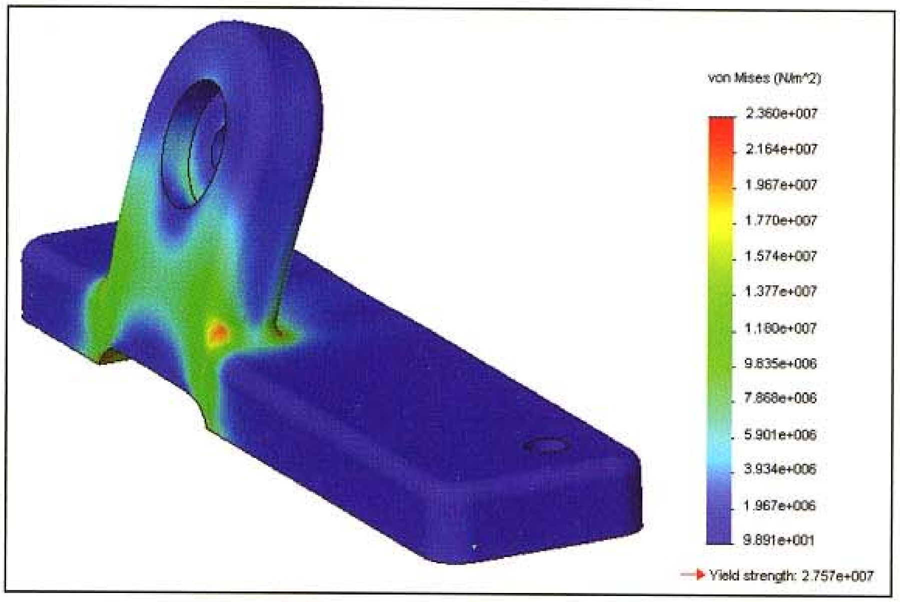
Create **High** quality mesh with the **Element size** of **2.198 mm** and the **Tolerance** of **0.109 mm**. (Slide the Mesh factor all the way to the right for these settings.)



The resulting mesh can be seen in the figure to the right. We observe significantly improved mapping of the model geometry.

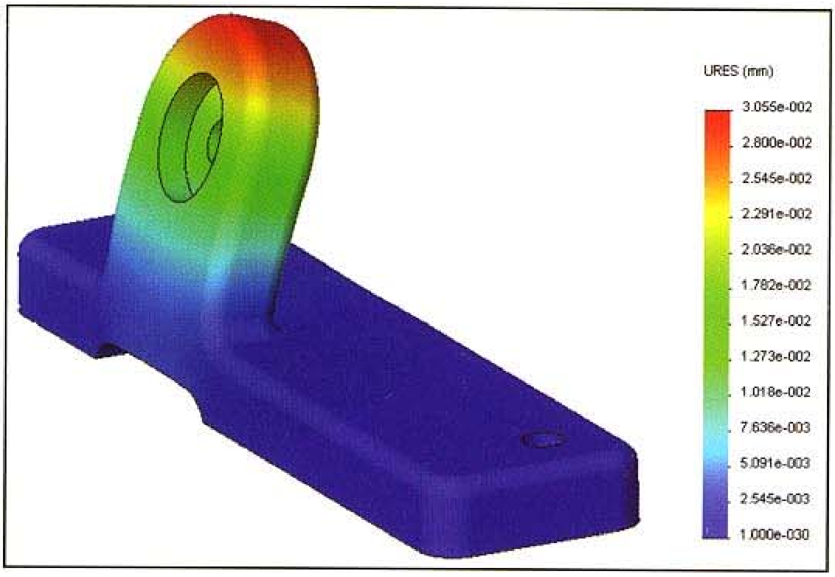
13 Run the study.

14 Plot stress results.



We now observe that the maximum von Mises stress increased from 16.3 MPa to 23.6 MPa, which is still below the material yield strength of the 27.5 MPa. This translates to a difference of nearly 31%.

15 Plot displacement results.



The above plot shows that the maximum displacement resultant increased from 0.03 mm to 0.0305 mm; a difference of approximately 1.6%.

Summary

In this exercise, we practiced the basic setup of the linear static study as well as the post processing features available in COSMOSWorks. We observed that the mesh quality has a significant impact on the results (especially the stress results). While the deviation in the resultant displacements obtained from the two studies was only 1.6%, the deviation for von Mises stresses was nearly 31%. The greater difference in the stresses is attributed to the following two phenomena:

- Displacements are the primary unknown in the finite element analysis and, as such, will always be significantly more accurate than strains and stresses. A relatively coarse mesh is sufficient for satisfactory displacement results, while significantly finer mesh is generally required for satisfactory stress results.
- The extreme values of the stresses occur in the high curvature regions where fillets are present. It is in these regions where our mesh needs to be refined for satisfactory results. This is a subject studied in Lesson 2.

Lesson 2
Static Analysis of an L-Bracket

Objectives

Upon successful completion of this lesson, you will be able to:

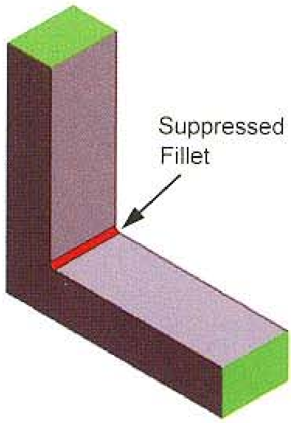
- Illustrate the differences between modeling and discretization errors
- Use Automatic transition option to mesh models
- Use mesh controls
- Describe when the lack of convergence of FEA results may occur
- Analyze model in different SolidWorks configurations
- Run multiple studies in a batch mode
- Extract reaction forces

Project Description

An L-shaped steel bracket is fixed at the top and a 200 lb. load is applied to the lower end face. We wish to evaluate the displacements and stresses in the model.

The corner of the bracket is rounded by a small fillet. Since the radius of the fillet is small compared to the overall size of the model, we decide to suppress it. As we soon prove, suppressing the fillet is ill advised.

We will investigate the effect of different mesh sizes on the maximum displacement and stress results. Rather than refining the mesh uniformly in the entire model, which is called *global mesh refinement*, we refine the mesh locally, where high stresses are located. This is called *local mesh refinement*.



Analyses of a Bracket Without the Fillet

1 **Open part.**
Open the SolidWorks part `L bracket`.

In the SolidWorks ConfigurationManager, examine the two configurations: `fillet` and `no fillet`.

Activate the `no fillet` configuration for the analysis.

2 **Define static study.**
Toggle from SolidWorks to the COSMOSWorks environment.

Create a study named `mesh1`.

In the **Mesh** list, click **Solid Mesh**.

In the analysis **Type** list, select **Static**.

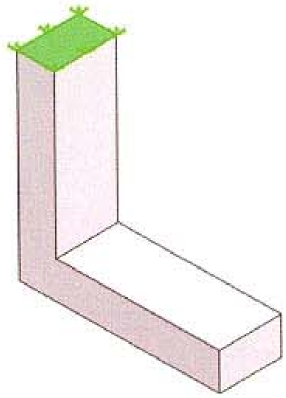
Click **OK**.

Note

The `L bracket` icon in the `Solids` folder already has a check mark next to the name of the assigned material because the material definition (AISI 304 steel) has been transferred from SolidWorks. Also, note that a sharp re-entrant corner takes the place of the suppressed fillet.

3 Apply fixed restraint.

Now apply a fixed support restraint to the top face using either **Fixed** or **Immovable** restraints.



To apply a fixed support restraint:

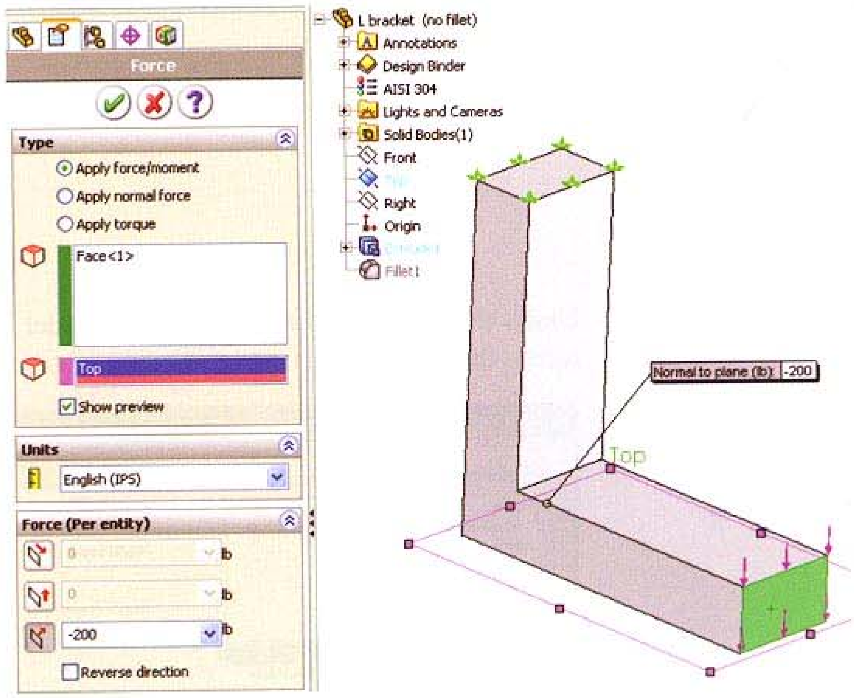
Right-click `Load/Restraint` and select **Restraints**.

In the restraint **Type** list, then select *either* **Fixed** or **Immovable**.

Click **OK**.

4 Apply force load.

Right-click `Load/Restraint` and select **Force**.



In the **Force** PropertyManager window, select the end face as the entity where force is to be applied, and then select `Top` plane as the reference plane to define the force direction.

The fly-out menu is useful for selecting the `Top` reference plane.

The direction of the bending load is normal to the `Top` reference plane. Therefore, enter **-200 lb** as the force component in the normal direction to the selected reference. Alternatively, enter **200** and select **Reverse Direction** check box in **Normal to Plane** force direction.

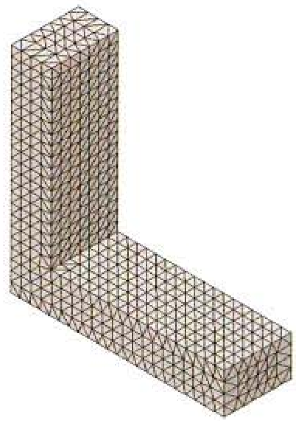
The definition of the mathematical model is now complete.

Click **OK**.

5 Mesh the model.

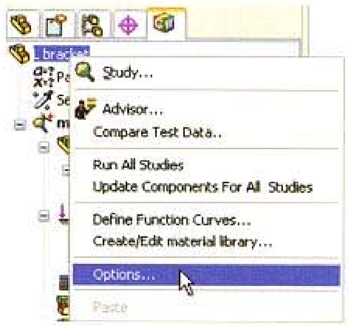
Verify that the meshing option is set to **High** quality, meaning that second order elements are created.

Mesh the model using the default **Element size** of **0.189 in** suggested by COSMOSWorks.

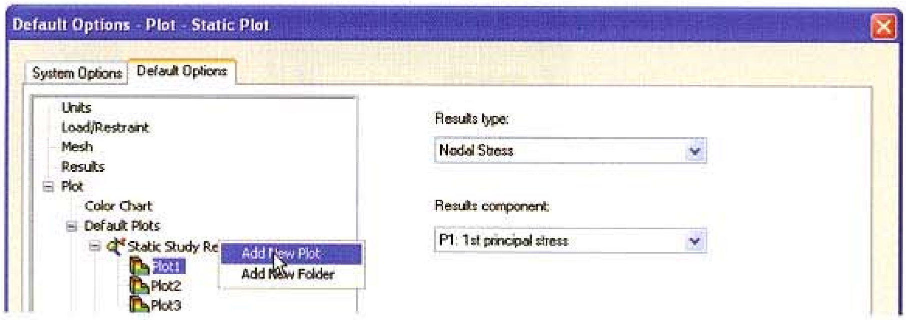


6 Set plot options.

Right-click the `L bracket` icon and select **Options.**



Under the **Default options** tab, in the folder `Plot`, `Default plots`, locate the subfolder `Static Study Results`.



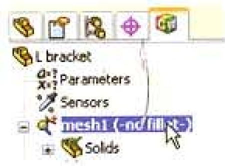
Make sure that an additional plot for the nodal 1st principle stress P1 is defined. If not, right-click on the `Static Study Results` folder and select **Add New Plot**.

Select **P1: 1st principle stress** under **Result component** and **Nodal stress** under **Result type**.

7 Create two new studies.

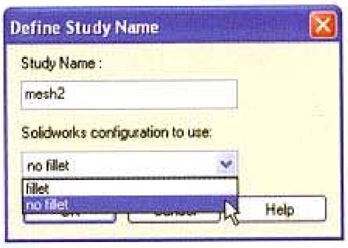
Study `mesh1` is now ready to be analyzed. However, we will create two more studies and analyze all three studies using the **Run all studies** command.

To copy the entire contents of `mesh1` into a new study, `mesh2`, drag the `mesh1` folder onto the `L bracket` part icon. This will open the **Define Study Name** dialog window.



In the **Study Name** field type `mesh2`.

In the **SolidWorks configuration to use** list, select **no fillet**.



Note In the **Define Study Name** dialog window, you can associate new studies to a particular SolidWorks configuration.

Click **OK**.

Analysis with Local Mesh Refinement

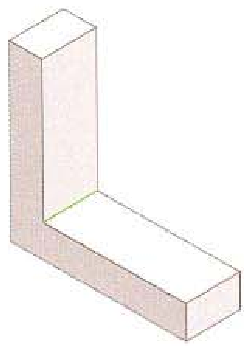
The study `mesh2` will investigate the effect of using smaller elements in the model on the results. In Lesson 1, we refined the mesh uniformly throughout the entire model by controlling the global element size.

In this lesson, we will use a different technique. Note that a stress concentration is located near the sharp re-entrant corner.

If you wish, you may run the study `mesh1` prior to running all studies in a batch mode to verify that the sharp edge indeed causes stress concentrations.

Knowing the location of high stress, we can refine the mesh locally in that area by applying local mesh controls.

8 Apply local mesh control for study `mesh2`.
To apply a mesh control, select the edge on which to apply mesh controls.

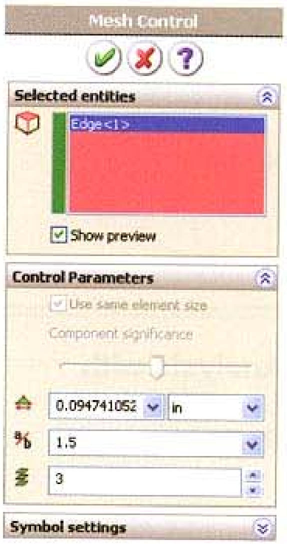


Right-click `Mesh` and select **Apply Mesh Control**. The **Mesh Control** PropertyManager appears.

As always, it is also possible to open the **Mesh Control** window first, and then select the desired entities on which to apply mesh controls.

Mesh controls allow you to control the element size locally on selected entities independent of the global element size. As compared to global mesh refinement, this is a more numerically efficient technique. Small elements are placed where needed, while portions of the model with no stress concentration are meshed with larger elements.

We will keep the suggested local **Element size** of **0.0947 in.**



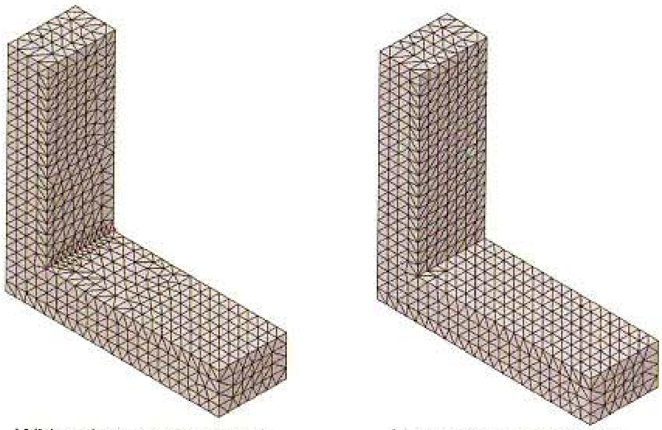
Keep the **Ratio** and **Layers** fields at their default values of **1.5** and **3**, respectively. (See the following discussion for an explanation of these options.)

Click **OK** to close **Mesh controls** PropertyManager.

9 Create mesh.

Create **High** quality mesh with the default global **Element size** of **0.189 in** and the default **Tolerance** of **0.00947 in.**

Note that smaller elements have been created along the edge where mesh control has been just applied.

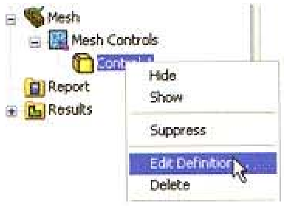


With edge mesh control　　　No edge mesh control

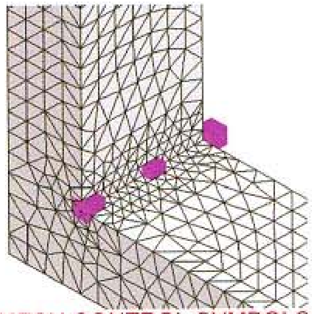
Mesh Controls

Mesh controls can also be applied to vertices, faces, or entire components of assemblies. Note that after mesh controls have been defined, the `Mesh` icon becomes a folder.

Mesh controls can then be edited using a shortcut menu displayed by right-clicking `Control-1` and select **Edit Definition** in the `Mesh` folder.



The mesh with applied control (also called mesh bias) features localized refinement along the edges.



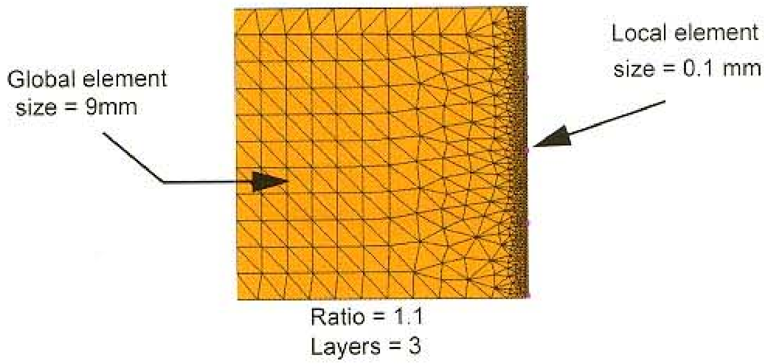
MESH CONTROL SYMBOLS

The **Ratio** parameter specifies the ratio between element sizes in consecutive transitional element layers. In our case, the default **Ratio** of **1.5** was used.

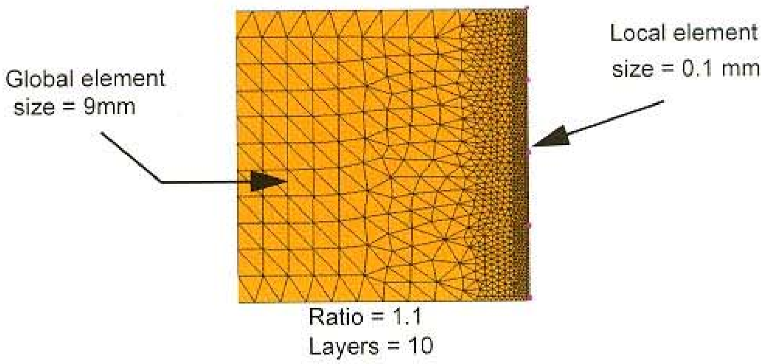
The **Layers** parameter controls the number of transitional element layers in which the specified **Ratio** is forced.

The following example will clarify the use of these two options.

a)



b)



In case *a)*, the specified growth ratio of 1.1 is applied to 3 layers. The element size at layer three is only $0.1 \times 1.1^3 = 0.13\text{mm}$, which is smaller than the global element size of 9 mm. The mesher, therefore, adds a few additional layers with a greater ratio to smoothly transition to the global element size of 9 mm.

In case *b)*, we force the specified ratio of 1.1 for 10 layers. Since at layer 10 the size of the element is still only $0.1 \times 1.1^{10} = 0.259\text{mm}$, the mesher adds a few additional layers with a greater ratio to arrive at the global element size of 9 mm.

If the number of layers specified at the mesh control is greater than the minimum number of layers needed to reach the global element size at the specified ratio, the mesher stops growing the element size once the global size is reached, i.e. the excessive layers are ignored.

Meshing must be done after controls are defined.

Mesh control symbols are displayed along the affected edge.

Mesh control symbols can be displayed or hidden by one of the following actions:

- Right-click `Mesh` and select **Hide All Control Symbols**
- Right-click `Mesh` and select **Show All Control Symbols**

The visibility of mesh control symbols can also be controlled individually for each mesh control.

Study `mesh2` is ready to be analyzed but we still need to prepare one more study before running all three studies using the **Run all studies** command.

---

10 Copy study `mesh2` into study mesh3.

11 Apply local mesh control for study `mesh3`.
In the `mesh3` study, right-click `Control-1` in the `Mesh` folder and select **Edit Definition**.

In the **Element size** box, enter **0.020 in** to locally refine the mesh along the sharp re-entrant edge. Keep the **Ratio** and **Layers** parameters at their default values of **1.5** and **3**, respectively.

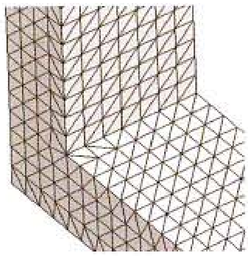
With this mesh control, we will create very small elements along the sharp re-entrant edge.

Click **OK**.

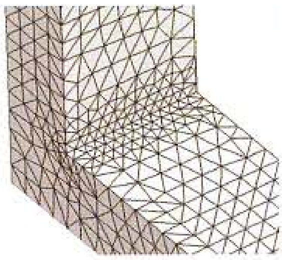
12 Mesh study `mesh3`.

Mesh study `mesh3` with **High** quality elements and the default global **Element size** of **0.189 in** and the default **Tolerance** of **0.0095 in**.

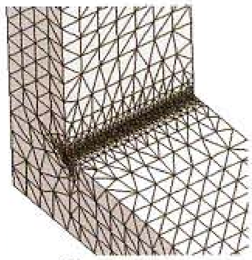
We now have three studies: `mesh1`, `mesh2` and `mesh3`. The only difference is mesh refinement along the sharp re-entrant edge.



Study: `mesh1`



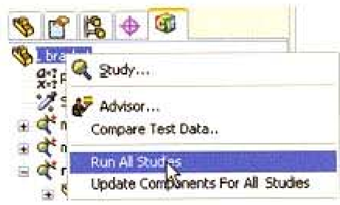
Study: `mesh2`



Study: `mesh3`

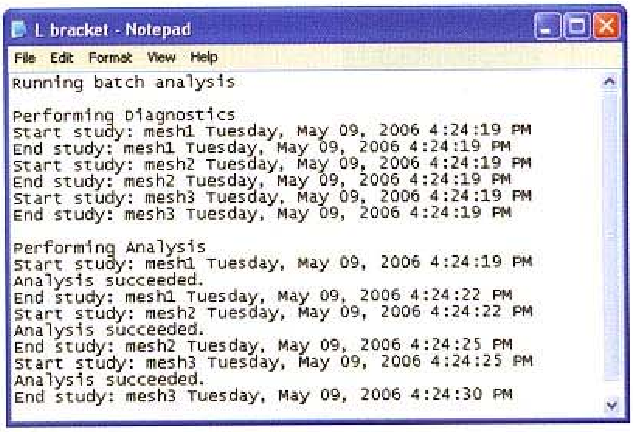
13 Run all studies.

To run all three studies, in the COSMOSWorks FeatureManager, right-click `L bracket` and select **Run all studies**.



Recall that in study `mesh1` we requested that, in addition to the default von Mises, resultant displacement, and equivalent strain result plots, P1: 1st principle stress plot be generated as well. Since `mesh2` and `mesh3` have been copied from `mesh1`, the same plots will be created in all three studies.

Once the analyses are completed, you may review the report in the `MSG` file located in the result folder.


L bracket - Notepad
File Edit Format View Help

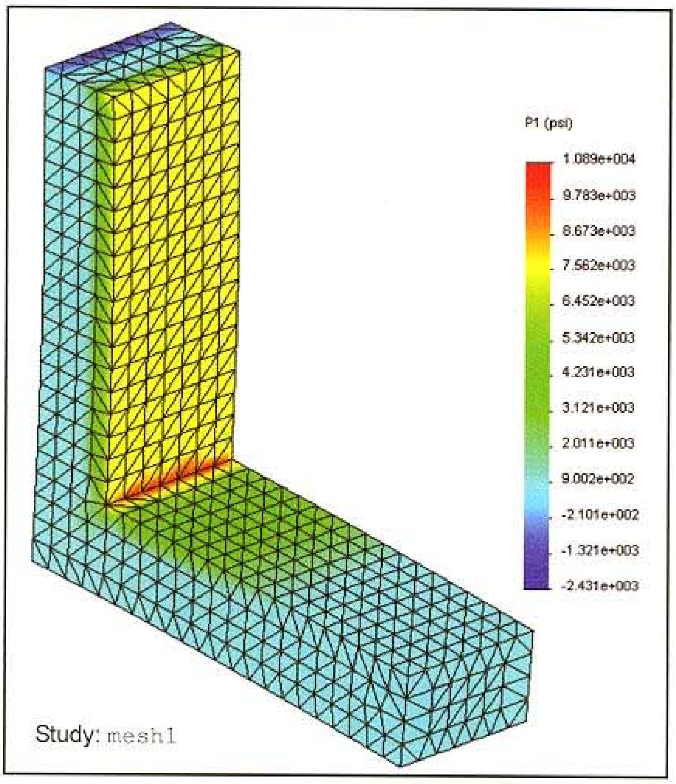
```
Running batch analysis

Performing Diagnostics
Start study: mesh1 Tuesday, May 09, 2006 4:24:19 PM
End study: mesh1 Tuesday, May 09, 2006 4:24:19 PM
Start study: mesh2 Tuesday, May 09, 2006 4:24:19 PM
End study: mesh2 Tuesday, May 09, 2006 4:24:19 PM
Start study: mesh3 Tuesday, May 09, 2006 4:24:19 PM
End study: mesh3 Tuesday, May 09, 2006 4:24:19 PM

Performing Analysis
Start study: mesh1 Tuesday, May 09, 2006 4:24:19 PM
Analysis succeeded.
End study: mesh1 Tuesday, May 09, 2006 4:24:22 PM
Start study: mesh2 Tuesday, May 09, 2006 4:24:22 PM
Analysis succeeded.
End study: mesh2 Tuesday, May 09, 2006 4:24:25 PM
Start study: mesh3 Tuesday, May 09, 2006 4:24:25 PM
Analysis succeeded.
End study: mesh3 Tuesday, May 09, 2006 4:24:30 PM
```

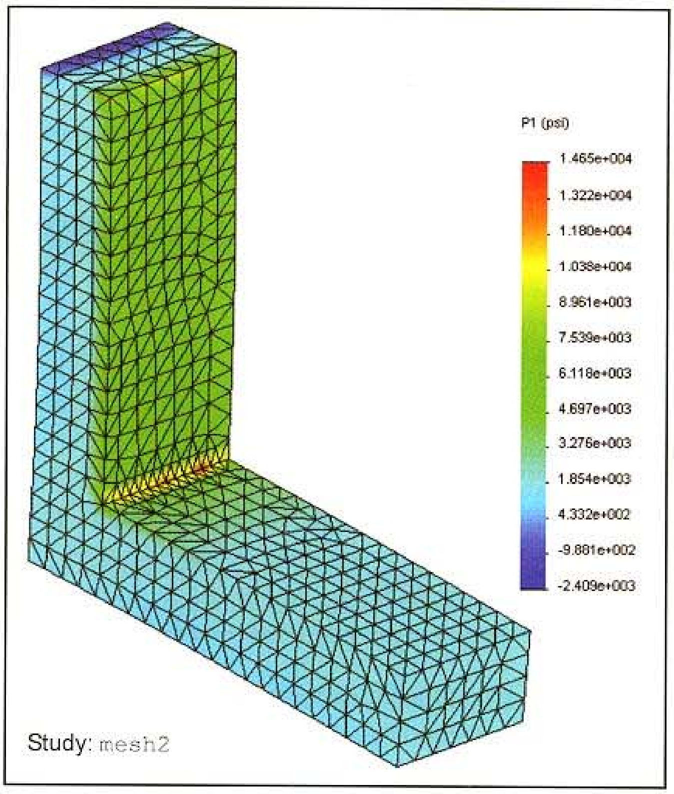
14 Plot principal stresses.

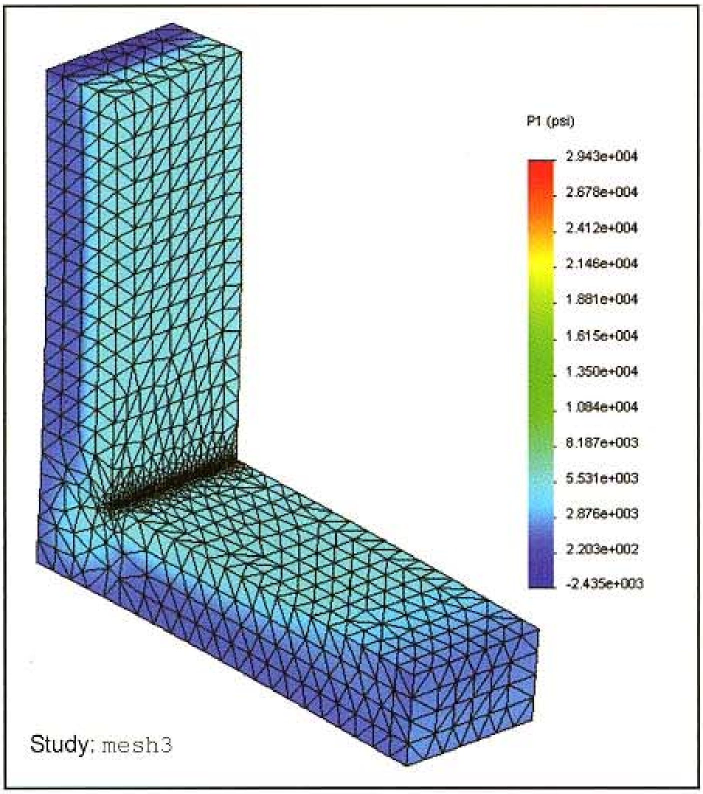
Request that the mesh be displayed with the plot by right-clicking the corresponding result plot and selecting **Settings**.

Then, under **Boundary options**, select **Mesh**.

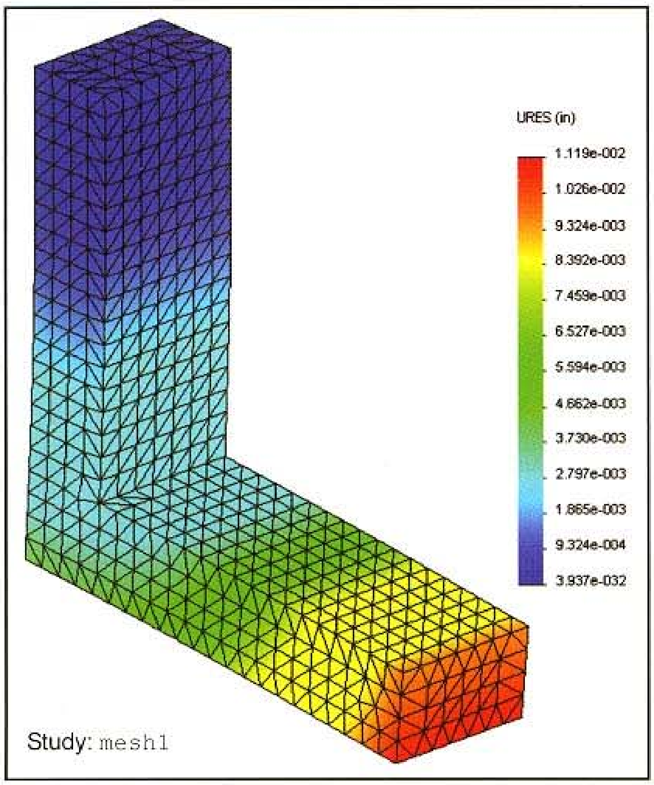
Click **OK**.

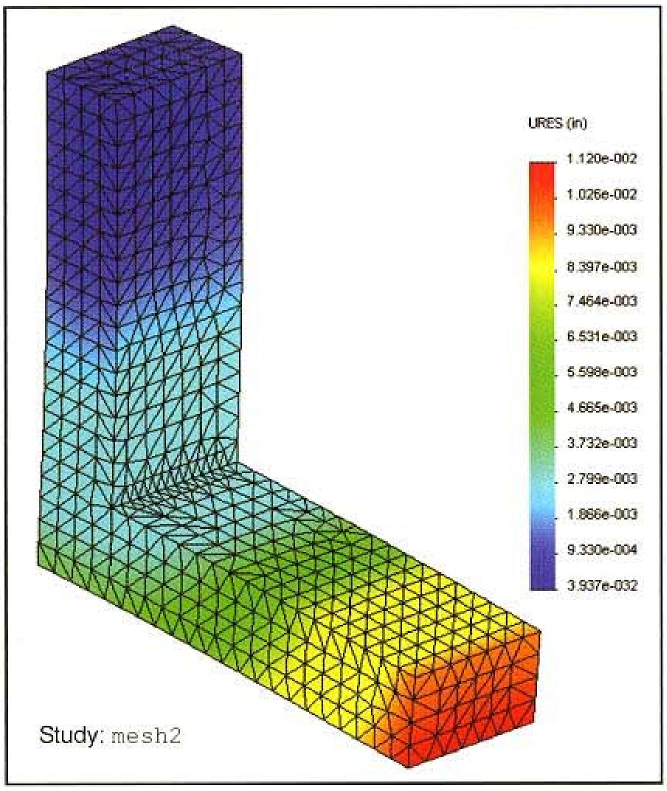


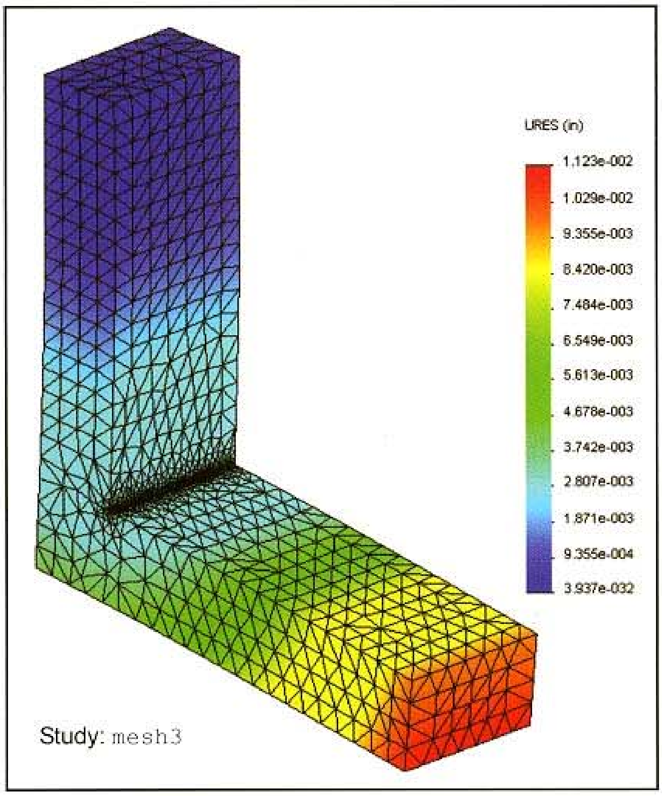




15 Plot resultant displacements.




URES (in)
1.120e-002
1.026e-002
9.330e-003
8.397e-003
7.464e-003
6.531e-003
5.598e-003
4.665e-003
3.732e-003
2.799e-003
1.866e-003
9.330e-004
3.937e-032
Study: mesh2


URES (in)
1.123e-002
1.029e-002
9.355e-003
8.420e-003
7.484e-003
6.549e-003
5.613e-003
4.678e-003
3.742e-003
2.807e-003
1.871e-003
9.355e-004
3.937e-032
Study: mesh3

Results Discussion

The study, `mesh1`, produces a maximum resultant displacement value of 0.01118 in and a maximum P1 stress result of 10,890 psi.

Reporting displacement results with five digits of accuracy is excessive. Uncertainties in loads, restraints, and material properties definition do not normally justify this level of accuracy.

We use five digits of accuracy so that we can compare the minute differences in the displacement results calculated in the three studies we undertake in this lesson.

Results Comparison

Let us summarize the maximum resultant displacement and maximum P1 stress results from `mesh1`, `mesh2` and `mesh3` studies in the following table:

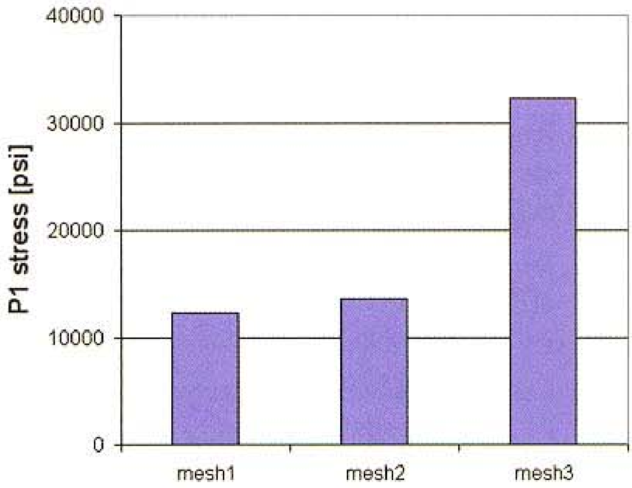
Study	Max. displ. [in.]	Increase in max. displ. [in.][%]	Max. P1 stress [psi]	Increase in P1 stress [psi][%]
`mesh1`	0.01118	-	10,890	-
`mesh2`	0.01196	0.00003 (0.26%)	14,650	3760 (34.5%)
`mesh3`	0.01122	0.00001 (0.08%)	29,430	14,780 (101%)

Note that each mesh refinement results in an increase in both the maximum displacement and the maximum stress. The increase in the displacement results is negligible and becomes less pronounced with successive runs. Thus, we can say that the displacement results converge.

If we continue this exercise of progressive mesh refinement, either locally near the sharp re-entrant as we did by means of the local mesh controls, or globally by reducing the global element size as we did in Lesson 1, we would note that the displacement results converge to a finite value and that even the first mesh is adequate if we are examining only displacement results.

Stress Singularities

Stresses, however, behave quite differently. Each subsequent mesh refinement produces higher stress results. Instead of converging to a finite value like the displacement results do, the stress results diverge.



With enough time and patience, we can produce results that show any stress magnitude. All that is necessary is to make the element size small enough!

The reason for divergent stress results is not that the finite element model is incorrect, but that the finite element model is based on the wrong mathematical model.

According to the theory of elasticity, stress in the sharp re-entrant corner is infinite; a mathematician would say that stress there is singular. The finite element model does not produce infinite stress results due to discretization errors, and these discretization errors mask the modeling error.

However, stress results are completely dependent on mesh size; therefore, they are totally meaningless.

If our objective is to determine the maximum stress, then the decision to suppress the fillet and analyze a model with a sharp re-entrant corner is a very serious mistake. The stress in a sharp re-entrant corner is singular, or infinite. The fillet, no matter how small it is, must be included in the model if we seek to find stresses in or near that fillet.

Analysis of Bracket with a Fillet

Therefore, we must repeat this lesson using a model with the fillet. Obtaining the correct model requires unsuppressing the fillet, which is done in SolidWorks by changing to the `fillet` configuration.

16 Change SolidWorks configuration.

In the SolidWorks configuration manager, activate configuration `fillet`.

Note

Note that after the configuration has been changed the studies `mesh1`, `mesh2` and `mesh3` are inaccessible. They are greyed-out. You can access them again only after activating the SolidWorks configuration corresponding to these studies. This is done in SolidWorks configuration manager. To change the configuration corresponding to a study, right-click the study icon and select **Activate SW configuration**.

17 Create new study.

Create a study `mesh4` by copying the `mesh1` study into the `mesh4` study.

Note

We copy the `mesh1` study and not the `mesh2` or `mesh3` studies for convenience because `mesh1` does not have mesh controls defined and `mesh4` does not require mesh controls.

If we use `mesh2` or `mesh3`, we have to delete the mesh controls in the `mesh4` study.

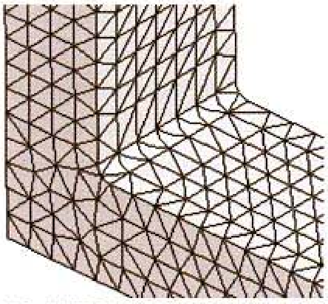
Automatic Transition

Because the fillet is a small feature compared to the overall size of the model, meshing with the default mesh settings produces an abrupt change in element size between the fillet and adjacent faces. To avoid this problem, we select the **Automatic transition** option in the meshing **Options**.

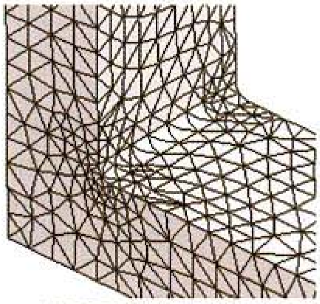
18 Mesh the model with Automatic transition option on.

Under the mesh **Options**, turn on the **Automatic transition** option.As in all previous studies, mesh the model with **High** quality elements and the default **Element size** of **0.189 in** and the default **Tolerance** of **0.0095 in**.

Compare the meshes created with and without the **Automatic transition** option.



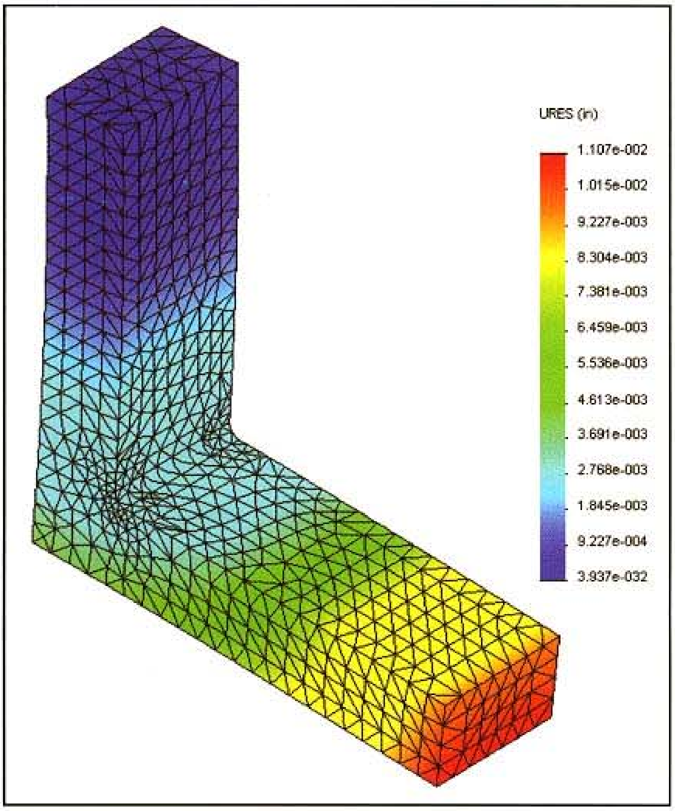
NO AUTOMATIC TRANSITION



AUTOMATIC TRANSITION

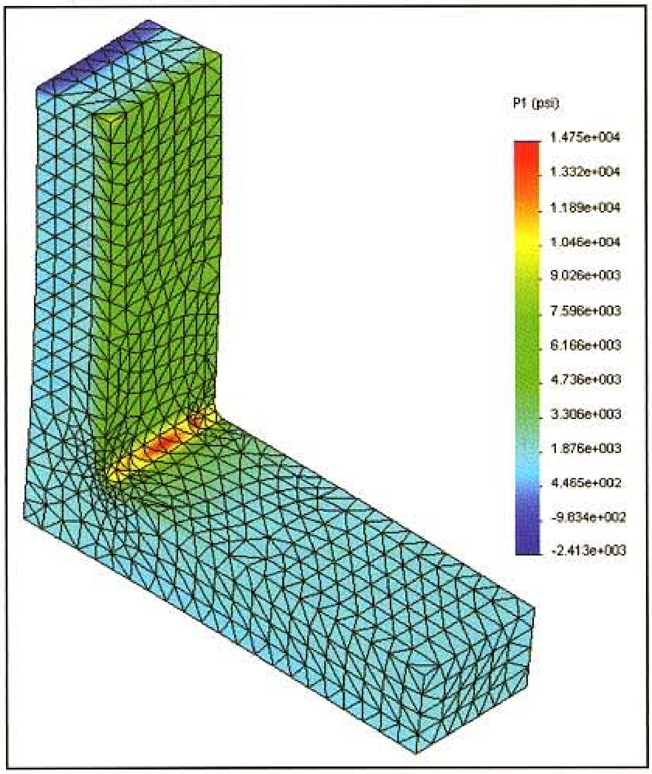
19 Run the analysis.

20 Plot Displacement results.



The maximum resultant displacement result (0.01107 in) reported for the `fillet` study differs only insignificantly from the earlier displacement results.

21 Plot principal stresses.



The stress results obtained by the model with the fillet indicate that the maximum **P1** stress is 14,750 psi.

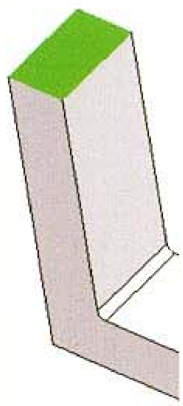
22 Extract reaction force.

Right-click on the `Results` folder and select **List Reaction Force**.

Select the face where the bracket is supported and click **Update**. Make sure the units are set to **English (IPS)**.

The **Reaction force (lb)** dialog will list the resultant of the reaction on the selected face (or faces, if more supported faces exist and are selected) as well as on the entire model.

We can see that the equilibrium is satisfied; the reaction force is equal to 200 lb, which confirms the equilibrium and the correctness of the solution.



Conclusion

These results are obtained by using the correct mathematical model. It does not make sense to debate which of the first three models produces the closest results to this one and, therefore, which one was "the best" among those three models. All models with sharp re-entrant edges are equally poor if we examine stresses on those edges.

Thus, if we are interested in stress at or near a sharp edge (or a sharp corner for shell element models), this edge must be modeled with a fillet, even if the fillet is very small.

Stress solution at a sharp re-entrant edge does not exist (is singular) and the stress results that are reported depend entirely on mesh size. Stress results in sharp re-entrant edges are, therefore, meaningless.

Summary

In this lesson, we illustrated what can go wrong when FEA is based on an incorrectly prepared model.

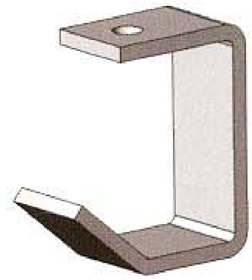
Using local mesh controls (rather than the global mesh controls implemented in Lesson 1) we obtained solutions for different meshes and revealed stress singularities at a sharp re-entrant corner.

We used this lesson to further discuss modeling and discretization error, meshing techniques, and also to illustrate the integration between SolidWorks and COSMOSWorks.

Exercise 2: Static Analysis of a C-bracket

Problem Statement

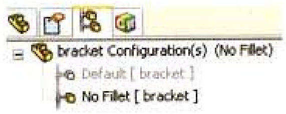
A hanging bracket mounted on the ceiling will be supporting a sign mounted on the bottom flange of the bracket. The sign will be mounted onto the bracket with a flat ribbon like cable. A 200 lb. force will be exerted on the bracket due to the weight of the sign and ribbon. We will evaluate the displacements and stresses for the bracket due to this loading. We are also interested in how modeling the bracket with and without fillets will effect our results.



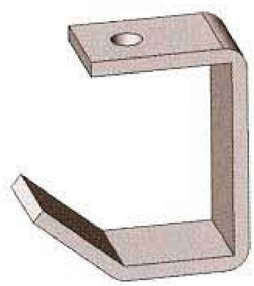
Analysis of Bracket with no Fillet

1 **Open part.**
Open the Solidworks part `bracket`. This file is located in the Exercise 2 directory.

2 **Specify active configuration.**
Activate the `No Fillet` configuration in the SolidWorks ConfigurationManager.



You will notice that the rounded inside edges become sharp re-entrant corners. By activating this configuration, all inner fillets are suppressed and will not be part of the analysis.



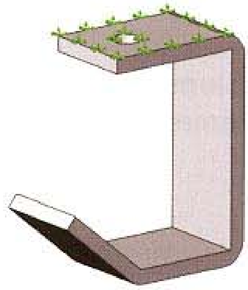
3 **Define a static study.**
Switch to the COSMOSWorks Manager and create a **Static** study named `no fillet 1` with a **Solid mesh**.

4 **Apply material properties.**
In the COSMOSWorks Manager, right-click on `Solids` and select **Apply Material to All**.

Select **Alloy Steel** from the `cosmos material` library.

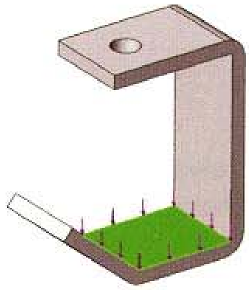
5 Apply fixed restraint.

Apply a **Fixed** restraint to the top face on the outer side of the bracket. We will assume that the compressive force of the screw is large enough to prevent any sliding or rotation about the screw.



6 Apply force.

Apply a **200 lb** normal force to the top face of the bottom flange. This force is due to the weight of the sign.



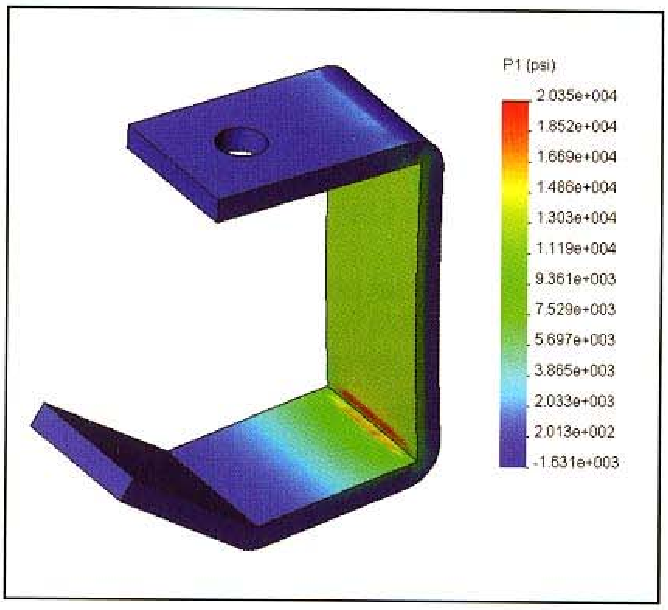
7 Mesh the model.

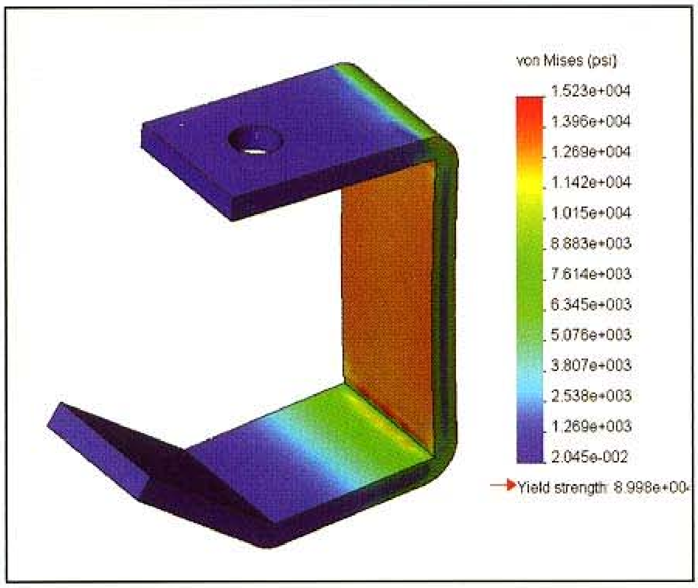
Mesh the model with the default element size. Use **High** quality elements.

8 Run the analysis.

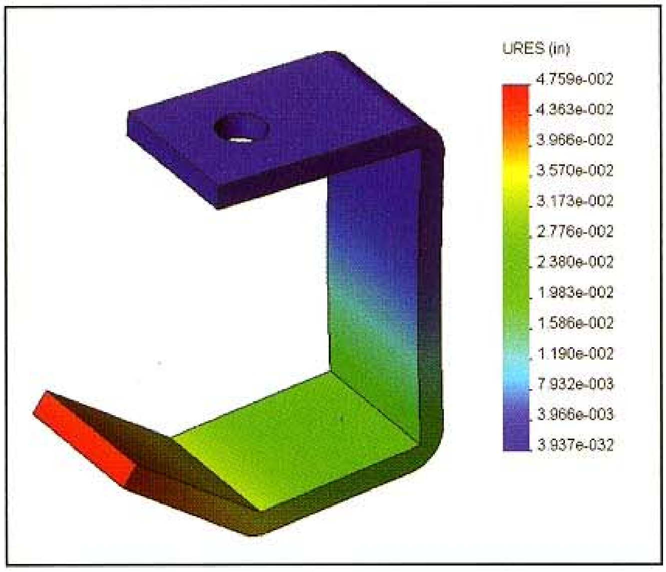
9 Plot stress results.

We find that the bracket has a maximum P1 stress of 20,350 psi and does not yield. However, there is a high stress concentration at the sharp corners.





10 Plot displacement results.



11 Create a new study.

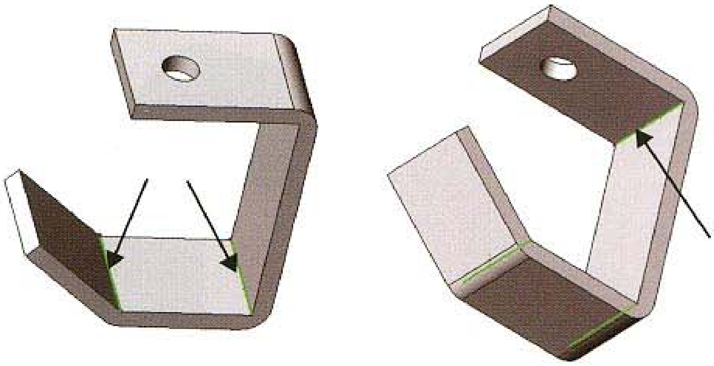
Create a **Static** study named `no fillet 2` with a **Solid mesh**.

12 Copy loads and restraints to new study.

Copy the materials, loads, and restraints from `no fillet 1` into the current study.

13 Apply mesh control.

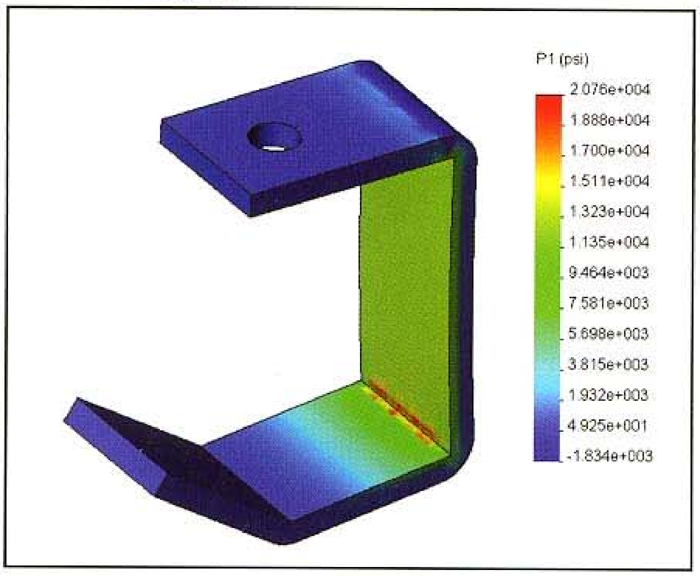
Apply mesh control to each of the three edges on the inner faces of the bracket. Use the default mesh control size.



14 Mesh the model.

Mesh the model with the default element size. We have created a finer mesh at the inside edges of the bracket, while the mesh sizes are coarser at all other locations in the bracket.

15 Run the analysis.



We find that the maximum P1 stress is 20,760 psi, which is slightly higher than the P1 value obtained in the previous study with no mesh control.

16 Create a new study.

Create a **Static** study named `no fillet 3` with a **Solid mesh**.

17 Copy loads and restraints to new study.

Copy the materials, loads, and restraints from `no fillet 1` into the current study.

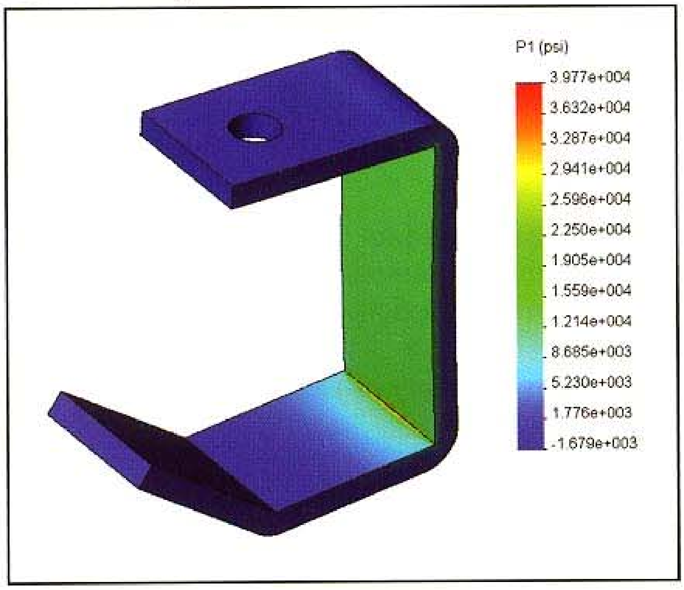
18 Apply mesh control.

In the COSMOSWorks Manager, under `Mesh`, `Apply Mesh Controls`, right-click `Control-1` and select **Edit Definition**.

Change the element size to **0.035** (in).

19 Mesh the model.
Mesh the model with the default element size. We have created a finer mesh at the inside edges of the bracket, while the mesh sizes are coarser at all other locations in the bracket.

20 Run the analysis.

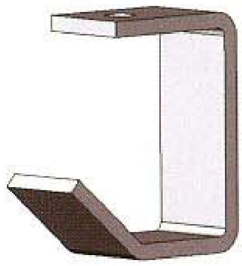


Again we find that the maximum P1 stress is significantly higher than the P1 value obtained in the previous study with a coarser mesh control. We see that although we are refining the mesh the stress results are not converging. This is due to the sharp re-entrant angles.

Analysis of Bracket with Fillet

We will now look at a model with fillets and analyze its solution.

21 Activate default configuration.
In SolidWorks ConfigurationManager activate the `Default` configuration to use the model with fillets.



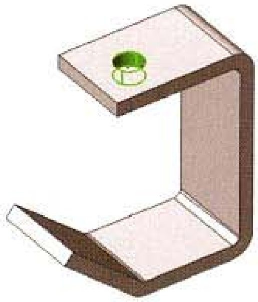
22 Create a new study.
Create a **Static** study named `fillet` with a **Solid mesh**.

23 Copy loads and restraints to new study.

Copy the materials, loads, restraints, and mesh from `no fillet 1` into the current study.

24 Apply mesh control.

Apply a mesh control of element size **0.02** (in) to the inner cylindrical surface of the hole.

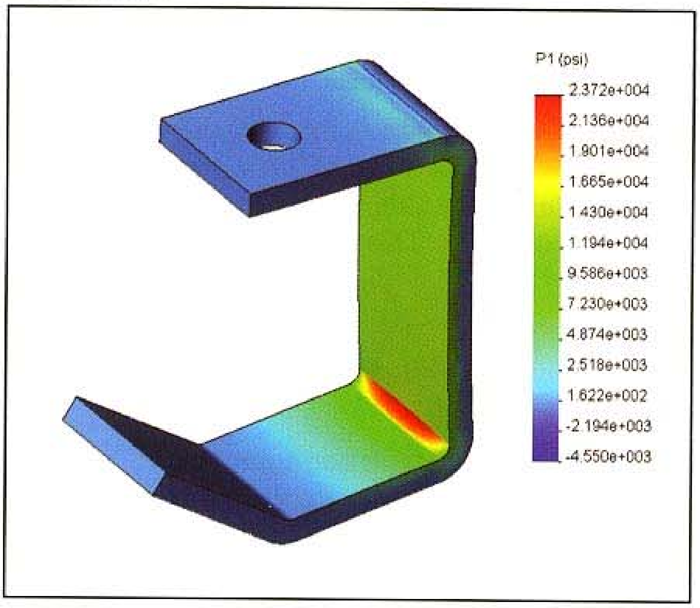


25 Mesh the model.

Compare the meshes created with and without the **Automatic transition** option.

26 Run the analysis with `Automatic Transition` mesh option.

27 Plot results.



The stress results obtained by the model with the fillet indicate that the maximum P1 stress is 23,720 psi which is significantly lower than the model without fillets.

Exercise 3: Static Analysis of a Bone Wrench

Problem Statement

A bone wrench will be analyzed for its stresses and deformations when subjected to loads resulting from regular working conditions. One side of the wrench is fixed, simulating a tight contact with a nut. The other side is subjected to a horizontal force exerted by an operator when tightening (loosening) the nut.

The analysis will conclude with a report generated in MS Word format.

Static Analysis

1 **Open part.**
Open the Solidworks part named `bonewrench.sldprt`. This file is located in the `Exercise 3` folder

2 **Specify COSMOSWorks options.**
Set the **Units** to **SI(MKS)** and the units of **Length** and **Stress** to **mm** and **N/m²**, respectively.

Request that all results be saved in the `exercise 3 results` subfolder in the Solidworks document folder. Request that the reports be saved in the same results folder.

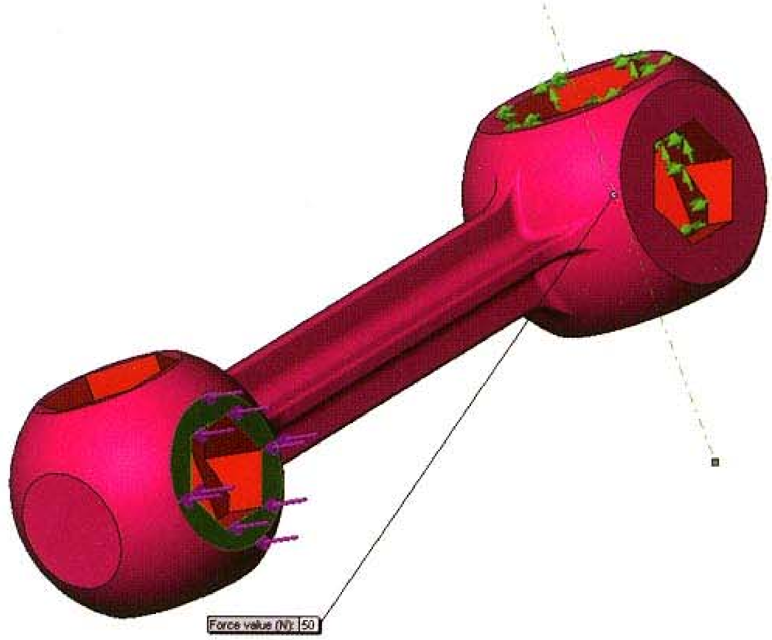
3 **Define a static study.**
Create a **Static** study named `Bone Wrench Analysis` with a **Solid mesh**.

4 **Apply material properties.**
Assign `Alloy Steel` material from the `cosmos materials` library to the part.

5 **Apply restraints.**
The tight contact between the wrench and the nut will be simulated by the application of **No Immovable** restraints on the faces (a total of eight faces), as shown in the figure below.

6 **Apply force.**

Apply a force of **150 N** exerted by an operator, as shown in the figure below.

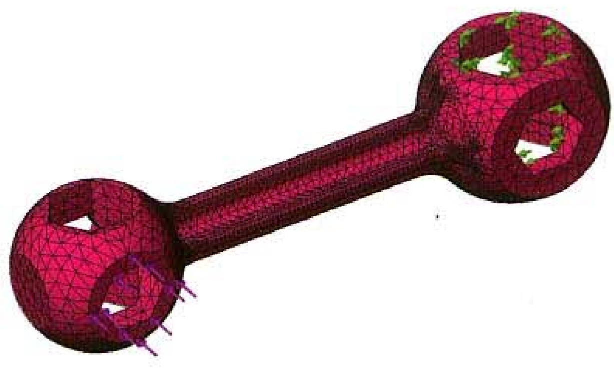


7 **Mesh the model.**

Mesh the model using **High** quality elements. Use the default **Element size** and **Tolerance** of **2.388 mm** and **0.119 mm**, respectively.

To improve the quality of the mesh in higher curvature regions, turn on the **Automatic Transition** option.

The resulting mesh can be seen in the figure to the right.

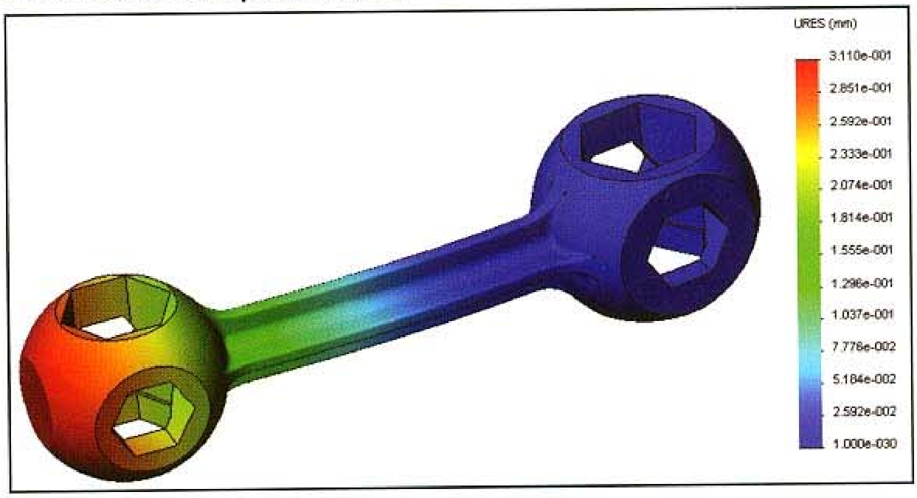


8 **Run the analysis.**

9 **Plot stress results.**

We observe that the resulting von Mises stress in the model is 24.7 MPa, which is well below the material yield strength of the 62 MPa.

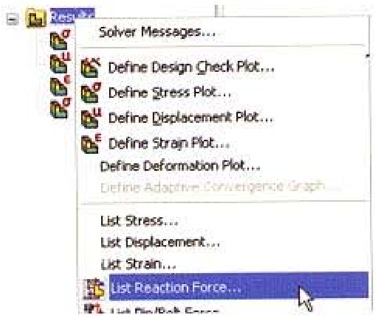
10 Plot resultant displacements.



The absolute values of the displacements are very small, with the maximum value of 0.3 mm.

11 Check the reaction moment.

Right-click on `Results` folder and select **List Reaction Force.**



Select all the faces where the model is restrained (a total of 8 faces).

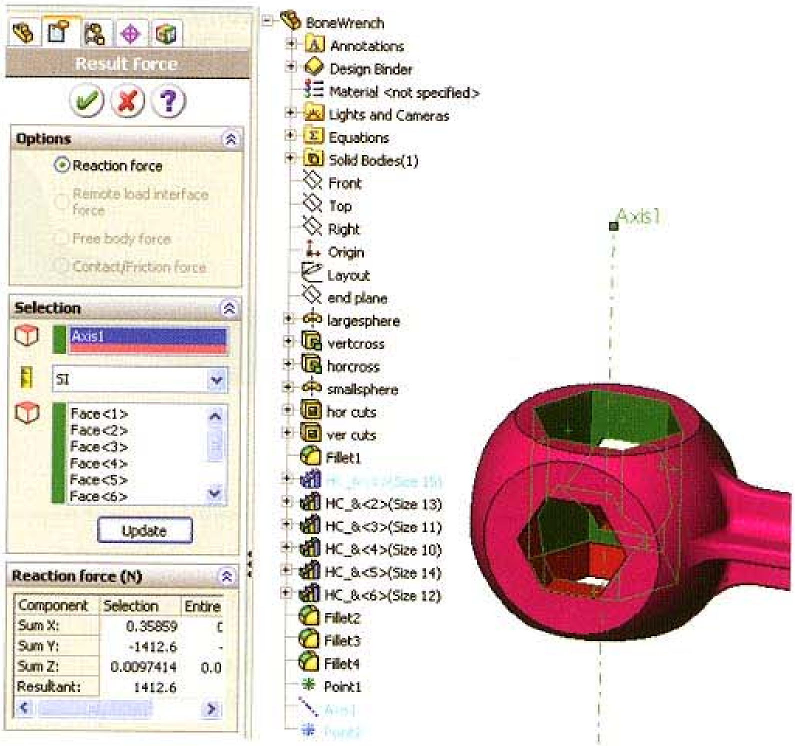
As shown in the figure below, in the **Plane, Axis or Coordinate system** field, select `Axis1`. COSMOSWorks will switch to the cylindrical coordination system defined by `Axis1`.

Note

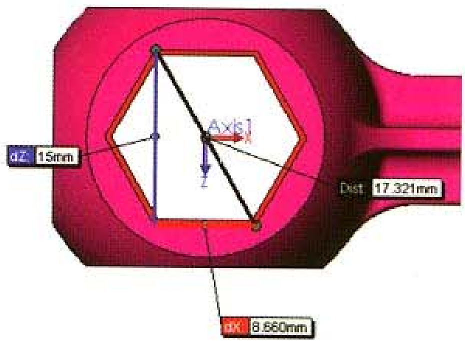
A full explanation of the cylindrical coordinate system will be given in Lesson 4.

Click **Update**.

The **Reaction force (N)** dialog reads **Sum Y: -1412.6 N**. This is the total value of the reaction force in the second cylindrical (circumferential) direction. To obtain a reaction moment, we have to multiply this value by a radius.



Because the opening is not circular, we will measure the outer and inner diameters and use the average as an approximation of the opening diameter.



The average diameter is $\frac{17.321 + 15}{2} = 16.16\text{mm}$.

Therefore, the total reaction moment is approximately equal to $\frac{16.16}{2} \times 1412.6 = 11414.16\text{Nmm}$.

To calculate the loading moment, let us measure the distance between the centroid of the applied load and `Axis1`.

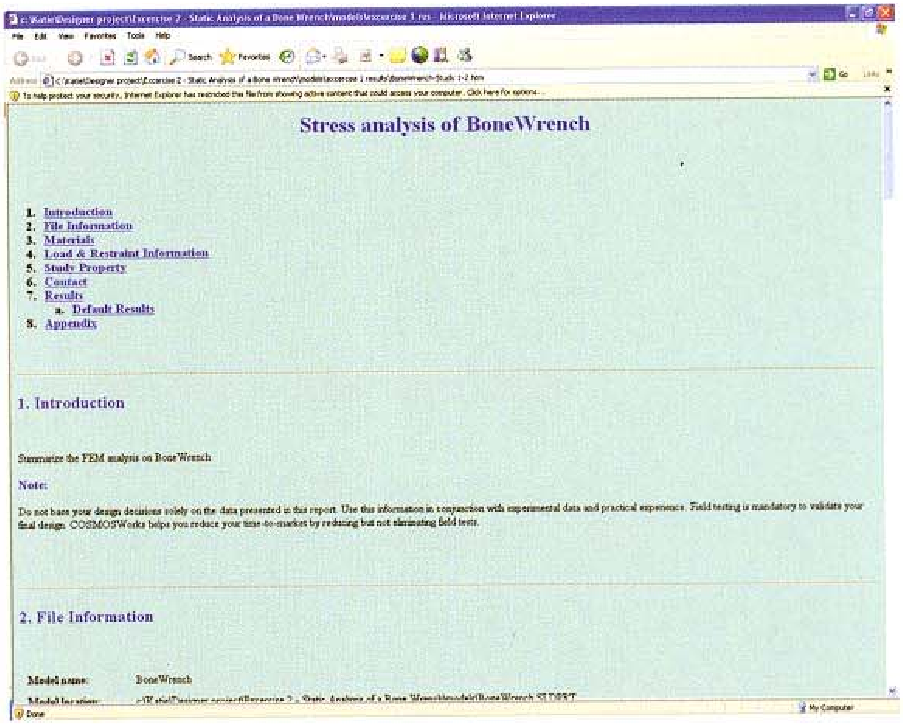
The measured distance is 75 mm. Therefore, the loading moment is equal to $75 \times 150 = 11250\text{Nmm}$, which confirms the equilibrium.

Note

The slight difference in the two values is not caused by the inaccuracy of COSMOSWorks computations. It is merely a consequence of the approximate calculation of the average diameter of 16.16 mm.

12 Generate report.

Generate a report in MS Word document format.



Analyze the report.

13 Close the part file.

Lesson 3
Contact/Gap Analysis
of Pliers

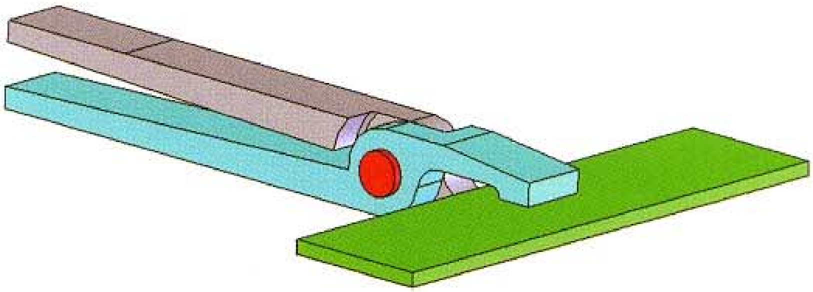
Objectives

Upon successful completion of this lesson, you will be able to:

- Perform structural analyses of simple assemblies
- Use the pin connector as a type of restraint
- Apply and define basic global and local Contact/Gaps conditions

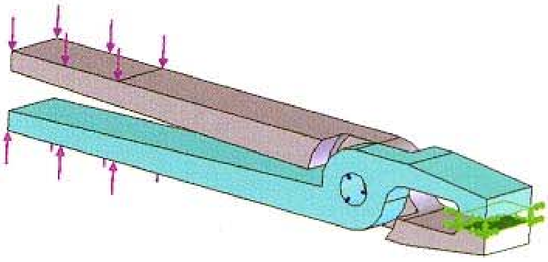
Project Description

In this lesson, we analyze a simple hand tool. It consists of four components: two identical `arms`, a `hinge pin`, and a piece of `flat` stock squeezed by pliers.



Our objective is to calculate the stresses that develop in the arms when a 50 lb. "squeezing" force is applied to the end of each arm.

We are not interested in the contact stresses that develop between the arm and hinge pin, nor are we interested in the contact stresses between the arms and the piece of flat stock.



Therefore, we can simplify the model by suppressing both the hinge pin and the flat stock and replacing them with the appropriate restraints.

Pliers with Global Contact

1 **Open assembly.**
Open the SolidWorks assembly file `pliers.sldasm`.

2 **Suppress `pin` and `flat`.**
Suppress these two components in the SolidWorks FeatureManager application. Then toggle to the COSMOSWorks FeatureManager.

3 **Set COSMOSWorks options.**
Set the global system of units to **English (IPS)** and the units of **Length** and **Stress** to **in** and **psi**, respectively.

Store the results in the subfolder named `results` under the SolidWorks document folder.

4 Create study.
Create a study named `pliers`. (**Static** analysis, **Solid** mesh.)

Note that there are two components in the `Solids` folder because there are two parts in the assembly to be analyzed.

Applying Materials to Assemblies

You can apply the same material to all components of an assembly or to each component individually.

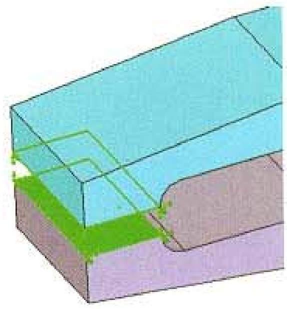
To apply material to the components, do *one* of the following:

- To apply the same material to all components, right-click `Solids` and select **Apply Material to All**.
- To apply material individually to each component (each component of an assembly may have different material properties), right-click `arm-1` (or `arm-2`) and select **Apply material to all bodies**.

5 Apply materials to components.
Apply **Alloy Steel** material properties to both arms.

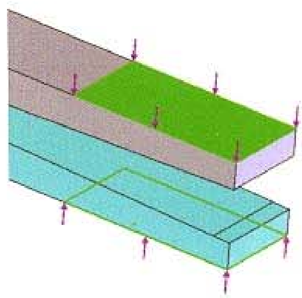
6 Apply fixed restraints.
Define **Fixed** or **Immovable** restraints on both jaws.

The applied restraints simulate the suppressed piece of squeezed flat stock and act as if pliers were bonded to it.



7 Apply force to handles.
Apply a 50 lb. force as to both handles. Note that the split faces facilitate load definition.

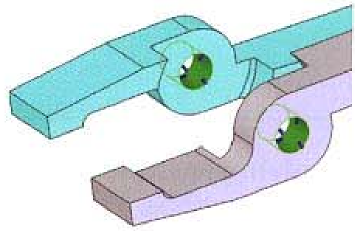
In the **Force** PropertyManager, select **Apply normal force**.



Pin connectors

To simulate a pin, we require that a cylindrical hole on one arm remains coaxial with the cylindrical hole on the other arm while the model deforms.

This condition allows the arms to rotate relative to each other and effectively acts as an "invisible pin".



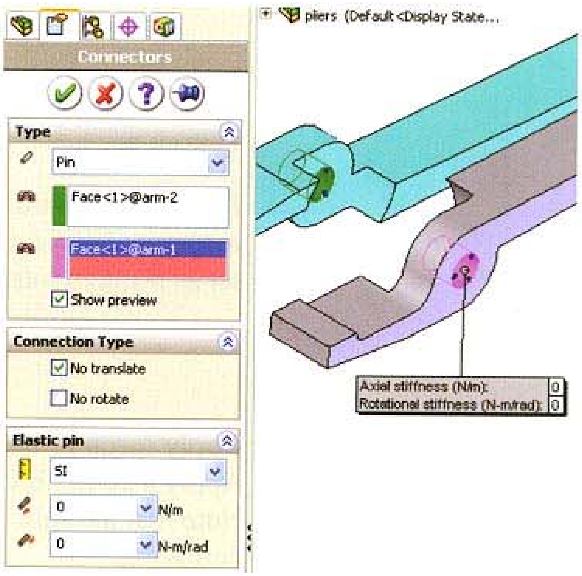
8 Define pin using pin connector.

Right-click `Load/Restraint` and select **Connectors**.

Select **Pin** from the available types of connectors.

Select the cylindrical face on one arm as **Cylindrical face(s) of component 1** and select the cylindrical face on the other arm as **Cylindrical face(s) of component 2**.

This selection is best done using an exploded view of the assembly.



In the **Connectors** PropertyManager, select **No translation** under **Connection Type**. This means that faces can rotate relative to each other but can not translate in the axial direction.

Note

Note that the pin connector also allows you to define elastic stiffness in both the axial and rotational directions. In our case, the rotational stiffness is set to **0**. (Axial stiffness is not specified because the **No translate** connection type was specified.)

Click **OK**.

Note

We have defined the loads and restraints, but we are not yet ready to mesh this assembly.

Global Contact Options

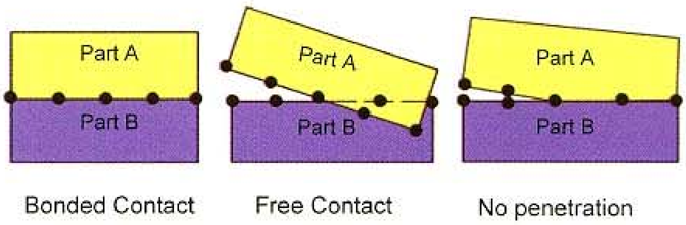
Notice a new icon named `Contact/Gaps` above the `Mesh` icon. We use this icon to define how the assembly components interact which each other.

Before proceeding with the model, let us examine the options available when defining component interaction.

Right-click `Contact/Gaps` and select **Set Global Contact.** The **Global Contact** window appears.

The **Global Contact** window specifies how to treat all touching faces in the assembly. You can override the global conditions by defining different conditions locally for selected pairs of faces or for selected components. The local contact conditions are discussed later in this lesson.

The available options for the global contact are: **Bonded**, **Free** and **No penetration.** These options are explained in the following figure and table.



Global Contact Types	
Touching faces: **Bonded** Options: **Compatible Mesh**	This is the default choice. Select this option when all touching faces are merged and the assembly behaves as one part. If touching faces are left as **Bonded**, the only difference between a part and an assembly is that in an assembly we can assign different material properties to individual components, while in a part the entire model must have the same material properties.

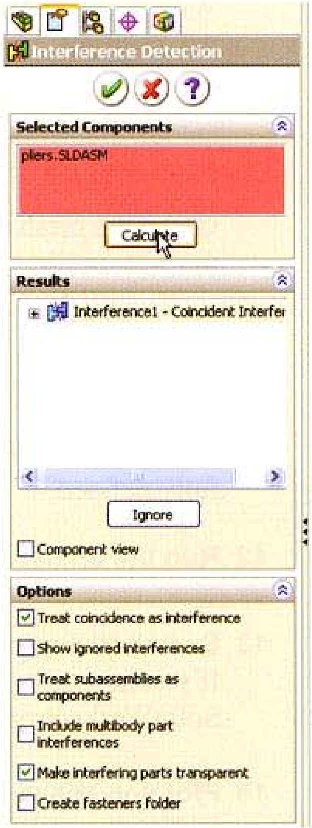
Global Contact Types	
Touching faces: **Bonded** Options: **Incompatible Mesh**	The program meshes each component independently. The touching faces are connected using bonding equations to make the assembly behave as one part. The touching faces are NOT merged. If meshing fails with the compatible mesh option, this option can help the meshing process to succeed. In general, the compatible mesh option produces more accurate results in the bonded regions. Compatible and Incompatible mesh options are discussed and practiced later on in the class.
Touching faces: **Free (no interaction)**	Select this option when the assembly is a series of unattached components with no structural connection between them.
Touching faces: **No penetration** Options: **Node to node**	Select this option when touching faces can come apart, but cannot penetrate each other. The touching faces must be of the same shape type: two flat faces, two cylindrical faces of the same radius, or two spherical faces of the same radius. The faces must share a common area, but do not have to be identical. The global value for the coefficient of friction can be specified in the study properties.

Important!

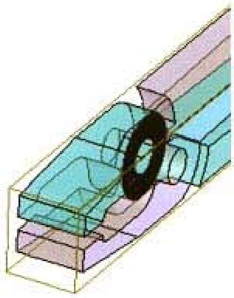
The global contact condition types will be applied only to the faces that are initially touching.

9 Check for existing interferences.

Under **Tools**, select **Interference Detection**. In the **Options** dialog, select **Treat coincidence as interference** and click **Calculate**.



We observe that one set of faces in the assembly is touching.



10 Set global contact option.

In order to allow the relative movement of the arms while the model deforms under the load, set the **Global Contact** condition to **Free (No interaction)** by right-clicking `Contact/Gaps` and selecting **Set Global Contact,** and then selecting **Free (No interaction)**.

Click **OK**.

Of course, this setting is valid only if we are sure that the faces do not penetrate each other during analysis.

Note

Setting the **Global Contact** condition to **No penetration** can also work in this lesson. The **No penetration (Options: Node to Node)** condition allows the contacting faces to slide along each other, but does not allow them to penetrate each other. However, a **No penetration (Options: Node to Node)** condition is used if a force pushes the arms against each other. Because there is no such force and we do not expect the arms to penetrate at this interface, the **No penetration (Options: Node to Node)** option complicates the model unnecessarily.

11 Mesh the model.

With the **Global Contact** condition set to **Free**, we are ready to mesh the assembly.

Mesh the assembly with **High** quality elements and the default **Element size** of **0.19 in**.

Under the mesh **Options**, activate the **Automatic transition** option.

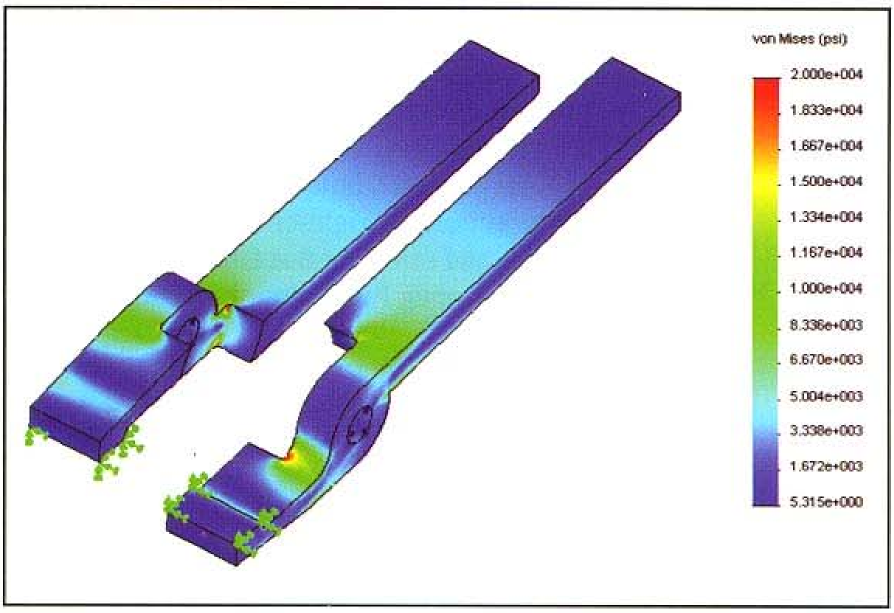
Important! Meshing must always be performed after the contact conditions are fully defined.

12 Run the analysis.

13 Switch to exploded view.

If you have not yet done so, switch to the exploded view using the SolidWorks PropertyManager application.

14 Plot von Mises stresses.



We want to see if the von Mises stresses in any portion of the model exceed 20,000 psi, which is our design stress. To determine whether the von Mises stresses exceed the maximum:

Display the von Mises stress plot by double-clicking on the `Stress1` plot icon.

While the plot is displayed, right-click `Stress1` and select **Chart options**.

Under **Display options,** select **Defined**, and then enter the minimum stress as **0** and the maximum stress as **20,000 psi**.

Click **OK.**

Right-click `stress1` and select **Settings**. Under **Fringe options**, select **Discrete**.

Click **OK**.

Viewing Assembly Results

Areas with stresses higher than 20,000 psi appear in red.

Note that an exploded view offers a very convenient way of examining the analysis results of an assembly, whereas, in normal viewing, components may obstruct the view.

Another way of reviewing results of an assembly is to hide some assembly components.

15 Hide one of the arms.

16 Define stress plot of one arm.
Toggle to COSMOSWorks, right-click the `Results` folder, and select **Define Stress Plot**.

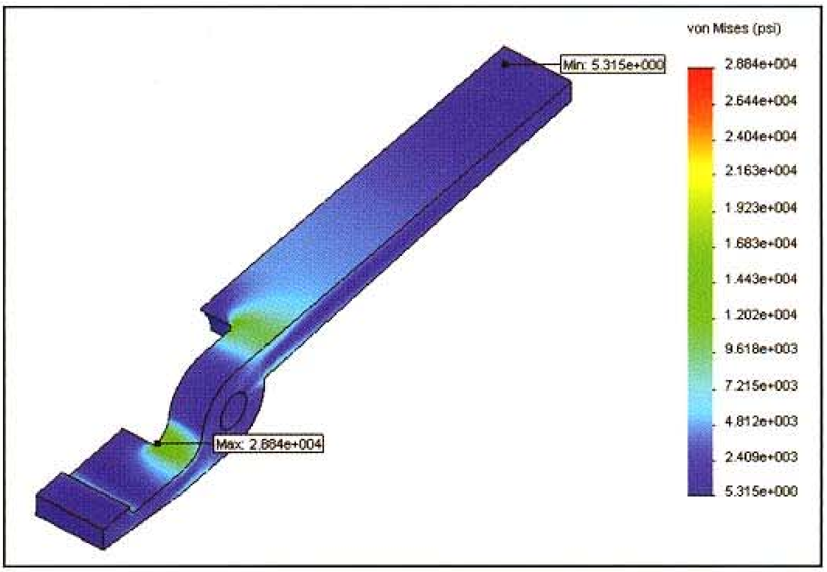
Click **OK.**

Note

You can also use the existing plot `Stress1` after hiding an assembly component arm.

In **Chart Options,** select **Show max. annotation** and **Show min. annotation**.

The maximum stress locations and their magnitudes are indicated for the displayed arm. If the locations of the maximum/minimum stress happen to be located on the hidden arm, then the annotation still refers to that hidden arm and appears to be "floating in air".



The maximum von Mises stress of approximately 29,000 psi is produced by normal operation of the pliers when a 50 lb. force is applied to the handles.

This load can be (perhaps with some difficulty) applied by hand.

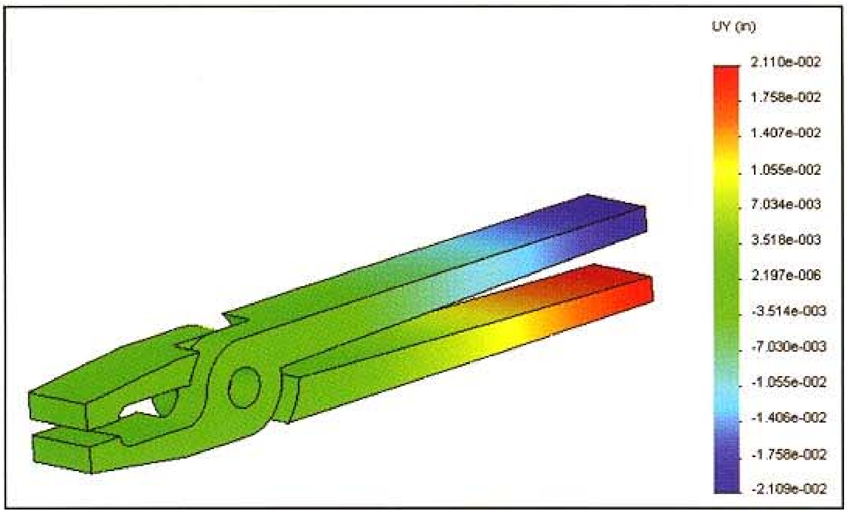
29,000 psi can easily be tolerated by the pliers' material, which has a yield stress of almost 90,000 psi.

We wish to determine the maximum stress that the pliers undergo when squeezing a 0.2 in. stock plate.

The maximum stress corresponds to the situation where the handles are blocked.

17 Show the hidden arm.

18 Create displacement plot.



To determine the force that brings the ends of the two handles together, we need to create a displacement plot showing the *y* component of the displacements.

Right-click the `Displacement1` plot icon and select **Edit definition**.

Select **UY** as the **Displacement Component**, and select **in** as the **Units.**

Under **Deformed shape**, select **True scale**. This option plots the deformation in 1:1 scale.

Click **OK**.

Required Force

We see that under the 50 lb. force, the end of each handle travels 0.021 in. Consequently, the distance between the two ends decreases by twice that amount, 0.0420 in.

Since the original distance is 0.6 in., the force magnitude must be increased by a factor of:

0.6 in. / 0.0420 in. = 14.3

Therefore, the force required to bring both arms in contact is equal to 14.3 x 50 = 715.5 lb. This is based on fundamental assumptions of linear analysis where the structural response is assumed to be proportional to the applied load.

Pliers with Local Contact

We will now load the pliers with a force that significantly exceeds 715.5 lbs to ensure that both arms come in contact. The appropriate definition of the load contact condition will ensure that the handles can come together, but cannot penetrate each other.

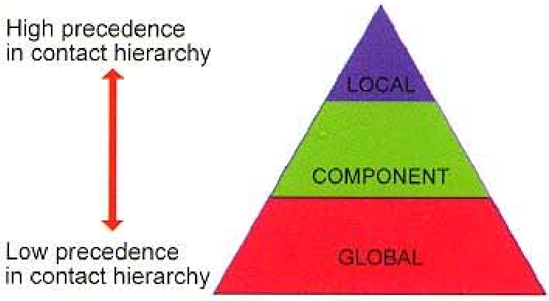
19 Create new study.
Copy the study `pliers` and name the new study `pliers contact`.

20 Edit force.
Edit the force magnitude to 2000 lb. This is an arbitrary magnitude based on our "rough" estimation of forces that will definitely bring the two arms together.

Component Contact options

The global contact condition remains the same (**free**) as in the previous study. However, now that the force is considerably larger in order to bring the two arms together, we need to specify a local contact condition that prevents their penetration.

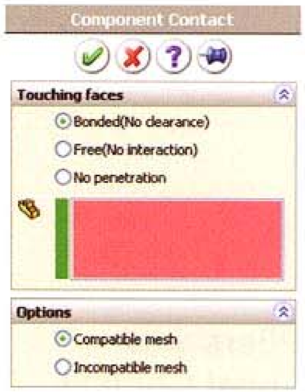
This local contact condition has precedence over the global contact. In general, the hierarchy of the contact conditions can be explained by the pyramid shown in the following figure.



Global conditions are overridden by component conditions, and both global and component conditions are overridden by local conditions.

The component contact conditions can be defined by right-clicking on the `Contact/Gaps` folder and selecting **Define Contact for Components.**

As can be seen in the figure to the right, the types of the component contacts are identical to the global contact condition types. After the components (parts of an assembly) are selected, the requested type of contact conditions are automatically generated on the components' touching faces.

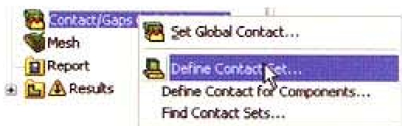


Important!

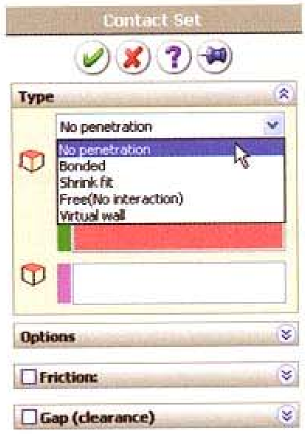
The component contact conditions will be applied only on the component faces initially touching the faces of other components in the assembly.

Local Contact Options

The local contact conditions can be defined by right-clicking on the `Contact/Gaps` folder and selecting **Define Contact Set**.



In addition to **Bonded**, **No penetration**, and **Free (No interaction)**, the local contact features two more contact types: **Virtual wall** and **Shrink fit**. Various features and options are available for each of these definitions and they will be explored both in this and in subsequent lessons.



The table below briefly describes the local contact condition types.

Local Contact Types	
No penetration	The faces (both initially touching and separated by a gap) may move away from each other but preserve the physical requirement that they may not penetrate each other. Friction coefficient and initial geometrical offset can be specified in the contact options. For the detailed description, please refer to the discussion on page 88.
Bonded	Touching faces: The selected faces will become bonded, similarly to the global and component level contact types. The compatibility of the mesh is governed by the corresponding component contact (if specified) or global contact compatibility setting. Faces separated by a gap: The generated mesh will always be incompatible irrespective of the component and global contact compatibility settings.
Shrink fit	The program creates a shrink fit condition between the selected faces. The faces may or may not be cylindrical. This condition requires that the two parts exhibit a finite volume interference.
Free	The selected pair of faces is free to move in any direction. Free faces can penetrate into each other, a physical impossibility. You should use this option only when you are absolutely sure that the specified loading will not cause the faces to penetrate.

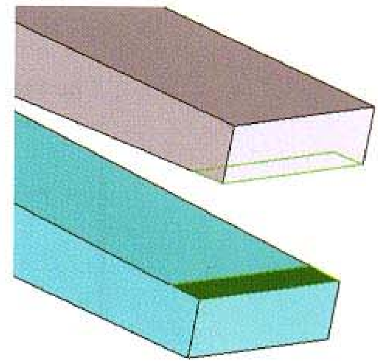
Virtual wall	This provides a sliding support in a way similar to Roller/Sliding restraint, except that a friction coefficient and wall elasticity can be specified.

Note

In this lesson, we will not use any component level contact. Rather, a local contact definition ensuring no penetration between the two arms will be specified.

21 Define contact set.

To define a contact zone between the ends of the two handles, we use the two small split faces on the inside of the handles to define a contact pair.



Right-click `Contact/Gaps` and select **Define Contact Set**.

In the **Contact Set** PropertyManager, select **No Penetration** as the desired type of contact. **No Penetration** is the most general type of contact.

The faces do not have to be the of the same type. For example, a flat face can contact a cylindrical face. Also, the faces do not have to touch each other to begin with; an initial gap may exist for surface contact. For the detailed description of the No penetration contact option, please refer to the following discussion.

Click one face to define it as the **Source,** and then click the other face to define it as the **Target**.

Select **Node to surface** under **Options**.

Click **OK**.

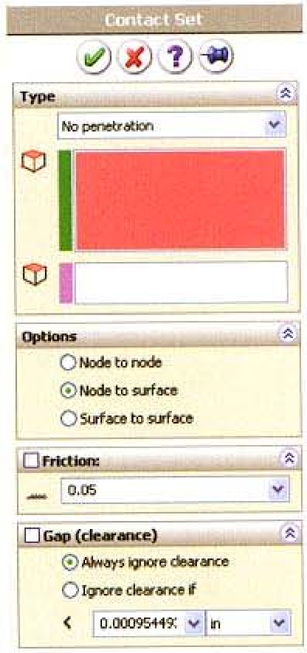
Note

The `Contact/Gaps` icon is now a folder that contains the local **Contact/Gaps** condition that we have just defined.

To edit the **Contact/Gaps** condition, right-click `Contact Set-1` under the **Contact/Gaps** folder and select **Edit Definition.**

No penetration local contact condition

No penetration is the first local contact definition that we use.



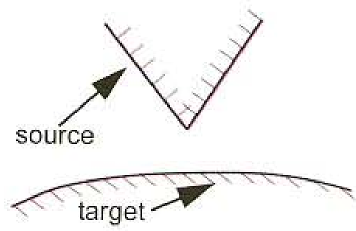
The **No penetration** condition is specified between the source and the target entities:

- Source: The source, represented by the nodes or surfaces of the finite element mesh, may be defined by vertices, edges, or faces. It is recommended that the mesh of the source be finer than that of a target.
- Target: The target face (no edges or vertices are allowed) should be larger and smoother than the geometry of the source. Based on the type of the **No penetration** contact, the target is represented by either nodes or surfaces of the finite element mesh.

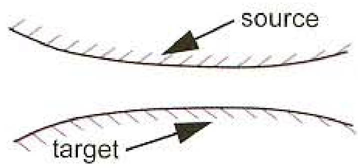
The **Options** dialog offers the following types:

- *Node to node*: Source and target entities must be initially in contact and no significant sliding may occur. This option may not be used when the Large displacements option is active.
- *Node to surface*: As the name suggests, the target is represented by surface entities, while the source is represented by the nodes of the finite element mesh. No restriction on the initial configuration is imposed, i.e. the source and the target do not need to be touching at the beginning of the analysis and sliding is permitted. Because the directions of the friction and normal forces are updated during the analysis, this option is valid for the Large displacements calculations.

This type of contact is able to describe complex contact configurations and behavior, but it requires substantially more computational effort.Typically, we would use this type of contact when edge-to-face configuration is expected.



- *Surface to surface*: This type of **No penetration** condition is the most general one. Both the target and the source are represented by the subsurfaces of the finite element mesh. As in the case of the *Node to surface* type, no restriction is imposed on the initial configuration and sliding is allowed. The directions of the contact and friction forces are updated during the analysis and this option is valid for the Large displacement computations. Typically, we would use this type of contact when a face-to-face configuration is expected.



- *Friction*: Any value of the friction coefficient is permitted.
- *GAP (clearance)*: In many applications, two entities cannot come into full contact due to the manufacturing limitations and the modeling approaches that we use. This feature restricts such two entities from coming closer than the initial geometrical offset. For a more detailed explanation, please refer to Lesson 6.

---

22 Mesh the model.

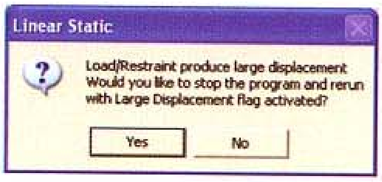
Mesh the model with **High** quality elements and the default element size. Keep the **Automatic transition** option **On**.

Note

Defining a local contact condition or, more generally, making any change to the contact conditions requires remeshing.

23 Run the analysis.

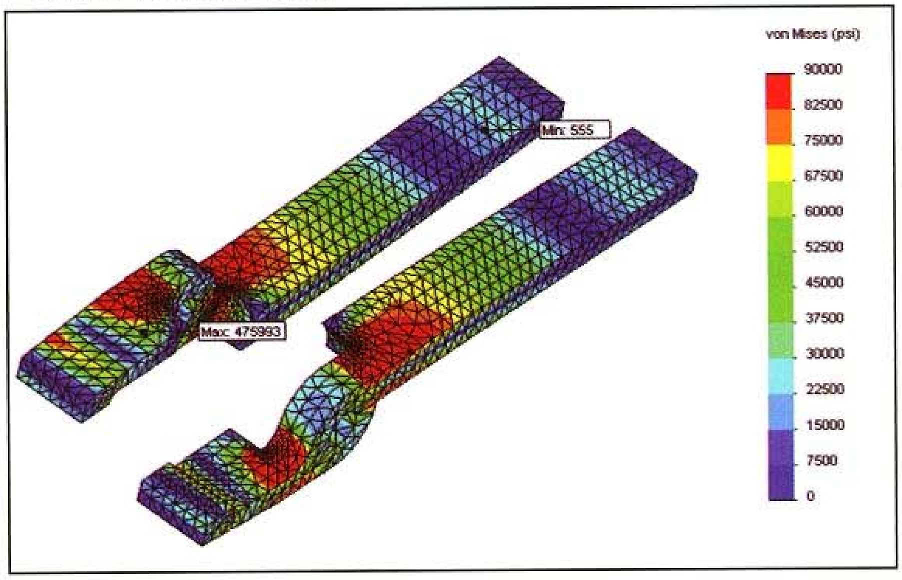
The large displacement dialog window will pop up. Click **No** to complete the analysis.



Note

The large displacement dialog box warns us that the large displacements of some parts in the assembly were detected. The large displacement computations are the subject of Lesson 12. At this point, we will ignore this fact.

24 Plot von Mises stresses.



After the analysis is complete, create a von Mises stress plot, with discrete fringes, the mesh showing, and the stresses scaled from 0 to 90,000 psi.

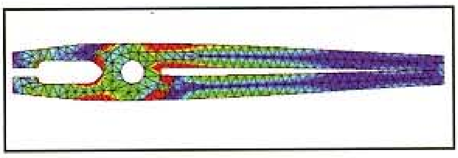
The region in red indicates the yielding material. We can observe that the maximum reported von Mises stress is approximately 470 ksi. This value is, of course, unrealistic. Yielding of the material indicates that a linear analysis is no longer valid and that a nonlinear analysis would be required.

Contact Stresses

After the handles are blocked, any further increase in force magnitude has little effect except for increasing the contact stresses where the handles touch.

Question: Can we analyze those contact stresses?

Answer: No, the element size in the contact area is much too large in comparison to the size of the contact area. This comparison is best seen in a side view.



The two handles touch only along the edge; therefore, the contact stresses are not modeled correctly. For accurate modeling of contact stresses, we need several elements along the length and width of the contact zone.

Summary

This was our first assembly analysis lesson.

The model was simplified by the use of a pin connector restraint, which eliminated the need to mesh the pin.

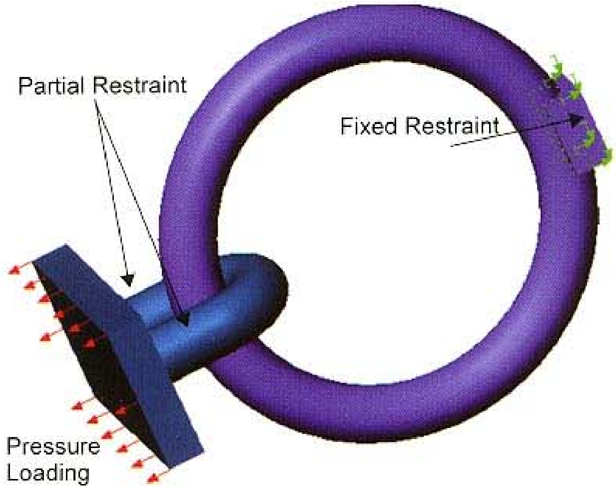
Global, component, and local contact conditions were introduced and discussed. In this lesson, we applied a **Free** global contact condition that prevented any interaction between the arms and a local **No penetration (Node to surface)** condition at the ends of the arms to prevent a penetration.

Finally, the limitations of linear material analysis were examined and contact stresses were introduced.

Exercise 4: Contact Analysis of a Two ring assembly

Project Description

A simple two ring assembly, in which the outer faces of the rings exert contact pressure on each other if tensile loading is applied, is considered for analysis. In this exercise, it is shown how models involving surface contact conditions can be easily set up and analyzed



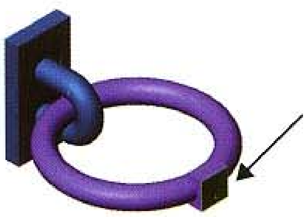
1 **Open Assembly.**
Open the Solidworks assembly `TwoRingsAssem.sldasm`.

2 **Define a static study.**
Create a **Static** study named `Pressure Loading` with a **Solid mesh**.

3 **Apply material properties.**
In the COSMOSWorks Manager, right-click on `Solids` and select **Apply Material to All**.

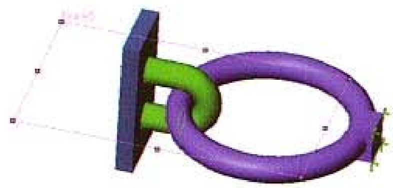
Select **Alloy Steel** from the `cosmos material` library.

4 **Apply fixed restraint.**
Apply a **Fixed** restraint to the back face of `TwoRingsPart1`.



5 Constrain `TwoRingsPart2` to move in the direction of the load.

Right-click `Load/Restraint` and select **Restraints**.



Under the restraint **Type**, select **Use reference geometry**.

Select `Plane2` from the SolidWorks flyout manager to specify the direction of the restraint.

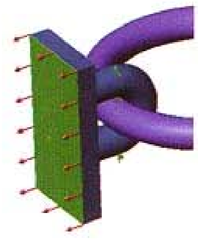
Select the three cylindrical surfaces to apply boundary condition.

Activate the Displacement components **Along plane Dir 2** and **Normal to plane**, and set the values to **0 in**.

Click **OK**.

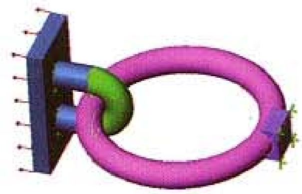
6 Apply pressure.

Apply a **500 psi** pressure normal to the surface of the `TwoRingsPart2`.



7 Define contact set.

Right-click `Contact/Gaps` and select **Define Contact Set**.



In the **Contact Set** PropertyManager, select **No Penetration** as the desired type of contact.

Click one face to define it as a **Source**, and then click the other face to define it as a **Target**.

Select **Node to surface** under **Options**.

Note It does not matter which face is selected as the source or target.

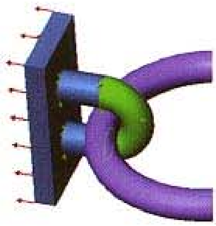
Click **OK**.

8 Apply mesh control.

Apply mesh control to surface on `TwoRingsPart2`.

In the element size box, enter a value of **2 mm**.

Take all other default mesh control settings.

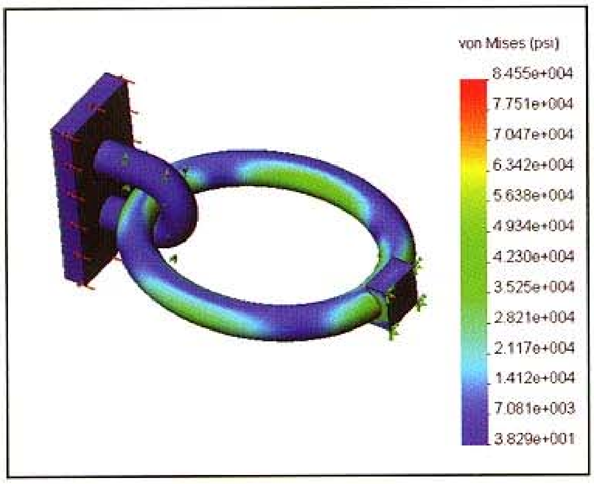


9 Mesh the model.

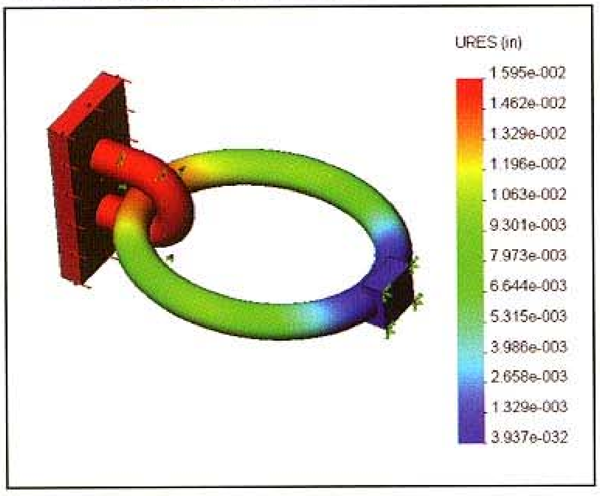
Mesh the model with the default element size. Use **High** quality elements.

10 Run the analysis.

11 Plot stress results.



12 Plot Displacement Results.



13 Animate Displacement Results.

Lesson 4
Shrink Fit Analysis of a Wheel Assembly

Objectives

Upon successful completion of this lesson, you will be able to:

- Analyze a shrink fit assembly
- Review stress results in local cylindrical coordinate system
- Present analysis results using eDrawings
- Locate problems with the help of the What's Wrong feature
- Use Soft springs and Inertial relief options to eliminate rigid body modes

Project Description

In this lesson we analyze a wheel assembly.

A rim with an inside radius of 2.382 in. is pressed on a hub with an outside radius of 2.391 in.

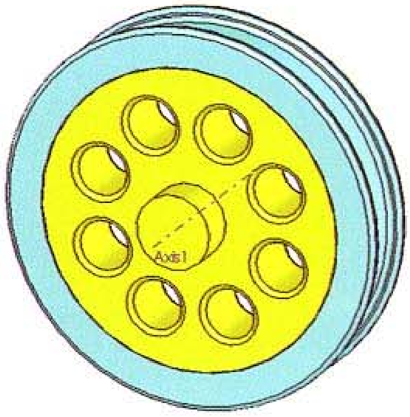
Our objective is to find the following stress results in both components:

- von Mises stress
- Hoop stress
- Contact stress

1 Open assembly.
Open the SolidWorks assembly `wheel assembly`.

Symmetry

Note that an axis, `Axis1`, has been defined in the assembly. We use it as a reference to produce plots of hoop stresses and contact stresses.

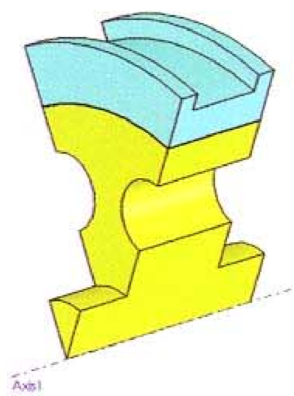


We can take advantage of the multiple symmetry of this assembly model and analyze 1/2, 1/4, or even 1/8 of the model.

Here, we analyze a 1/8 section of the model.

2 Unsuppress cut in model.
Unsuppress the cut in the assembly, this is the last feature in SolidWorks FeatureManager, named `cut 1/8`.

3 Suppress rounds.
To defeature the model, suppress the rounds in both parts (look for the feature `rounds` in rim and `round 1`, `round 2` in hub part).



Defeaturing

With this modification to the CAD assembly model, we have departed from the original CAD geometry and we are now analyzing geometry specifically created for the purpose of analysis.

Suppression of the rounds has left some sharp re-entrant edges.

These are permissible only because we do not intend to examine stresses along these edges or in their vicinity.

Shrink Fit Analysis

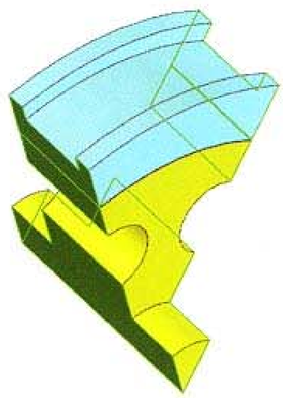
4 **Create static study.**
Toggle to the COSMOSWorks environment, and create a static study study named `shrink fit` with a solid mesh.

5 **Set COSMOSWorks options.**
Set the global system of units to **English (IPS)** and set the units of **Length** and **Stress** to **psi** and **in**, respectively.

6 **Review material properties.**
Observe that the `Solids` folder holds two icons, corresponding to the `hub` and `rim` components of the assembly, and that the material properties have been automatically transferred from SolidWorks.

Examine each part individually to confirm that the hub material is **Plain Carbon Steel** with a yield stress of 32,000 psi (220 MPa) and the rim material is **Alloy Steel** with a yield stress 90,000 psi (620 MPa).

7 **Define symmetry restraints.**
We use a 1/8 section of the wheel assembly, but want valid results for the complete model. Therefore, we need to simulate the remaining 7/8 of the assembly model. Applying symmetry boundary conditions to the radial faces created by the cut make the 1/8 section behave as if the wheel was still complete.



Apply symmetry boundary conditions to all the faces that were created by the radial cut.

Symmetry boundary conditions on both sides of the radial cut can be created in one step.

Select faces from both components on both planes of symmetry: four faces on the `hub` and two faces on the `rim`.

Right-click `Load/Restraint` and select **Restraints**.

Under **Type**, select **Symmetry**.

Click **OK**.

Rigid body mode

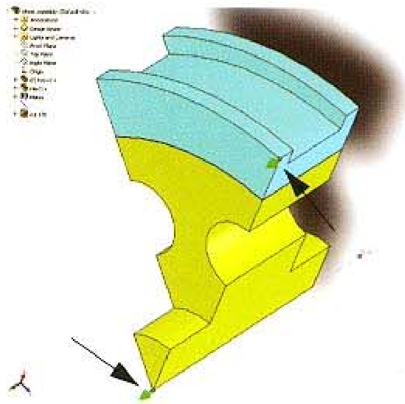
With the symmetry restraints applied, the model can still move in the axial direction. Thus, one rigid body mode remains unconstrained.

To eliminate this rigid body motion, it is enough to restrain just one vertex on each of the components (the total of two vertices) in the axial direction. Note that each part must be constrained individually because parts can slide in the axial direction, the shrink fit contact is frictionless.

This is actually an artificial restraint simply for the purpose of removing rigid body motion, which is not allowed in structural FEA and causes the solver to crash.

Alternatively, we can use the soft spring feature which is specified in the study properties. We will demonstrate this option in the second part of this lesson.

8 Eliminate rigid body mode in the model.
Restrain the model by one vertex on each of the two assembly components.



Select one vertex on the rim and one on the hub (any vertex), right-click `Load/Restraint` and select **Restraints**.

Under **Type**, select **Use reference geometry**.

Using the fly-out menu, select `Axis1` as reference plane.

Under **Translations**, specify that the displacement in the direction along the axis **(Axial)** is equal to **0**.

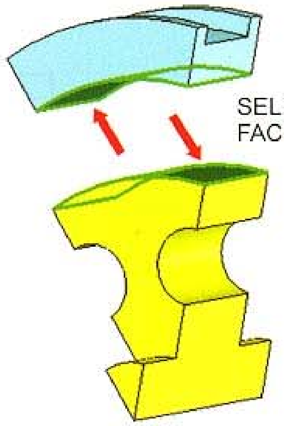
Now the assembly is fully restrained; it has no unconstrained rigid body modes. Any other movement of the assembly must be associated with the deformation.

Shrink Fit Contact Condition

Because the rim diameter is smaller than the wheel diameter, there is an interference in the SolidWorks assembly. COSMOSWorks eliminates this interference by "stretching" the rim and "squeezing" the wheel if we define the contact conditions between the interfering faces as **Shrink Fit**. **Shrink Fit** is one of the several types of local **Contact/Gap** conditions available in COSMOSWorks.

9 Define shrink fit condition.

To define a **Shrink Fit**, right-click `Contact/Gaps` and select **Define Contact Set**.



The **Contact Set** window appears.

Select **Shrink Fit**, **Node to surface**, from the available types of contact conditions.

Define one face (such as the face of hub) as the source and the other face (such as the face on rim) as the target.

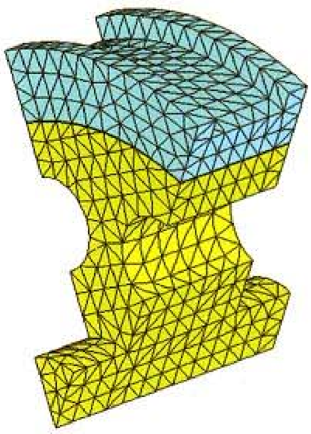
Click **OK**.

Note

Under the **Friction** dialog, we could specify the coefficient of friction. In this analysis we will assume no friction.

10 Mesh the model.

Create a **High** quality mesh with the default **Element size** of **0.15 in**.



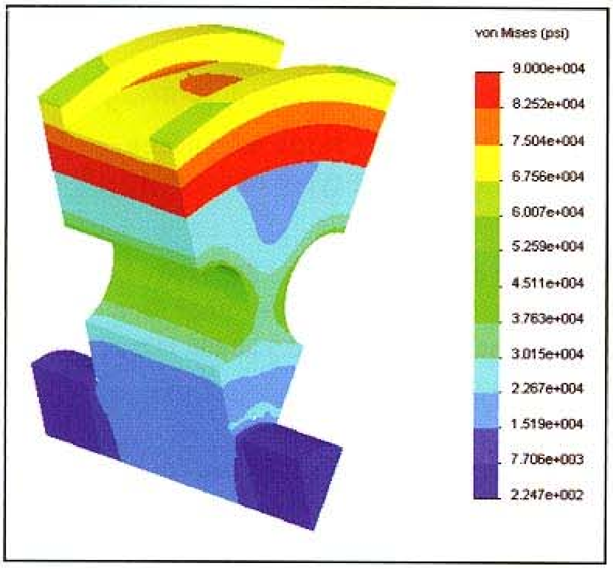
Note that, along the axial direction of the two faces in contact, eight elements have been created, which is adequate for this particular analysis.

If we expected high gradients in the contact stress distribution, then more elements along the contacting faces would be required to model contact stresses.

11 Run the analysis.

Note that the solution takes longer than if the model were treated as **Bonded**.

12 Plot von Mises Stresses.
Create and show the von Mises stress plot.



Under **Settings**, change the **Fringe** type to **Discrete**.

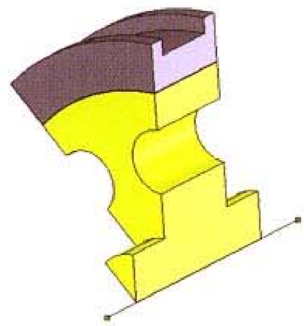
Make sure that the deformed shape is in the **True scale.**

Right-click the `Stress-1` icon in the `Results` folder and select **Chart Options**. Under **Display Options** select **Defined** and set the maximum stress legend to **90,000 psi**, which is the yield stress of the rim material. Also, familiarize yourself with other ways of modifying the plot which are available in **Chart Options.**

The von Mises stress results indicate that a portion of the rim experiences stresses above the material yield stress.

Plot Results in Local Coordinate System

13 Select axis for stress plot reference.
Now let us prepare a stress plot showing hoop (circumferential) stresses. For this we must present stress results in a cylindrical coordinate system with the z axis aligned with the axis of the wheel assembly.



Right-click on the `Results` folder and select **Define Stress Plot**.

From the SolidWorks Flyout Feature Manager select `Axis1`. This is to be used as a reference in the stress plot we are now creating.

Verify that `Axis1` appears as the selected reference in the **Plane, Axis or Coordinate System** box of the **Stress Plot** window.

Defining Cylindrical Coordinate Systems

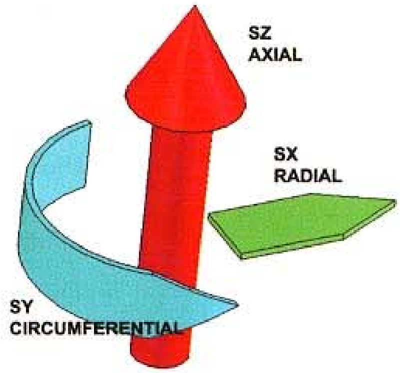
Any axis defines a cylindrical coordinate system whose first direction is radial, second is circumferential, and third is axial.

Therefore, `Axis1` defines radial, circumferential, and axial directions associated with the axis position.

Using an axis as a reference redefines the meaning of the stress components SX, SY, and SZ, which are normally associated with the directions of the global coordinate system.

If an axis is used as a reference, definitions of SX, SY, and SZ undergo the following changes:

- SX becomes the stress component in the radial direction
- SY becomes the stress component in the circumferential direction
- SZ becomes the stress component in the axial direction



14 Plot hoop stresses.
Because we are defining a stress plot showing hoop stresses, select the **SY** stress component.

The SY stress component points in the circumferential direction, which is the hoop stress.

Under **Settings**, select **Discrete** as the **Fringe Option**.

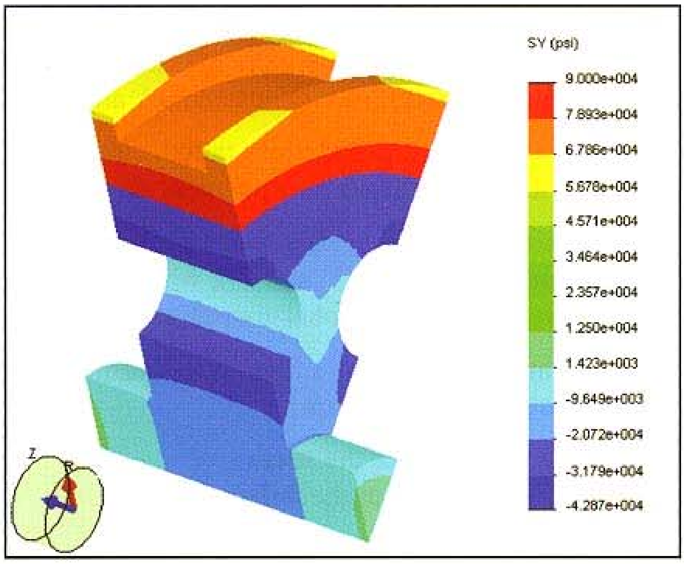
Set the **Units** to **psi**.

Under **Chart Options**, select **Defined** and set the maximum value of the stress legend to **90,000 psi**.

Set the **Deformed shape** scale to the **True scale**.

Note

When a local cylindrical system is specified in the definition of the result plots, the familiar triad icon is replaced with a new symbol denoting a cylindrical system.



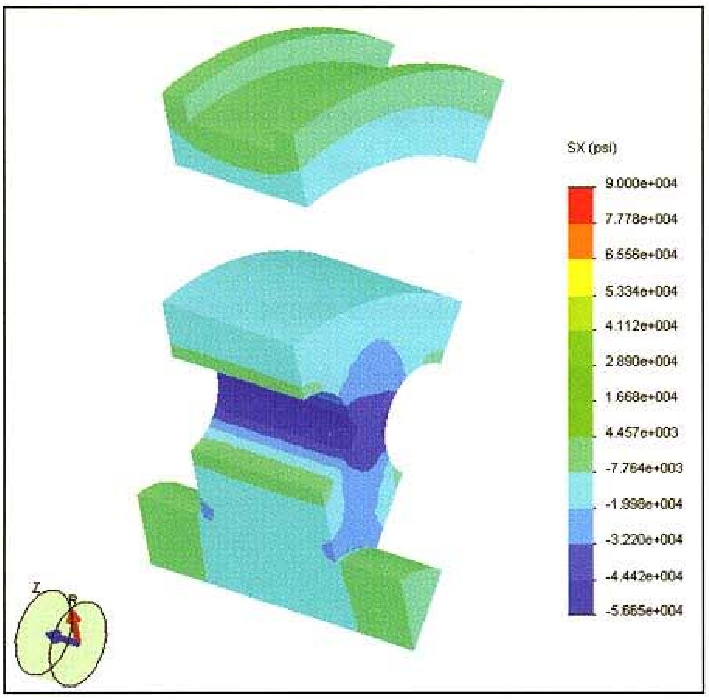
15 Plot contact stresses.

Use exploded view. To display a plot showing the contact stress on contacting surfaces:

Display an exploded view and select `Axis1` as a reference. Set the **Deformed shape** scale to **True scale**.

Plot the SX component of stress with respect to `Axis1`.

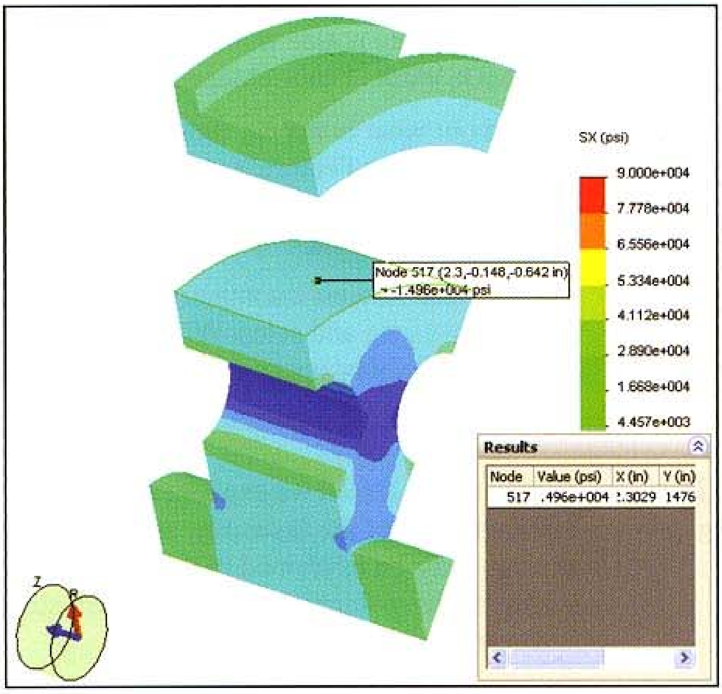
The SX stress component, which corresponds to the direction normal to the two faces in contact, is the radial direction and, hence, SX is the contact stress.



16 Probe stress results.

To probe the stress plot for detailed stress results, right-click the `plot` icon and select **Probe**. An undeformed plot is required if we want to probe for detailed stress results.

Stresses on both surfaces are, of course, equal. The negative sign denotes stress towards the surface.



Saving all plots

Each plot created in a study can be saved individually in any of several formats available. To view the list of available formats, right-click any plot icon, select **Save as** to open the **Save as** window, and examine the options in the **Save as type** menu.

Most likely you will find the **eDrawings** format the most useful when communicating the COSMOSWorks analysis results. Analysis results stored in an **eDrawings** format can then be viewed with an **eDrawings** viewer, which is available for free at www.solidworks.com.

Rather than saving result plots individually, it is also possible to save them in one step either in **JPEG** or **eDrawings** format. Right-click the study or the `Results` folder to invoke the pop-up menu and select either **Save all plots as JPEG files** or **Save all plots as eDrawings.**

Using **JPEG** format, individual files will be created for each plot that has been defined in the study. Using **eDrawings** format, all plots will be stored in one file. In both cases, files will be located in the COSMOSWorks report folder. (The report folder can be selected in **COSMOSWorks, Options, Default Options (New study)**, under **Results**.)

What's wrong feature

Occasionally, when defining COSMOSWorks model or analyzing results you may notice warning symbols showing in COSMOSWorks FeatureManager.

To find out what is wrong, right-click the item under a study (in COSMOSWorks FeatureManager) accompanied by the warning sign and select **What's wrong** to open the **What's Wrong** window. This window will list a description of the problem for that item only.

You can also request a summary of all the problems in a study. Right-click the `Study` and select **What's wrong** to list all problems in the study. This inquiry opens a window listing all problems in the study.

Analysis with Soft springs

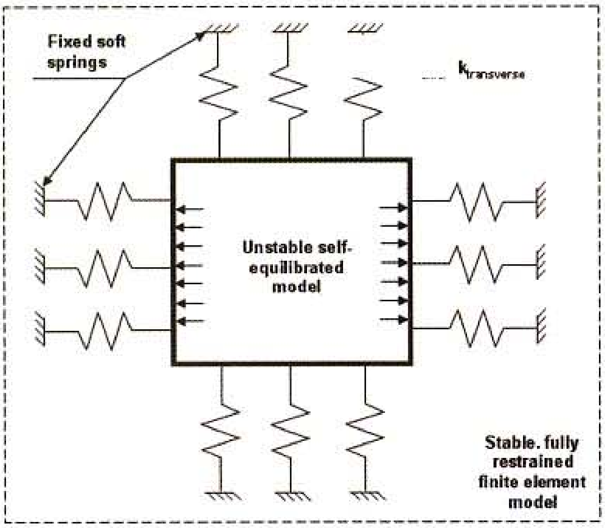
Earlier in this lesson we explained that to prevent rigid body movement along the axis of the assembly, at least one vertex on both the rim and the hub must be restrained in the axial direction. Without these restraints the assembly would have zero stiffness along the axial direction.

Now we'll present an alternative way of preventing rigid body movements in the model without restraining the two vertices.

Soft springs option

Theoretically, we do not expect the model to sway in the axial direction due to the action of some external loading (none that would cause such action exists in our model). All of the loads, which are applied in the form of the shrink fit contact condition, are inherently balanced. Finite element method, however, does not recognize this fact and a small inaccuracy, a numerical error, or mesh asymmetry may cause the model to displace uncontrollably in axial direction. All such cases can be stabilized by the Soft spring option.

When this option is activated, the model is surrounded by springs with stiffnesses that are negligible relative to the stiffness of the model (see the following figure). The finite element model is then stabilized and restrained against all rigid body motions.



Note that the above procedure works as long as the model is self-equilibrated, or the net magnitude of the external load is so small that the soft springs are able to compensate for it.

Inertial relief option

Another method of preventing rigid body movements is Inertial relief. Rather than adding artificial stiffness to counteract the load imbalance, as is done by selecting **Use soft springs to stabilize model**, this option adds an artificial balancing load eliminating any load resultant along unrestrained directions.

This options should not be used with the intention to stabilize an analysis where gravity, centrifugal or some thermal loads are defined.

In our case, both the **Use soft springs to stabilize model** and the **Use inertial relief** solver options can be used.

---

17 Create new study.
Copy the existing study `shrink fit` into a new study called `soft springs`.

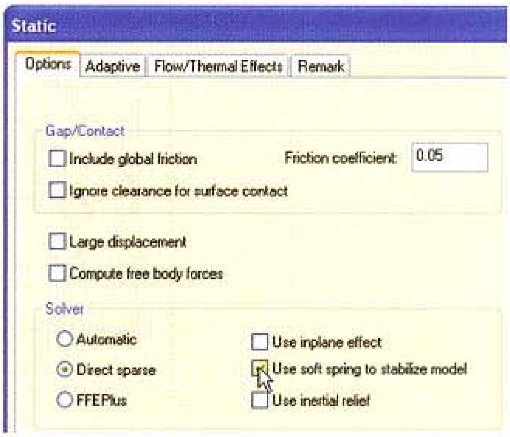
18 Suppress axial restraint.
Under `soft springs` study, right-click `Restraint-2` and select **Suppress** to release the axial constraint.

19 Select soft spring option to stabilize model.
Right-click the study `soft springs` and select **Properties**.

Under the **Options** tab, activate the **Use soft springs to stabilize model** option.

Select the **Direct sparse** solver.

Click **OK.**



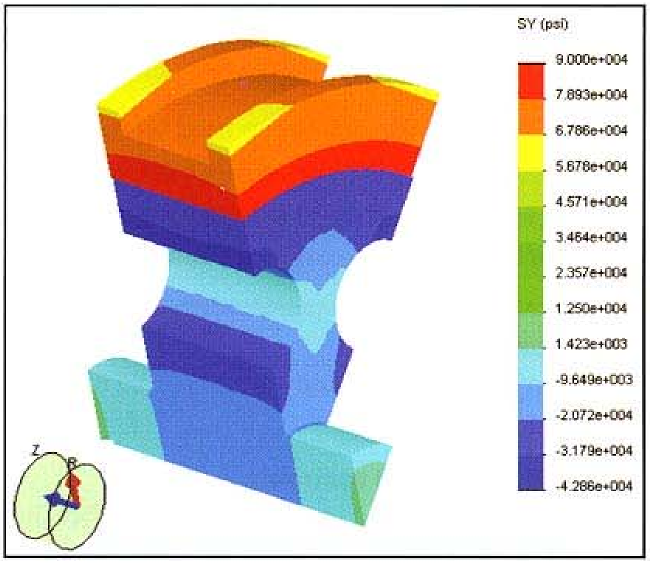
20 Run the study.

21 Plot hoop stresses.
Plot the distribution of hoop stresses.

Set the **Units** to **psi**.

Under **Settings**, select **Discrete** as the **Fringe Option**.

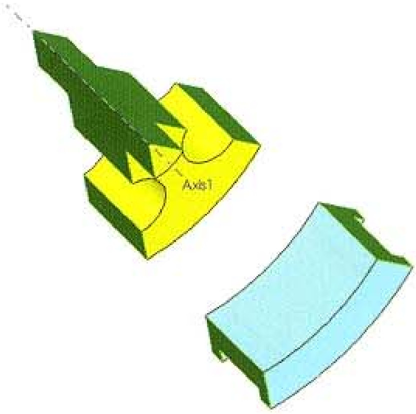
Set the maximum value of the stress legend to **90,000 psi**.



Comparing the results above with the corresponding plot in the previous study we see that they are indeed identical.

Summary

Interference between assembly components is allowed only if a **Shrink fit** condition is present in the assembly.



The results of a study with a **Shrink fit** condition are best viewed using a 1:1 scale of deformation.

To see results on contacting faces, use exploded view.

The results for axi-symmetric parts are best viewed in cylindrical coordinate systems.

This observation particularly applies to stress results other than von Mises stress. Von Mises stress, as a scalar value, is insensitive to the choice of reference coordinate system.

To prevent the rigid body motions the model must be stabilized in the axial direction. The most straight forward method is to restrain one vertex (point) on each model in the axial direction. Alternatively, we used the **Use soft springs to stabilize the model** option which surrounds the model with a layer of soft springs to provide a minimum stiffness in unrestrained directions.

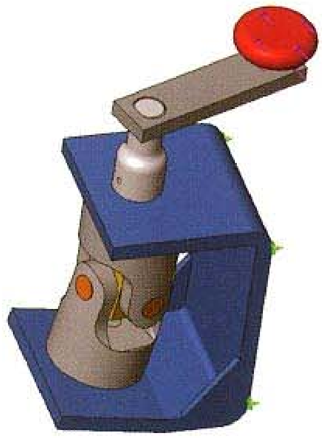
Lesson 5
Static Analysis of a Differential Assembly

Objectives

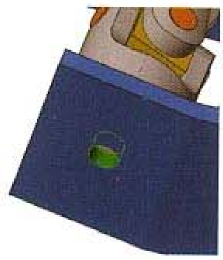
Upon successful completion of this lesson, you will be able to:

- Analyze more complex solid mesh assemblies with various contact conditions
- Use initial clearance in definitions of local No penetration contact conditions
- Auto-generate the local contact definitions
- Analyze and judge the quality of solid finite element mesh
- Use the Remote Load feature to simplify the analysis
- Use and define Design Check plot

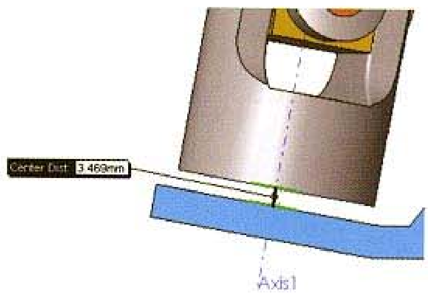
Problem Statement



The differential assembly shown in the figure below is used to transmit a torque from the vertical direction to the inclined direction indicated by `Axis1`. The assembly is welded to a secondary structure at the four locations on the back side of the bracket. The torque is generated by applying a 2.5 N horizontal force to the handle. (In the top view, this force is perpendicular to the handle arm.)



The shaft along `Axis1`, rigidly connected to the bottom yoke (`Yoke_female`) and passing through the bottom opening on the bracket, is not modeled. It is assumed that, due to improper manufacturing and elevated temperature from the friction in the shaft/bracket contact, this interface becomes temporarily blocked and the shaft consequently transfers all the torque to the bracket. (Further increase in the torque would loosen this connection and the differential assembly would begin to rotate.)



Furthermore, the geometry of the shaft ensures that the `Yoke_female` and the `RevBracket` do not come any closer than the initial clearance of 3.469 mm (see the figure to the right).

The goal of the analysis is to obtain the distribution of stresses and strains in the components of the differential.

Local Contact Conditions

In the first part of the lesson we will define all of the appropriate contact conditions without the help of COSMOSWorks.

1 **Open assembly.**
Open SolidWorks assembly `differential` from the `Lesson 5` folder.

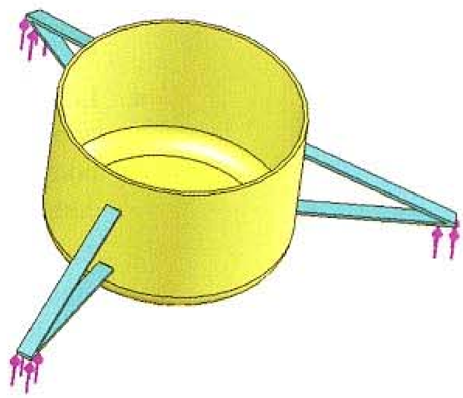
2 **Set COSMOSWorks options.**
Set the global system of units to **SI (MKS)** and set the units of **Length** and **Stress** to **mm** and **N/mm^2 (MPa)**, respectively.

Store the results in the SolidWorks document folder in the `Lesson 5 results` subfolder.

Because we are not interested in the stresses and the deformations of the crank arm, shaft, and knob, we will simplify the analysis by using a remote load feature.

Remote Load

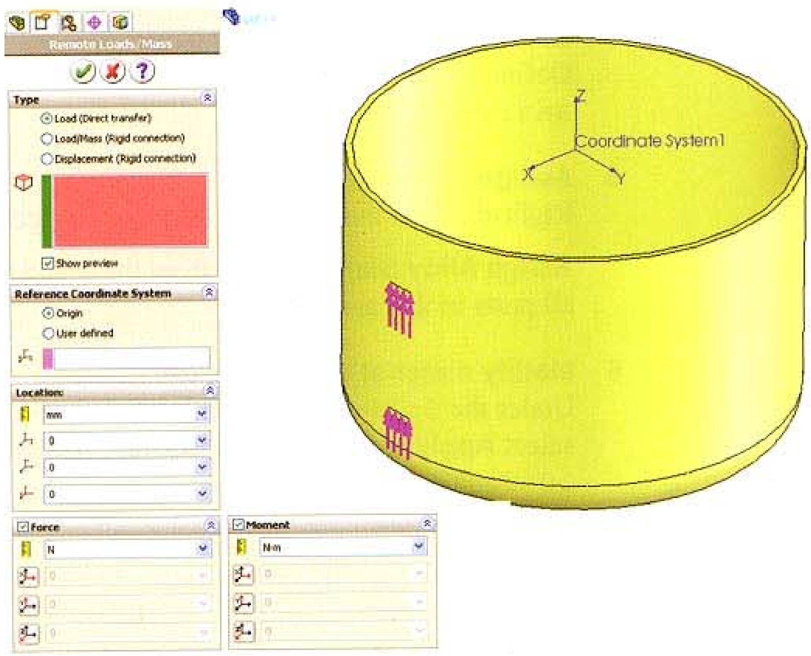
The **Remote Load** option found in the `Load/Restraints` menu allows the user to simplify certain assemblies prior to meshing, often helping reduce the size of the mesh. For example, examine a pot supported by three outriggers. A load is applied to the tip of each outrigger.



We are not interested in the deformations and stresses of the outriggers and we wish to concentrate solely on the analysis of the pot.

Taking advantage of the **Remote Load** option, we avoid modeling the outriggers and still are able to apply loads to the pot as if the outriggers were present.

Instead of analyzing the assembly, we analyze the pot as a part. Using the **Remote Load** menu, we apply a load to the split faces marking the area where the outriggers are attached to the pot.



The point of application of the **Remote Load** must be defined in the coordinate system selected in the **Remote Loads** menu.

There are three ways in which **Remote Load** can be defined:

- **Load (Direct transfer)** is applicable if the omitted component (the outrigger) can be assumed to be much more flexible than the analyzed part (the pot). The load that would have been applied to the outrigger is applied to the split faces of the pot and is expressed by means of equivalent loads and moments.
- **Load/Mass (Rigid connection)** is applicable if the omitted component is very rigid and can be assumed to displace as a rigid body. In this case the faces where the loads are applied are connected by invisible rigid bars to the point of load application.

  This option also allows you to specify a remotely located, isolated mass. This feature is useful when the gravity (or another constant acceleration load) is included or when performing a frequency analysis.
- **Displacement (Rigid connection)**, the third option in the **Remote Load** definition, is also applicable when the omitted component is very rigid and can be assumed to displace as a rigid body; however, the load needs to be applied as a prescribed displacement. In this case, the faces where the loads are applied are also connected by invisible rigid bars to the point of the load application.

---

3 **Activate configuration `Without_crank`.**
This step suppresses the crank shaft, arm, and knob.

4 **Define static study.**
Define a new **Static** study with **Solid mesh**. Name it `stress analysis`.

5 **Assign material.**
Right-click on the `Solids` folder and select **Apply material to All**.

Assign **Alloy Steel** material from the `cosmos materials` library to all parts in this assembly.

6 **Modify material of the bracket.**
Under the `Solids` folder, right-click on `RevBracket-1` part and select **Apply Material to All Bodies**. Assign `Aluminum Alloy 1060` to this part.

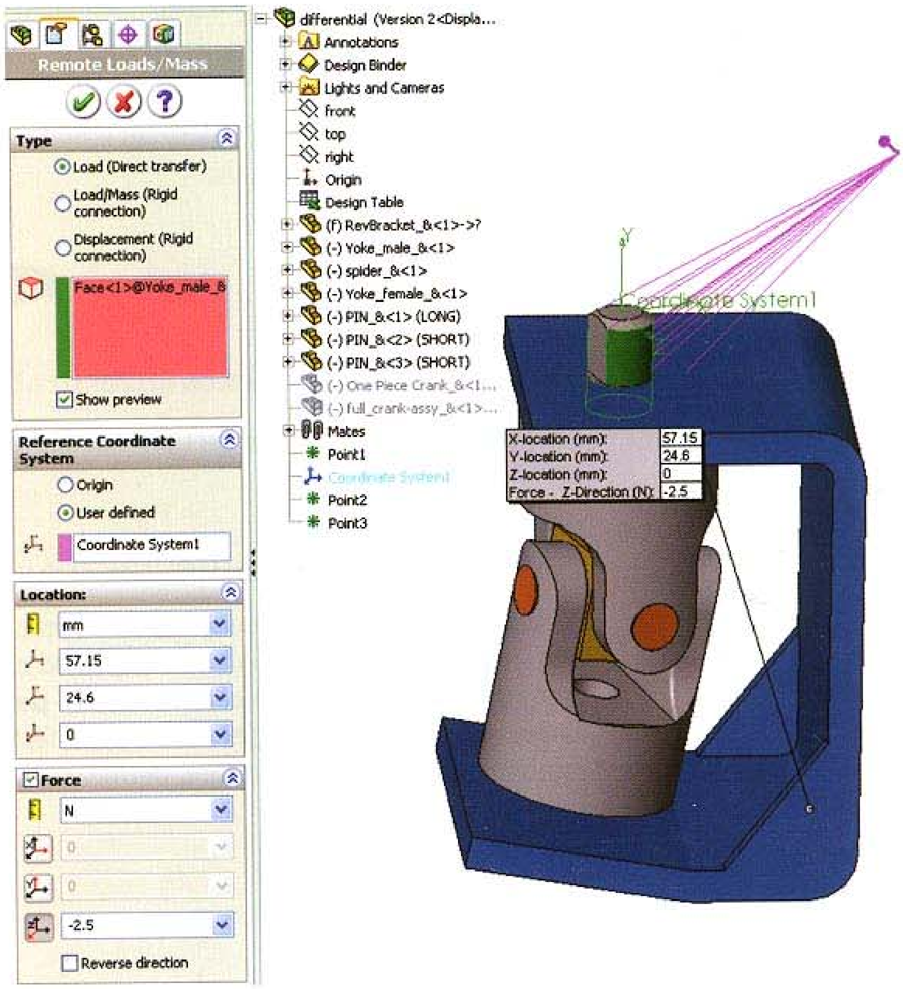
7 Define remote load.

Right-click `Load/Restraint` and select **Remote Load.**

In reference to the following figure, specify **Load (Direct transfer)** under the **Type** dialog, and select the face on the `Yoke_male` knob where the torque will be transmitted.

Note

By selecting **Load (Direct transfer)**, we are accounting for the possibility of having looser connections between the crank sub-assembly and the `Yoke_male` part, and for the fact that the `crank shaft`, `arm` and `knob` are manufactured from material that is softer than Alloy Steel.



Under the **Reference Coordinate System** dialog, specify **User defined** and select `Coordinate System 1` from the SolidWorks fly-out menu. The remote location of the force will be specified in this local coordinate system.

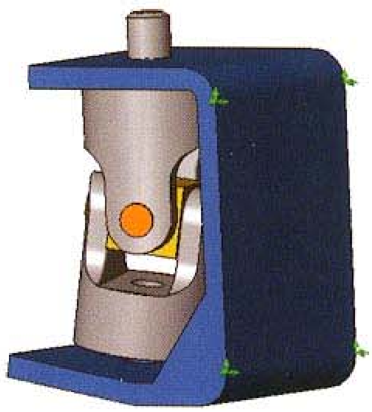
In the **Location** dialog, enter the following coordinates: **x-location: 57.15 mm**, **y-location: 24.6 mm**, and **z-location: 0 mm**.

In the **Force** dialog, specify **-2.5 N** in **z-direction**. (Note that the components of the force are also specified in the local coordinate system `Coordinate System 1`.)

Click **OK**.

Rename this remote force definition to `Remote load`.

8 Restrain bracket.
Apply **Immovable** restraint to the four vertices on the back face of the bracket, as shown in the figure.



Rename this restraint to `Immovable bracket`.

Note

This restraint simulates the fact that the bracket is welded onto a secondary stiff structure.

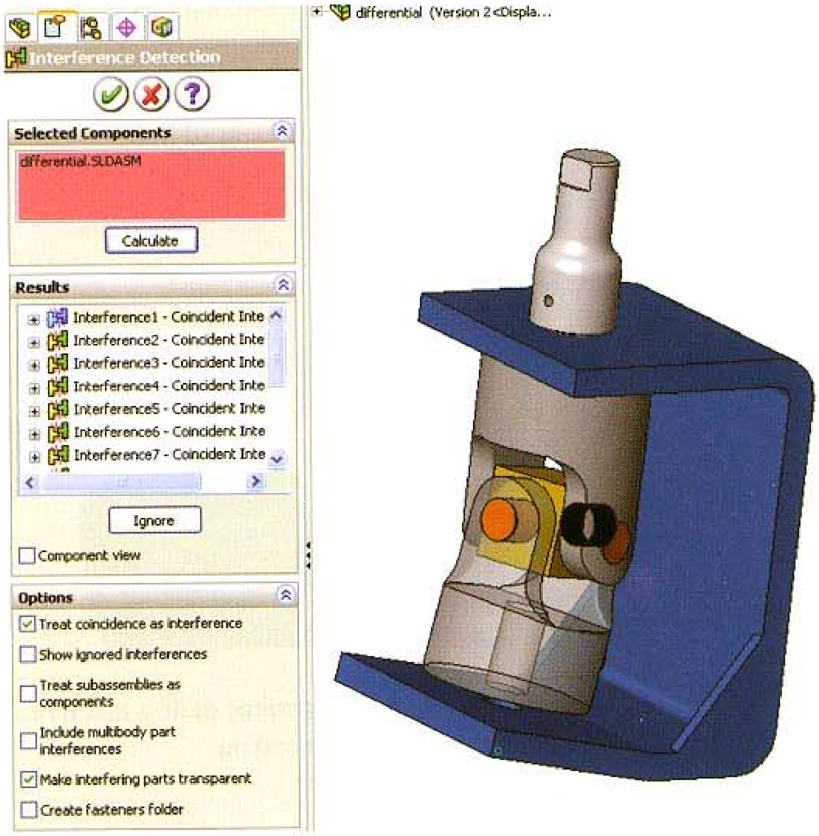
9 Explode the view.
Explode the view for easier definition of the contact conditions.

As you will see, there are numerous contact conditions in this assembly that must be specified. Due to the expected mutual slide of the parts in contact, the global and the component **No penetration** condition types can not be used (both of these conditions feature a **Node to Node** option only). We will, therefore, define each contact condition individually.

10 Locate all contacts in the assembly.
Under **Tools**, select **Interference Detection**. Under the **Options** dialog, activate the **Treat coincidence as interference** field.

Click **Calculate**.

Browse through and analyze the identified contact interfaces.



11 Modify global contact.
Set the global contact to **Free**.

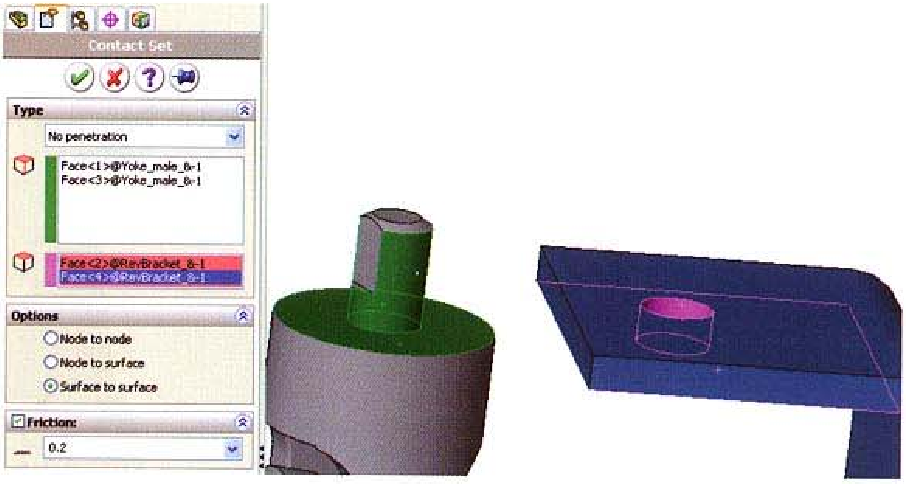
Note

By modifying this condition we want to make sure that no two parts will be bonded. All contacting faces will have local **No penetration** condition defined.

12 Define `Yoke_male` and `RevBracket_&` contacts.
Right-click on the `Contact/Gap` folder and select **Define Contact Set**. Specify **No penetration** under **Type** and **Surface to Surface** under **Options**.

Select the two faces on the `Yoke_male` part as the **Source**, and the two faces on the `RevBracket` as the **Target** (see the following figure).

Activate the **Friction** dialog by clicking the appropriate check box and enter the friction coefficient of **0.2**.

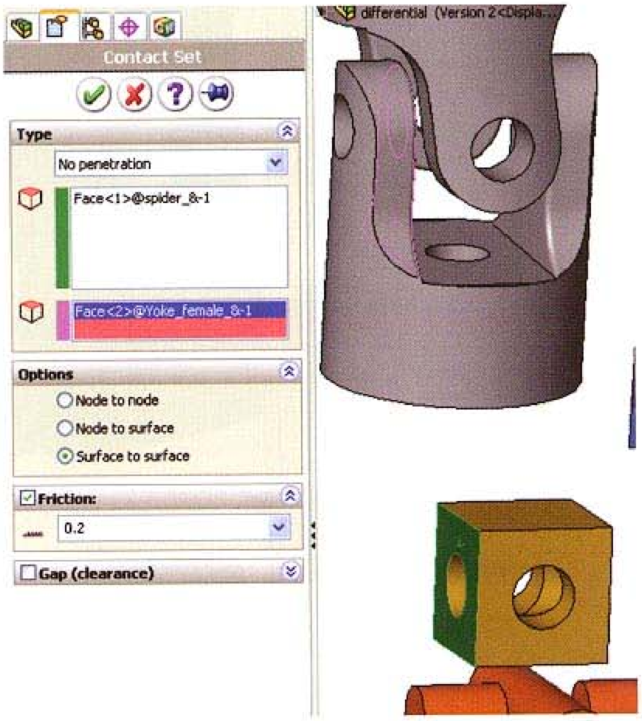


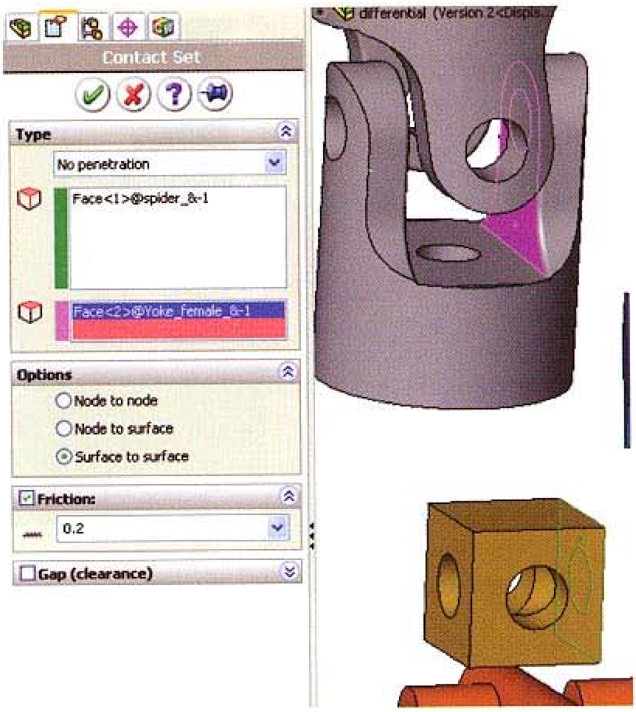
Note

Leave the **Gap (clearance)** dialog inactive. This option will be discussed later in this lesson.

13 Define `Yoke_male` and `Spider_&` contacts.

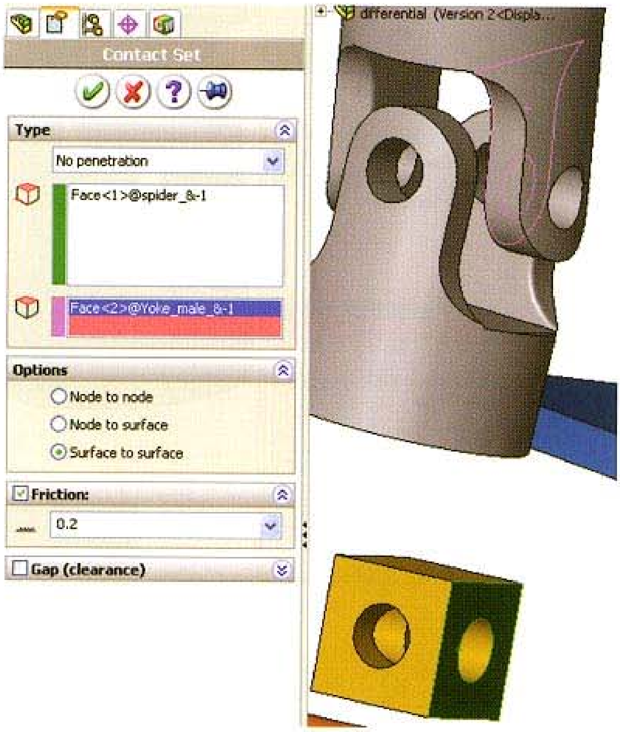
Define **No penetration**, **Surface to Surface** contact sets with the friction coefficient of **0.2**, as shown in the following figures.

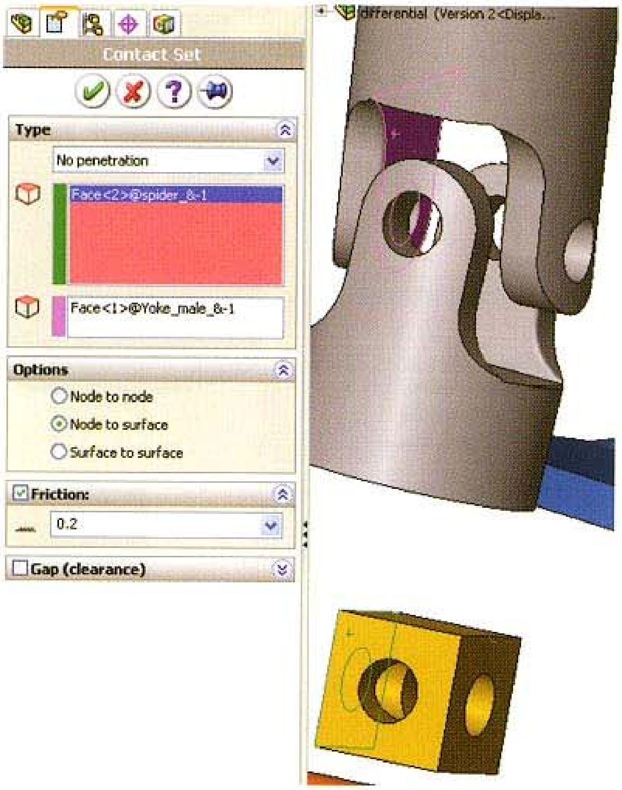




14 Define `Yoke_female` and `Spider_&` contacts.

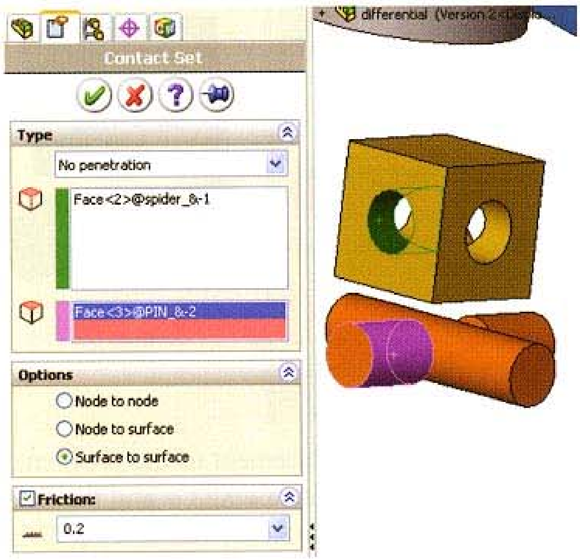
Define **No penetration**, **Surface to Surface** contact sets with the friction coefficient of **0.2**, as shown in the following figures.

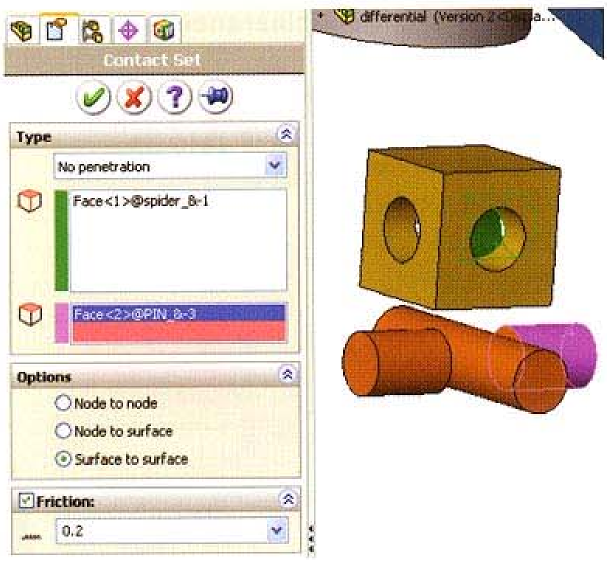


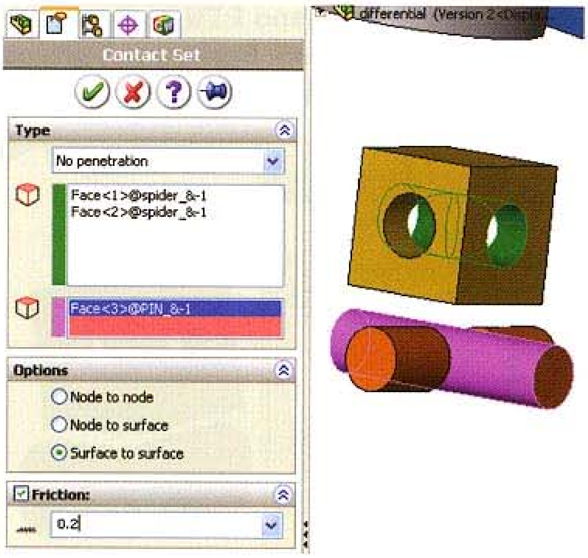


15 Define `Spider_&` and `PIN` contacts.

Define **No penetration**, **Surface to Surface** contact sets with the friction coefficient of **0.2**, as shown in the following figures.



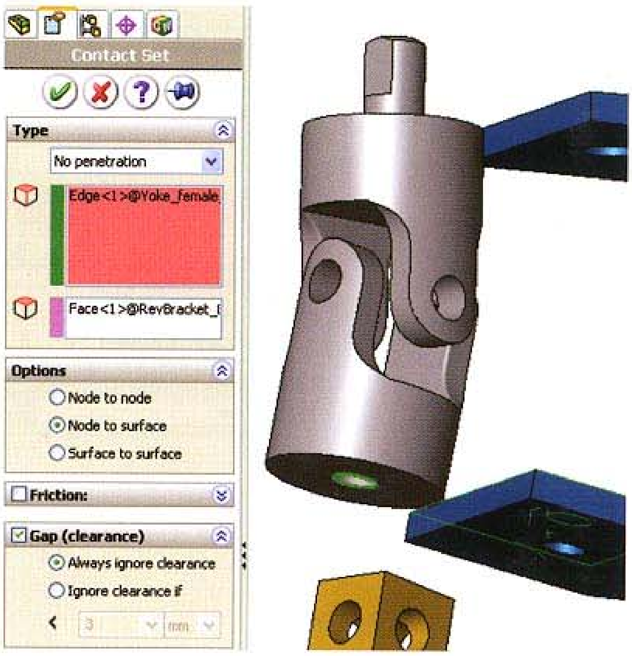




As specified in the statement of the problem, the geometry of the shaft ensures that the `Yoke_female` and the `RevBracket` do not come any closer than the initial manufacturing distance of 3.469 mm. We will simulate this constraint with the help of the **No penetration** contact condition with **Gap (clearance)** settings.

16 Define `Yoke_female` vs. `RevBracket` contact.

Define a **No penetration, Node to Surface** contact set with the rim of the cylindrical opening on the `Yoke_female` as a **Source** and the face on the `RevBracket` as a **Target**.



Activate the **Gap (clearance)** option box and select **Always ignore clearance**. (See the discussion on the **Gap (clearance)** option.)

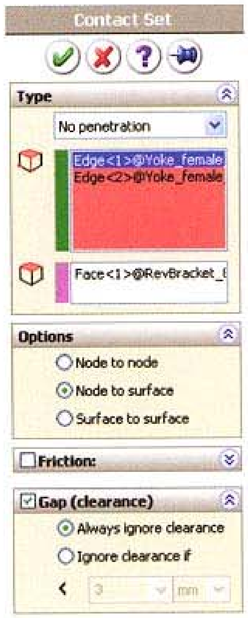
Note

We specified a **Node to Surface** contact since our source entity is an edge. The Surface to Surface option would not be appropriate in this case.

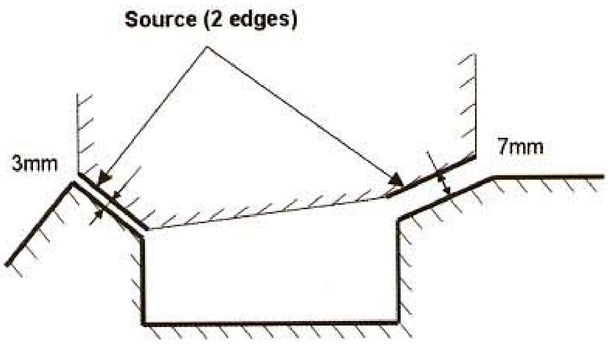
The above contact condition ensures that the two entities will not come any closer than the initial manufacturing distance of 3.469 mm, but allows them to separate.

Gap (clearance) option

This feature enforces the contact offset equal only to the initial geometrical distance between the target and the source. The options of this feature enables the user to select whether the initial geometrical offset should be applied to all of the source and target entities within a specific contact set or only to those with the initial separation distance smaller than the **User defined value**.



Let us demonstrate this feature on the following example:



- If the **Always ignore clearance** option is activated in the **Gap (clearance)** dialog, no node along the specified source edges will be allowed to come closer than their respective initial geometrical separations of 3 mm and 7 mm. All the points along the source edges will, however, be permitted to separate further.
- If **Ignore clearance if < 4 mm** is specified, for example, the 3 mm contact will behave as described above, while the 7 mm contact will be allowed to fully close (if the appropriate load is specified).

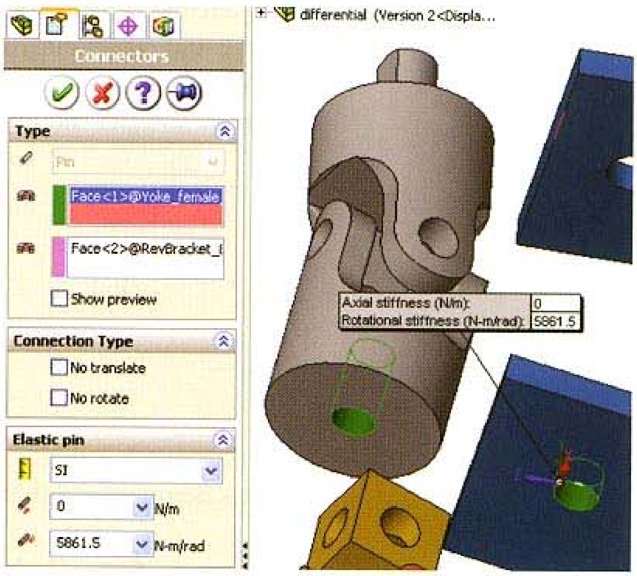
The last restraint must ensure that the cylindrical openings on the `Yoke_female` and `RevBracket` remain aligned and that these two openings remain connected for the transmission of the torque (they are physically connected by a shaft). Without this condition, the differential is free to rotate and our solution may fail or be inaccurate.

17 Define `Yoke_female` and `RevBracket` connector.

Specify a Pin connector between the cylindrical openings of the `Yoke_female` and that of a `RevBracket` (see the figure below).

In the **Connection Type** dialog, make sure that both the **No translate** and **No rotate** options are NOT selected.

In the **Elastic pin** dialog, enter **0 N/m** for the **Axial stiffness** and **5861.15 N-m/rad** for the **Rotational stiffness**.



Rotational and Axial stiffness

Assuming a cylindrical shape with a constant cross-section, the rotational stiffness can be calculated using the formula:

$$K_{ROT} = \frac{JG}{L}$$

where $J = \frac{\pi r^4}{2}$ is the polar moment of inertia for the circle with radius r, G is a shear modulus of the material, and L is the length of the shaft connecting our two points (effective length of the shaft). Substituting our values into the above equation, we obtain $K_R = 5861.15$ N-m/rad.

The axial stiffness of a cylindrical shaft with a constant cross-section can be calculated using the formula:

$$K_{AXIA} = \frac{EA}{L}$$

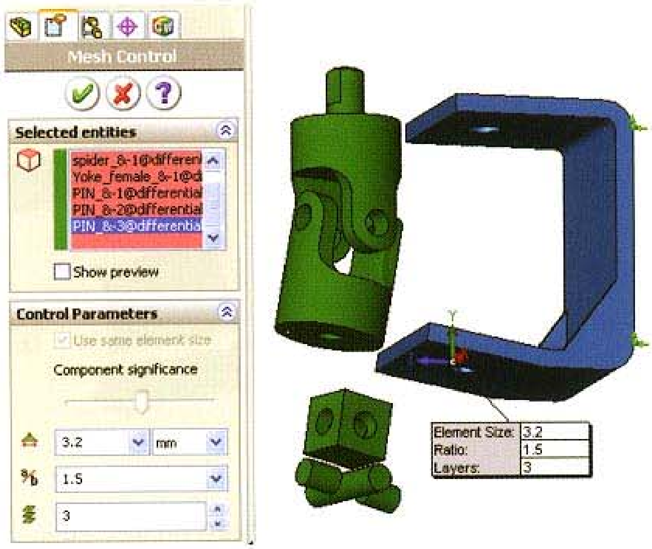
where E is the Young's modulus and $A = 2\pi r^2$ is the cross-sectional area of the circle with radius r.

All of the necessary contact conditions have been defined.

Our main concerns are the components of the differential; namely, `Yoke_male`, `Yoke_female`, `Spider`, and the `Pins`. We will, therefore, request a finer mesh for these parts.

18 Apply mesh control to selected parts.

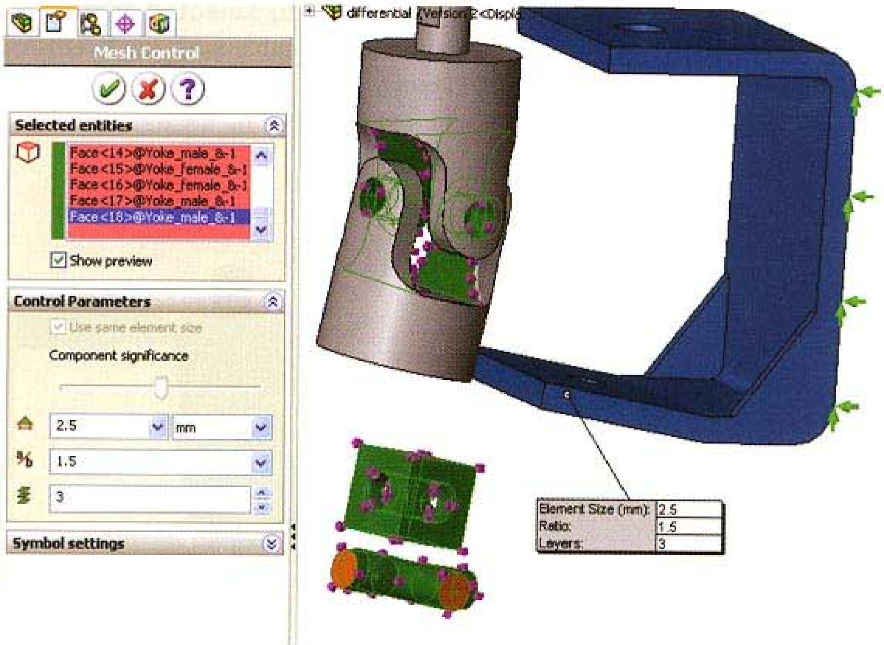
Apply a local mesh control with the **Element size** of **3.2 mm**, **Ratio** of **1.5**, and **Layers** equal to **3** to `Yoke_male`, `Yoke_female`, `Spider`, `Pin_&<1>(LONG)`, `Pin_&<2>(SHORT)`, and `Pin_&<3>(SHORT)`.



For more accurate contact representation, the touching faces on the above five parts will be discretized with fine mesh.

19 Apply mesh control to faces.

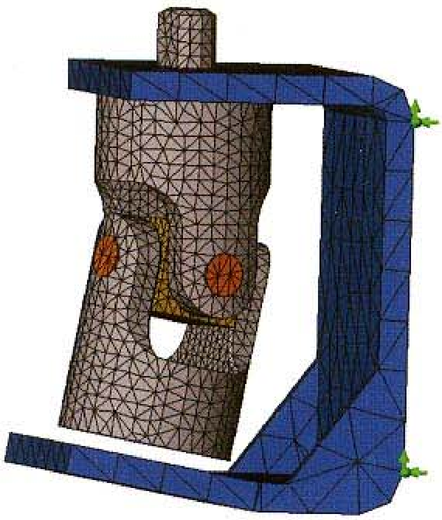
Apply a local mesh control with the **Element size** of **2.5 mm**, **Ratio** of **1.5**, and **Layers** equal to **3** to all the faces in contact pertaining to all five parts of our interest (those that we used in the previous mesh control).



20 Mesh the assembly.

Mesh the assembly with **Draft** quality elements and the default **Element size** of **10.9 mm** and a **Tolerance** of **0.545 mm**.

The resulting mesh is shown in the figure below.



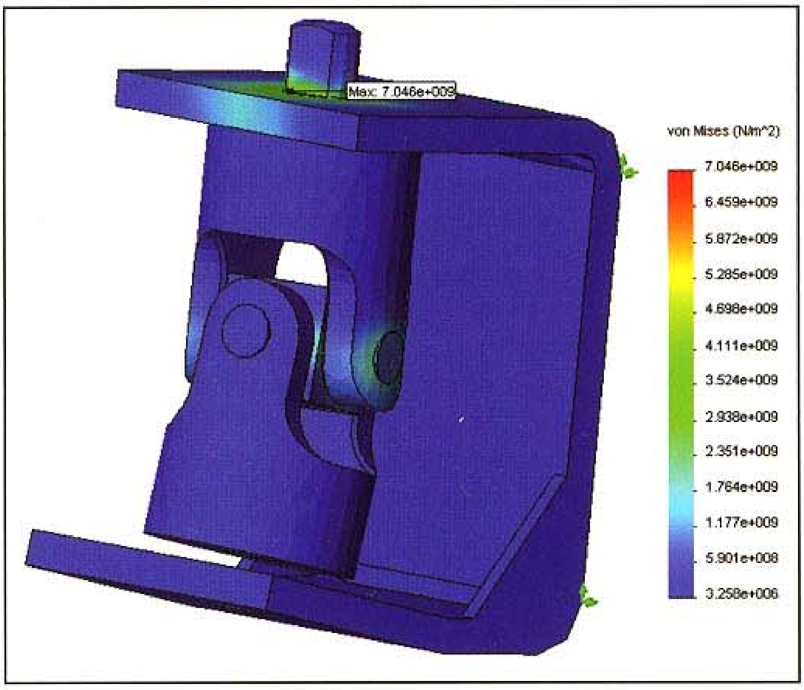
21 Set properties of study.
Specify **Direct Sparse** solver for this analysis.

Note

The **Direct Sparse** solver is specifically chosen because we have a larger number of contacts and connectors, but the size of the model is still rather small. We expect this solver to be more efficient than an FFEPlus iterative solver. In general, however, with the increasing size of the problem, the FFEPlus iterative solver becomes more efficient and would be our choice instead. For more information on solvers, please consult Appendix A.

22 Run the study.
The study will run for approximately five minutes and complete successfully.

23 Display and animate stress results.
Display and animate the distribution of the von Mises stresses in the model.



We observe that the stresses in many parts of our model are well above the yield strength of Alloy steel (620 MPa).

Note that our mesh is rather coarse and that **Draft** quality elements were used. For reliable stress results, we would have to refine the mesh and use **High** quality elements. We may also use a more efficient technique to define the local contact sets.

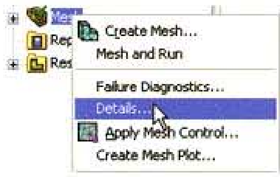
High Quality Mesh Analysis (Optional)

In the second part of this lesson, we analyze the quality of the current mesh, generate a new High quality mesh, auto-generate the contact conditions, and post-process the results of the refined study.

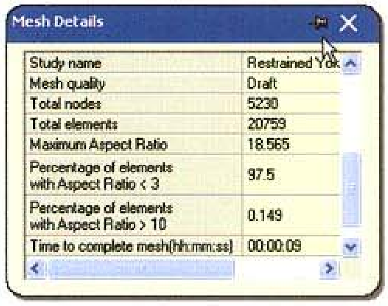
First, let us have a look at the quality of our current mesh.

24 Analyze details of current coarse Draft mesh.

Right-click on the `Mesh` folder and select **Details**.

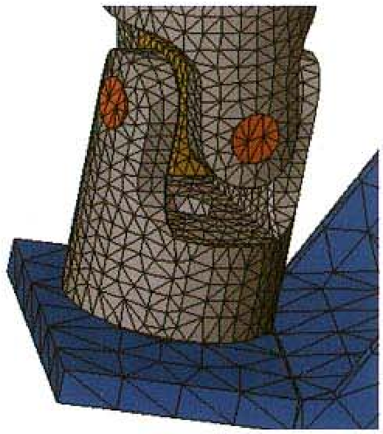


The dialog window lists the basic information about our current mesh. Scroll all the way down and note the three rows that provide information about the **Aspect Ratio**. (For the detailed information on the Aspect Ratio, please consult Appendix A.)



The **Maximum Aspect Ratio** of 18.5 is rather large and the **Percentage of elements with Aspect Ratio < 3** of 97.5 is acceptable.

The overall judgement about our mesh can be achieved by simple visual inspection. Two Draft quality elements per thickness of the Yoke's walls, as well as in through-thickness direction in the pins, are not enough for reliable stress and strain results.



In general, we would require at least four Draft (three High) quality elements in the through-thickness directions when stresses or strains are of any concern. A fairly coarse mesh of the `RevBracket` is not a major concern. We would, however, still require a minimum of one (High quality) or two (Draft quality) elements through the thickness. The contact interfaces do not necessarily need to be refined any further, especially if in High quality elements, unless contact stresses are of any importance.

25 Define new study.

Define a new **Static** study named `stress analysis-refined`.

26 Copy loads, restraints and mesh.

Copy the `Load/Restraint`, `Solids`, and `Mesh` folders to our new study.

27 Define local contact sets.

We will take advantage of the automatic contact generation feature in COSMOSWorks.

Right-click on the `Contact/Gaps` folder and select **Find Contact Sets**.

Under the **Options** dialog, check the **Touching faces** box.

In **Select components**, select all the parts and subassemblies in our assembly. You can multi-select all of the components by windowing the assembly.

Click **Find faces**.

All detected contact sets are listed in the **Results** dialog. You can browse through and view each by clicking on it.

Select all found contact sets under the **Results** dialog. Under **Type** and **Options**, select **No penetration** and **Surface to Surface**, correspondingly.

Click the **Create** contact sets button. All the contact sets will be generated and listed under the `Contact/Gaps` folder.

Click **OK** to close the **Find Contact Sets** dialog.

At this point, we would have to check each contact set and specify the friction coefficient. Additionally, we would have to manually create the last **No penetration** contact set with the initial geometrical offset between the `Yoke_female` and `RevBracket`. As this would be just a repetition of the steps from the previous study, we will copy all of the contact definitions from the previous study and remesh quickly instead.

28 Delete all contact sets in study `stress analysis-refined`.

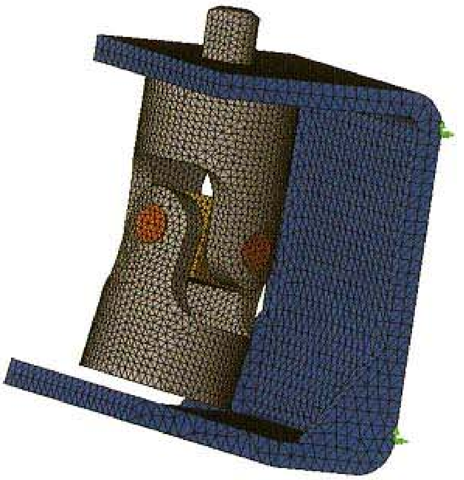
29 Copy all contact sets.
Copy all of the contact sets from the study `stress analysis` to the study `stress analysis-refined`.

30 Modify mesh control on parts.
Modify the mesh control on all the parts. Specify the **Element size** of **1.8 mm**, **Ratio** of **1.5**, and **Layers** equal to **3**.

31 Suppress mesh control on contact faces.
The mesh control on the parts is already fine enough. Further refinement of the contact faces is not necessary unless the contact stresses are of interest.

32 Mesh the model.
Mesh the model with **High** quality elements, a global **Element size** of **3.2 mm**, and a **Tolerance** of **0.16 mm**.

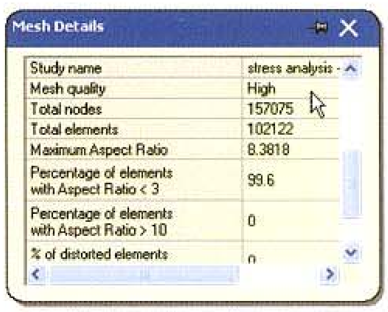
The resulting mesh can be seen in the following figure.



We observe that the recommended numbers of elements in through-thickness directions for all components are now satisfactory.

33 Display the mesh details.
We note that the Maximum Aspect Ratio decreased to an acceptable value of 8.312 and that the Percentage of elements with Aspect Ratio < 3 increased to 99.6.

Meshes with these parameters can be regarded as very good.



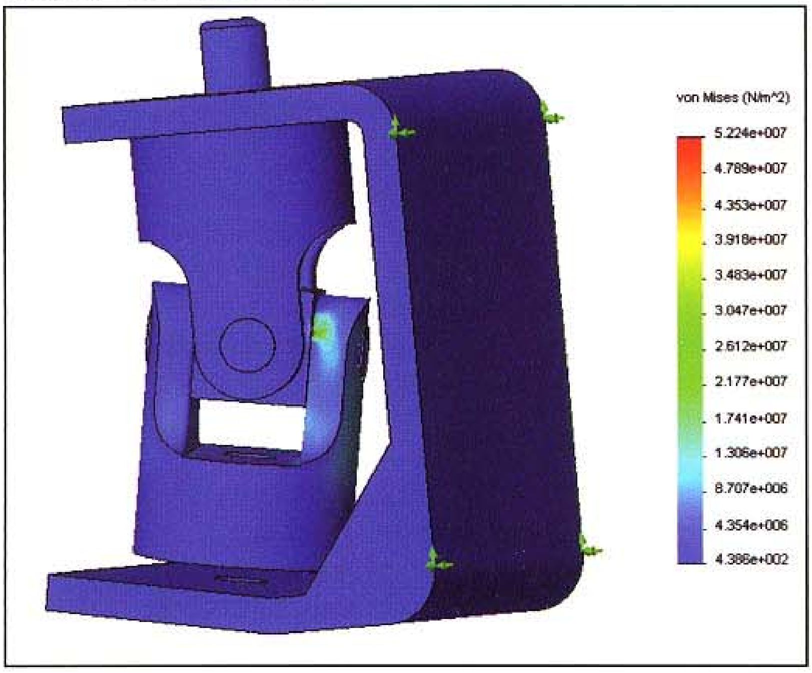
Also note that the number of **Total nodes** increased 31 times to 157075.

Due to the time required, this study has been calculated and its results can be found in the `completed and calculated` subfolder of the `Lesson 5` directory on the CD accompanying this training manual.

34 Save and close this assembly.

35 Open the SolidWorks assembly `differential_completed`.
Open `differential_completed` assembly from the `completed and calculated - High Elements` subfolder in the `Lesson 5` directory.

36 Display stress results.



The plot indicated the maximum von Mises stress of only 52.2 MPa, which is significantly smaller than the yield stress of the Alloy steel (620 MPa). It can be seen that this stress is close to the contact regions, while the stress within the parts is significantly smaller.

We would, therefore, conclude that this assembly is designed with the sufficient factor of safety.

Design check plot

A **Design Check Plot** can be used to conveniently plot the distribution of the factor of safety in the assembly. The procedure on how to define this postprocessing plot and some of the features are shown and discussed in the following step.

37 Plot a factor of safety distribution.

Right-click on the `Results` folder and select **Define Check Plot**.

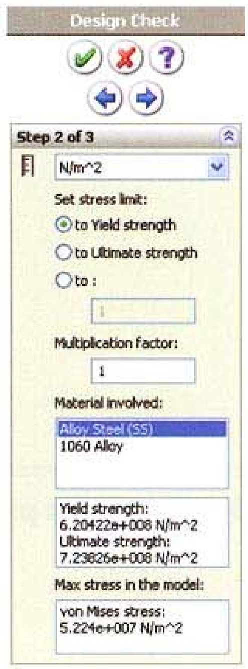
As nearly all standards require the use of von Mises stress for the calculation of the factor of safety, choose **Max von Mises stress** in the first window.

Note

The inequality $\frac{\sigma_{vonMises}}{\sigma_{Limit}} < 1$ shown in the dialog window is not the definition of the factor of safety and the user should not be confused by this expression. It is a definition of the von Mises yield criterion used by the software to identify material points that experience yielding (with factor of safety < 1). Users should ignore this expression at this time. It will become clearer as the user becomes more proficient with the software and theory.

Click **Next**.

The second dialog window specifies the material constant that will be used as a comparison against the von Mises stress (selected in the previous window). Again, since most of the standards specify the material yield stress, select **Set stress limit to Yield strength**.

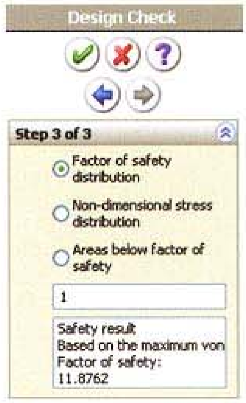


Note

We also remind the user that our computations are valid within the scope of the linear elasticity traditionally limited by the yield stress point on the material stress-strain curve.

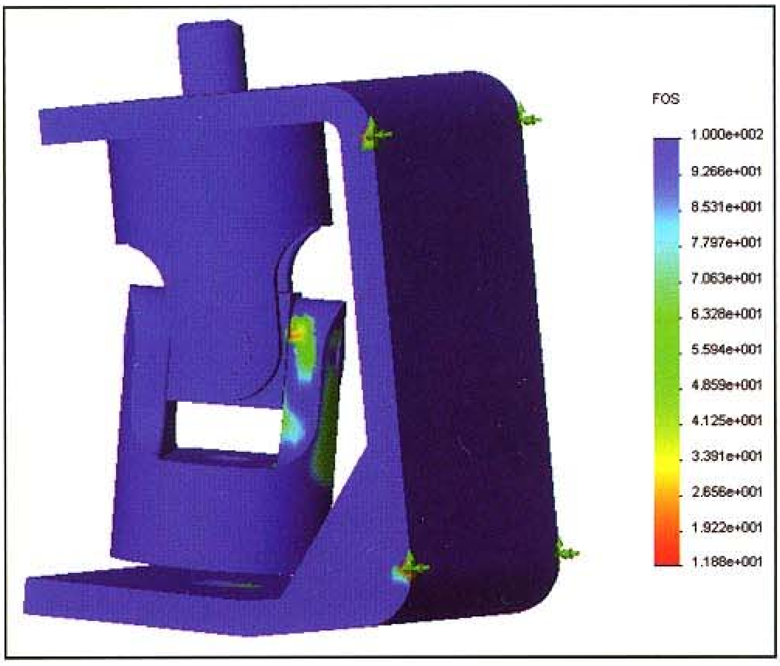
Also note that the bottom of this dialog lists the materials used in the assembly along with their corresponding yield strength information. The units can be set at the top of this dialog window.

Click **Next**.



The third window allows the user to specify the quantity to plot. Select **Factor of safety distribution**.

Click **OK** to generate the plot.



We can see that the lowest value of the factor of safety is 11.9, which can be viewed as rather conservative.

38 Close the assembly.

Summary

In this lesson, we analyzed an advanced solid mesh assembly with various contact conditions and connectors. The creation of the local contact sets was shown and practiced.

The pin connector with specified rotational stiffness was used to simulate a real shaft. The remote load feature was used to remotely apply the load without a need to model the linking parts.

We analyzed the quality of the finite element mesh and discussed the optimum size of elements with respect to the characteristic dimensions of the model.

Finally, a new postprocessing feature, **Design Check Plot**, was introduced. In this lesson, we used this feature to plot the distribution of the factor of safety and discussed various options available in the definition of this plot type.

Lesson 6
Shell Analysis of a Pulley

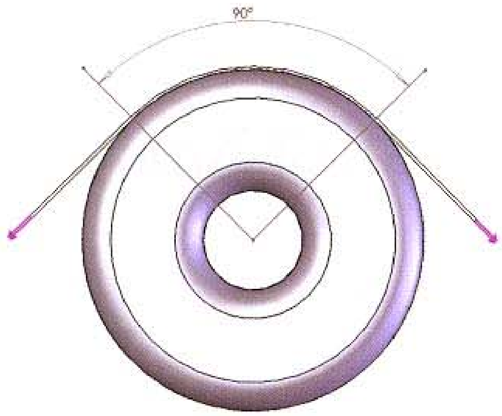
Objectives

Upon successful completion of this lesson, you will be able to:

- Create a mid-plane shell element mesh
- Create a shell mesh from selected surfaces
- Perform structural analyses and analyze results using shell elements on parts
- Evaluate mesh adequacy to model stress concentrations

Project Description

In this lesson, we analyze a stamped steel pulley used as an idler pulley in an automotive belt drive. It is known that the vertical resultant force exerted on the pulley by the belt is 500 N. (It can therefore be found from the force equilibrium that the corresponding belt force is equal to 353.55 N.) Our goal is to determine the deformations and stresses that develop in the pulley. Because one dimension of the pulley geometry (thickness) is much smaller that the other dimensions, meshing this geometry with solid elements is not practical.



Model Preparation

To mesh the model properly with solid elements, we would need to place two layers of tetrahedral elements across the thickness. Such a mesh calls for a very small element size and consequently results in a very large number of elements.

We will mesh this model with solid elements later for comparison, but first we will use shell elements. We will construct a shell mesh using two distinct modeling techniques: **Shell mesh using mid-surfaces** and **Shell mesh using surfaces**.

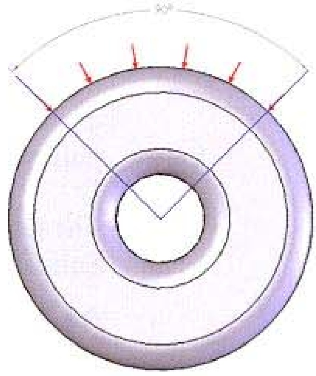
We can take advantage of the symmetry of the geometry, loads, and restraints by analyzing one half of the pulley while simulating the missing half with symmetry boundary conditions.

The pulley is simple enough to analyze without using symmetry. Idealizations (namely, using shell elements in place of solids and using one half of the model in place of the whole model) are utilized here only as a learning example.

1 **Open part.**
Open the SolidWorks part file `pulley`. This file is located in the `Lesson 6` directory.

Note

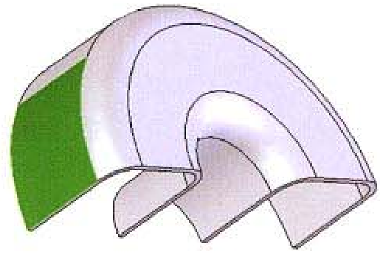
Split lines define a face extending over a 90° section of the pulley. This area is where we apply a belt load as pressure exerted on the pulley.



2 Make symmetry cut.

Unsuppress the cut (the last feature, named `symmetry`) in the SolidWorks FeatureManager.

Toggle to the COSMOSWorks environment.



3 Set COSMOSWorks options.

Set the global system of units to **SI (MKS)** and the units of **Length** and **Stress** to **mm** and **N/mm^2 (MPa)**, respectively.

Store the results files in the `results` folder in the SolidWorks document folder.

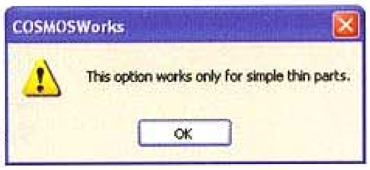
Shell mesh using mid-plane surfaces

4 Create study.

Create a study named `pulley shells-midplane`. Select **Static** as the **Analysis type** and select **Shell mesh using mid-surfaces** as the **Mesh type**.

Working with Midsurface Shells

COSMOSWorks issues a warning that **Shell Meshing in mid-planes** works only for simple, thin parts.



This is because an automatic extraction of the midplane for more complicated geometries may be impossible. A different approach is required in such cases and will be practiced in the second part of this lesson.

As we will see, the pulley is indeed a simple thin part for which **Shell Mesh using mid-surfaces** works well.

Note that the icon next to `Body1` is a surface expressing the fact that a solid geometry of the pulley is represented through its midplane surface.

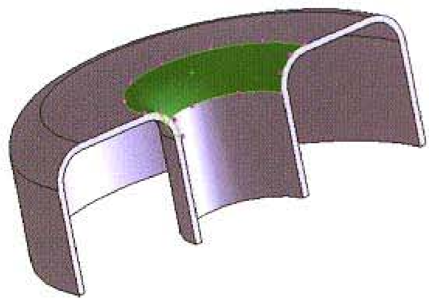
Note

The material definition (AISI 1020) is automatically transferred from SolidWorks.

An important observation is that we do not have to define the thickness of this surface. The 2 mm thickness is based upon the geometry of the SolidWorks part.

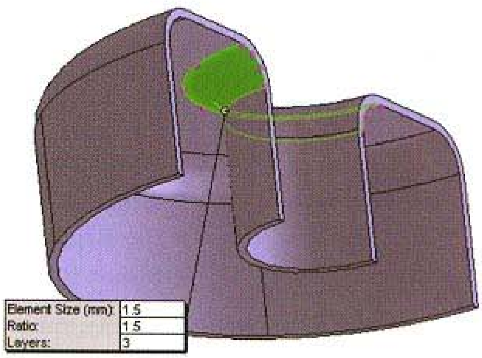
For now, let us skip the definition of loads and restraints and go directly to the mesh creation.

5 **Mesh control.**
Apply a mesh control on the rounded face (see the figure) with the default **Element size** of **1.5 mm**. Keep the **Ratio** and **Layers** options at their default values of **1.5** and **3**, respectively.



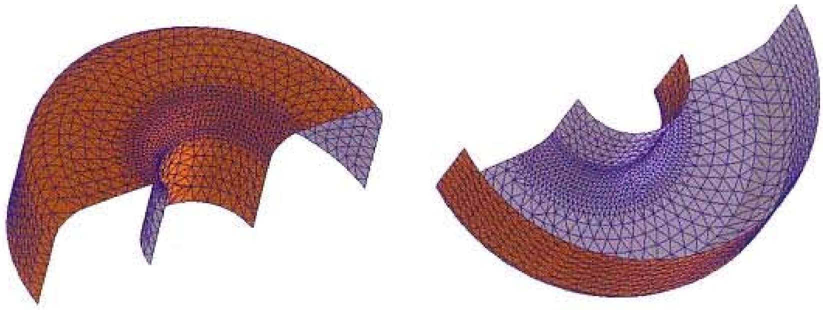
Note

Alternatively, we could select the inner rounded face (see the figure). Both conditions would indicate an identical mesh control applied to the rounded face located on the mid-surface.



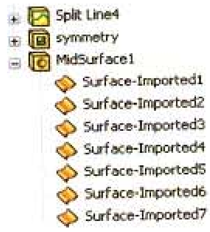
6 **Create mesh.**
Create a **High** quality mesh with the global **Element size** of **4.5 mm** and the **Tolerance** of **0.225 mm**. Make sure that the **Automatic Transition** option is off.

The resulting mesh can be seen in the following figures.



Note

When the **Create Mesh** command is issued, SolidWorks first creates a surface named `Midsurface1`, located in the midplane of the model.

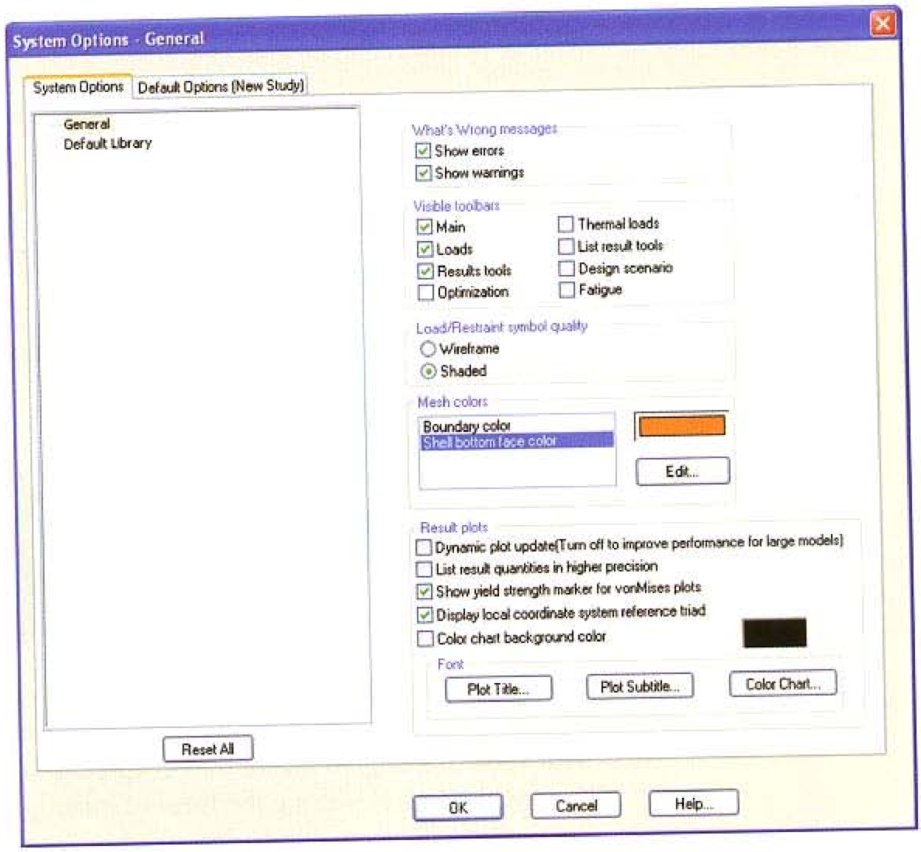


`Midsurface1` appears in the SolidWorks FeatureManager as an imported feature.

This surface is meshed with the shell elements.

Shell Mesh Colors

A quick examination of the shell mesh reveals that the shell elements on each side of the mid-surface have different colors. To interpret the meaning of the colors assigned to the mesh, go to the COSMOSWorks **Options** menu and display the **System Options** tab.

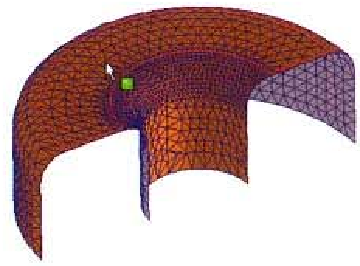


The **Mesh colors** section indicates that the shell bottom faces are marked with an orange color. The shell top faces assume the part color, gray in this case. This fact is very important in the postprocessing phase.

Furthermore, we see that the inside and outside are colored uniformly (i.e. gray and orange colors do not alternate on any side). Such aligned shell mesh is required for correct postprocessing.

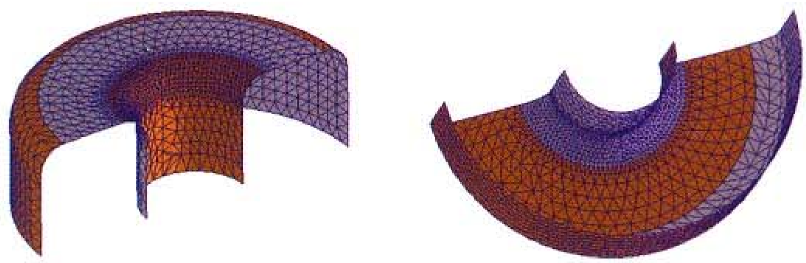
In some cases, however, the shell mesh does not need to be aligned at the end of the meshing phase. In such cases, the mesh need to be aligned manually. We can practice flipping the top and bottom of the shell mesh with our current mesh.

7 **Flip the mesh on one face.**
Left-click on the face indicated in the figure.



Right-click on `Mesh` and select **Flip shell elements.**

The result of this operation can be seen below.

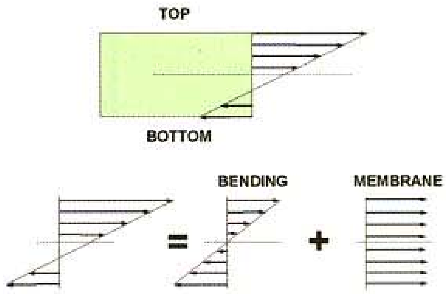


We observe that the colors are not uniform and the shell mesh is misaligned. While the finite element computations would be correct with such misaligned mesh, the postprocessing would show meaningless results along the lines of misalignment. The importance of the shell mesh alignment is described in the discussion section below.

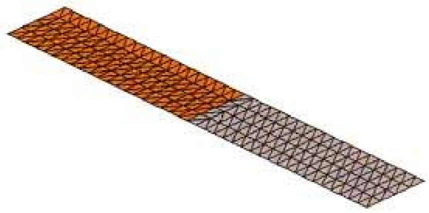
Shell Element Alignment

Before proceeding further, we need to explain why shell element alignment is important. Shell elements can model bending; therefore, most often, different stress results are reported at the top and bottom of shell elements.

Using the postprocessing options for shell elements, we can choose to display the stresses on the **top** or **bottom**. Additionally, stress distribution in the through-thickness direction can be divided into two components: **bending** and **membrane**. All four options are demonstrated in the figure to the right.

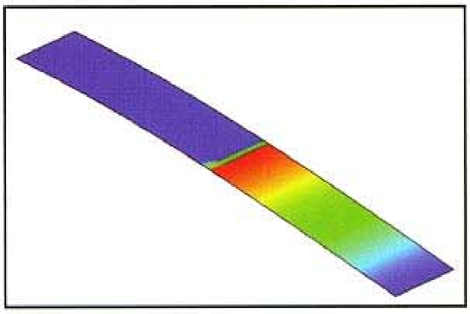


Let us depart for a moment from the pulley and examine a rectangular cantilever beam meshed with misaligned shell elements.



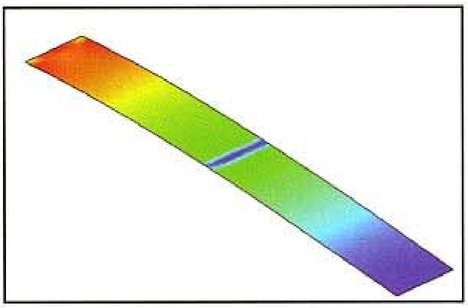
Say we want to display the P1 stress (maximum principal stress) at the top of the beam.

Because the shell element orientation in the model is inconsistent, the stress plot is in error.



Now, instead of the P1 stress, we plot the von Mises stress.

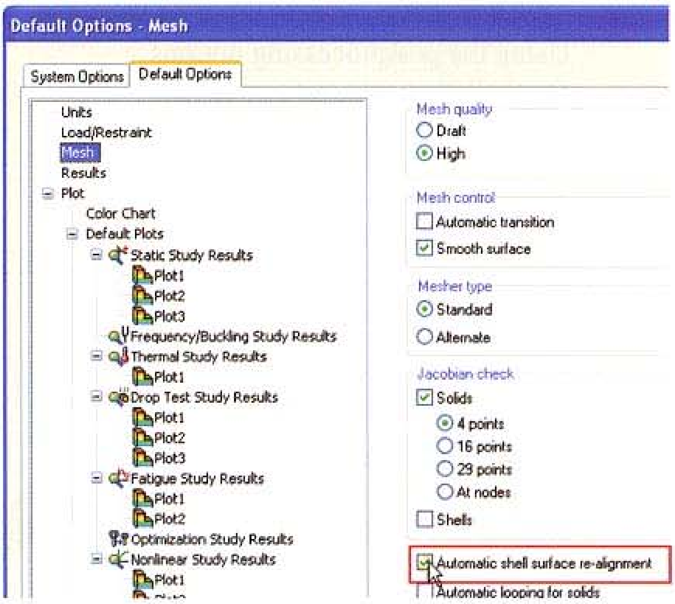
The fact that the plot is erroneous again becomes obvious along the line of misalignment.



This error is because stresses are averaged before the von Mises stress is calculated. Averaging between the results on the top and the results on the bottom of the shell elements across the misalignment line results in nearly zero stress. Having explained the importance of shell element alignment, we now return to the pulley lesson.

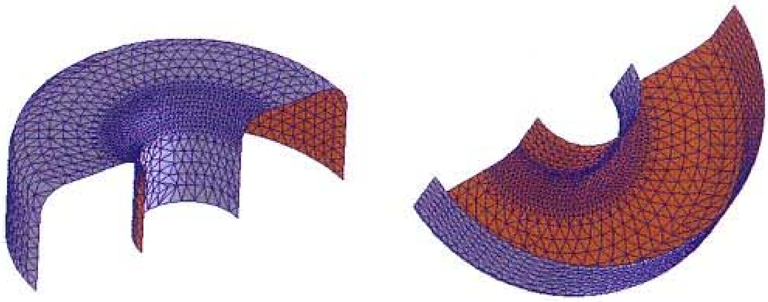
Automatic shell surface re-alignment

Turn on this option to automatically align the surface of the generated shell mesh. It can be found under the COSMOSWorks default options.



8 **Align mesh.**
Flip the mesh so that the bottom of the shell mesh faces the inside of the pulley model. Make sure that the mesh remains aligned, i.e. colors on both the inside and outside must remain uniform.

The resulting mesh can be seen in the following figure.

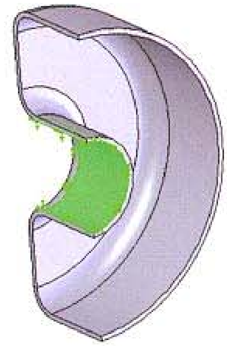


Note

Provided that the mesh is aligned, there is no need to flip the mesh. In our case, we flipped the entire mesh so that the bottom of the shell mesh coincides with the inside of the pulley. The postprocessing then becomes more intuitive.

9 Apply fixed restraint.
Select the face on the inside semi-cylindrical face and apply a **Fixed** restraint.

Although the restraint is applied to a face, it is transferred to the shell element nodes.

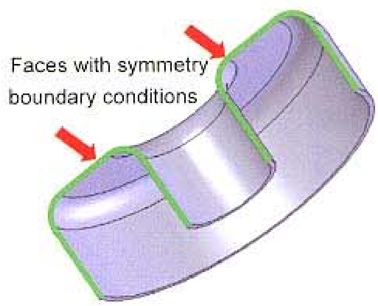


Note

Because the selected face is cylindrical, the default choice of restraint is **On cylindrical face**. Do not use this choice.

Also note that **Fixed** and **Immovable** restraints are not the same restraint conditions as they were in previous lessons. We are working with shell elements, which have six degrees of freedom and need to differentiate between **Fixed** and **Immovable** types of restraints.

10 Apply symmetry restraint.
Select the two faces in the plane of symmetry, and apply a **Symmetry** restraint.



Symmetry Restraints

This restraint simulates the half of the pulley that is missing from the model.

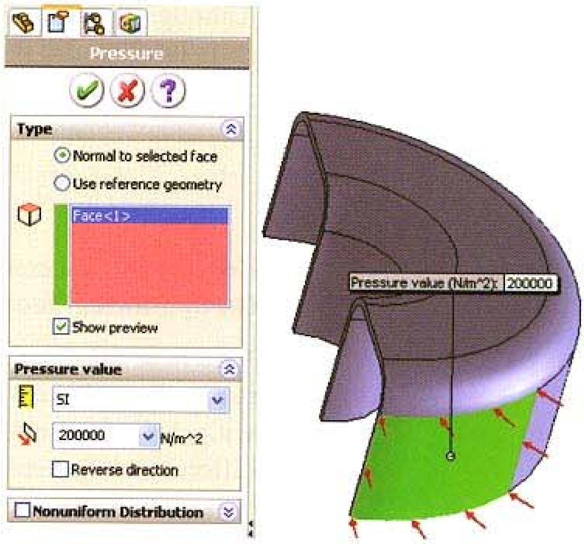
The restraint is transferred to the edge of the mid-surface that is meshed with shell elements.

Alternatively, rather that applying a **Symmetry** restraint to a face, we can apply the same condition manually to either one of the two edges of this face. The restraint will still be transferred to the edge of the surface that is meshed with shell elements. See the discussion in the second part of this lesson for more information on symmetry restraints.

11 Define pressure load.

Right-click `Load/Restraint` and select **Pressure**, and then select **Normal to selected face**.

Select the face upon which the pressure load needs to be applied, and define a pressure load equal to **200,000 N/m² (Pa)**.



Click **OK** to save the definition.

Note

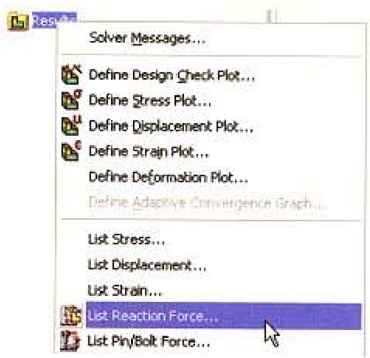
Notice that we do not load the model by the forces in the belt. Rather, we apply a pressure value simulating the presence of the belt.

Also, you may ask how we know that this 200,000 Pa pressure results in the desired 500 N reaction force in the vertical direction. A linear static analysis with an arbitrary magnitude of the pressure was run ahead of time. Based on the reaction force magnitude obtained in this study, we were able to scale the pressure to 200,000 Pa to obtain the 500 N vertical reaction. We already used a similar proportionality principle in Lesson 3.

12 Run the analysis.

13 List reaction forces.

Right-click the `Results` folder and select **List Reaction Force**.



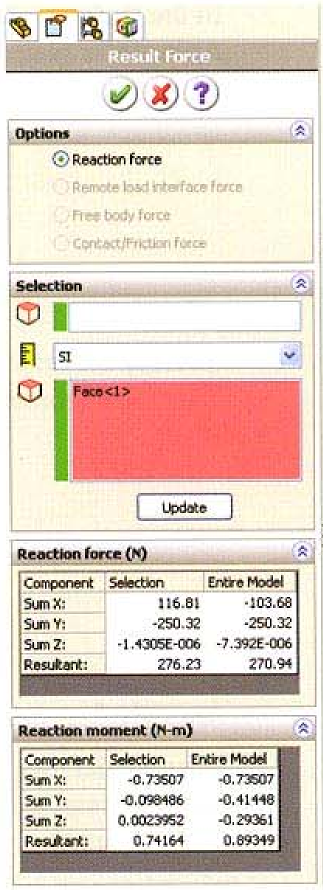
Select the supported face.

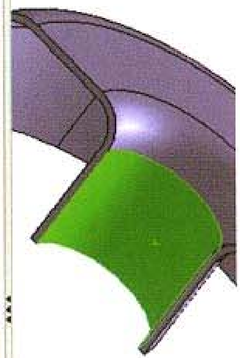
Set the **Units** to **SI**.

The **Reaction force (N)** and **Reaction moment (N-m)** dialogs show the reaction resultants for the selected face in the global cartesian coordinate system.

We observe, for example, that the global y and x components of the reaction force resultant are approximately 250 N and 116 N, respectively.

The reaction moments resultants about the x and y axes are -0.73 N-m and -0.1 N-m, respectively.





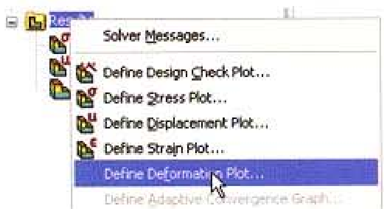
Note that 250 N vertical resultant force for half of the pulley corresponds to the 500 N vertical reaction for the entire model.

The moment reactions reflect the fact that the pulley or the load is asymmetrical with respect to the xy and xz planes. As expected, the resultant reaction moment about the global z-axis is nearly zero, i.e. the pulley as well as the load are symmetrical with respect to the yz plane.

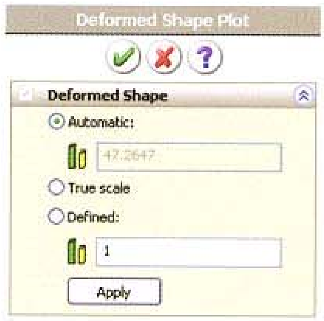
14 Animate deformation plot.

To verify that the symmetry boundary conditions are correct, animate the deformed geometry plot.

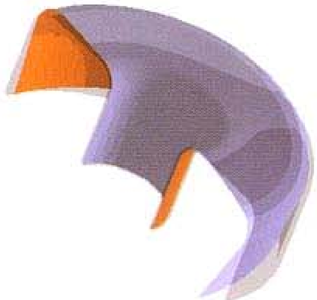
Right-click on the `Results` folder and select **Define deformation plot**.



In the **Deformed Shape** dialog, select **Automatic** scale.



Overlay the undeformed shape over the deformed geometry.



Animate the plot.

Observe that the edge where the symmetry boundary conditions were applied remains in its original plane (remains flat) while the pulley is deforming.

The best way to report such results is to save the animation file in AVI format.

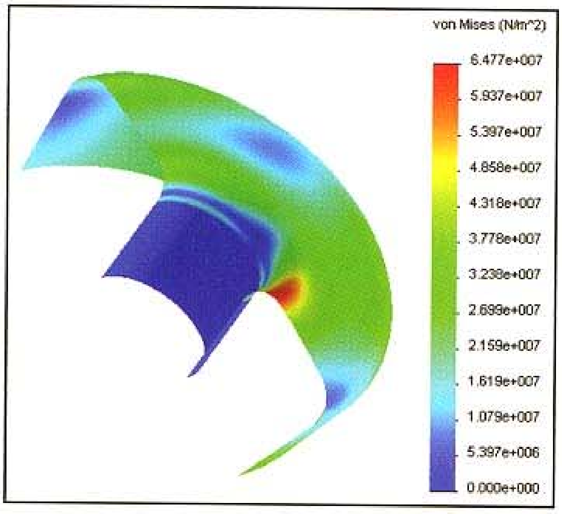
Deformation plot

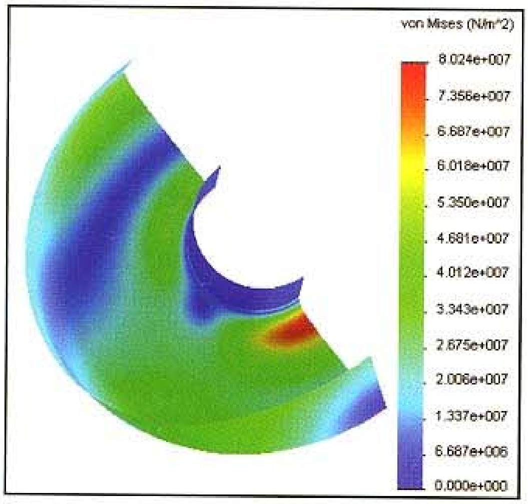
Deformation plot displays the deformed geometry using SolidWorks model colors. No contours, such as displacements or stresses, are plotted.

15 Plot von Mises stresses.

Display the distribution of the von Mises stress. Edit the definition of the plot to make sure that the results correspond to the **top** face of the shell mesh.

Analogously, define a new plot for the distribution of the von Mises stresses on the **bottom** of the shell mesh.





We observe that the maximum von Mises stresses on the top and bottom of the shell mesh are 64.7 MPa and 80 MPa, respectively.

Shell mesh using surfaces

If the geometry is rather complicated, the automatic extraction of the midplane may not be possible. In such instances, the shell sheets must be defined manually. This procedure will be demonstrated in the second part of this lesson.

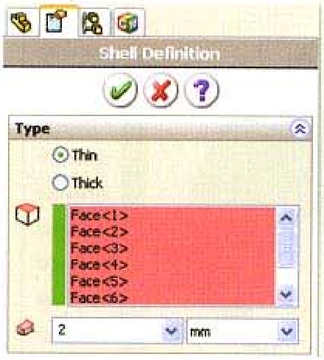
16 Define new study.

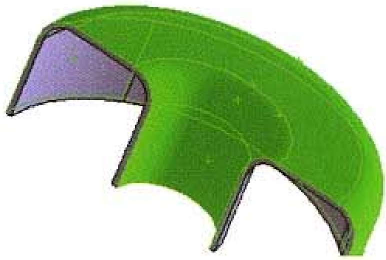
Define a new Static study named `Pulley shells-surfaces`. Specify **Shell mesh using surfaces** as a **Mesh type**.

Note that the study folder now contains a new folder named `Shells` (the `Solids` folder is not present).

17 Define shell sheets.

Right-click on the `Shells` folder and select **Define By Selected Surfaces**.





Select all the outside surfaces of the pulley.

Specify the **Thickness** of **2 mm** and **Thin** shell formulation.

Note

When using the **Shell mesh using surfaces** mesh type option, the thickness must be specified manually.

Also note that, as soon as the sheets are defined, the material information (AISI 1020) is directly transferred from SolidWorks.

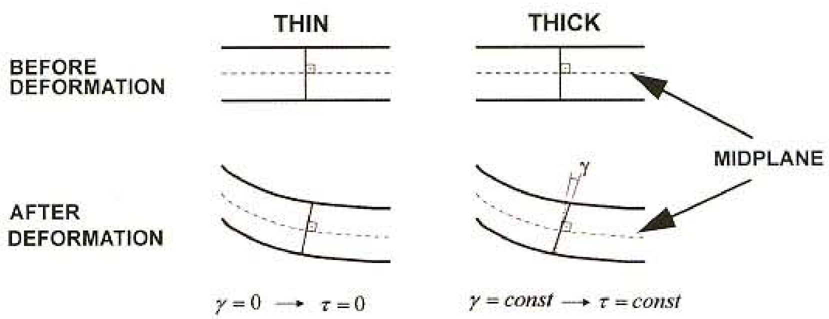
A `Shell-1` icon is created in the `Shells` folder. `Shell-1` groups all the faces that will be meshed with thin shell elements 2 mm in thickness.

Note

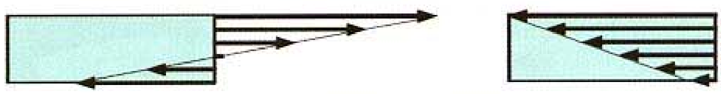
We could also create `Shell-2`, `Shell-3`, and so on, with different thicknesses and different shell element definitions (thin or thick) if applicable.

Thin vs. Thick Shells

- **Thin shell element** technology assumes that the cross-section perpendicular to the midplane remains straight and also perpendicular to the midplane at the end of the deformation (Kirchhoff theory). As a consequence, this shell element ignores the shear deformation and stress in the through-thickness direction. Thin, membrane-like structures with the span to thickness ratio larger than 20 can be accurately modeled using this element.
- **Thick shell element** technology assumes that the cross-section perpendicular to the midplane remains straight after the deformation takes place, but it is no longer perpendicular to the deformed midplane (Mindlin theory). As a consequence, this element assumes constant distribution of the shear deformation in the through-thickness direction. Thicker shells where shear effects become noticeable can be accurately modeled using this element.



In both cases the distribution of the normal bending stresses can be seen as linear.

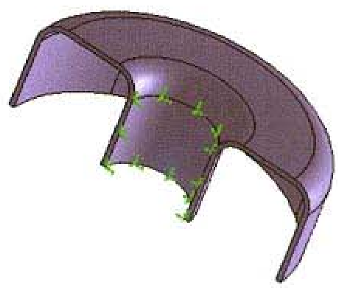


LINEAR DISTRIBUTION OF IN-PLANE STRESS IN BOTH THIN AND THICK SHELL

---

18 Apply fixed restraint.

Apply a fixed restraint to the semi-cylindrical face, as indicated in the figure.

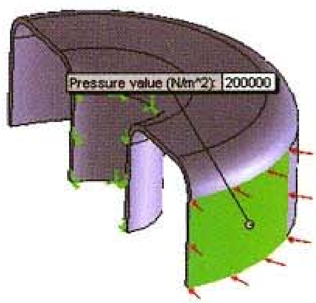


Note

The choice of whether to apply this restraint on the inside or outside face is no longer arbitrary. Since the shell sheets have been defined using the outside faces, we must remain consistent and apply the boundary condition on the outside face as well.

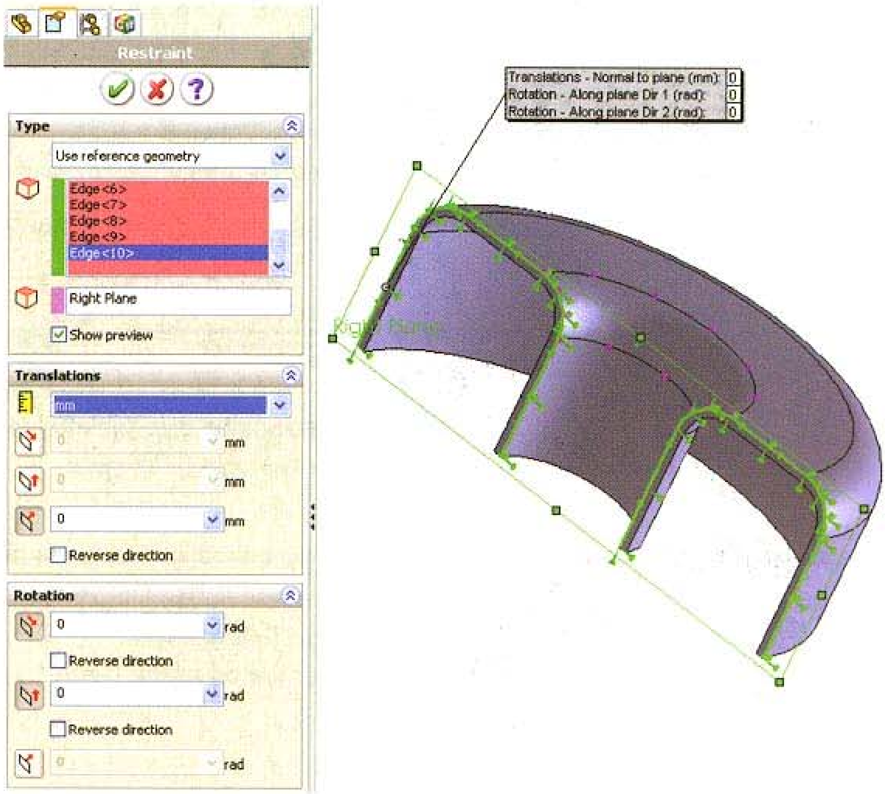
19 Apply pressure.

Apply Normal to selected face pressure to the face indicated in the figure. Specify the magnitude of 200,000 PA.



20 Apply symmetry restraint.

In the case of the **Shell mesh using surfaces** mesh type, the symmetry condition must be specified manually.



Apply **Restraint**, **Use reference geometry** type.

Select all the outside edges located on the symmetry plane.

Select `Right plane` as your reference entity.

Enter **0 mm** in the **Normal to selected plane** field.

Enter **0 rad** in both **Along plane Dir 1** and **Along plane Dir2** fields.

Click **OK** to save this restraint.

Applying Symmetry Restraints

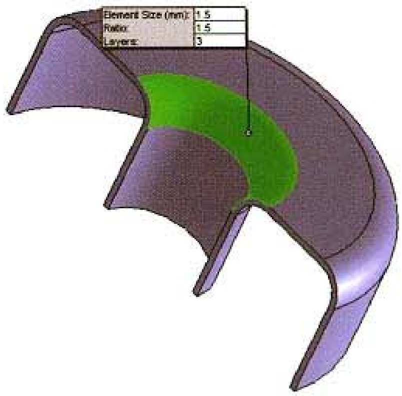
To illustrate a rule for applying symmetry boundary conditions, consider a point on the plane of symmetry. Any displacement that moves the point out of this plane must be restricted. Furthermore, any rotation that inclines the plane of the cut from the plane of symmetry must be restricted as well.

The following table summarizes symmetry boundary conditions in the three principal planes.

	Symmetry Boundary Conditions Plane of Symmetry		
	xy	*yz*	*xz*
***x* translation**	free	constrained	free
***y* translation**	free	free	constrained
***z* translation**	constrained	free	free
***x* rotation**	constrained	free	constrained
***y* rotation**	constrained	constrained	free
***z* rotation**	free	constrained	constrained

21 Apply mesh control.

Apply a mesh control with a default **Element size** of **1.5 mm** to the rounded face, as indicated in the figure. Keep the **Ratio** and the **Tolerance** at their default values of **1.5** and **3**, respectively.

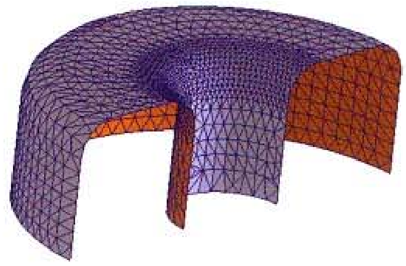


22 Mesh the model.

Mesh the model with **High** quality elements and the global **Element size** of **4.5 mm**.

Make sure that the shell mesh is aligned. Note that the orange color identifies the bottom of the mesh. To be consistent with the first part of this lesson, orient the mesh so that the bottom of the shell mesh coincides with the inside of the solid pulley.

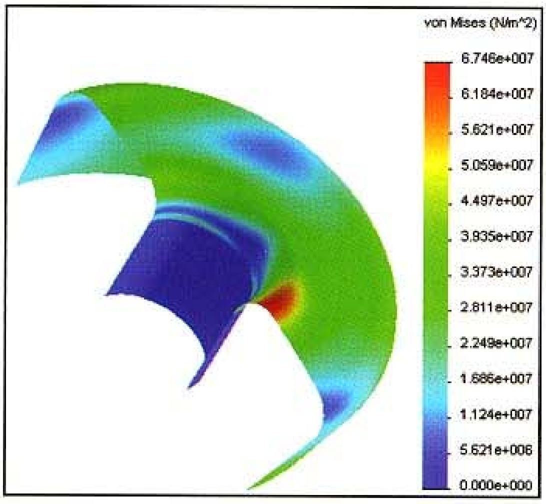
The resulting mesh can be seen in the figure to the right.

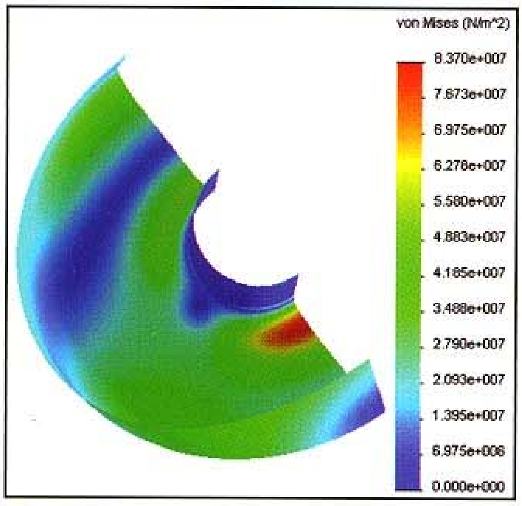


23 Run the analysis.

24 Plot von Mises stress.

Plot the von Mises stresses on both the bottom and the top of the shell mesh.





We observe that the maximum von Mises stresses on the **top** and **bottom** are **67.5 MPa** and **83.7 MPa**, respectively. This translates to a difference of 4% and 4.2% when compared to the first part of this lesson, where the mesh was located exactly at the midplane.

In many applications, the midplane extraction is not a trivial process. Given the fact that shell mesh is applicable to thin sheet-like structures, the above modeling error is rather small and commonly acceptable.

Solid vs. shell elements

The first two parts of this lesson introduced us to shell elements. Having investigated shell specific modeling techniques and shell element properties, we will now compare the results with those produced by using solid elements.

Why? All solid bodies can be meshed using solid elements. In the following section of this lesson we, therefore, demonstrate why the use of shell elements is beneficial.

Shell Element vs. Solid Element Modeling

25 Create new study.
Create a new static study named `pulley solids` and select **Solid Mesh** as the **Mesh type**.

Note

You cannot copy study `pulley shells` into `pulley solids` because they are of different types. You can, however, copy the `Load/Restraint` folder from one study to another.

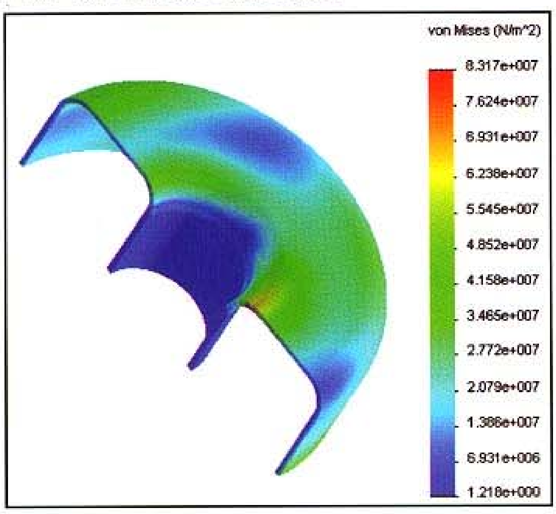
26 Create mesh.
Mesh the model with **High** quality elements and the default global **Element size** of **3mm**.

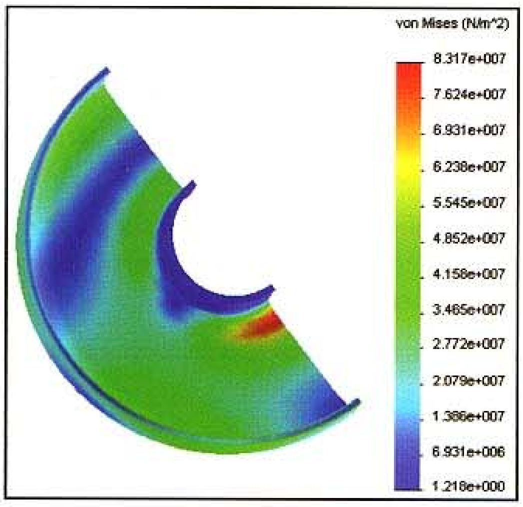
27 Run the analysis.

Note

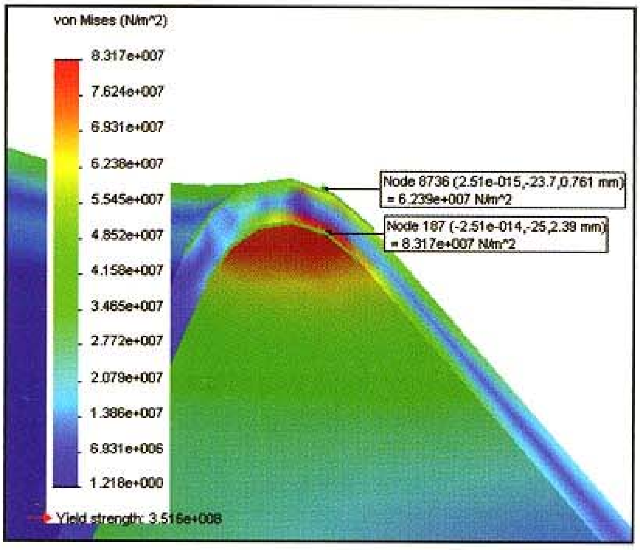
You can merge these two steps, mesh and run, if you select the **Run analysis after meshing** check box in the **Mesh** PropertyManager.

28 Plot von Mises stresses.





The maximum stresses on the outside and inside faces of the solid pulley are approximately 67 MPa and 83.2 MPA, which compares quite well to the solutions obtained from the shell studies.



Note, however, that this mesh features one element through the thickness while two layers of high quality elements are recommended as a minimum.

Refined solid mesh

29 Create new study.
Create a copy of the `pulley solids` study and name the new study `pulley solids dense`.

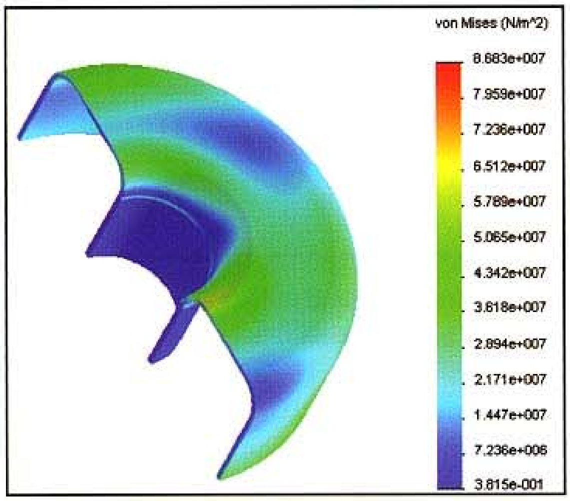
30 Mesh the model.

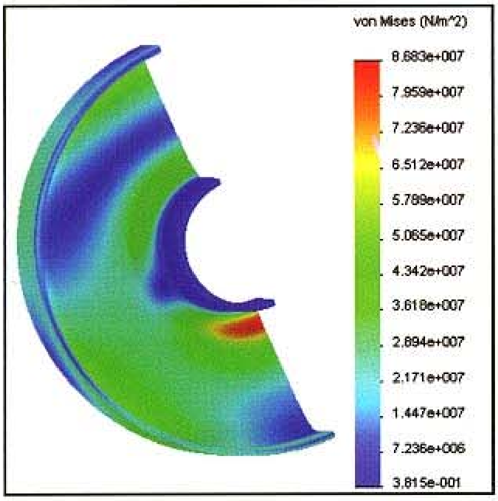
Mesh the model with an **Element size** of **1 mm**. This element size ensures that two layers of elements are placed across the wall thickness, which is 2 mm.

31 Run the analysis.

32 Plot von Mises stresses.

Create a von Mises stress plot.





The maximum von Mises stresses on the outside and inside are 61 MPa and 86.8 MPa. The dense solid element mesh reports a more "regular" shape of the stress concentration.

Results Comparison

The following table compares the displacement and stress results from the four studies that we solved in this lesson.

Study	Displacement [mm]	von Mises Stress [MPa]	D.O.F.
`pulley shells - midplane`	0.306 (bottom)	80 (bottom)	14,538
`pulley shells - surface`	0.338 (bottom)	83.7 (bottom)	25,398
`pulley solids`	0.315	83.2	56,259
`pulley solids dense`	0.318	86.8	878,652

Computational Effort

The summary table also lists the number of degrees of freedom in each model. While the model is open, locate the file with the OUT extension in the COSMOSWorks database to view this information. The number of degrees of freedom can be used as a measure of computational effort required to obtain the solution. Lower computational effort is directly related to the time and cost required to run a study.

Note that the models produce practically identical displacement results.

Compare the number of DOF for the `pulley solids` and `pulley solids dense` studies to see that the dense model is 14 times larger. The stress results of both solid models are within 5%, demonstrating that the model with two layers of elements is not really necessary in this case.

The reason for the 8% maximum difference in the stress results between shell and solid models is that shell elements can not account for the shift of the neutral bending layer towards the inside of the pulley curvature.

We must then conclude that solid elements provide more accurate results when analyzing models with highly curved walls in bending.

Summary

The pulley lesson introduces us to shell elements and familiarizes us with concepts such as shell element thickness and orientation.

Both the **Shell mesh using mid-surfaces** and **Shell mesh using surfaces** modeling techniques were introduced and used to build the finite element model.

Symmetry boundary conditions were used in both modeling techniques. While the convenient **Symmetry** restraint type was used in the case of **Shell mesh using mid-surfaces**, it is not applicable for **Shell mesh using surfaces**. Manual application of this restraint was introduced and practiced as well.

The concept of mesh adequacy was also addressed, and the results and modeling differences between shell and solid element models were discussed. It was concluded that solid elements may produce results that are slightly more accurate than those produced by shell elements, provided the solid mesh is sufficiently fine. This can, however, lead to a substantial increase in the problem size which may become intractable.

Exercise 5: Shell Analysis of a Bracket Using Selected Surfaces

Project Description

A sheet metal bracket has been designed to support a side load of 100 lb (load applied in the x-direction of the global coordinate system).

We consider two design configurations:

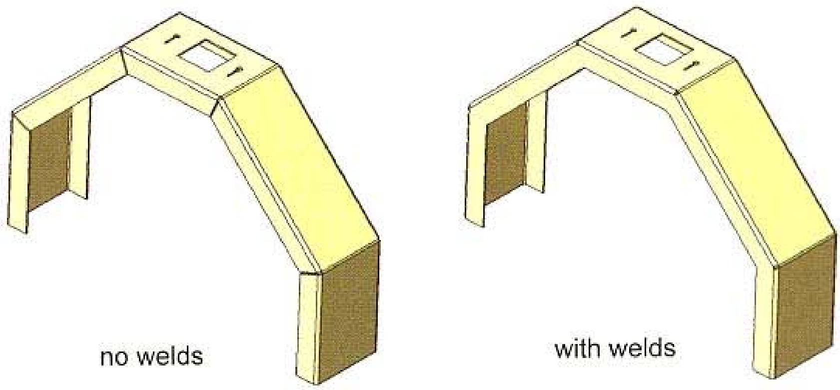
- One without any welds
- One with welds connecting the mitre flanges

We compare the maximum displacements and maximum von Mises stresses for these two configurations.

1 **Open part.**
In SolidWorks, open the part file `horseshoe` and examine the two configurations:

- `no welds`
- `with welds`

In the `with welds` configuration, a total of eight extrusions have been added to connect the mitre flanges.



Analysis of Bracket With No Welds

The geometry lends itself to meshing with shell elements, but is too complex for the **Shell mesh using mid-surfaces** type of mesh. You can try creating a shell element mesh using mid-surfaces to see why this method does not work. To create a finite element mesh, we must use the **Shell mesh using surfaces** method of creating a shell element mesh.

2 **Activate the `no welds` configuration.**

3 **Create study.**
Create a study named `no welds analysis`.

Select **Static analysis** as **Analysis type** and **Shell mesh using surfaces** as the **Mesh type**.

4 Define shell sheets.
Right-click `Shells` and select **Define By Selected Surfaces**.

Select all the exterior faces.

Enter **0.12 in.** as the shell element thickness.

Select **Thin** as the shell element type.

Click **OK** to accept your selections.

5 Define material properties.
Assign **Galvanized Steel** to the faces in the `Shell-1` group.

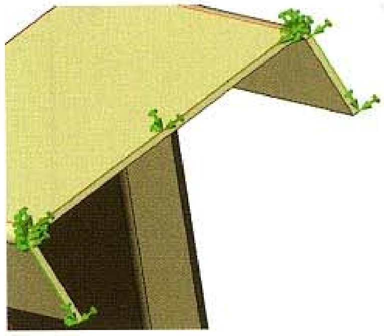
6 Create mesh.
Specify **High** quality elements and accept the default **Element size** of **0.538 in**.

Make sure the **Automatic Transition** option is cleared in **Mesh Options**.

Note

Shell elements should be consistently aligned to ensure correct stress averaging along the boundaries' separating faces.

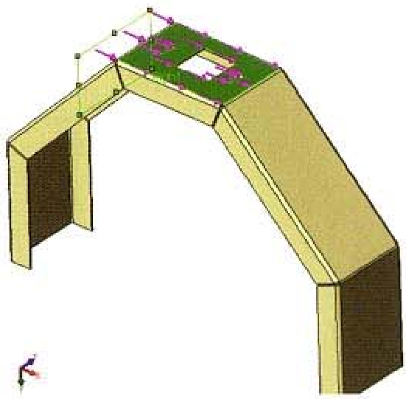
7 **Apply fixed restraints.**
Select the outside edges of the end faces on both sides of the bracket and apply a **Fixed** restraint.



Note

Having created a shell element mesh on the outside faces of the bracket, it becomes clear why restraints must be applied to the outside edges. If we apply restraints to the entire end faces, the restraints are not transferred from the geometry to the shell elements, as there is no mesh on these faces.

8 **Apply 100 lb force.**
Apply a load to the top face of the model (`Base-Flange3`) as indicated in the figure.



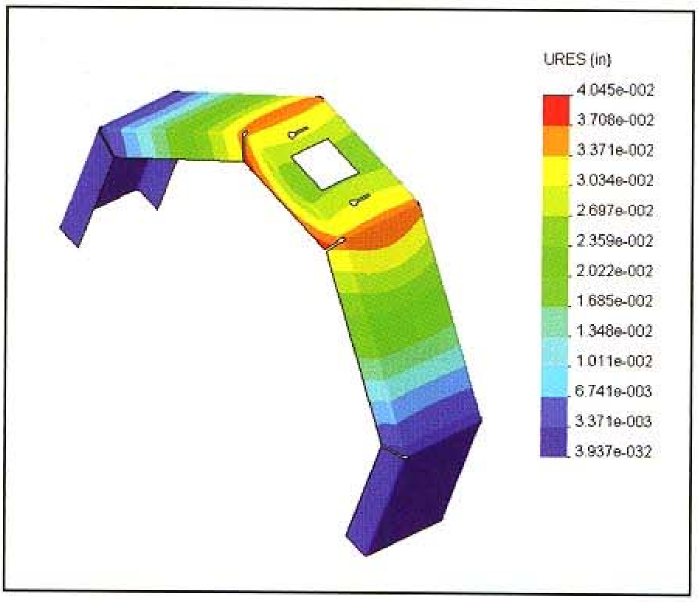
Moment Load

The **Force** definition window also allows you to apply a load in the form of moment.

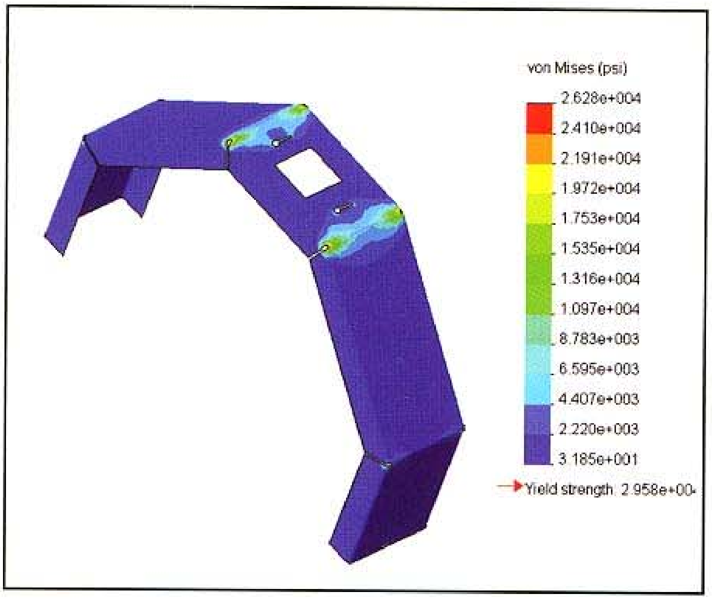
This option is possible because shell elements have six degrees of freedom (three translational and three rotational) and, thus, can be loaded with either a force or a moment.

9 **Run the analysis.**

10 Plot resultant displacements and von Mises stresses.



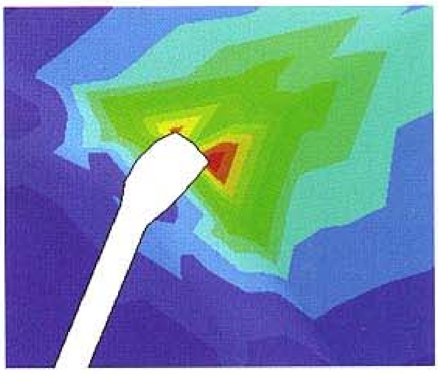
Use a **True scale** deformation factor.



The highest stresses in the configuration without welds are located at the sharp re-entrant edges of the sheet metal bracket.

Note

As you recall from Lesson 2 (`L bracket`), the numerical values of these stress results are meaningless, as they are singular at those locations.



Analysis With Welds

11 Activate the `with welds` configuration.

12 Create new study.
Create a study named `with welds analysis` using the same study options as in the previous analysis.

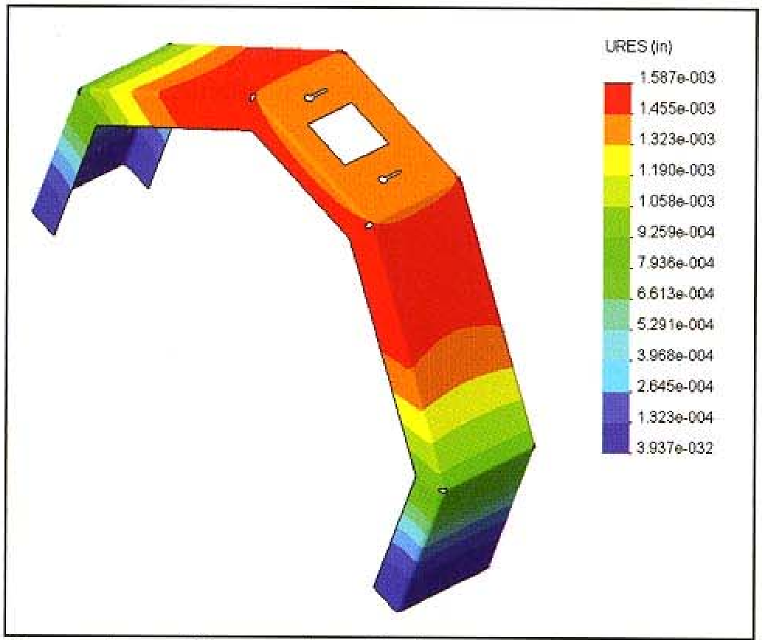
13 Copy loads and restraints.
Even though the study `no weld analysis` is inactive, you can copy loads and restraints.

14 Define shell sheets.
Define all of the external faces as sheets. Specify **0.12 in** as the **Shell thickness** and **Thin** shell technology.

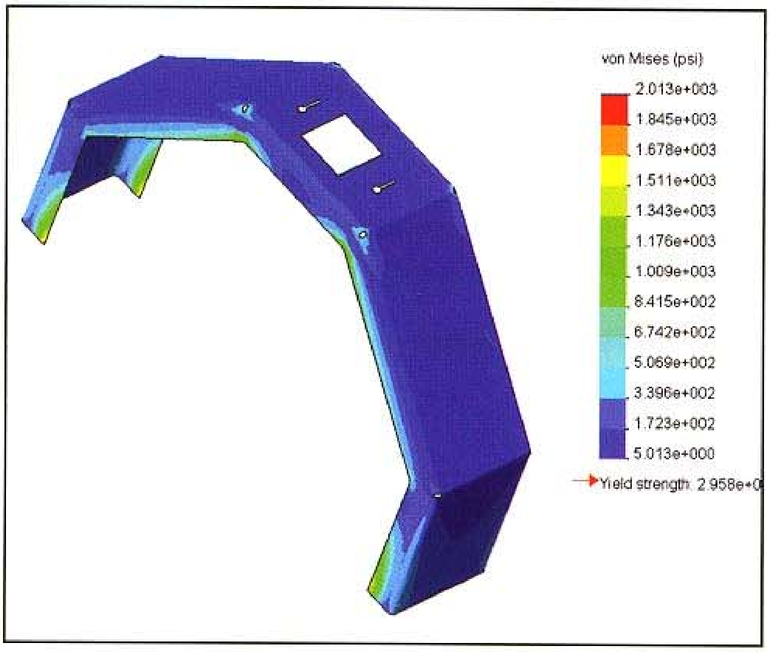
15 Mesh the model and align shell elements.
Use **High** quality elements with the default **Element size**.

16 Run the analysis.

17 Plot resultant displacements and von Mises stresses.



Comparison of the resultant displacements between the model without welds and the model with welds shows a reduction of the maximum displacement from 0.04 in. to 0.0016 in.



The maximum von Mises stress results are not easily comparable because of stress singularity in the model without welds. We can, however, observe that the maximum von Mises stress magnitude is 2,013 psi, which is below the yield stress of the galvanized steel (29,579 psi).

18 Save and close the model.

Lesson 7
Connectors, Special Supports and Contacts

Objectives

Upon successful completion of this lesson, you will be able to:

- Perform analysis on different configurations of a SolidWorks model using COSMOSWorks
- Use pin connectors and hinges
- Use virtual wall local contact condition
- Use bolt connectors
- Use spring connectors
- Use spot weld connectors
- Apply restraints in a local coordinate system
- Analyze results in a local coordinate system

Connectors

Mate definitions in the SolidWorks assembly do not translate into contact definitions in COSMOSWorks. Therefore, from the point of view of COSMOSWorks, the components of the assemblies are unattached until we define the proper contact conditions or connectors describing interactions between the assembly components.

The main purpose of this lesson is to practice the various connectors available in COSMOSWorks.

Note

Depending on the type of connector used, certain contact conditions have to be defined between connected components.

The following table lists the available connector options:

Connector Type	Definition
Rigid	Defines a rigid link between the selected faces. The connected faces do not deform.
Spring	Connects a face on a component (or solid body) to a face on another component (or solid body) by distributed springs with the specified normal and shear stiffness. The stiffness values may be entered as distributed or total values. The two faces must be either planar and parallel to each other, or cylindrical and coaxial. You can specify a pre-load for the spring connector. The following types are available: **Compression Extension** **Compression only** **Extension only**
Pin	Connects cylindrical faces of two components. The following two options are available in the pin definition: 1. **No Translation**. Specifies a pin that prevents relative axial translation between the two cylindrical faces. 2. **No Rotation**. Specifies a pin that prevents relative rotation between the two cylindrical faces. The stiffness values corresponding to the axial and rotational directions may be specified as well.

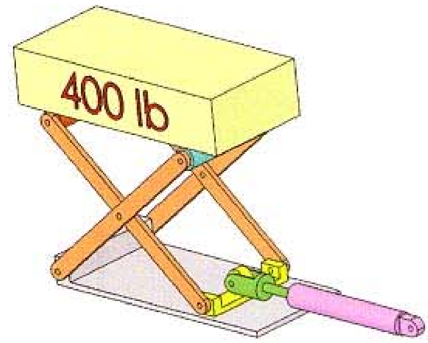
Connector Type	Definition
Elastic support	Defines an elastic foundation between the selected faces of a part or assembly and the ground. The faces do not have to be planar. A distributed stiffness at a point on the face represents the stiffness density associated with an infinitely small area around the point. The tangential and normal stiffness components are assumed constant and directed in the directions tangential and normal to the face at every point.
Bolt	Defines a bolt connector between two components or between a component and the ground. Bolts both with and without nuts are supported. Material specifications directly from the material libraries and various preload options are available.
Spot weld	Defines a connector simulating spot weld between two solid faces or two shell faces.
Link	Ties any two locations on the model by a rigid bar that is hinged at both ends. The distance between the two locations remains unchanged during deformation. Link does not restrict rotations at both ends.

Project Description - Hinges, Virtual wall, and Elastic support

A scissor lift used to lift a 400 lb. weight is operated by an external hydraulic cylinder connected to a slider traveling on a base.

The load is assumed to be evenly distributed between the two rollers which, in turn, evenly split the load between the arms. This way each arm is loaded with a 100 lb. force.

We would like to find the displacements and stresses in the lift components at the collapsed position of the lift arms. We are not interested in contact stresses in the pin joints, nor the stresses in the base.



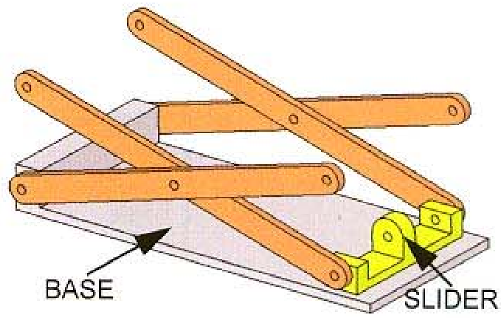
1 **Open assembly.**
Open the assembly model `lift` and familiarize yourself with the `collapsed` and `extended` configurations of this assembly.

The goal of this analysis is to analyze the assembly in the collapsed configuration.

2 **Activate the configuration `collapsed`.**

Note

Note that the weight, hydraulic cylinder, connecting pins, and many other details are not modeled, and the SolidWorks assembly `lift` depicts the scissor lift in a somewhat idealized way.



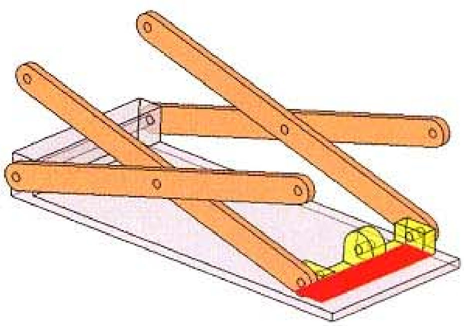
3 **Create study.**
Create a study named `collapsed`. (**Static** analysis, **Solid** mesh.)

4 **Assign material properties.**
Specify **Plain Carbon Steel** for all of the components.

5 **Check for assembly interferences.**
Under **Tools**, select **Interference Detection**.

Activate the **Treat coincidence as interference** option.

Click **Calculate**.



We observe that there are the only two parts in the assembly with touching faces.

Note

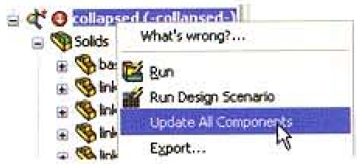
Since we are not interested in the deformations and stresses in the base, we will suppress this part to simplify our mesh. At the same time, however, we must correctly represent the contact condition with the corresponding friction forces. This can be achieved by using a **Virtual wall** contact condition type, described in the discussion that follows.

6 **Suppress the `base` part.**

7 **Update all components.**

Because the `base` part was suppressed after the study `collapsed` was defined, we have to update the study components.

Right-click on the study `collapsed` and select **Update All Components**.

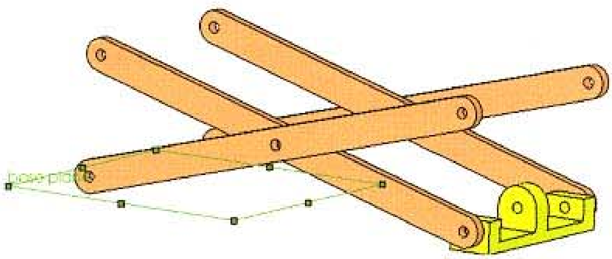


Note

Note that the component `base` is now show in the color gray or not present at all, identifying it as inactive.

Virtual Wall

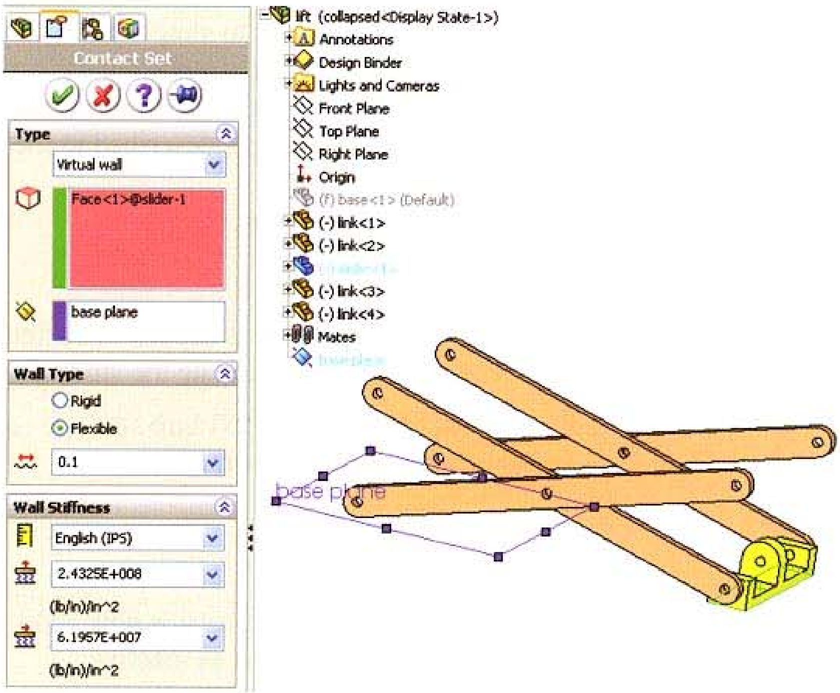
In general, the **Virtual wall** contact condition is specified between a face/edge/vertex on a source part and a target reference plane commonly representing the surface of the suppressed part. In our case, this contact exists between the bottom face of the slider and the base plane (modeling the top surface of the suppressed base).



It is possible to specify whether the target plane represents a rigid or flexible object. If the latter option is selected, both the axial and the transverse stiffness values must be specified. For information on how to correctly calculate these values, consult the knowledge base article titled Axial and tangential stiffness for virtual wall and drop test.

8 Define Virtual wall.

Right-click on the `Contact/Gaps` folder and select **Define Contact Set**.



In the **Type** dialog, specify **Virtual wall**.

Select the bottom face on the `slider` as a **Source** and the `base plane` as a **Target**.

Specify the **Friction Coefficient** of **0.1**. Under **Wall Type**, select **Flexible**.

Enter the values of **2.4325e8 lb/in/in^2** and **6.1958e7 lb/in/in** as **Axial stiffness** and **Tangential stiffness**, respectively.

Click **OK** to save the virtual wall settings.

Hinge Restraint

The connection between the lift arms and the base has to be simulated as hinges.

The **Hinge** type of restraint suppresses radial and axial translations, which are defined in the local cylindrical coordinate system associated with the cylindrical surface.

Exactly the same restraint can be defined using the **On cylindrical face** type of restraint where we restrain the radial and axial displacement components.

Note

Using the **Hinge** restraint, we assume that the base is rather stiff and does not deform. If the elastic behavior of the base must be accounted for, it would have to be included in the analysis.

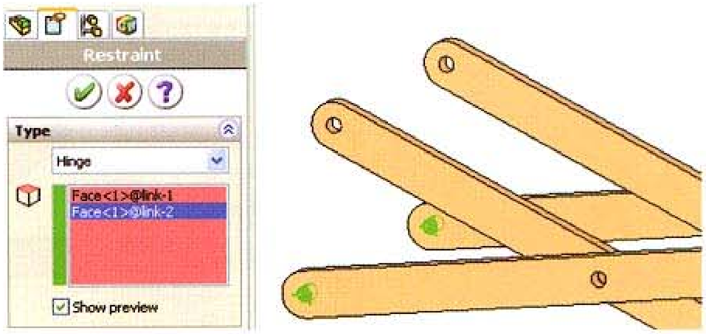
---

9 Define hinge restraint.

Under the `Load/Restraint` folder, select **Restraints** and then select **Hinge**.

Select the two cylindrical faces initially connected to the base.

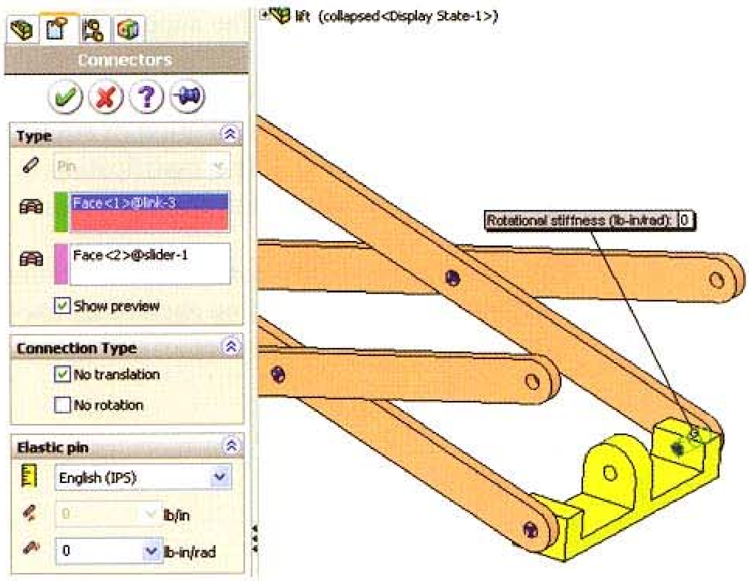
Click **OK**.



10 Define Pin connectors.

Define two rigid **Pin** connectors between the four arms, and two rigid **Pin** connectors between the `arm` and `slider`.

For all the pins, allow the relative rotation and restrain the relative translation between the connected components. For rigid pins, keep both of the stiffness fields at zero.

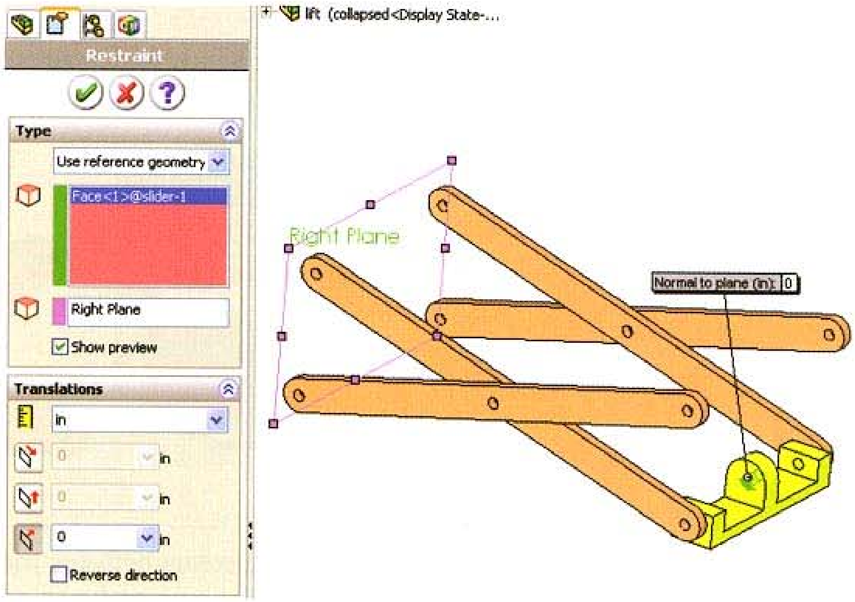


Note

For an explanation of the Pin connector, please refer to Lesson 4.

11 Define restraint on `slider` cylindrical face.

To model the support provided by the hydraulic cylinder, select the cylindrical face on the cylinder lug of `slider` (`Cut-Extrude3`) and right-click `Load/Restraint` and select **Restraints**.



Select **Use reference geometry** and the `Right` plane as a reference. Restrain translations in the normal direction.

Click **OK**.

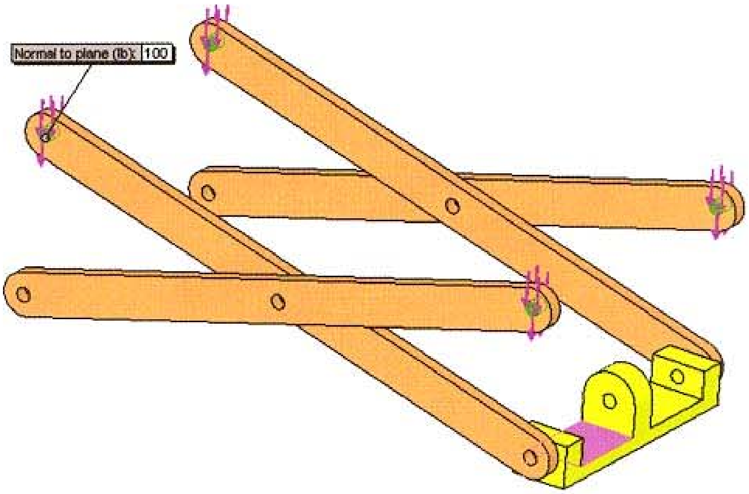
This definition simulates the support offered by the hydraulic cylinder.

Note

Note, however, that by applying restraints to the entire cylindrical face we ignore the realistic distribution of stresses between the cylinder pin and the lug. This modeling simplification is acceptable because we do not intend to investigate the contact stresses in the lug.

The model is now fully constrained even though the assembly components are not touching each other except for the one pair of faces with local **Node to node** contact conditions.

12 Apply force on `link` components.



Apply a downward force of **100 lb** to each of the four cylindrical openings at the free ends of the four `link` components. The total weight distributed equally between all four locations is thus 400 lb.

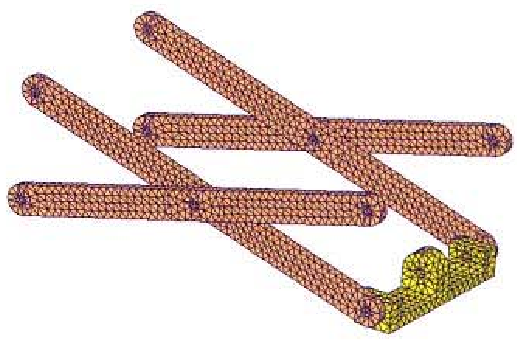
Bearing Load

Applying the load to the entire cylindrical hole is an acceptable simplification because we do not intend to analyze contact stresses developing between the arms and roller pins.

Note that there is a more accurate way to apply load to a cylindrical hole that still does not require contact stress analysis. It is called a **Bearing Load**.

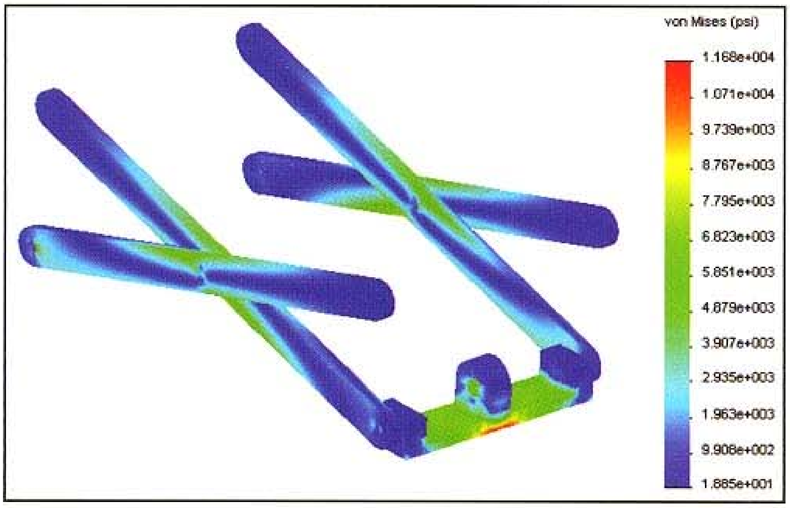
A load defined as a bearing load is applied to a portion of the cylindrical face (this requires splitting the face), and its variation is described by a cosine function to simulate the contact pressure distribution.

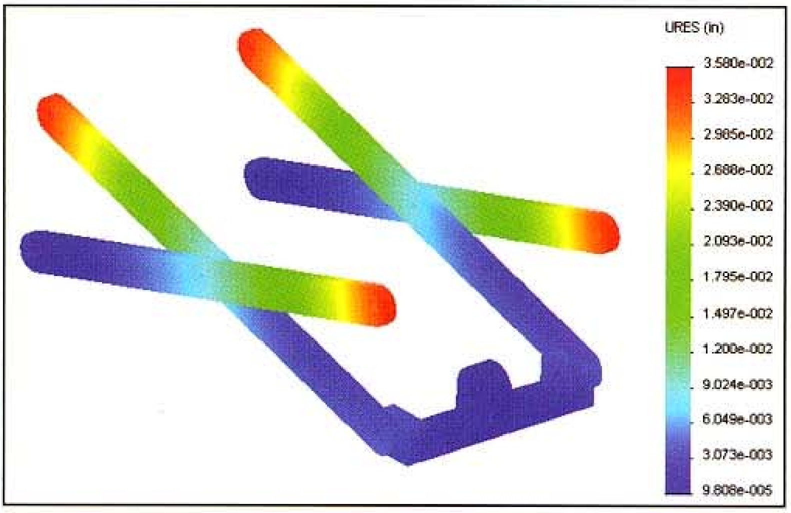
13 Create mesh.
Mesh the model with **High** quality elements and without **Automatic transition**. Use the default **Element size** and the **Tolerance** of **0.427 in** and **0.021 in**, respectively. The resulting mesh can be seen in the figure below.



14 Run the analysis.

15 Plot von Mises stress and resultant displacement.





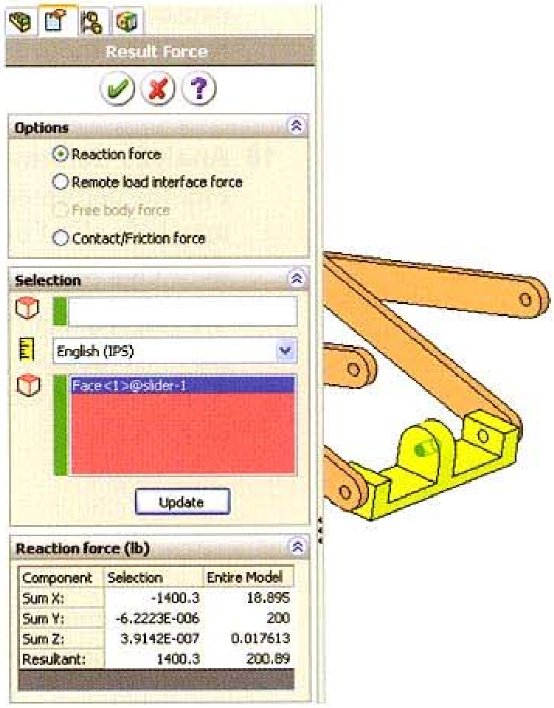
We observe that the model is not yielding and the resultant displacements are rather small.

16 List lug hole reaction force.

To determine the cylinder force required to maintain the equilibrium of the collapsed scissor lift under the 400 lb. weight, list the cylinder lug hole.

Right-click the `Results` folder and select **List Reaction Force**.

The reaction force in the *x*-direction, which is the direction of the hydraulic cylinder, is approximately 1,400 lb.



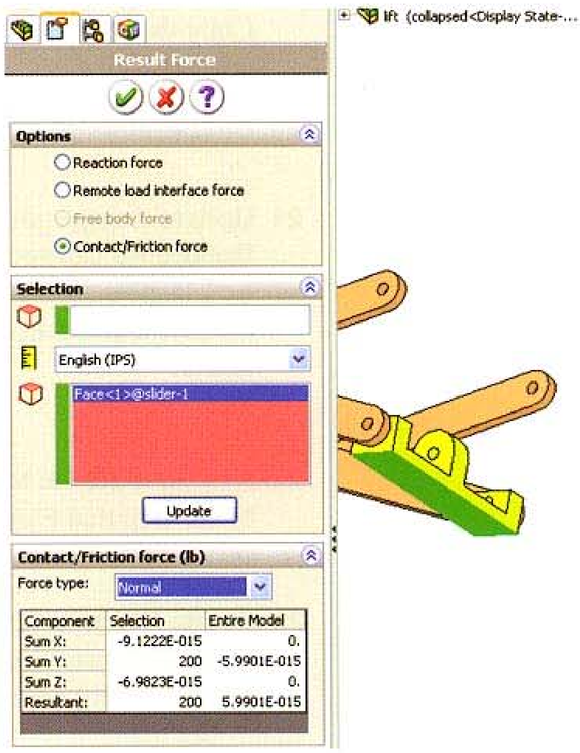
17 List contact and friction forces.

Right-click the `Results` folder and select **List Contact/Friction Force**.

Select the bottom face on the `slider` and click **Update**. In the **Contact/Friction** dialog, specify **Total**.

The **Normal force** (*y*-component) is equal to 200 lb., which amounts to half of the total load (the second half is carried by the two hinge restraints).

The **Friction force** (*x*-component) is equal to -18.9 lb. We can easily verify that this result is correct by calculating 200 x 0.1 = 20 lb.



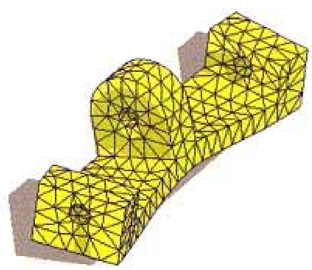
Note

The small difference between the theoretical and numerical values is caused by a rather coarse mesh at the contact interface. You may try to refine the mesh and observe the friction force approach the value of 20 lb.

18 Analyze deformation of `slider`.

Plot the deformed shape of the slider in highly magnified scale. (You may hide all of the remaining components in SolidWorks to view the deformed shape more clearly).

We can see that the middle part of the `slider` detaches from the base; the contact is provided only by very small areas on both sides.



Also note that, to accurately model the contact stresses, a highly refined mesh would be required.

Analysis with base (optional)

The results from the previous study can be verified by running the same analysis with the base included in the finite element model.

19 Create new study.

Copy the study `collapsed` into a new study named `collapsed with base`.

20 Unsuppress part `base`.

21 Update components.

Update All Components in the study `collapsed with base`. You should see the part `base` listed under the `Solids` folder.

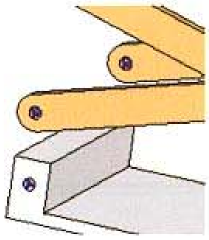
22 Assign material to part `base`.

Make sure that **Plain Carbon Steel** material is assigned to the part `base`.

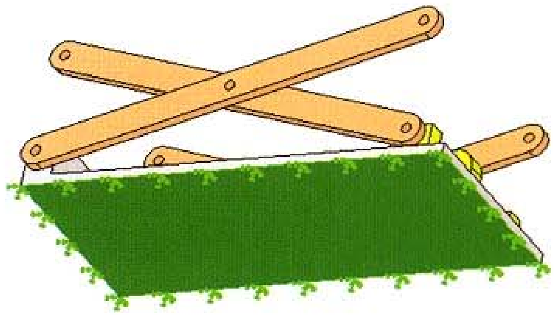
23 Delete Hinge restraints.

Since the elastic base is now included in the model, the hinge restraints are no longer needed. Delete them.

24 Define additional Pin connectors.
Define the **Pin** connectors between the `link` parts and the `base`. You may explode the view to ease the definition. These connectors are replacing the hinges deleted in the previous step.



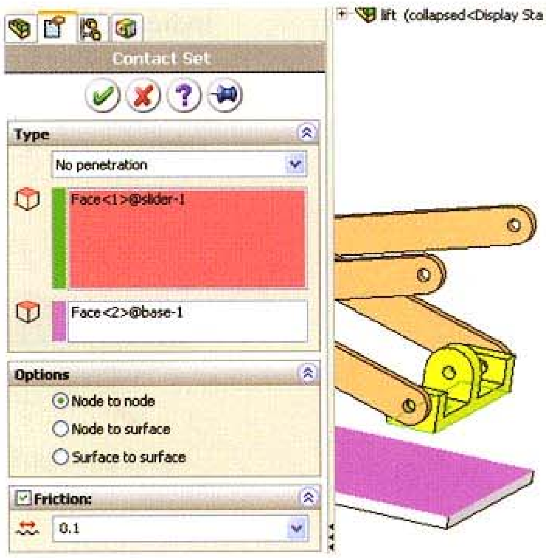
25 Define fixed restraint.
To define a support, on the bottom face of the `base`, create a **Fixed** (or **Immovable**) restraint.



26 Redefine the contact.
Change the contact set definition to **No penetration**, select **Node to Node** and set the **Friction Coefficient** of **0.1**.

Keep the bottom face of the `slider` as a **Source** and specify the top face of the `base` as a **Target**.

Click **OK**.



27 Create mesh.
Mesh the assembly with **High** quality elements and the default **Element size** of **0.6 in**. Do not use **Automatic transition**.

28 Run the analysis.

29 List lug hole force.
The reaction at the lug hole is 1400.2 lb., which agrees with our previous results.

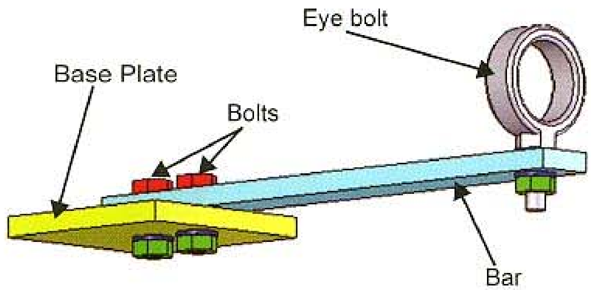
30 List contact and friction forces.

The normal and the friction forces at the `slider & base` interface are 200 and -18.9 lb., which also agree with the previous results.

31 Save and close the assembly model `lift`.

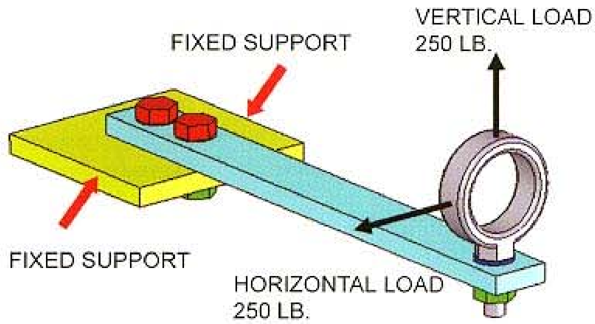
Problem Description - Using Bolt connectors

A `bar` is attached to a `base plate` with two loose fitting bolts: bolt diameter is 0.5", hole diameter is 0.5625".



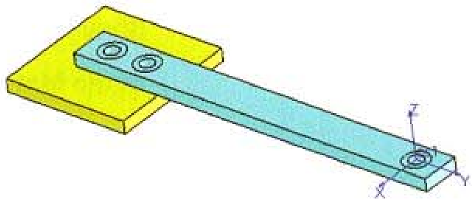
The `base plate` is supported along both sides. The `eye` bolt is loaded in vertical and horizontal directions with 250 lb forces, as indicated in the figure below. It is assumed that the `eye` is rather stiff and provides a nearly rigid connection between the forces and the strip.

Both the `bar` and the `base plate` are manufactured from steel AISI 1020.



1 **Open Assembly.**
Open the assembly model `bolt joints`.

Notice that the bolts, nuts and washers have been suppressed. This is because in this exercise we will use bolt connectors. To account for the missing eyebolt, we will apply the horizontal load as a remote load on the cantilever beam.



2 **Create Study.**
Create a study named `two bolts - torque preload` (**Static** analysis, **Solid mesh**).

3 **Show Exploded View.**

4 **Create Bolt Connector.**
In the exploded view, right-click `Load/Restraints` folder and select **Connectors**.

In the **Type** list, click **Bolt Connector**.

Under **Type**, select **With nut**.

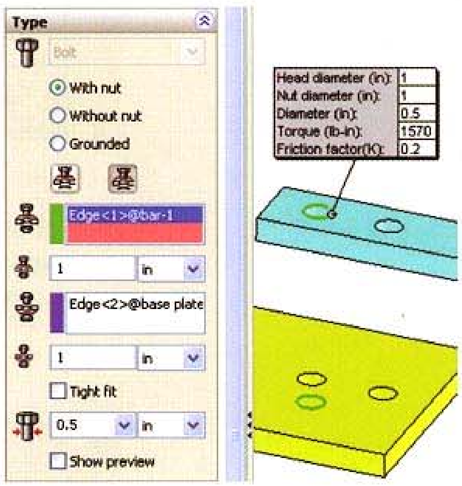
Activate the **Head/Nut diameter** option.

Select the Head and Nut diameters.

Clear the **Tight fit** check box.

Enter **1 in** for the head and nut diameters.

In the **Diameter** box, enter the value of **0.5 in**.



Under **Material**, select **Library**.

Click **Select Library**.

Select **Alloy Steel** from the `cosmos material` library.

In the **Material** window, click **OK**.

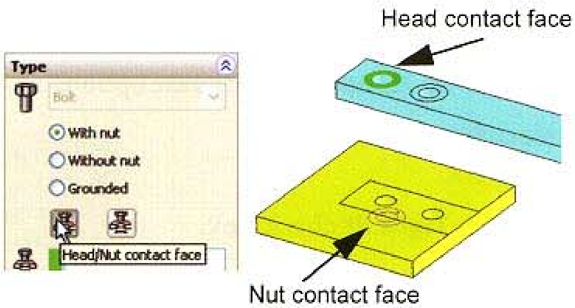
Under **Preload**, select **Torque**.

In the **Torque** box, enter a value of **1570** (lb.-in.). Specify the **Friction factor** of **0.2**.

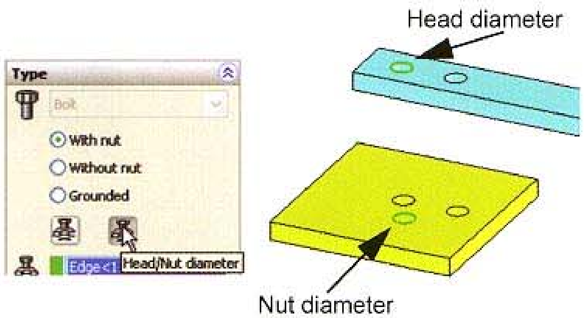
You can verify with hand calculations that the corresponding axial bolt force is 15,700 lb. Consequently, bolt tensile stress equals 79,960 psi which is 88% of the yield strength of **Alloy Steel**.

Bolt head and nut contact faces

The head and nut contact faces have to be included in the definition of the bolt connector. The contact faces can be generated manually in SolidWorks using split lines, in which case, the option **Head/Nut contact faces** would be activated.



If no contact faces are created in SolidWorks, COSMOSWorks will generate them automatically. Select the **Head/Nut diameter** option to activate this functionality.



Bolt Tight fit and Diameter

The **Tight fit** option controls not only whether the bolt shank is in direct contact with the hole, but also whether the walls of the bolt hole may deform or not.

- If the stiffness of the bolt material is significantly smaller than the stiffness of the material of the bolted components, the presence of a rather soft bolt shank will not have any substantial effect on the deformation of the hole walls. In such a case, the **Tight fit** option should be cleared.
- If the stiffness of the materials are comparable, or the stiffness of the bolt material is greater than the stiffness of the material of the bolted parts, the **Tight fit** option should be activated.
- If the diameter of the bolt is smaller than the diameter of the bolt hole, the **Tight fit** option should always be cleared. In this case, the stiffness characteristics of the materials are not important.

Bolt pre-load

Bolt pre-load can be defined directly by entering an axial force or indirectly as a torque. When the torque (T) value is entered, COSMOSWorks calculates the axial bolt force, which is the corresponding bolt pre-load, using the following formula:

$$F = \frac{T}{K \times D}$$

where D is the diameter of the bolt, and K is the friction factor (also commonly known as a torque coefficient).

The exact formula for the friction factor K is rather complicated and can be found in Mechanical Engineering Design by J.E. Shigley (1986). However, the value of $K = 0.2$ is a very good approximation for most practical cases.

---

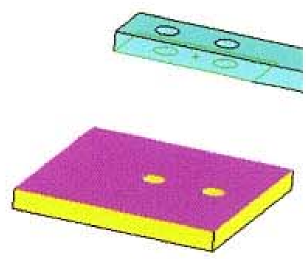
5 Define second bolt connector using aforementioned properties.

6 Define Contact conditions

For correct modeling of bolted connections we need to define a contact condition between the two assembly components.

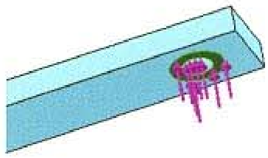
Because we expect a horizontal slide along the interface, a local **No penetration, Node to Surface** contact condition is required.

Define a **No penetration, Node to Surface** contact set between the two components, as indicated in the figure.



7 Apply vertical load.

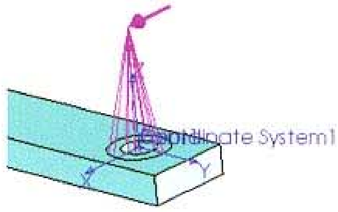
Apply a vertical upward load of **250 lb** to the split face surrounding the hole at the free end of the cantilever beam.



8 Apply remote load.

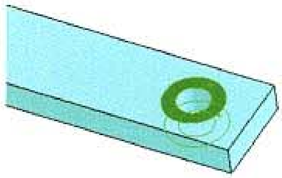
Apply a horizontal **250 lb** force, as indicated in the figure.

Use local `Coordinate System1` to specify the location of the force (**0**, **0**, **2 in**) and the magnitude (**250 lb**, **0**, **0**).



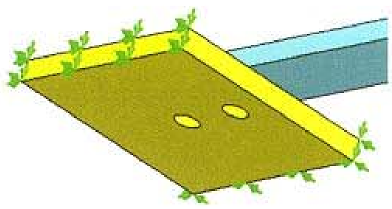
As mentioned in the beginning of this problem, we assume that the `eye` bolt is rather rigid. Thus, use the **Load/Mass Rigid connection** option.

Select both the bottom and the top contact faces as **Faces for Remote Load**. This reflects the reality in which most of the loads are transmitted through the friction forces between the bolt head/nut and the bar.



9 Apply Immovable restraint.

Apply **Immovable** restraints to the two side faces on the `base plate`.

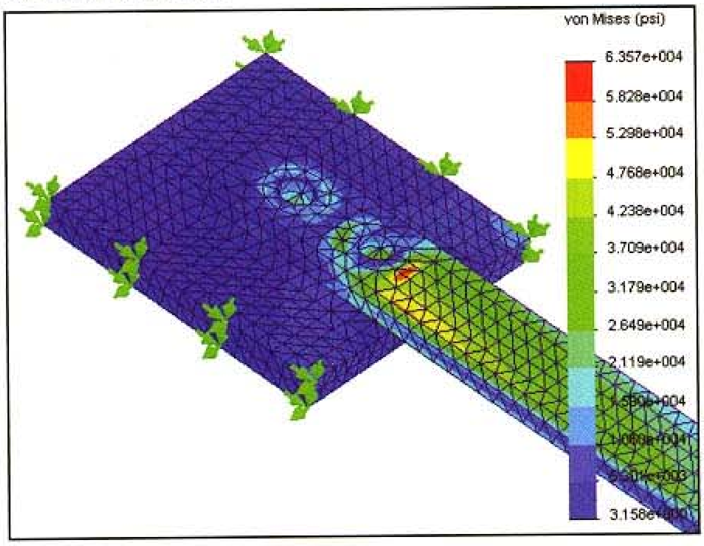


10 Mesh assembly.

Create **High** quality mesh with the default **Element size** of **0.285 in.**

11 Run the analysis.

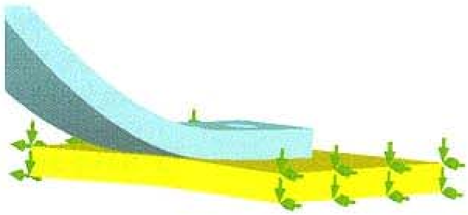
12 Review Results.



Review the area where the highest stresses are located and notice that the size of the "hot spot" is smaller than that of the element size. Therefore, stresses in this area are reported with a large error.Mesh refinement would be required to obtain accurate maximum stress results.

13 Plot details of deformation.

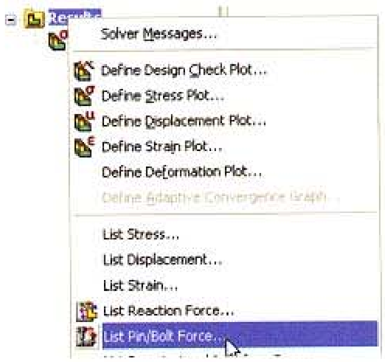
Analyze the details of the deformation using a **Deformation** result plot in magnified scale.

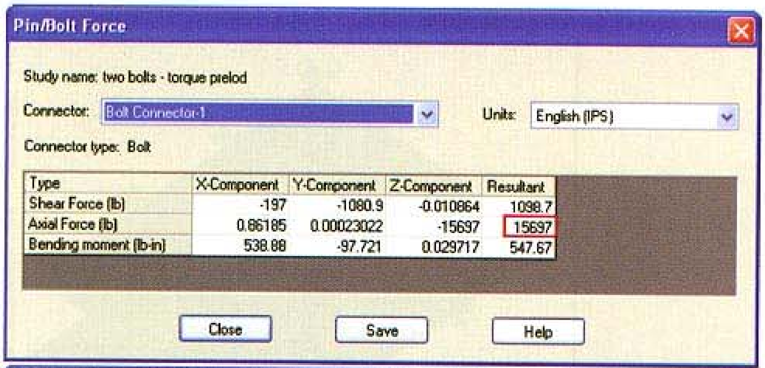


It can be seen that the `bar` and the `base plate` separate from each other.

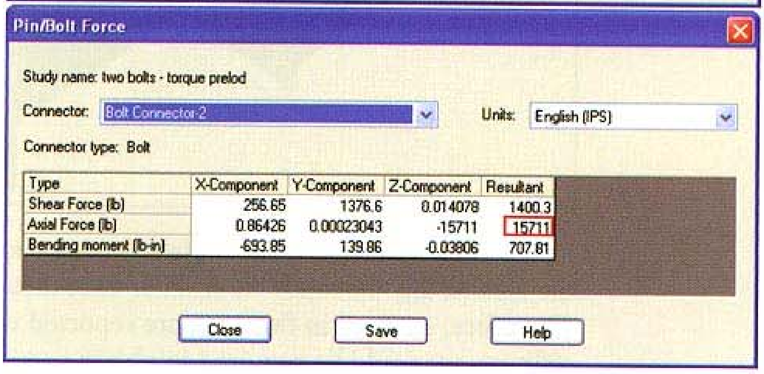
14 Review bolt forces.

To review bolt forces, right-click the `Results` folder and select **List Pin/ Bolt Force**.





Type	X-Component	Y-Component	Z-Component	Resultant
Shear Force (lb)	-197	-1080.9	-0.010864	1098.7
Axial Force (lb)	0.86185	0.00023022	-15697	15697
Bending moment (lb-in)	538.88	-97.721	0.029717	547.67



Type	X-Component	Y-Component	Z-Component	Resultant
Shear Force (lb)	256.65	1376.6	0.014078	1400.3
Axial Force (lb)	0.86426	0.00023043	-15711	15711
Bending moment (lb-in)	-693.85	139.86	-0.03806	707.81

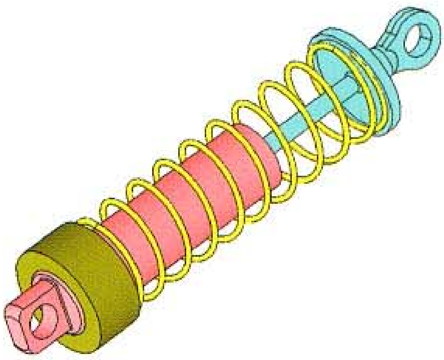
Bolt Connector-1 and **Bolt Connector-2** are the first two bolt connectors that we defined. The corresponding axial bolt forces are 15697 lb. and 15711 lb.

As compared to the bolt preload of 15,700 lb., the effect of the external load is very low. A negligible change in bolt axial load is desirable if we want to avoid bolt loosening.

15 Save and close the assembly.

Project Description - Stress Analysis of a Shock Absorber

A miniature shock absorber consists of a tube, a plunger, a clamp, and a helical spring. In this section, we investigate the stresses that develop in the plunger collar when the assembly is compressed with a 3 N force.



Stresses in the helical spring are not of interest. Therefore, we remove the spring from the model, and replace it with an equivalent spring connector.

Calculate Compressive Spring Stiffness

First, the stiffness of the helical spring must be calculated. For this we analyze the spring individually.

To calculate the compressive spring stiffness:

1 **Open part.**
Open the SolidWorks part `spring copy.prt`.

Note

For convenient application of restraints and loads, disks have been added to both ends of the spring. The distance between the disks corresponds to the active length of the un-compressed spring.

2 **Set COSMOSWorks options.**
Set the system of **Units** to **SI (MKS)** and the units of **Length** and **Stress** to **mm** and **N/m^2 (Pa)**.

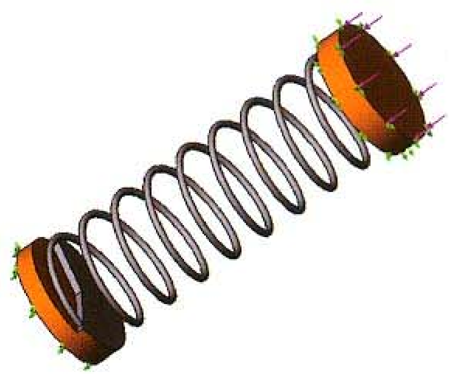
3 **Create study.**
Toggle to COSMOSWorks and create a study named `spring stiffness` (**Static** analysis, **Solid** mesh).

4 **Review material properties.**
The material properties (**Alloy Steel**) are transferred from SolidWorks.

5 **Apply Fixed restraint.**
Apply a **Fixed** restraint to the end face of one disk (item 1).

6 **Apply radial restraint.**
Apply a restraint in the radial direction to the cylindrical face of the other disk (item 2).

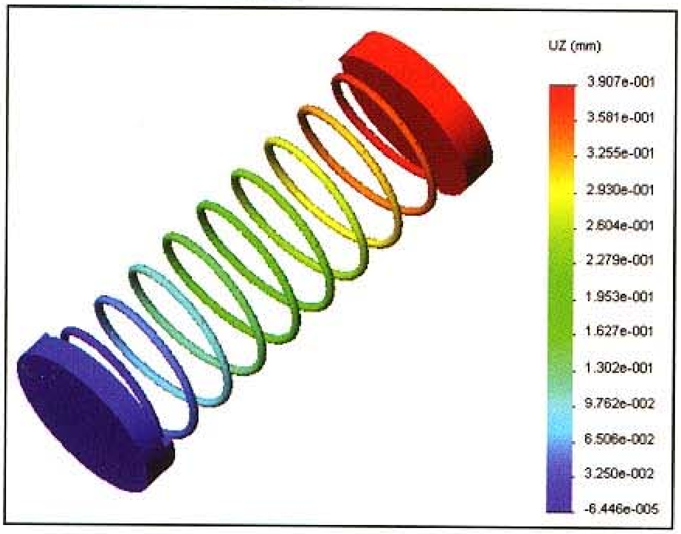
This restraint only allows the spring to be compressed (or expanded) in its axial direction.



7 **Apply compressive force.**
Apply a 0.1 N compressive force to the end face of the disk with the cylindrical face constrained in the radial direction.

8 **Mesh the model and run the analysis.**
Use **High** quality elements with the default **Element size** and **Tolerance** of **1.394 mm** and **0.07 mm**, respectively.

9 Plot *z* displacements.



Displacement results indicate an axial displacement of 0.391 mm. The axial displacement is in the *z* direction.

Coil Spring Axial Stiffness

Therefore, the axial stiffness of the spring is 255.7 N/m. ($k = f/x$).

We use this result to define the spring connector in the next model using the equation $f = kx$, where $k=255.7\ N/m$.

Alternately, we could use an approximate formula for the stiffness of a helical spring (Mechanical Vibrations by S. S. Rao, 1995).

$$K_{AXIA} = \frac{Gd^4}{8nD^3}$$

where G is the material shear modulus, d is the diameter of the wire, D is the mean coil diameter, and n is the number of active turns.

Substituting our values (n = 8.75, d = 1 mm, D = 17 mm, and G = 7.9e10 N/m) into the above formula gives an axial stiffness of approximately 230 N/m. We will, however, use our accurate result of 255.7 N/m.

Analyze Shock Absorber Assembly

Having determined the stiffness of the helical spring, we can proceed with the analysis of the whole shock absorber.

10 Open assembly.

Open the assembly file named `shock` and suppress the helical spring (part file `Front Spring`).

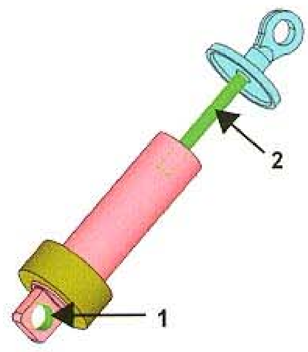
11 Create study.

Create a study named `shock assembly` (**Static** analysis, **Solid** mesh).

12 Apply Immovable restraints.

Apply an **Immovable** restraint to the cylindrical face of the eye belonging to the `Shock Tube` (item 1).

This restraint fully restrains the `Shock Tube` component.



13 Restrain `Shock Plunger`.

Apply restraints in the radial and circumferential directions to the cylindrical face of the eye (item 2) belonging to the part `Shock Plunger`.

Use **On cylindrical face** as the **Type** of restraint.

Note

With these three constraints, the assembly model is left with one rigid body motion. The `Shock Plunger` can slide in and out of the tube because the `Shock Plunger` and `Shock Tube` are disconnected. We connect these two parts with a spring connector.

14 Define spring connector.

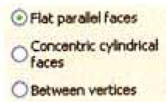
Right-click `Load/Restraint` and select **Connectors**.

Select **Spring** from the list of available connector types.

Spring connector types

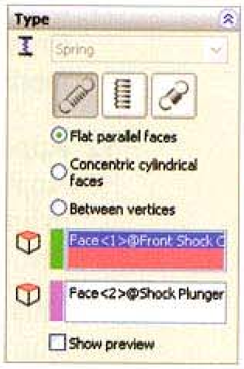
Under **Type**, we can specify whether the spring is active in both compression and tension, compression only, or tension only.

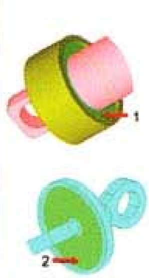
The options **Flat parallel faces**, **Concentric cylindrical faces**, and **Between vertices** specify the characteristics of the spring end entities.



Under **Type**, select **Compression Extension** with **Flat parallel faces**.

As shown, specify the selected face on the `Shock Tube` (item 1) as the **Planar Faces of Component1** and the face on the `Shock Plunger` (item 2) as the **Parallel Faces of Component2**.



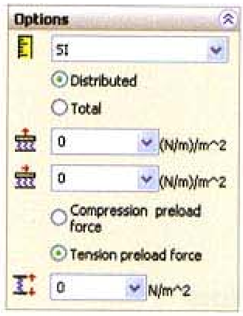


Note Which face is selected as planar and parallel is not important.

Spring connector options

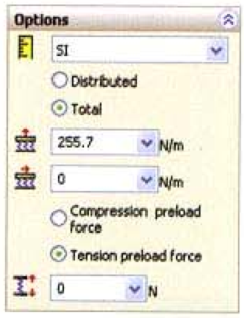
Under **Options** we can specify the **Normal** and **Tangential** spring stiffness values. Both quantities can be expressed as **Total** (N/m or lb/in), or **Distributed** in the units of (N/m)/m^2 or (lb/in)/in^2, for example.

Both the **Compression preload** and **Tension preload** can be input.



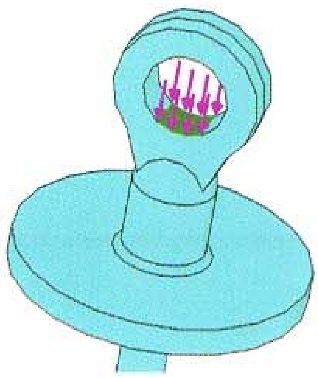
Under **Options**, select **Total** stiffness and enter **255.7 N/m** in the **Normal** direction. The `Shock Tube` and `Shock Plunger` are now connected.

Click **OK**.



15 Apply force to shock plunger.
Apply a **3 N** load to the split face on the cylindrical face of the `Shock Plunger` ear in the direction of the rod.

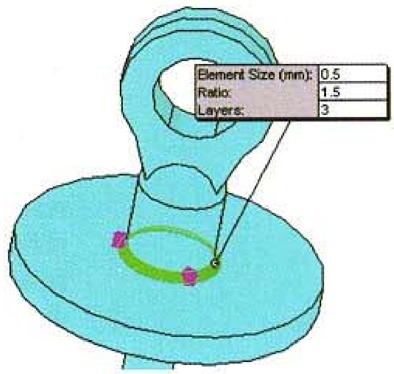
The load is applied normal to `Plane1`.



16 Apply mesh control.

Apply mesh control to the fillet face on the `Shock Plunger` where a higher stress concentration can be expected.

Specify a local **Element size** of **0.5 mm** and the default **Ratio** and **Layers** of **1.5** and **3**, respectively.



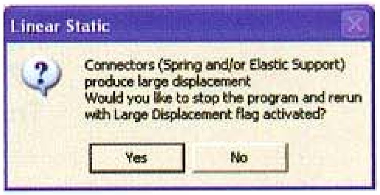
17 Mesh the model.

Mesh the model with **High** quality elements with the **Element size** of **1 mm** and the **Tolerance** of **0.05 mm**.

18 Run the analysis.

Run the analysis and note that the solver issues a warning about large displacements.

Click **No**. The analysis will then complete.



Large Displacement Warning

In this case, we ignore the warning because the large displacements caused by elasticity introduced by the spring connector are translations only. The rotations of the assembly components are restricted and the deformations are very small.

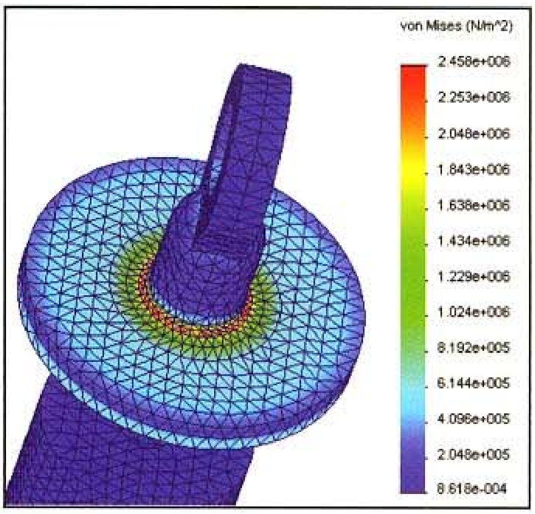
You may run the solution with and without the **Large Displacement Contact/Connector** flag activated to verify that both runs produce the same results.

Large displacement analysis is a subject of Lesson 12.

19 Plot von Mises stresses.

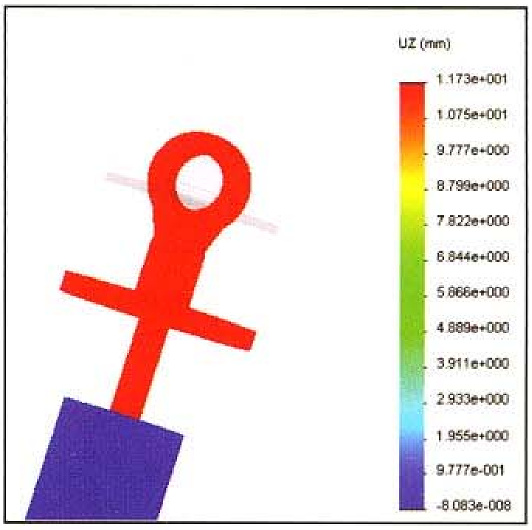
We note that the maximum stresses of 2.5 MPa are well below the yield strength of the Alloy Steel (620 MPa).

For more accurate stress results, we could further refine the mesh around the fillet, but that is not the main purpose of this project.



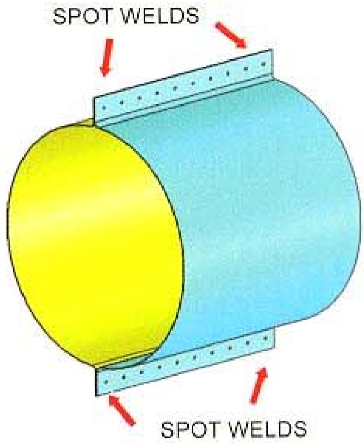
20 Plot displacements.

Plot the distribution of the **UZ: Z Displacement** in the **True Scale**.



We observe the maximum displacement of 11.7 mm, which is in accordance with the linear spring equation ($u_z = F/k = 3/255.7 = 0.0117$ m = 11.7 mm).

Problem Description - Using Spot Welds

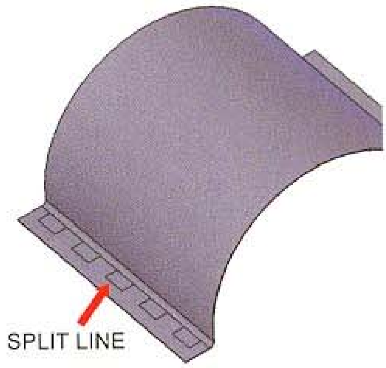


A tube is fabricated out of two sheets of galvanized steel 0.04" thick. The two pieces are joined by 10 spot welds on each side. The spot welds are spaced 1" apart and the diameter of each spot weld is 0.125".

We use FEA to investigate torsional stiffness of the assembly by finding the torque required to twist the tube by 1^o.

The twist angle of 1^o is arbitrary. We are not attempting to duplicate any real life test conditions. We intend to use the results of this numerical test to compare different spot welds configurations. The two-piece design is the first configuration we test.

1 **Open Assembly.**
Open the assembly model `tube solid` and examine two configurations: `complete tube` and `half tube`. The assembly consists of two identical parts, `tube 30`. Examine part model `tube 30` and note a split line added to locate the positions of spot welds.



2 **Activate `complete tube` configuration.**

3 **Set COSMOSWorks Options.**
Set the system of **Units** to **English (IPS)**, the units of **Length** and **Stress** to **in** and **psi**, respectively.

4 **Create study.**
Create a study named `tube solid` (**static**, **solid mesh**).

5 **Review Material properties.**
Verify that the material definition (**Galvanized Steel**) has been transferred from SolidWorks to COSMOSWorks.

Defining Spot Welds

To define a **Spot weld**, two faces which are connected by the spot weld and it's location need to be specified. This location can be specified on either one of these two faces.

To specify the spot weld location we can use a reference point (it must be assembly reference point, not part reference points) or a vertex. In this lesson we will use vertices created from split lines.

6 Define Spot Welds

Spot welds are defined as connectors. To define a spot weld, right-click `Load/Restraints` folder and select **Connectors**.

In the **Type** list, click **Spot welds**.

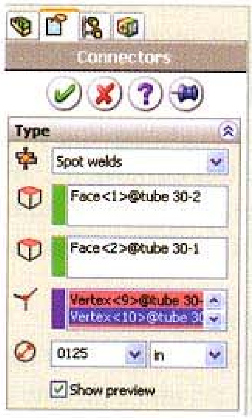
Select **Spot weld first face**, as shown in the figure.

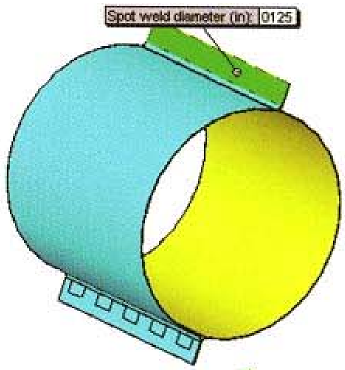
Then select the connected face on the other part (see the figure) and select all **Spot weld locations**.

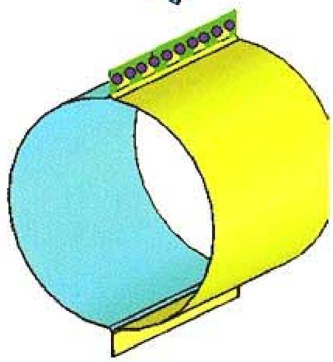
In the **Spot weld diameter**, enter **0.125 in**.

This way, all spot weld locations on one side are defined in a single restraint.

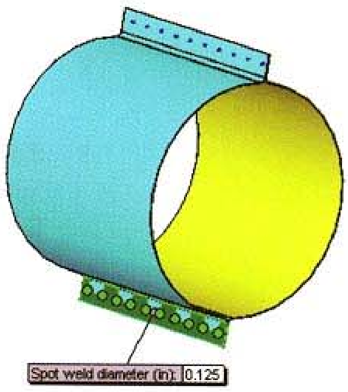
Click **OK**.







7 Similarly, apply spot welds at all the other 10 locations on the other side of the tube.



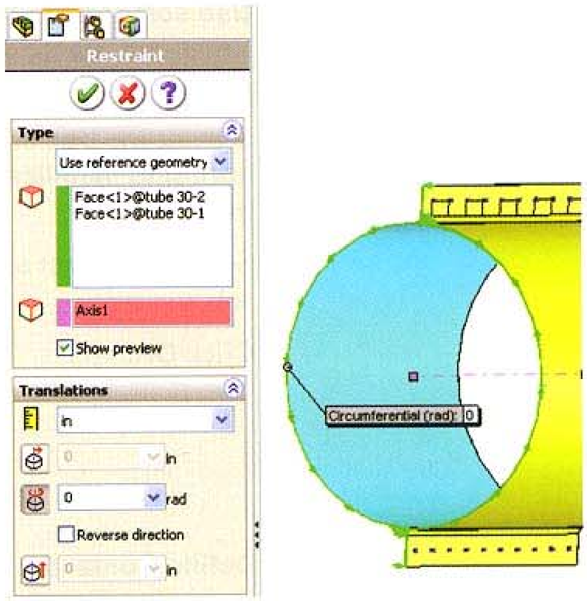
8 Apply restraints.

Now apply restraints to the faces at each end of tube.

Select the **Use reference geometry** type of restraint and select the assembly axis as reference geometry. This way, the directions of restraints are aligned with the cylindrical coordinate system defined by this axis. The first component is radial translation, the second is circumferential rotation (expressed in radians), and the third is axial translation.

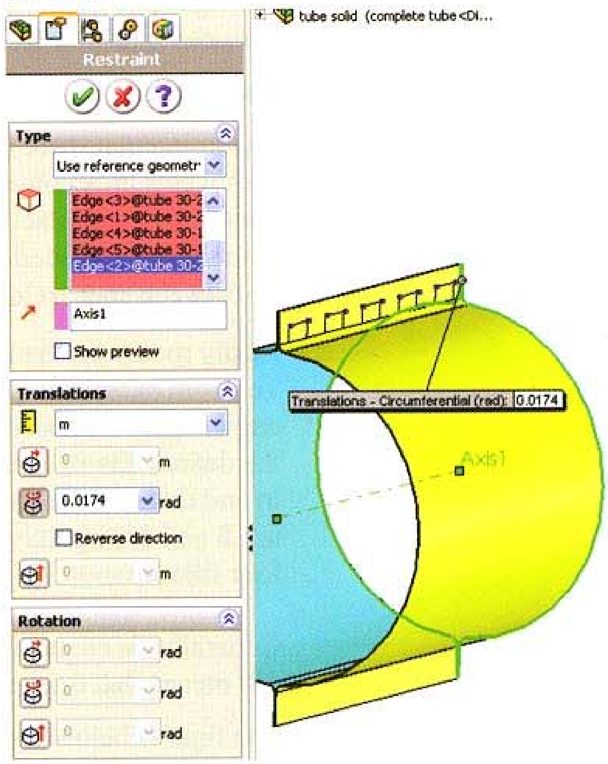
Select the two faces on one side of the tube and restrain the **Circumferential** displacement component (enter **0 rad**).

Click OK.



9 Prescribe rotation at the other end.

Analogously to the previous condition, apply a 1° (**0.0174 rad**) **Circumferential** displacement to the two faces on the opposite end.



Note

Note that the prescribed displacements defined on both ends of the tube do not restrain the assembly in the axial direction. The model can move in axial direction as a rigid body without experiencing any deformation. The model will be stabilized using the **soft springs** option.

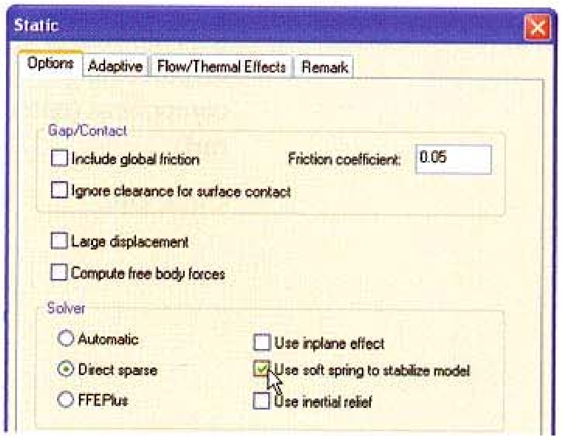
10 Use soft springs to stabilize the model.

Right-click on the study `tube solid` and select **Properties**.

Under the **Options** tab, activate the **Use soft spring to stabilize model** option.

Select the **Direct sparse** solver.

Click **OK**.



11 Define Contact.

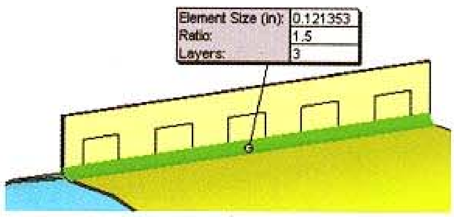
Before meshing the model we make an important modeling decision: we assume that the two halves of the tube interact with each other only through spot welds. Consequently, we model the contact condition between touching faces of the two halves as **Free**.

Set this option in the **Global Contact** PropertyManager.

Assuming that there is no interaction other than specified through the spot welds, this conveniently simplifies the model because we do not have to solve contact conditions. However, it is a reasonable assumption considering that thin spot welded sheets often "come apart" in-between spot welds.

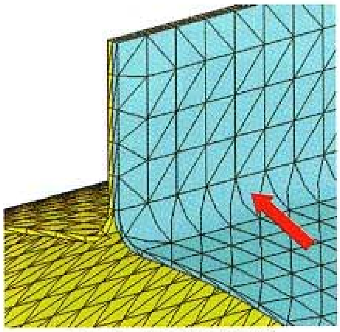
12 Apply mesh control.

To avoid excessive element turn angle, apply a mesh control with the default **Element size** of **0.121 in** and the **Ratio** and **Layers** equal to**1.5** and **3**, respectively, to all four fillet rounds.

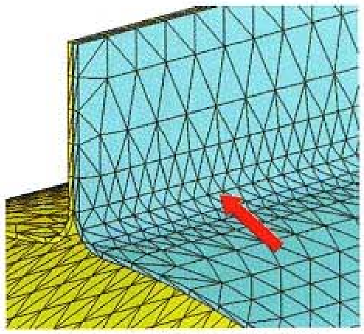


Generally an element turn angle of 45° or less is preferred. Turn angle 45° means that one element face "wraps" over a 45° arc.

The figures below show the mesh with and without the mesh control definitions.



NO MESH CONTROL
ELEMENT TURN ANGLE 90 DEG



MESH CONTROL APPLIED
ELEMENT TURN ANGLE 30 DEG

Spot Welds - Stress concentrations

Note that the mesh has only one element across the wall thickness. Generally two layers of second order elements are recommended. One layer is acceptable for the analysis of deformations but may produce high stress error in detailed stress results.

We accept one layer of elements because we intend to use this model for the analysis of deformations, not stresses. Besides, models with **Spot welds** connectors are not suitable for detailed stress analysis. **Spot welds** connector models point to point connections, which mathematically results in infinite stresses near the spot weld.

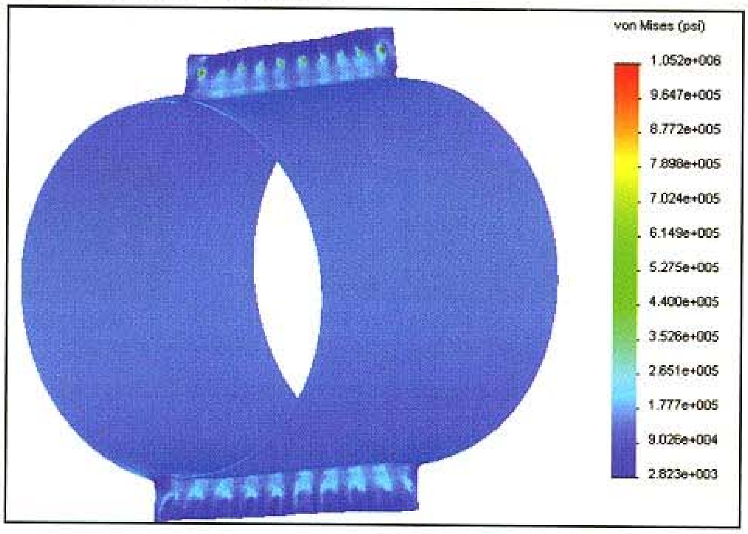
Models with **Spot welds** are suitable for analysis of deformations and global stresses, which is our intention in this model.

Also, the model geometry would be better meshed with shell elements than with solid elements. We use solid elements to practice **Spot welds** connectors with solid geometries. Later in this lesson we will solve the same model using shells.

13 **Mesh the model.**
Create a **High** quality mesh with a default **Element size** of **0.243 in.**

14 **Run the analysis.**

15 Plot von Mises stresses.



Von Mises stress results indicate high stress near spot welds. As we said before, any stress results near spot welds are unreliable.

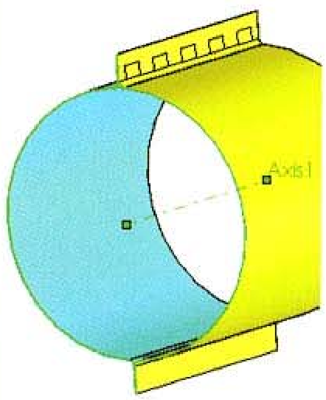
Resulting torque extraction

To calculate the resulting torque, we first list the *y*-component of the reaction force in the cylindrical coordinate system and multiply it by the radius (see Exercise 3 for an additional example).

16 List reaction force in cylindrical system.

List the *y*-component of the reaction force in the cylindrical coordinate system defined by `Axis1` (see the following figure).

The circumferential component of the reaction force is **1.068e5 lb**.



17 Calculate resulting torque.
The average radius is 4.98 in. Therefore the resulting torque T is:

$$T = 1.068e5 \text{ lb.} \times 4.98 \text{ in.} = 527.9e3 \text{ lb.} \times \text{in.}$$

Spot Welds - Shell mesh (optional)

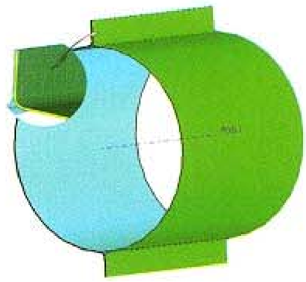
We can compare the results of the previous study with the results of a study using shell elements. This study will also show the use of spot welds for shell sheets that are not in direct contact.

18 Create new study.
Define a new **Shell mesh using surface, Static** study named `tube shells`.

19 Define shell sheets.
Define one sheet for each half of the tube. Specify the **Thickness** of **0.04** and use **Thin** shell technology.

Note

Use the outside faces to define the shell sheets.



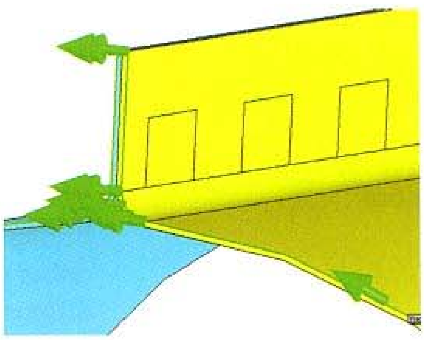
20 Assign materials.
Make sure that **Galvanized steel** is assigned to both sheets.

21 Apply restraints.
Apply a 1° circumferential displacement to all the edges on one side.

Restrain the outer edges on the other side against the circumferential displacement (rotation).

Note

Make sure that you apply the restraints on the outer edges where the shell sheets were previously defined.

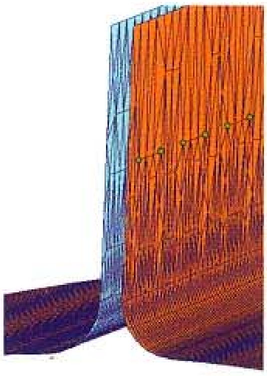


22 Define spot welds.
Similarly to the previous study, define the spot welds.

Spot welds on sheets not in contact

Notice that the two faces welded via the spot welds are not in direct contact.

This feature of COSMOSWorks can be used extensively in more complex shell and mixed assemblies.



23 Apply mesh controls.
Similarly to the previous study, apply mesh controls to the four fillet rounds. Use the default **Element size** of **0.12 in** and the **Ratio** and **Layers** of **1.5** and **3**, respectively.

24 Mesh the assembly.
Create a **High** quality mesh with a default **Element size** of **0.364 in**.

25 Apply soft springs and specify solver type.
Similarly to the previous study, activate the **Use soft spring to stabilize model** option.

Specify the **Direct sparse** solver.

26 Run the analysis.

27 List reaction force and calculate torque.
Using the identical procedure as in the previous study, list the circumferential component of the resultant reaction force.

The resulting force is 1.056e5 lb. Therefore, the corresponding torque is 1.056e5 lb. x 5 in. = 528e3 lb. x in., which corresponds very well to the results obtained in the study `tube solid`.

Note

We used a radius of 5 in. rather than the average radius of 4.98 in. because the shell sheets were defined using the outside tube faces. A rather small difference in the radii would translate to a very small difference in the resulting torques and can thus be neglected.

This fact was, however, already in our modeling assumption: a small difference in the inner and outer radii enabled us to use the outer faces for the definition of the shell sheets rather than the real midsurface, the extraction of which can be cumbersome.

Summary

The main focus of this lesson was the use of connectors, special supports, and contacts in an analysis with COSMOSWorks.

In the `lift` assembly model, by using pin connectors we avoided modeling the pins themselves, and by using a virtual wall we eliminated the need to mesh the base. These features allowed us to simplify the model.

The hinge support was used to model the connection between the arms and the base. This approach, however, did not produce accurate stress results around the pin holes and hinge supports, and we could not account for the flexibility of the base.

Accurate stress analysis of these regions would require modeling the pins and the base explicitly in a contact stress problem.

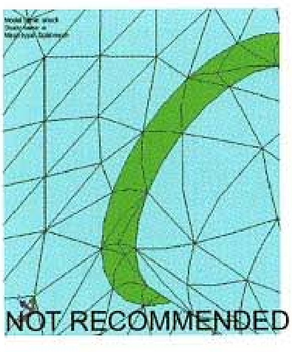
We saw how friction can also be used in the virtual wall contact analysis.

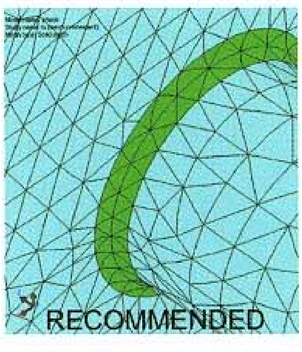
In the `bolt joints` assembly model, we used bolt connectors to simplify the model by eliminating the nuts and bolts from the assembly. This approach is acceptable if only the bolt forces, for the subsequent bolt design using various tables, are of interest. The bolts have to be modeled if the real detailed stress distribution within the bolts and their vicinity is required. This is, however, rarely the case.

In the `shock` assembly model, we analyzed an assembly and used a spring connector to simplify the model by eliminating the spring component. Of course, this approach is acceptable only if the spring itself is of no interest in the analysis.

Using the `tube solid` assembly and part files, we analyzed the use of Spot Welds. In both the solid and shell element models, we observed that spot welds result in very high unrealistic stress concentrations that have a localized effect only. While they have no effect on the solution of the entire structure and can thus be ignored, they greatly simplify the problem and reduce the required computational time.

We also discussed the fact that for accurate stress results in rounds and fillets, a 90° fillet should be meshed with a minimum of two layers of elements.





Lesson 8
Mixed Meshing - Analysis of an Impeller

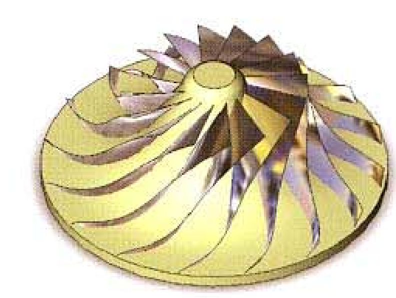
Objectives

Upon successful completion of this lesson, you will be able to:

- Discuss mesh compatibility problems in mixed solid and shell element meshes
- Use mixed meshes for different types of analysis
- Understand the concept of compatible vs. incompatible meshing

Mixed Meshing

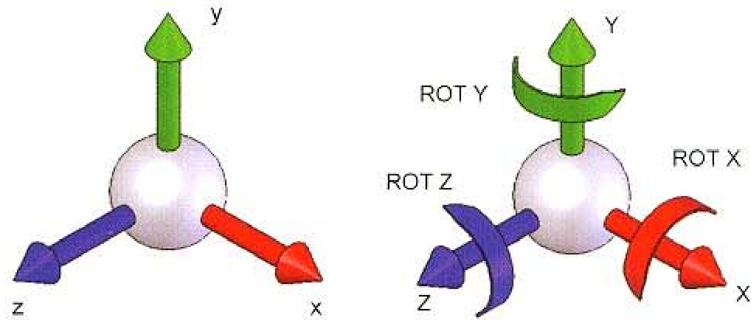
Review the impeller geometry to notice that it consists of a "chunky" core with thin blades attached to it.



The core would mesh well with solid elements while the thin blades would be well represented by shell elements. We will use the mixed mesh capabilities of COSMOSWorks to construct a mesh where solid elements "coexist" with shell elements in the same study. However, we will see that it takes extra efforts to ensure mixed mesh compatibility.

You will recall from "Introduction to FEA" in the first chapter of this volume that nodes of a solid element have three degrees of freedom, meaning that node displacement is fully described by three translational components.

You will further recall that nodes of a shell element have six degrees of freedom. Displacement of a shell element node is described by three translational components and three rotational components.



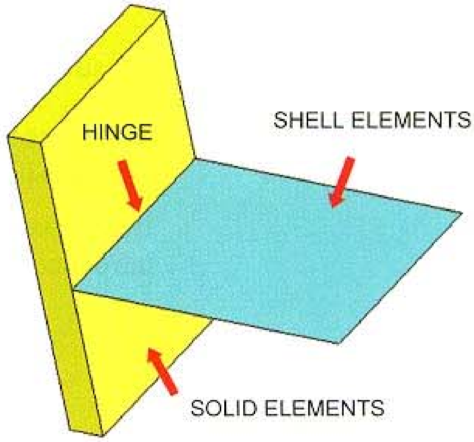
DEGREES OF FREEDOM OF A NODE OF A SOLID ELEMENT

DEGREES OF FREEDOM OF A NODE OF A SHELL ELEMENT

We usually show these displacement components (or degrees of freedom) as aligned with global coordinate system. However, degrees of freedom may be presented in any coordinate system.

Because nodes of solid elements do not have rotational degrees of freedom as compared to nodes of shell elements, an attempt to connect shell and solid elements results in an unintentional hinge along the common edge.

The rotational degrees of freedom of shell element nodes have nothing to "hold on to" at the interface with solid element nodes. Therefore, these rotations remain unconstrained, forming a hinge along the connecting edge.



With a hinge joint present, we have a discontinuous displacement field (discontinuity of rotations) and possible rigid body nodes in the model.

The incompatibility between shell and solid elements that lead to unintentional hinges is not specific to COSMOSWorks. It occurs in any FEA software every time we try to connect elements of different type having nodes with different numbers of degrees of freedom.

Modeling Issues

In COSMOSWorks, an "untreated" connection between solid elements and shell elements not only creates a hinge, but solid and shell portions of the mesh remain completely detached.

- To connect them, we must define proper local contact conditions along all connecting edges. This restriction has two important consequences for the end user:

  - The finite element mesh will always be incompatible across a contact interface

  - The bonded entities (regardless whether they are solid-to-solid, solid-to-shell, or shell-to-shell) do not need to be in direct contact.

  The above two consequences are important in simplifying the meshing of the solid geometries using mixed mesh in COSMOSWorks.

  Models intended for mixed meshes may need to be prepared specifically with this purpose in mind.

- Portions intended for shell meshing may (do not have to) be modeled as surfaces.

- Shell elements could be created from designated faces of a solid geometry, but this makes the solid geometry unavailable for meshing with solid elements. Thus, every thin solid intended for shell meshing must be an independent body.

- Shell and solid mesh entities do not have to be in direct contact.

This benefit, stemming out from the need to define a local contact condition for all the contacting edges, has an important modeling advantages, as we will see in the next few lessons.

Supported Analysis Types

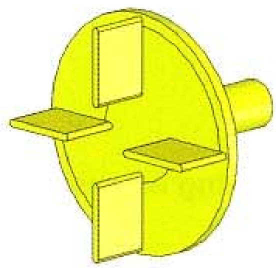
Mixed meshing is available for Static, Frequency, Buckling, Thermal and Non-linear studies.

Before we proceed with the analysis of an impeller, we will first introduce the modeling concepts described in the above paragraphs.

Mixed Mesh: single body parts

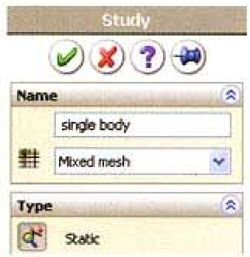
In this section we will learn that solid geometries intended for solid and shell geometries must not be together in a single body.

1 **Open part.**
Open part model `rotor 01`.



2 **Create Mixed Mesh study.**
Create a mixed mesh `static` study by selecting **Mixed mesh** as the mesh type.

Name this study `single body`.

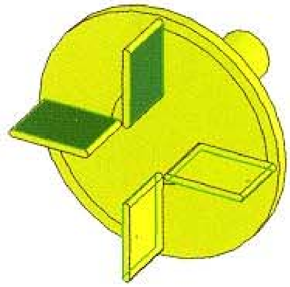


3 **Review Study.**
Note that, in addition to the familiar `Solids` folder, the study also contains a `Shells` folder.

Note

Notice that the `Solids` folder contains only a single part.

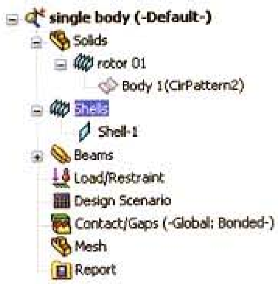
4 **Define Shell surfaces.**
Right-click the `Shells` folder and select **Define by Selected Surfaces** to indicate which surfaces should be meshed with shell elements.



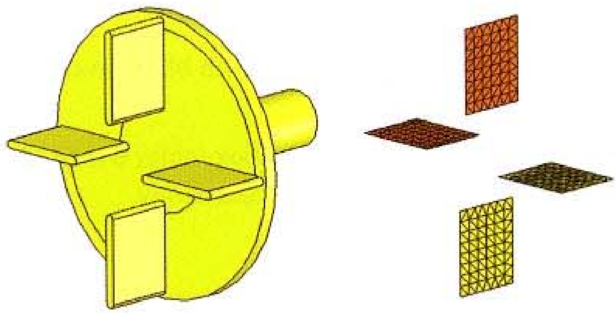
Select one side face on each blade, designating them for meshing with shell elements.

In the **Shell thickness** box, enter **6 mm**.
Keep the default **Thin** shell technology even though it is not important in this study.

Once the **Shell Definition** PropertyManager has been closed, notice that all solid geometry has been deleted and the `Solids` folder contains only surface features!



5 **Mesh the Model.**
Mesh this model with the default mesh size. After meshing this model, notice that the mesh consists of four detached pieces for only the shell surfaces.



This example illustrates that we cannot just mesh faces of solid models with shells. Instead, we must prepare a combination of solid and surface geometry for a model specifically intended for mixed mesh modeling.

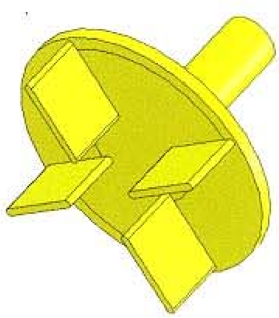
To illustrate this, we will analyze the model `rotor 02`, where blades are represented by a surface geometry.

6 **Close `rotor 01` part.**

Mixed Mesh: multi body parts

7 Open part.

Open part model `rotor 02a`.



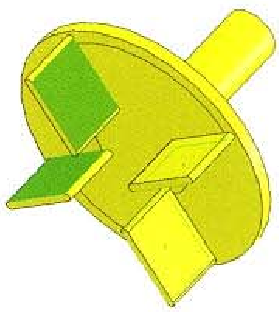
8 Create Mixed Mesh study.

Create a **Mixed mesh** study named `multi body part`.

Note that we now have the same part with each solid geometry as an independent body.

9 Define Shell surfaces.

Right-click the `Shells` folder and select **Define by Selected Surfaces** to indicate which surfaces should be meshed with shell elements.



Select one side face on each blade to define it as a shell sheet.

In the **Shell thickness** box, enter **6 mm**. Keep the default **Thin** shell technology.

Click **OK**.

Notice that once the definition of the shells has been completed, all the corresponding bodies have been deactivated.

Note

Remember that the model in the `multi body part` study `rotor 02` is not ready to be solved. In order to keep the mixed mesh in one piece, we would have to define local bonded contact conditions between the solid face and the blade shells.

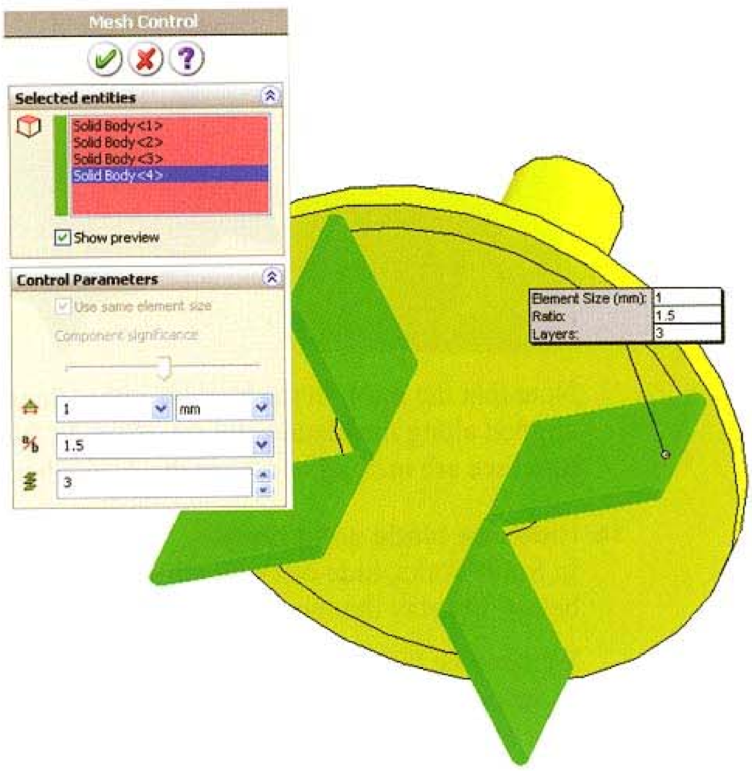
Also, we could question how the nodes of the finite element are aligned along the contact lines. This is a subject of the next section.

Compatible / Incompatible Meshing

In the following exercise, we will learn the difference between compatible and incompatible meshing. We will use solid mesh assembly for this purpose.

10 Define a new study.
Define a new **Static, Solid mesh** study named `rotor2a-solid`.

11 Apply mesh control.
Apply mesh control with the **Element size** of **1 mm** and the default **Ratio** and **Layers** to all four blade bodies.

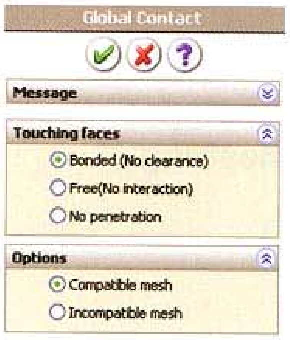


Compatible mesh

When compatible mesh is requested, the parts in the assembly are meshed so that a smooth mesh transition between any two parts is achieved. The nodes along the interface are then imprinted one upon another and merged to ensure the bonding. If compatible meshing fails at some interface, the software will attempt to generate incompatible mesh for the two parts involved.

12 Set global compatible bonding.

Right-click on the `Contact/Gaps` folder and select the **Set Global Contact** option.



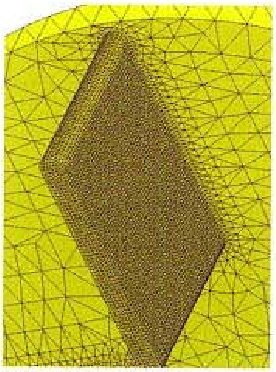
Make sure that **Bonded** is selected under the **Touching faces** dialog.

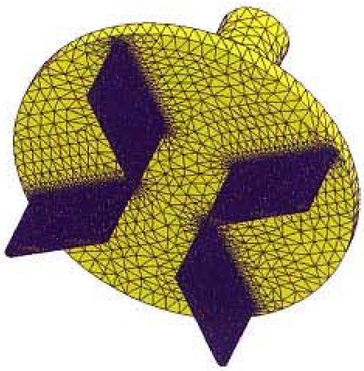
Under **Options**, activate **Compatible mesh**.

13 Mesh part.

Mesh this multi body part with a global **Element size** of **8.158 mm**.

The resulting mesh can be seen below.

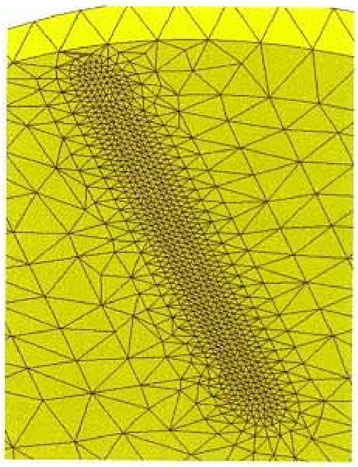




Note that the mesh in the blade is rather dense and the nodes are nicely aligned along the blade/solid interface. In fact, the nodes along this interface are merged to ensure the prescribed bonding.

14 Hide one blade solid body.

In SolidWorks, hide one of the blade bodies, as shown in the figure below. Display the mesh in COSMOSWorks.



We observe that a node-to node correspondence is also forced along the entire blade/solid interface. The nodes are subsequently merged to ensure the required bonding.

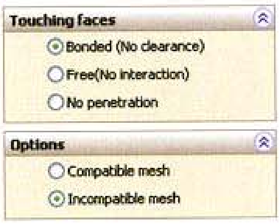
While merging the nodes is the most accurate way to ensure the bonding between two touching bodies in a part (or two parts in an assembly), it poses additional constrains on the mesher. All bodies and parts must be meshed together, causing the operation to become more complex and take longer.

Incompatible mesh

Alternately, we may prescribe the mesher to mesh every body/part independently and ensure the bonding via the constraint equations (additional mathematical expressions).

15 Set global incompatible bonding.

Edit the setting of the global bonding and change the **Options** to **Incompatible**.

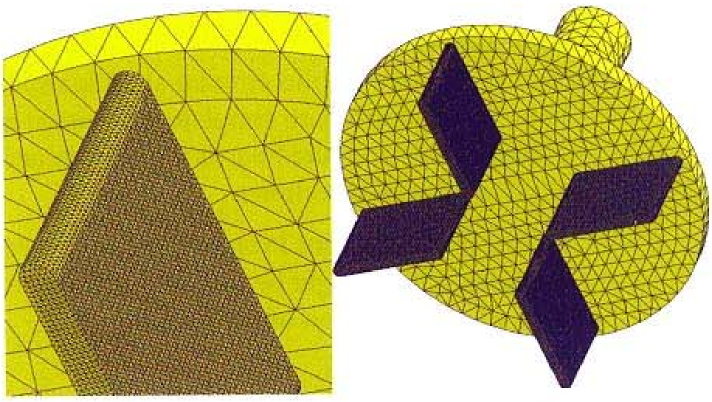


16 Show all the blades.

Display the hidden blade.

17 Mesh part.

Mesh the part with the default **Element size** of **8.158 mm**.



Observe that the solid body of the rotor and the blades are meshed independently and the nodes along the interface are not aligned. The bonding is ensured by means of the additional constraint equations.

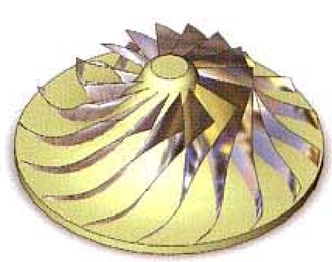
Shell and Mixed mesh: compatible and incompatible meshing

The user selection on whether to generate globally compatible and incompatible contact mesh is available for initially touching faces in solid meshing only. In the case of mixed and shell meshing, the generated mesh will always be incompatible across the contact interfaces, irrespective of the user's specification. Even if the nodes look aligned across the interface, the bonding will be incompatible.

Additionally, whenever a local contact condition is defined (regardless of the mesh type) the mesh will be incompatible across the interface.

Mixed Mesh: Analysis of an Impeller

A turbocharger impeller spins with a constant speed of 20,000 RPM.



The centrifugal force causes expansion of the impeller diameter.

We must determine the increase in diameter of the impeller and the location of the maximum magnitude of von Mises stresses.

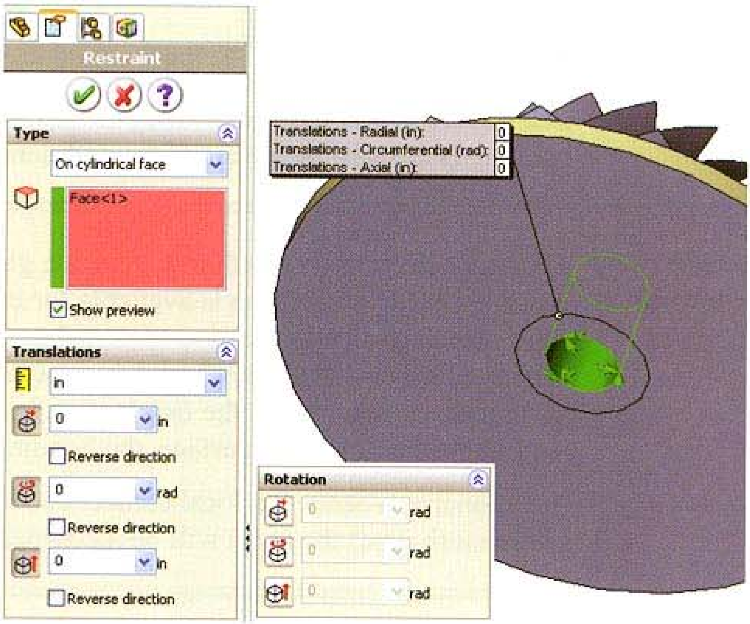
18 Open Part file.
Open the SolidWorks part file `impeller 01`.

In this assembly each blade is represented by a surface, not a solid geometry (however, this not a requirement). The surface edge attached to the solid geometry is coincidental with a curve associated with solid geometry.

19 Create Mixed Mesh study.
Create a **Static** study named `centrifugal 01` with **Mixed mesh** elements.

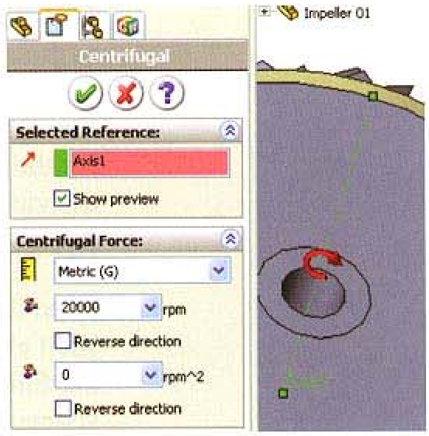
20 Apply Restraints.
We assume that a solid circular shaft is rigidly connected to the impeller. Therefore, to simulate a presence of the shaft, select the cylindrical face of the hole at the bottom and restrain circumferential, axial and radial directions.

Notice that **Restraint** PropertyManager gives the option of restraining rotations which might be required if we were applying restraints to shell element portion of the model.



21 Apply a centrifugal load of 20,000 RPM.

Right-click `Loads/Restraints` and select **Centrifugal**.



From the SolidWorks floating FeatureManager, select `Axis1` as the **Selected Reference**.

In the **Units** list, select **Metric (G)**.

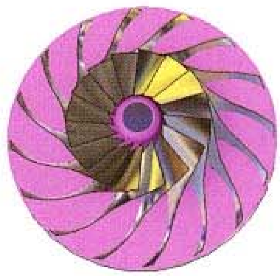
In the **Angular Velocity** box, enter **20,000 rpm**.

Click **OK**.

We do not apply any pressure to impeller blades because stress induced by pressure is negligible as compared to stresses caused by centrifugal forces.

22 Assign Shell surfaces.

Right-click the `Shells` folder and select **Define By All Reference Surfaces**.



In the **Shell Definition** PropertyManager, all the blades in the model will be selected in the **Selected entities** list. Alternatively, if you select **Define by selected surfaces** to define surfaces, you need to select the sixteen blades individually.

Under **Type**, select **Thin** shell formulation.

In the **Shell thickness** box, enter **1 mm**.

Click **OK**.

Upon closing the **Shell Definition** PropertyManager, COSMOSWorks creates `Shell-1` in the `Shells` folder. `Shell-1` includes all surfaces in the model. It would also be possible to define more than one component in the `Shells` folder, i.e. if different blades had different thicknesses.

23 Apply Material.

Material needs to be applied in two steps: to components of `Solids` folder and to components of `Shells` folder. The easiest way to apply material is to right-click each folder and select **Apply Material to All**. Use `Alloy steel` material found in COSMOSWorks material library.

24 Apply Mesh control.

For accurate stress results apply mesh control to the cylindrical face where support has been applied and to the split face surrounding it. The split face has been defined specifically for the purpose of applying mesh controls in the vicinity of restraint.

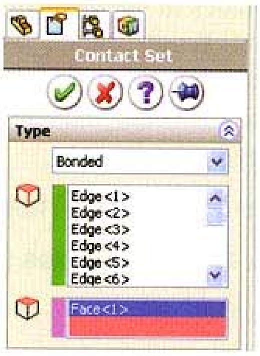
Use default mesh control parameters.

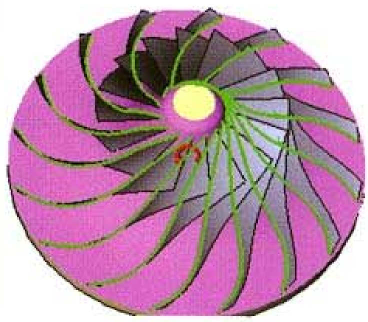
Even though we could mesh the model now, we still must assure that shell elements are properly connected to solid elements. Without that, the finite element model would consist of seventeen detached pieces: sixteen blades meshed with shells elements and solid core meshed with solid elements.

25 Define connection between shells and solids.

To bond shell elements to solid elements we define local contact conditions.

Right-click the `Contact/Gaps` folder and select **Define Contact Set**.





Under contact **Type**, select **Bonded**.

In the **Source** list, select edges of all blades contacting the solid body.

In the **Target** list, select the face of solid geometry contacting the edges.

Click **OK**.

26 Set Global contact.

Right-click the `Contact/Gaps` folder and select **Set Global contact**.

Under **Touching faces**, select **Bonded (No clearance)**.

Under **Options**, select **Incompatible mesh**.

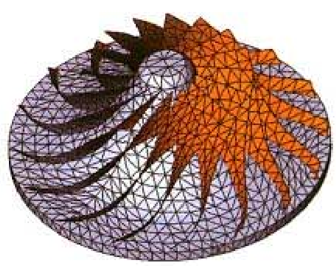
Click **OK**.

Note

The specification of the incompatible mesh here is only for the sake of completeness. Whenever a mixed mesh is specified, the bonding will be incompatible by default.

27 Create Mesh.

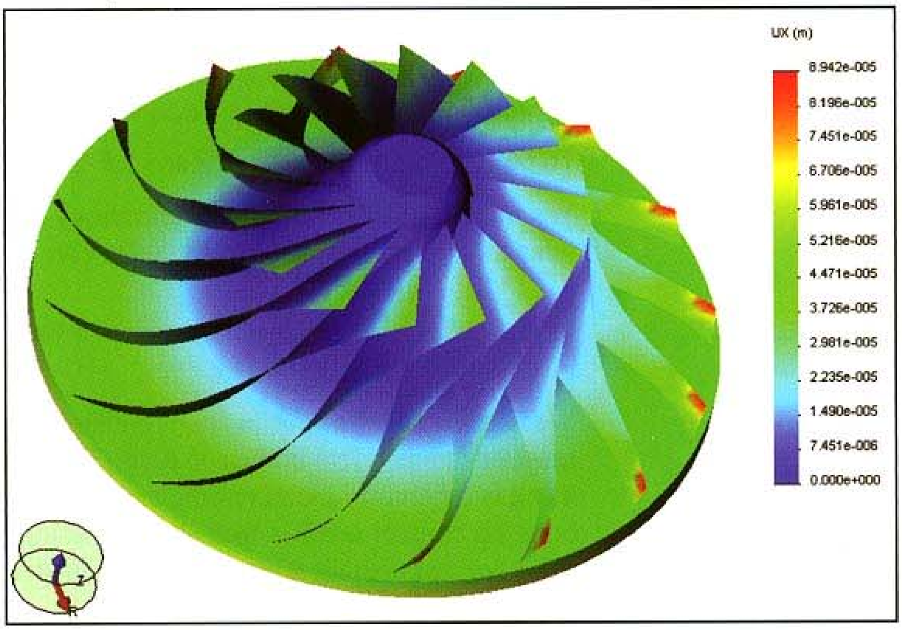
You may now create a **High** quality mesh with the default **Element size** of **10.58 mm.**



28 Run analysis.

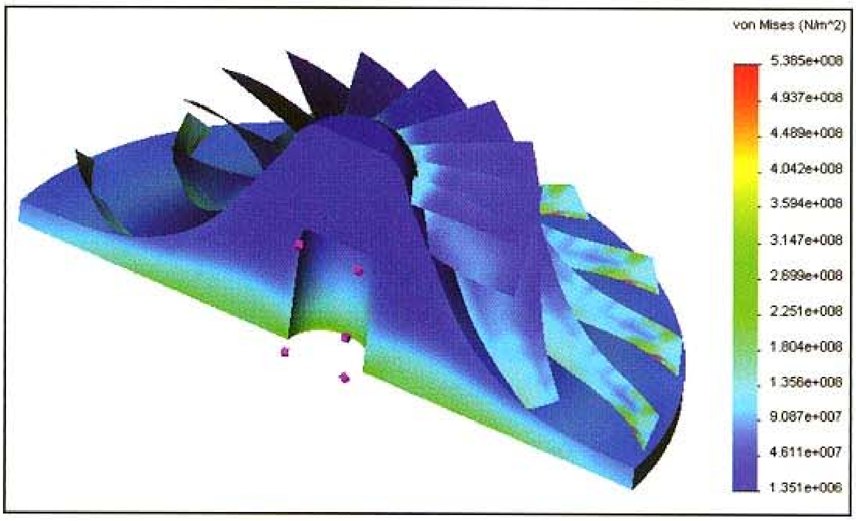
When running the analysis, use the **Direct Sparse** solver, which is faster than iterative solvers (FFEPlus) for solving mixed mesh models of this size.

29 Plot displacement results.



To view radial displacement results, construct a plot showing **UX component** displacement. Use the `Axis1` of the model as a reference so **UX** becomes the radial component of displacement.

30 Plot `von Mises stress`.



To plot von Mises stress we don't need to use any reference geometry because von Mises stress is a scalar stress measure.

To better picture stress results around the support, use a section view.

We can observe the stress concentrations near the end of the blades where they are connected to the solid core.

Summary

In this lesson we practiced and discussed the issues related to mixed meshing in COSMOSWorks.

We learned that the entities (parts) intended for the shell and solid mesh must be independent bodies or parts. The global bonding in mixed mesh is inherently incompatible and must be accompanied by local bonding contact sets for each shell/solid interface. However, solid-to-solid bonded contact does not need to be defined locally.

We also became familiar with the concept of compatible and incompatible meshing in solid and mixed mesh models.

Lesson 9
Vehicle Suspension Analysis Using Design Scenarios

Objectives

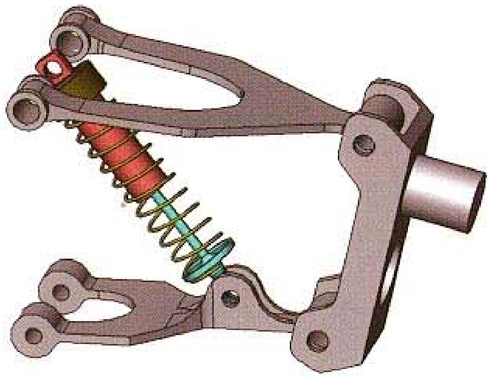
Upon successful completion of this lesson, you will be able to:

- Understand and use the Design Scenario feature to analyze trends when specific parameters are varied.
- Find optimum value of some design parameters

Project Description

The vehicle suspension assembly shown in the figure below can be subjected to a multitude of loading variations during its operating conditions. In this lesson, the assembly will be analyzed when subjected to the following four conditions:

- stationary vehicle
- vehicle moving at constant acceleration on a smooth road
- vehicle moving on a bumpy road
- vehicle moving at a constant speed on a smooth road and turning on banked road



All of the suspension components are manufactured from Alloy Steel. The goal of the analysis is also to adjust the thickness of the lower `arm` to an optimal value.

Suspension Design: Multiple load cases

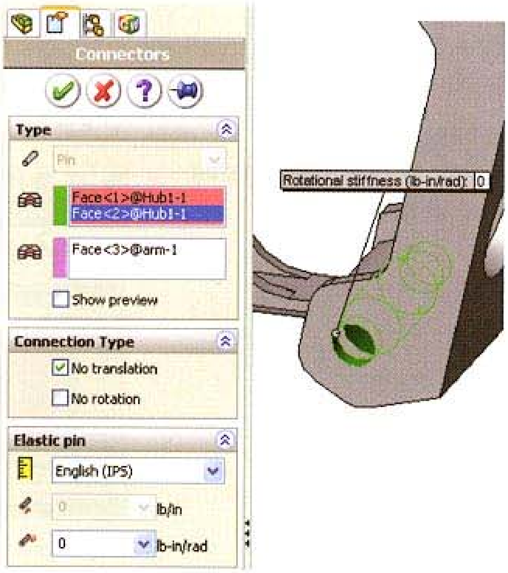
1 **Open assembly `suspension.sldasm`.**
This assembly is located in the `Lesson 9` folder

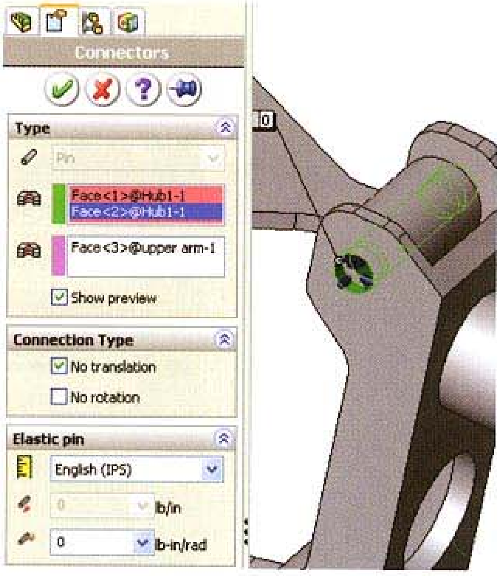
2 **Set COSMOSWorks options.**
Set the global system of units to **English (IPS)** and the units of **Length** and **Stress** to **in** and **psi**, respectively.

3 **Create new study.**
Create a new **Static, Solid mesh** study named `Multiple loads`.

4 **Suppress `Front Spring` component.**
The component will be replaced by an elastic spring connector.

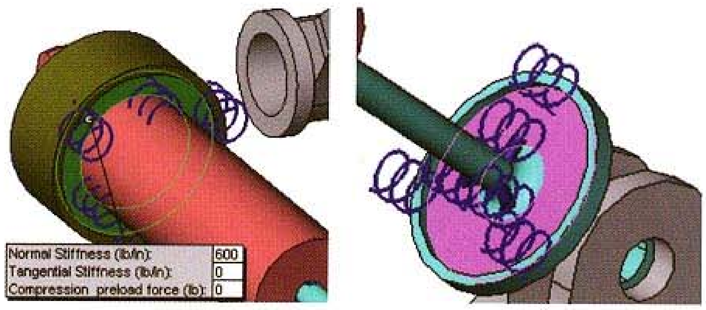
5 **Update components.**
Right-click on the `Multiple loads` study and select **Update All Components**.



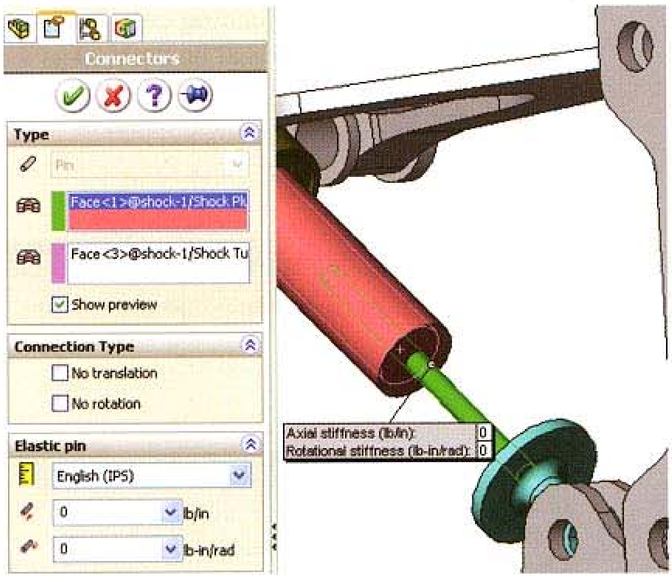


10 Define a spring connector.

As shown in the figure, define a spring connector with the **Total Normal stiffness** of **600 lb/in** between the `Front Shock Clamp` and the `Shock Plunger`.



11 Restrain the motion of the `Shock Plunger`.



Define a **Pin** connector between the outer cylindrical face of the `Shock Plunger` and the inner cylindrical face of the `Shock Tube`. Make sure that both the **No translation** and **No rotation** fields are deselected.

This condition is required to ensure the correct relative displacement between the `Shock Plunger` and the `Shock Tube`. Without this condition a **No penetration** contact would be required.

Design scenarios

Design scenarios can be conveniently used to analyze an assembly in which the loads, geometry or material constants are to be treated as design variables. Results, such as displacements or stresses, can then be graphed as functions of the design variables.

A design scenario is defined in two steps:

First, a list of the parameters (design variables) must be specified by right-clicking on the `Parameters` folder. A multitude of parameter types is available: loads, geometrical features, material constants, and others.

Second, a design scenario in which sets (combinations) of the parameters along with their numerical values are created by right-clicking on the `Design Scenario` icon.

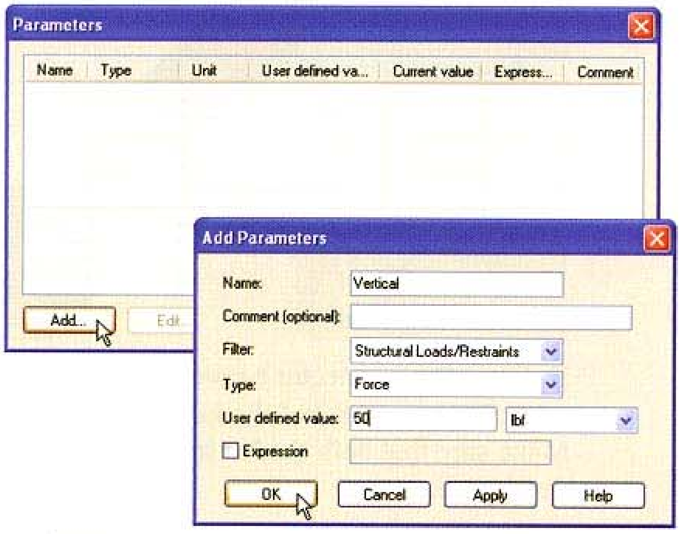
Note

In some cases, such as when a load or a material constant is used as a parameter, a definition of the load or of the material constant must be linked to the corresponding parameter. This intermediate step will be practiced in this lesson.

12 Specify load parameters.

In this lesson, multiple loading conditions corresponding to various vehicle travel scenarios will be defined as a design scenario.

Right-click on the `Parameters` folder and select **Edit/Define**.



In the **Parameters** dialog window click **Add**.

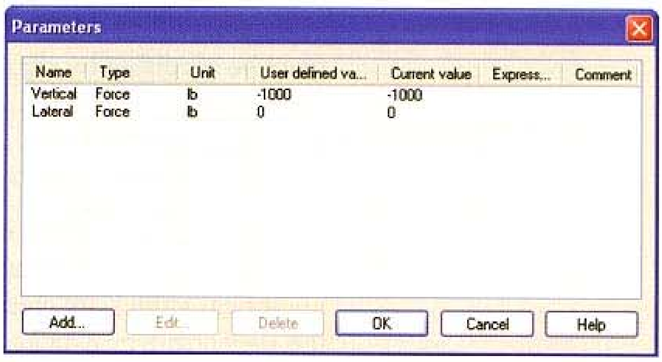
In the **Add Parameters** dialog window, enter **Vertical** as **Name**. Select **Structural Loads/Restraints** under **Filter** and **Force** under **Type**.

Under **User defined value,** enter **50 lbf**.

Click **OK** to close the **Add Parameters** dialog window.

Similarly, define the second parameter named **Lateral** with **0 lbf** as the **User defined value**.

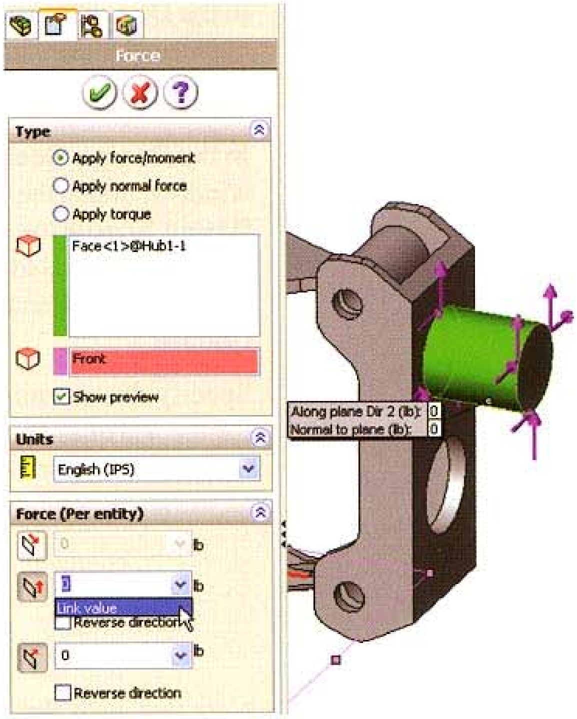
Two **Force** parameters have been defined in this step: **Vertical** and **Lateral**.



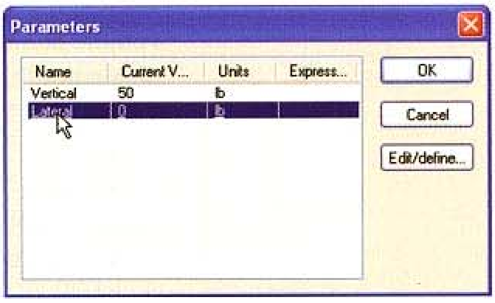
13 Define the Vertical and Lateral forces.

Apply force on the cylindrical face, as indicated in the figure. Use the `Front` plane as a reference.

In the **Along plane Dir2** field, select **Link value**.



In the **Parameters** dialog window, select **Lateral** to link this parameter to the corresponding component of the force.

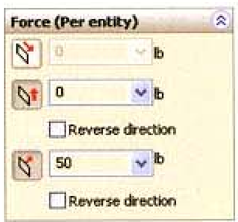


Note

Notice that the user defined value of 0 lb is shown in the appropriate field with a distinct background color.

Repeat the procedure and link the **Normal to Plane** force component to the parameter **Vertical**.

Both load components are now linked to the design scenario parameters.



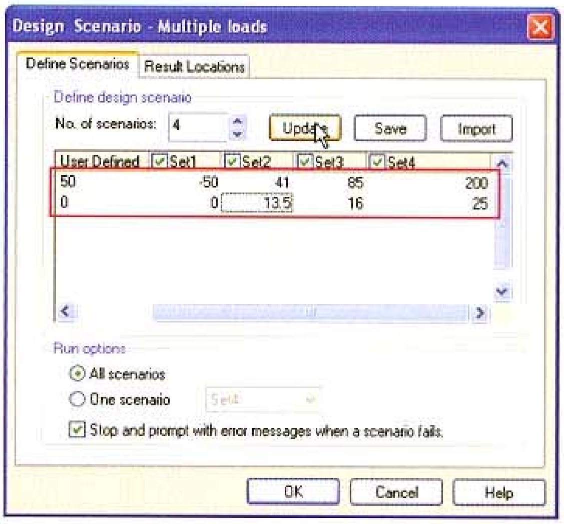
14 Define Design Scenario.

In the Design Scenario definition, the specified parameters are assigned numerical values and their combinations (sets) are created.

Right-click on the `Design Scenario` folder and select **Edit/Define**.

In the **Design Scenario** window, under the **Define Scenarios** tab, increase the **No. of scenarios** to **4** and click **Update.**

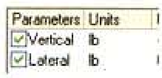
Specify the magnitudes for the **Vertical** and **Lateral** force parameters, as shown in the figure.



Note

Note that each **Set** represents a load case corresponding to the specific vehicle travel conditions: Set 1 corresponds to the loading when a vehicle is stationary, Set 2 to a vehicle moving at constant acceleration on a smooth road, Set 3 to a vehicle moving on a bumpy road, and Set 4 to a vehicle moving at constant speed on a smooth curving and banking road.

Activate both parameters by checking the boxes in the **Parameters** column.

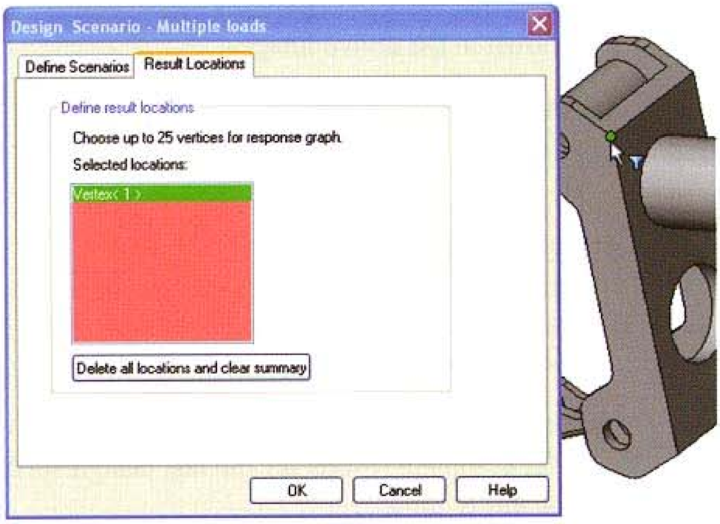


Under **Run** options, select **All scenarios** (if desired, only selected scenarios may be requested).

Design Scenario results

The Design Scenario feature automatically generates and runs multiple studies corresponding to each Set. As the amount of data can easily become excessive, certain storage restrictions have been imposed:

- For each study (design scenario Set), only the assembly global extreme values and the results at selected vertices for the von Mises, P1, P2 and P3 stresses, resultant displacements, stress intensity, and equivalent strains are stored.
- For the last Set, the full results for the entire model are available and can be plotted.

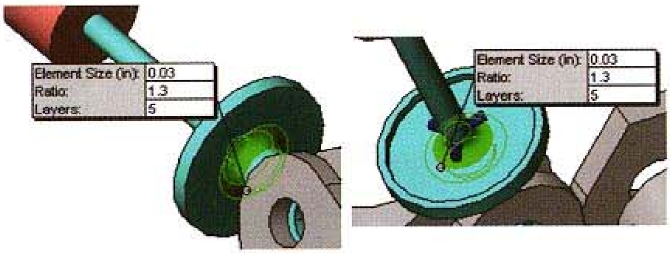


Under the **Result Locations** tab, select the vertex indicated in the figure. This vertex will be used to compare the hub displacements from all the design sets.

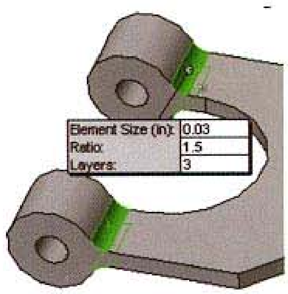
Click **OK** in the **Design Scenario - Multiple loads** dialog window.

15 Refine mesh at higher curvature regions.

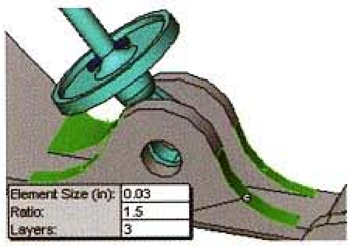
Apply mesh controls at the two fillets on the `Shock Plunger`.



Apply mesh control at the four neck fillets on the lower `arm`.



Apply mesh control to the four fillets on the lower `arm` located close to the `shock x arm` pin connector.



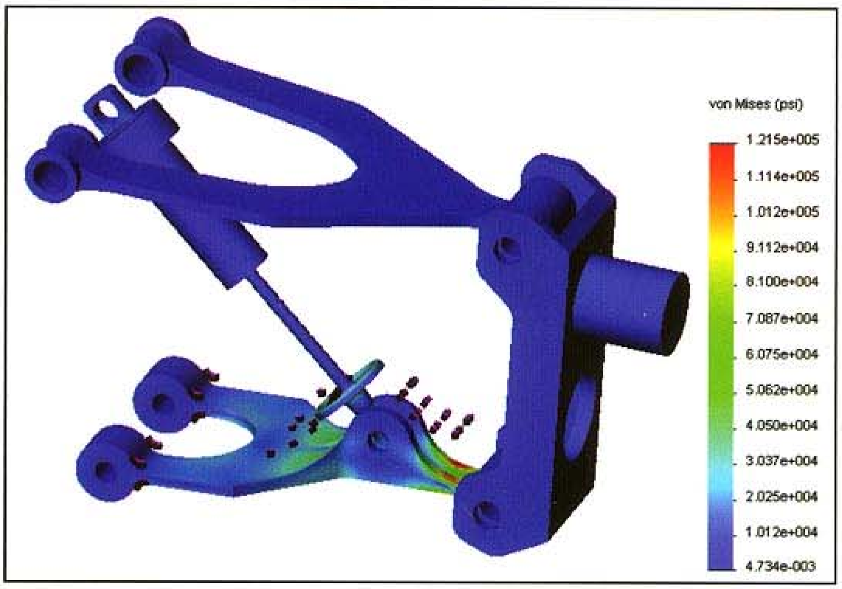
Note that the local **Element sizes**, **Ratio** and **Layers** parameters are shown in the above images.

16 Mesh the assembly.
Create a **High** quality mesh with the default global **Element size** of **0.147 in**.

17 Run Design Scenario.
Right-click on the `Multiple loads` study folder and select **Run Design Scenario**.

18 Plot stress results for the last Set.
The complete results for the last design scenario set are available for the entire model.

Define a **von Mises** plot at the **Automatic** deformation magnification scale.



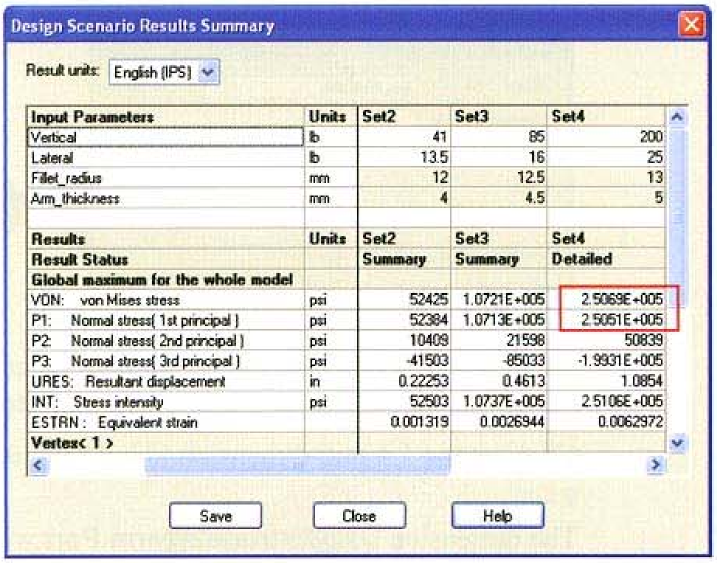
We see that the von Mises stresses in Set 4 exceed the yield strength of Alloy Steel (90 ksi).

Animate the plot to observe the deformation of the shock assembly.

19 Analyze global extreme result values.

Right-click on the `Results` folder and select **Show Design Scenario Summary**.

A dialog window shows the global extremes (regardless of the location) and the results at the selected locations.



Input Parameters	Units	Set2	Set3	Set4
Vertical	lb	41	85	200
Lateral	lb	13.5	16	25
Fillet_radius	mm	12	12.5	13
Arm_thickness	mm	4	4.5	5
Results	**Units**	**Set2**	**Set3**	**Set4**
Result Status		**Summary**	**Summary**	**Detailed**
Global maximum for the whole model				
VON: von Mises stress	psi	52425	1.0721E+005	2.5069E+005
P1: Normal stress(1st principal)	psi	52384	1.0713E+005	2.5051E+005
P2: Normal stress(2nd principal)	psi	10409	21598	50839
P3: Normal stress(3rd principal)	psi	-41503	-85033	-1.9931E+005
URES: Resultant displacement	in	0.22253	0.4613	1.0854
INT: Stress intensity	psi	52503	1.0737E+005	2.5106E+005
ESTRN : Equivalent strain		0.001319	0.0026944	0.0062972
Vertex< 1 >				

We can again see that the last study (Set 4) reports the same largest magnitude for the von Mises stresses (250 ksi). The resulting displacements and equivalent strains reaching the values of 1 in and 0.27% are also the largest in Set 4. Thus, we can conclude that the last set, Set 4 (corresponding to the loading when the vehicle travels at a constant speed on a smooth, curved and banked road), represents the most extreme case and the shock assembly will be designed to withstand this loading.

Also, note that the four design scenario sets could be easily specified as four unique studies. The advantage of the design scenario becomes more apparent if the number of Sets (in this example, load cases) increases.

Suspension Design: Geometry modification

Since Set 4 was identified as the worst load combination when the model exhibits yielding, the assembly geometry will now be modified with the help of design scenarios. A safety factor of 1.3 on von Mises stresses and the maximum resultant displacement of the `hub` component equal to 1 in will be required.

20 Create new study.

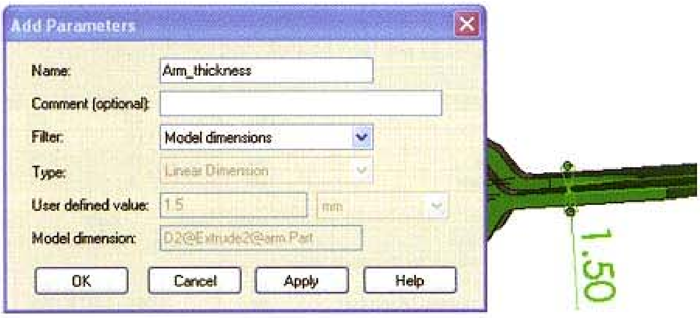
Copy the study `Multiple load` into a new study named `Geometry design`.

21 Define geometrical parameter.

Define an additional geometrical parameter that will be used in this study.

Right-click on the `Parameters` folder and select **Edit/Define**.

In the **Parameters** dialog window click **Add**.



In the **Add Parameters** dialog window enter **Arm_thickness** as a **Name** and select **Model dimensions** as **Filter**.

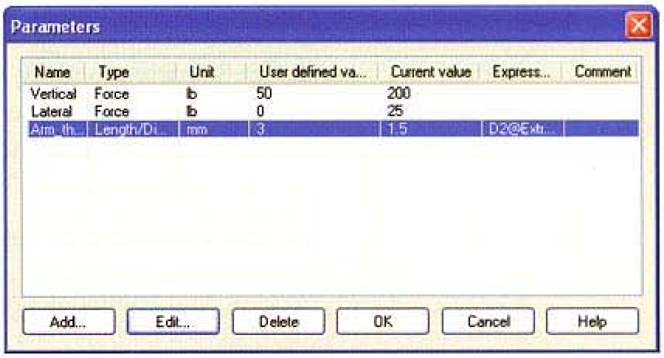
Click on the 1.5 mm dimension identifying the thickness of the lower `arm`.

The dimension **D2@Extrude2@arm.Part** will then be shown in the **Model dimension** field

Click **OK** to close the **Add Parameters** window.

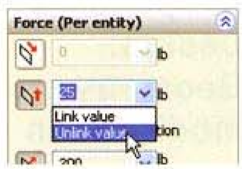
A total of three parameters are now defined.

Click **OK** to close the **Parameters** dialog window.



22 Modify the load.

In the study `Geometry`, edit the force definition and unlink the values of the force components from the parameters.



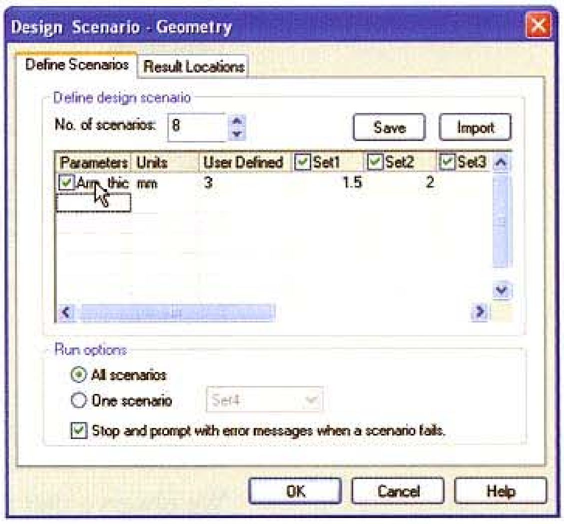
Enter a value of **25 lb** in the **Along plane Dir2** field and **200 lb** in the **Normal to plane** field. This corresponds to the worst loading case in which the vehicle travels at a constant speed on a smooth, curved and banked road (Set 4 in the previous design scenario).

23 Modify the design scenario.

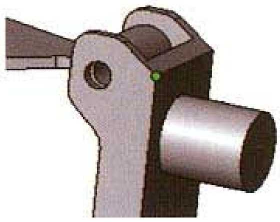
Right-click on the `Design scenario` folder and select **Edit/Define**.

Under the **Design Scenario** tab, specify eight Sets with the following values for the parameter **Arm_thickness** used in this study:

1.5, 2, 3, 4, 5, 6, 7, and 8mm.



Under the **Results Locations** tab, make sure that the indicated vertex is selected in the **Selected locations** field.



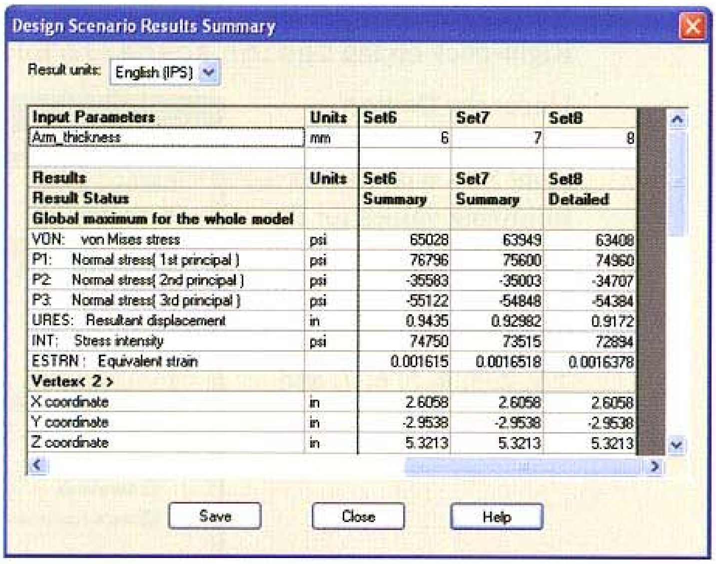
24 Mesh the model.

Create a **High** quality mesh with the default global **Element size** of **0.147 in**.

25 Run design scenario.

26 Review global extremes of the results.

Right-click on the `Results` folder and select **Show Design Scenario Summary**.



Design Scenario Results Summary

Result units: English (IPS)

Input Parameters	Units	Set6	Set7	Set8
Arm_thickness	mm	6	7	8
Results	**Units**	**Set6**	**Set7**	**Set8**
Result Status		**Summary**	**Summary**	**Detailed**
Global maximum for the whole model				
VON: von Mises stress	psi	65028	63949	63408
P1: Normal stress(1st principal)	psi	76796	75600	74960
P2: Normal stress(2nd principal)	psi	-35583	-35003	-34707
P3: Normal stress(3rd principal)	psi	-55122	-54848	-54384
URES: Resultant displacement	in	0.9435	0.92982	0.9172
INT: Stress intensity	psi	74750	73515	72894
ESTRN: Equivalent strain		0.001615	0.0016518	0.0016378
Vertex< 2 >				
X coordinate	in	2.6058	2.6058	2.6058
Y coordinate	in	-2.9538	-2.9538	-2.9538
Z coordinate	in	5.3213	5.3213	5.3213

Save Close Help

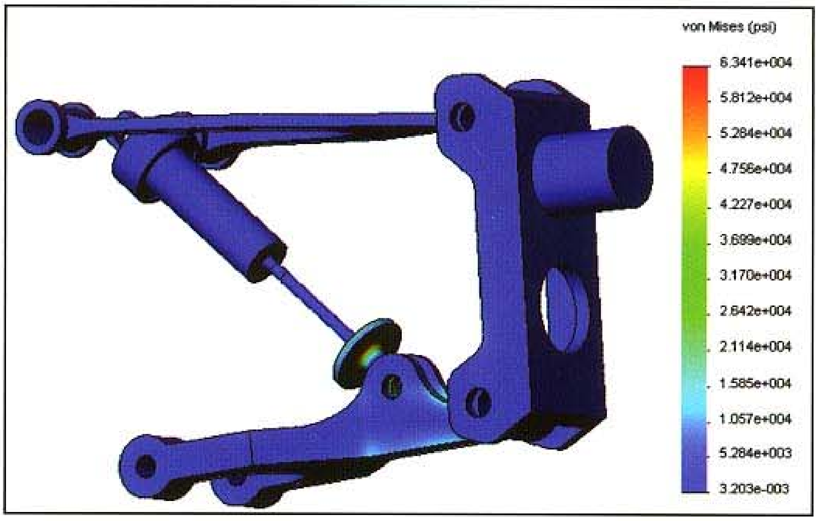
We observe that the maximum value of the von Mises stresses was reduced to 63.4 ksi. This value satisfies our requirement on the minimum von Mises stress factor of 1.3 (90 ksi / 1.3 = 69 ksi).

The maximum resultant displacement remained nearly unchanged at 0.92 in.

Close the **Design Scenario Results Summary** window.

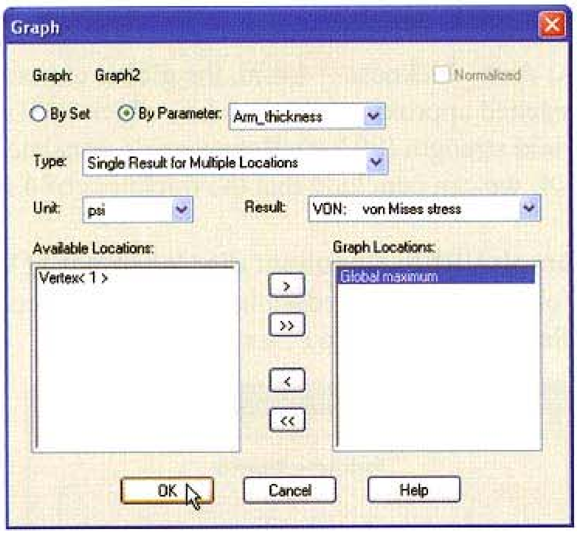
27 Display von Mises plot.

Show the von Mises stress plot for the last set (Set 8) when the thickness of the lower `arm` was 8 mm.



We can see the already indicated maximum von Mises stress of 63 ksi. Notice, however, that the location of the maximum stress shifted to the fillet on the plunger. Varying model dimensions cause the stresses to redistribute as the stiffness of the components change relative to each other.

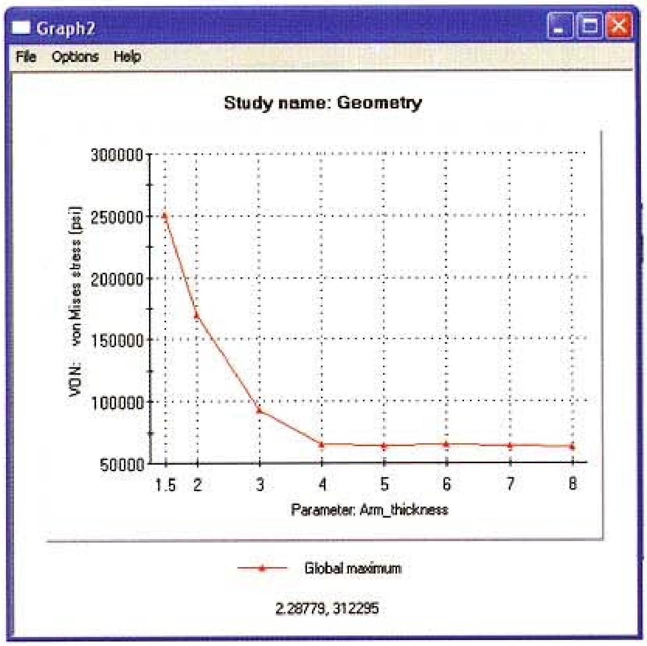
28 Graph the global extreme for von Mises stresses.
Right-click on the `Results` folder and select **Define Design Scenario Graph**.



For the ordinate of the graph, select **By Parameter** (**Arm_thickness** will appear in the selection field as it is the only parameter in this study). Under **Result**, select **VON: von Mises stress** and **psi** for the **Units**.

In the **Graph Locations** field, select **Global maximum**.

Click **OK** to show the graph.



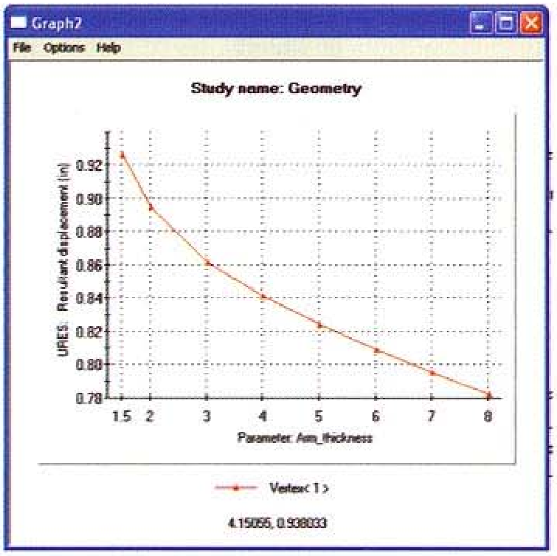
The above graph shows the variation of the global extreme value of the von Mises stress with the thickness of the lower `arm`.

We observe that any increase in the thickness of the lower `arm` above 4 mm does not deliver any substantial decrease in the global von Mises stress magnitude (the maximum value location shifted from the lower `arm` to the `Plunger`). We can, therefore, conclude that the thickness of 4 mm is optimal.

At Arm_thickness = 4 mm, the global extreme of von Mises stress reached approximately 64.3 ksi, which is 71.4% of the Alloy Steel yield strength (90 ksi). Because this translates in a factor of safety of 1.4, we can conclude that the thickness of 4 mm is optimal.

29 Graph URES: Resultant displacement at `Vertex 1`.

Following the procedure in the previous step, graph **URES: Resultant displacement** at `Vertex 1`.



We can observe that the increase in the thickness of the lower `arm` does have a substantial effect on the resultant displacement at `Vertex 1`, even for higher values. At Arm_thickness = 4 mm, the resultant displacement is equal to 0.84 in., which is less than 1 in., and the design therefore passes this criterion as well.

30 Save and close the assembly.

Note

Alternatively, we could analyze the variety of material combinations used for different parts in this assembly. Using the identical procedure that was practiced in this lesson, we would first define the parameters for the material properties (such as Young's modulus or Yield strength). Then, we would define a design scenario in which the combinations as well as the numerical values of the material parameters would be specified.

Summary

This lesson introduced and practiced the Design Scenario feature that allows the user to study various trends in the design when specific design parameters are defined. This feature has many practical applications, some of which were practiced in this lesson. Namely, it was used to study various load cases simulating various travel conditions of a small vehicle and to find an optimum value of the thickness of one of the suspension components.

The design scenario study is defined in two steps:

First, a list of the parameters (design variables) must be specified by right-clicking on the `Parameters` folder. A multitude of parameter types is available: loads, geometrical features, material constants, and others.

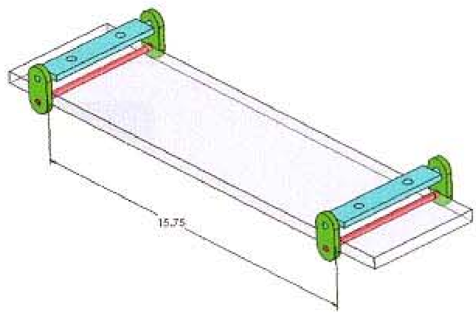
Second, a design scenario in which sets (combinations) of the parameters along with their numerical values are created by right-clicking on the `Design Scenario` icon.

It is apparent that if an optimum combination of a larger number of design parameters is desired, the process may be rather lengthy and cumbersome. In such case, a full automatic Optimization modulus of COSMOSWorks must be used.

Exercise 6: Analysis of a Platform Using Design Scenarios

Project Description

A rectangular platform made of a plastic material, Nylon 6/10, is supported by two steel rods protruding through the platform width.

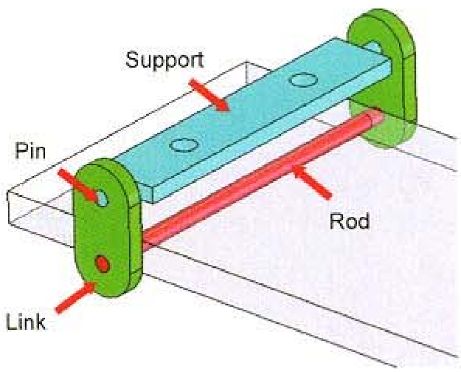


Note that the rods are suspended by short links, which themselves are pin-supported. The distance between the pins attached to the platform may change when the platform experiences deformation.

This type of support makes it possible to use linear analysis for the study of deflections and stresses of the platform.

The platform assembly is subjected to an acceleration of 100 G.

We wish to find the distance between the two steel pins that minimizes the platform deflection.



Platform analysis

1 Open part.
Open the SolidWorks part file `platform`.

Note

We assume that the steel rods are rigid and that we are not interested in the contact stresses between the pins and the platform. These assumptions allow us to exclude the rods from the analysis model.

Since the rods are represented by properly applied supports, the analysis is conducted using the SolidWorks part file `platform`, rather than the assembly file `platform assembly`.

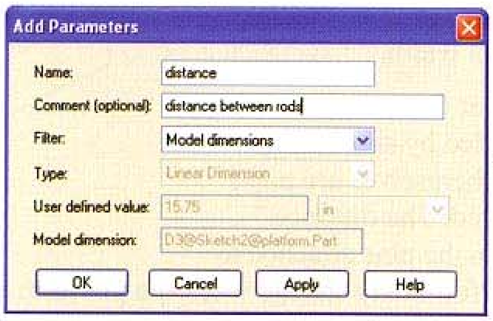
2 Show feature dimensions.
In the Solidworks FeatureManager, right-click `Annotations` and select **Show Feature Dimensions**.

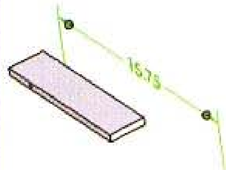
Note

In this exercise, we take advantage of platform double symmetry to analyze one quarter of the model.

3 Make symmetry cut.
Unsuppress the `double symmetry` feature of SolidWorks.

4 Define parameters.
Define a **Linear dimension** parameter.

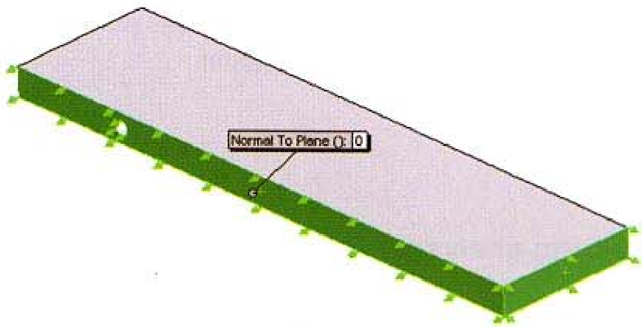




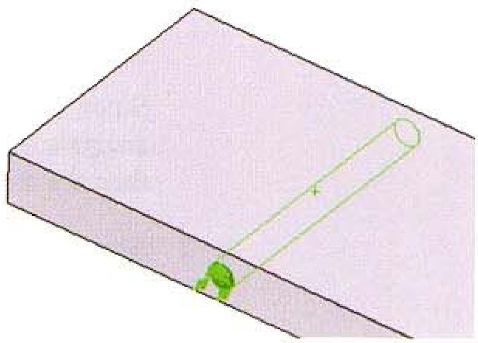
Select a **15.75 in** dimension that defines the distance between the two hinges.

5 Create a study.
Create a **Static**, **Solid mesh** study named `100G`.

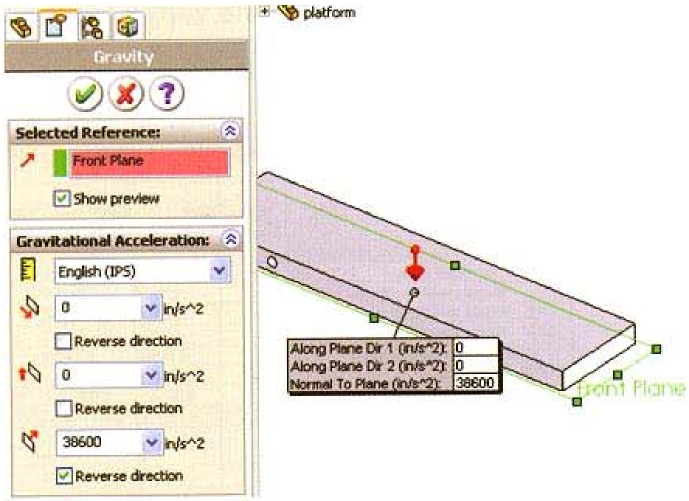
6 Define symmetry restraints.
Apply symmetry boundary conditions to the two faces created by the cut.



7 Define restraint to simulate rod support.
Apply a **Hinge** restraint to the cylindrical face as indicate in the figure.



8 Apply gravity load.
To apply a gravity load, right-click `Load/Restraint` and select **Gravity**.



Select the `Front` plane as the reference to define the direction of gravitational acceleration.Make sure the **Unit** field is set to **English (IPS)**.

Enter **38600** (this value is one hundred times the gravitational acceleration) in the direction normal to the `Front` plane. Load orientation is controlled by the +/- sign.

Click **OK**.

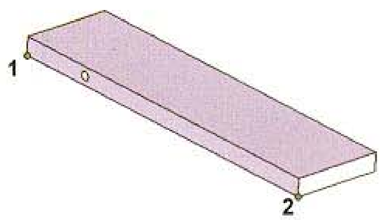
Note

Mass density is a required material property when using gravity loading.

9 **Define design scenario.**

Define a design scenario with 10 Sets and the following values for the parameter **distance**: **19**, **17**, **15**, **13**, **11**, **9**, **7**, **5**, **3**, and **1 in**.

For **Results Locations**, select the two indicated vertices (number 1 at the edge of plate and number 2 in the center of the bottom face).

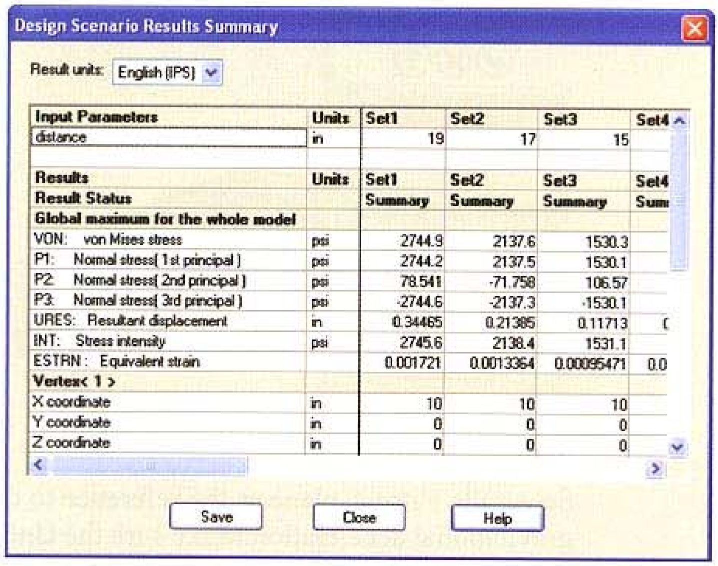


10 **Mesh the model.**

Create a **High** quality mesh with the default **Element size** of **0.231 in** and the **Automatic transition** feature on.

11 **Run design scenario.**

12 View global extremes and vertex location results.

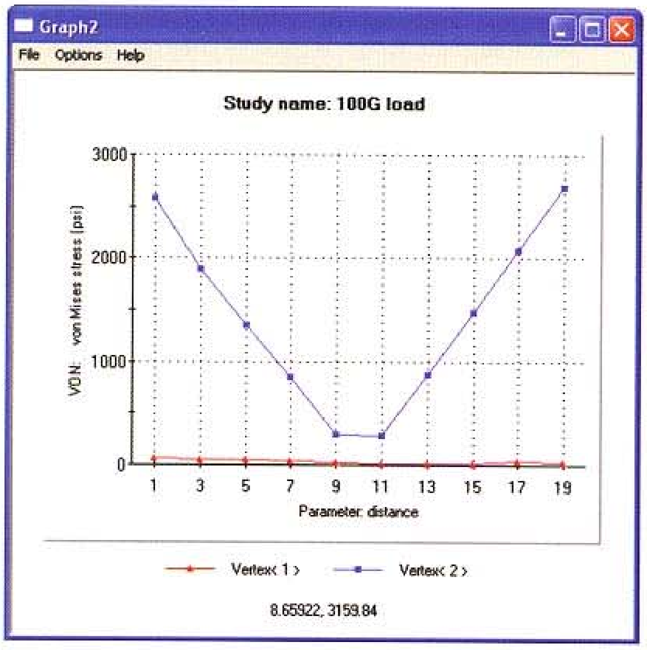


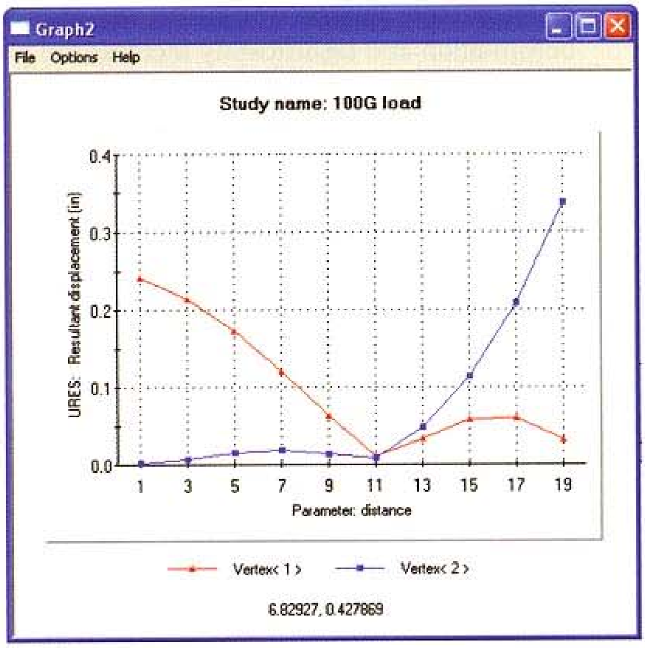
Input Parameters	Units	Set1	Set2	Set3	Set4
distance	in	19	17	15	
Results	**Units**	**Set1**	**Set2**	**Set3**	**Set4**
Result Status		**Summary**	**Summary**	**Summary**	**Sum**
Global maximum for the whole model					
VON: von Mises stress	psi	2744.9	2137.6	1530.3	
P1: Normal stress(1st principal)	psi	2744.2	2137.5	1530.1	
P2: Normal stress(2nd principal)	psi	78.541	-71.758	106.57	
P3: Normal stress(3rd principal)	psi	-2744.6	-2137.3	-1530.1	
URES: Resultant displacement	in	0.34465	0.21385	0.11713	(
INT: Stress intensity	psi	2745.6	2138.4	1531.1	
ESTRN: Equivalent strain		0.001721	0.0013364	0.00095471	0.0
Vertex< 1 >					
X coordinate	in	10	10	10	
Y coordinate	in	0	0	0	
Z coordinate	in	0	0	0	

Examine the results, and note that the results are available for the `Vertex1` and `Vertex2` locations as well.

13 Graph results at Vertex1 and Vertex2 locations.

Define design scenario graphs for the variation of the von Mises stress and the resultant displacement against the **distance** parameter.



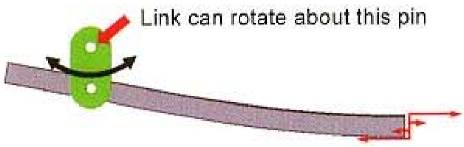


We observe that both the von Mises stresses and the resultant displacements are minimized when the distance between the two supporting rods is 11 in. (Set 5). The corresponding values are 283 psi and 0.01 in.

Limitations of Linear Analysis

At the beginning of this exercise, we stated that the platform was suspended by floating links. The links themselves are pin-supported and rotate about supporting pins. For this reason, the distance between the rods may change when the platform experiences deformations under the prescribed load.

Consequently, in the assumption of small displacements, a platform suspended by floating links does not develop any tensile stresses. It resists the load only with bending stresses.



If the links are rigidly supported and the rods are unable to move closer together, tensile stresses develop in addition to bending stresses.

These tensile stresses, also called membrane stresses, are the result of deformation and significantly increase the platform stiffness.

Question: What does this have to do with linear analysis?

Answer: Linear analysis assumes that the structure stiffness does not change with deformation, and the solution is based on the original stiffness calculated before any deformations occurred.

Therefore, linear analysis cannot account for additional stiffness created by membrane stresses that develop during the deformation process.

Even if we intended to model rigid hinges, the results would still have pertained to floating hinges and the platform stiffness would have been underestimated.

To differentiate between floating and rigid hinges we use nonlinear geometry analysis, which is available in COSMOSWorks Advanced Professional.

Lesson 10
Static Analysis of a Support Bracket

Objectives

Upon successful completion of this lesson, you will be able to:

- Use the h-adaptive solution method
- Use the p-adaptive solution method
- Compare results obtained using h-adaptive and p-adaptive solution method
- Use symmetry boundary conditions
- Use the Graph tool

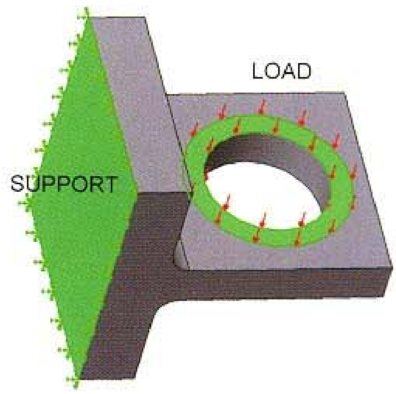
Project Description

A hollow cantilever bracket is supported along the face at the back side. The SolidWorks model dimensions are in inches; therefore, we use the IPS system of units in this exercise.

A load of 5,000 lb is uniformly distributed to the split face that surrounds the cylindrical hole.

We must determine the location and maximum magnitude of von Mises stresses.

1 **Open part.**
Open the SolidWorks part file `support bracket`.



2 **Unsuppress features.**
In the SolidWorks FeatureManager window, unsuppress the feature named `symmetry cut`.

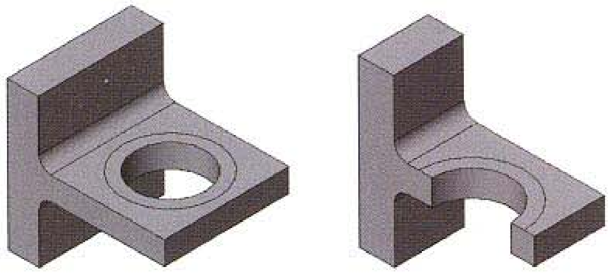
Geometry Preparation

Note that the bracket geometry has been defeatured to make meshing easier; the external cosmetic fillets have been suppressed. While these details do not complicate this model to the point of preventing us from meshing or solving it, we use the model with suppressed features to emphasize the fact that defeaturing is often necessary for more complex models.

Symmetry

Due to the symmetry of the bracket geometry, loads, and supports, we can simplify the finite element model by analyzing only one half of its geometry. We can simulate the missing half with symmetry boundary conditions. These conditions are defined on the faces created by the cut in the plane of symmetry.

By analyzing half of the model, we radically depart from the CAD geometry. We decide to analyze half of the model, not because the full model is too complex to handle, but to practice the use of symmetry boundary conditions. Using symmetry boundary conditions is a very useful modeling technique to simplify models when it is really required.



You are encouraged to use the full model later to perform the same analysis.

3 **Define a study.**
Define a study named `standard` (**static analysis, solid mesh**).

Note

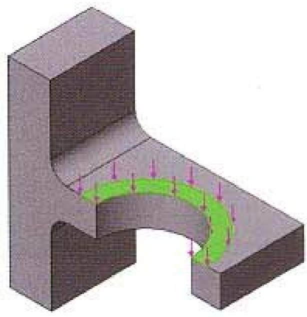
This study will provide results we need as a reference when comparing different solution methods. The study name `standard` reflects the fact that we use the "regular" solution method where mesh does not change during the solution process. This is how we have solved all previous lessons in this training manual.

We do not need to define material properties because the material (AISI 304) was defined earlier in SolidWorks and has been transferred automatically to COSMOSWorks.

Note that you can define the load as a force or pressure as desired.

To define loads and restraints do *one* of the following:

4 **Apply load as a force.**
Apply **2,500** lb. force to the split face around the hole. Note that only one half of the force magnitude must be applied because the model represents only one half of the geometry.



In the **Force** PropertyManager, select the **Apply normal force** check box.

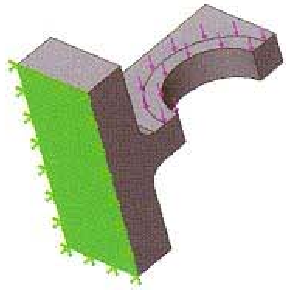
Or...

5 **Apply load as a pressure.**
Right-click `Load/Restraint` and select **Pressure**. The total area of the split face around the hole is 1.546 in^2, therefore, the pressure magnitude would be 1617 psi.

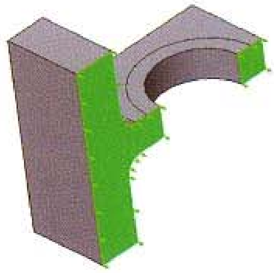
In the **Pressure Value** PropertyManager, enter **1617 psi** as the pressure magnitude. Because pressure is defined as force per unit of area, the pressure magnitude does not need to be divided in half when applied to one half of the model.

Under **Pressure Type**, select the **Normal to selected face** check box.

6 Apply Immovable restraints to the face at the back.



7 Apply Symmetry Boundary Conditions.
Now we apply symmetry boundary conditions to the faces created by the cut in the plane of symmetry.



To apply symmetry boundary conditions:

Right-click `Load/Restraint` and select **Restraints**.

From the list of available restraint types, select **Symmetry**.

Click **OK**.

8 Mesh the `standard` study,
Create a **High** quality mesh with the default element size.

9 Run the analysis.
Run the solution for the `standard` study.

To verify that the symmetry boundary conditions work as expected, animate one of the results plots. While the model deforms, verify that the faces with symmetry boundary conditions applied to them remain flat and there is no movement of these faces in the direction normal to the plane of symmetry.

Types of Mesh adaptive solutions

The use of symmetry boundary conditions is a subtopic of this exercise. Our main learning objective is to introduce two new solution methods.

The results of the `standard` study serve as a basis for comparison between three different solution methods used in this lesson:

1. Standard solution (the one we have just completed)
2. **h-adaptive** solution
3. **p-adaptive** solution

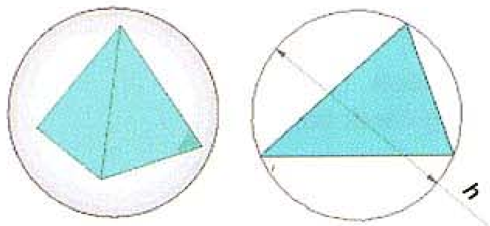
h-adaptive solution method

Before we explain the **h-adaptive** solution method, recall that any solution obtained using the Finite Element Analysis depends on how the analyzed model has been meshed.

We may re-phrase the above observation to say that the FEA data of interest depends on the choice of discretization. Thus, changing the mesh parameters (global or local mesh controls) will affect the FEA results. This is because different meshes (different choices of discretization) will cause different discretization errors.

Discretization errors can be estimated by making systematic changes to the mesh and studying the impact of these changes in the area of interest. This process is called "convergence process".

One way to make systematic changes to the mesh is to modify the element size through mesh refinement. Because ***h*** denotes the characteristic element size, the convergence process through mesh refinement is called "h convergence process". In this process, the size of the elements is gradually reduced.



You will recall that we have already conducted the h convergence process in `Lesson 1` and `2` in the COSMOSWorks Designer tutorial.

In `Lesson 1`, we refined the model uniformly, meaning the entire model was meshed with the same element size. That size was reduced in different studies. In `Lesson 2`, we used mesh controls to refine the mesh only in areas where we deemed it necessary.

The convergence process we conducted in `Lesson 1` and `Lesson 2` required us to define several studies with different meshes, run the analysis, and summarize the results. These were informative but rather tedious exercises.

Now, using **h-adaptive** solution method we will automate the h convergence process.

h-Adaptivity Study

10 Create new study for h-adaptive solution.
Copy study `standard` into a new study and name it `h adaptive`.

11 Set h-adaptive solution parameters.
Right-click `h adaptive` and select **Properties**.

Click the **Adaptive** tab.

Note

The **Adaptive** solution tab is available only for static analysis and solid mesh elements.

Under the **Adaptive method** option, select **h-adaptive**.

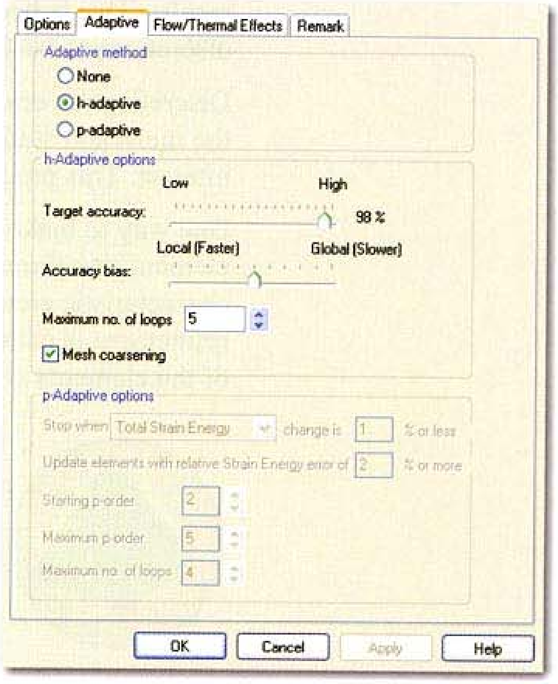
Under the **h-Adaptive options**, accept the default values for **Target accuracy**.

Also, accept the slider location at the center for **Accuracy bias**.

In the **Maximum no. of loops** box, enter **5**.

Select the **Mesh coarsening** check box.

Click **OK**.



h-adaptivity options

What is going to happen when we solve the study with the above settings?

COSMOSWorks will solve the same model several times, each time using more refined meshes. The mesh refinements will be performed automatically, no user intervention is required.

How many mesh refinements will be performed?

Considering that we have set the **Maximum number of loops** to **5**, COSMOSWorks will solve for the original mesh and then perform several other mesh refinements. Looping will terminate when **Target accuracy** is obtained or if the **Maximum number of loops** is reached. **Maximum number of loops** of **5** means that the solution may consist of a maximum of six steps: the original mesh and five refinements.

Target accuracy is the accuracy of strain energy norm (RMS strain energy) in the model. We set it at 98% which means that looping stops if the difference in the strain energy norm between the two consecutive loops drops below 2%.

Target Accuracy

The **Target accuracy** is based on the total strain energy in the model. This is a global measure of the discretization error. As such, it is largely insensitive to localized errors, even if those errors are high.

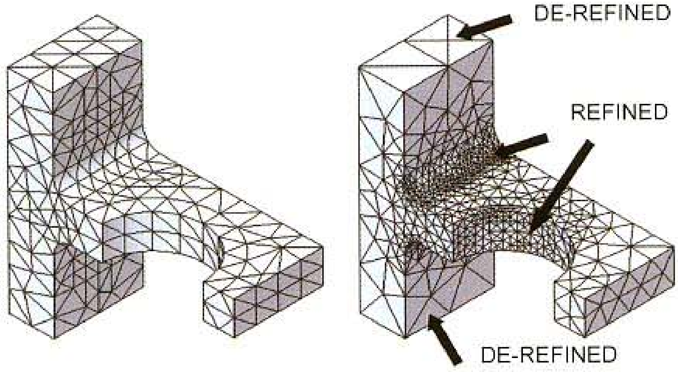
Accuracy Bias

To account for the local errors, looping is also controlled by an **Accuracy bias**.You can move the **Accuracy bias** slider to the left **(Local)** to instruct the program to concentrate on getting accurate peak stress results, meaning that local areas with high strain energy errors will receive "preferential treatment" (mesh will be highly refined in those areas). Or, you can move the slider to the right **(Global)** to instruct the program to compute overall accurate results with respect to lower strain energy errors. We do not have explicit control over the magnitude of local strain energy error.

From `Lesson 2` you will recall that stress singularity occur at the locations of concentrated forces and sharp re-entrant corners. The stresses at these locations diverge to infinity as smaller mesh elements are used.

Therefore, for models with such singularity, it is recommended to move the **Accuracy bias** slider to the right **(Global)**. This way high, but localized, strain energy errors will be ignored; the solver will not adjust mesh refinement pattern to reduce these errors.

Local Accuracy bias usually produces results faster than **Global Accuracy bias.**



ORIGINAL MESH

MESH AFTER COMPLETION OF H-ADAPTIVE SOLUTION

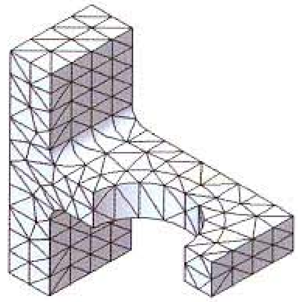
When **h-adaptive** solution is used, you can start with a coarser original mesh size. This mesh is a starting point and COSMOSWorks refines it as needed during the solution process. Additionally during the mesh refining process, the mesh may be "de-refined" if **Mesh coarsening** is selected, as it has been in our study.

The mesh may become coarser in some locations if the **h-adaptive** solver decides that the initial user defined mesh is "too fine", meaning it is excessively refined giving low stress gradients in these locations.

The mesh is not refined uniformly but only where needed to keep strain energy errors low. We may say that mesh adapts to the stress patterns. This gives the "adaptive" name to **h-adaptive** solution method.

12 Create mesh for `h-adaptive` study.

Mesh the model with a coarse mesh and the **Element size** of **0.5 in**. Make sure **High** quality elements are used.



This mesh is not acceptable for standard solution techniques because there are not enough elements to capture the complex stress gradients around the fillets.

13 Run the `h-adaptive` study.

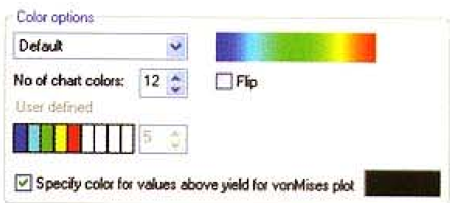
Run the solution for the `h adaptive` study. Notice that the solution progresses in steps corresponding to the number of mesh refinements.

To help re-visualize the stress results, the regions where the material yields will be plotted in a distinct color.

14 Activate distinct color for yielding regions.

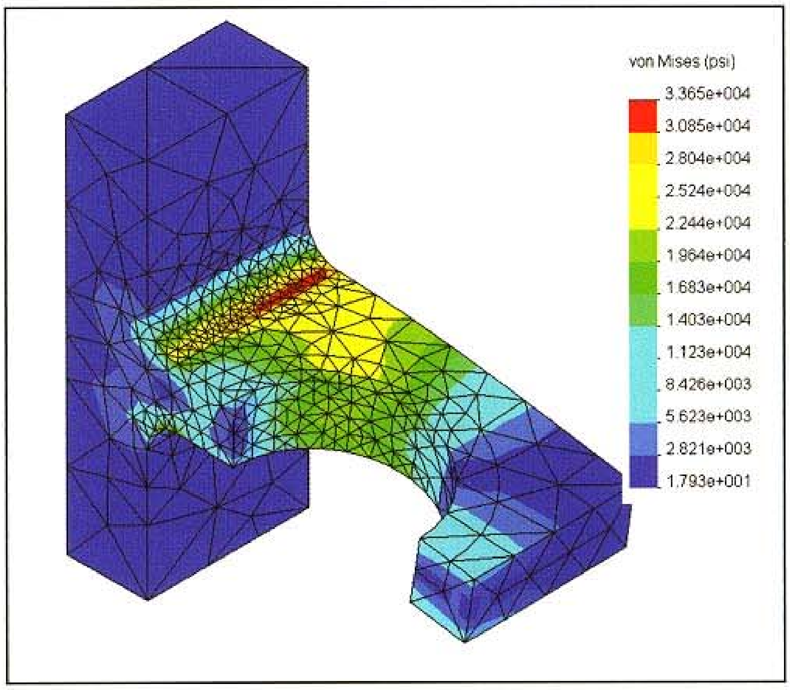
Right-click the `support bracket` part and select **Options**. Under the `Default Options (New Study)` folder, select the `Plot, Color Chart` folder.

Activate the **Specify color for values above yield for von Mises plot** feature. Note that this color is gray by default.



Click **OK**.

15 Plot Von Mises Stress.
Define a new von Mises stress plot. Under **Settings**, select **Mesh** for the **Boundary options**.



The stress plot shows the maximum von Mises stress of 33.2 ksi, which slightly exceed the yield strength of AISI 304 steel.

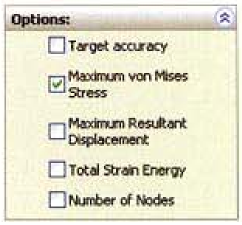
Note that the yielding regions are shown in a distinct color.

Displaying mesh superimposed on the plot confirms that the mesh has indeed been refined where stress concentrations are located and de-refined in "quiet" portions of the model.

Note

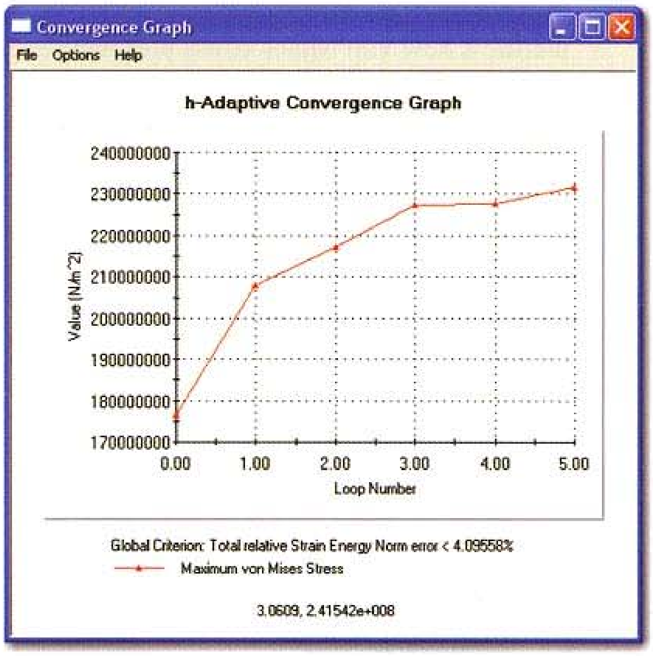
Any plot (stress, displacement, strain, and so on) in the results folders of the `h adaptive` study displays the final result, or the last performed step of the h adaptive solution process. In addition to displaying the final plot results, we can also access the history of the iterative solution.

16 Create convergence graph.
Right-click on the `Results` folder and select **Define Adaptive Convergence Graph**.



Under **Options**, select the **Maximum von Misses stress** check box. Clear the **Target Accuracy** check box.

Click **OK**.



Review h-adaptive solution

Let us review the graph and make a few observations about the **h-adaptive** solution.

- The **h-adaptive** solution has been obtained in five steps: the first using the original mesh and the next four steps with automatically refined meshes.
- In each loop the mesh is refined further.
- The 2% strain energy error we specified in properties of the `h adaptive` *study is NOT the stress error.*
- The maximum number of loops (6) has not been reached. Therefore, we know that the strain energy error fell below 2% after the fourth mesh refinement.

incorrect

- Based on the coordinates of graph points we can explicitly calculate von Mises stress error.
- The stress units in the graph are N/m^2, regardless of what units are used in the model.

If we are interested in von Mises stress, why can't we specify the error in terms of von Mises stress? In other words, why don't we use von Mises stress rather than the total strain energy as a convergence criterion?

The reason why the total strain energy is used as a convergence criterion is because the total strain energy ("total" means in the entire model) always shows monotonic convergence without local "plateaus", which might lead to premature termination of the convergence process.

Please review displacement results of study `h adaptive` before proceeding.

Having obtained a solution with the **h-adaptive** solution method we now solve the same model using the **p-adaptive** solution method.

p-Adaptivity Study

P-adaptive solution requires the use of a different type of finite element called p-element. Before we begin, we need to explain what p-elements are and what they do.

p-adaptive solution method

In `Lesson 1`, we said that COSMOSWorks uses three types of elements: tetrahedral solids, triangular shells, and beams. Each can be defined as either a:

- First order element (draft quality)
- Second order element (high quality)

Further, recall that first order elements model a linear (or first order) displacement and constant stress distribution, while second order elements model a parabolic (second order) displacement and linear stress distribution.

We now have to amend the above paragraphs. Besides first and second order solid tetrahedral elements, COSMOSWorks also has higher order tetrahedral solid elements (up to the 5^{th} order) meaning that a polynomial of the 5^{th} order can be used to model a displacement field inside the element, along its faces and edges. These elements are available when the **p-adaptive** solution method is used.

The order of elements used in the **p-adaptive** solutions is not pre-defined, but can be upgraded automatically during the iterative solution process without our intervention. These elements with upgradeable order are called p-elements.

17 Create p-element study.
To begin this exercise:

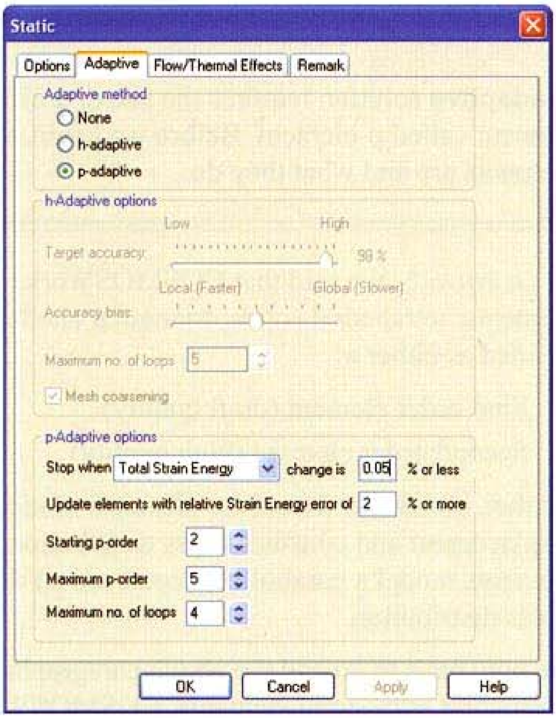
Create a study named `p elements`. The study name reflects the use of high order p-elements. (Specify **Static** as **Analysis type** and **Solid mesh** as **Mesh type**.)

18 Copy loads and restraints from previous study.

19 Define p-element method and its options.

To utilize p-elements in the analysis, right-click the `p elements` study folder and select **Properties.**

Under the **Adaptive** tab in the **Properties** window, select the **p-adaptive** box. Note that this option is available only for static analysis and only when using solid elements.



Set **Starting p-order** to **2**, which means that all elements are first defined as second order elements.

Set **Maximum p-order** to **5**. The **p-adaptive** solution runs in iterations, called loops, and with each new loop, the order of elements increases. The highest order available is the 5^{th} order, but the actual highest order we use can be lower and is defined by **Maximum p-order**.

Set **Maximum no. of loops** to **4**.

Under **p-adaptive options**, in the **change is** text box, enter **0.05**.

Click **OK**.

Looping continues until the change in **Total Strain Energy** between the two consecutive iterations is less than 0.05%, as specified in the **p-adaptive options** area. If this requirement is not satisfied, then looping stops when the elements reach the highest available order, which in our case is the 5^{th} order. Note that it takes four iterative loops to reach a 5^{th} order element. Investigate other choices in the **p-adaptive** options area.

Why are we specifying this high accuracy requirement (0.05%) for the total strain energy error? Actually, we do not expect that the solution will satisfy this requirement. We want to force the solver to complete all four steps so we can analyze graphs consisting of four, rather than two or three points.

The **p-adaptive** solution process is conceptually similar to the already performed **h-adaptive** iterative process of mesh refinement. Both add degrees of freedom to the model, one by mesh refinement, the other by element order upgrade.

The difference between **h-adaptive** and **p-adaptive** solution methods is that, in **h-adaptive**, mesh changes while element order stays the same, while in **p-adaptive**, mesh stays the same but element order changes.

h vs. p Elements

Let us pause to explain some terminology:

Question: Why are upgradeable elements called p-elements?

Answer: The iterative process that we are currently discussing does not involve mesh refinement. While the mesh remains unchanged, the element order changes from the initial 2^{nd} order all the way to 5^{th} order (or less if the convergence criterion is satisfied sooner).

The element order is defined by the order of polynomial functions that describe the displacement field in the element. Because the polynomial (p) order experiences change, the process is called the p convergence process, and the upgradeable elements we use are called p-elements.

Question: Why is the p convergence process called a **p-adaptive** solution, and what exactly does "adaptive" mean?

Answer: Adaptive means that not all p-elements are necessarily upgraded during the solution process.

Indeed, as you see in the **p-adaptive** options area, **Update elements with relative Strain Energy error of _% or more** means that only those elements not satisfying the above criterion are upgraded. We say, therefore, that element upgrading is "adaptive", or driven by the results of consecutive iterations.

This is in close analogy to **h-adaptive** solution (performed earlier in this lesson), where the mesh was refined during consecutive loops.

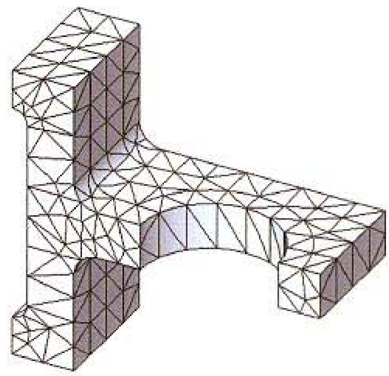
We are now sufficiently familiarized with p-elements to proceed with **p-adaptive** solution.

20 Create mesh.
Right-click `Mesh` and select **Create Mesh**.

Under **Options**, click **At nodes**.

Under **Jacobian Check for solid**, select **At nodes**.

21 Mesh the model and run the analysis.
Create a **High** quality mesh intended for p-elements with an **Element size** of **0.5 in**.



Note

Considering that a p-adaptive solution is used, we can manage with a coarser mesh.Create a **High** quality mesh intended for p-elements with an **Element size** of **0.5 in**.

Check the **Run analysis after meshing** box to combine the mesh and run steps into one.

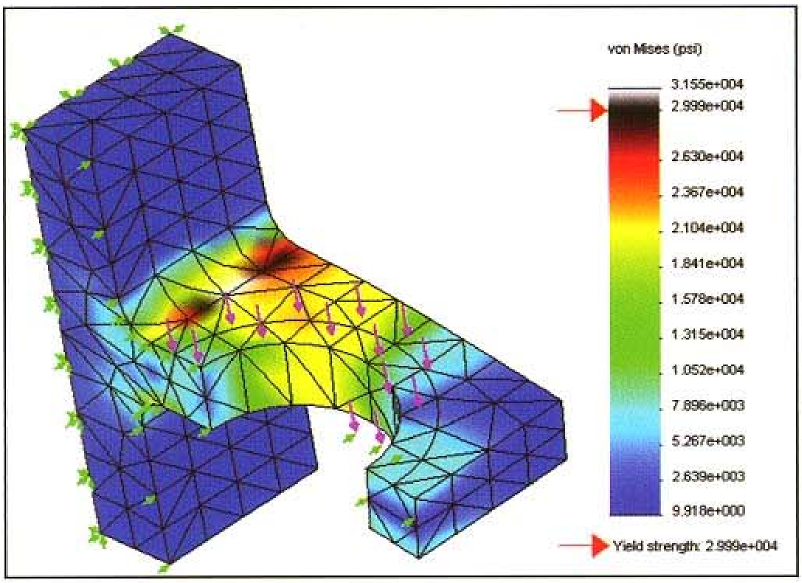
Note

This mesh is not acceptable for use with h-elements because there are not enough elements to capture the complex stress field, especially near the rounds. However, using higher order p-elements is equivalent to refining an h-element mesh, so that even this coarse mesh delivers accurate results.

Run the analysis, and notice that the solution progresses in steps corresponding to the number of element-order upgrades.

22 Display von Mises stress plot.
Now that we have solved the study with p-elements, we display a von Mises stress plot.

To set the plot settings, right-click the stress plot and select **Settings**. Select **Discrete** for the **Fringe options** and select **Mesh** for the **Boundary options**.



The resulting stress plot shows a maximum von Mises stress of 31,550 psi, which is just above the yield stress of AISI 304.

23 Animate results plot.
To verify that the symmetry boundary conditions work as expected, animate one of the results plots. While the model deforms, verify that the faces with symmetry boundary conditions applied to them remain flat and there is no movement of these faces in the direction normal to the plane of symmetry.

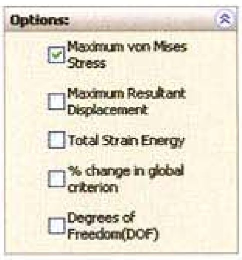
Note

Any plot (stress, displacement, strain, and so on) in the results folders of the `p elements` study displays the final result, or the last step of the **p-adaptive** solution process. In addition to displaying the final plot results, we can also access the history of the iterative solution.

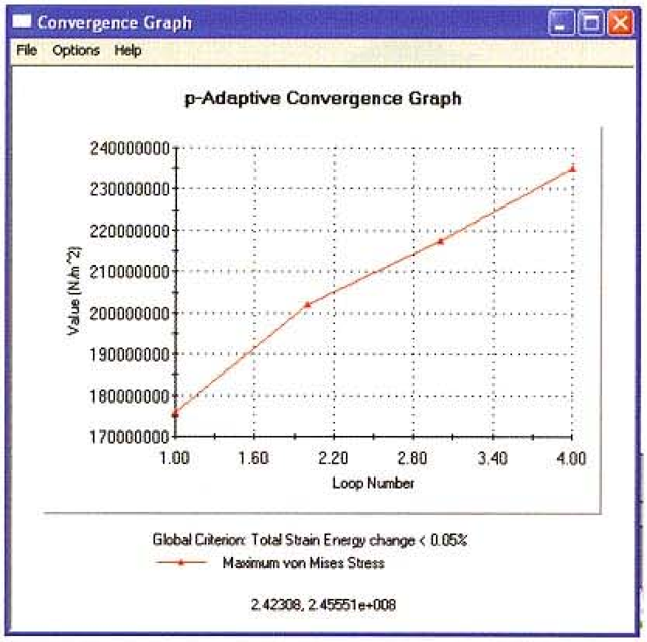
24 Create a convergence graph.
Right-click the `Results` folder and select **Define Adaptive Convergence Graph**.

The **Convergence Graph** PropertyManager opens. Here you can specify what information to display on the graph.

Because we are primarily interested in the maximum von Mises stress, select **Maximum von Mises stress** in the **Options** window.



Click **OK**.



Let us review the graph and make a few observations about the **p-adaptive** solution.:

- The **p-adaptive** solution has been obtained in four steps: the first using the second order elements, and the next three with higher order element up to the 5th order.
- The 0.05% strain energy error we specified in the properties of the `p adaptive` study is NOT stress error.
- We do not know if the strain energy error fell below 0.05% in the fourth step because the maximum number of loops has been reached. We have reached the maximum available element order, thus, we do not know if looping would have continued otherwise.
- Based on the coordinates of graph points we could explicitly calculate von Mises stress error.
- The stress units in the graph are N/m^2. SI units are internally used by COMOSWorks for calculations, regardless of what units are used in the model.

Now, let us summarize the results of all three studies executed in this lesson. Recall that information on the number of degrees of freedom is taken from the OUT file corresponding to the given study in the COSMOSWorks data base.

Solution type	Max. resultant displacement [in.]	Max. von Mises stress [psi]	# D.O.F.
Standard	0.01702	30390	31887
h-adaptive	0.01705	33200	27255
p-Adaptive	0.0168	34100	57636

Displacement results are practically the same. Stress results are within 9%. Considering that a highly concentrated stress is rather difficult to model with any solution technique, this accuracy is satisfactory. Standard solution appears to be the most economical; it had the shortest solution time.

Having completed the exercise with three solution methods, we note that **h-adaptive** and **p-adaptive** solution methods are very close, conceptually. Upgrading the element order in the p convergence process adds degrees of freedom to the model, which is a direct analogy to adding degrees of freedom by mesh refinement in the h convergence process.

This explains why we can use a coarse mesh for both **h-adaptive** and **p-adaptive** solution. The degrees of freedom that are "missing" in the initial mesh are added in the process of iterative solution either by mesh refinement or by element order upgrade, and produce an analogous effect to using a standard solution technique with properly refined mesh.

h vs. p Elements - Summary

	Solution type	
	h-adaptive	**p-adaptive**
Element order	2nd order, does not change during the solution	Changes during the solution process from 2nd to max. 5th to satisfy the accuracy requirements
Mesh adaptivity	Mesh is changed by refinement (both element size and location of refinement) and adapts to the pattern of stress distribution found in the model. High stress gradients are meshed with more refined mesh.	Mesh does not change. Element order adapts to the pattern of stress distribution found in the model. High stress gradients are meshed with higher element order.
Global error control	Total Strain Energy (called Target accuracy)	Total Strain Energy RMS Resultant Displacement RMS von Mises stress
Local error control	Local strain energy (called Accuracy bias)	Local strain energy
Maximum number of loops	Six: The first one with the original mesh and remaining five with refined meshes	Four: The first one with 2nd order elements, the last one with 5th order elements

25 Examine chart options.

While examining the graph, use this opportunity to investigate the graph options, which offer various ways to format and display the graph. The graph options are too numerous and too detailed to discuss here, but are self-explanatory.

Which Solution Method is Better?

- The **standard** solution using h elements?
- The **h-adaptive** solution using h elements?
- The **p-adaptive** solution using p elements?

Generally, with the standard solution method using second order h-elements, we obtain a reasonably accurate solution within a reasonably short time.

Experience indicates that the standard solution method utilizing second order h-elements offers the best combination of accuracy and computational efficiency.

For this reason, the automesher in COSMOSWorks is tuned to meet the requirements of an h-element mesh intended for the standard solution method.

Both **h-adaptive** and **p-adaptive** methods involve iterative solutions that stop either when the accuracy requirement has been satisfied or when the maximum allowed number of iterations has been reached. In this lesson, we requested a very low error to make sure that the solver goes through the maximum number of iterations. This way we obtained solutions with low, but not explicitly known, error.

Try running `h adaptive` and `p adaptive` studies again with relaxed accuracy requirements so that the solver does not use up all available loops.

If the convergence graph shows:

- less than six iterations have been performed for h adaptive solution

or...

- less than four iterations have been performed for p adaptive solution

This means that the solution has stopped because your accuracy requirements have been satisfied and not because the maximum number of loops has been reached.

Summary

Both **h-adaptive** and **p-adaptive** solution methods are significantly more time-consuming. Therefore, these solution methods are reserved for special cases where the solution must have narrowly specified accuracy.

The adaptive solution methods are also great learning tools, leading to a better understanding of element order, the convergence process, and discretization error. For this reason, you are encouraged to repeat some of the lessons presented in this volume using the adaptive solution technique of your choice.

Lesson 11 Thermal Stress Analysis of a Bimetal Strip

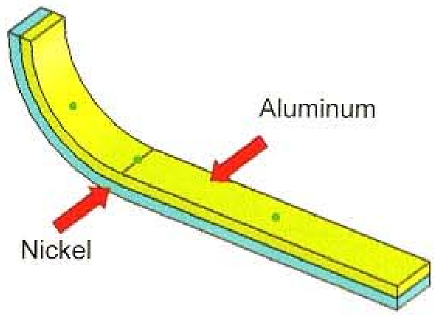
Objectives

Upon successful completion of this lesson, you will be able to:

- Perform a static analysis with a temperature load
- Define temperature dependent material properties
- Use sensors to retrieve results at desired locations
- Use soft springs option in thermal stress analysis
- Save the deformed shape of the model
- Examine results in local coordinate systems

Project Description

An aluminum strip, glued with a nickel strip into a bimetal assembly, is at room temperature (25°C). Without any constraints on its deformation, the bimetal strip is then heated to 280°C (536°F). Due to the difference in the coefficients of the thermal expansion of aluminum (200 W/m K) and nickel (43 W/m K), the bimetal deforms.



Our objective is to find the deformations that arise due to the different thermal expansions of aluminum and nickel, and the minimum required strength of the bonding glue material.

It is required that the numerical simulation is followed by an experiment. Six tensometers, oriented along the longitudinal direction, will be attached to the surface of the tested model (three on the top of each part) as shown in the figure to measure the surface deformation. To allow for the correlation between the numerical and experimental data, sensors will be defined in the same locations in the finite element model.

The deformed assembly will then be saved as a SolidWorks model for further design applications.

Deformation Analysis of Bimetal Assembly

Open assembly.
Open assembly file named `bimetal` with component parts: `al` (aluminum) and `ni` (nickel).

2 **Create study.**
Create a study named `bonded`. (**Static** analysis, **Solid** mesh.)

Note

The Ni and Al properties are automatically transferred from the SolidWorks assembly. Because the model is exposed to an elevated temperature, the material constants have to be modified accordingly. The following two tables show the dependence of the material constants on the temperature

	Variation of SIGYLD [Pa] on temperature				
	Room	100°C	204°C	260°C	316°C
Inconel 702 Nickel Alloy	406.9e6	-	356e6	-	326e6
2014 - T6 Aluminum Alloy	378.6e6	330.5e6	210e6	119.8e6	44.4e6

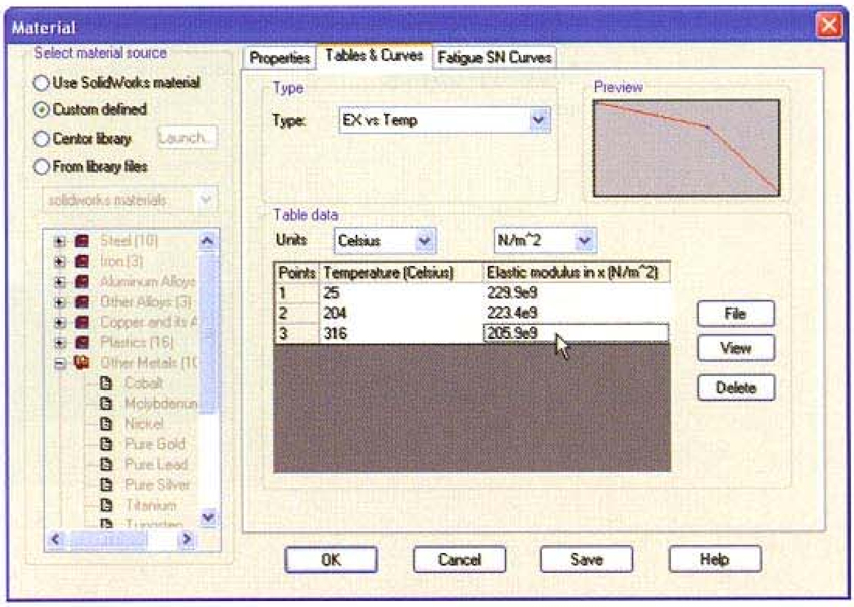
	Variation of EX [Pa] on temperature				
	Room	100°C	204°C	260°C	316°C
Inconel 702 Nickel Alloy	229.9e9	-	223.4e9	-	205e9
2014 - T6 Aluminum Alloy	71.9e9	70.6e9	64.1e9	50.8e9	50.5e9

3 **Assign material properties to Ni strip.**
Right-click on the folder `ni-2` (under the folder `Solids`) and select **Apply material to all bodies.**

The material dialog window displays default Ni material constants at room temperature.

Under the **Select material source** section, select **Custom defined**.

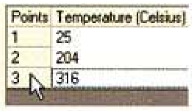
In the **Tables & Curves** tab, select the **EX vs Temp** curve under **Type**. Select **Celsius** degrees and **N/m^2** under the **Table data** section.



Enter the available data points from the table above defining the dependence of the Young's modulus on temperature for Inconel 702 Nickel Alloy.

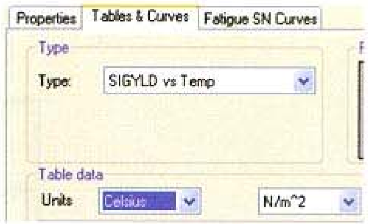
Note

To add a new line in the Table data definition, double-click on the last row.

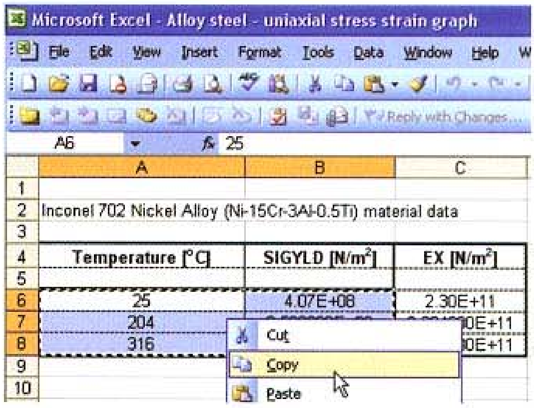


The number of data points can be excessive in many cases. In COSMOSWorks, the data can be conveniently copied from other programs such as Excel.

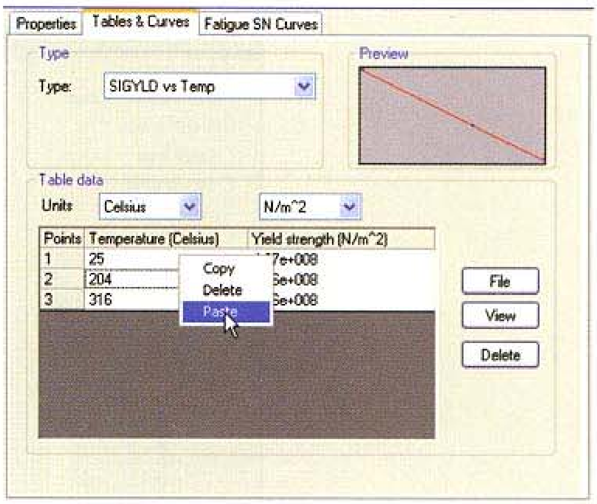
Under **Type**, select **SIGYLD vs Temp**, set the **Units** to degrees **Celsius** and **N/m^2** (Pa).



Open the Excel file `materialdata.xls` located in the lesson directory, and navigate to the sheet with the Inconel 702 Nickel Alloy data. Right-click on the corresponding data and select **Copy**.

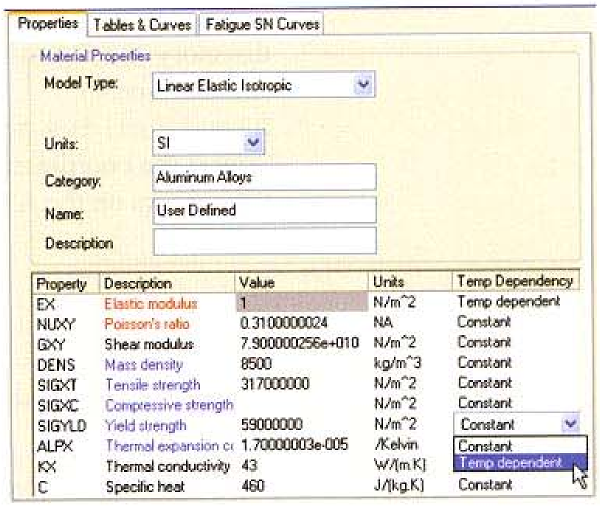


Paste the data into the Table data area of the Material dialog window.



Switch back to the **Properties** tab. Under the **Temp dependency** column, click on the corresponding cells for **EX** and **SIGYLD**, and select **Temp dependent**.

Notice that the cells for **EX** and **SIGYLD** under the **Value** column show **1** and become inaccessible.



Click **OK** to confirm the definition of the Inconel 702 Nickel Alloy properties.

Note

We will assume that the thermal expansion coefficient remains constant in a given temperature range.

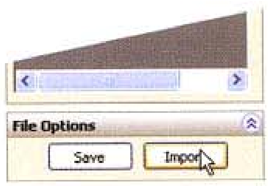
4 **Assign material to Al strip.**
Follow the procedure above and assign the same (**SIGYLD** and **EX**) temperature dependent material constants for 2014-T6 Aluminum Alloy. The data can be again found in the `materialdata.xls` Excel file located in the lesson directory.

Note

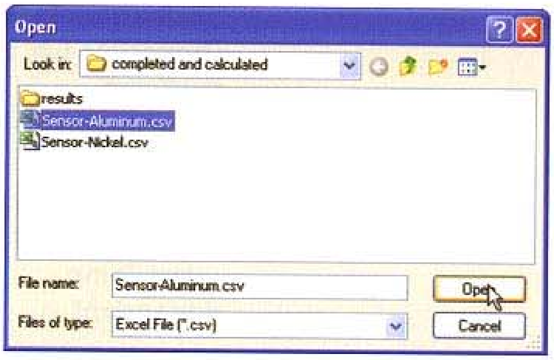
Note that, as always, when analyzing an assembly in COSMOSWorks, the default global contact conditions are set to **Bonded**. As we analyze glued components, bonded contact is appropriate.

5 **Define sensors on Al strip.**
Right-click the `Sensors` folder and select **Edit/Define**.

In the **Sensors** dialog, under **File Options** click **Import**.

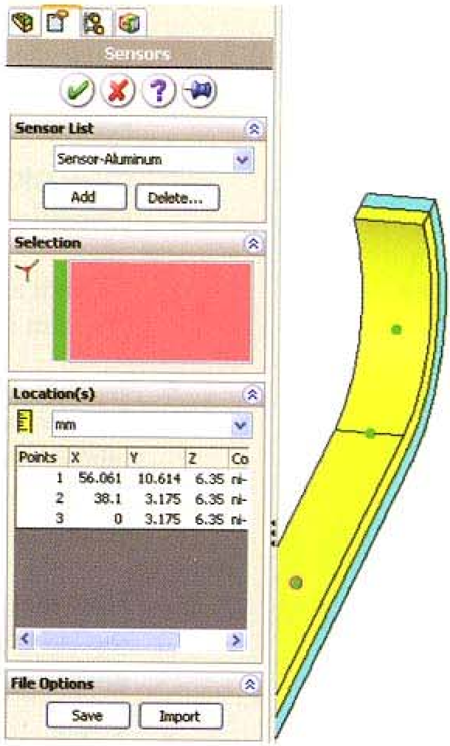


Browse to the lesson directory and **Open** the file `Sensor-Aluminum.cvs`. to import the coordinates on the sensors on the `Al` strip.



Click **OK** to save the sensor locations.

Note that the name of the sensor is identical to the name of the file, `Sensor-Aluminum`. Also, the locations of the sensors will be shown in the model.



6 Define sensors on Ni strip.
Follow the procedure above to define the sensors on the `Ni` strip.

7 Apply temperature load.
Right-click `Load/Restraint` and select **Temperature**. Use the SolidWorks fly-out menu to select both components of the assembly whose names are visible in the **Faces, Edges, Vertices for Temperature** field.

Enter a temperature of **280°C**. This definition states that the temperature of both assembly components is uniformly elevated/lowered to 280°C from the reference temperature at zero strain.

Click **OK**.

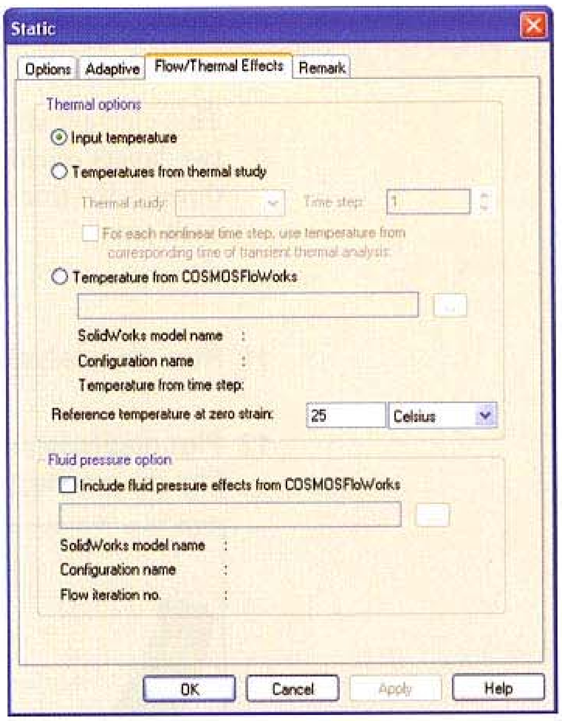
8 **Define zero strain temperature.**

Right-click the bonded study folder and select **Properties**. The **Static** window opens.

Select the **Flow/Thermal Effects** tab.

In the **Thermal options** area, select **Input temperature** (this is the default choice). The input temperature is the 280°C that we defined earlier.

Enter 25°C as the **Reference temperature at zero strain**. This temperature corresponds to the room temperature, and we assume that no strain exists in the model at this temperature, due to the structural loads and boundary constraints.



Importing temperatures from COSMOSWorks thermal study or COSMOS FloWorks

In the previous dialog window, notice that the temperature loads can be also imported from the COSMOSWorks thermal study or directly from the CFD (Computational Fluid Dynamic) simulation in COSMOSFloWorks.

For the stress analysis it is also possible to import the distributions of the fluid pressures from COSMOSFloWorks.

9 Stabilize the model.

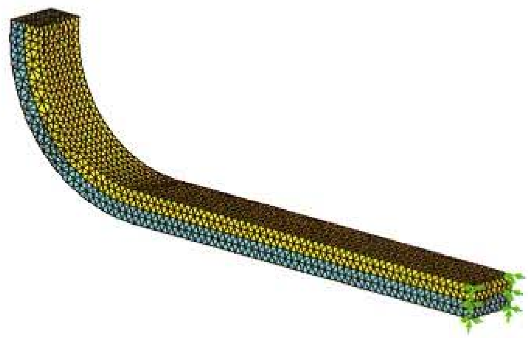
Because the strip's deformation should be unconstrained, we cannot apply an external boundary condition. Since the model is in the state of thermodynamic equilibrium and is not subjected to any external force loads, we can use the soft spring option to stabilize the model.

Right-click on the bonded study folder and select **Properties**. Under the **Options** tab, activate the **Use soft spring to stabilize model** option.

10 Mesh the model.

Create a **High** quality mesh with the global **Element size** of **1.5 mm**.

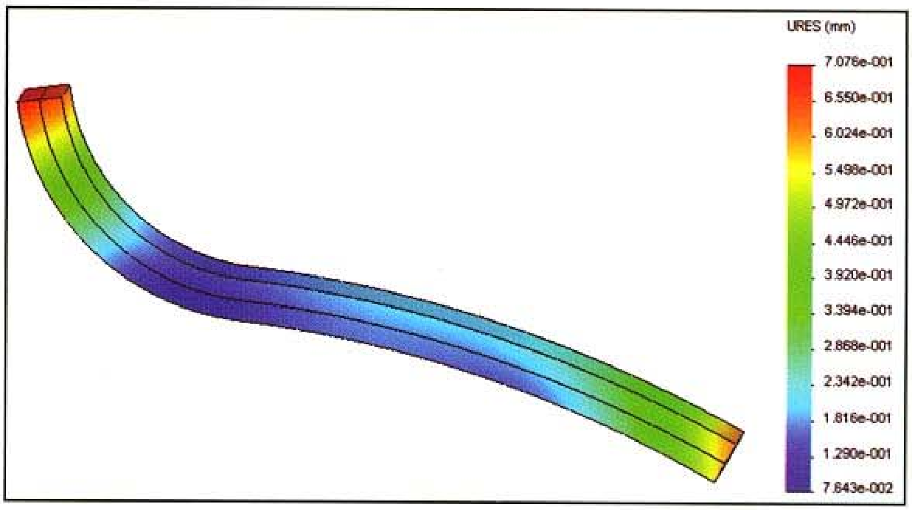
This element size creates two layers of elements through the thickness of each part.



11 Run the analysis.

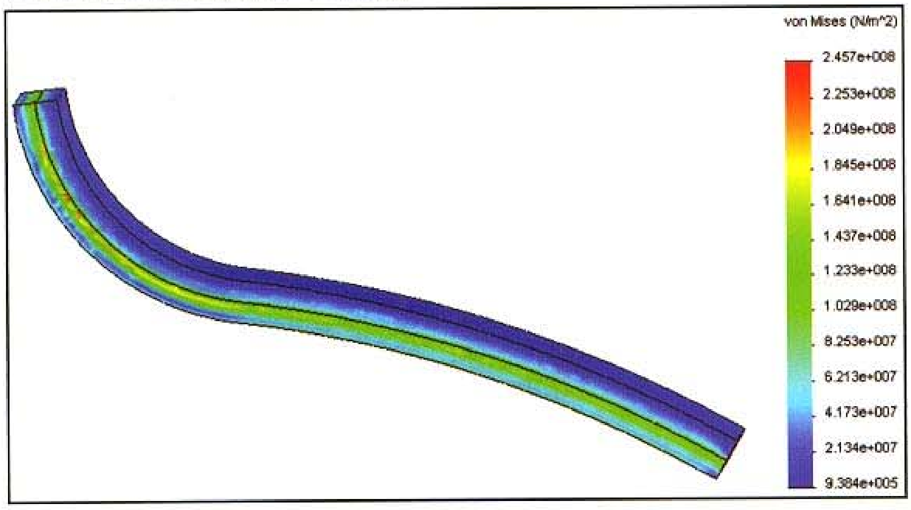
12 Plot displacements.

Plot the resultant displacements (in 1:1 deformation scale).



We observe that the maximum displacement at the tip of the bimetallic strip is 0.7 mm.

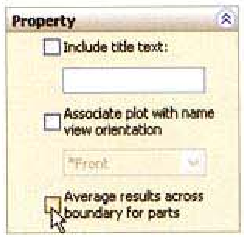
13 Plot von Mises stress results.

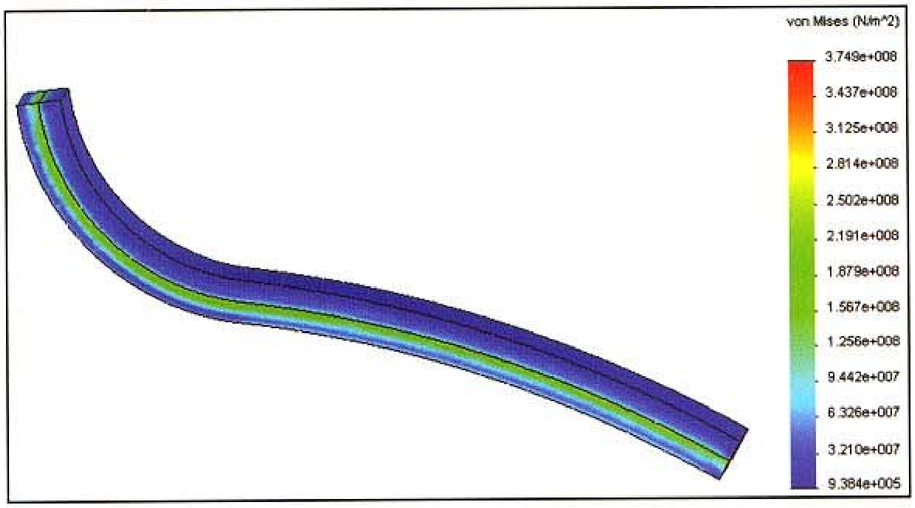


This plot can be misleading, however. As we learned in Lesson 1, the von Mises stresses are obtained by averaging the stress values extrapolated to the nodes from all the adjacent elements. In this case, the stresses at the interface are averaged between two distinct parts. To obtain the correct distribution of the von Mises stresses, we have to disable the averaging across the part boundaries.

Right-click on the stress plot and select **Edit Definition**.

In the **Property** dialog, clear the **Average results across boundary for parts** box.

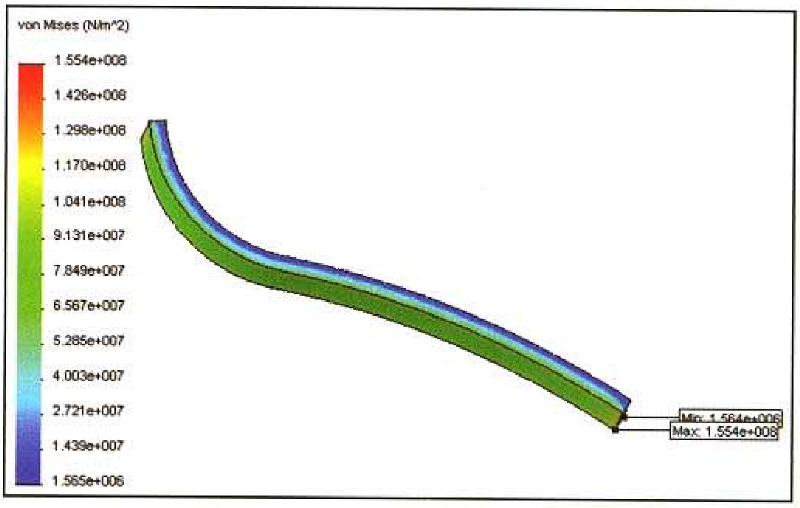


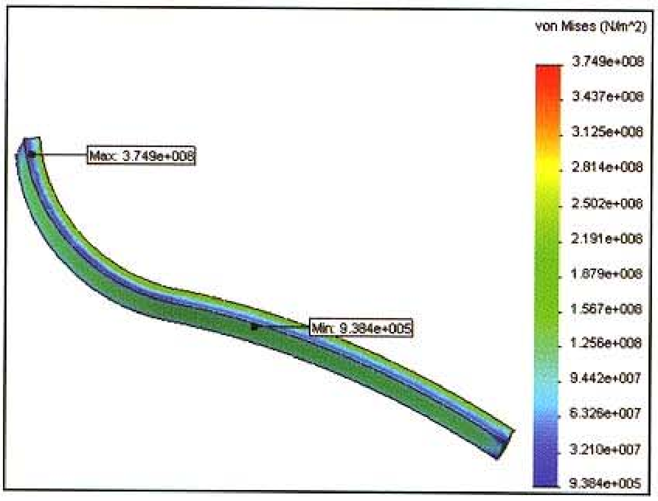


The new von Mises plot above shows the correct distribution. We can see that with the averaging across the boundaries option disabled, the maximum value jumped to approximately 375 MPa in some interface regions.

To view the maximums in each of the two parts separately, hide the other part in SolidWorks and plot the distribution of the von Mises stress for the displayed part only. Also, the extreme values for the shown parts only can be requested as described below.

Under **Chart Options**, activate the **Show min annotations**, **Show max annotations**, and **Show Min/Max range on shown parts only** options.





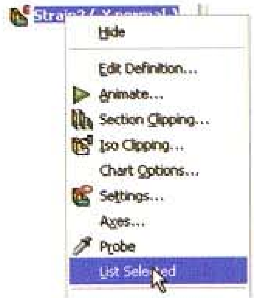
We can see that the maximum von Mises stresses in the Aluminum and Nickel Alloy parts, 155 and 375 MPa, are well above their corresponding yield strengths at 280°C (93 and 335 MPa). This indicates that both parts are yielding.

The accurate solution to the above problem can, therefore, be obtained using a nonlinear modulus of COSMOSWorks, where full stress-strain curves for the given alloys would have to be specified.

We will ignore the fact that the parts are yielding and continue with the lesson. In the next part, we would like to analyze the interface layer and find the minimum required strength of the bonding material

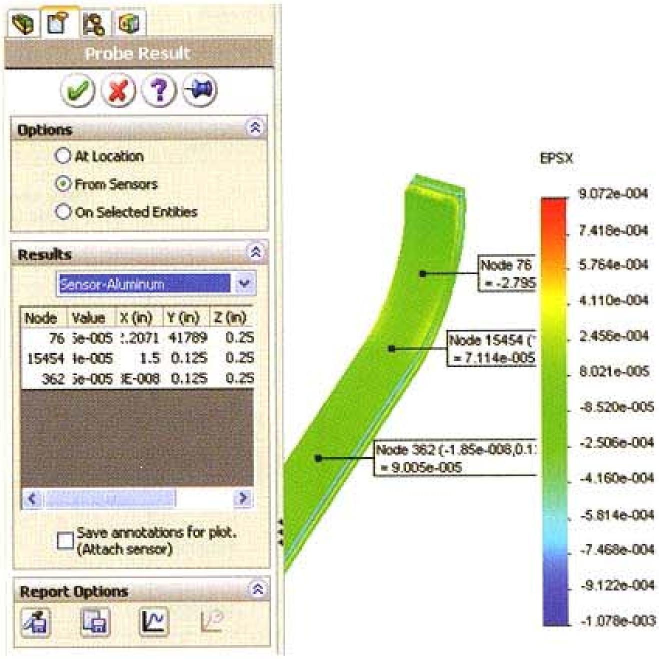
14 Show strain results at the sensor locations.
Define a new strain plot for the **ESPX: X Normal strain** component.

Right-click on the newly defined strain plot and select **List selected**.



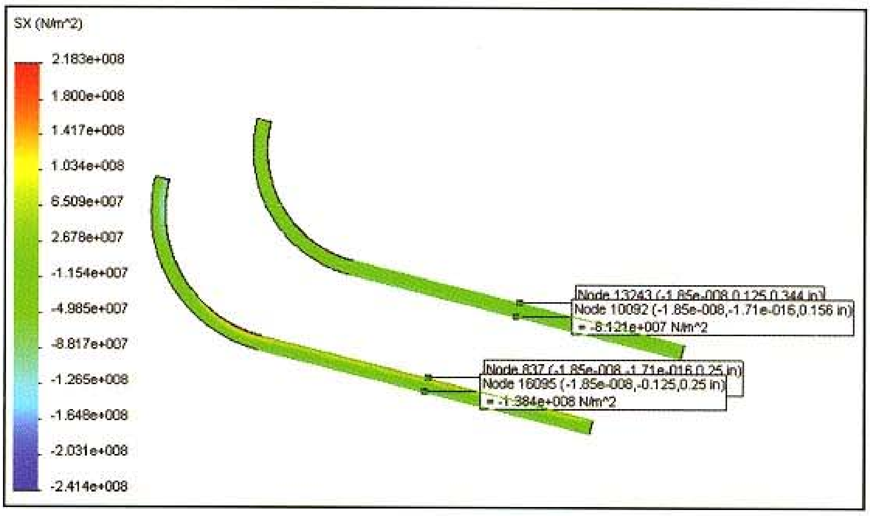
Select **From sensors** under **Options** and `Sensor-Aluminum` under **Results**.

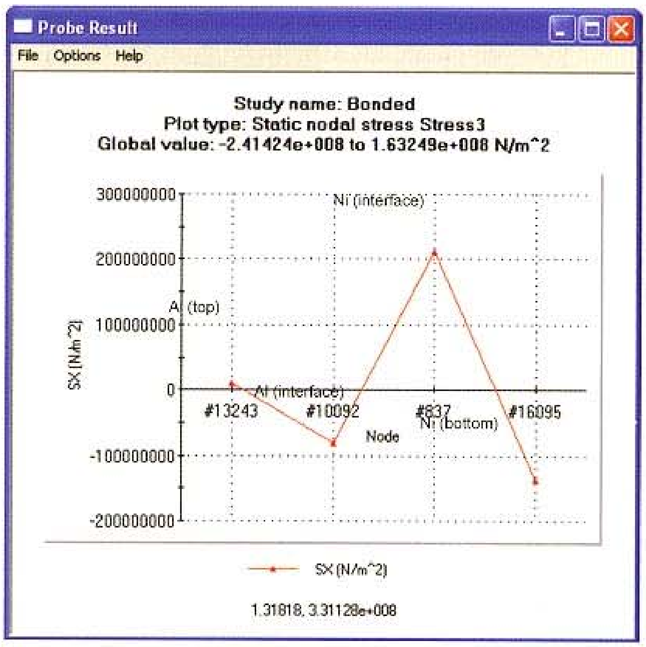
The normal strain values will be listed in the table and shown on the model.



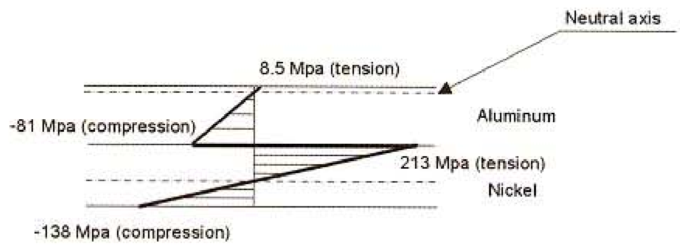
Note that the buttons under **Report Options** let you graph the results at the sensor locations or save them in `.cvs` file for further processing. Also, it is possible to include the results at all sensor locations in the study report.

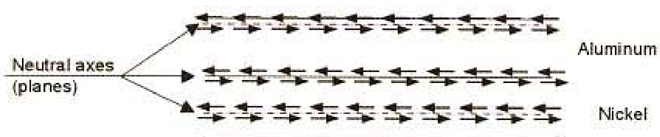
15 Plot distribution of normal stress SX.
Define a new stress plot for the **SX: X Normal stress** component. In exploded view, analyze the through-thickness variation of the **SX** normal stress (use the Probe feature to path plot the variation of the **SX** stress through the thickness).





The results and the graph above indicate the following variation of **SX**.





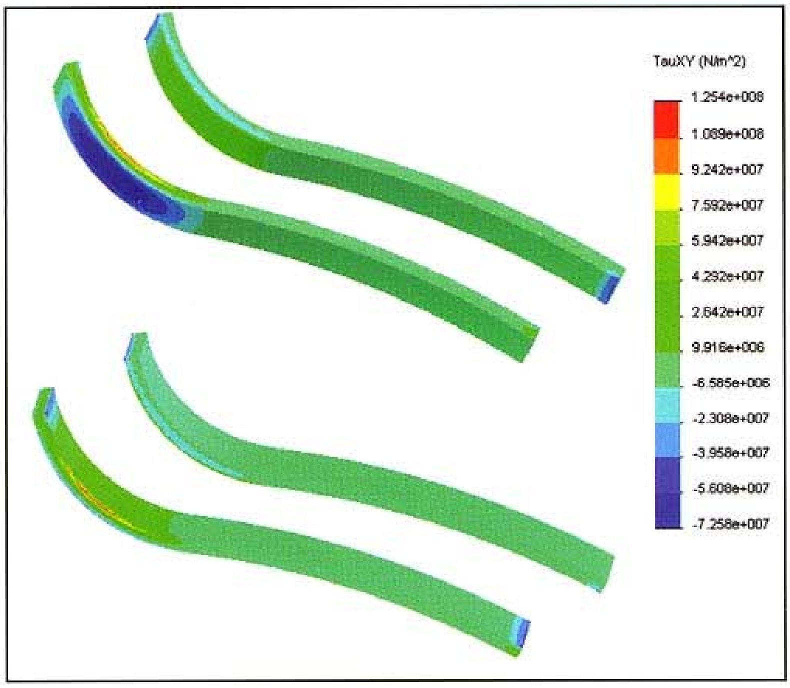
We observe that at the interface the normal stresses change abruptly from -81 MPa (compression) in Aluminum to 213 MPa (tension) in Nickel. Furthermore, we can observe three neutral axes (planes) where the normal stresses are at zero. Two of these axes (planes) are clearly indicated in the figure. The third axis coincides with the interface plane, where the normal stresses change abruptly from -81 MPa (compression) in Aluminum to 231 MPa (tension) in Nickel. All three locations are accompanied by the local extremes of shear stress that may delaminate the glued strips (see the figure above).

Since our main goal is to obtain the required strength of the bonding material, we will focus on the interface location. The bonding material must be capable of resisting the shear stress at the Aluminum/Nickel interface.

Reviewing the *Interpretation of FEA results* section of the Introduction reveals that we must plot the τ_{xy} component of the stress. This corresponds to the **TXY: Shear stress in the Y dir on YZ plane** component.

16 Plot interface shear stress.

Define a new plot of **TXY: Shear stress in the Y dir on YZ plane** stress. Display the plot in the exploded view and, under **Settings**, request the **Discrete** fringe option.



We can observe that the interface shear stress on the side of Aluminum and Nickel are identical, i.e. the equilibrium is satisfied. The discrete plot conveniently shows that the maximum value of shear stress (ignoring the localized stress concentrations at the tip of the straight section) equals approximately 10 MPa. This would be the minimum required strength of the glue in shear for this application.

Question

We concluded that the minimum required strength of the glue is approximately 10 MPa. Inspection of the τ_{xy} plot above shows that the bent part should experience much larger shear stress. Why?

For the answer review the following section.

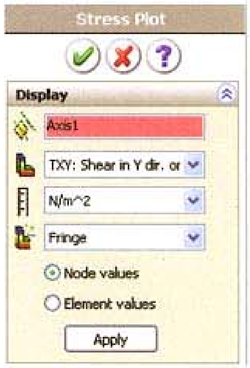
Examining Results in Local Coordinate Systems (Optional)

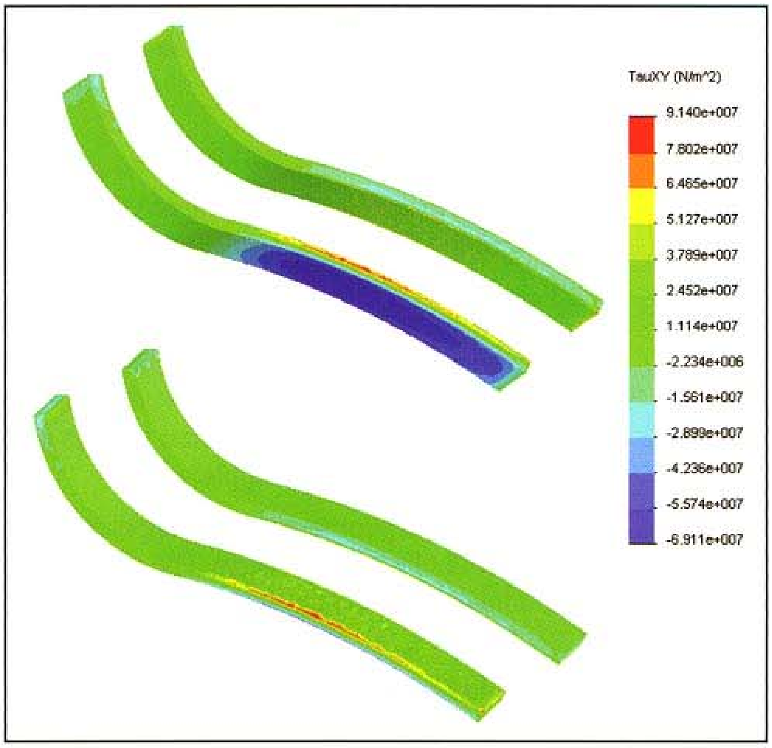
τ_{xy} in the previous figure is referencing the global coordinate system. In the bent section, the global x, y and z axes are no longer aligned with the interface geometry. In other words, the τ_{xy} distribution in the bent section no longer represents the interface shear stress.

For the correct representation of the interface shear stress in the bent section, we have to switch to the appropriate coordinate system aligned with the geometry.

17 Interface shear stress at the bent location.

Create a **Discrete** τ_{xy} plot with `Axis1` used as a reference. `Axis1` defines a local cylindrical coordinate system aligned with the geometry of the bent section.





Note

In the cylindrical coordinate system, the r, θ, and z sequence of coordinate axes corresponds to the x, y, and z sequence in the Cartesian coordinate system. By specifying τ_{xy} in the plot definition dialog with `Axis1` specified as a reference, we request a plot of $\tau_{r\theta}$, which is the interface shear stress.

The above plot of the interface shear stress shows that the bent section experiences larger shear. The indicated value is approximately 24 MPa. As this is greater than the value of 10 MPa obtained from the straight section, we conclude that this is the minimum required shear strength of the bonding material.

Note

Since the bonding material will also have to resist the normal stress we would have to verify the interface normal stress in the proper coordinate systems. It can be checked that this stress is significantly smaller in this case (approximately 5MPa) and will not govern.

Saving Model in its Deformed Shape

We end this lesson by saving the deformed shape as a new SolidWorks model so that it can be used as an assembly component to check for interference, and so on.

18 Create deformed plot.
Right-click on the `Results` folder and select **Define Deformation Plot**. (Make sure the plot is defined in 1:1 scale.)

19 Save deformed plot as a VRML.
Right-click the `plot` icon and select **Save As**.

The **Save As** window opens.

Specify **VRML** as the target format.

Place the saved VRML file where you can find it. By default, the file is saved in the folder that holds the automatically created reports.

Click **OK**.

20 Open the VRML file in the SolidWorks environment.
In the options area of the SolidWorks **File**, **Open** window, select **Import as Solid Body**. Note that it takes some time to read the VRML file into the SolidWorks application.

The deformed geometry model appears as an imported feature in the SolidWorks FeatureManager. The deformed shape can now be examined with the standard SolidWorks tools.

Summary

A simple bimetal assembly was analyzed when subjected to the elevated temperature. To eliminate the effect of external supports, the **Use soft springs to stabilize model** option was used.

At elevated temperatures, the values of some of the material properties may vary considerably. In this lesson, we practiced the definition of the temperature dependent yield strength and Young's modulus.

The main goal of the lesson was to obtain the minimum required bond strength of the interface glue. To obtain this value, a complex distribution of the normal stress SX was studied, and the definition of the neutral axes (planes) was introduced. Furthermore, the corresponding component of the shear stress was plotted.

In the bent section, the curved geometry required the introduction of the local cylindrical coordinate system. The interface shear stress was then plotted with respect to this local coordinate system.

Since the experimental verification of the numerical results was required a set of sensors were defined to extract the deformation results at the specified tensometer locations.

Lastly, exporting of the deformed geometry as a VRML file was shown and discussed.

Lesson 12
Beam Elements-
Static Analysis of a
Conveyor Frame

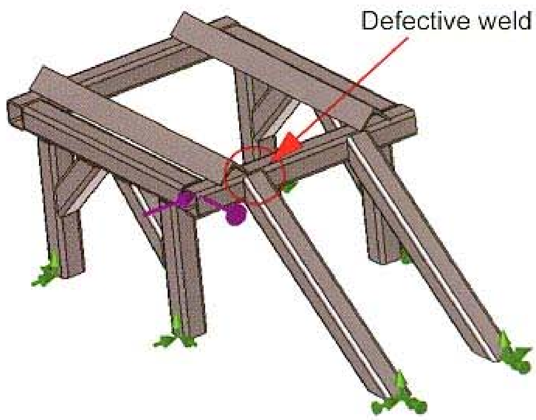
Objectives

Upon successful completion of this lesson, you will be able to:

- Use beam elements to analyze weldment models
- Define appropriate beam joint connections reflecting the real situation
- Post-process the results from beam elements

Project Description

The simplified model of a conveyor frame (shown in the figure) is manufactured from Alloy steel with all the joints welded.



During the inspection, it was found that the weld at the indicated joint became defective and was not capable of transmitting the moments. The faulty frame is to be analyzed when subjected to the operating loading conditions (combination of an isolated force and a moment). All six legs of the frame are bolted to the ground but only the two inclined legs can actually transmit the moments to the floor.

1 **Open part `conveyor frame`.**
The part is located in the Lesson 12 directory.

2 **Set COSMOSWorks options.**
Set the global system of units to **English (IPS)** and the units of **Length** and **Stress** to **in** and **psi**, respectively.

3 **Create new study.**
Create a new study named `frame` (**Static, Beam mesh**).

Note

Notice a new folder `Beams` along with one subfolder `Joint group1`.

The frame can be analyzed using both solid and shell elements, both resulting in an excessive number of elements. Also, the construction of the mesh along with the corresponding contact conditions may take some time. In this lesson, we will use beam elements: a fact that will allow us to greatly simplify the model with a minimum sacrifice on the side of the accuracy.

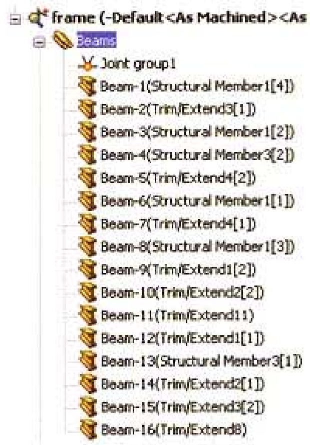
Beam elements

Beams are another class of structural elements where all of the cross-sectional characteristics are accounted for during the derivation of the element stiffness matrix. As a beneficial consequence, these cross-sectional characteristics do not need to be reflected in the finite element mesh, thus, greatly simplifying the model preparation and analysis.

In general, the beam element has two nodes with six degrees of freedom in each node. For more information, consult the Introduction chapter in this manual.

4 Define beam elements.
Right-click on the folder `Beams` and select **Treat all structural members as beams.**

Sixteen beam elements will be automatically generated.



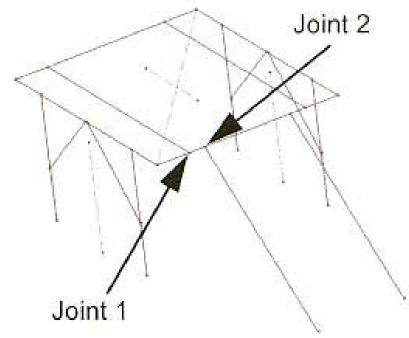
Note

You can also define each beam element individually by right-clicking on the folder `Beams` and selecting **Add beam**.

5 Specify material.
Assign **Alloy Steel** for all beam elements.

Beam joints: locations

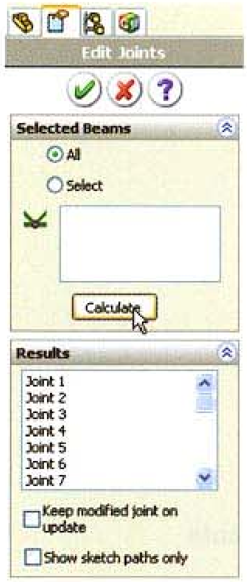
As described in the previous discussion, all of the beam cross-sectional characteristics are already included as parameters during the derivation of the beam element stiffness matrix. The resulting mesh is, therefore, made out of lines connected by joints. The lines forming the geometry for meshing are identical to the sketches used to generate the structural weldment members. For our case, the corresponding sketches are shown in the figure.

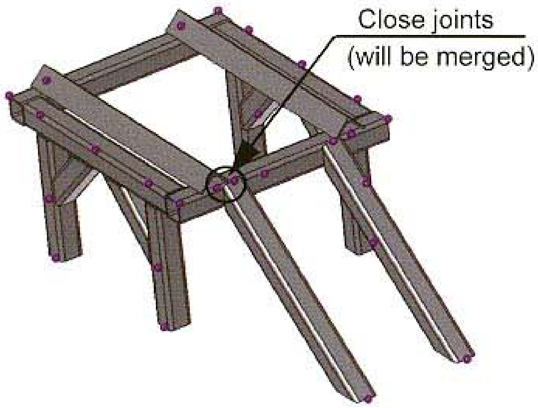


The joints define the straight segments that will be meshed with beam elements. While the detection of the joints is fully automated in COSMOSWorks, the relative position of some joints may be too close and we may wish to merge them, i.e. merge two straight segments into one. In the figure above, the joints 1 and 2 are relatively close and can be merged.

6 Calculate joints.
Under the `Beams` folder, right-click on `Joint group1` and select **Edit.**

In the **Selected Beams** dialog, select **All** and click **Calculate**. The generated joints will be shown in the **Results** dialog. Their location will be displayed in the model as well.

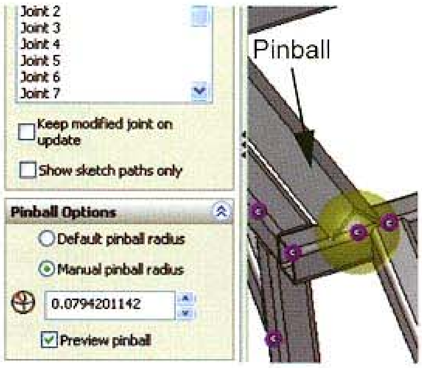




Note

The joints identified in the above figure are relatively close and will be merged.

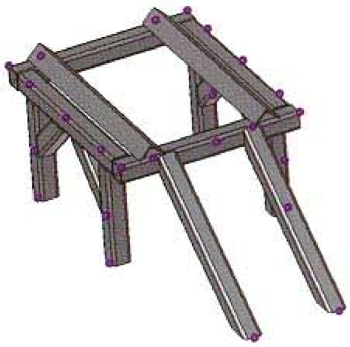
In the **Pinball Options** dialog, check the **Preview pinball** box. With your mouse, drag the pinball to the location of any of the joints to be merged.



Activate the **Manual pinball radius** option and, either using the numerical field or your mouse, increase the diameter of the pinball so that it encloses the other joint (the corresponding diameter of the pinball is approximately 0.08 in).

Click **Calculate** to update the joint locations.

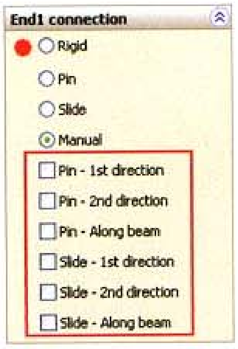
Click **OK** to accept and close the **Edit Joints** dialog.



Beam joint types

Each beam end point features six degrees of freedom that may be restrained or released to reflect various structural connection configurations. COSMOSWorks offers the following options to connect the end point of the beam element to the joint:

- **Rigid** - All six degrees of freedom are tied to the joint. This connection type transfers all force as well as all of the moments from the beam element to the joint (and vice versa).
- **Pin** - Only three degrees of freedom are tied to the joint. The connection is not able to transmit the moments from the beam to the joint (and vice versa).
- **Manual** - A custom designed connection type can be generated.

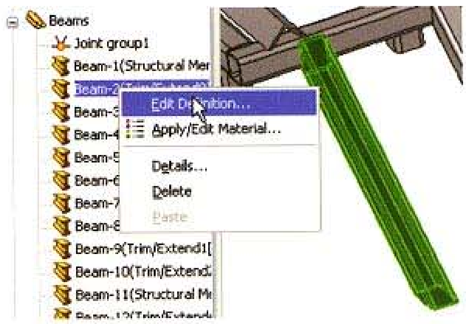


The faulty weld in the indicated joint eliminates the possibility to transfer moments between the beam element and the joint. The moment ties can be released by specifying the pin connection type.

7 **Define faulty weld joint.**

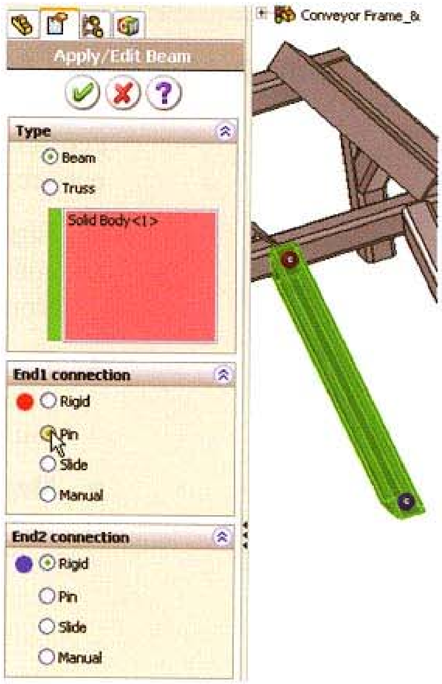
Under the `Beams` folder, right-click on the beam element corresponding to the inclined member with the faulty weld and select **Edit definition**.

Note that the two end points are graphically shown as red and blue circles.



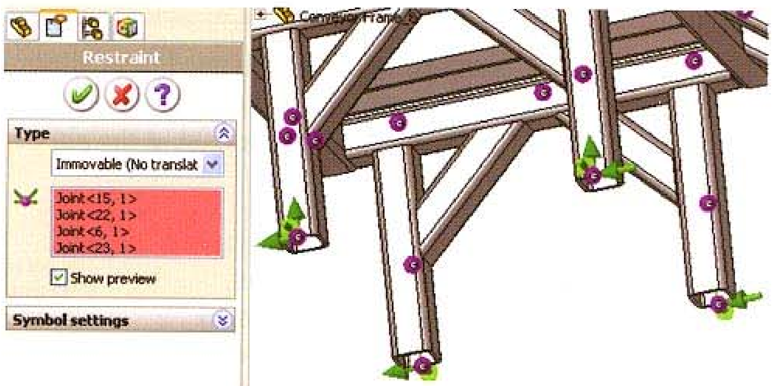
Select **Pin** for the upper connection between the joint and the beam.

Click **OK** to confirm the settings.



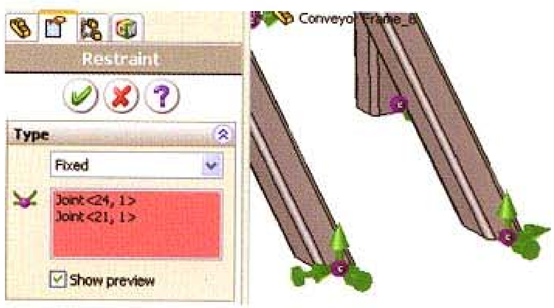
8 Restrain the vertical legs.

Apply an **Immovable** restraint to the bottom joints on all four vertical legs.



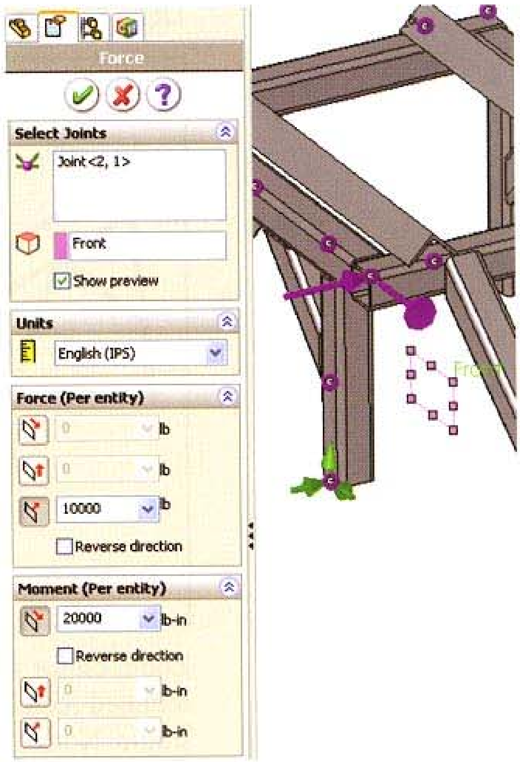
9 Restrain the inclined legs.

Apply a **Fixed** restraint to the bottom joints on the two inclined legs.

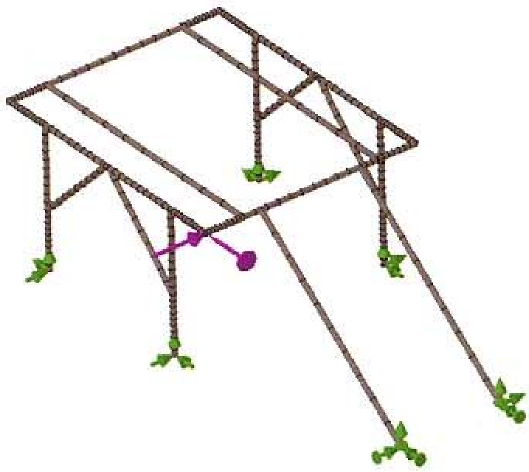


10 Define loads on corner joint.

Apply a **10,000 lb** force and a **20,000 lb in** moment to the corner joint. The force and the moment are oriented in the **Normal to Plane** and **Along plane Dir1** directions with reference to the `Front` plane, respectively.



11 Mesh the model.

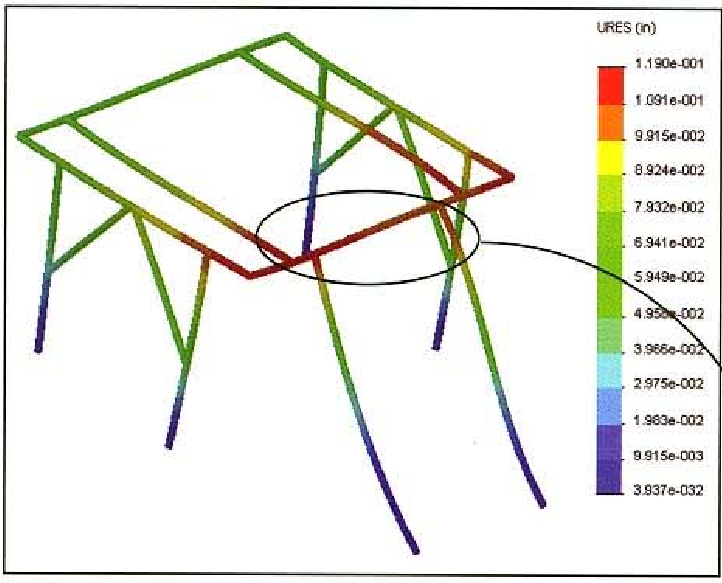


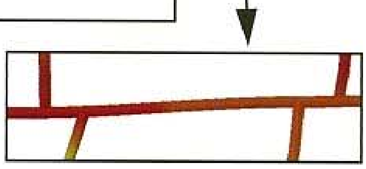
12 Run the analysis.

Note

Notice how quickly the study completes. If solid or shell elements were used instead, the computations would take considerably longer.

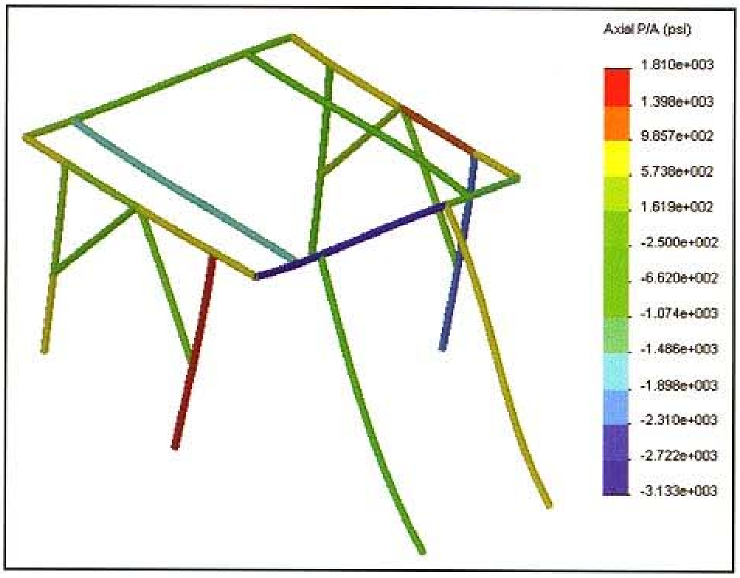
13 Plot resulting displacements.
Define a **RES: Resultant displacement** plot.





Zoom closer to the section where both inclined members connect the top of the frame. Notice that the member with the faulty weld rotated at the joint location, while the other member remains perpendicular irrespective of the structural deformations. This indicates that the faulty weld connection does not transmit moments, indeed.

14 Plot axial normal stress.
Define an **Axial** stress plot.

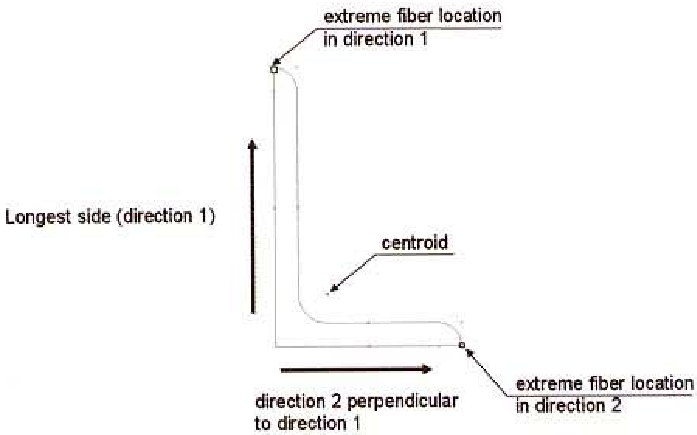


The Axial stress plot indicates a component of normal stress evenly distributed across the cross-section of the beam element caused by normal (axial) force. We observe a maximum value of approximately 1.8 ksi. The component is relatively small in this case.

Cross-section 1st and 2nd directions

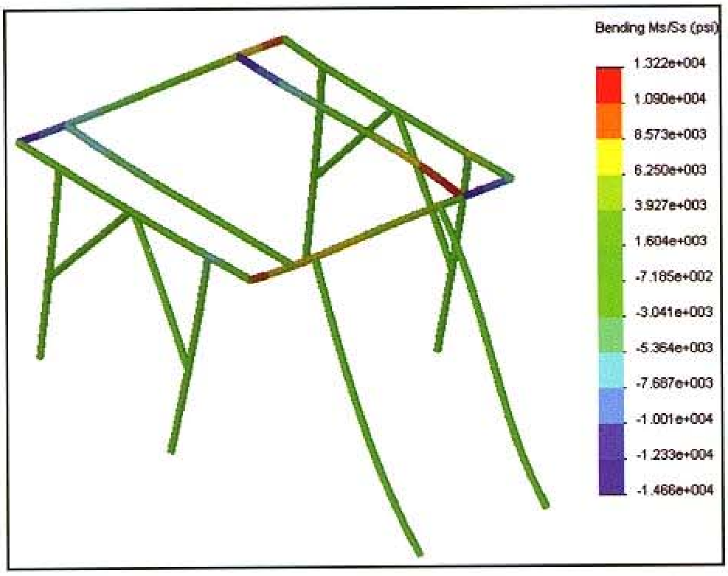
To post-process the bending component of the normal stress, 1st and 2nd directions must be specified.

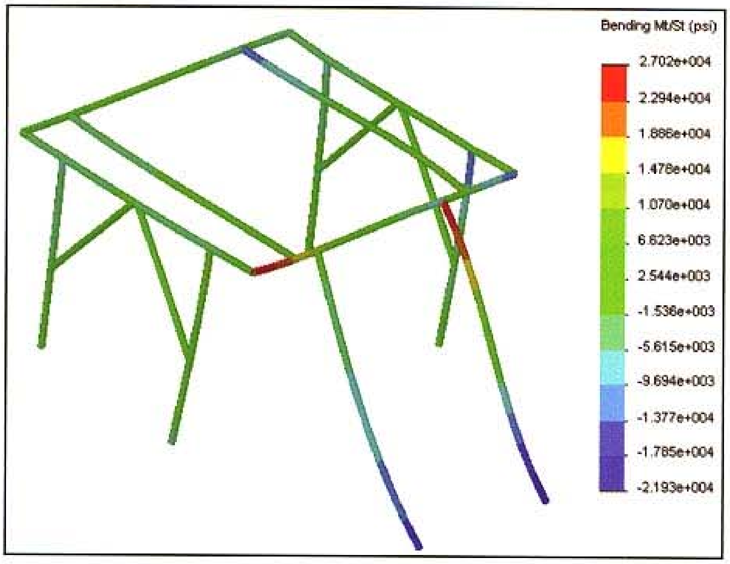
1st direction is defined along the longest side of the cross-section, and 2nd direction is perpendicular to it.



15 Plot normal stress due to bending.

Define **Bending in local direction1** and **Bending in local direction2** stress plots.





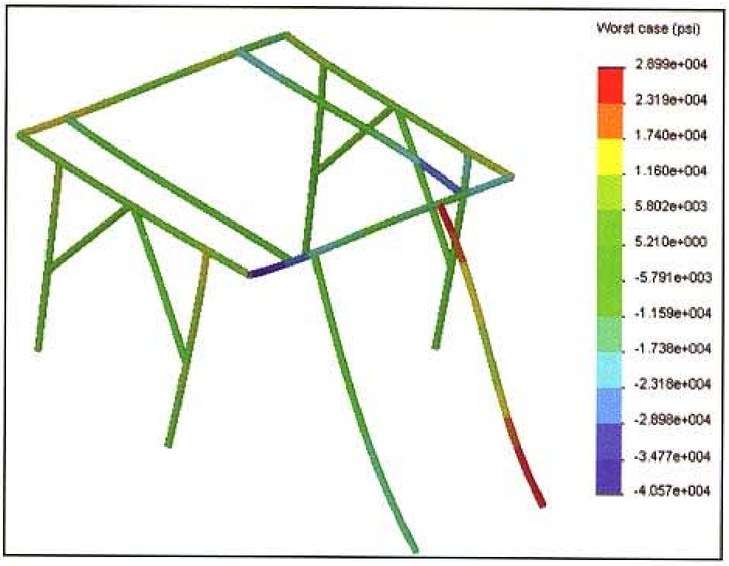
These plots indicate the maximum value of the component of normal stress (extreme fibers location) caused by the bending moment. We can observe a significantly greater value (than in the Axial stress plot) of 27.2 ksi.

Note

Similarly, **Bending in local direction2** shows the maximum value of the component of normal stress in the direction2 caused by the bending moment.

A total normal stress experienced by a cross-section is equal to the sum of the axial and bending components: the worst case stress plot.

16 Plot the extremes of the total normal stress.
Define a **Worst case** stress plot.



This plot adds the Axial and the Bending in local direction1 normal stresses. It is the plot of the most extreme normal stress experienced by the beam cross-sections.

We can see that the maximum normal stress of 29 ksi is still significantly smaller than the yield strength of Alloy Steel (90 ksi).

17 Save and close the model.

Summary

In this lesson, we analyzed a conveyor frame model constructed using the SolidWorks weldment feature. Since all the structural members were thin and long, we used beam elements. Their use can greatly simplify the analysis and make the computations significantly faster.

The model preparation consists of beam element and joint definition steps, both of which are automated in COSMOSWorks. If any two joints are generated too close, relative to the position of the remaining joints, they may be merged.

Because beam elements feature six degrees of freedom at each end, various possibilities for the joint / beam element connection exist. The connection types were discussed and practiced as well.

Lesson 13 Large Displacement Analysis of a Clamp

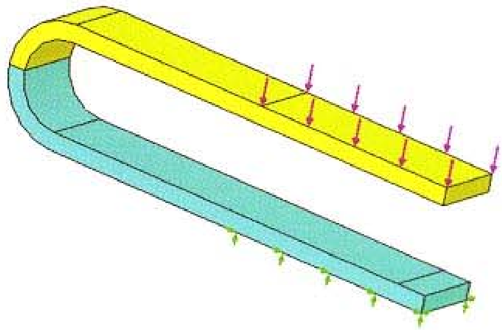
Objectives

Upon successful completion of this lesson, you will be able to:

- Understand the difference between geometrically nonlinear (large displacement and geometrically linear (small displacement) analyses
- Perform geometrically nonlinear (large displacement) analysis
- Assess limitations of the linear material model

Project Description

A clamp is bent with a 3,000 lb. force applied to one arm while the other arm rests on a rigid support (such as a steel block or concrete foundation).



It is known that this load deforms the clamp significantly and brings both of the arms in contact.

The goal is to ascertain whether this load causes the arms to touch and if the clamp remains permanently bent after removal of the load.

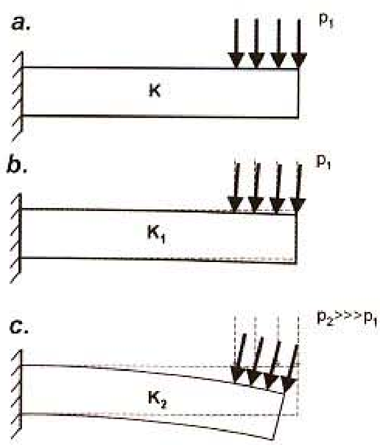
Small vs. Large displacement analysis

As explained in the beginning of this manual, COSMOSWorks Designer computations are limited to the small displacement class of problems (geometrically linear analysis). Here we will show that this limitation is actually not present and COSMOSWorks Designer is capable of solving large displacement nonlinear problems as well.

In small displacement analysis, it is assumed that the shape of the model before and after the deformation took place is nearly identical.

Consider a cantilever beam loaded by a pressure, as shown in figure *a* below. First, let us assume that our load is small in relation to the stiffness of the beam, resulting in deformations that are barely noticeable (figure *b* below). The stiffness of the deformed beam, $[\mathbf{K}_1]$ (which is a function of geometry and the material), will be nearly identical to the original stiffness of the undeformed beam, $[\mathbf{K}]$.

We can, thus, conclude that $[\mathbf{K}] \approx [\mathbf{K}_1]$, and that the linear elastic solution $[\mathbf{K}]\{\mathbf{u}\} = \{\mathbf{F}\}$ is valid as long as the above assumption is acceptable.



If the same beam is loaded with a significantly larger pressure, its deformation will become large. Because of the significant change in the geometry, the stiffness of the this beam, $[\mathbf{K}_2]$, is considerably different, and the linear elastic solution is no longer acceptable.

Cases *b* and *c* are commonly referred to as small displacement and large displacement problems, respectively.

Large displacement problems are of a nonlinear nature. They are significantly more complicated because they require a gradual increase of the load in small increments and elaborate iterational schemes to converge the solution to the equilibrium. They are very sensitive to the selection of the various analysis parameters and their solution requires some experience.

Small Displacement Linear Analysis

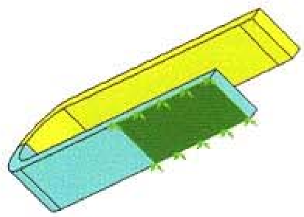
First, we will attempt to solve this problem as linear (with the assumption of small displacements).

1 **Open assembly.**
Open the assembly `clamp`, and define a study named `small displacements`. (**Static** analysis, **Solid** mesh.)

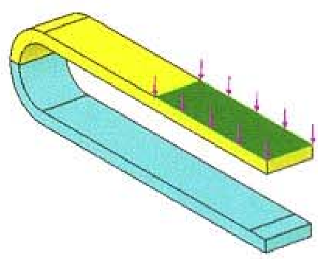
2 **Review material properties.**
The material properties of **Alloy Steel** are automatically transferred from SolidWorks.

3 **Apply restraint.**
Locate the two faces created by the split lines on the outside of the arms.

Apply an **Immovable** restraint to one face.

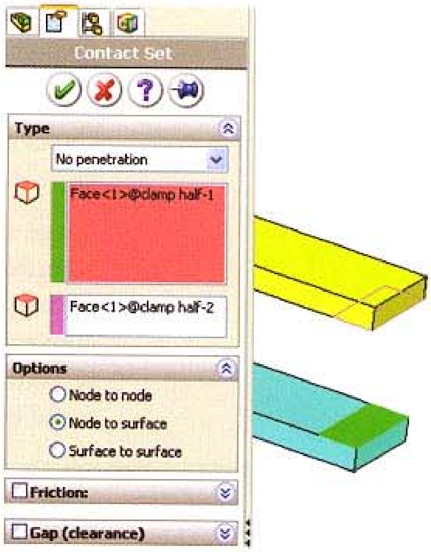


4 **Apply force.**
Apply a **3,000 lb.** force (use the **Apply normal force** option) to the other face.



5 Define surface contact set.

Define a **No penetration, Node to Surface** contact set between the two small faces at the end of the clamp arms.



6 Mesh assembly.

Mesh the assembly with **High** quality elements and the default element size of **0.31 in**.

7 Specify Direct Sparse solver.

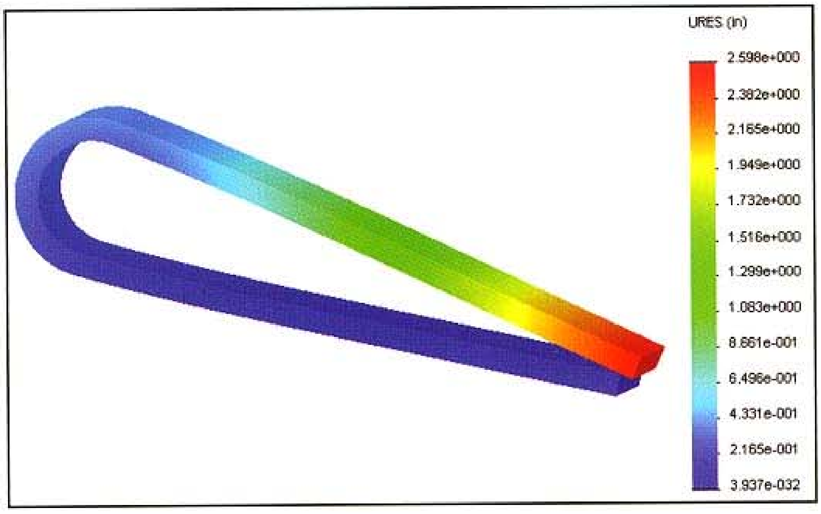
Direct sparse solver is considerably faster for this type and size of problem.

8 Run the analysis.

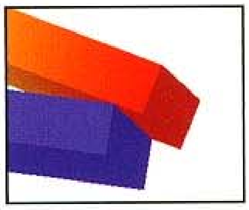
COSMOSWorks solver detects the large displacements in the model and issues a warning window (see the figure to the right).

Click **No** to complete the analysis as linear with small displacements.

9 **Plot resultant displacements.**
Create a **URES: Resultant displacement** plot in **True** deformation scale.



A quick review of displacement results reveals that the loaded arm has overextended the fixed arm. Obviously, this result is incorrect.



Results Discussion

The ignored warning of the solver along with the incorrect displacement results are sufficient reasons to invalidate the produced results. Therefore, we do not have to analyze the stresses.

Large Displacement Nonlinear Analysis

To obtain the correct solution, we must use large displacement formulation.

Before running the large displacement analysis let us go over the differences between the contact solutions in small and large displacement analyses.

Contact solution in Small and Large Displacement Analyses

In small displacement analysis, the normals to the contact areas do not change directions during the loading. This implies that the direction of the normal and friction forces remains fixed as well.

Contrarily, in the large displacement analysis, the directions of the normal and friction forces are updated during the deformation process. Because of the potential significant displacements and sliding in the contact regions during the large displacement analysis, the Node-to-Node (No penetration) contact option should not be used.

For more information on the contact solution in a geometrically nonlinear analysis, consult the COSMOSWorks Nonlinear training manual.

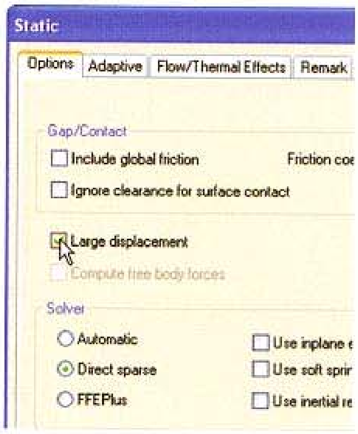
10 Create new study.

Copy the study named `small displacements` into a new study named `large displacements`.

11 Set study properties.

Right-click study `large displacements` and select **Properties**. Under the **Options** tab select **Large displacement**.

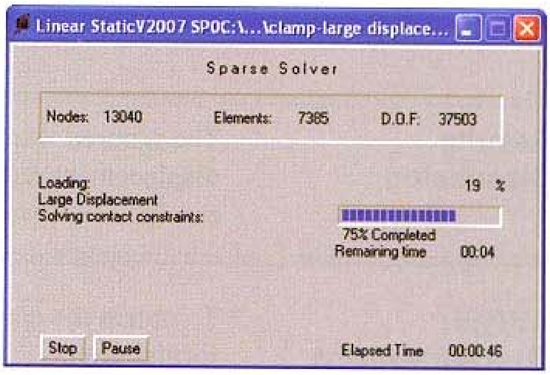
Click **OK**.



12 Run the analysis.

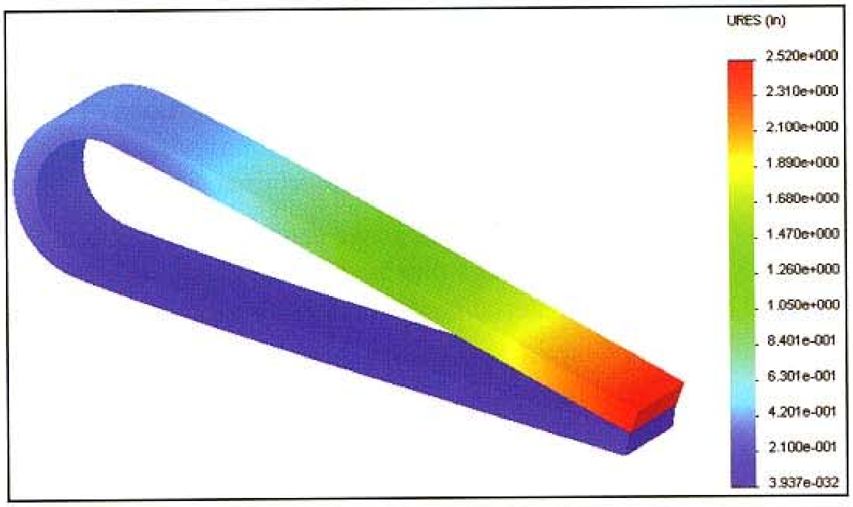
Note that the solution takes significantly longer due to the extra time required to increment the load in steps, as discussed earlier.

The progress dialog window shows the percentage of the total load applied as well as the solution progress at the current step.

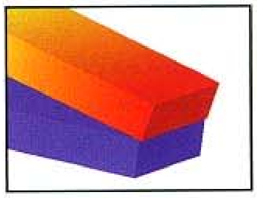


13 Plot resultant displacements.

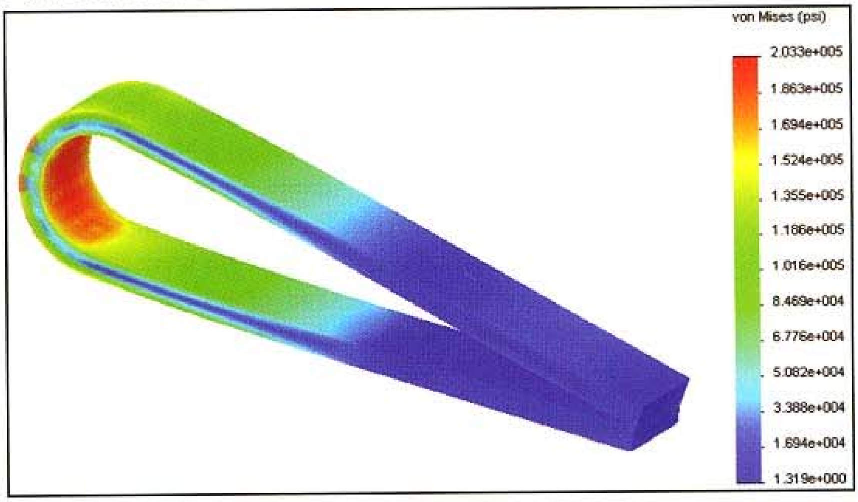
Plot the distribution of **URES: Resultant displacements**.



We can observe that the detail of the displacements at the tip location is what we would expect, i.e. the tip edges of the contact surfaces are nearly touching.



14 Plot von Mises stresses.



Stress results are also consistent with what we expect to see in a bending problem such as this.

Note

No high stress shows in the contact region because the mesh size is too large to capture these localized contact stresses.

Observing that contact area is very small (in fact, the solution presents it as a line contact) we conclude that our choice **Node to surface** option in **No penetration** contact set definition is correct.

While stress results obtained using the large-displacement contact option are generally correct, a closer examination reveals some problems.

Permanent Deformation

It is evident that significant portions of the clamp experience stresses above the yield stress. Therefore, after the load is removed, the clamp will not return to its original shape.

This is as far as we can take this problem using a linear material model (the large displacement analysis is geometrically nonlinear, but the material model is linear elastic).

To determine the shape and residual stresses in a permanently deformed clamp after the load has been removed, the analysis must include a nonlinear material model. This option is available in COSMOSWorks Nonlinear modulus which is part of the COSMOSWorks Advanced Professional suite.

COSMOSWorks Advance Professional

The presence of a geometrically nonlinear solver (large displacement option) in COSMOSWorks Designer provides the user with the very powerful feature to solve problems out of the scope of the geometrically linear static study. However, the solution of these problems in general requires the correct setup of various parameters and solver options. Because a large displacement modulus of COSMOSWorks Designer uses a predefined set of parameters, its solution success is limited.

All of the features and options of the advanced nonlinear solver are available in COSMOSWorks Nonlinear modulus which is part of the COSMOSWorks Advanced Professional suite. Furthermore, multiple advanced material models are available in COSMOSWorks Advanced Professional only. Users who wish to take their COSMOSWorks expertise to the next level are encouraged to inquire about COSMOSWorks Advanced Professional suite and to take a COSMOSWorks Nonlinear training course.

Summary

In this lesson, we ventured into the next level of FEA analysis and discussed and practiced the basic characteristics of the geometrically nonlinear (large displacement) analysis. The limitations of the geometrically linear (small displacement) analysis were discussed as well.

We first attempted to solve the problem using a small displacement formulation, but erroneous displacement results indicated the need to consider this analysis as a large displacement problem.

In the large displacement problem, the load was applied in steps and the model stiffness was updated during the deformation process. This process took longer to solve, but was required to obtain accurate results.

Stress results indicated that the clamp will remain permanently deformed after the load has been removed, but for a quantitative analysis of this deformation a nonlinear material analysis (available in COSMOSWorks Advanced Professional suite) would be required.

Finally, the significance of a COSMOSWorks Advanced Professional suite upgrade for users interested in nonlinear FEA was discussed.

Appendix A
Meshing, Solvers, and Tips & Tricks

Meshing Strategies

Meshing, more precisely called discretization, is what converts a mathematical model into a finite element model ready for solution.

As a finite element method, meshing accomplishes two tasks. First, it replaces a continuous model with a discrete one. Meshing, therefore, reduces the problem to one with a finite number of unknowns suitable for solution with an approximate numerical technique. Second, it represents the desired solution (e.g., displacements or temperatures) with an assembly of simple polynomial functions defined individually for elements. See the Introduction to FEA section of the manual for a description of this process.

For the user, meshing is a necessary step towards the problem solution. Many new FEA users expect meshing to be a fully automated process requiring little, in any, input from the user. With experience comes the realization that meshing is often a demanding task.

The history of development of commercial FEA software witnessed many attempts to make meshing invisible to FEA users, but this has not been a successful approach.

While the meshing process has been simplified and automated, it is still not a "hands-off" task that runs in the background. As FEA users, we require a means to interact with the meshing process.

COSMOSWorks finds the fine balance by isolating us from those issues that are purely meshing-specific, but providing us control over meshing when needed.

Geometry Preparation

Ideally, we use SolidWorks geometry, toggle to COSMOSWorks, where we define the type of analysis and material, apply loads and restraints, and then we mesh the geometry and obtain the solution.

This approach works well for simple models. More complex geometry requires preparation before it can be meshed. In the process of geometry preparation for FEA, we depart from manufacturing-specific, CAD geometry and construct geometry intended specifically for analysis. We call this geometry FEA geometry.

We differentiate between CAD geometry and FEA geometry based on their different requirements:

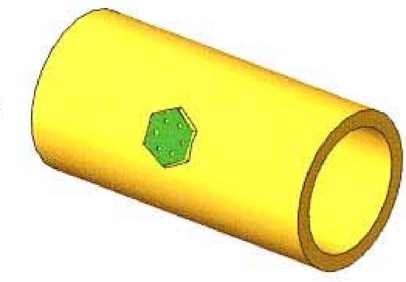
CAD Geometry	FEA Geometry
Must contain all information necessary for manufacturing	Must be meshable
	Must allow for creation of a mesh that will correctly model the data of interest
	Must allow for creation of a mesh solvable within a reasonable time

Often CAD geometry does not satisfy the requirements of FEA geometry. CAD geometry serves as a starting point in the process of FE model preparation, but is only seldom used for FEA without modifications.

We now describe several actions performed on manufacturing-specific, CAD geometry in order to convert it into FEA-specific geometry.

Defeaturing

CAD geometry contains all the features necessary to make a part. Many of those features are unimportant for analysis and should be suppressed prior to meshing.



At best, leaving such features results in an unnecessarily complicated mesh and a long solution time. At worst, it may prevent the mesher from completing its task.

Of course, determining which features to exclude and which to include in the FE model requires careful engineering judgment. The small size of a feature as compared to the overall size of the model does not always justify its exclusion. For example, very small internal fillets should be retained in the model if the objective of the analysis is to find stresses in the area of the round.

Idealization

Idealization modifies CAD geometry more substantially than defeaturing. Idealization may, for example, involve converting 3D, solid-CAD geometry into surface geometry suitable for subsequent meshing with shell elements.

COSMOSWorks creates surface geometry automatically if **Shell mesh using mid-surfaces** is selected as the **Mesh type**.

CAD geometry meshed with solid elements

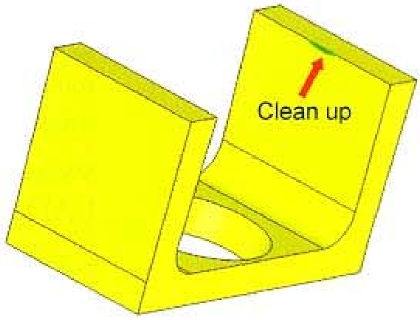
Idealized geometry meshed with shell elements

It is also possible to create a shell-element mesh on selected faces or mesh the surface geometry constructed specifically for FEA.

Note that idealization creates an abstract geometry (zero thickness surface) suitable exclusively for analysis.

Clean-up

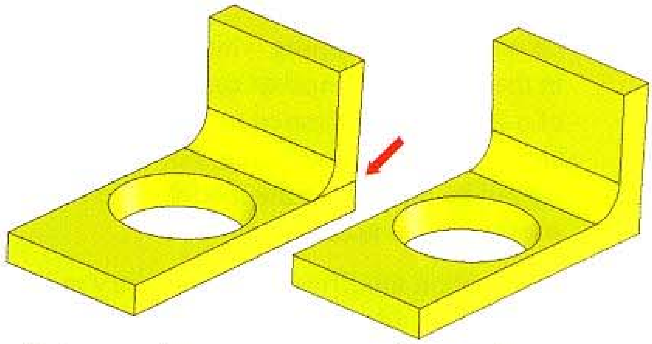
Clean-up refers to issues of geometry quality that must be dealt with to enable correct meshing.



Geometry that is adequate for manufacturing purposes may contain features that either do not mesh or force the mesher to create a large number of elements or create distorted elements. Examples include very short edges and/or faces. Those small features must be removed or the automesher tries to mesh them.

Mesh creation may also fail because of quality issues, including multiple entities, floating solids, and other quality problems.

To avoid creating elements with tangent edges (see see *Mesh Quality* later in this appendix), geometry faces may have to be merged.



Before merging After merging

Mesh Quality

Creating a solid-element mesh can be likened to a process of filling up a volume with tetrahedral elements, while creating a shell-element mesh can be likened to filling up a surface with triangles.

Recall from the Introduction to FEA section of this manual, that in the vast majority of problems, the second-order, tetrahedral elements and second-order, triangular elements map to curvilinear geometry and are much easier to work with when meshing and analyzing.

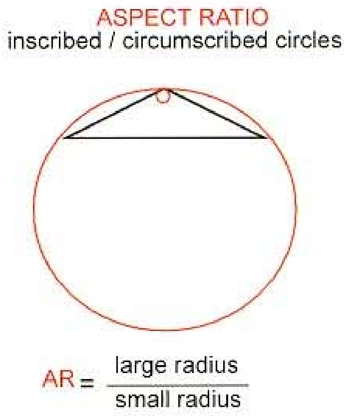
This observation exemplifies the fact that elements experience distortion during meshing, which brings us to the issue of mesh quality.

While elements are almost always distorted in the process of mapping to geometry, excessive distortion leads to element degeneration.

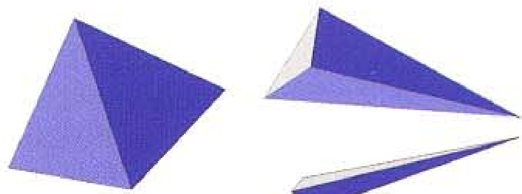
Mesh degeneration can often be prevented by controlling the default element size or applying local mesh or component controls. We have practiced mesh controls in many lessons. Now we discuss the most important forms of element distortion.

Aspect Ratio Check

Numerical accuracy is best achieved by a mesh with uniform, perfect, tetrahedral or triangular elements whose edges are equal in length. For a general geometry, it is not possible to create a mesh of perfect, tetrahedral elements. Due to small edges, curved geometry, thin features, and sharp corners, some of the generated elements can have some edges much longer than others. When the edges of an element become much different in length, the accuracy of the results deteriorates.



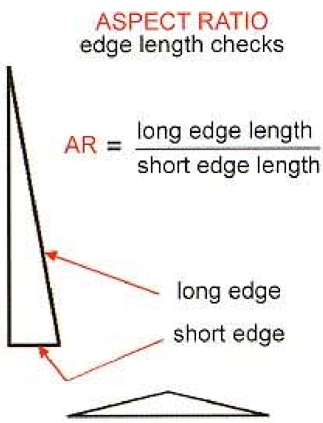
The aspect ratio of a perfect, tetrahedral element is used as the basis for calculating aspect ratios of other elements. The aspect ratio of an element is defined as the ratio between the longest edge and the shortest normal dropped from a vertex to the opposite face normalized with respect to a perfect tetrahedral. By definition, the aspect ratio of a perfect tetrahedral element is 1.0. The aspect-ratio check is automatically used by the program to check the quality of the mesh and assumes straight edges connecting the four corner nodes.



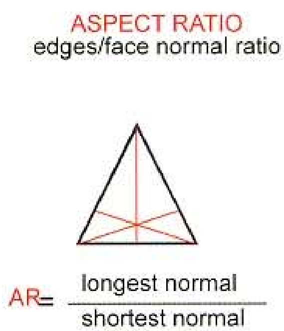
Correct element shape

Excessively distorted elements

As part of the aspect-ratio check, COSMOSWorks performs an edge-length check, a radius of inscribed and circumscribed radius check and a length of normals check.



This aspect-ratio measure does not recognize "flat" elements as bad.



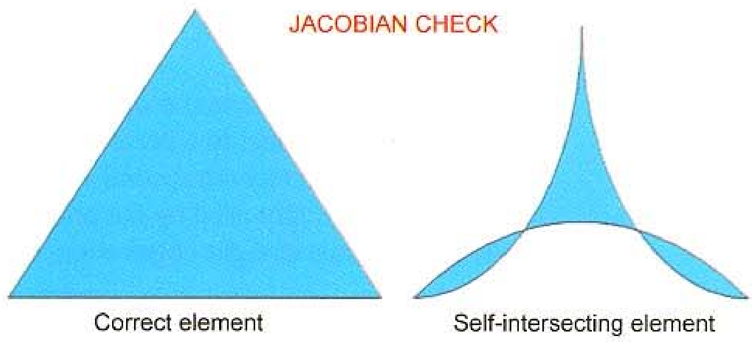
Jacobian Check

Second-order elements map to curved geometry much more accurately than linear elements of the same size. The mid-side nodes of the boundary edges of an element are placed on the actual geometry of the model. In sharp or curved boundaries, placing the mid-side nodes on the actual geometry can result in generating distorted elements with edges overlapping each other.

The Jacobian of an extremely distorted element becomes negative. An element with a negative Jacobian causes the analysis program to stop.

The Jacobian check is based on a number of points located within each element. COSMOSWorks gives you a choice to base the Jacobian check on 4, 16, or 29 Gaussian points or **At Nodes**.

The Jacobian ratio of 1.0 is given to a parabolic, tetrahedral element with all mid-side nodes located exactly at the middle of the straight edges. The Jacobian ratio increases as the curvatures of the edges increase. The Jacobian ratio at a point inside the element provides a measure of the degree of distortion of the element at that location. COSMOSWorks calculates the Jacobian ratio at the selected number of Gaussian points for each tetrahedral element.

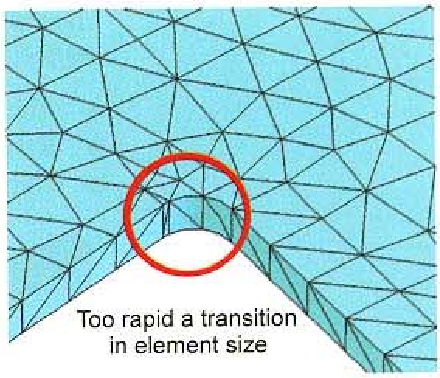


It is generally the case that a Jacobian ratio of 40 or less is acceptable. COSMOSWorks adjusts the locations of the mid-side nodes of distorted elements automatically to ensure that all elements pass the Jacobian check.

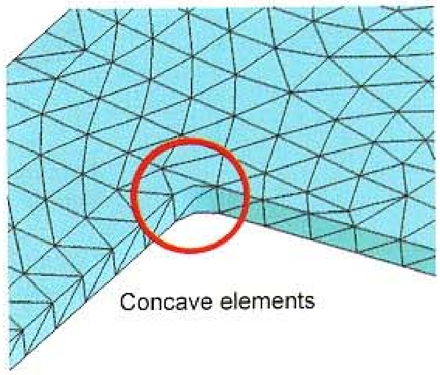
Even if this check of mesh quality does not issue warnings, avoiding elements that are too "concave" is generally good practice. This can be accomplished by using mesh controls or adjusting the global element size.

Note

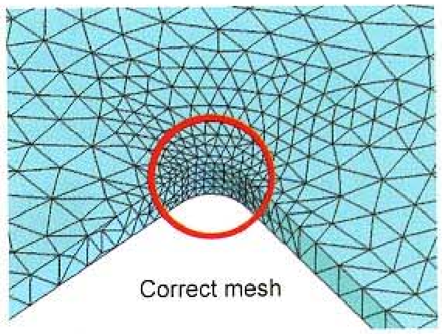
COSMOSWorks tries to place two elements over a 90° arc. This, combined with global elements that are too large, may lead to very small elements placed next to large elements.



If arc is larger than 90°, one element is placed over the arc leading to the creation of elements with "concave" faces.



Applying mesh controls (here to the round face) allows for the creation of a correct mesh.

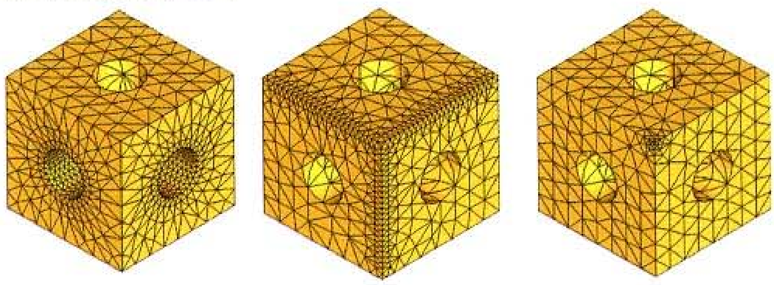


Mesh Controls

We have practiced the use of mesh controls in many lessons. For easy reference, we review them now.

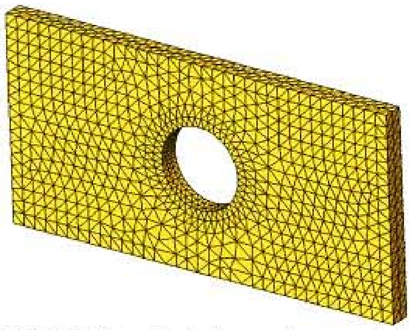
Generally, mesh controls can be applied to faces, edges, vertices, and assembly components.

Mesh control applied to:

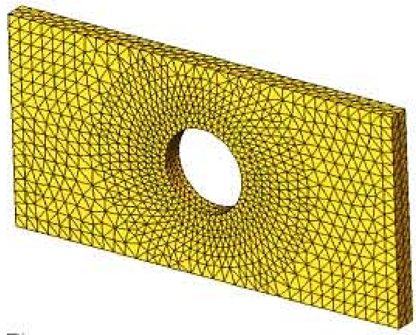


The definition of mesh controls applied to a part consists of specifying the following:

- Element size on the selected entity
- Ratio of element size between the layers
- Number of element layers to be affected by local refinement



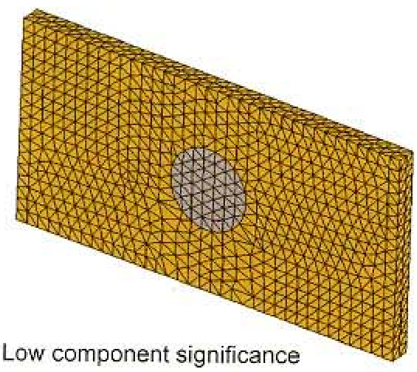
Element size ratio between layers = 1.5
Number of transition layers = 3



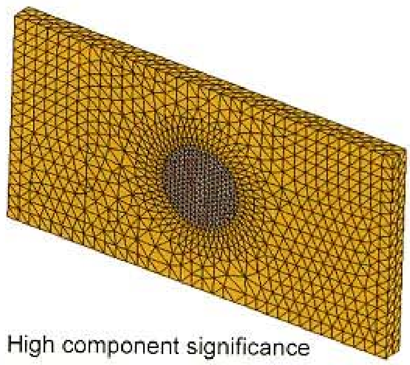
Element size ratio between layers = 1.1
Number of transition layers = 6

The definition of mesh controls applied to a component consists of specifying the **Component significance**, which instructs the mesher, based on the position of the slider, to use a different element size for each selected component.

The left end of the slider corresponds to using the default global-element size of the assembly. The right end of the slider corresponds to using the default element size if the component is meshed independently.



Low component significance



High component significance

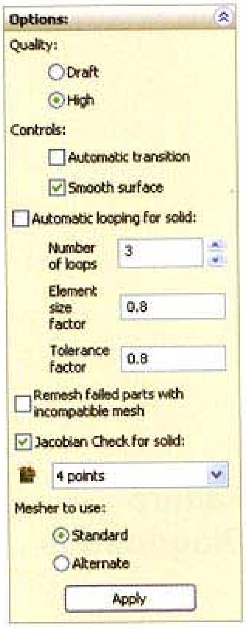
If the option **Use the same element size** is selected, then all selected components are meshed with the same element size as specified in **Mesh Control** window.

Automatic Looping

Many meshing problems can be solved by using a smaller element size. Using a smaller element size, of course, results in longer solution times.

To find the largest element size that still meshes, we can use **Automatic looping**, specified in the meshing options.

Automatic looping instructs the mesher to automatically mesh the model again using a smaller, global element size. You control the maximum number of trials allowed and the ratio by which the global element size and tolerance are reduced each time.



Meshing Stages

Meshing proceeds in three steps:

- Evaluating the geometry
- Processing the boundary
- Creating the mesh

Meshing problems may arise at any step.

During the first step, evaluating the geometry, COSMOSWorks checks the geometry imported from SolidWorks. Geometry import is completely transparent to the user.

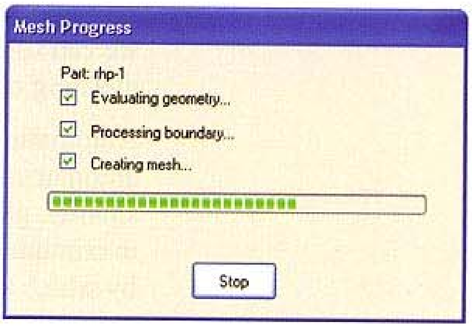
The actual meshing of a solid component consists of two phases. When processing the boundary, the mesher places nodes on the boundary. This phase is called surface meshing. If the this phase is successful, the third phase, creating the mesh, starts as the volume is filled with tetrahedral elements.

If meshing fails when evaluating the geometry, the most likely cause is a geometry error. To verify if geometry error is the cause, export the geometry as an IGES to see if the error message, "Failed to process trimmed surface entity" is displayed. If this message appears, send the part to SolidWorks support for diagnosis of the geometry problem.

When processing the boundary, if meshing fails before the progress indicator reaches the right end of the progress bar, the failure is due to an error meshing at least one face. Right-click `Mesh`, and select **Failure Diagnostics** to find the face causing problems. Use a split line or mesh control to help mesh that face.

When processing the boundary, if meshing fails after the progress indicator has reached the right end of the progress bar, but before the second check mark appears, then remesh with the tolerance of the element size increased from the 5% default to 10%. You can continue increasing the tolerance if 10% fails, but do not exceed 25% tolerance.

If the mesh fails when creating the mesh, the failure is occurring in the volume-filling stage. Reduce the tolerance from 5% to 1% of the element size. If the mesh still fails, reduce the element size by 25% and set the tolerance to 1%.

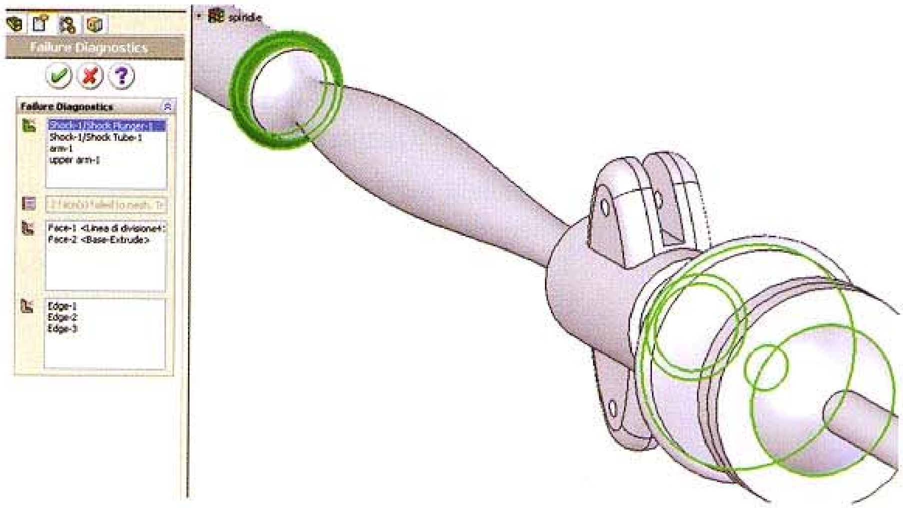


Failure Diagnostics

When meshing fails, COSMOSWorks displays a message and stops unless the automatic mesh looping is active. A failure diagnostics tool is provided to help you locate and resolve solid-meshing problems.

The Failure Diagnostics PropertyManager lists the components, faces, and edges that fail. It also highlights the failed entities in the model window.

To review the entities that prevented successful meshing, right-click `Mesh` and select **Failure Diagnostics**.



The offending entities are listed in **Failure Diagnostics** window and highlighted in the graphics window.

The **Failure Diagnostics** tool is available for a solid-element mesh, but not for a shell-element mesh.

Tips for Meshing Parts

Check for underdefined sketches.

Use SolidWorks Utilities to find sliver faces, knife edges, and so on.

For meshing failures on faces, create a shell study and select only the failed face. Then try various element sizes until that face meshes.

If the mesh failure diagnostics do not provide enough information to determine the exact location of the problem, successively cut portions off the model to isolate the region of failure, or roll back the SolidWorks models until the model meshes.

Tips for Meshing Assemblies

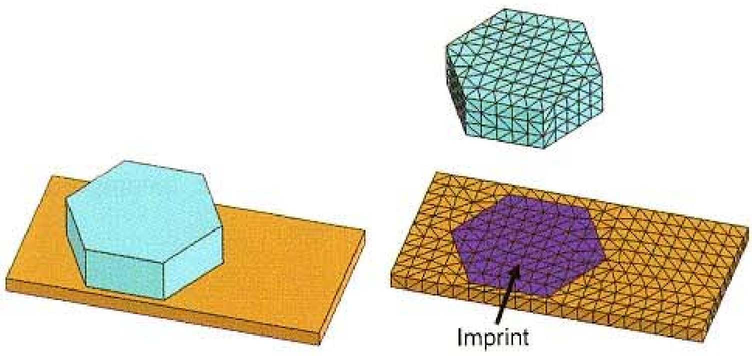
Select **Tools, Interference Detection** to determine where parts interfere and where faces touch (coincident). Remember that interference is allowed only if the shrink fit contact condition is defined.

Do not model line contact (such as a cylinder tangent to a plate) or point contact (such as the top of a cone touching a plate) between assembly components. The area of contact should be > 0.

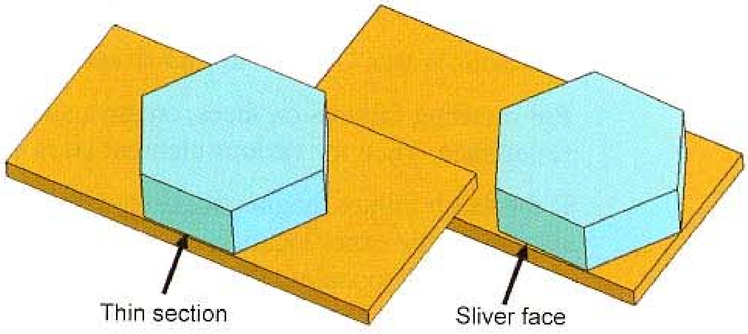
Note that when COSMOSWorks meshes an assembly, "imprints" are made on all touching faces, allowing the nodes from both components to align.

If bonded contact conditions have been defined, then the same node is shared by both components. If node to node or surface conditions have been defined, two coincident nodes are created and joined by gap elements. Gap elements remain invisible to the user.

Note that the color of the imprint in the following illustration has been modified in a graphics program to make it clearly visible.



Beware of imprints that cause sliver faces, thin annular faces, or faces with multiple "lobes" connected by thin sections.

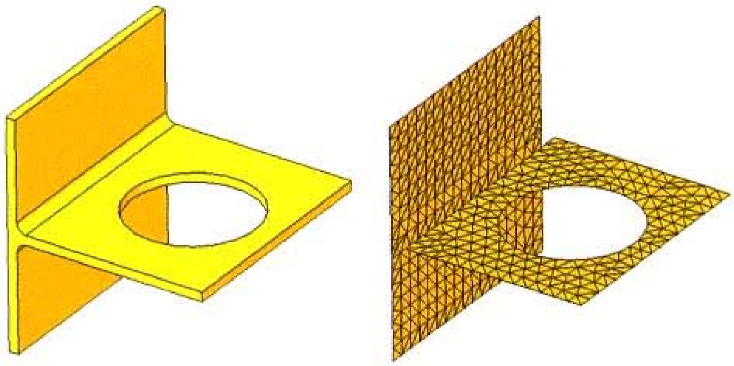


Tips for Using Shell Elements

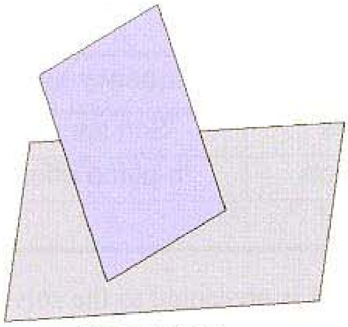
Shell meshing uses only the surface meshing phase; no volume filling occurs.

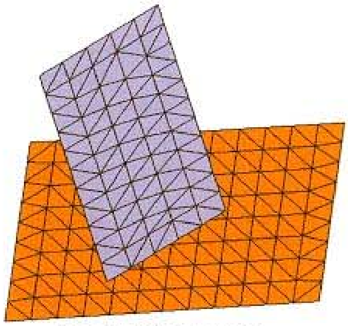
Although the use of shell elements results in a simpler model that solves faster than a corresponding solid-element model, preparation of a shell-element mesh is more time consuming as compared to a solid-element mesh.

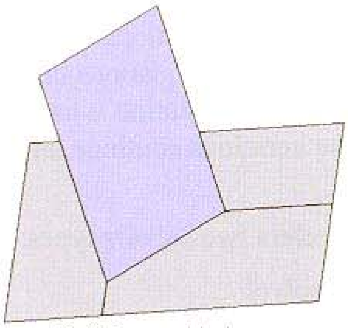
Meshing in mid-planes often results in disjointed meshes.

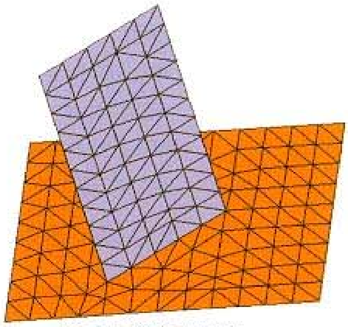


If surface geometry is to be meshed, split lines may be used where surfaces meet to ensure alignment of the nodes and, consequently, mesh compatibility. However, incompatible mesh with misaligned nodes is also allowed!


No split lines


Incompatible mesh

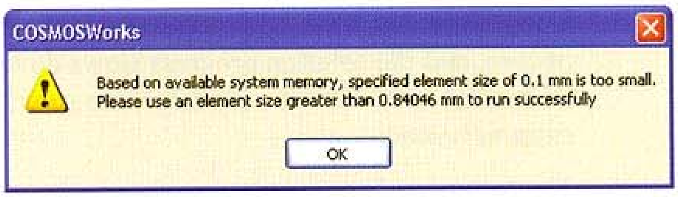

Split lines added


Compatible mesh

Hardware Considerations in Meshing

Meshing is the most critical step on the way to obtaining a solution. The maximum mesh size, meaning the smallest, element size that can be used, depends on the size of RAM memory.

While the simple rule "the more the better" applies, we recommend 2 GB for working with real-life, complex models.



Solvers in COSMOSWorks

Having successfully meshed the model we are only one step away from obtaining a solution.

Generally, if a model can be meshed, it will solve; solving is a less critical step than meshing.

However, several problems can arise. The solver may find problems with model definition, such as no definition of material or loads. The kinds of issues that prevent solution depend, of course, on the type of analysis (static, frequency, and so on).

The solver may also detect rigid body motions due to insufficient restraints. Rigid body motions can be dealt with using solver options, such as **Use soft spring to stabilize model** or **Use inertial relief**.

Available solver options depend on the type of analysis.

Static analysis	Frequency analysis	Buckling analysis
Soft springs	Soft springs	Soft springs
In-plane effects	In-plane effects	
Inertial relief		

The meshed model is presented to the solver in the form of a large number of linear algebraic equations. Those equations can be solved with two classes of solution methods: direct and iterative.

Direct methods solve the equations using exact numerical techniques. Iterative methods solve the equations using approximate techniques where, in each iteration, a solution is assumed and the associated errors are evaluated. The iterations continue until the errors become acceptable.

COSMOSWorks offers two solvers types:

- Direct Sparse solver
- FFEPlus (iterative)

Choosing a Solver

In general, all solvers give comparable results if the required solver options are supported. While all solvers are efficient for small problems (25,000 degrees of freedom or less), big differences in performance (speed and memory usage) occur in solving large problems.

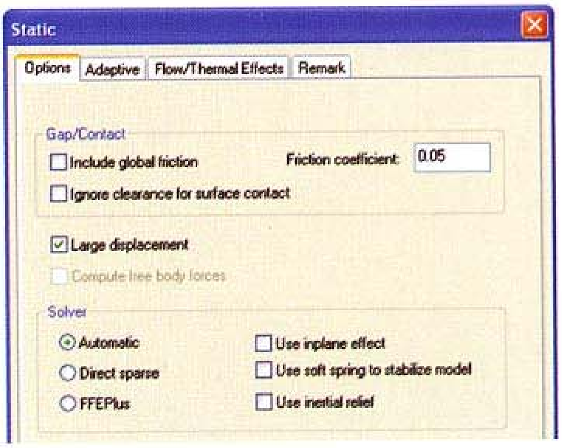
If a solver requires more memory than available on the computer, the solver uses disk space to store and retrieve temporary data. When this situation occurs, a message appears saying that the solution is going out of core, and the solution progress slows down. If the amount of data to be written to the disk is very large, the solution progress can be extremely slow.

The following factors help you choose the proper solver:

- **Size of the problem**
  In general, FFEPlus is faster in solving problems with degrees of freedom (DOF) over 100,000. This solver becomes more efficient as the problem gets larger.
- **Computer resources**
  The Direct Sparse solver, in particular, becomes faster with more memory available on your computer.
- **Analysis options**
- **Element type**

- **Material properties**
  When the moduli of elasticity of the materials used in a model are very different (like Steel and Nylon), iterative solvers are less accurate than direct methods. The Direct Sparse solver is recommended in such cases.

A solver can be selected in the study properties. Since the choice of the most suitable solver requires some experience, an automatic selection has been implemented as well. Use this option if you are not sure which solver is best suited for your analysis.



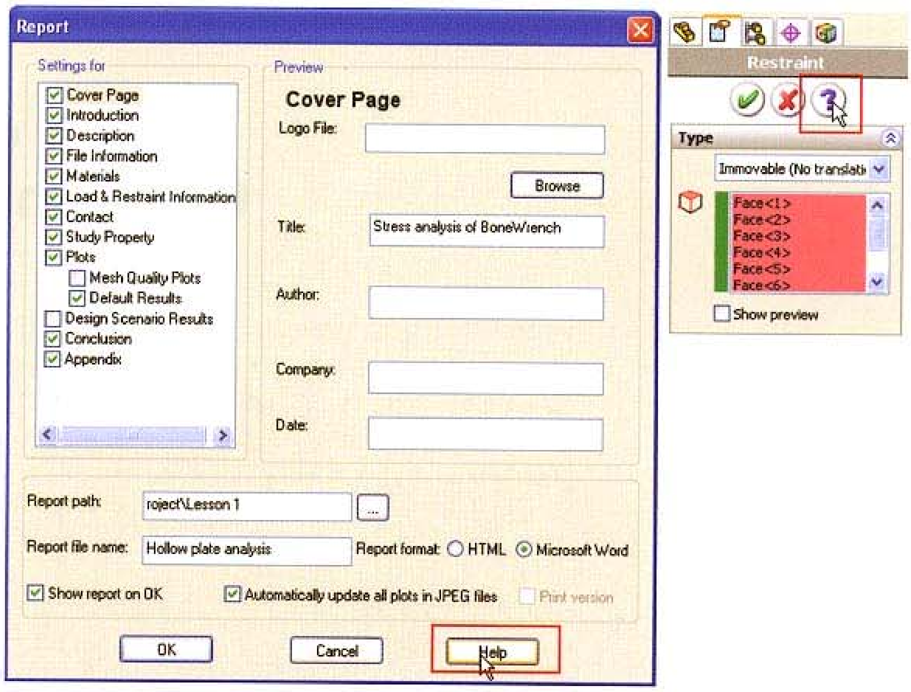
Appendix B
Customer Help and Assistance

Customer help and assistance

COSMOSWorks features extensive apparatus to help you with various information needs.

COSMOSWorks Help

Nearly every dialog window contains a help icon. Use this as your initial help option. Here you will find answers to most of the common questions relevant to the desired topic or COSMOSWorks function.

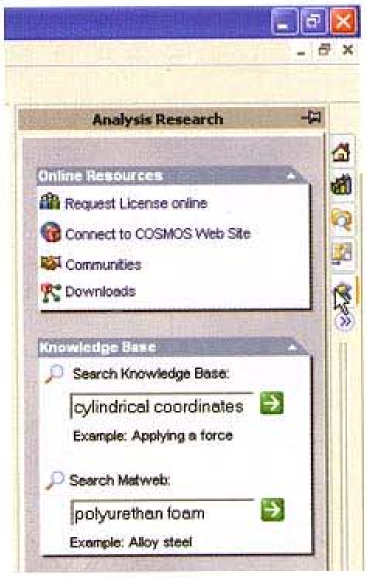


The above figures show how to access COSMOSWorks help files from the most common dialogs that you may encounter in COSMOSWorks.

Online Resources

Right-click on **COSMOSWorks** and select **Research** to display the **Analysis Research** dialog.

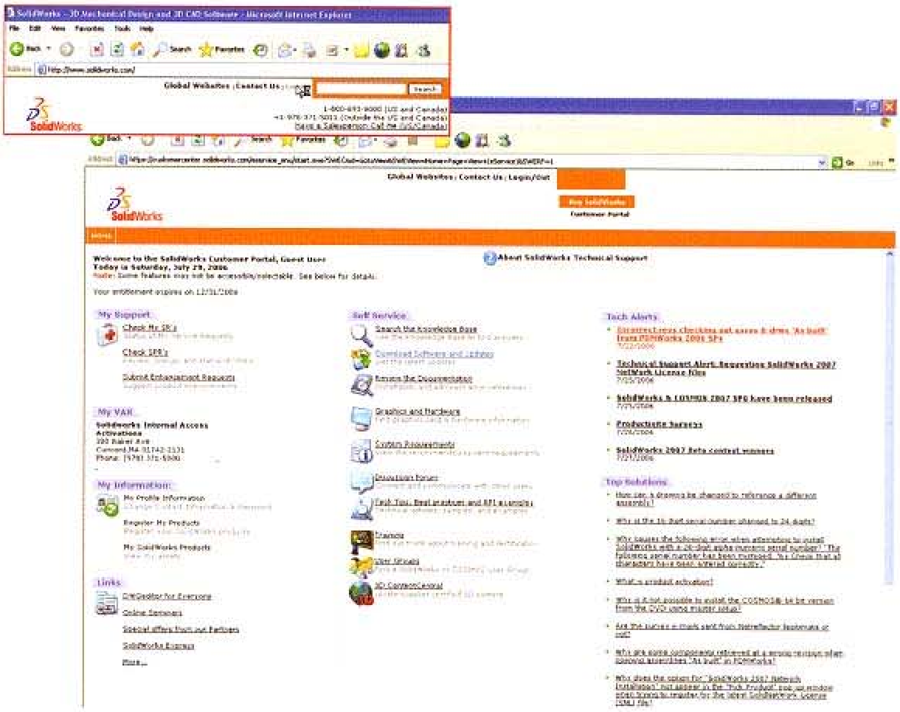
This area contains countless resource information in organized and accessible form. The functionality of some of the links in the **Analysis Research** dialog is described in the following text.



- **Search Knowledge Base**
  This database contains numerous targeted, well-organized and maintained articles on various topics in analysis theory, COSMOSWorks usage and troubleshooting, licensing, and many other practical areas. We strongly encourage users to use this feature as often as possible. Valid subscription and internet connection are required to access this information database.
- **Search Matweb**
  A search query in this field will take you directly to the free online material database matweb.com (free Premium membership is also available). Internet connection is required to use this feature
- **Downloads**
  The latest upgrades and service packs (SP) can be downloaded from the web download page that can be accessed via this feature.

Customer Support Portal

Complete customer account information and customer service and maintenance links can be accessed via the Customer service portal accessible from www.solidworks.com website. The portal allows you to submit service and enhancement requests, search the knowledge base, view the information on the online seminars and various discussion forums and much more.



Customer Phone Support

Subscription customers have access to the dedicated phone and email technical support. Please contact your local VAR for the local technical support telephone number and email address. A customer serial number is always required.

Index