CosmosWorks Displayed:
Basic Theory and Applications

Prof. Ed Akin
Rice University
Houston, Texas

akin@rice.edu

September 10, 2007

1 CosmosWorks Overview

1.1 Capabilities

The CosmosWorks software offers several types of studies including:

Static: Static (or Stress) studies calculate displacements, reaction forces, strains, stresses, failure criterion, factor of safety, and error estimates. Available loading conditions include point, line, surface, acceleration and thermal loads are available. Elastic orthotropic materials are available.

Thermal: Thermal studies calculate temperatures, temperature gradients, and heat flow based on heat generation, conduction, convection, contact resistance and radiation conditions. Thermal orthotropic materials are available.

Frequency: A body tends to vibrate at natural, or resonant, frequencies. For each natural frequency, the body takes a certain shape called the mode shape. Frequency analysis calculates the natural frequencies and the associated mode shapes.

Buckling: A buckling study calculates a load factor multiplier for axial loads to predict when the axial loads will cause large catastrophic transverse displacements. Slender structures subject to axial loads can fail due to buckling at load levels far lower than those required to cause material failure.

Fatigue: Fatigue studies evaluate the consumed life of an object based on fatigue events. Repeated loading weakens materials over time even when the induced stresses are low. The number of cycles required for failure depends on the material and the stress fluctuations. Those data are provided by a material curve, called the S-N curve, which depicts the number of cycles that cause failure for different stress levels.

Optimization: Optimization studies automate the search for the optimum design based on an initial geometric design. Optimization studies require the definition of the following:

Objective. State the objective of the study, for example, the minimum material to be used.

Design Variables. Select the dimensions that can change and set their allowed ranges.

Behavior Constraints. Set the conditions that the optimum design must satisfy. For example, you can require that a stress component does not exceed a certain value and the natural frequency be within a specified range.

Nonlinear Static: When the assumptions of linear static analysis do not apply, you need to use nonlinear studies to solve the problem. The main sources of nonlinearity are: large displacements, nonlinear material properties, and contact. Nonlinear studies calculate displacements, reaction forces, strains, and stresses at incrementally varying levels of loads and restraints.

Nonlinear Thermal: Cosmos solves a nonlinear thermal problem based on temperature dependent material properties, restraints and sources.

Drop Test: Drop test studies evaluate the impact effect of dropping the design on a rigid floor. You can specify the dropping distance or the velocity at the time of impact in addition to gravity. The program solves a dynamic problem as a function of time using explicit integration methods. After the analysis is completed, you can plot and graph the time history of the displacements, velocities, accelerations, strains, and stresses.

Like most commercial finite element systems CosmosWorks has many capabilities and the average user only uses a few of them. Table 1-1 lists those abilities that the author thinks are most useful. Those with an asterisk are illustrated in this book. CosmosWorks comes with a very good set of tutorials. Here Cosmos will be introduced by examples intended to show basis capabilities generally not covered in the tutorials. Also some tutorials do not have the space to discuss good engineering practice. Here one goal is to introduce such concepts based on the author's decades of experience in applying finite element (FE) methods. For example, you should always try to validate a FE calculation with approximate analytic solutions and/or a different type of finite element model. This is true for even experienced persons using a program with which they have not had extensive experience. Sometimes you might just misunderstand the supporting documentation and make a simple input error.

In the author's opinion, the book by Adams and Askenazi [1] is one of the best practical overviews of the interaction of modern solid modeling (SM) software and general finite element software (such as CosmosWorks), and the many pitfalls that will plague many beginners. It points out that almost all FE studies involve assumptions and approximations and the user of such tools should be conscious of them and address them in any analysis or design report. You are encouraged to read it.

1.2 Element Types and Geometries

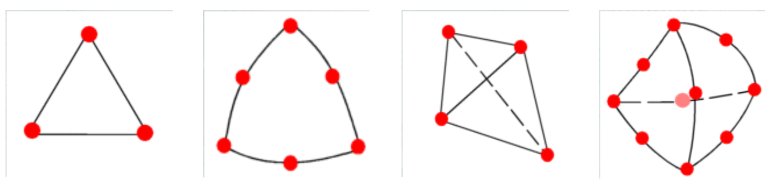
CosmosWorks currently includes solid continuum elements and curved surface shell elements (thin and thick). The shells are triangular with three vertex nodes or three vertex and three mid-edge nodes (Figure 1-1). The solids are tetrahedra with four vertex nodes or four vertex and six mid-edge nodes. They use linear and quadratic interpolation for the solution based on whether they have two or three nodes on an edge. The linear elements are also called simplex elements because their number of vertices is one more than the dimension of the space.

You should always examine the mesh before starting an analysis run. The size of each element indicates a region where the solution is approximated (piecewise) by a polynomial. Most finite element systems, including CosmosWorks, use linear or quadratic complete polynomials in each element. You can tell by inspection which is being used by looking at an element edge. If that line has two nodes the polynomial is linear. If it has three nodes then the polynomial is quadratic.

Table 1-1: Selected CosmosWorks capabilities

*	**Angular acceleration**		**Nonlinear analysis**
*	Angular velocity		Optimization analysis
*	Assembly analysis	*	Orthotropic materials
*	Body loads: gravity & centrifugal		p-adaptive analyses
*	Buckling load factors & modes	*	Plot customization
	Connectors (Bearing, bolt, etc.)	*	Prescribed nonzero restraints
	Contact analysis of with friction	*	Principal stresses
*	Deformation plot	*	Probe and list-by-entity tools
*	Directional pressure and force		Publish eDrawings of results
*	Directional restraints	*	Reaction force result plot

*	Displacement plots		Remote mass
	Drop test analysis		Remote restraints
*	Dynamic section and iso plots	*	Restrain edges, faces & vertices
*	Edit material library	*	Result graphs and listings
	Element result plots		S-N Fatigue curves
	Export to ANSYS, NASTRAN, etc.	*	Special torque, remote, bearing loads
	Factor of safety calculation & plot	*	Strain and displacement analyses
	Fatigue analysis and plots	*	Stress analysis
*	Fixed restraints on faces	*	Stress contour plots
*	Force on edges, faces & vertices		Stress error estimate
*	Frequency analyses	*	Symmetry restraints
	h-adaptive analyses	*	Temperature distribution
*	Heat flux result plots	*	Temperature gradient plots
*	Heat sources		Temperature-dependent properties
*	Heat transfer analysis		Thermal contact resistance
	Import COSMOSFloWorks loads	*	Thermal stress analysis
	Import COSMOSMotion loads		Thermostat controlled heat generation
	Interference or shrink fit	*	Thin parts, sheet metal using shells
	Large displacement analysis	*	Von Mises equivalent stress
*	Multibody part analysis	*	Weldment analysis using beam elements



Shell elements Solid elements

Figure 1-1 CosmosWorks shell and solid element types

Let T(x, y, z) denote an entity to be interpolated and let x, y, and z be the local element edge coordinates. You can relate the number of nodes on an element to the number of polynomial coefficients (a, b, c, …) in the local element spatial approximation, as outlined below:

1. *Linear element type*:

   Straight edge line, or straight bar – 2 nodes, $T(x) = a + b\,x$;
   Straight edged triangle shell, or tetrahedron face – 3 nodes, $T(x, y) = a + b\,x + c\,y$;
   Straight edged, flat faced, tetrahedron – 4 nodes, $T(x, y, z) = a + b\,x + c\,y + d\,z$. Therefore, the solution gradient in this type of element is constant and many elements are required to get good results. In CosmosWorks a mesh of linear elements is called a "Draft Mesh". These are also called *complete linear* elements.

2. Quadratic element type:

   Edge line – 3 nodes, $T(x) = a + b\,x + c\,x^2$;
   Curved triangular shell, or tetrahedron face – 6 nodes,
   $T(x,y) = a + b\,x + c\,y + d\,x^2 + e\,xy + f\,y^2$;
   General curved tetrahedron – 10 nodes,
   $T(x, y, z) = a + b\,x + c\,y + d\,z + e\,x^2 + f\,xy + g\,y^2 + h\,xz + i\,yz + j\,z^2$.

These are called complete quadratic elements and their gradients are complete linear polynomials like those above. Therefore, the solution gradient in these elements varies linearly in space and fewer elements are required for a good solution. CosmosWorks refers to quadratic elements as a "Quality Mesh". The above comments refer to flat shells loaded only in their plane. When shells are loaded normal to their plane a more complicated set of interpolations are used to include bending behavior.

These polynomial interpolations within an element mean that the primary unknown (generalized displacements or temperature) is continuous within the element (C^∞) and across element interfaces (C^0). But, the gradient of the primary unknown is discontinuous across elements, whereas the exact gradient value is continuous in a homogeneous material. The amount of discontinuity between elements is reduced as the element size is reduced. For example, the two colored quadratic surfaces in Figure 1-2 could represent the temperature distribution through the two adjacent (white) elements. Tangent to the common edge the temperature and its tangential slope would both be continuous, but the slope normal to that interface is not continuous. Thus, the temperature gradient is discontinuous across each interface, but continuous everywhere inside any single element. Actually, the element interpolations are also used to determine the shape of each element.

For a quadratic shell, the SolidWorks sends the physical x,y,z coordinates of each the six nodes to CosmosWorks for building the shell geometry, as seen on the left of Figure 1-3. Similarly, the element nodes on all surfaces of a solid are defined by SolidWorks and then the CosmosWorks mesh generator builds the interior tetrahedrons by working in from the bounding surfaces. While the edges seen in Figure 1-3 would be defined exactly in SolidWorks as circles, and they “look like” circles in the finite element mesh they are actually piecewise approximations of a circle. In other words, the quadratic edge of an element is a segment of a parabola passing through the three edge nodes that is used to approximate a segment of a circular arc through the same three nodes. The geometric error is easy to compute and it increases rapidly with the enclosed angle for the edge.

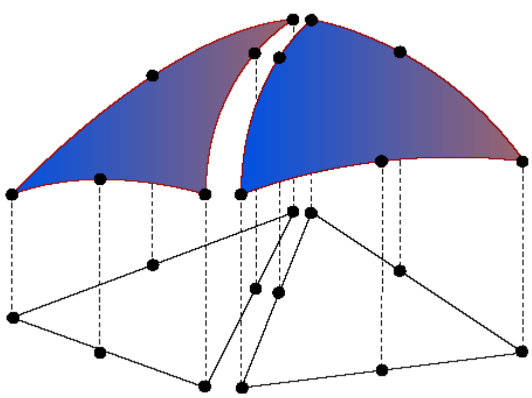


Figure 1-2 Exploded view of two quadratic faces

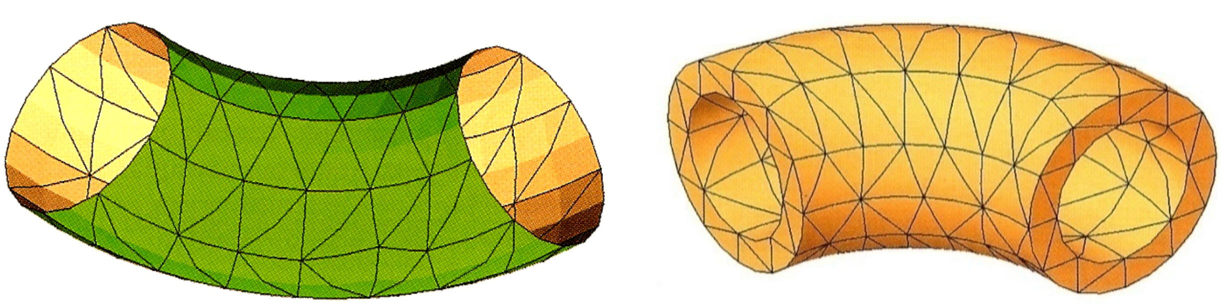


Figure 1-3 Piecewise quadratic surface and solid

1.3 Common modeling errors

As noted above, FE models often have small geometric errors. They can be reduced by mesh refinement and are usually much less important than other sources of error. Probably the most common source of error is in selecting the restraint approximations to be applied to a model. Usually a restraint is applied to a region where surrounding material has been removed and it is necessary to replace the missing material with a restraint. Keep in mind that the removed material must be capable of supplying the assumed restraint or you may introduce a very large error. Sometimes you should move the restraints further away from the part of main interest by including a small region of the supporting material to which you apply the restraints.

Many tutorials and examples assume fixed supports for simplicity. True fixed supports are extremely rare. They require zero movement of the support region. That in turn means that the removed material (represented by the restraint) must be able to develop large reaction forces (and/or moments). A fixed support assumes the material can convey both tension and compression reaction forces locally as needed. Yet some supports can only convey tension while others can only resist compression. Fixed support assumptions tend to under estimate the stresses in the part of interest, but over estimate the resisting stresses (reactions) in the removed material replaced by our simplified engineering assumptions (the restraint type).

Loadings are also not as clear elementary examples suggest. Is a force applied as a point load, a line load, a surface load, etc. That is, where and how a load is applied is usually an assumption. Likewise, the magnitude of a force or other load source may be a reasonable guess or it may be given by established design codes. In thermal studies the source terms, like convection coefficients, vary over a wide range and you may have to run different studies with the high and low values. If two or more regions have different convection fluids then you may have to consider many studies using a Monte Carlo approach to combining the different ranges of data to find the likely worst case.

The nature of the equations being solved is such that the computed reactions are essentially always equal and opposite to the resultant actual applied loads, not necessarily the loads you though that you applied. Reaction data are available in Cosmos and you should always check them.

Common “standardized” materials have mechanical and thermal properties that are relatively well known and are built into the Cosmos materials library. However, even those materials have some range in their values that are not represented in a single number stored in a table.

Many important properties, like the modulus of elasticity, are experimentally measured to only two or three significant figures. Yet a materials table frequently gives average values or values converted from other units to a misleading six or seven significant figures. So usually the computed displacements are only accurate to three or four digits and the stresses to two or three digits.

For all these reasons you must always employ a Factor of Safety (FOS). Remember that a FOS of unity means that failure is eminent; it does not mean that a part or assembly is safe. In practice you should try to justify $1 < FOS < 8$. Several consistent approaches for computing a FOS are given in mechanical design books [9].

2 Concepts of Stress Analysis

2.1 Introduction

Here the concepts of stress analysis will be stated in a finite element context. That means that the primary unknown will be the (generalized) displacements. All other items of interest will mainly depend on the gradient of the displacements and therefore will be less accurate than the displacements. Stress analysis covers several common special cases to be mentioned later. Here only two formulations will be considered initially. They are the solid continuum form and the shell form. Both are offered in CosmosWorks. They differ in that the continuum form utilizes only displacement vectors, while the shell form utilizes displacement vectors *and* infinitesimal rotation vectors.

Stress transfer takes place within, and on, the boundaries of a solid body. The **displacement vector**, ***u***, at any point in the continuum body has the units of meters [m], and its components are the primary unknowns. The components of displacement are usually called *u, v,* and *w* in the x, y, and z-directions, respectively. Therefore, they imply the existence of each other, ***u*** ↔ *(u, v, w).* All the displacement components vary over space. As in the heat transfer case, the gradients of those components are needed but only as an intermediate quantity. The displacement gradients have the units of [m/m], or are considered dimensionless. Unlike the heat transfer case where the gradient was used directly, in stress analysis the multiple components of the displacement gradients are combined into alternate forms called **strains**. The strains have geometrical interpretations that are summarized in Figure 2-1 for 1D and 2D geometry.

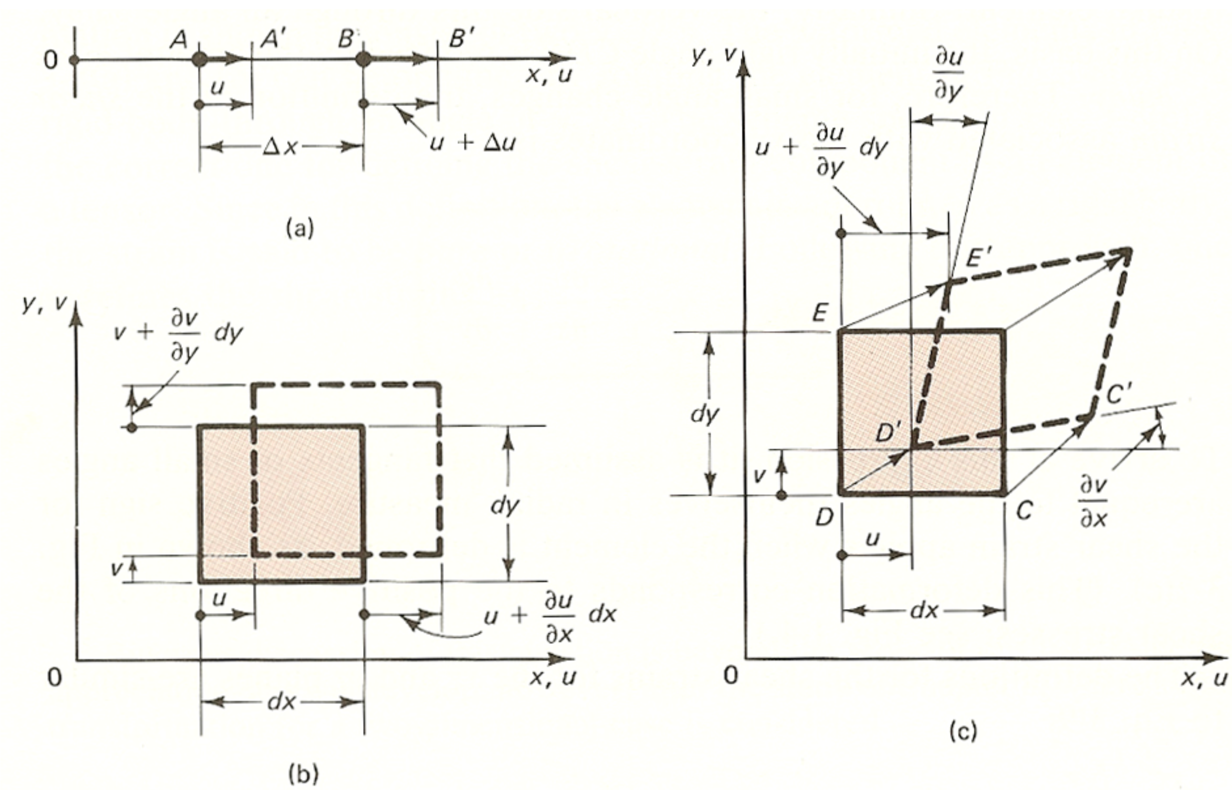
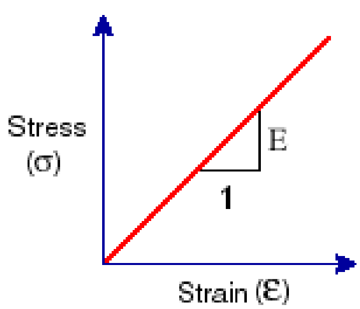


Figure 2-1 Geometry of normal strain (a) 1D, (b) 2D, and (c) 2D shear strain

In 1D the normal strain is just the ratio of the change in length over the original length, $\varepsilon_x = \partial u / \partial x$. In strain2D and 3D both normal strains and shear strains exist. The normal strains involve only the part of the gradient terms parallel to the displacement component. In 2D they are $\varepsilon_x = \partial u / \partial x$ and $\varepsilon_y = \partial v / \partial y$. As seen in Figure 2-1 (b), they would cause a change in volume, but not a change in shape of the rectangular differential element. A shear strain causes a change in shape. The total angle change (from 90 degrees) is used as the engineering definition of the shear strain. The shear strains involve a combination of the component of the gradient that is perpendicular to the displacement component. In 2D the engineering shear strain is $\gamma = (\partial u / \partial y + \partial v / \partial x)$, as seen in Figure 2-1(c). Strain has one component in 1D, three components in 2D, and six components in 3D. They are commonly written as a column vector in finite element analysis, $\varepsilon = (\varepsilon_x \quad \varepsilon_y \quad \gamma)^T$.

The above geometrical data (the strains) will be multiplied by material properties to define a new physical quantity, the **stress**, which is directly proportional to the strains. This is known as **Hooke's Law**: **$\sigma = E\ \varepsilon$,** (see Figure 2-2) where the square **material matrix**, **E**, contains the elastic modulus, and Poisson's ratio of the material, . The stresses are written as a corresponding column vector, **$\sigma = (\sigma_x \quad \sigma_y \quad \tau)^T$**. Unless stated otherwise, the applications illustrated here are assume to be in the linear range of a property.

The 2D and 3D stress components are shown in Figure 2-3. The normal and shear stresses represent the normal force per unit area and the tangential forces per unit area, respectively. They have the units of [N/m^2], or [Pa], but are usually given in [MPa]. The generalizations of the engineering strain definitions are seen in Figure 2-4. The **strain energy** (or **potential energy**) stored in the differential material element is half the scalar product of the stresses and the strains. Error estimates from stress studies are based on strain energy.



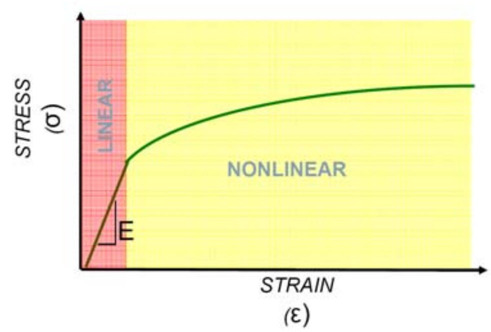
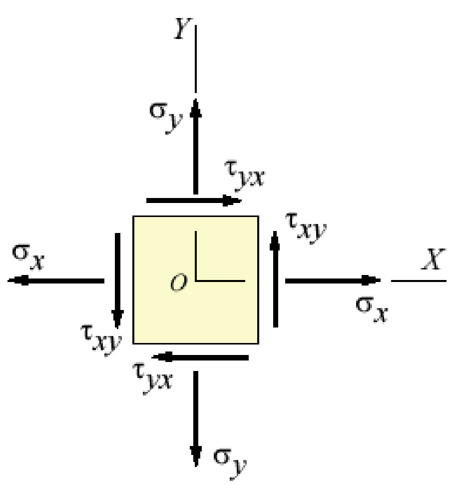


Figure 2-2 Hooke's Law for linear stress-strain



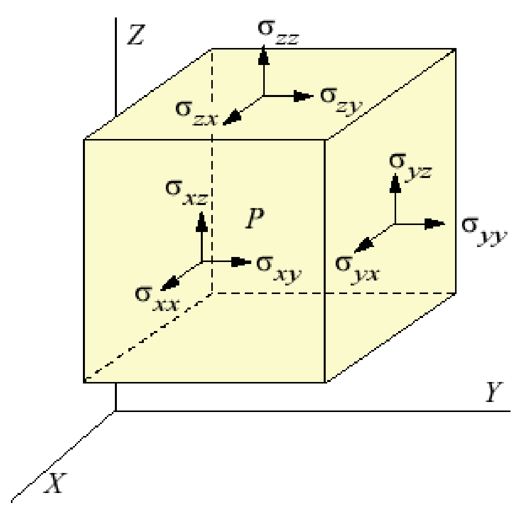


Figure 2-3 Stress components in 2D (left) and 3D

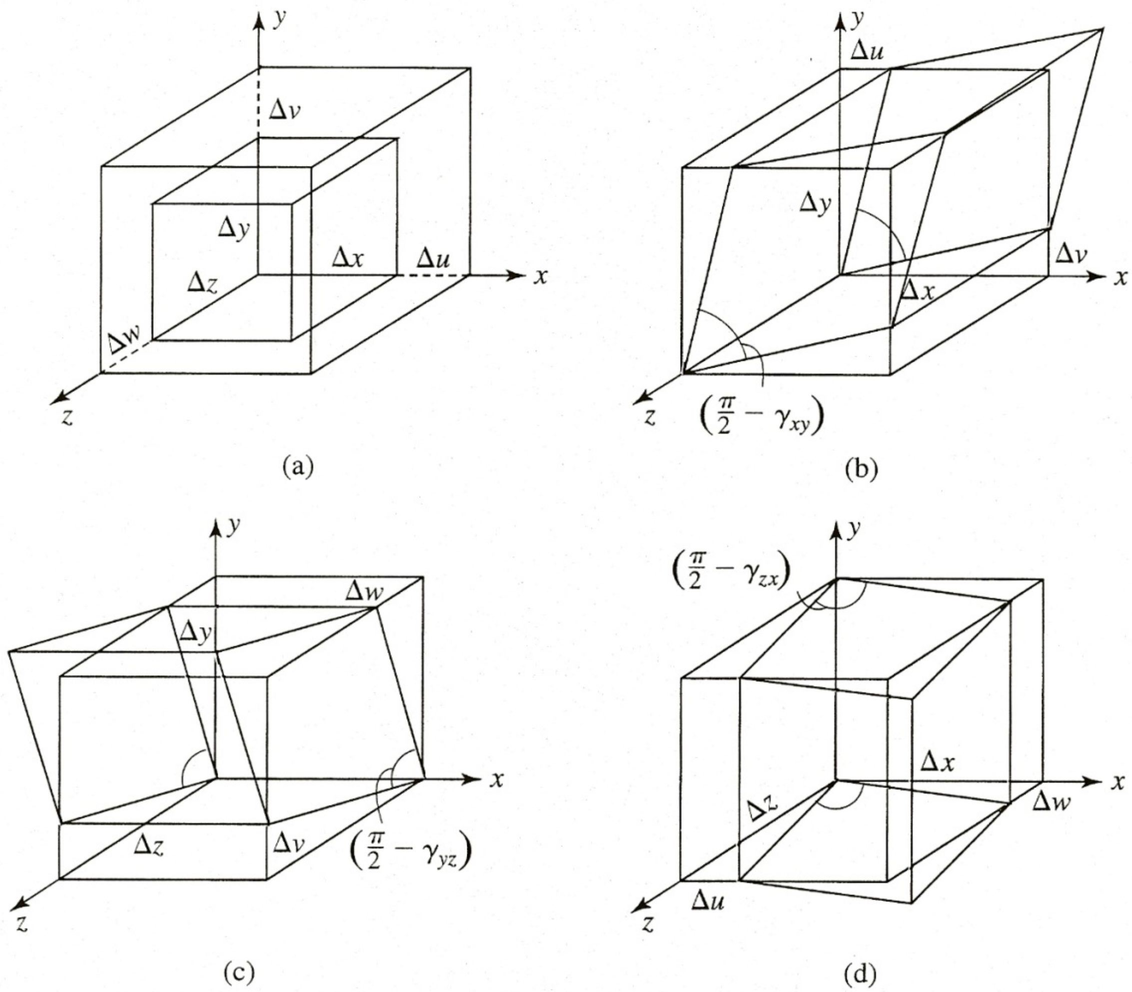


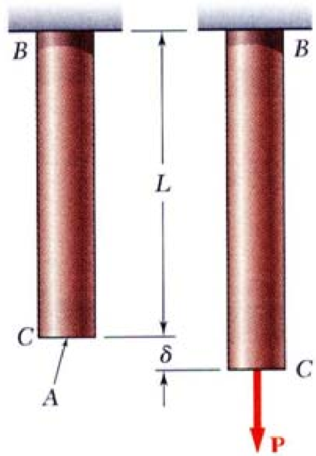
Figure 2-4 Graphical representations of 3D normal strains (a) and shear strains

2.2 Axial bar example

The simplest available stress example is an axial bar, shown in Figure 2-5, restrained at one end and subjected to an axial load, P, at the other end. Let the length and area of the bar be denoted by L, and A, respectively. Its material has an elastic modulus of E. The axial displacement, u (x), varies linearly from zero at the support to a maximum of δ at the load point. That is, u (x) = x δ/ L, so the axial strain is ε_x = ∂u / ∂x = δ / L, which is a constant. Likewise, the axial stress is everywhere constant, σ = E ε = E δ / L which in the case simply reduces to σ = P / A. Like many other more complicated problems, the stress here does not depend on the material properties, but the displacement always does. You should always carefully check both the deflections and stresses when validating a finite element solution.

Since the assumed displacement is linear here any finite element model would give exact deflection and stress results. However, if the load had been the distributed weight of the bar the displacement would have been quadratic in *x* and the stress would be linear in *x*. Then a quadratic element mesh would give exact stresses and displacements everywhere, but a linear element mesh would not.

The elastic bar is often modeled as a linear spring. In introductory mechanics of materials the axial stiffness of a bar is defined as k = E A / L, where the bar has a length of L, an area A, and is constructed of a material elastic modulus of E.



σ = P / A, δ = P L / E A

Figure 2-5 A linearly elastic bar with an axial load

2.3 Structural mechanics

Modern structural analysis relies extensively on the finite element method. It's most popular integral formulation, based on the variational calculus of Euler, is the Principle of Minimum Total Potential Energy. Basically, it states that the displacement field that satisfies the essential displacement boundary conditions and minimizes the total potential energy is the one that corresponds to the state of static equilibrium. This implies that displacements are our primary unknowns. They will be interpolated in space as will their derivatives, the strains. The total potential energy, Π, is the strain energy, U, of the structure minus the mechanical work, W, done by the applied forces. From introductory mechanics, the mechanical work, W, done by a force is the scalar dot product of the force vector, **F**, and the displacement vector, **u**, at its point of application.

The well-known linear elastic spring will be reviewed to illustrate the concept of obtaining equilibrium equations from energy formulations. Consider a linear spring, of stiffness k, that has an applied force, F, at the free (right) end, and is restrained from displacement at the other (left) end. The free end undergoes a displacement of Δ. The work done by the single force is

$$W = \vec{\Delta} \circ \vec{F} = \Delta_x \ F_x = u\,F.$$

The spring stores potential energy due to its deformation (change in length). Here we call that strain energy. That stored energy is given by

$$U = \frac{1}{2} k \, \Delta_x^{\,2}$$

Therefore, the total potential energy for the loaded spring is

$$\Pi = \frac{1}{2} k \, \Delta_x^{\,2} - \Delta_x \ F_x$$

The equation of equilibrium is obtained by minimizing this total potential energy with respect to the unknown displacement, Δ_x. That is,

$$\frac{\partial \Pi}{\partial \Delta_x} = 0 = \frac{2}{2} k \, \Delta_x - F_x$$

This simplifies to the common single scalar equation

$$k\,\Delta_x = F,$$

which is the well-known equilibrium equation for a linear spring. This example was slightly simplified, since we started with the condition that the left end of the spring had no displacement (an essential or Dirichlet boundary condition). Next we will consider a spring where either end can be fixed or free to move. This will require that you both minimize the total potential energy and impose diven displacement restraints.

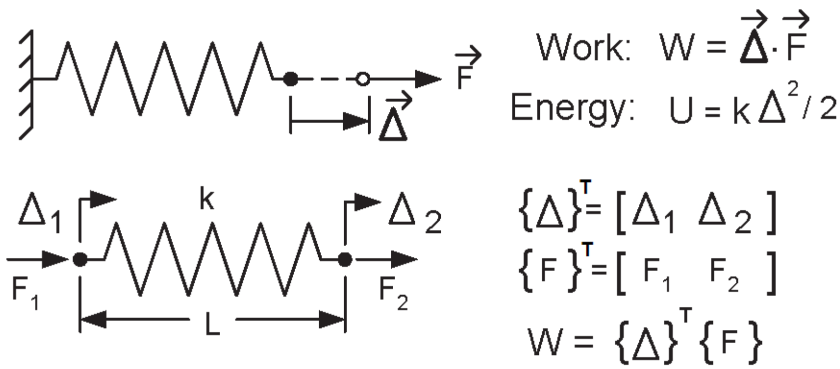


Figure 2-6 The classic and general linear spring element

Now the spring model has two end displacements, Δ_1 and Δ_2, and two associated axial forces, F_1 and F_2. The net deformation of the bar is $\delta = \Delta_2 - \Delta_1$. Denote the total vector of displacement components as

$$\vec{\Delta} = \{\Delta\} = \begin{Bmatrix}\Delta_1\\ \Delta_2\end{Bmatrix}$$

and the associated vector of forces as

$$\vec{F} = \{F\} = \begin{Bmatrix}F_1\\ F_2\end{Bmatrix}$$

Then the mechanical work done on the spring is

$$W = \{\Delta\}^T\{F\} = \Delta_1 F_1 + \Delta_2 F_2$$

Then the spring's strain energy is

$$U = \frac{1}{2}\{\Delta\}^T[k]\{\Delta\} = \frac{1}{2}k\,\delta^2,$$

where the spring "stiffness matrix" is found to be

$$[k] = k\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}.$$

The total potential energy, Π, becomes

$$\Pi = \frac{1}{2}\{\Delta\}^T[k]\{\Delta\} - \{\Delta\}^T\{F\} = \frac{k}{2}\begin{Bmatrix}\Delta_1\\ \Delta_2\end{Bmatrix}^T\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}\begin{Bmatrix}\Delta_1\\ \Delta_2\end{Bmatrix} - \begin{Bmatrix}\Delta_1\\ \Delta_2\end{Bmatrix}^T\begin{Bmatrix}F_1\\ F_2\end{Bmatrix}.$$

Note that each term has the units of energy, i.e. force times length. The matrix equations of equilibrium come from satisfying the displacement restraint and the minimization of the total potential energy with respect to each and every displacement component. The minimization requires that the partial derivative of all the displacements vanish:

$$\frac{\partial \Pi}{\partial\{\Delta\}} = \{0\}\text{, or } \frac{\partial \Pi}{\partial \Delta_j} = 0_j\text{ , } j = 1, 2\text{ .}$$

This represents the first stage system of algebraic equations of equilibrium for the elastic system:

$$k\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} \Delta_1 \\ \Delta_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix}.$$

These two symmetric equations do not yet reflect the presence of any essential boundary condition on the displacements. Therefore, no unique solution exists for the two displacements due to applied forces. Mathematically, this is clear because the square matrix has a zero determinate and can not be inverted. If all of the displacements are known, you can find the applied forces. For example, if you had a rigid body translation of $\Delta_1 = \Delta_2 = C$ where C is an arbitrary constant you clearly get $F_1 = F_2 = 0$. If you stretch the spring by two equal and opposite displacements; $\Delta_1 = -C$, $\Delta_2 = C$ and $F_1 = -2\,k\,C$, and $F_2 = 2\,k\,C$ which is equal and opposite to F_1 .

2.4 Equilibrium of restrained systems

Like the original spring problem, assume the right force, F2, is known, and the left displacement, Δ_1, has a given (restrained) value, say Δ_{given} . Then, the above matrix equation represents two unique equilibrium equations for two unknowns, the displacement Δ_2 and the reaction force F_1. That makes this linear algebraic system look strange because there are unknowns on both sides of the "=". You can correct that by re-arranging the equation system. First, multiply the first column of the stiffness matrix by the known Δ_{given} value and move it to the right side:

$$k\begin{bmatrix} 0 & -1 \\ 0 & 1 \end{bmatrix}\begin{Bmatrix} 0 \\ \Delta_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} - k\begin{Bmatrix} 1 \\ -1 \end{Bmatrix}\Delta_{given}$$

and then move the unknown reaction, F_1, to the left side

$$\begin{bmatrix} -1 & -k \\ 0 & k \end{bmatrix}\begin{Bmatrix} F_1 \\ \Delta_2 \end{Bmatrix} = \begin{Bmatrix} 0 \\ F_2 \end{Bmatrix} - k\begin{Bmatrix} 1 \\ -1 \end{Bmatrix}\Delta_{given}.$$

Now you have the usual form of a linear system of equations where the right side is a known vector and the left side is the product of a known square matrix times a vector of unknowns. Since both the minimization and the displacement restraints have been combined you now have a unique set of equations for the unknown displacements and the unknown restraint reactions. Inverting the 2 by 2 matrix gives the solution:

$$\begin{Bmatrix} F_1 \\ \Delta_2 \end{Bmatrix} = \frac{-1}{k}\begin{bmatrix} k & k \\ 0 & -1 \end{bmatrix}\left(\begin{Bmatrix} 0 \\ F_2 \end{Bmatrix} - k\begin{Bmatrix} 1 \\ -1 \end{Bmatrix}\Delta_{given}\right)$$

$$\begin{Bmatrix} F_1 \\ \Delta_2 \end{Bmatrix} = \begin{Bmatrix} -F_2 \\ F_2/k + \Delta_{given} \end{Bmatrix}$$

so that $F_1 = -F_2$ always, as expected, and if $\Delta_{given} = 0$, as originally stated, then $\Delta_2 = F_2/k$.

2.5 General equilibrium matrix partitions

The above small example gives insight to the most general form of the algebraic system that results from only minimizing the total potential energy: a singular matrix system with more unknowns than equations. That is because there is not a unique equilibrium solution until you also apply the essential (Dirichlet) boundary conditions on the displacements. The algebraic system can be written in a general partitioned matrix form that more clearly defines what must be done to reduce the system to a solvable form by utilizing essential boundary conditions.

For an elastic system of any size, the full matrix equations obtained by minimizing the energy can always be rearranged into the following partitioned matrix form:

$$\begin{bmatrix} K_{uu} & K_{ug} \\ K_{gu} & K_{gg} \end{bmatrix} \begin{Bmatrix} \Delta_u \\ \Delta_g \end{Bmatrix} = \begin{Bmatrix} F_g \\ F_u \end{Bmatrix}$$

where Δ_u represents the unknown nodal displacements, and Δ_g represents the given essential boundary values (restraints) of the other displacements. The stiffness sub-matrices $\mathbf{K_{uu}}$ and $\mathbf{K_{gg}}$ are square, whereas $\mathbf{K_{ug}}$ and $\mathbf{K_{gu}}$ are rectangular. In a finite element formulation all of the coefficients in the $\mathbf{K}$ matrices are known. The resultant applied nodal loads are in sub-vector $\mathbf{F_g}$ and the $\mathbf{F_u}$ terms represent the unknown generalized reactions forces associated with essential boundary conditions. This means that after the enforcement of the essential boundary conditions the actual remaining unknowns are Δ_u and $\mathbf{F_u}$. Only then does the net number of unknowns correspond to the number of equations. But, they must be re-arranged before all the remaining unknowns can be computed.

Here, for simplicity, it has been assumed that the equations have been numbered in a manner that places rows associated with the given displacements (essential boundary conditions) at the end of the system equations. The above matrix relations can be rewritten as two sets of matrix identities:

$$K_{uu}\Delta_u + K_{ug}\Delta_g = F_g$$

$$K_{gu}\Delta_u + K_{gg}\Delta_g = F_u.$$

The first identity can be solved for the unknown displacements, Δ_u, by putting it in the standard linear equation form by moving the known product $K_{ug}\Delta_g$ to the right side. Most books on numerical analysis assume that you have reduced the system to this smaller, nonsingular form before trying to solve the system. Inverting the smaller non-singular square matrix yields the unique equilibrium displacement field:

$$\Delta_u = K_{uu}^{-1}(F_g - K_{ug}\Delta_g).$$

The remaining reaction forces can then be recovered, if desired, from the second matrix identity:

$$F_u = K_{gu}\Delta_u + K_{gg}\Delta_g.$$

In most applications these reaction data have physical meanings that are important in their own right, or useful in validating the solution. However, this part of the calculation is optional.

2.6 Component Failure

Structural components can be determined to fail by various modes determined by buckling, deflection, natural frequency, strain, or stress. Strain or stress failure criteria are different depending on whether they are considered as brittle or ductile materials. The difference between the two material behaviors is determined by their response to a uniaxial stress-strain test as illustrated in Figure 2-7. You need to know what class of material is being used. CosmosWorks, and most finite element systems, default to assuming a ductile material and display the distortional energy failure theory which is usually called the Von Mises stress, or effective stress, even though it is actually a scalar. A brittle material requires the use of a higher factor of safety.

As illustrated in Figure 2-8, the solid elements have three translational degrees of freedom (dof) as nodal unknowns, for a total of 12 or 30 dof. The shell elements have three translational degrees of freedom as well as three rotational degrees of freedom, for a total of 18 or 36 dof. The difference in dof types means that moments or couples can only be applied directly to shell models. Solid elements require that couples be indirectly applied by specifying a pair of equivalent pressure distributions, or an equivalent pair of equal and opposite forces at two nodes on the body.

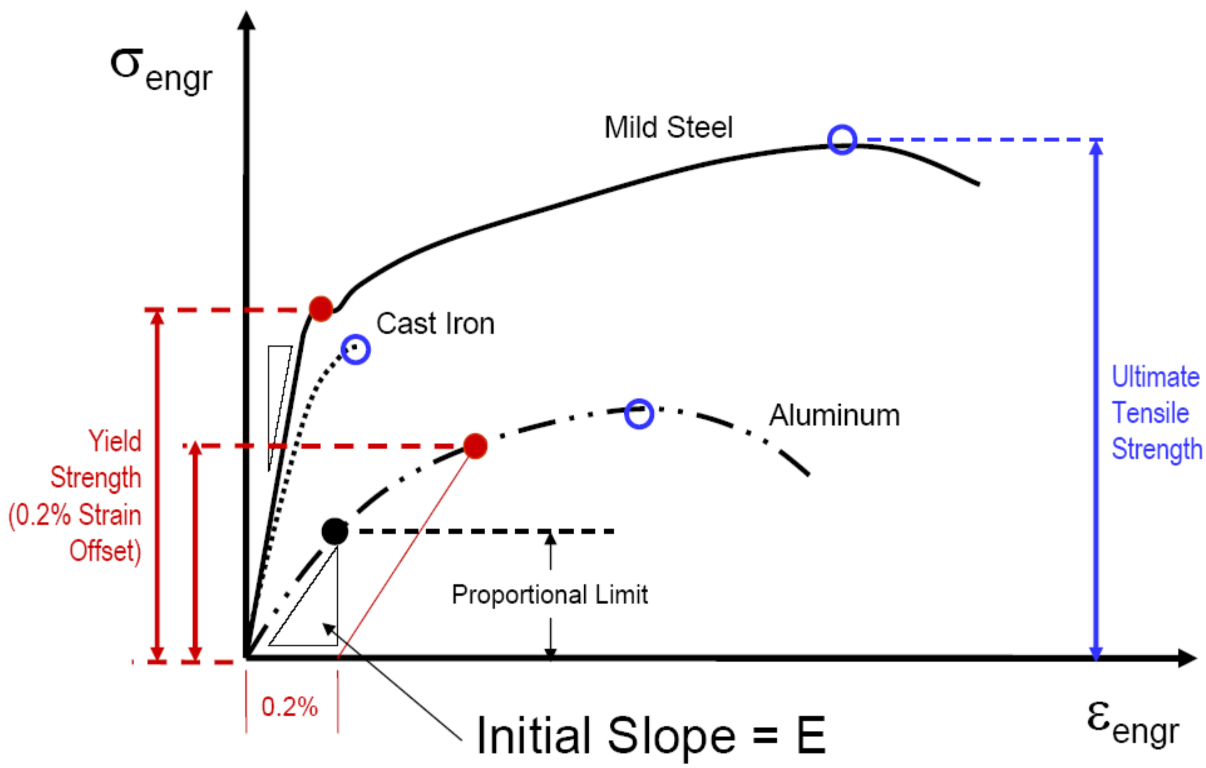
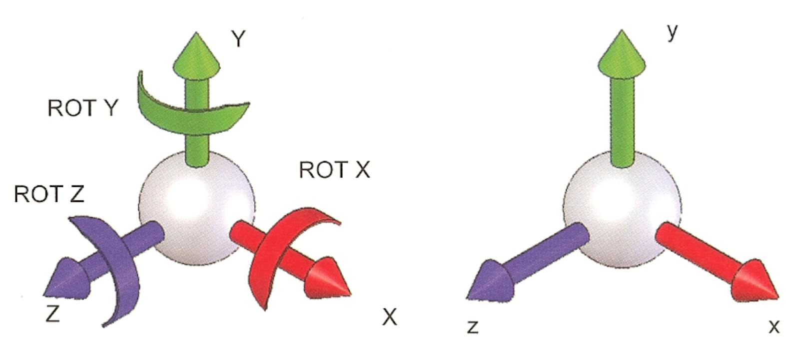


Figure 2-7 Axial stress-strain experimental results



Shell node   Solid node

Figure 2-8 Nodal degrees of freedom for frames and shells; solids and trusses

2.7 Element type selection

Even with today's advances in computing power you seem never to have enough computational resources to solve all the problems that present themselves. Frequently solid elements are not the best choice for computational efficiency. The analysts should learn when other element types can be valid or when they can be utilized to validate a study carried out with a different element type. Cosmos offers a small element library that includes bars, trusses, beams, frames, thin plates and shells, thick plates and shells, and solid elements.

The shells and solid elements are considered to be continuum elements. The plate elements are a special case of shells with no initial curvature. Let h denote the typical thickness of a component while its typical length is denoted by l. The thickness to length ratio, h/l, gives some guidance as to when a particular element type is valid for an analysis. When h/l is large shear deformation is at its maximum importance and you should be using solid elements. Conversely, when h/l is very small transverse shear deformation is not important and thin shell elements are probably the most cost effective element choice. In the intermediate range of h/l the thick shell elements will be most cost effective. The thick shell elements are extensions of thin shell elements that contain additional terms to account for transverse shear deformations.

The overlapping h/l ranges for the three continuum element types are suggested in Figure 2-9. The thickness of the lines suggests those regions where a particular element type is generally considered to be the preferred element of choice. The overlapping ranges suggest where one type of element calculation can be used to validate a calculated result obtained with a different element type. Validation calculations include the different approaches to boundary conditions and loads required by different element formulations. They also can indirectly check that a user actually understands how to utilize a finite element code.

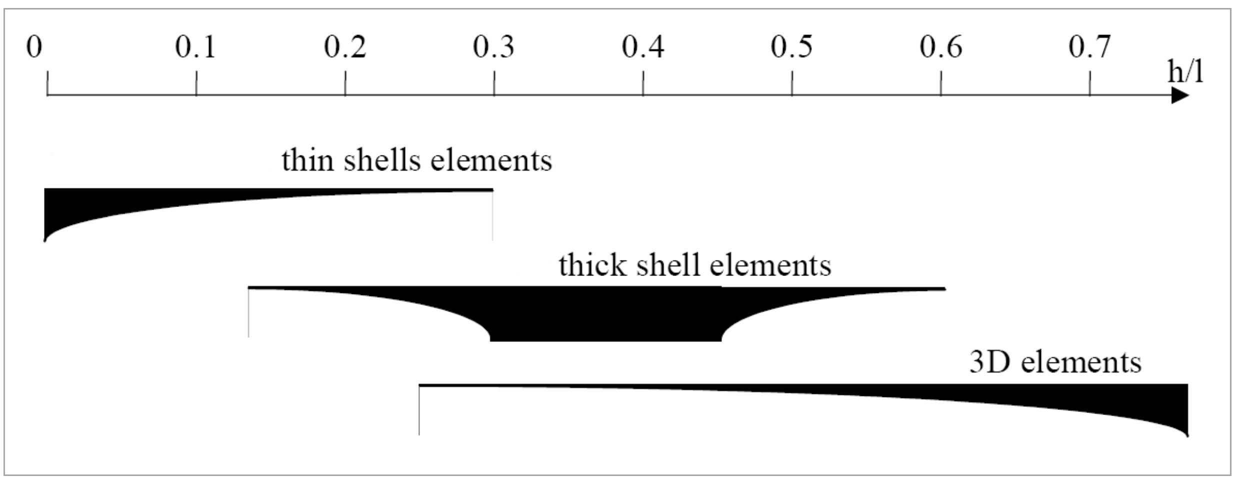
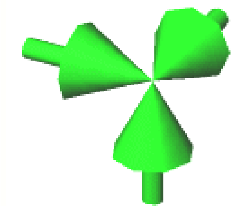
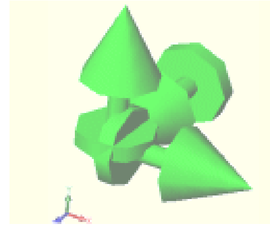


Figure 2-9 Overlapping valid ranges of element types

2.8 CosmosWorks restraint and load symbols

The symbols used in CosmosWorks to represent a single translational and rotational dof at a node are shown green in Figure 2-10. The symbols for the corresponding forces and moment loadings are shown pink in that figure. Since finite element solutions are based on work-energy relations, the above word "corresponding" means that their dot product represents the mechanical work done at the point. When a model can involve either translations or rotations as dof they are often referred to as generalized displacements. The CosmosWorks nodal symbols for the unknown generalized displacement dof's for the solid nodes (top) and shell nodes are seen in



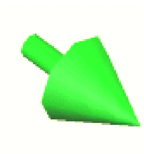


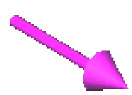
Node of solid or truss element:
All three displacements are zero.

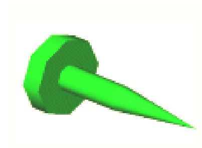
Node of frame or shell element:
All three displacements and all three rotations are zero.

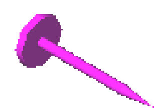
Figure 2-11. You almost always must supply enough restraints to prevent any model from undergoing a rigid body translation or rigid body rotation.

For simplicity many finite element examples incorrectly apply complete restraints at a face, edge or node. That is, they enforce an **Immovable** condition for solids or a **Fixed** condition for shells. Actually determining the type of restraint, as well as where the part is restrained is often the most difficult part of an analysis. You frequently encounter the common conditions of symmetry or anti-symmetry restraints. You should under understand symmetry plane restraints for solids and shells.







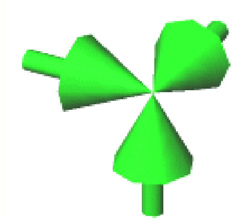


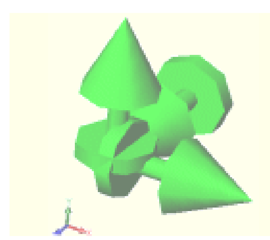
Displacement Force Rotation Couple

Figure 2-10 Single component symbols for restraints and loads

2.9 Symmetry dof on a plane

A plane of symmetry is flat and has mirror image geometry, material properties, loading, and restraints. Symmetry restraints are very common for solids and for shells. Figure 2-12 shows that for both solids and shells, the displacement perpendicular to the symmetry plane is zero. Shells have the additional condition that the in-plane component of its rotation vector is zero. Of course, the flat symmetry plane conditions can be stated in a different way. For a solid element translational displacements parallel to the symmetry plane are allowed. For a shell element rotation is allowed about an axis perpendicular to the symmetry plane and its translational displacements parallel to the symmetry plane are also allowed.





Node of solid or truss element:
All three displacements are zero.

Node of frame or shell element:
All three displacements and all three rotations are zero.

Figure 2-11 Fixed restraint symbols for solids (top) and shell nodes

Node of a solid or truss element:
Displacement normal to the symmetry plane is zero.

Node of a frame or shell element:
Displacement normal to the symmetry plane and two rotations parallel to it are zero.

Figure 2-12 Symmetry requires zero normal displacement, and zero in-plane rotation

2.10 Available material inputs for stress studies

Most applications involve the use of isotropic (direction independent) materials. The available mechanical properties for them in Cosmos are listed in Table 2-1. It is becoming more common to have designs utilizing anisotropic (direction dependent) materials. The most common special case of anisotropic materials is the orthotropic material. Any anisotropic material has its properties input relative to the principal directions of the material. That means you must construct the principal directions reference plane or coordinate system before entering orthotropic data. Mechanical orthotropic properties are subject to some theoretical relations ships that physically possible materials must satisfy (such as positive strain energy). Experimental material properties data may require adjustment before being accepted by Cosmos.

Table 2-1 Isotropic mechanical properties

Symbol	Label	Item
E	EX	Elastic modulus (Young's modulus)
ν	NUXY	Poisson's ratio
G	GXY	Shear modulus
ρ	DENS	Mass density
σ_t	SIGXT	Tensile strength (Ultimate stress)
σ_c	SIGXC	Compression stress limit
σ_y	SIGYLD	Yield stress (Yield strength)
α	ALPX	Coefficient of thermal expansion

Table 2-2 Orthotropic mechanical properties in principal material direction

Symbol	Label	Item
E_x	EX	Elastic modulus in material X direction
E_y	EY	Elastic modulus in material Y direction
E_z	EZ	Elastic modulus in material Z-direction
ν_{xy}	NUXY	Poisson's ratio in material XY directions
ν_{yz}	NUYZ	Poisson's ratio in material YZ directions
ν_{xz}	NUXZ	Poisson's ratio in material XZ directions
G_{xy}	GXY	Shear modulus in material XY directions
G_{yz}	GYZ	Shear modulus in material YZ directions
G_{xz}	GXZ	Shear modulus in material XZ directions
ρ	DENS	Mass density
σ_t	SIGXT	Tensile strength (Ultimate stress)
σ_c	SIGXC	Compression stress limit
σ_y	SIGYLD	Yield stress (Yield strength)
α_x	ALPX	Thermal expansion coefficient in material X

α_y	ALPY	Thermal expansion coefficient in material Y
α_z	ALPZ	Thermal expansion coefficient in material Z

NUXY, NUYZ, and NUXZ are not independent

2.11 Stress study outputs

A successful run of a study will create a large amount of additional output results that can be displayed and/or listed in the post-processing phase. Displacements are the primary unknown in a Cosmos stress study. The available displacement vector components are cited in Table 2-3 and Table 2-4, along with the reactions they create if the displacement is used as a restraint. The displacements can be plotted as vector displays, or contour values.

Table 2-3 Output results for solids, shells, and trusses

Symbol	Label	Item	Symbol	Label	Item
U_x	UX	Displacement (X direction)	R_x	RFX	Reaction force (X direction)
U_y	UY	Displacement (Y direction)	R_y	RFY:	Reaction force (Y direction)
U_z	UZ	Displacement (Z direction)	R_z	RFZ	Reaction force (Z direction)
U_r	URES:	Resultant displacement magnitude	R_r	RFRES	Resultant reaction force magnitude

Table 2-4 Additional primary results for solids, shells, and trusses

Symbol	Label	Item	Symbol	Label	Item
θ_x	RX	Rotation (X direction)	M_x	RMX:	Reaction moment (X direction)
θ_y	RY	Rotation (Y direction)	M_y	RMY	Reaction moment (Y direction)
θ_z	RZ	Rotation (Z direction)	M_z	RMZ:	Reaction moment (Z direction)
			M_r	MRESR	Resultant reaction moment magnitude

The strains and stresses are computed from the displacements. The stress components available at an element centroid or averaged at a node are given in Table 2-5. The six components listed on the left in that table give the general stress at a point (i.e., a node or an element centroid). Those six values are illustrated on the left of Figure 2-13. They can be used to compute the scalar von Mises failure criterion. They can also be used to solve an eigenvalue problem for the principal normal stresses and their directions, which are shown on the right of Figure 2-13. The maximum shear stress occurs on a plane whose normal is 45 degrees from the direction of P1. The principal normal stresses can also be used to compute the scalar von Mises failure criterion.

Table 2-5: Nodal and element stress results

Symbol	Label	Item	Symbol	Label	Item
σ_x	SX	Normal stress parallel to x-axis	σ_1	P1	1st principal normal stress
σ_y	SY	Normal stress parallel to y-axis	σ_2	P2	2nd principal normal stress
σ_z	SZ	Normal stress parallel to z-axis	σ_3	P3	3rd principal normal stress
τ_{xy}	TXY	Shear in Y direction on plane normal to x-axis	τ_I	INT	Stress intensity (P1-P3), twice the maximum shear stress
τ_{xz}	TXZ	Shear in Z direction on plane normal to x-axis			
τ_{yz}	TYZ	Shear in Z direction on plane normal to z-axis	σ_{vm}	VON	von Mises stress (distortional energy failure criterion)

The von Mises effective stress is compared to the material yield stress for ductile materials. Failure is predicted to occur (based on the distortional energy stored in the material) when the von Mises value reaches the yield stress. The maximum shear stress is predicted to cause failure when it reaches half the yield stress. Cosmos uses the shear stress intensity which is also compared to the yield stress to determine failure (because it is twice the maximum shear stress). The first four values on the right side of Table 2-5 are often represented graphically in mechanics as a 3D Mohr's circle (seen in Figure 2-14).

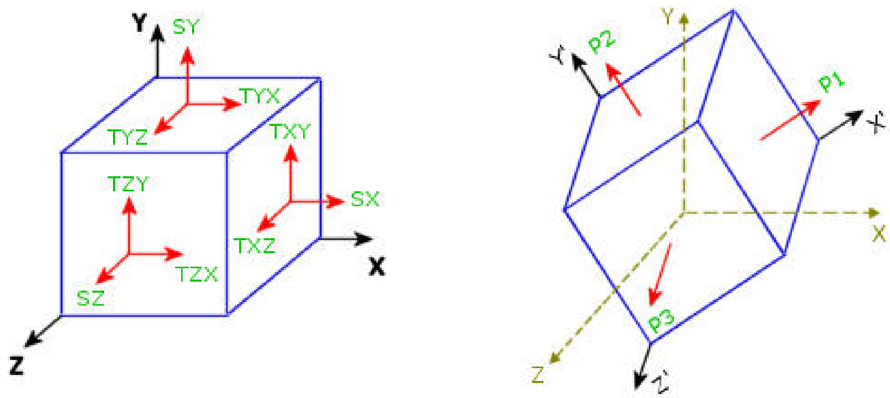


Figure 2-13 The stress tensor and its principal normal values

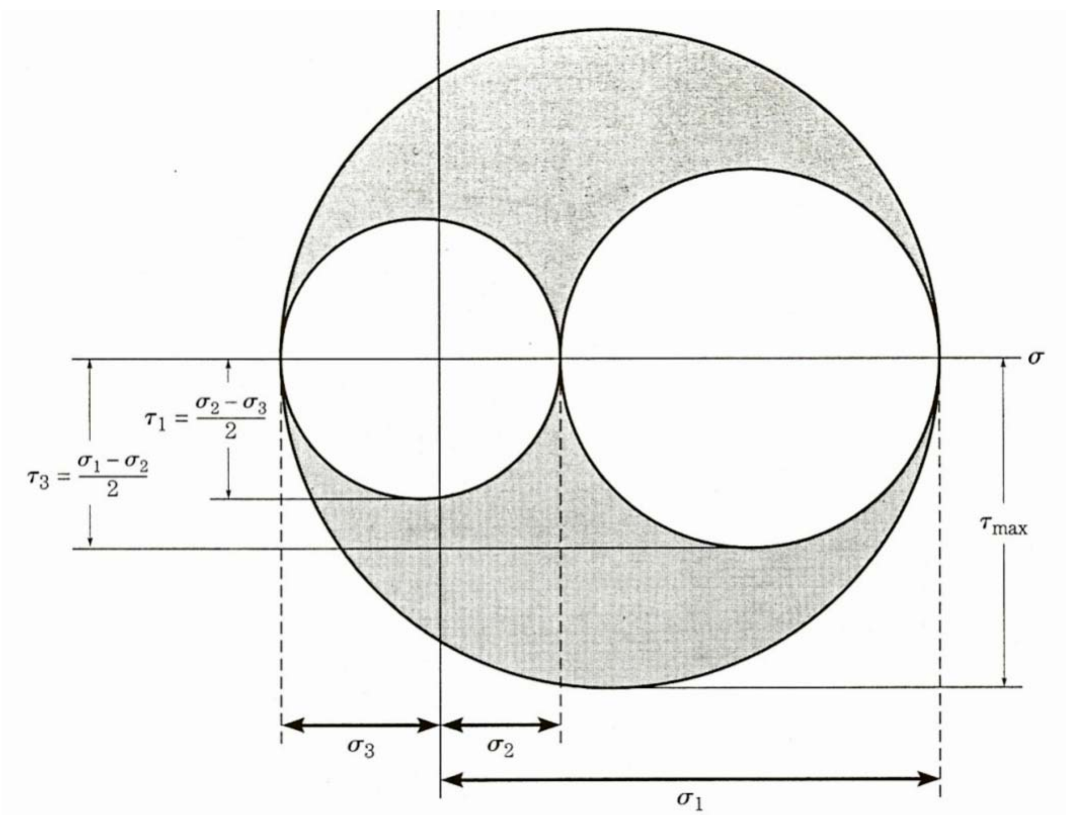


Figure 2-14 The three-dimensional Mohr's circle of stress

If desired, you can plot all three principal components at once. The three principal normal stresses at a node or element center can be represented by an ellipsoid. The three radii of the ellipsoid represent the magnitudes of the three principal normal stress components, P1, P2, and P3. The sign of the stresses (tension or compression) are represented by arrows. The color code of the surface is based on the von Mises value at the point, a scalar quantity. If one of the principal stresses is zero, the ellipsoid becomes a planar ellipse. If the three principal stresses have the same magnitude, the ellipsoid becomes a sphere. In the case of simple uniaxial tensile stress, the ellipsoid becomes a line.

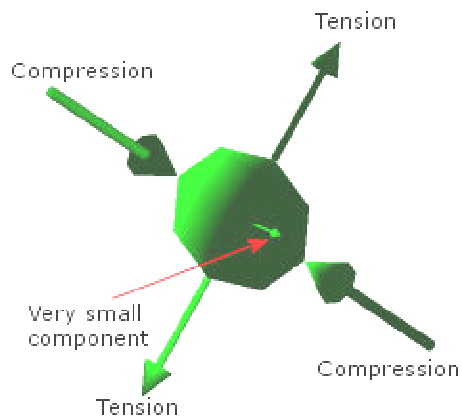


Figure 2-15 A principal stress ellipsoid colored by von Mises value

The available nodal output results in Table 2-5 are obtained by averaging the element values that surround the node. You can also view them as constant values at the element centroids. That can give you insight to the smoothness of the approximation. For brittle materials you can also be interested in the element strain results. They are listed in Table 2-6. Table 2-7 shows that you can also view the element error estimate, ERR which is used to direct adaptive solutions, and the contact pressure from an iterative contact analysis.

Table 2-6 Element centroidal strain component results

Sym	Label	Item	Sym	Label	Item
ε_x	EPSX	Normal strain parallel to x-axis	ε_1	E1	Normal principal strain (1st principal direction)
ε_y	EPSY	Normal strain parallel to y-axis	ε_2	E2	Normal principal strain (2nd principal direction)
ε_z	EPSZ	Normal strain parallel to z-axis	ε_3	E3	Normal principal strain (3rd principal direction)
γ_{xy}	GMXY	Shear strain in Y direction on plane normal to x-axis	ε_r	ESTRN	Equivalent strain
γ_{xz}	GMXZ	Shear strain in Z direction on plane normal to x-axis	SED	SEDENS	Strain energy density (per unit volume)
γ_{yz}	GMYZ	Shear strain in Z direction on plane normal to y-axis	SE	ENERGY	Total strain energy

Table 2-7 Additional element centroid stress related results

Label	Item
ERR	Element error measured in the strain energy norm
CP	Contract pressure developed on a contact surface

3 Mesh control in CosmosWorks

3.1 Introduction

You must plan ahead when building a solid model so that it can be used for realistic finite element load and/or restraint analysis cases. You often do that in the solid modeling phase by using lines or arcs to partition lines, curves, or surfaces. This is called using a "split line" in SolidWorks and CosmosWorks. There is also a "split part" feature that is similar except that it cuts a part into multiple sub-parts. You sometimes need to do that for symmetry or anti-symmetry finite element analysis so that we can analyze the part more efficiently. Here, the concept of splitting surfaces for mesh control will be illustrated via stress analysis.

3.2 Initial analysis

The split line concepts for mesh control will be illustrated via the first tutorial "Static Analysis of a Part", using the CosmosWorks example file Tutor1.sldprt which is shown in Figure 3-1. (Remember to save it with a new name by putting your initials at the front.) Consider that tutorial to be a preliminary analysis. You should recall that there was bending of the base plate near the loaded vertical post. However, the original solid mesh had only one element through the thickness in that region and would therefore underestimate the bending stresses there. You need to control the mesh there to form 5 to 6 layers to accurately capture the change in bending stress through the thickness.

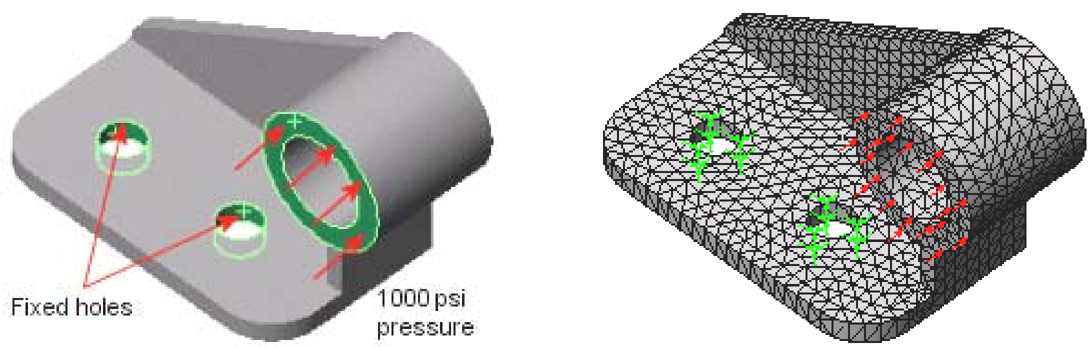


Figure 3-1 Original part, restraints, load, and mesh

The original deformed shape, in Figure 3-2, is shown relative to the undeformed part (in gray). You see bending at the end of the base and deflection of the part bottom back edge in the direction of the (unseen) supporting object below it. From the original effective stress plot in Figure 3-3 and

Figure 3-4 you can see that large regions, within the red contours, have exceeded the material yield stress. Actually, the maximum value is over 133,000 psi, or about 1.5 times the yield stress, on the top and bottom of the base. Clearly, this part must have its material or dimensions changed and/or new support options must be utilized.

3.3 Structural restraint options in CosmosWorks

Solids and shells must be restrained and loaded in different ways since shells also have rotational degrees of freedom and solids do not. Table 3-1 lists the current restraint options for solids within CosmosWorks. Note that they only involve components of the displacement vectors. The loading options for solids are given in Table 3-2. They involve only forces, pressures, temperature differences, and accelerations. No pure couples (moments) can be specified at a node. Of course, loadings statically equivalent to applied couples can be specified by spatially varying pressures.

Once the restraints, loads, and materials have been specified the mesh must be generated, as discussed in the next section. For complicated parts it is sometimes wise to generate the mesh first. The mesh generation phase can fail for various reasons (mainly bad solid models or insufficient memory resources). If it does fail the time spent on loads and restraints may be wasted. When a mesh exists too, you proceed on to running the analysis. Then you must conduct and document the post-processing the analysis results. Those phases will be introduced in the following general introduction to Cosmos for stress studies.

Table 3-1 Restraints for solid stress analysis

Restraint Type	Solid Element Definition
Fixed or Immovable	All three translations are zero on face, edge, or vertex.
Hinge	On a cylindrical face, only the circumferential displacement is allowed.
On cylindrical face	The cylindrical coordinate displacements normal to and/or on the cylindrical surface are given.
On flat face	Displacements normal to and/or tangent to a flat face are given.
On Spherical face	The spherical coordinate displacements normal to and/or on the spherical surface are given.
Roller/Sliding or Symmetry	Two displacements tangent to a flat face are allowed.
Use reference geometry	A face, edge, or vertex can translate a specified amount relative to a reference plane and axis.

Table 3-2 Load conditions for solid stress analysis

Load Type	Solid Element Definition
Apply force	The total force on a face, edge, or vertex is given relative to a single edge or axis direction.
Apply normal force	The total force normal to a face, at its centroid, is specified and converted to an equivalent pressure.
Apply torque	The total torque on a face is specified with respect to an axis and converted to an equivalent pressure.
Bearing Load	On a cylindrical surface give the total force in a Cartesian X or Y direction to convert to a sine distribution pressure.
Centrifugal	The angular acceleration and angular velocity are given about an axis, edge, or cylindrical surface.
Connectors	See CosmosWorks help files.
Gravity	The gravitation acceleration value is given and oriented by an axis, edge, or a direction in or normal to a selected plane.
Remote load	See CosmosWorks help files.
Temperature	Not recommended. Transfer from thermal analysis.

Begin by reconsidering the restraints utilized. In many problems the restraints are unclear or questionable and you need to consider other restraint cases. You previously restrained all the three translations on the two small cylindrical surfaces. That makes those surfaces perfectly rigid. That would be almost impossible to build. It is likely that the holes were intended to be bolt holes. Then the applied backward (-*z*) pressure load would probably require the development of tension reaction forces along the back (-*z*) half of the bolt cylinder. That would not happen because an air gap would open up. Also, a bolt usually applies a restraint to the surface under the bolt head (in addition to a bolt bearing load on its cylindrical part), and that is not in the original choice of restraints. Furthermore, the bracket seems to be attached to some rigid object under it and therefore you would expect the back edge of the bracket to be somehow supported by that object when that region of the part deforms as seen in Figure 3-2. The original effective stresses are in Figure 3-3 and Figure 3-4.

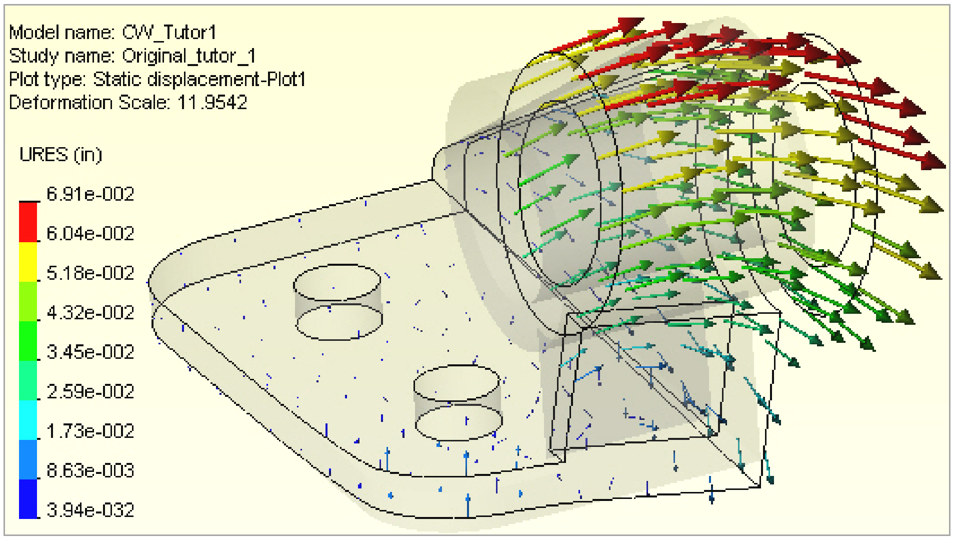


Figure 3-2 Original part deflections

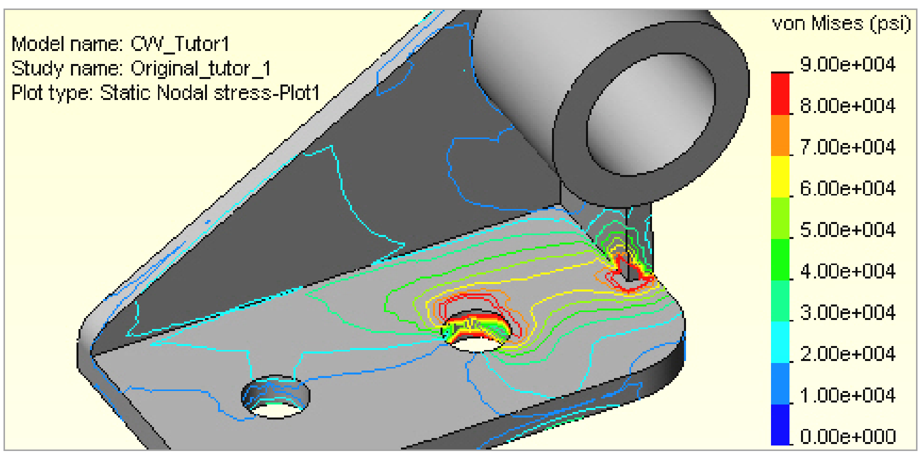


Figure 3-3 Original top surface effectives stresses

As a new possible restraint set, assume that the bolt heads are tight and act on a small surface ring around each hold. That could provide rigid body translation restraints in 3 directions. Each bolt would prevent three translations and the pair of them combines to also prevent three rigid body rotations. Thus they combine to prevent all six possible rigid body motions (RBM). If the bolts were not tight then only a normal displacement (along the bolt axis) would be restrained on the base top surface (and two RBM would remain).

Bearing loads on the bolt shaft can be found by an iterative process in CosmosWorks, but for early studies you can assume a small cylindrical contact at the most positive *z* (front) location. The previously computed bending of the part is also assumed to cause contact with the supporting object below, and thus a *y*-translational restraint, along the bottom back edge of the part. To accomplish those types of restraint controls you need to "split" the surfaces and transition the mesh in those regions to get better results. Thus, you need to form two smaller surface rings for the bolt heads, and split the cylinders into smaller bearing areas.

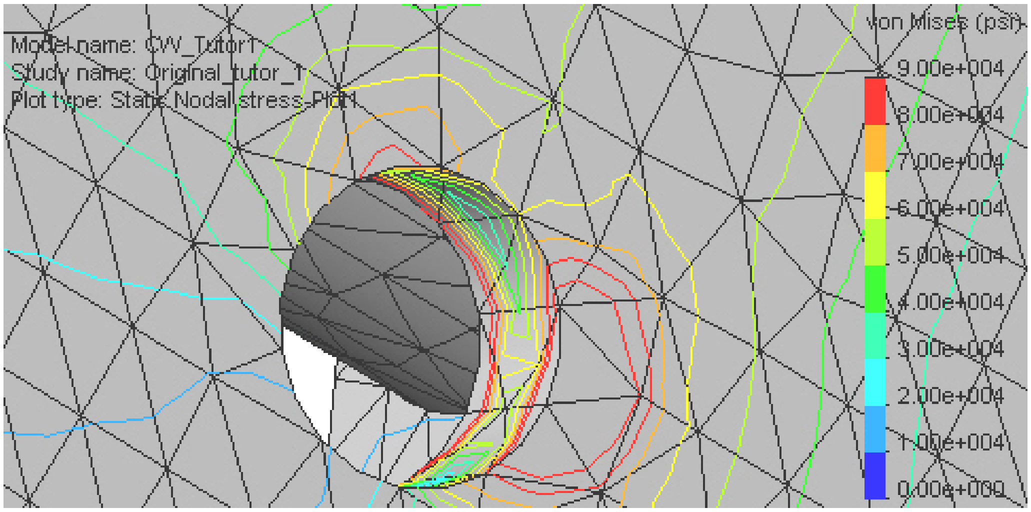
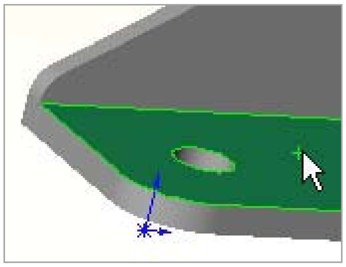


Figure 3-4 Original bottom surface effective stresses

3.4 Splitting a Surface or Curve

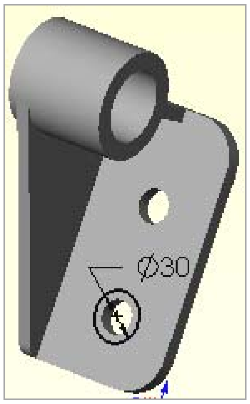
To avoid changing the master tutorial file, open the part and then "save as" a new file name on the desktop. To introduce the required split lines:

1. Select the top surface of the base by moving the cursor over it until its boundary turns green and **Insert Sketch**.

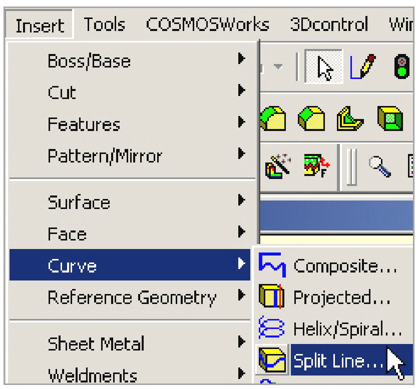


2. Pick the **Sketch** icon and pick the **Circle** option to form a bolt head washer area.

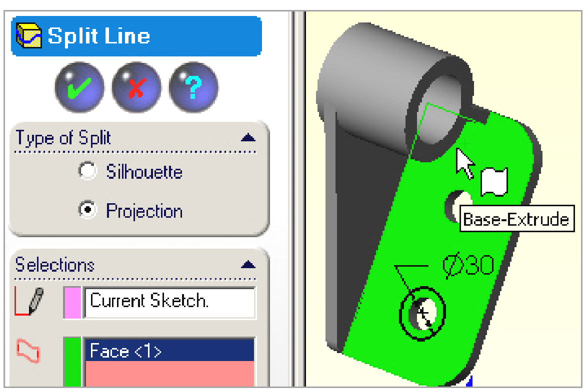
3. Place the cursor on the hole edge to "wake up" the center point. Draw a larger circle on the surface. Set its diameter to 30 mm and click **OK**. That does not change the surface; it just adds a circle to it.



4. To split the surface go to the top menu bar and select **Insert → Curve → Split Line**. The **Type** will be a projection.



5. Next, pick the surface(s) to be cut by this curve. Here it will be the top of the base plate again so select it and click **OK**. Repeat the above two processes for the second hole.



Now you will see that moving the cursor around shows a new circle and a new ring of surface area that could be used to enforce restraints or loads. The new surface areas are shown in Figure 3-5. Later you will use these newly created ring areas as a bolt head (washer) restraint region.

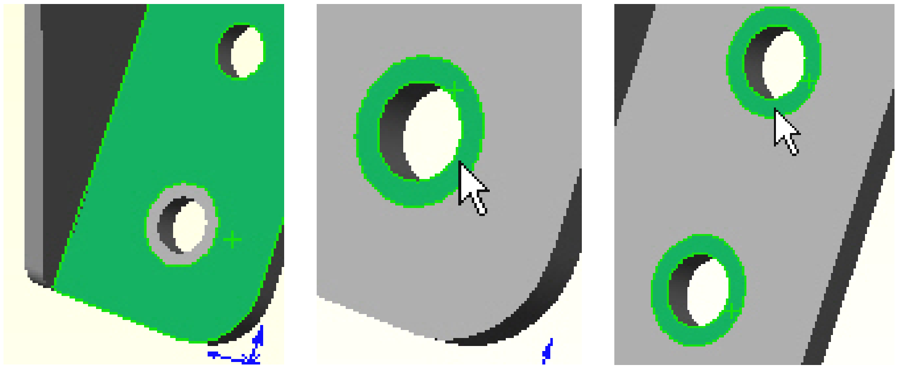
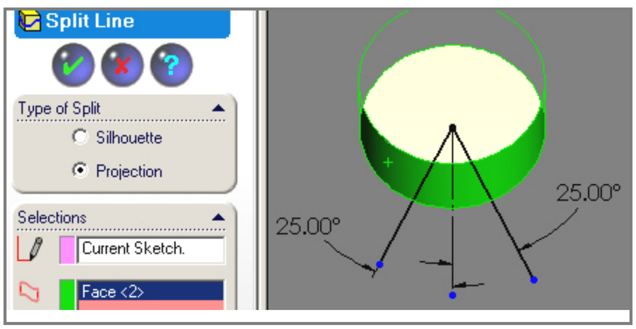


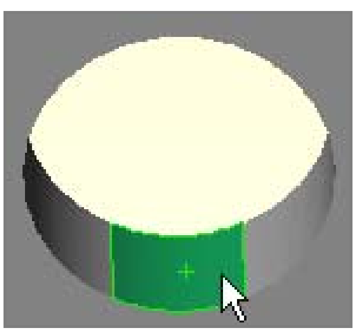
Figure 3-5 The original surface is now three surfaces

To form a vertical bearing region on the bolt hole sidewall:

1. Select the top surface and use **Sketch → Line**. "Wake up" the center point again. Use it to draw a straight line forward from the center that crosses the circle.



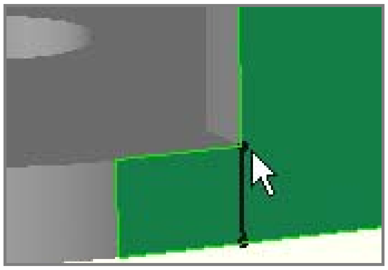
2. Convert it to a construction line and at two other radial lines offset by about 25°.
3. Select **Insert → Curve → Split Line** and pick the cylinder of the first bolt hole, click **OK**. That creates a new load bearing surface that could be restrained and/or control the mesh.



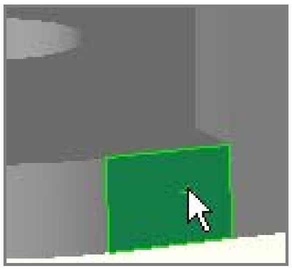
4. Repeat those operations for the second bolt hole.

The above surface splits were constructed to give more flexibility in applying various displacement restraints. You will need another surface split to help exercise engineering control over the revised mesh. You want the "L" shaped side area to have the leg split off so you can control a bending mesh there:

1. Right click on the (+*x*) right side face, **Insert Sketch→Sketch→Line**.
2. "Wake up" the left vertical line to put in a short vertical line that crosses the base.



3. Use **Insert →Curve→ Split Line** and select the "L" face, click **OK**.



Now you see it has become two rectangles. You can control the number of layers in the smaller one to account for local bending.

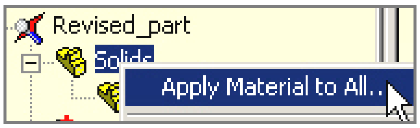
All of these new surface areas and lines can have different local mesh sizes prescribed to control our mesh generation. That is a standard feature of CosmosWorks, but you must supply the engineering judgment as to where any part needs the extra line or surface divisions for applying loads (or heat sources) and restraints in an analysis. True point loads or point moments are unusual in practice. They should be replaced by reasonable loading areas (that require split curves) and pressure distributions.

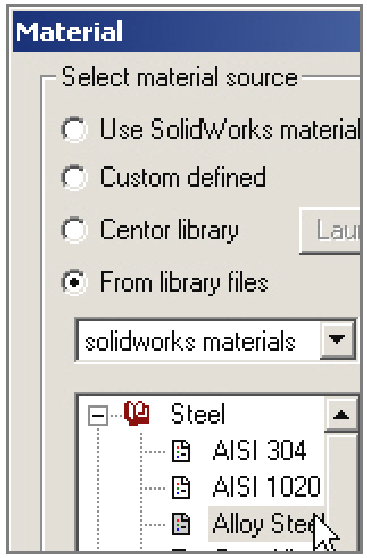
3.5 Beginning CosmosWorks study

Activate the CosmosWorks Manager by clicking on its icon (4th from the left):

1. Then start a static, solid part study.

2. Pick alloy steel as the part material. Note that the yield stress is about 90,000 psi.

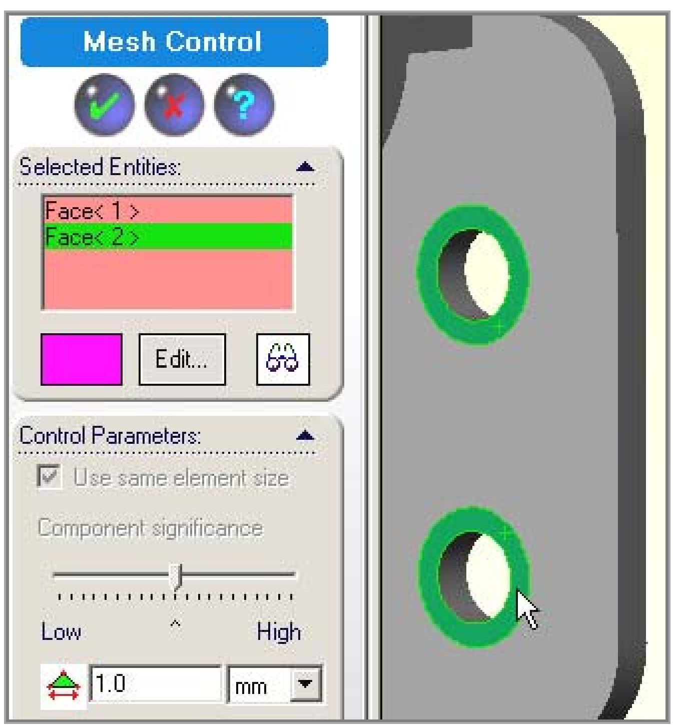




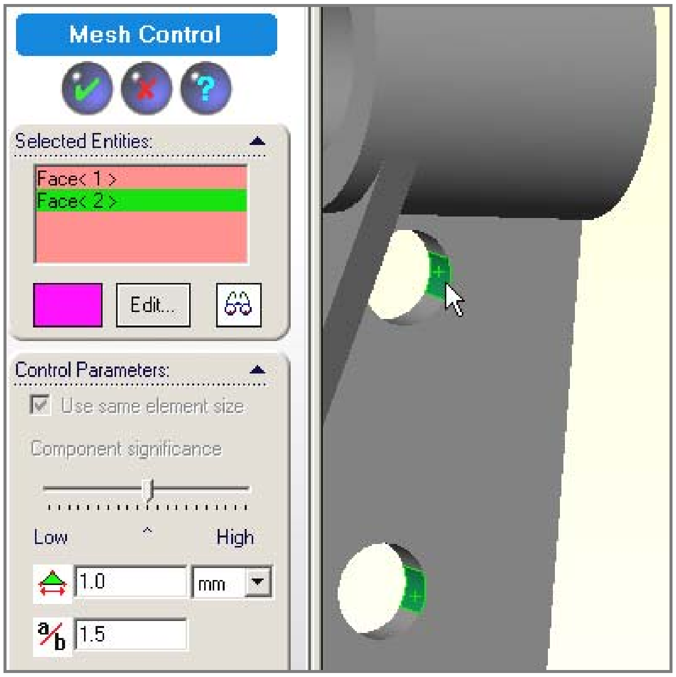
3.6 Mesh control

Wherever translational displacement restraints are applied, reaction forces are developed and localized stress concentrations are likely. Therefore, you want to assure that small elements are created in such regions. This process is referred to as mesh control. It is required in almost every study. Invoke it with:

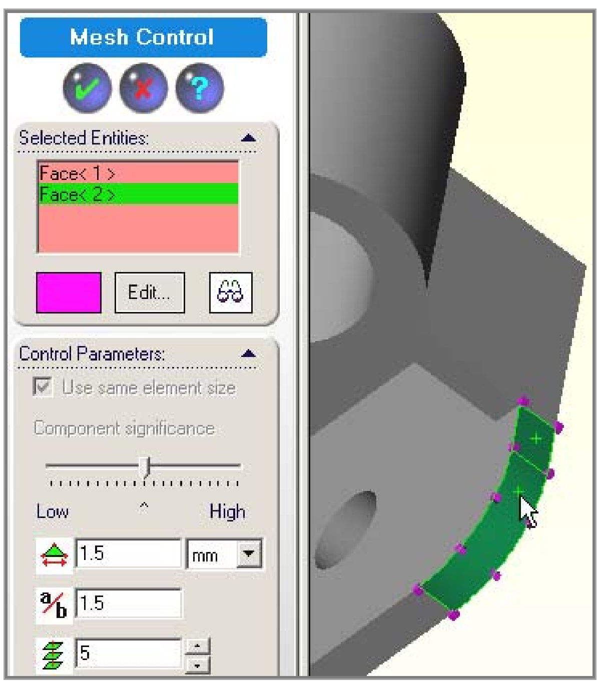
1. Right click on **Mesh→ Apply Control** to bring up the **Mesh Control panel**. The default element size of about 3 mm needs to be changed in the new regions created above. Select the region around the two bolt washers by picking those two surfaces, and setting the desired size to 1mm.



2. Likewise, for the two bearing surfaces within the smaller vertical cylindrical holes set the size to about 1.5 mm.



3. In the final corner region you need several elements through the thickness of curved front corner to accurately model the local bending seen in the first study. Select the small split rectangle and the adjacent cylindrical corner and use an element size of about 1.5 mm.



3.7 Mesh preview

Use **Mesh→Create** to generate the mesh. Then examine it and increase or decrease the local sizes specified above so that it looks acceptable, as shown in

Figure 3-6. Note that the mesh makes a smooth transition from the smaller element sizes to the larger default size in the far body regions. An additional refinement near the corner of the base and rectangular shaped leg (below the loaded tube) would also be wise. You should always preview the mesh before running the solution.

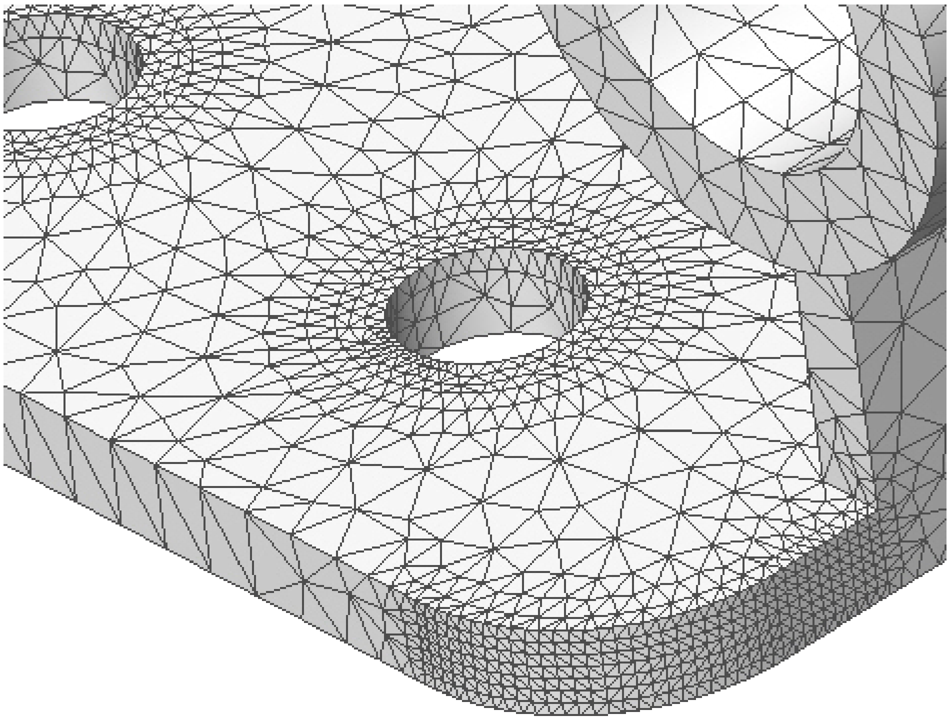


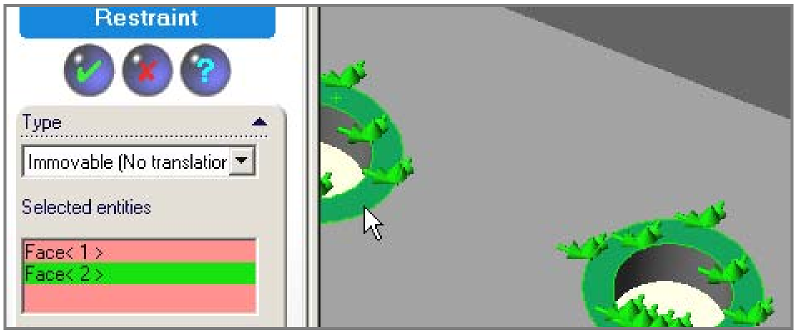
Figure 3-6 Controlled mesh sizes for the second model

3.8 Restraints

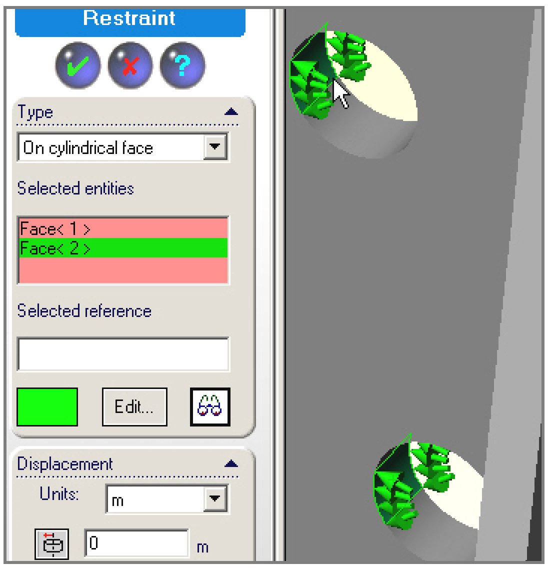
The new restraints will be enforced by beginning with the new surface areas representing the bolt washer contact regions:

1. Select **Load/Restraint→ Restraints** and restrain the washer areas against all three translation directions (solid elements do not have rotational dof).

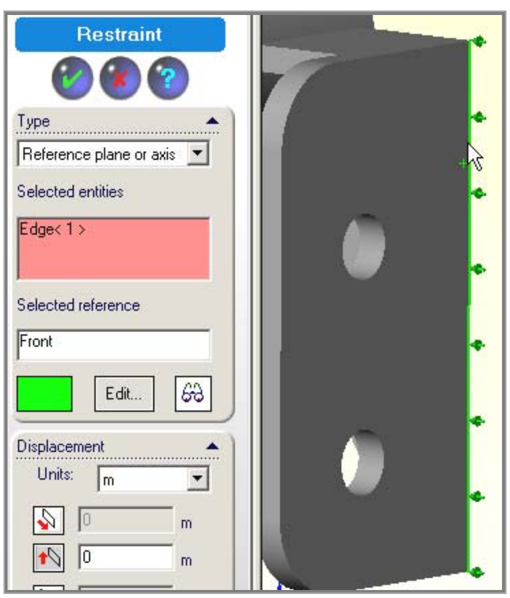
2. In the **Restraint panel** pick the two concentric ring faces as the **Selected Entities** and chose **Immovable** as the **Type**. That prevents all six possible rigid body motions (RBM) that can occur with a solid part.



3. Next select the two cylindrical bolt bearing faces and **restrain radial motion** on the **cylindrical face Type**. That prevents *z* translational RBM and the *y* rotational RBM.



4. Finally, this study will assume the bottom back edge helps support the part. Select that back edge line and choose the **reference plane Type**. Restrain the edge line displacement, relative to the front face, by picking the vertical direction (parallel to the front plane) and preview the restraint arrow to verify the correct choice. That prevents RBM in *y* translation and *z* rotation.

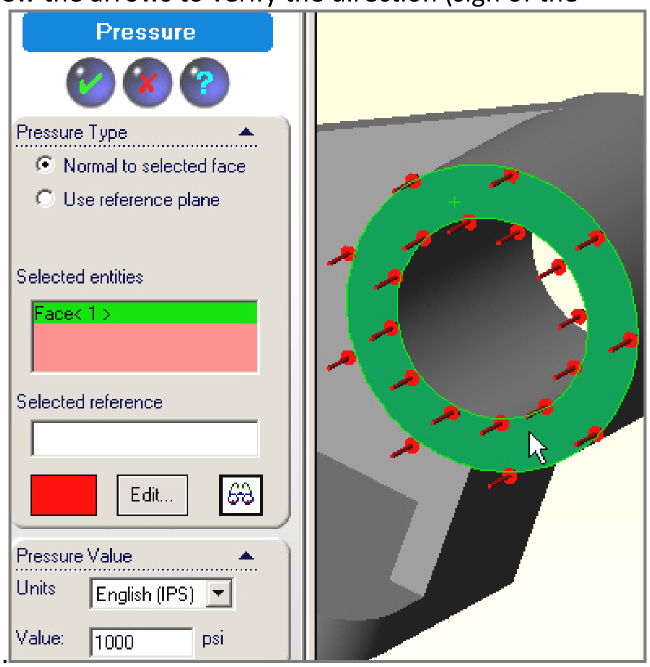


3.9 Pressure loading

This part has one pressure loading on the large front tube face. Impose it with:

1. **Load/Restraint→Pressure**.

2. In the **Pressure panel** give the **Pressure Type** as normal to the face, set the units and value (1,000 psi) and preview the arrows to verify the direction (sign of the pressure).



Clearly, the resultant applied force will be that pressure times the selected surface area. Since SolidWorks is a parametric modeler remember that if you change either diameter of the tube the area and the resultant force will also change. If you mean to specific the total force then use **Load/Restraint→Force** and give the total force on that area. It would not change with a parametric area change.

To check the total force caused by the above pressure (after the following displacement solution phase) you can check the reaction force since it must be equal and opposite. (You can do that in post-processing by right clicking on the **Displacement report→Reactions**.)

3.10 Run the study

Right click on the study name and select **Run.** Proceed to post-processing the results as shown below.

3.10.1 Displacement results

The displacements are shown amplified in

Figure 3-7 along with the gray undeformed part. As with the first approximation, the main bending region is the rounded front corner of the base plate. The mesh there is now fine enough to describe the flexural stress and shear stress as they change through the thickness. The default contour plot style for viewing the displacements is a continuous color variation. These are pretty and should be used at some point in a written stress analysis report, but they can hide some useful engineering checks.

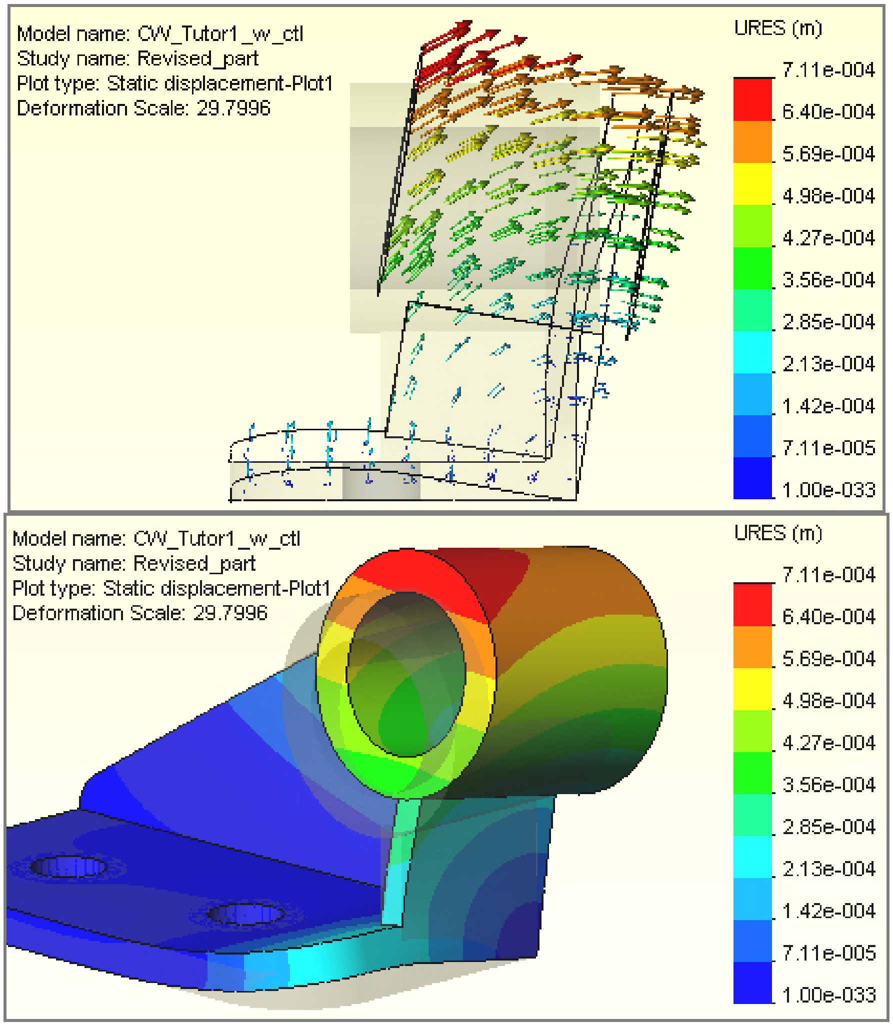


Figure 3-7 Amplified part displacements

To create a more informative displacement vector plot:

1. Right click in the graphics area. Then pick **Edit Definition→Vector→Line→OK**. When the displacement vector plot appears use **Vector Plot Options** to dynamically control the clarity. Rotate this view (hold down the center mouse button and move the mouse) to find the most informative orientation. That is probably the right side view seen in the top of
2. Figure 3-7.
3. Next, convert to a discrete contour display of the magnitude of the displacement vector (bottom of that figure). It is more difficult to interpret than vector plots.

3.10.2 Effective stress

The effective stress distribution on the top of the base is shown in Figure 3-8. There two different contour options are illustrated. The line contour form tends to be more useful if preliminary reports are being prepared on black and white printers. The material yield stress was 90,000 psi, so every region colored in red is above yield. In that figure the **Edit Definition** feature was used to manually specify the stress range and the number of discrete colors. Rotating the part shows that there are other regions of the part above the yield stress. On the bottom corner of the base the assumed line support has also caused high local stresses, as seen in Figure 3-9. In that figure you should also note that the bottom edge of one bolt bearing surface shows a small region of yielding. That should be examined in more detail and other views.

Line or point restrains are not likely to exist in real part components. You could revise that region by putting in a split line to create a narrow triangular support area. That would be more accurate, if the resulting part stresses there are compressive. Otherwise, you would need to use a contact surface. Using contact surfaces where gaps can develop requires a much more time consuming iterative solution. However, the important thing is to attempt an accurate model of the part response, not an easy model.

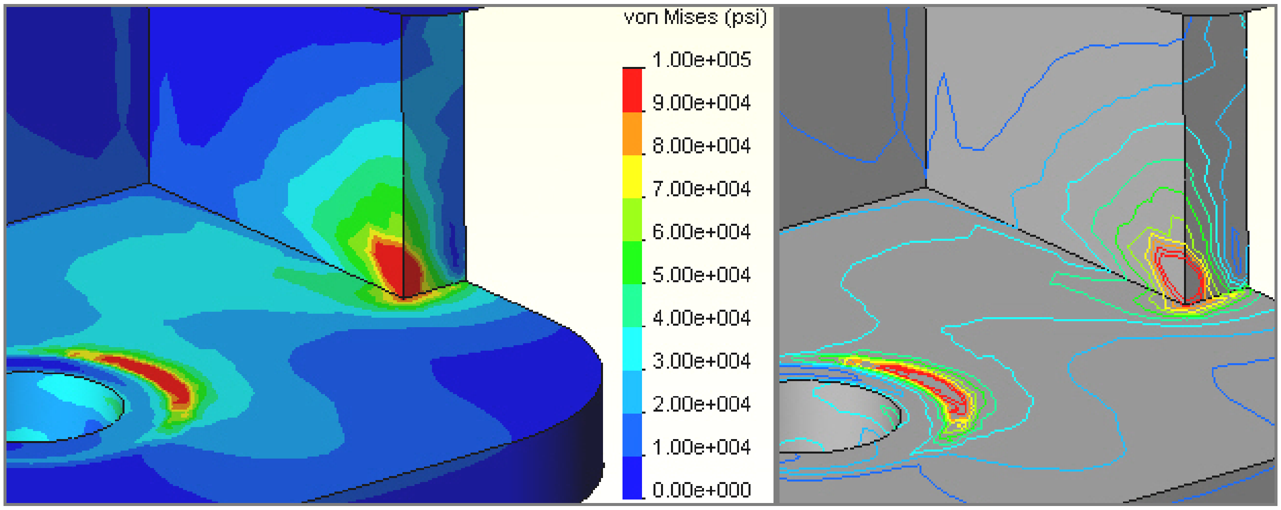


Figure 3-8 Failure criterion on the base top surface

3.10.3 Maximum principal stress

The extreme values of the stress tensor at any point are known as the principle stresses (eigenvalues of the stress tensor). For 3D parts here are three normal principle stresses and a maximum shear stress. The principle normal stress components have both a magnitude and direction. They can be represented as directed line segments with two end arrow heads used to indicate tension or compression. The maximum principal stress vector plot, in Figure 3-10, shows the highest tension stress (and is a failure criterion for some materials). Here, it occurs in the yielding region where the base joins the vertical tube support. Those high stress concentrations could (and should) be reduced by adding fillets along the associated edges along the re-entrant corners.

The main goal of this study was to illustrate the usefulness of split surfaces, and the need for engineering judgment in making restraint assumptions. The locations and type of restraints are usually the least well know aspect of a part analysis or design. Load conditions are probably the next least reliable information. Use friendly software to investigate various combinations of loads and restraints to get the safest results.

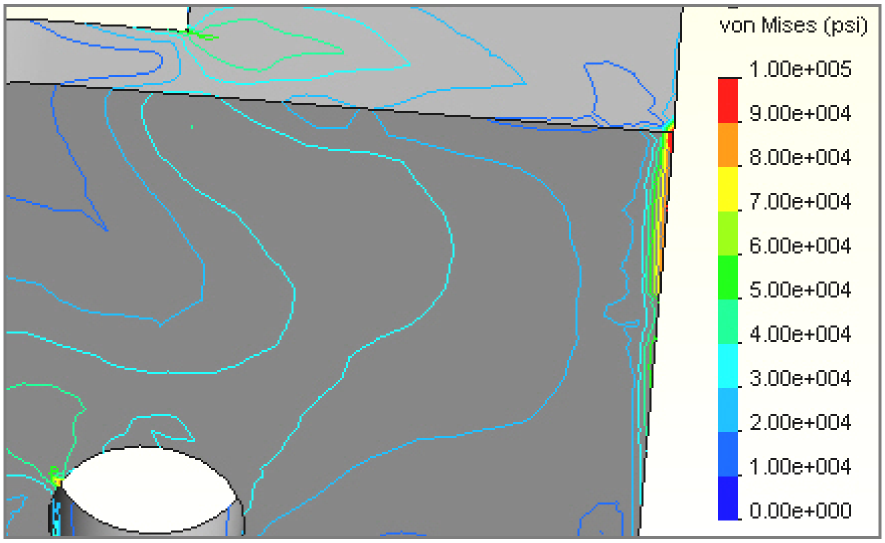


Figure 3-9 Failure criterion in the base bottom region

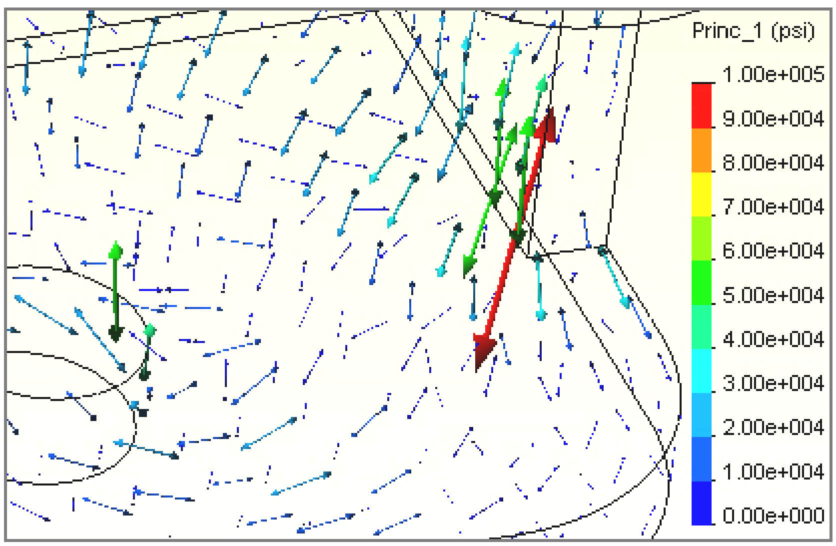


Figure 3-10 The maximum principal stress vectors

3.11 Other aspects of mesh generation

CosmosWorks has a very powerful solid mesh generator for tetrahedral elements. Almost every analysis requires the engineer to employ judgment on where to apply controls to the mesh. Usually the mesh needs to be refined around restraint regions, load regions, and reentrant regions of the solids. Figure 3-11 illustrates that you can control element sizes on faces, edges, and vertices. The earlier example showed that you should plan ahead while building your solid to allow for expected mesh control needs and insert split lines into your solid to allow for additional entities to be selected on your model. Mesh control lets you specify the desired element size, the rate at which neighboring elements increase in size, and how many layers of transition elements you want around the selected geometric entity.

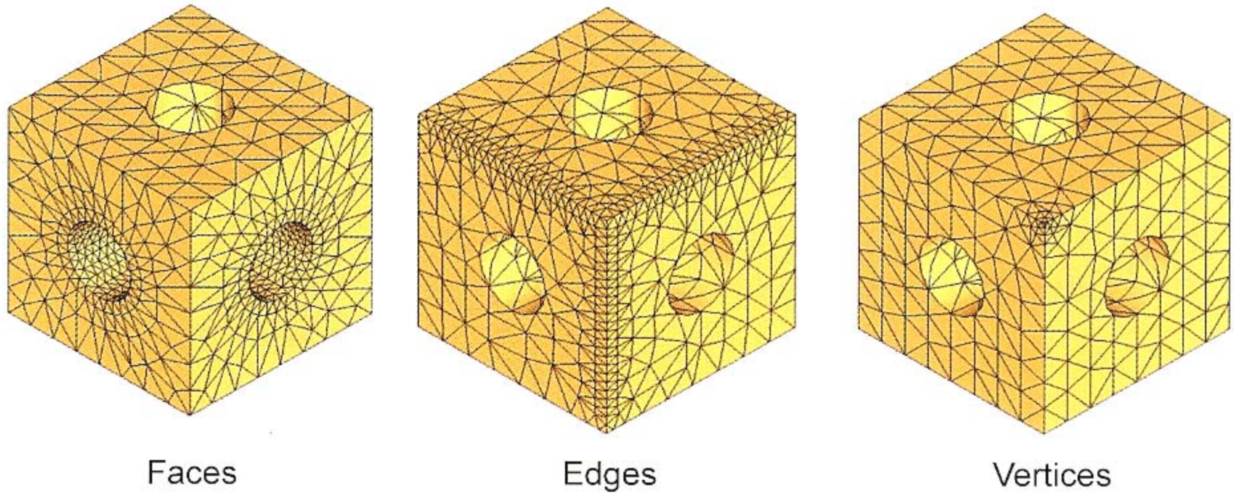


Figure 3-11 Options for local mesh control

Any sharp (no filet) reentrant corner in theory causes infinite derivatives at the corner in heat transfer and stress analysis studies. A fillet removes the infinite stresses, but interior fillets usually need a reasonable amount of mesh refinement (see Figure 3-12). Exterior fillets are less important in getting accurate results and are sometimes suppressed in finite element studies. Split lines are optional for some solids but are often mandatory to properly join and align shell meshes as seen in Figure 3-13. This is also important when connecting shells to solids.

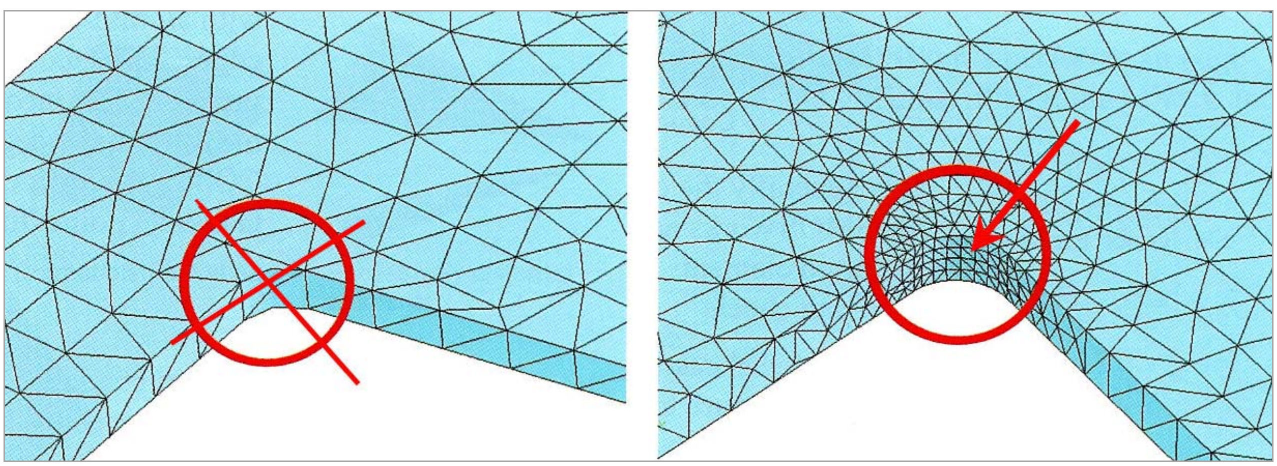


Figure 3-12 Avoid a small number of elements along arcs

Bad solid modeling practices are probably the most common cause of failure of the mesh generation. The mesh generator begins with the edges of a solid. It divides them into segments corresponding to the requested element sizes. Then it fills the faces with triangles of the proper size. From the triangles it proceeds inward to fill the volume with tetrahedrons. If the edge lines are much smaller that the requested element size (Figure 3-14), or if they join at very small angles then the mesh generator tries to use tiny elements to match the poor local part geometry. If it is possible to do that the mesh is still likely to fail due to insufficient computer memory. Part flaws are often too small to see without zooming in on the part.

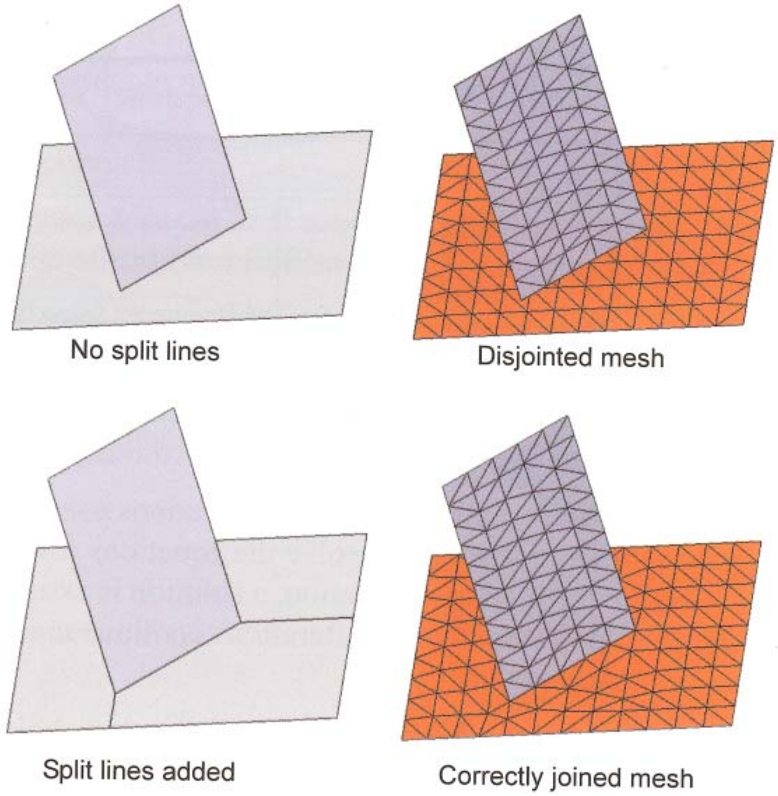


Figure 3-13 Split lines give you more mesh control

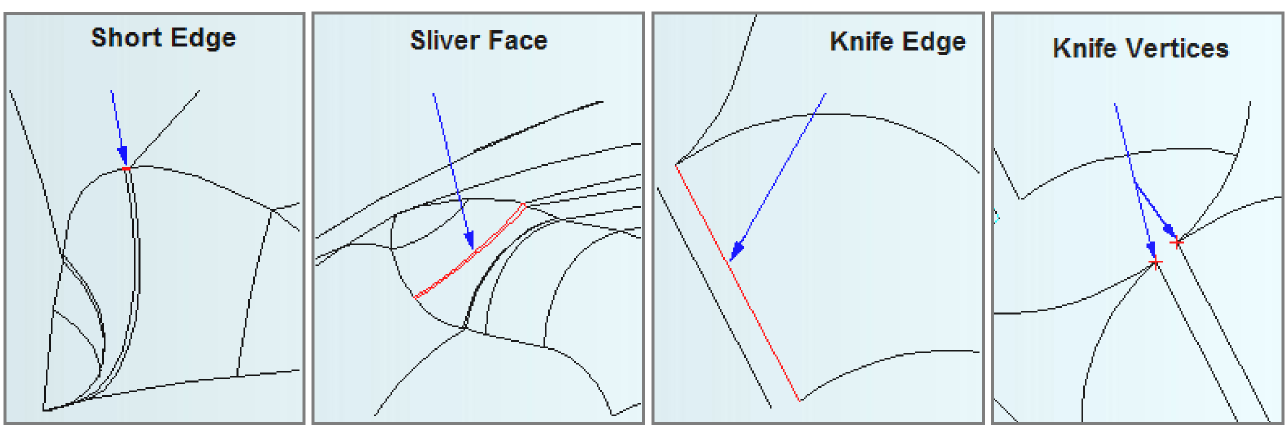


Figure 3-14 Bad solid models cause meshing failures

4 General Solid Stress Analysis

4.1 Introduction

Every part, at some level, can be thought of as a 3D solid. That is the default analysis mode of CosmosWorks. You must start every study by building a solid, even if you in turn model it as a shell or frame. Therefore, you need to learn the numerous options that are available to support such studies. To validate the results of a 3D solid study you often need to use a beam, frame or shell model. For the proper assumptions those lower dimensional studies can be quite accurate, and are almost always much less demanding of computer resources. You will find that you never have large enough computer resources and you will have to learn how to use symmetry, anti-symmetry, beams, frames, shells, and trusses to reduce some problems to a size that can be solved with your available resources.

4.2 Flexural analysis of a Zee-section beam

4.2.1 Introduction

In this study you will validate your understanding of the use of Cosmos by solving a cantilever beam and comparing the FEA results to that predicted by mechanics of materials theory. The constant cross-section is a zee-shape in the x-y plane as seen in Figure 4-1. It extends in the z-direction for a length of L = 500 mm. The thickness of the section is t = 5 mm, each flange has a length of a = 20 mm, and the web has a depth of h = 2a = 40 mm. At the free end it is loaded by a distributed force parallel to the y-axis (i.e., vertical). A detailed outline of the construction of the solid part model is given in Appendix A.

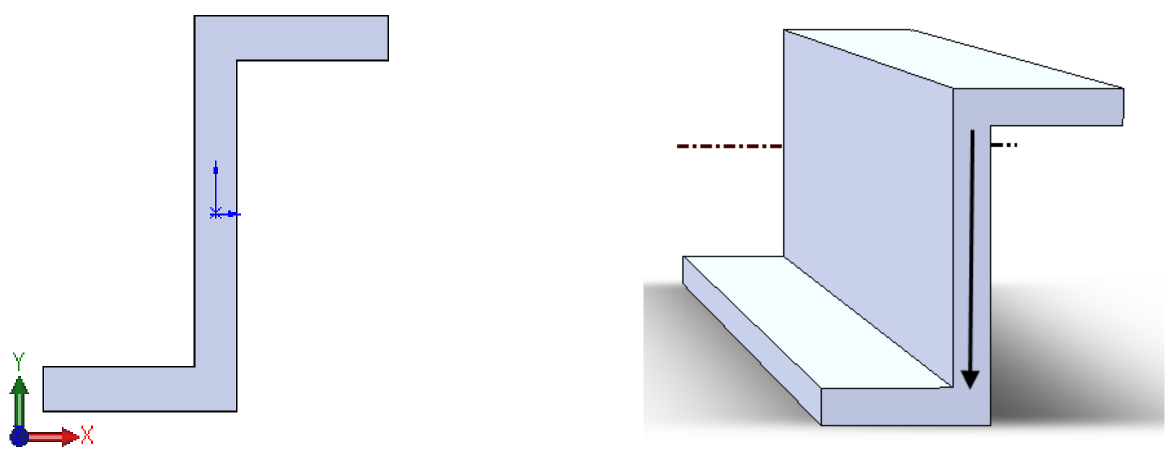


Figure 4-1 A Zee section straight beam solid

4.2.2 Validation estimate

Before you start a FEA study you should try to get a reasonable approximation of the stresses and deflections to be obtained. This can be an analytic equation for a similar support and loading case, a FEA beam model compared to a continuum solid model, a one or two element model that can be solved analytically, etc.

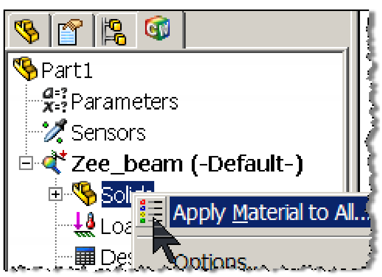
The cantilever is horizontal and has a vertical load of P = 500 N. Therefore, it causes a bending moment, about the x-axis of M = P (L – z), where z is the distance from the support. Recall (*for symmetric sections*) that such a loading causes a linear flexural stress (σ_z) that varies linearly through the depth. It is zero at the neutral axis (here parallel to the x-axis at the section centroid) and has a maximum constant tension along the top edge, and a corresponding constant compression along the bottom edge (parallel to x). The load P causes a varying moment and a transverse shear stress (τ) that varies parabolically through the depth and has its maximum at the neutral axis. Those stress behaviors are sketched with the section in Figure 4-2. The flexural and shear stress equations are $\sigma_z = M\, y / I_x$ and $\tau = P\, Q / t\, I_x$ where I_x is the second moment of the section and Q is the first moment of the section at y. For this section $I_x = 2/3\, t\, a^3$. The maximum constant tension will occur at y = a + t/2, while compression occurs at -y. Likewise, the end deflection of the beam in the vertical (y) direction will be $U_y = PL^3 / 3EI_x$. With that review and its predictions you can now proceed with the FEA study.

Figure 4-2 Flexural (left) and shear stresses from symmetric beam theory

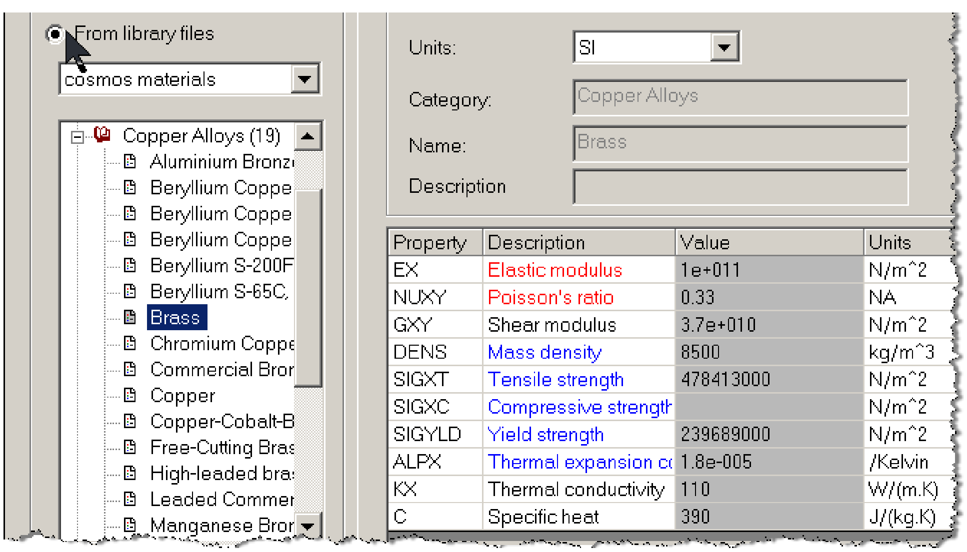
4.2.3 Cosmos study

Select the CosmosWorks icon to open its **manager menu**:

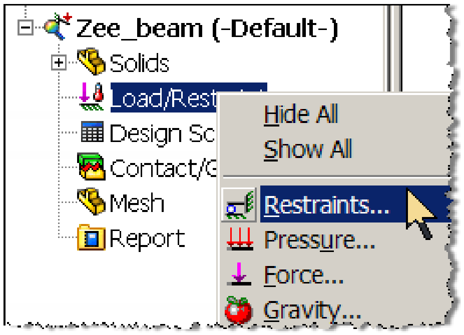
1. Right click on the **Part Name→Study**
2. Select **Solid Mesh**, and enter the new **Study Name** (Zee_beam).
3. Right click on **Solids→Apply Material to All**.



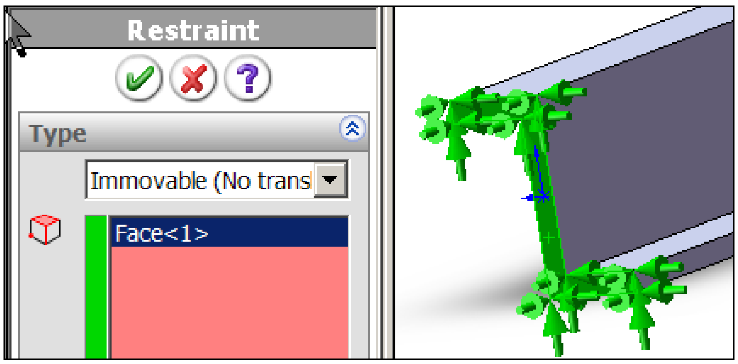
4. In the **Material panel** pick **From library files**, and select SI units.
5. Select **Copper Alloys→Brass** and review the properties (and significant figures)



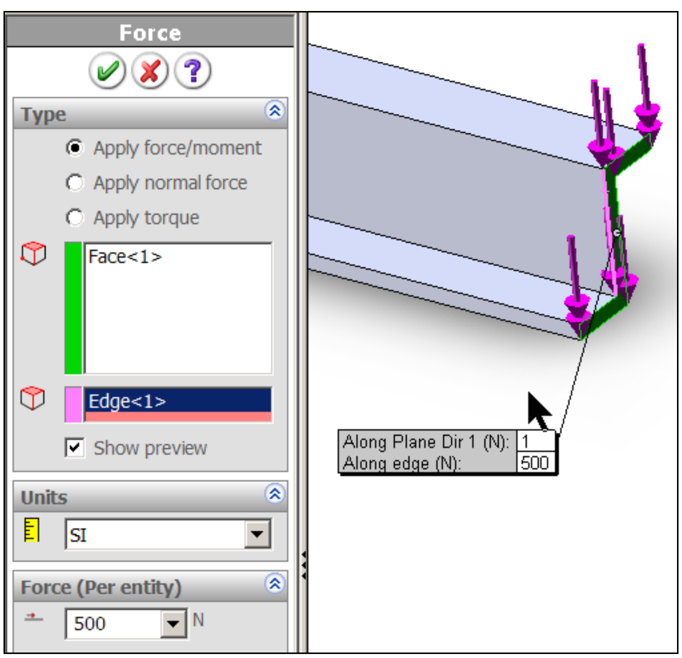
6. Right click on **Load/Restraint→Restraints** to open the **Restraint panel**.



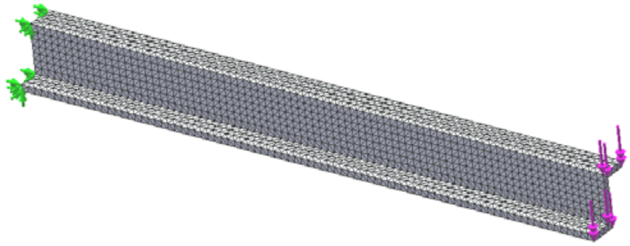
7. Pick **Immovable** (no displacements) and select one end **face** of the beam.



8. Right click on **Load/Restraint→Loads→Apply Force**. Select the other end **face,** a vertical **edge** for the direction, set the **value** at 500 N.



9. Next create a default mesh, right click on **Mesh→Create Mesh→OK.**



Since local flange bending is not expected to be high, the default mesh with only one quadratic solid through the thickness should be acceptable. Otherwise you should have at least three elements through the thickness in a region of expected local bending stresses. There are enough elements through the height and length of the solid to model the high bending expected near the **immovable** (cantilever) restraint. Having reviewed and accepted a default mesh we execute the problem and recover selected results:

1. In the **CosmosWorks manager** menu select the **Study Name→Run**.
2. When the **Results** list appears right click on **Stress→Edit Definition**.
3. In the **Stress Plot** panel select **SZ: Z normal stress** as the component and **Fringe** as the display type. SZ was selected as the first display since it is the normal stress component parallel to the beam axis that you would validate with beam theory.
4. Optionally control the stress display by right clicking in the graphics area **Settings→ Settings panel→Discrete** fringe options, and to better see the stress differences:
5. Right click in graphics area **Chart Options→Chart Options panel→5** color levels.

The resulting stress contours (Figure 4-3) have a maximum value of about 152 MPa, in both tension and compression. The contour spatial distributions are not what you would expect from symmetric beam theory. That theory predicts the flexural stress contours on the top and bottom to be parallel to the restraint wall (perpendicular to the beam axis). Yet the actual contours are almost parallel to the beam axis. In other words, symmetric beam theory predicts a neutral axis (NA), at the beam half depth and parallel to the flange.

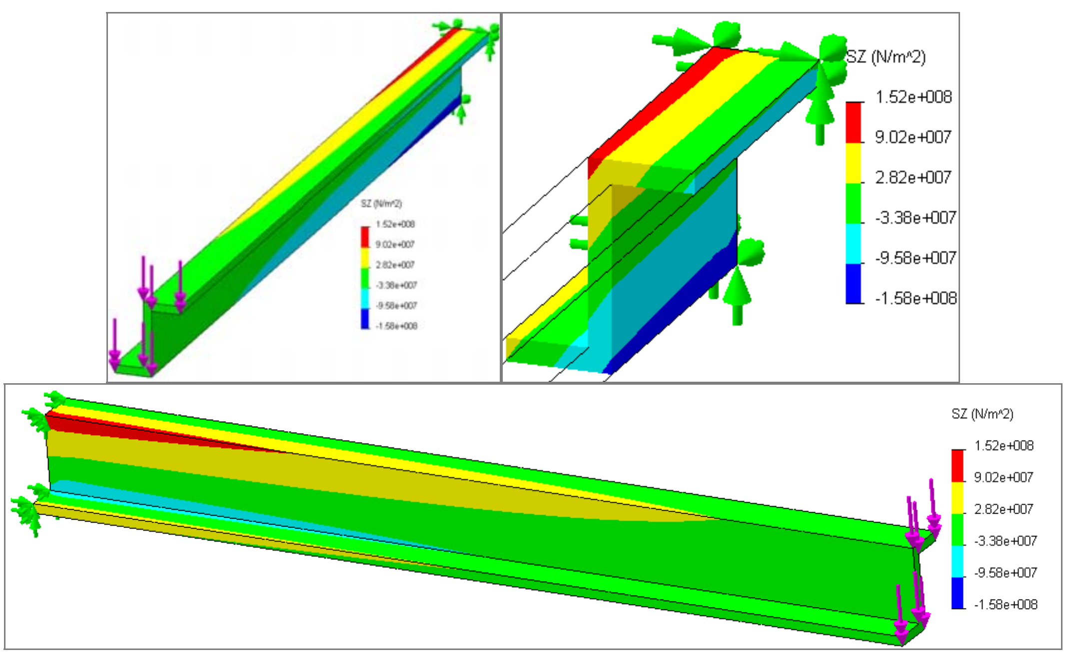


Figure 4-3 Axial flexural stress levels

That is, the NA would be expected to be parallel to the global X–axis. Instead of zero normal stresses there, they are zero along an inclined line rotated about 55 degrees w.r.t. the X-axis. The actual NA is highlighted In

Figure 4-4. Points above the NA are in tension here and those below are in compression. Figure 4-5 shows a similar distribution for the von Mises stress.

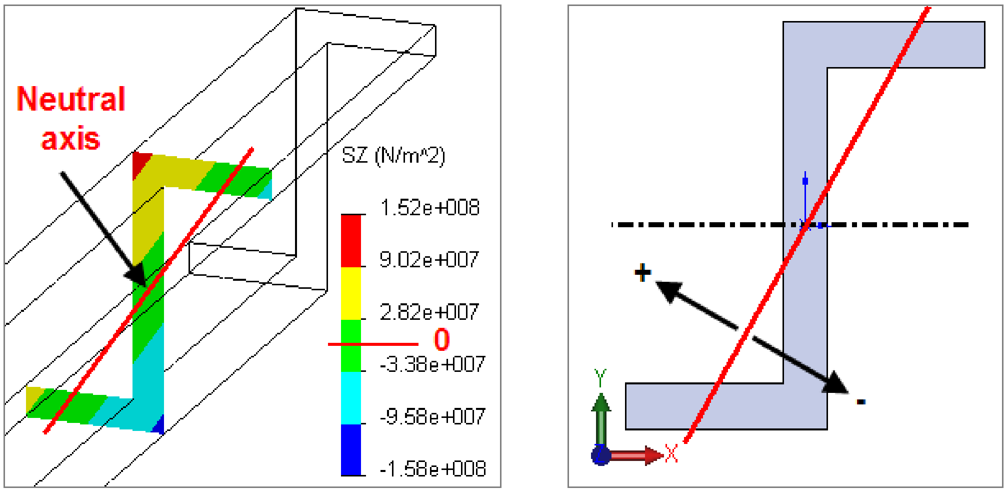


Figure 4-4 The bending neutral axis of the section

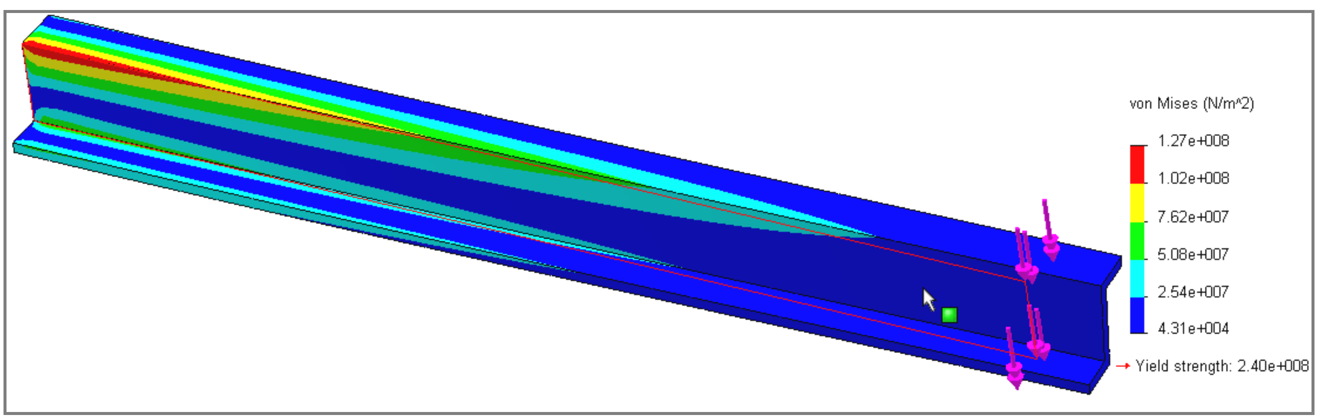


Figure 4-5 The von Mises effective stress distribution

Since the stress distribution is quite different from symmetric beam theory you should also look at the deflections in detail. Symmetric beam theory says that the deflection is a maximum at the free end and lies in the z-y (side) plane and there is no deflection in the top (z-x) plane.

However, Figure shows that there are significant displacements out of the plane of the beam web and resultant loads. The graphs in Figure 4-7 verify that the horizontal (top view) deflections are larger than the (side view) vertical deflections. Therefore, there was something wrong with the 1D mechanics of materials concept that was selected to validate the solid finite element study (or you made an error in using Cosmos).

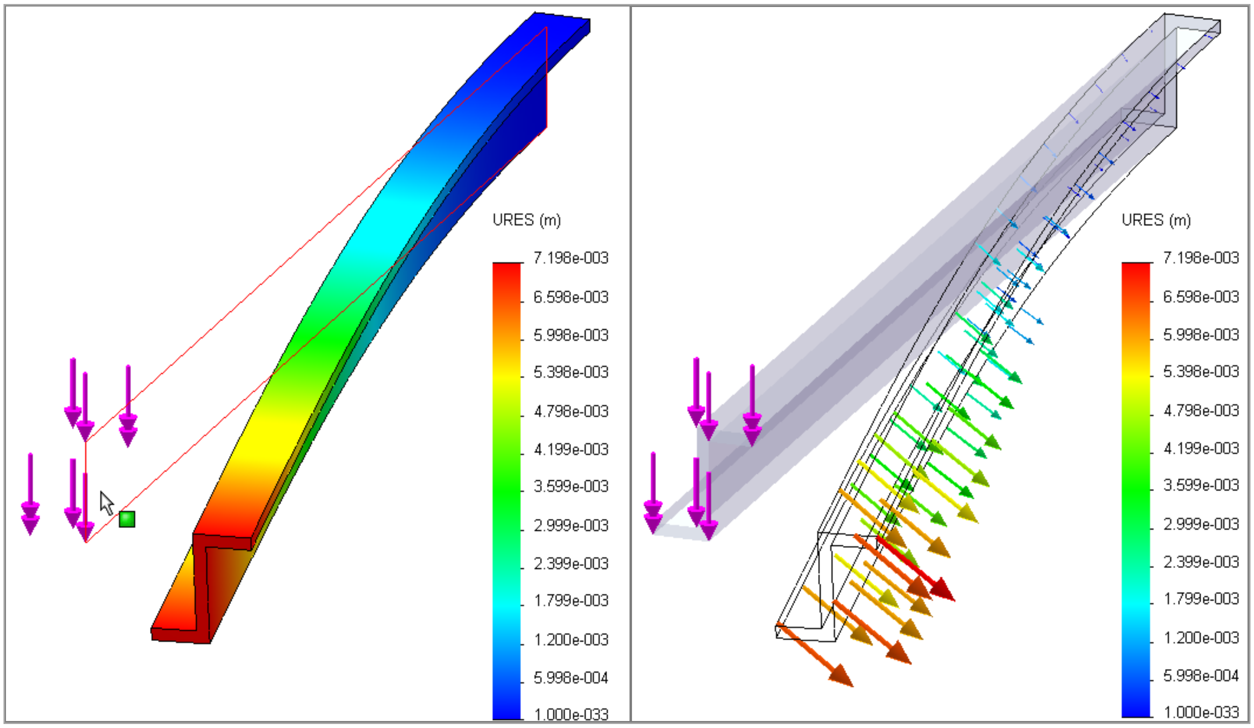


Figure 4-6 Out of plane displacements of Zee member

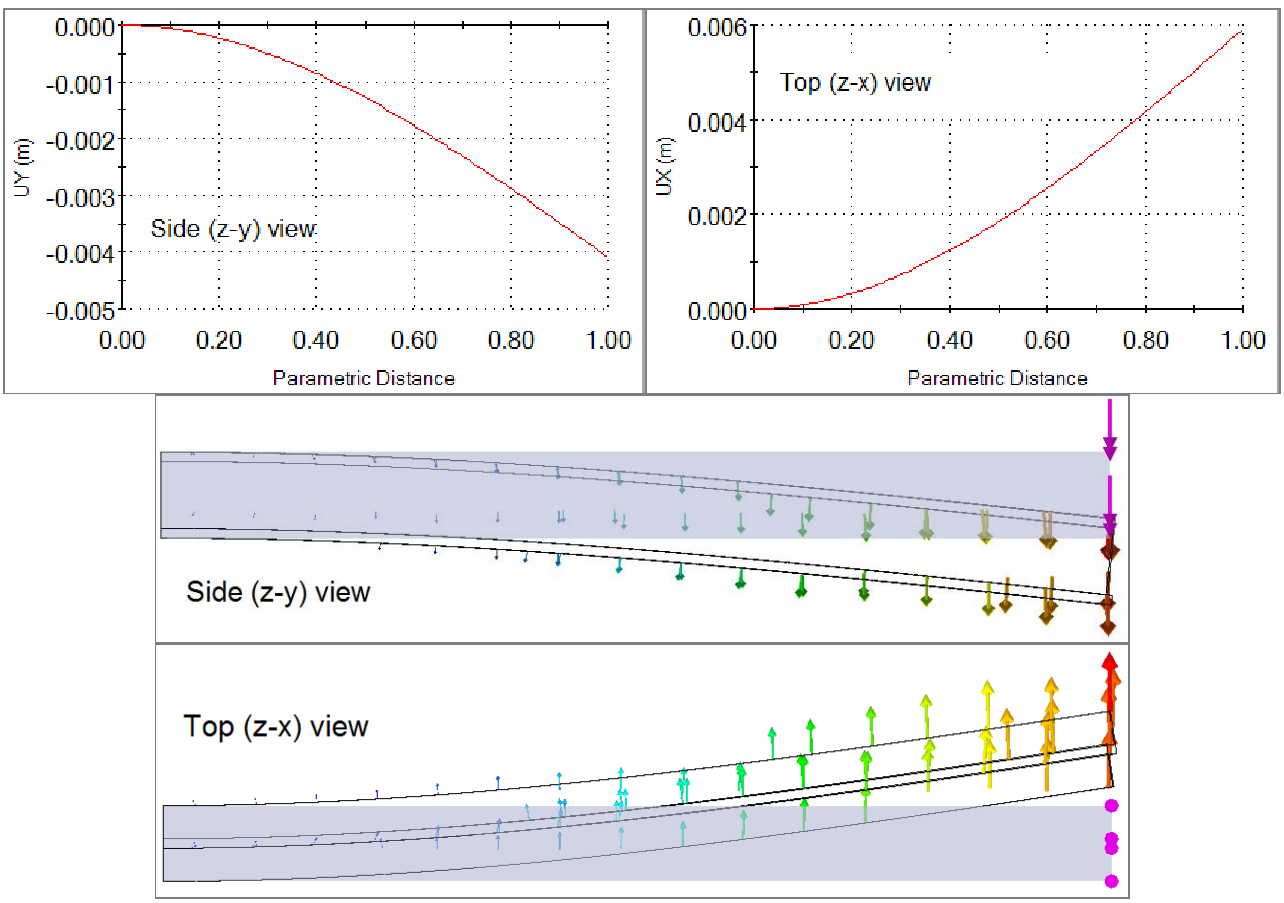


Figure 4-7 True shape displacement data for Zee member

4.2.4 Results comparison and re-validation

The simplified predicted deflection of U_y = 0.0078 m is almost twice as large as U_y = 0.0042 m found in Figure 4-7. That figure also shows a value of U_x = 0.0059 m while the simplified theory predicts zero. Likewise, the simplified normal stress estimate predicts a constant stress along the top and bottom edges of the beam, but that was not observed in the solid solution. Of course the FE results and the validation predictions do not agree! The simplified 1D mechanics relations are only valid for straight symmetric sections. Usually those sections have two planes of symmetry. But they must have at least one symmetry plane, so as to make the product of inertia vanish ($I_{xy} \equiv 0$). The current section does not even have one symmetry plane. Its geometric inertias are $I_x = 2/3\ t\ a^3$, $I_y = 3/8\ t\ a^3$, $I_{xy} = -\ t\ a^3$, and its cross-sectional area is $A = 4\ t\ a$. The unsymmetrical 1D beam theory predicts that the NA axis will rotate from the x-axis by an angle of $\alpha = \tan^{-1}(-I_{xy} / I_x) = 56.3$ degrees, which seems to agree with

Figure 4-4. Any time $I_{xy} \neq 0$, one must employ non-symmetric beam theory for bending. The general theory [10] states that the cross-sectional normal stress varies according to the equation:

$$\sigma_z = [(M_x I_y - M_y I_{xy})y - (M_y I_x - M_x I_{xy})x] / (I_x I_y - I_{xy}^2),$$

but since in this case $M_y = 0$

$$\sigma_z = M_x (I_y\, y - I_{xy}\, x) / (I_x I_y - I_{xy}^2) = P\, L\, (6\, y - 9\, x) / (7\, t\, a^3),$$

which reduces to the original stress estimate only for $I_{xy} = 0$. On the top most horizontal line of the flange (y = a) the above stress is estimated to vary from a corner tension of about $P\ L / 7\ t\ a^2$ to a compression stress of 3/2 of that value. The more general 1D displacement predictions are

$$U_y = -\,P(L - z)^3 I_y / 6\, E(I_x I_y - I_{xy}^2) + C_0\, z + C_1 = P/(7\, E\, t\, a^3)[(L - z)^3 + 3L^2 z - L^3],$$

$$U_x = P(L - z)^3 I_{xy} / 6\, E(I_x I_y - I_{xy}^2) + C_2\, z + C_3 = -\,3/2\, U_y$$

which yield maximum values of $U_{y\,max} = -\,2\ P\ L^3 / 7\ E\ t\ a^3$, $U_{x\,max} = 3/2\ U_{y\,max}$ and $U_{max} = 0.515\ P\ L^3 / E\ t\ a^3$. For the given dimensions the above estimates reduce to $\sigma_{z\,corner}$ = 17.9e7 MPa at the restraint wall and U_y = -0.0045, U_x = 0.0067, and U_{max} = 0.0081, meters at the free end, respectively. The new validation estimates agree with the solid study results reasonably well. The results do suggest using a finer mesh in the corners near the restraint wall.

4.3 Shoulder implant-cement-bone assembly

4.3.1 Introduction

This is a simplified preliminary analysis of half of an artificial shoulder joint prosthesis to be cemented into the clavicle bone. It is assumed that the spherical ball in the proximal arm joint will contact the implant surface off center and apply a uniform pressure to it. A glue layer of specified thickness is required to be placed between the implant and the bone. The conceptual design is seen in Figure 4-8 where the implant is gray, the cement yellow, and the bone is purple. To assure that all three components mate properly they will be constructed in a series of coupled solid bodies. Our goal is to mate the bodies, prepare a mesh, and obtain the stresses in the implant and bone.

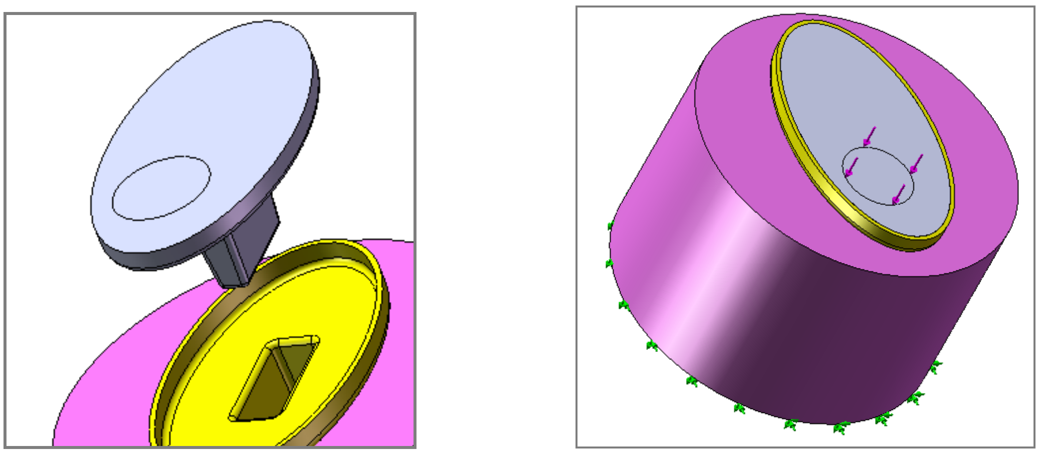
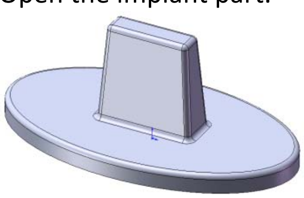


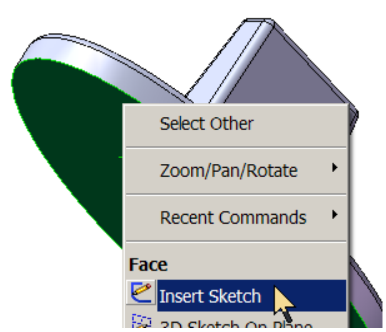
Figure 4-8 Conceptual shoulder implant

4.3.2 Implant loading area

Assume that the implant part has been constructed and that you need to specify its loading surface for this load case. The contact pressure area is assumed to be circular and offset from the center of the elliptical bearing face. Note that the two offset center dimensions could be defined as design parameters in Cosmos for automating the evaluation of several possible loading positions. Construct the loading area:

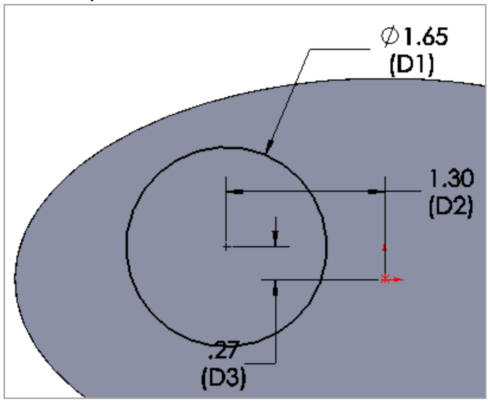
1. Open the implant part.



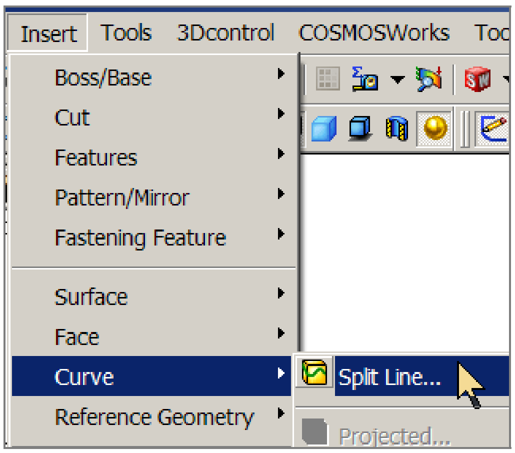


2. Select the front face and insert a sketch.

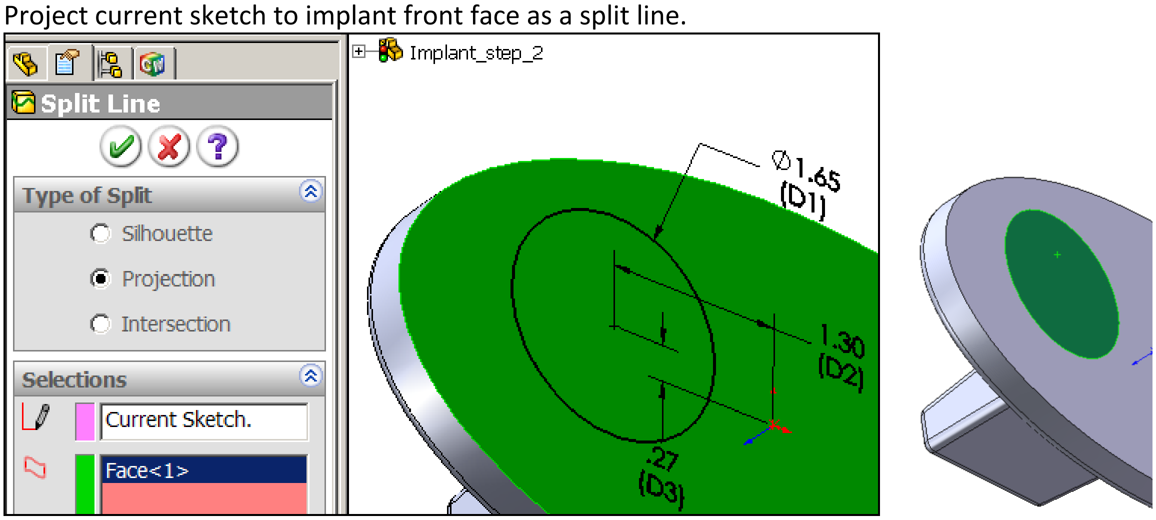
3. Locate pressure area on the front face.



4. Activate a split line feature.



5. Project current sketch to implant front face as a split line.



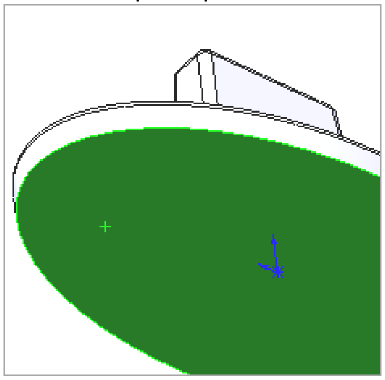
This defines the desired circular loading area. After preliminary studies you could include the spherical joint and use a nonlinear contact analysis to define the true contact size.

A surgical tool will be provided to ream out a hole in the bone to allow for a uniform thickness of bone cement. That is, the bone hole is offset from the final desired implant position by a constant amount. Having saved the above changes to the surface of the implant the implant solid will be used to define the cement and bone solids.

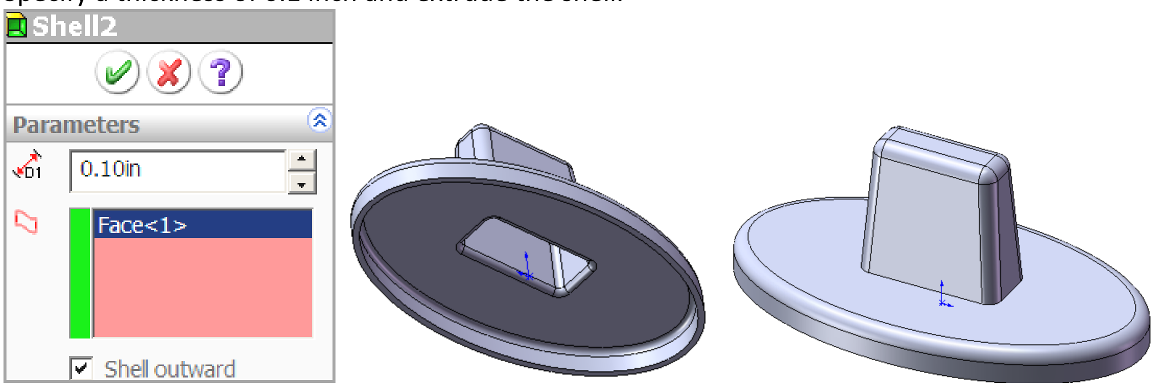
4.3.3 Construct the cement layer

The bone cement is to be applied, with a constant thickness, to the outside of the implant. That is easily modeled here as a "shell outward" extrusion:

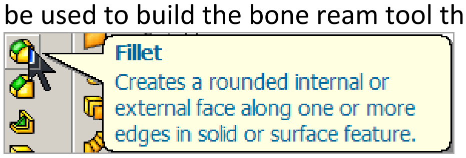
1. Pick the implant part and identify the front face that is to remain open in the new cement part.



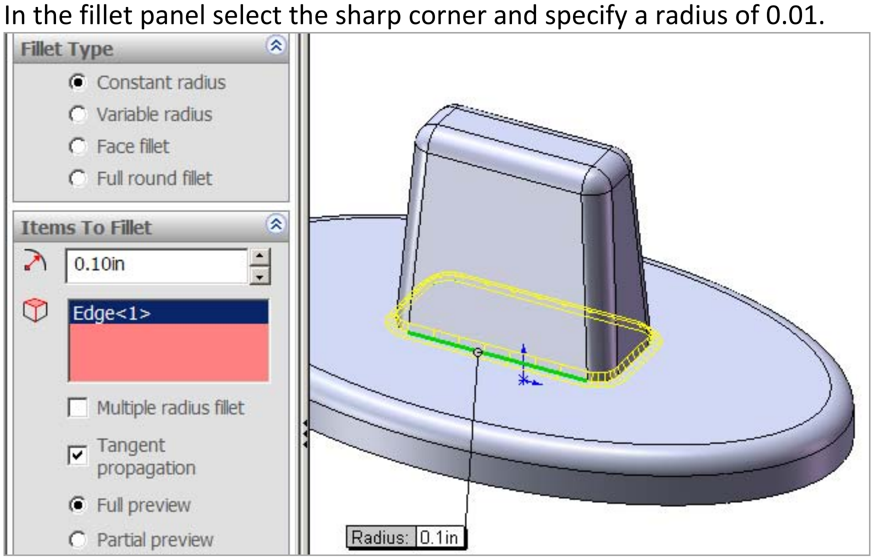
2. Specify a thickness of 0.1 inch and extrude the shell.



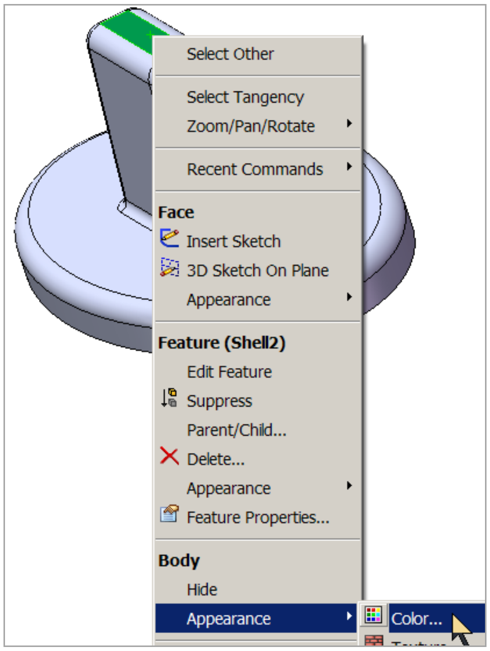
3. The bone side of the cement shows a sharp edge around the implant post. Since that surface will also be used to build the bone ream tool that edge should be rounded. Select the fillet icon.

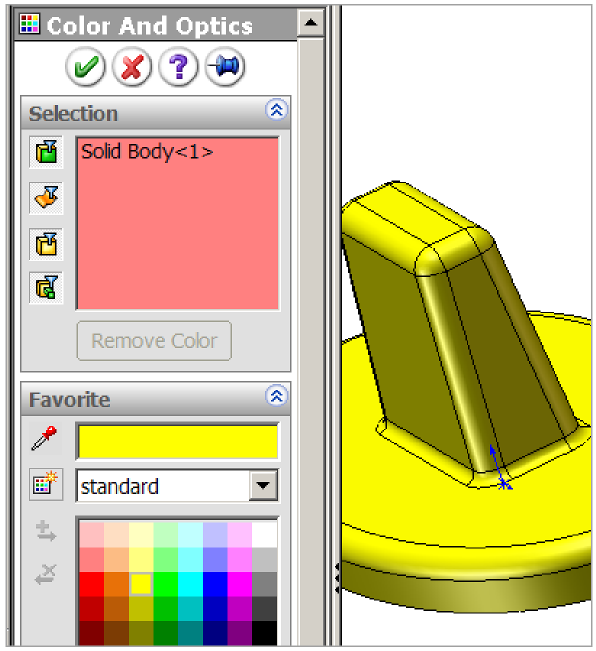


4. In the fillet panel select the sharp corner and specify a radius of 0.01.



5. Change the color of this part. Click on a surface, select **Body→Appearance→Yellow→OK**.



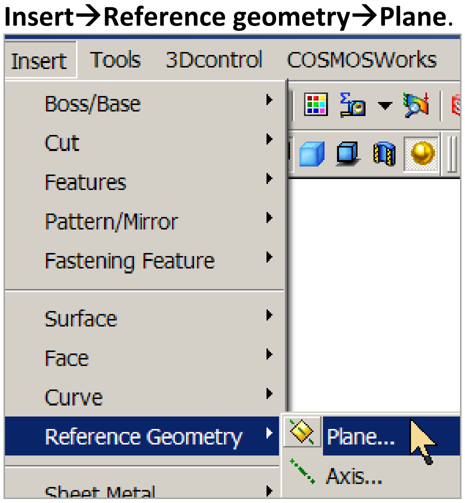


The resulting cement part is named and saved. That part is not closed because it also defines the enlarged hole shape that must be cut from the bone (a Boolean operation). The implant part is hidden to reduce the screen clutter.

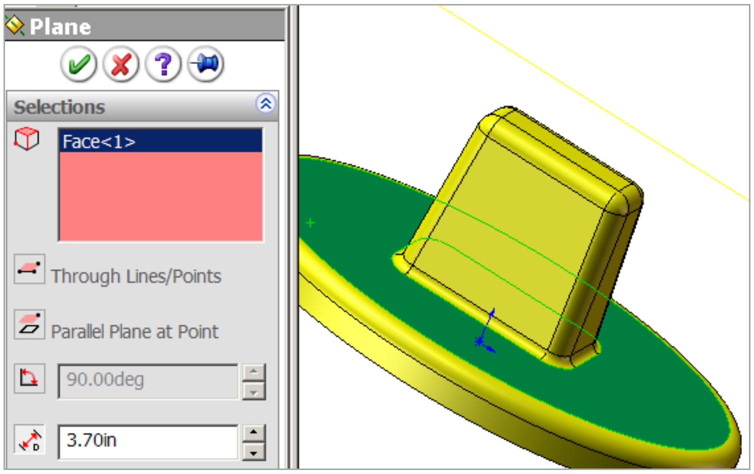
4.3.4 Add a mating bone solid

The bone in this region is about twice as thick as the implant post and will be approximated as a cylinder. The cement and implant are intended to go into the center of the supporting bone. Thus, you can define the cylindrical bone part relative to the back center of the cement so as to make it easy to mate the assembly later. To make the upcoming Boolean operations easier to view it is best to build a construction plane that is offset from, but parallel to, the back of the cement part:

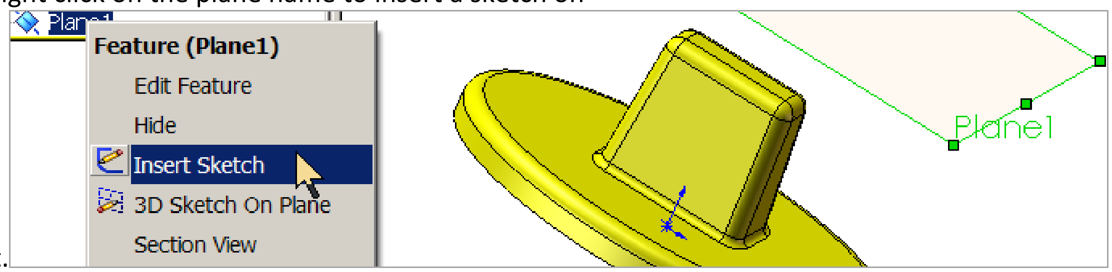
1. **Insert→Reference geometry→Plane**.



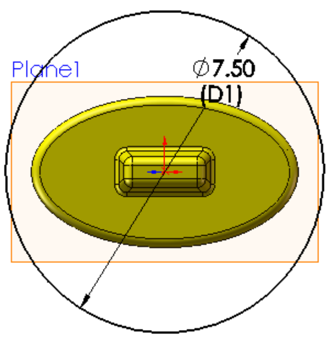
2. Select a flat face of the cement and offset a plane from it to be used to build the approximation of the bone. The default name is Plane1.



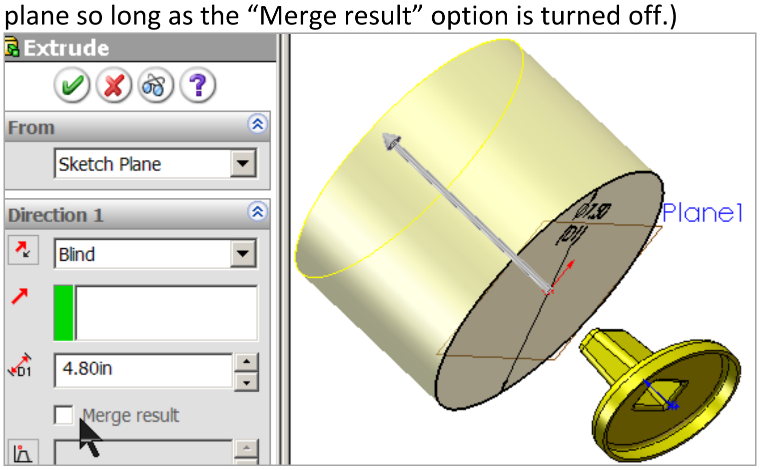
3. Right click on the plane name to insert a sketch on it.



4. Select a Normal to view and sketch a circle centered on the stems of the implant and cement.



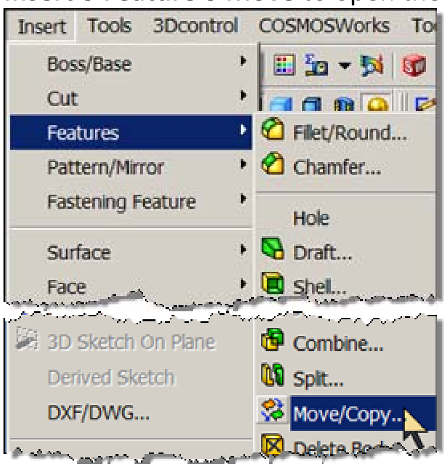
5. Extrude the circular area to form the bone model and to position it for the Boolean cut will be used to represent the effect of the bone reamer. (This operation could have been done without the reference plane so long as the "Merge result" option is turned off.)



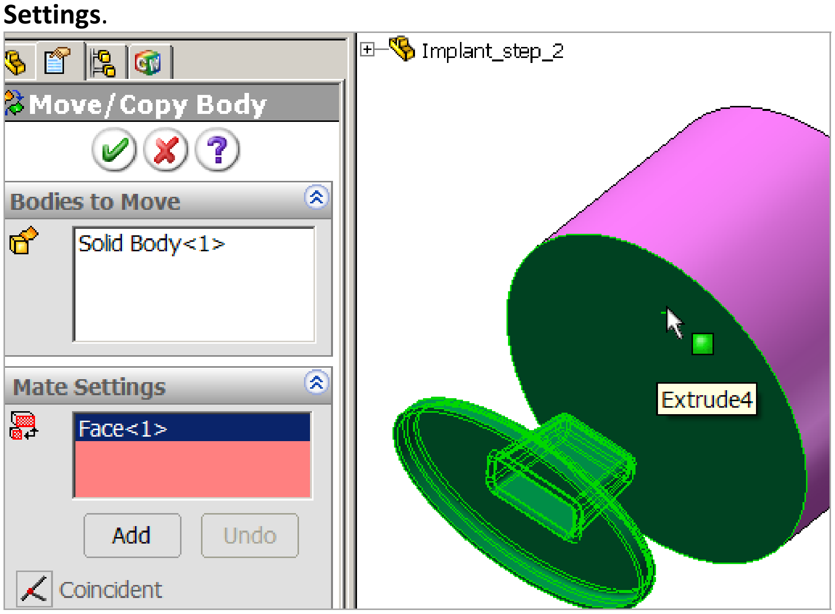
4.3.5 Prepare to remove the cement volume from the bone

To prepare to ream the necessary hole in the bone you must "Move" the reamer flat face to mate with bone flat face:

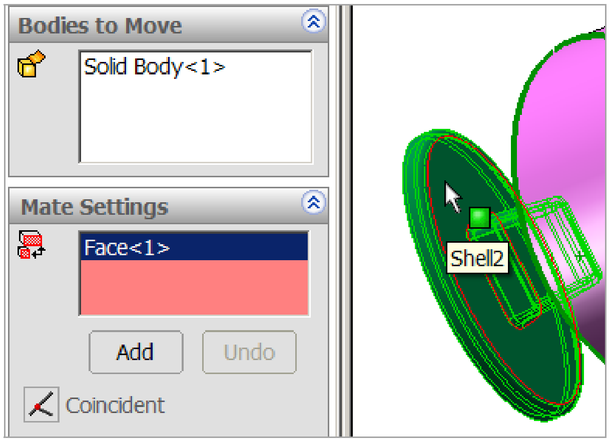
1. **Insert→Feature→Move** to open the **Move/Copy panel.**

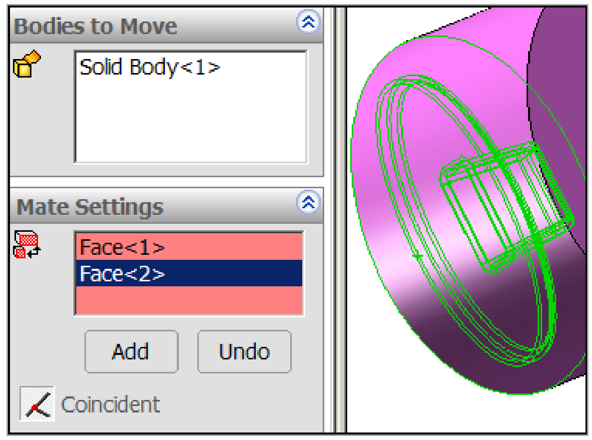


2. Select the cement as the **Body to Move**, and the back oval face of the cement as the first **Mate Settings**.



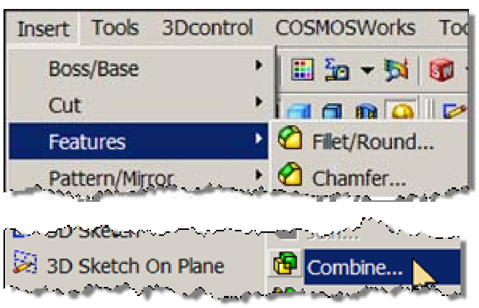
3. Select the front of the bone cylinder as the second mating face and mark them as **Coincident** to position of the reamer within the bone. (Actually, the first cement face select should have selected the front face of the cement, so that the new artificial joint face would match the original natural bearing face location.)

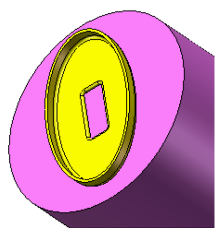




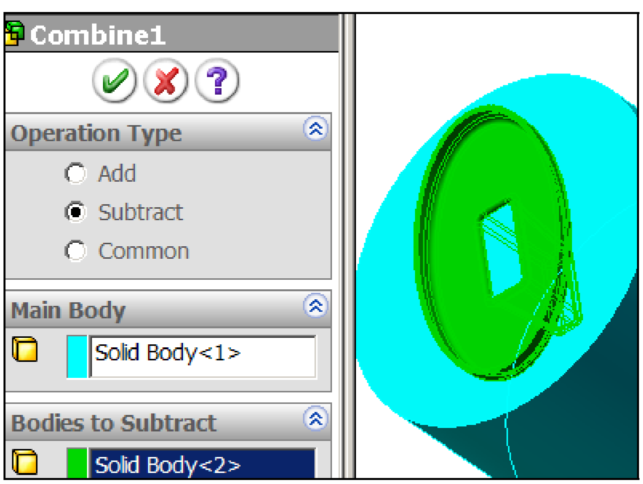
Finally, you are ready to ream the bone and complete the third part that will be needed for the assembled stress analysis. Surprisingly, neither SolidWorks nor CosmosWorks help files have a reference to the Boolean constructive solid geometry operations of union, intersection, and difference. Instead, they call those the "**Combine**" operations and name them, **Add**, **Common**, and **Subtraction**, respectively. Since the Subtraction (difference) operation is non-commutative you must use care in which of the two bodies you select first.

1. Begin with another view of the reamer part relative to the original bone part. Select **Insert→Features →Combine** to access the Boolean operators in the **Combine panel**.

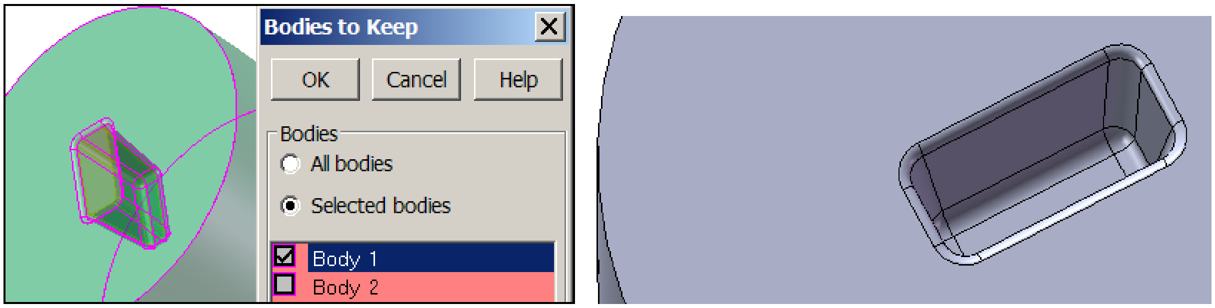




2. Set the **Operation Type** to **Subtract**, select the bone as the **Main Body** to be cut. Select the cement (reamer) as the **Body to Subtract**. Click **OK**.



3. Upon completion of the Boolean subtract operation check **Body1**, in the **Bodies to Keep panel**, as the third part to be named and saved for the assembly.

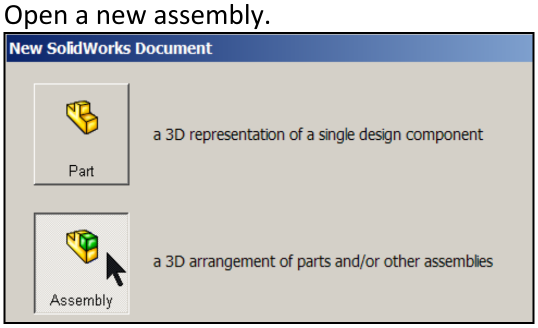


Now the three parts need to be joined to form an assembly to carry out a finite element study are available. In reality, the bone cement is mixed and placed in the reamed bone hole and the implant is pushed in to force out the excess cement. The mixed cement chemically reacts and generates a very large amount of heat (heat power in Cosmos terminology) and can cause temperatures above the boiling point of water (and blood) if the thickness is too large. A thermal study should be also done, but only a stress analysis will be shown here.

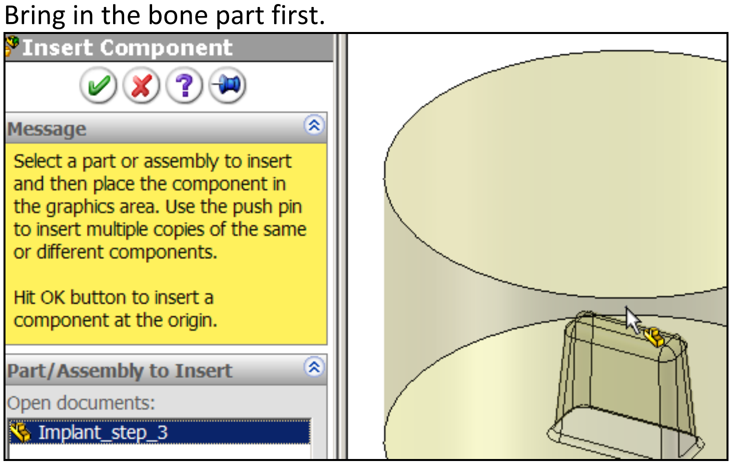
4.3.6 Build an assembly for stress analysis

To conduct a stress analysis, or thermal analysis, of the interaction of these three parts you must construct an assembly containing all three parts:

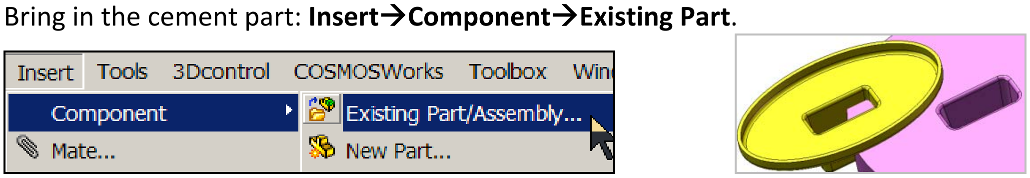
1. Open a new assembly.



2. Bring in the bone part first.

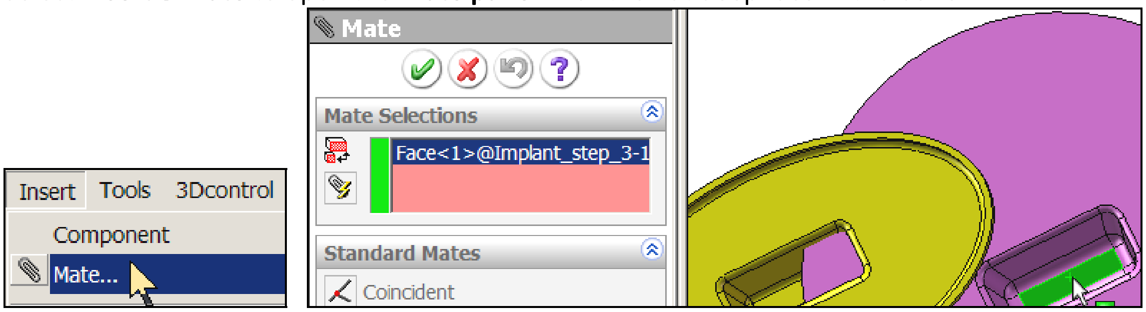


3. Bring in the cement part: **Insert→Component→Existing Part**.

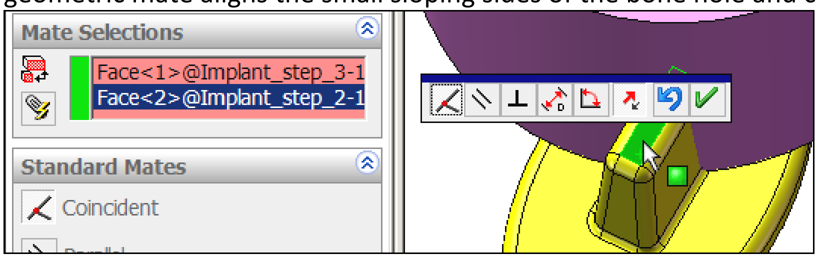


Arrange the cement part so that it properly mates to the bone in a geometric sense:

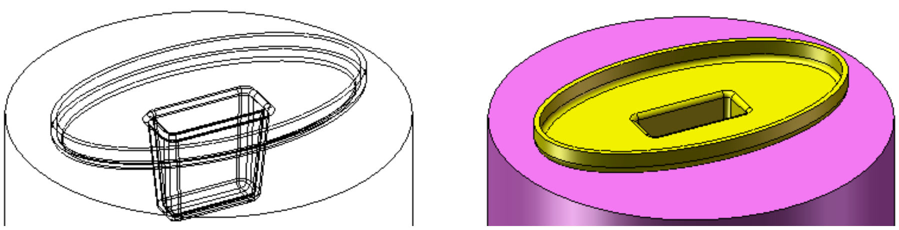
1. Select **Insert→Mate** to open the **Mate panel**. Pick the flat deep face in the bone.



2. Pick the mating small flat face of the cement, check **Coincident**, click **OK**. Then mate the widest interior sloping face of the hole to the corresponding outer face of the cement. The third and final geometric mate aligns the small sloping sides of the bone hole and cement parts.



3. Verify that these two parts are in the correct relative geometric positions.

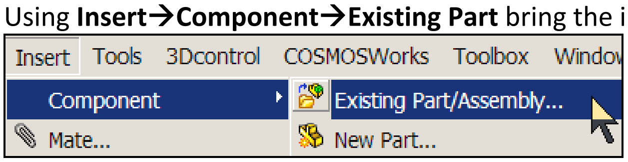


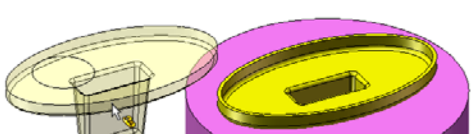
4. To be safe it is wise to save this (or any) intermediate assembly.

Be alerted here that just because a geometric mate exists in SolidWorks does not mean that they will be connected in Cosmos. In Cosmos they might be bonded together, or they may be allowed to move so that gaps open between them, etc. As you will see later, you need to consciously select the physically correct interaction between such surfaces in the "Contact/Gaps" option in CosmosWorks.

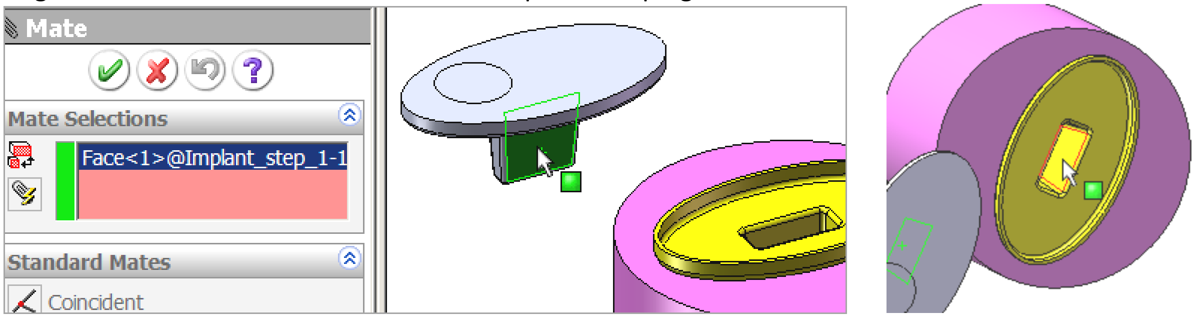
Now, add the implant solid to the assembly:

1. Using **Insert→Component→Existing Part** bring the implant into the assembly.

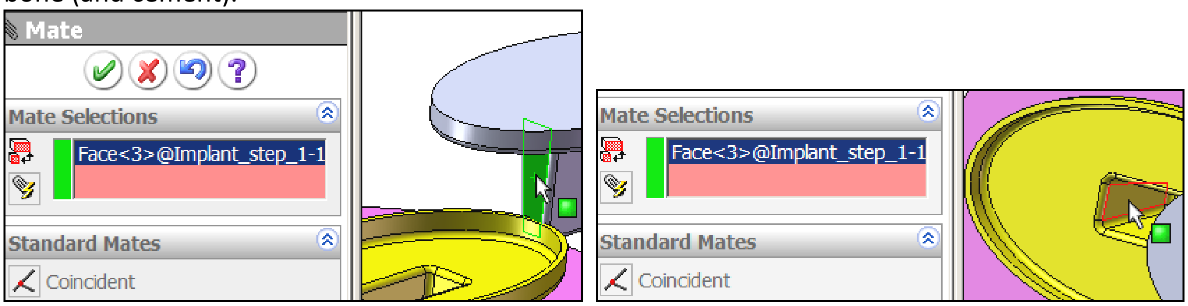




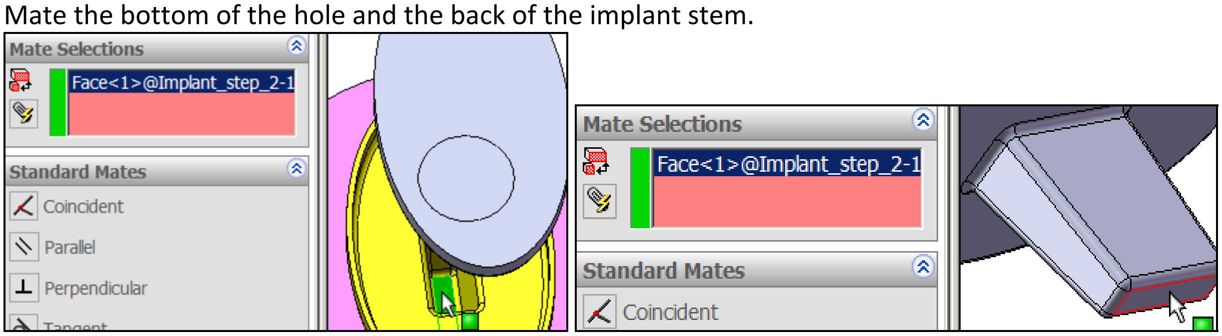
2. Begin to mate three of its surfaces. Mate a pair of sloping faces.



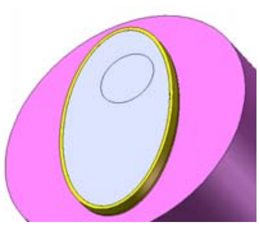
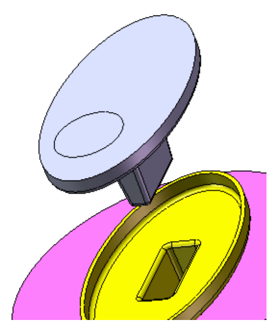
3. Mate the other pair of sloping faces. This centers the implant directly above the center hole of the bone (and cement).



4. Mate the bottom of the hole and the back of the implant stem.



5. This completes the placement of the three structural parts in their relative positions. Now you are ready to leave SolidWorks.

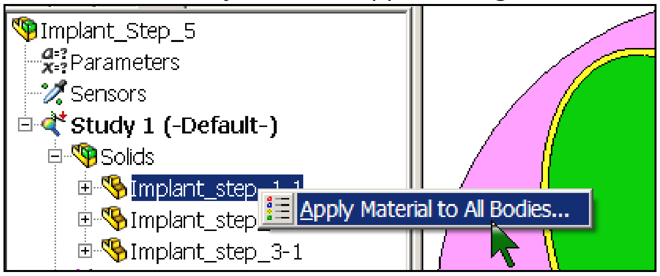



4.3.7 Begin the assembly stress analysis

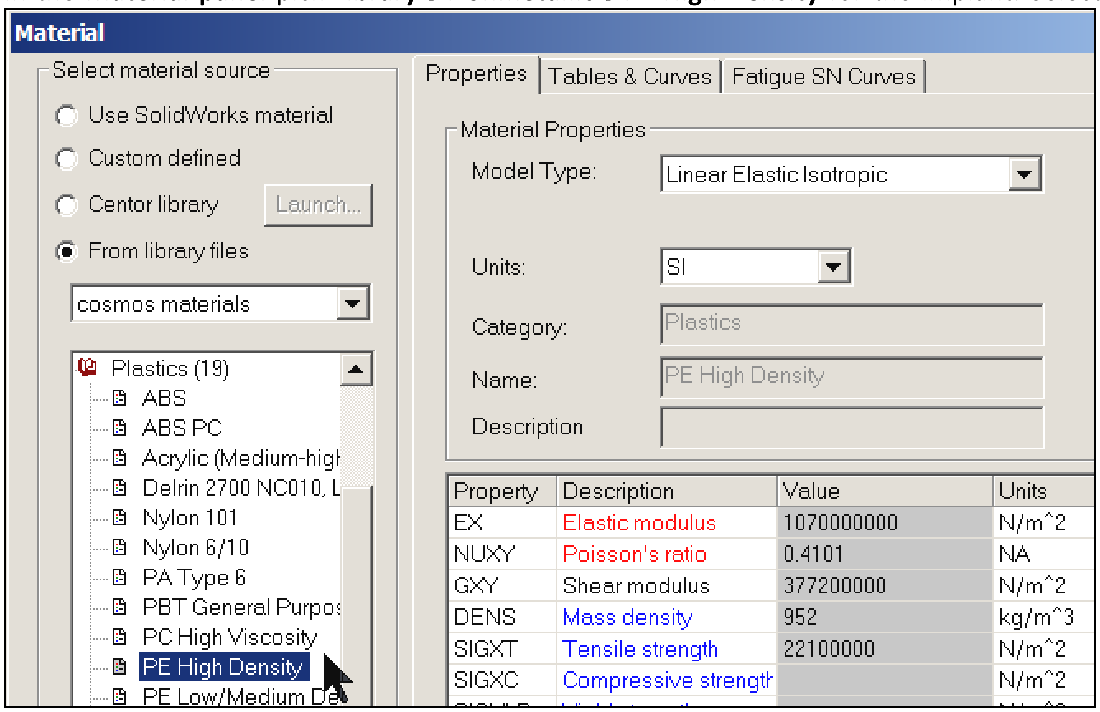
Save the bone/cement/implant completed assembly. Select the Cosmos icon to begin the assembly stress analysis. In CosmosWorks:

1. Right click on the **Part_name→Study, Name** the part (bone_implant here), pick **Static** analysis, and for the **Model Type** pick solid.

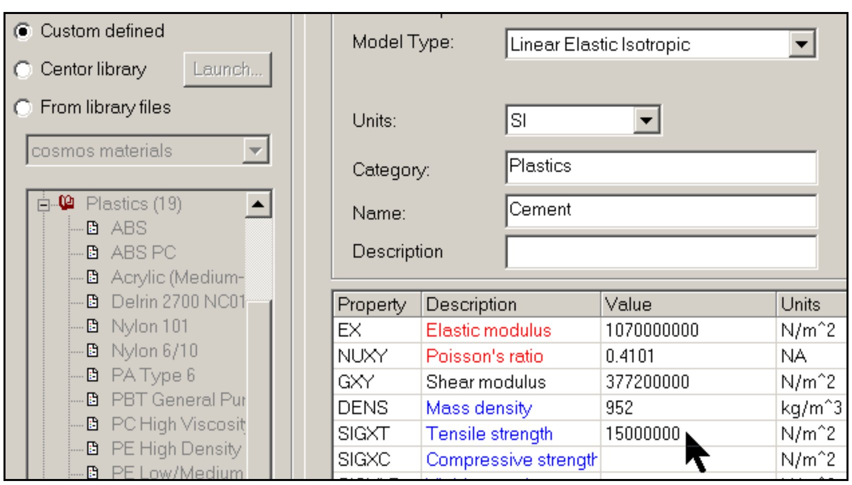
2. When the **Study Menu** appears right click on **Solids→Implant_Step_1→Apply Materials.**



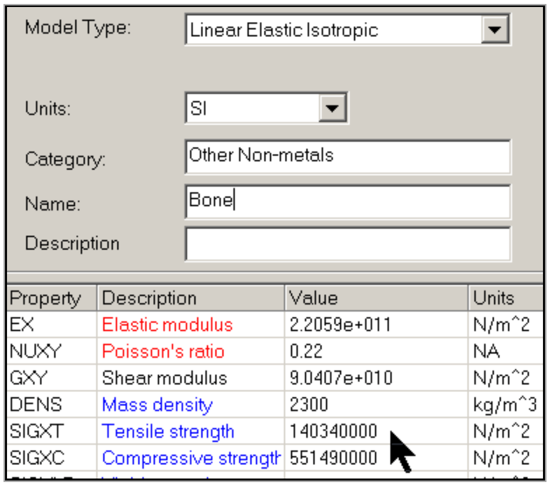
3. In the **Material panel** pick **Library→Nonmetallic→PE High Density** for the implant. Select **Units→SI**.



4. For the bone cement in the **Material panel** pick **Custom defined**. Enter the values for the elastic modulus, Poisson ratio, and tensile strength.



5. For the bone in the **Material panel** pick **Custom defined**. Enter the elastic modulus, Poisson ratio, and tensile strength. (Custom compressive strengths should also be given for the cement and bone if those data are available.)



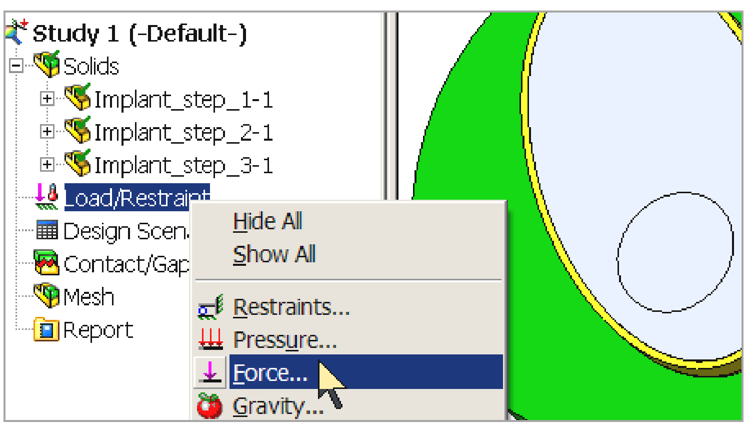
4.3.8 Set the material interface relations

Having defined each of the three material region properties you should next declare how their interfaces displace with respect to each other. The previous geometric matting of the surfaces just made them geometrically adjacent. They are not yet structurally connected. In the Cosmos assembly you will see that the **Contact/Gaps** icon is located above the usual Mesh icon location. That icon must be used to bond the touching faces: Right click **Contact/Gaps**, turn on **Touching Faces Bonded**.

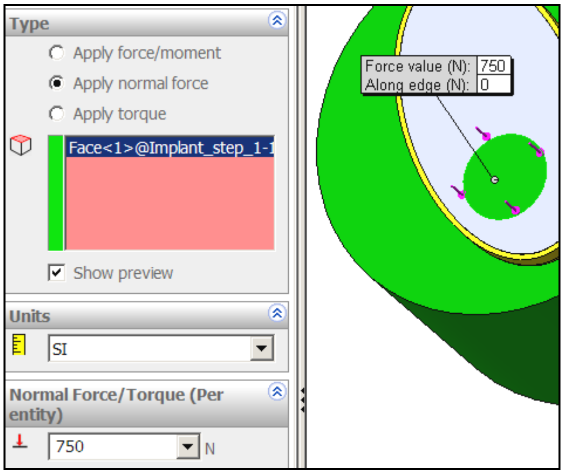
That selection means that the materials are structurally bonded together and do not just look like their displacements will agree on their touching surfaces. Here you intended for the surfaces to be literally glued together so this is the correct type of material interactions. However, that is not always the case so you should get in the habit of always visiting the Contact/Gaps option and think about how each pair of geometrically mated surfaces are expected to act in a structural (or thermal) sense.

Next you will specify the force and restraints on the implant:

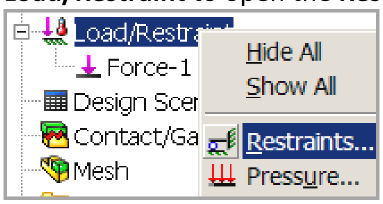
1. The free face force is expected to come from the ball joint on the arm. A biomechanical analysis of the muscles and ligaments results in a load of 750 N. Select **Load/Restraints→Force**.



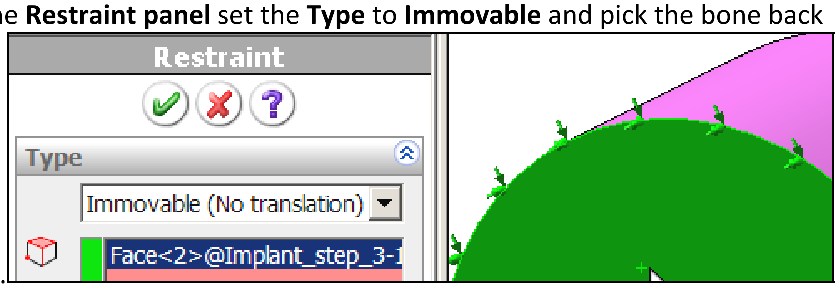
2. Select the circular area on the implant face and apply a pressure giving a resultant force of 750 N.



3. The back of the bone is initially assumed to be fully supported by the surrounding bone. Select **Load/Restraint** to open the **Restraint panel**.



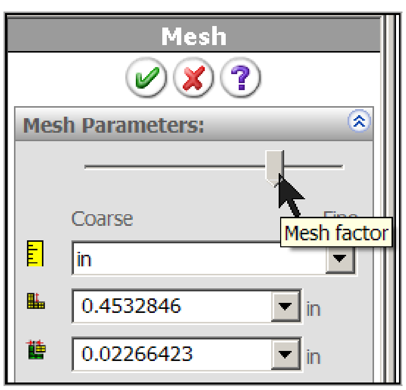
4. In the **Restraint panel** set the **Type** to **Immovable** and pick the bone back face.

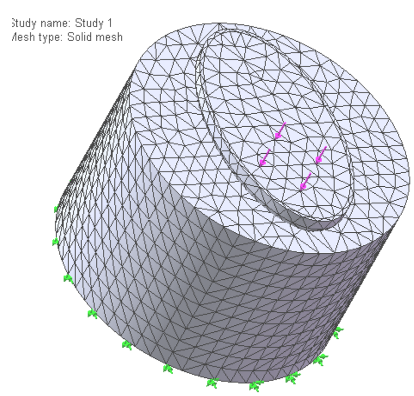


4.3.9 Mesh preview

For a quick initial study you can accept the default mesh from the mesh generator. However, you should expect high displacement and stresses within the contact circle and use the mesh control options to refine that region in the next study. To generate the default initial mesh, without mesh controls:

1. Right click **Mesh→Create**.





2. Now select **Study Name→Run** to carry out the displacement calculations and their post-processing.

4.3.10 Viewing selected results

The post-processing usually begins by checking the reactions and viewing the displacements of the assembly. The displacements, seen in Figure 4-9, are quite localized. The corresponding external and internal von Mises stresses, on a plane through the minor axis of the implant component, are seen in Figure 4-10. The initial interest is focused on the stress levels at the implant/cement interface as shown in the major axis slice given in

Figure 4-11**.** To better illustrate those stresses, the implant is shown exploded away from the cement in Figure 4-12. It is also wise to review the maximum shear stress at that interface. It is seen in Figure 4-13.

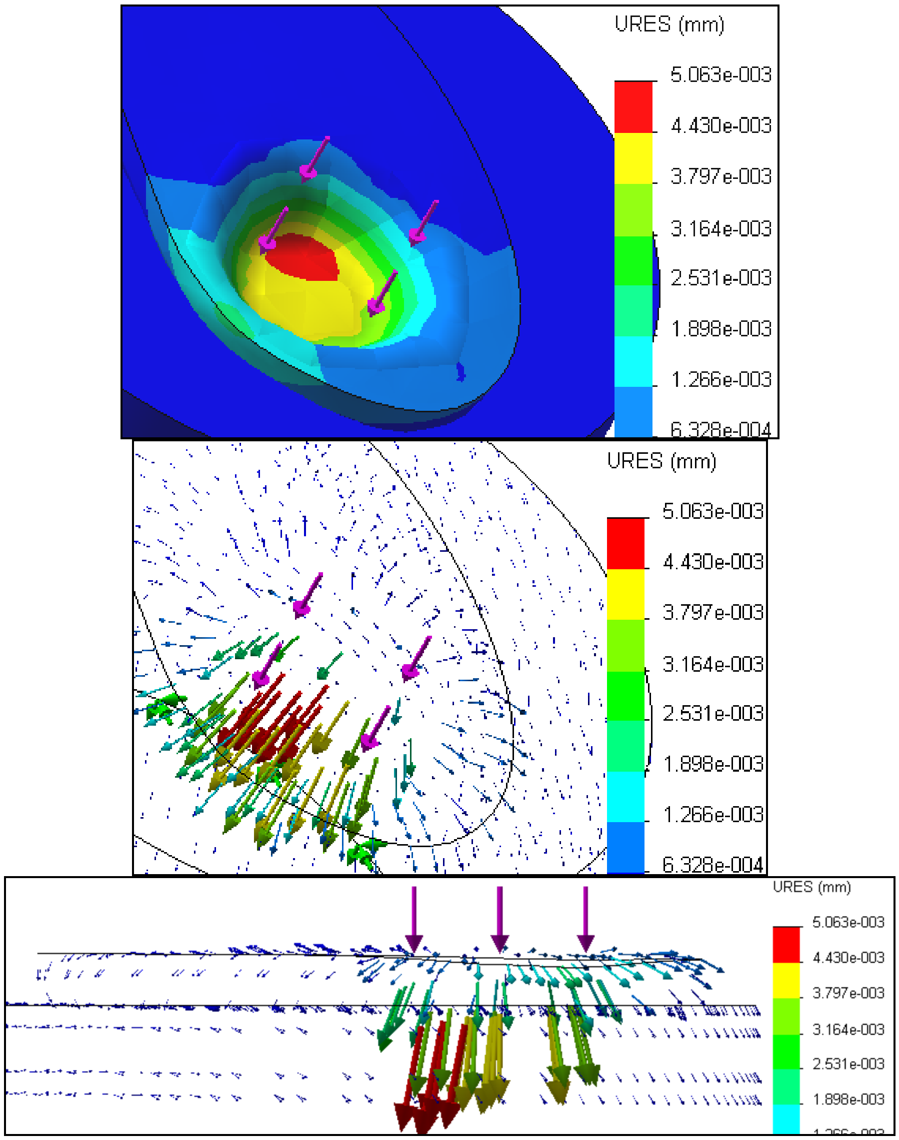


Figure 4-9 Local displacements near the contact region

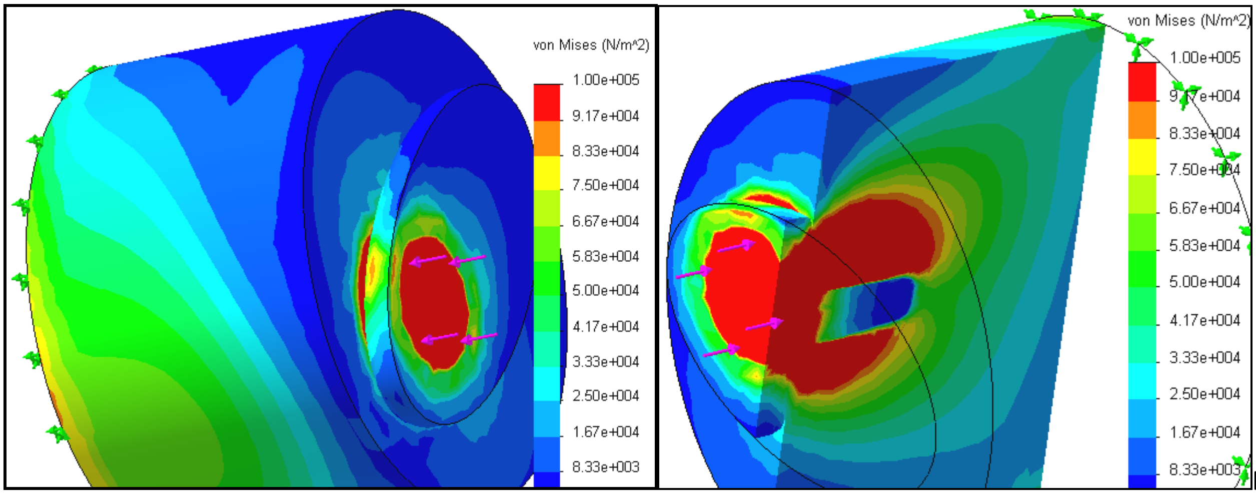


Figure 4-10 External and internal stresses along the minor axis plane

For this problem you would want to examine additional slices through the assembly to view the stress results. You should also hide the implant and bone parts so that you can see the stresses on both the inside and outside of the cement component.

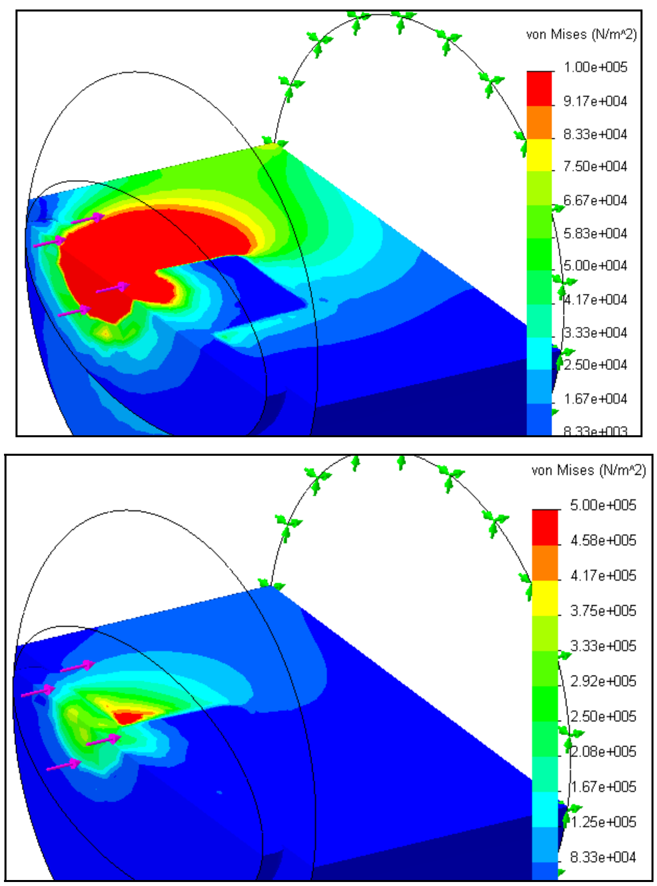


Figure 4-11 Von Mises stress on a plane through the implant major axis

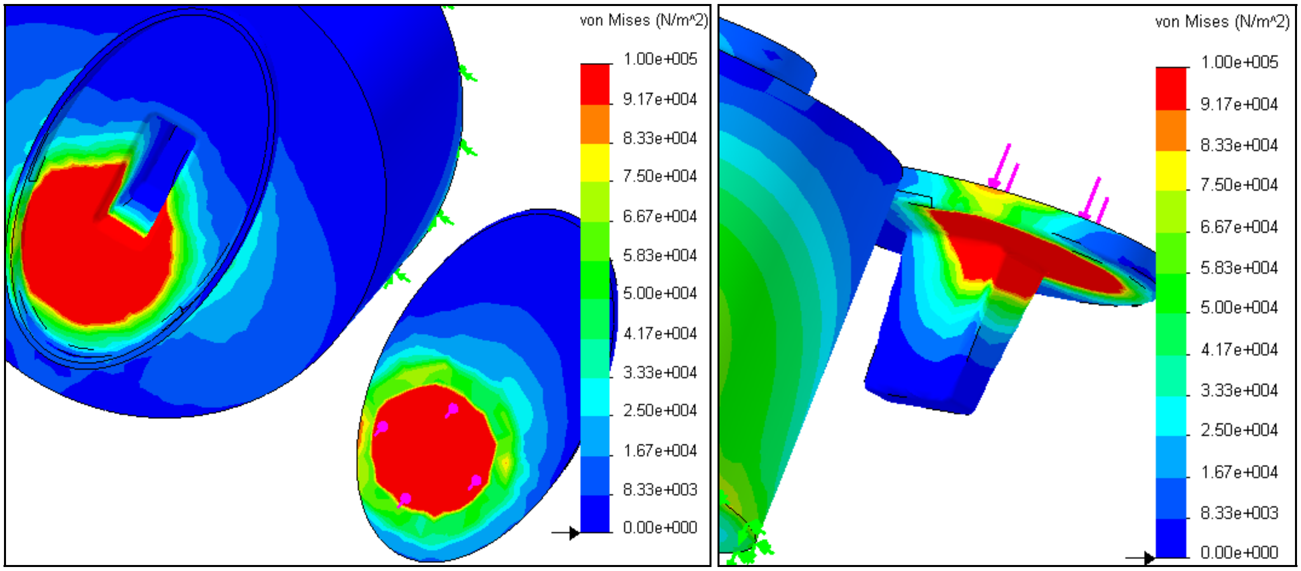


Figure 4-12 An exploded view of the implant/cement stress

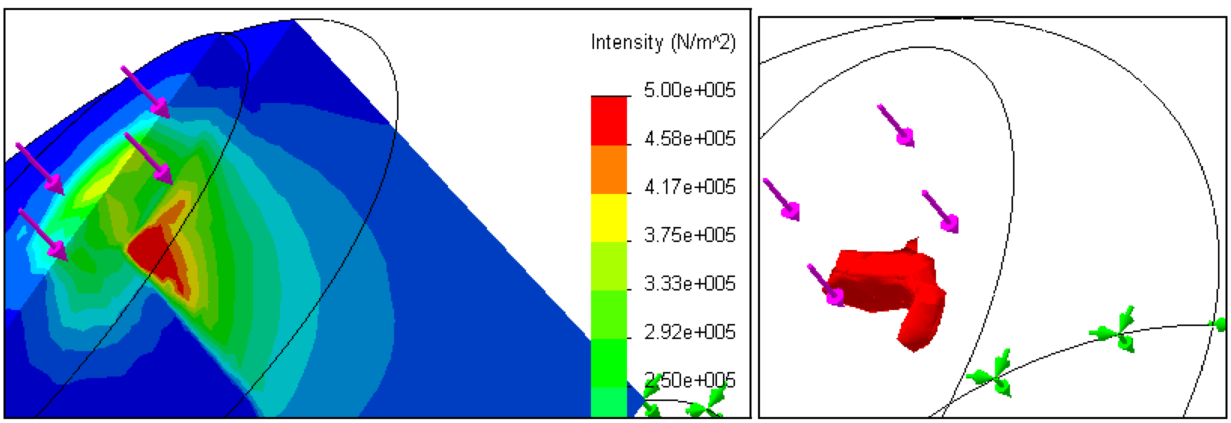


Figure 4-13 The intensity (twice the maximum shear stress)

5 Plane Stress Analysis

5.1 Introduction

Generally you will be forced to utilize the solid elements in CosmosWorks due to a complicated solid geometry. To learn how to utilize local mesh control for the solid elements it is useful to review some two-dimensional (2D) problems employing the triangular elements. Historically, 2D analytic applications were developed to represent, or bound, some classic solid objects. Those special cases include plane stress analysis, plane strain analysis, axisymmetric analysis, flat plate analysis, and general shell analysis. After completing the following 2D approximation you should go back and solve the much larger 3D version of the problem and verify that you get essentially the same results for both the stresses and deflections.

Plane stress analysis is the 2D stress state that is usually covered in undergraduate courses on mechanics of materials. It is based on a thin flat object that is loaded, and supported in a single flat plane. The stresses normal to the plane are zero (but not the strain). There are two normal stresses and one shear stress component at each point (σ_x, σ_y, and τ). The displacement vector has two translational components (*u_x*, and *u_y*). Therefore, any load (point, line, or area) has two corresponding components.

The CosmosWorks "shell" elements can be used for plane stress analysis. However, only their in-plane, or "membrane", behavior is utilized. That means that only the elements in-plane displacements are active and available to be restrained. To create such a study you need to construct the 2D shape and extrude it with a constant thickness that is small compared to the other two dimensions of the part. Then begin a mid-surface shell study.

Before solid elements became easy to generate it was not unusual to model some shapes as 2.5D. That is, they were plane stress in nature but had regions of different constant thickness. This concept can be useful in validating the results of a solid study if you have no analytic approximation to use. Since the mid-surface shells extract their thickness automatically from the solid body you should use a mid-plane extrude when you are building such a part.

One use of a plane stress model here is to illustrate the number of elements that are needed through the depth of a region, which is mainly in a state of bending, in order to capture a good approximation of the flexural stresses. Elementary beam theory and 2D elasticity theory both show that the longitudinal normal stress (σ_x) varies linearly through the depth. For pure bending it is tension at one depth extreme, compression at the other, and zero at the center of its depth (also known as the neutral-axis). When the bending is due, in part, to a transverse force then the shear stress (τ) is maximum at the neutral axis and zero at the top and bottom fibers. For a rectangular cross-section the shear stress varies parabolically through the depth. Since the element stresses are discontinuous at their interfaces, you will need at least three of the quadratic (6 node) triangles, or about five of the linear (3 node) triangles to get a reasonable spatial approximation of the parabolic shear stress. This concept should guide you in applying mesh control through the depth of a region you expect, or find, to be in a state of bending.

5.2 Simply supported beam, Load case 1

This will be illustrated with a simple rectangular beam plane stress analysis. Consider a beam of rectangular cross-section with a thickness of *t = 2 cm*, a depth of *h = 10 cm*, and a length of *L = 100 cm*. Let a uniformly distributed downward vertical load of *w = 100 N/cm* be applied at its top surface and let both ends be *simply supported* (i.e., have *u_y = 0* at the neutral axis) by a roller support. In addition, both ends are subjected to equal moments that each displaces the beam center downwards. The end moment has a value of *M = 1.25e3*

N-m. The material is aluminum 1060. This is a problem where the stresses depend only on the geometry. However, the deflections always depend on the material type.

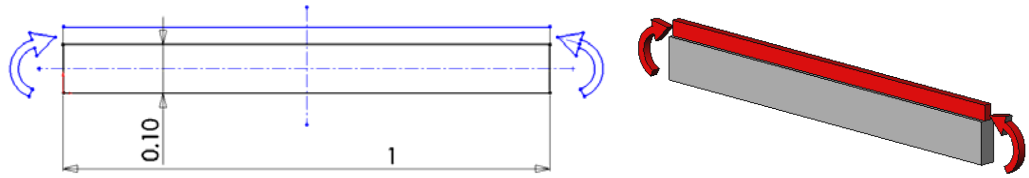


Figure 5-1   A simply supported beam with line load and end moments

It should be clear that this problem is symmetrical about the vertical centerline (why that is true will be explained shortly should it not be clear). Therefore, no more than half the beam needs to be considered (and half the load). Select the right half. The beam theory results should suggest that an even more simplified model would be valid due to anti-symmetry (if we assume half the line load acts on both the top and bottom faces). The 3D flat face symmetry restraint was described earlier. The 2D nature of this example provides insight into how to identify lines (or planes in 3D) of symmetry and anti-symmetry, as shown in Figure 5-2.

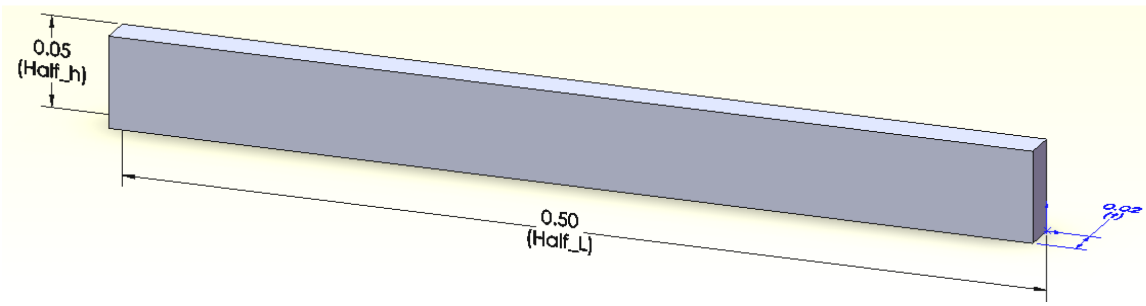


Figure 5-2 One-quarter of the beam

5.2.1   Symmetry and anti-symmetry restraints

A process for identifying displacement restraints on planes of symmetry and anti symmetry will be outlined here. Assume that the horizontal center line of the beam corresponds to the

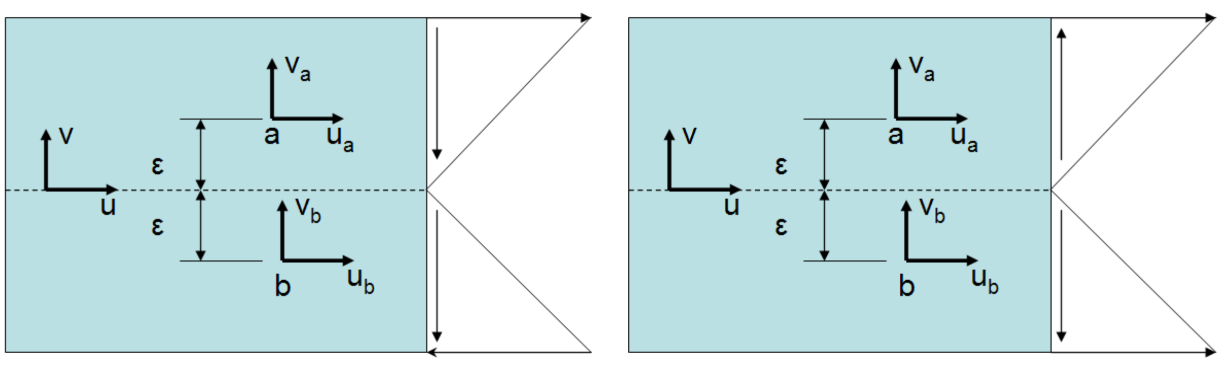


Figure 5-3 Anti-symmetric (u=0, v=?), and symmetric (u=?, v=0) displacement states

dashed centerline of the anti-symmetric image at the left in Figure 5-3. The question is, what, if any, restraint should be applied to the *u* or *v* displacement component on that line. To resolve that question imagine two mirror image points, *a* and *b*, each a distance, *ε*, above and below the dashed line. Note that both the upper and lower half portions are loaded downward in an identical fashion, and they have the same horizontal end supports, . Therefore, you expect v_a and v_b to be equal, but have an unknown value (say $v_a = v_b = ?$). Likewise, the horizontal load application is equal in magnitude, but of opposite sign in the upper and lower regions.

Therefore, you expect $u_b = -u_a$. Now let the distance between the points go to zero ($\varepsilon \rightarrow 0$). The limit gives $v = v_a = v_b = ?$, so v is unknown and no restraint is applied to it. The limit on the horizontal displacement gives $u = u_b = -u_a \rightarrow 0$, so the horizontal displacement can be restrained to zero if you with to use a half depth anti-symmetric model. Another way to say that is: ***on a line of anti-symmetry the tangential displacement component is restrained to zero***.

The vertical centerline symmetry can be justified in a similar way. Imagine that the right image in Figure 5-3 is rotated 90 degrees clockwise so the dashed line is parallel to the beam vertical symmetry line. Now u represents the displacement component tangent to the beam centerline (i.e., vertical). The vertical loading on both sides is the same, as are the vertical end supports, so the vertical motion at a and b will be the same (say $u_a = u_b = ?$). In the limit, as the two points approach each other $u = u_a = u_b = ?$, so the beam vertical centerline has an unknown tangential displacement and is not subject to a restraint. Now consider the displacement normal to the beam vertical centerline (here v). At any specified depth, the loadings and deflections in that direction are equal and opposite. Therefore, in the limit as the two points approach each other $u = u_b = -u_a \rightarrow 0$, so the displacement component normal to the beam vertical centerline must vanish. Another way to state that is: ***on a line of symmetry the normal displacement component is restrained to zero.***

5.2.2 Part loadings

From the above arguments, the 2D approximation can be reduced to one-quarter of the original domain. The other material is removed and replaced by the restraints that they impose on the portion that remains. Now your attention can focus on the applied load states. The top (and bottom) line load can be replaced either with a total force on the top surface, or an equivalent pressure on the top surface, since CosmosWorks does not offer a load-per-unit-length option. Unfortunately, either requires a hand calculation that might introduce an error. The less obvious question is how to apply the end moment(s).

Since the general shell element has been force to lie in a flat plane, and have no loads normal to the plane, its two in-plane rotational dof will be identically zero. However, the nodal rotations normal to the plane are still active (in the literature they are call drilling freedoms in 2D studies). That may make you think that you could apply a moment, M_z, at a node on the neutral axis of each end of the beam. In theory, that should be possible, but in practice it works poorly (try it) and the end moment should be applied in a different fashion. One easy way to apply a moment is to form a couple by applying equal positive and negative triangular pressures across the depth of the ends of the beam. That approach works equally well for 3D solids that do not have rotational degrees of freedom.

The maximum required pressure is related to the desired moment by simple static equilibrium. The resultant horizontal force for a linear pressure variation from zero to p_{max} is

$F = A\, p_{max} / 2$, where A is the corresponding rectangular area, $A = t\,(h/2)$, so $F = t\, h\, p_{max} / 4$. That resultant force occurs at the centroid of the pressure loading, so its lever arm with respect to the neutral axis is $d = 2(h/2)/3 = h/3$ (for the top and bottom portions). The pair of equal and opposite forces form a combined couple of $M_z = F\,(2d) = t\, h^2\, p_{max} / 6$. Finally, the required maximum pressure is

$$\boldsymbol{p_{max} = 6\, M_z / t\, h^2.}$$

To apply this pressure distribution in CosmosWorks you must define a local coordinate system located at the neutral axis of the beam and use it to define a variable pressure. However, the CosmosWorks nonuniform pressure data requires a pressure scale, $\boldsymbol{p_{scale}}$, times a non-dimensional function of a selected local coordinate system. Here you will assume a pressure load linearly varying with local y placed at the neutral axis: $\boldsymbol{p}\,(y) = \boldsymbol{p_{scale}} * y$ (with y non-dimensional). This must match $\boldsymbol{p_{max}}$ at y = h / 2, so

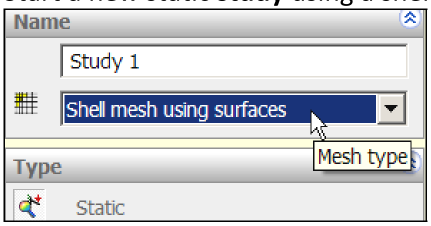
$$\boldsymbol{p_{scale} = 2\, p_{max} / h = 12\, M_z / t\, h^3.}$$

Since it is often necessary to apply moments to solids in this fashion this moment loading will be checked independently against beam theory estimates before applying the line load. Here, $p_{max} = 3.75e7\ N/m^2$, $p_{scale} = 7.5e8$.

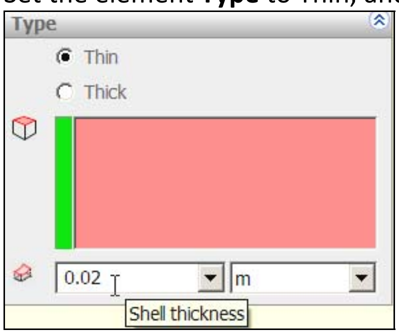
5.2.3 CosmosWorks plane stress model

The beam theory solution for a simply supported beam with a uniform load is well known, as is the solution for the loading by two end moments (called pure bending). In both cases the maximum deflection occurs at the beam mid-span. The two values are $v_{max} = 5\ w\ L^4 / 384\ EI$, and $v_{max} = M\ L^2 / 8\ EI$, respectively. Here the centerline deflection due only to the end moment is $v_{max} = 1.36e\text{-}3\ m$. For a linear analysis and the sum of these two values can be used to validate the centerline deflection. Next, the one-quarter model, shown in Figure 5-2, will be built, restrained, and loaded:

1. Start a new static **study** using a shell mesh.

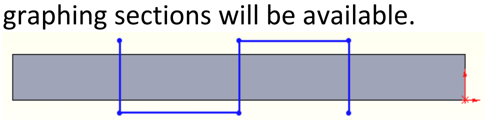


2. **Shells→Defined by Selected Surfaces**. Select the front face to open the Shell Definition panel.
3. Set the element **Type** to Thin, and the **Shell Thickness** to be *0.02 m*.

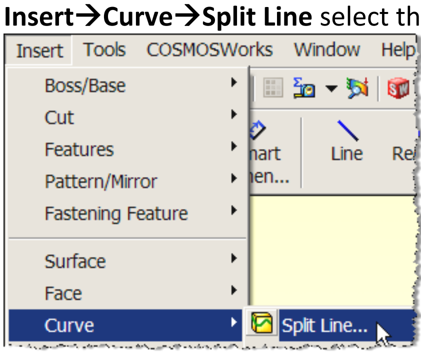


Since the stresses through the depth are going to be examined here, you should plan ahead and insert some split lines on the front surface to be used to list and/or graph selected stress and deflection components:

1. Right click on the front face, **Insert Sketch**.
2. Insert a **line** segment that crosses the face at the interior quarter points. Including the end lines, five graphing sections will be available.



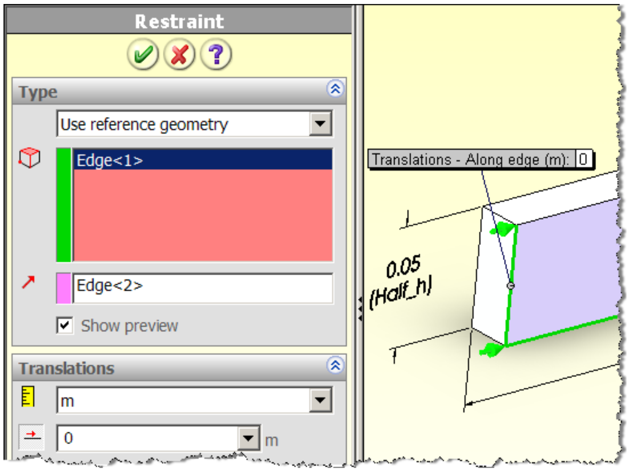
3. **Insert→Curve→Split Line** select the body faces, click **OK.**



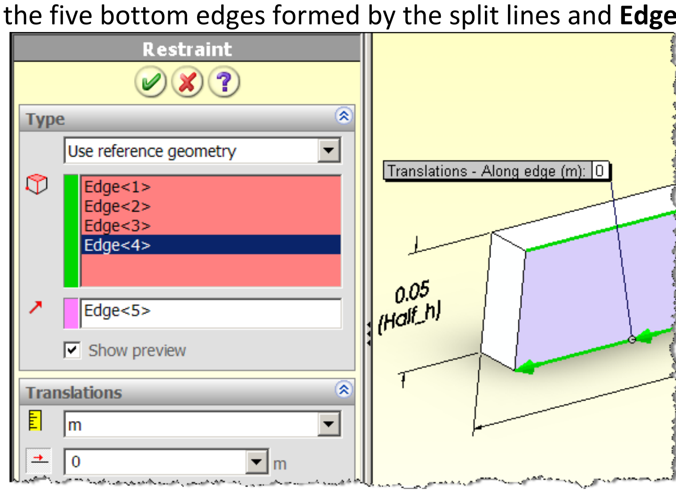
5.2.4 Edge restraints and loads

Remember that shells defined by selected surfaces must have their restraints and loads applied directly to the edges of the selected surface. First the symmetry and anti-symmetry restraints will be applied. Since the shell mesh will be flat it is easy to use its edges to define directions for loads, or restraints:

1. **Load/Restraint→Restraints** opens the **Restraint panel**.
2. The zero horizontal (*x*) deflection is applied as a symmetry condition on the edge corresponding to the beam vertical centerline; **Use reference geometry**, select **Edge1** to restrain and **Edge5** for the direction.



3. Apply the anti-symmetry condition along the edge of the neutral axis. **Use reference geometry**, select the five bottom edges formed by the split lines and **Edge5** for the direction.

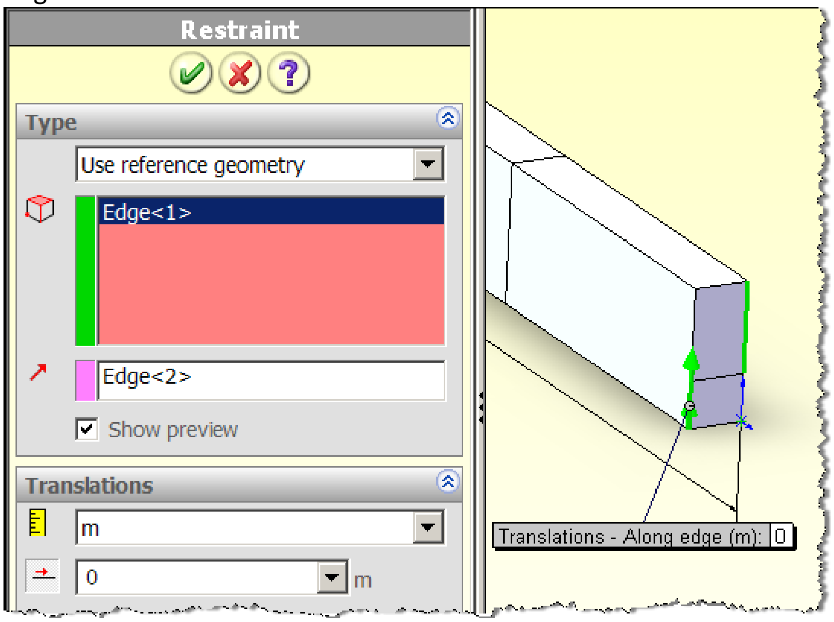


At the simply supported end it is necessary to assume how that support will be accomplished. Beam theory treats it as a point support, but in 2D or 3D that causes a false infinite stress at the point. Another split line was introduced so about one-third of that end could be picked to provide the vertical restraint required. This serves as a reminder that where, and how, parts are restrained is an assumption. So it is wise to investigate more than one such assumption. Software tutorials are intended to illustrate specific features of the software, and usually do not have the space for, or intention of, presenting the best engineering judgment. Immovable restraints are often used in tutorials, but they are unusual in real applications.

Apply the vertical end restraint:

1. **Load/Restraint→Restraints** opens the **Restraint panel**.

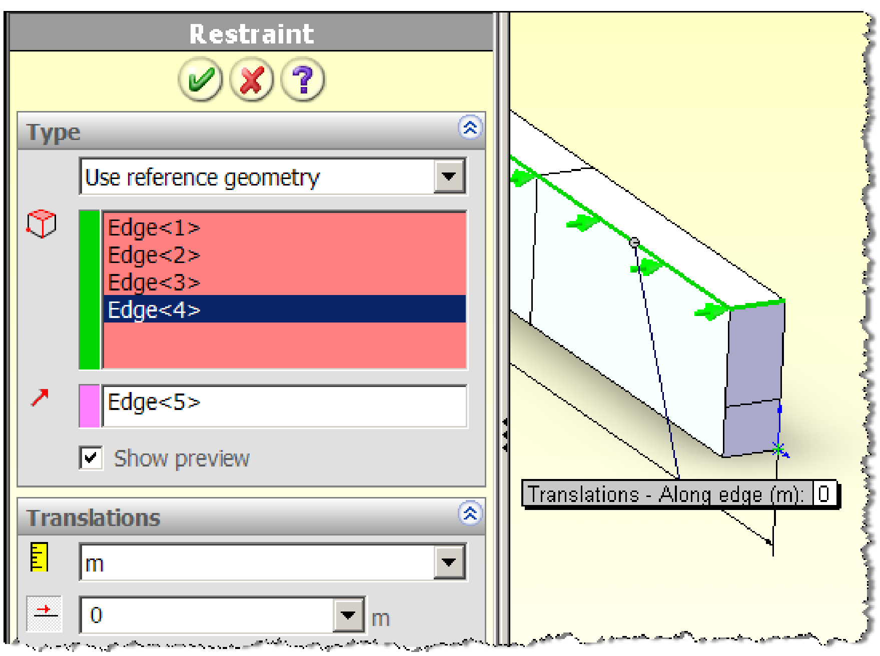
2. **Use reference geometry**, select the lower front vertical edge line to restrain, and the back vertical edge as the direction.



5.2.5 Rigid body motion restraint

Since a general shell element is being used in a plane stress application it still has the ability to translate normal to its plane and to rotate about the in-plane axes (*x* and *y*). Those three rigid body motions must also be eliminated in any plane stress analysis:

1. **Load/Restraint→Restraints** opens the **Restraint panel**.
2. **Use reference geometry**, select the five segments of the top (or bottom) edge of the beam to restrain. Zero the *z*-translation by picking the edge parallel to the z-axis as the direction. The combination of all the plane stress supports and the rigid body motion restraint is seen in Figure 5-4.



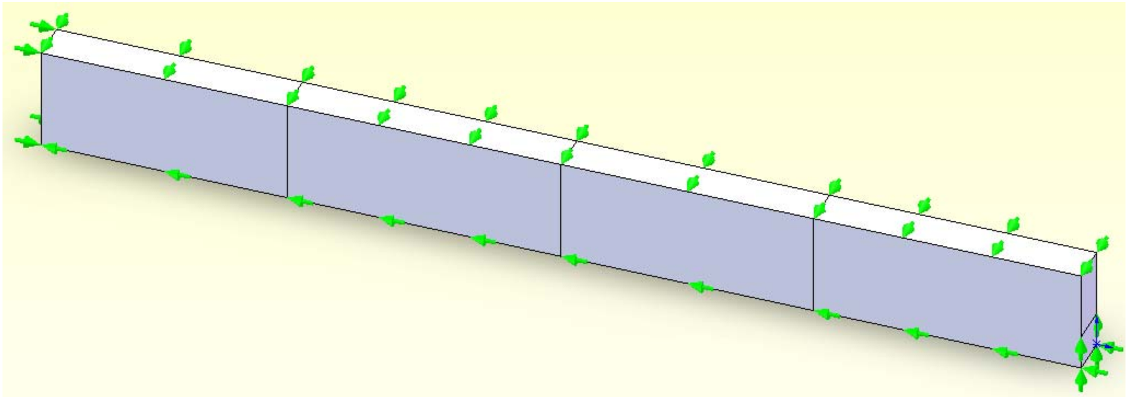
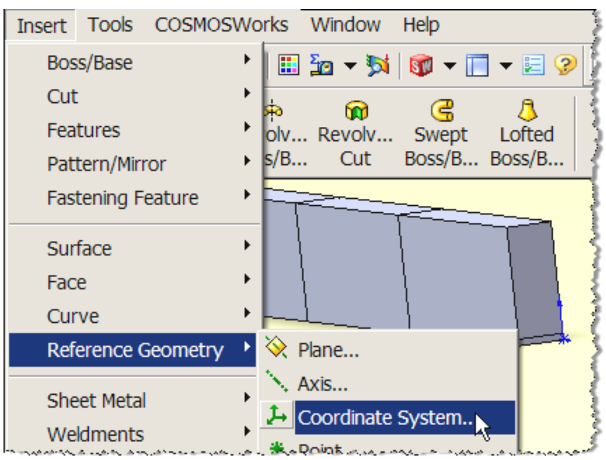


Figure 5-4 The active symmetric, anti-symmetric, and RBM displacement restraints

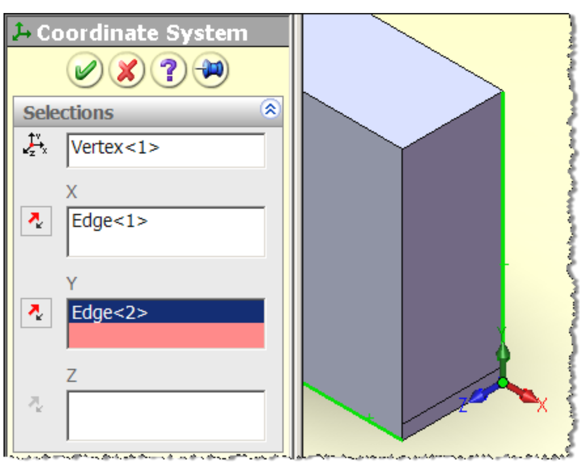
5.2.6 Moment application as a nonuniform pressure

A linear variation of equal and opposite pressures, relative to the neutral axis, can be used to apply a statically equivalent moment to a continuum body that does not have rotational degrees of freedom. It also has the side benefit of matching the normal stress distribution in a beam subjected to a state of pure bending. That usually requires the user to define a local coordinate system at the axis about which the moment acts. In this case, it must be located at the neutral axis of the beam:

1. Select **Insert→Reference Geometry→Coordinate System** to open the **Coordinate System panel**.



2. Right click one end of the neutral axis to set the **origin** of Coordinate System 1.



3. If the *y*-axis is vertical click **OK**, else pick part edges to orientate the *y*-axis.

The application of the non-uniform pressure is applied at the front vertical edge at the simple support in the **Pressure panel** of Figure 5-5. A unit **pressure value** is used to set the units and the magnitude is defined by multiplying that value by a non-dimensional polynomial of the spatial coordinates of a point, relative to local **Coordinate System 1** defined above.

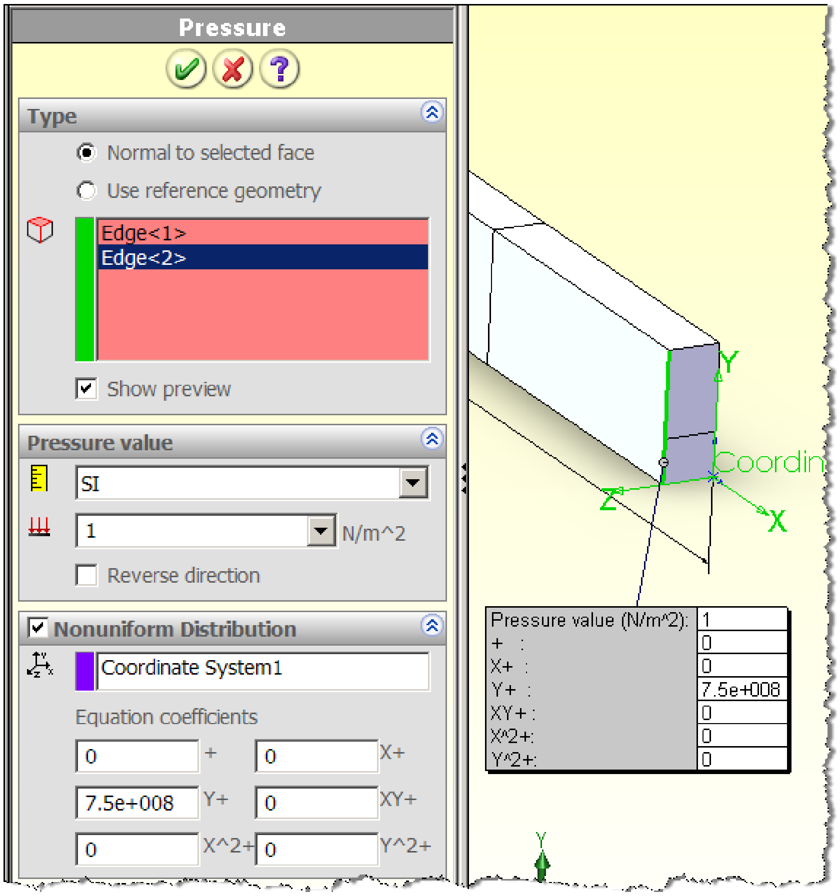


Figure 5-5 Applying the required linear pressure

5.2.7 Mesh and run the study

Having completed the restraints and loads the default names in the manager menu have been changed (by slow double clicks) to reflect what they are intended to accomplish (left of Figure 5-6). Also, the mesh has been designed to be crude so as to illustrate how mesh control is needed in regions of solids subjected to local bending. After checking the mesh set the material to 1060 Aluminum and **Run** the study.

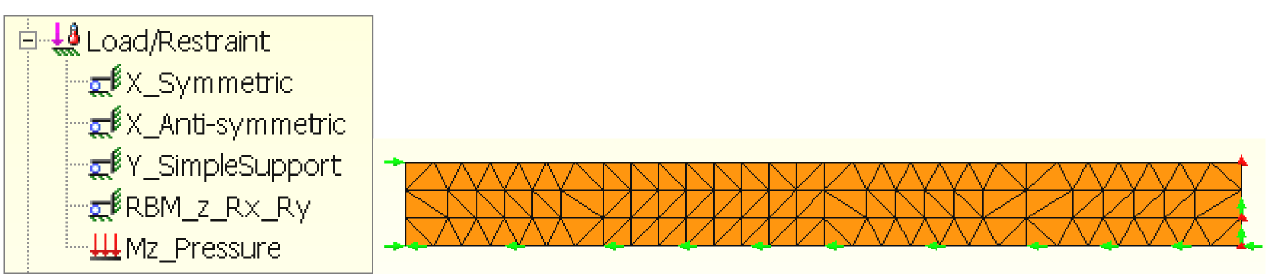


Figure 5-6 Beam quarter model shell mesh

5.2.8 Post-processing and result validation

The maximum vertical deflection and the maximum horizontal fiber stress will be recovered and compared to a beam theory estimate in order to try to validate the FEA study. For this simple geometry and pure bending moment the beam theory results should be much more accurate than is usually true. As stated above, the maximum vertical deflection at the centerline is predicted by beam theory to be $v_{max} = 1.36e\text{-}3\ m$. The resultant displacement vectors are seen in Figure 5-7. They are seen to become vertical at the centerline. A probe displacement result at the bottom point of the vertical centerline line gives $v_{FEA} = 1.36e\text{-}3\ m$, which agrees to three significant figures with the elementary theory. A detail view of the support region (bottom of Figure 5-7) shows that the displacement vectors close to the restraint are basically rotating about that end.

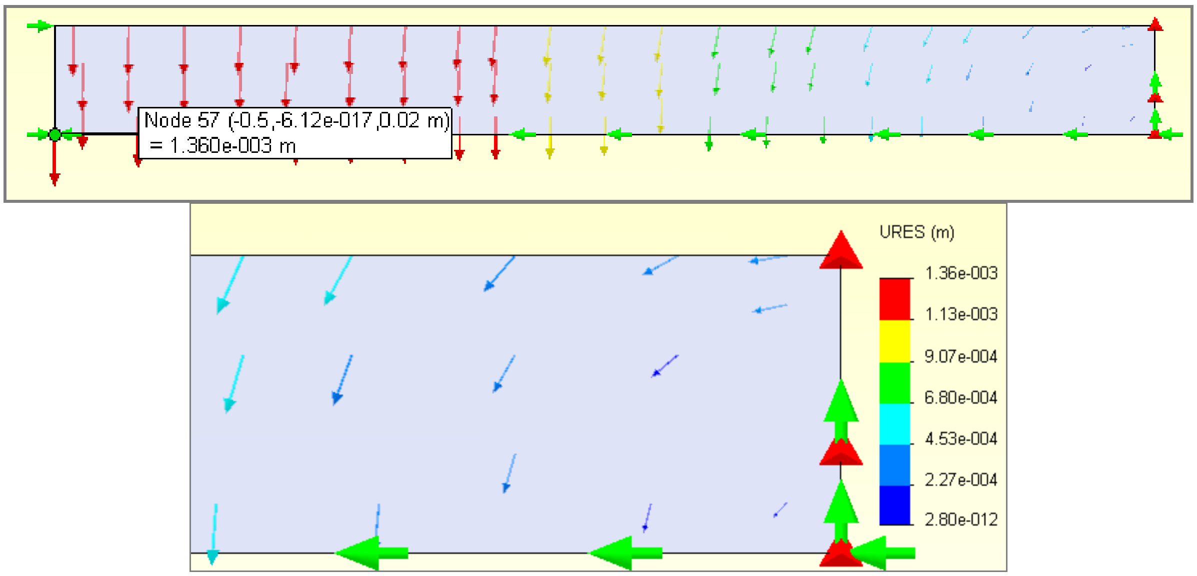


Figure 5-7 Displacement vectors in the quarter anti-symmetry beam

For bending by end couples only, the elementary theory states that the horizontal fiber stress is constant along the length of the beam and is equal to the applied end pressure. That is, the top fiber is predicted to be in compression with a stress value $\sigma_x = p_{max} = 3.75e7\ N/\text{m}^2$. That seems to agree with the contour range in Figure 5-9 and indeed, a stress probe there gives a value of $\sigma_x = -3.77e7\ N/\text{m}^2$. Beam theory gives a linear variation, through the depth, from that maximum to zero at the neutral axis. To compare with that, a graph of along the quarter point split line is given in Figure 5-9. It shows that the seven nodes along the edges of the three quadratic elements have picked up the predicted linear graph quite well. For the next load case of a full span line load the shear stress (that is zero here) will be parabolic and the corresponding graph will be less accurate for such a crude mesh.

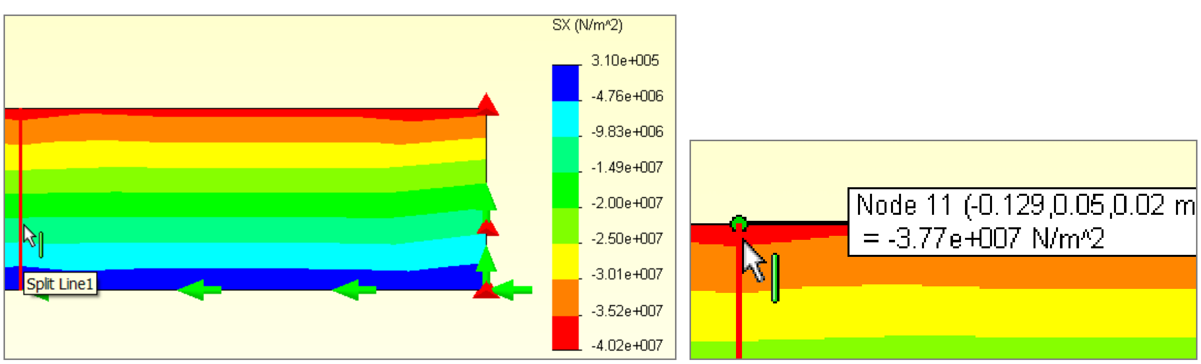


Figure 5-8 Horizontal stress along the right L/8 span segment

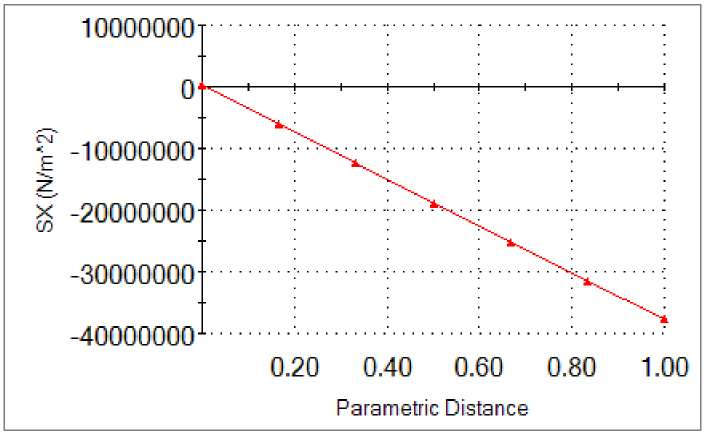
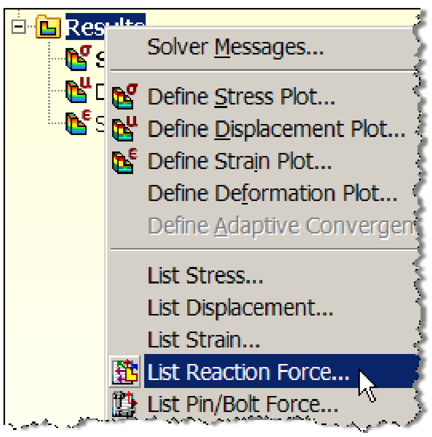


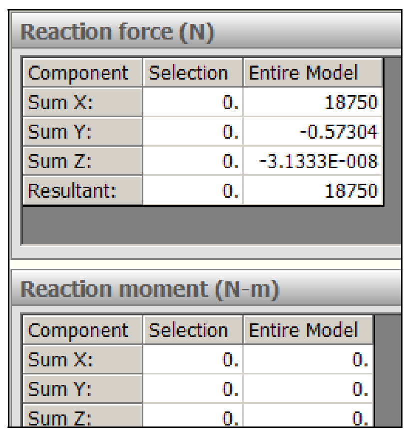
Figure 5-9 Graph of horizontal stress at vertical lineL/8 from the support

5.2.9 Reaction recovery

For this first load case, the only external applied load is the horizontal pressure distribution. It caused a resultant external horizontal force that was shown above to be *F=18,750 N*. You should expect the finite element reaction to be equal and opposite of that external resultant load. Check that in the manager menu:

1. Right click **Results→ List Reaction Force** to open the panel with the reaction forces and moments.





Reaction force (N)

Component	Selection	Entire Model
Sum X:	0.	18750
Sum Y:	0.	-0.57304
Sum Z:	0.	-3.1333E-008
Resultant:	0.	18750

Reaction moment (N-m)

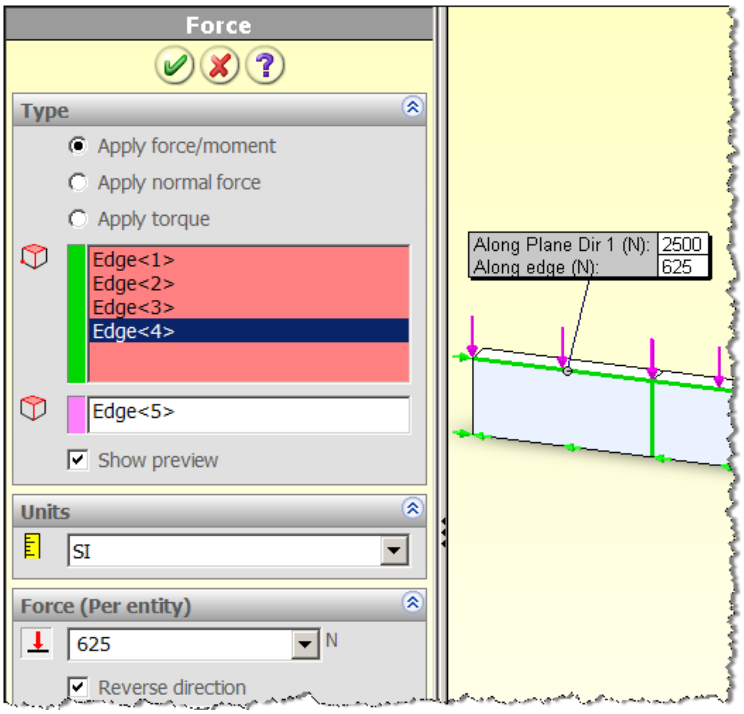
Component	Selection	Entire Model
Sum X:	0.	0.
Sum Y:	0.	0.
Sum Z:	0.	0.

2. Examine the horizontal (*x*) reaction force above and verify that its sum is *18,750 N.* (The sum of the moments is often confusing because they are computed with respect to the origin of the global coordinate system, and most programs never mention that fact.)

To test your experience with CosmosWorks, you should now run this special case study as a full 3D solid subject to the same end pressures. You will find this model was quite accurate. While planning 3D meshes you can get useful insights by running a 2D study like this. Also, a 2D approximation can be a useful validation tool if no analytic results or experimental values are available. They can also be easier to visualize. Of course, many problems require a full 3D study but 1D or 2D studies along the way are educational.

5.3 Load case 2, the transverse line load

Having validated the moment load case, the line load will be validated and then both load cases will be activated to obtain the results of the original problem statement. First, go to the manager menu, right click on the moment pressure load and suppress it. Next you open a new force case to account for the line load. Recall that the line load totaled *10,000 N*. Since the part has been reduced to one-fourth, through the use of symmetry and anti-symmetry, you only need to distribute *2,500 N* over this model. There are two ways to do that for selected surface shell formulation of any plane stress problem. They are to apply that total as either a line load, or to distribute it over the mesh face as a tangential shear traction (which is the better way). Figure 5-10 (left) shows the Apply Force approach. That approach has been made less clear by the way the split lines were constructed. The top of the beam has been split into four segments and this method applies a force *per entity.* Therefore, a resultant force of *625 N* per edge segment is specified. Had the split lines not had equal spacing you would have to measure each of their lengths and go through this procedure four times (the pressure approach avoids that potential complication)..



Reaction force (N)

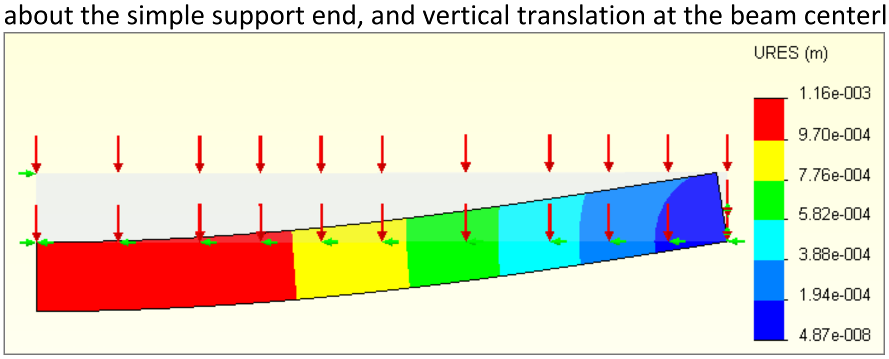
Component	Selection	Entire Model
Sum X:	0.	-0.36609
Sum Y:	0.	2500.3
Sum Z:	0.	-0.00016134
Resultant:	0.	2500.3

Figure 5-10 Second loadcase line load

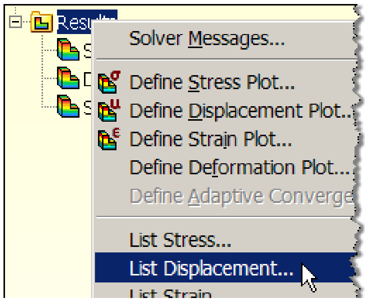
With this second load case in place the study is simply run again with the same restraints and mesh. Since this loading procedure was potentially confusing, the first information recovered from the results file was the force reaction data, on the right in Figure 5-10. Those data verify that the total vertical force on the quarter model was *2500 N*, as desired. Thus, only a series of quick spot checks of the results are carried out before moving on to the true problem where both load cases are activated.

The beam theory validation result, for this line load, predicted a maximum vertical centerline deflection of $v_{max} = 1.13e\text{-}3\ m$. The plane stress maximum deflection was extracted:

1. Double click on Plot1 under displacements. The contoured magnitude shows a rotational motion about the simple support end, and vertical translation at the beam centerline, as expected.



1. Right click in the manager menu **Results→List Displacements** to open the **Displacements List window**.



2. Under **List Options** select **Absolute Max,** click **OK**.

3. When the list appears note that the maximum deflection is *1.16e-3 m* at the vertical centerline position. That is very close to the initial validation estimate.

Node	X (m)	Y (m)	Z (m)	URES (m)
57	-0.5	0	0.02	1.16355e-003
250	-0.5	0.00833333	0.02	1.16348e-003

The numerical value of the maximum horizontal fiber stress was listed in a similar manner. The maximum compression value, in Figure 5-11, of $\sigma_x = -4.04e7\ N/m^2$ compares well with the simple beam theory value of -*3.75e7* N/m^2, being about a 7% difference. Since the mesh is so crude the beam stress is probably the most accurate and the plane stress value will match it as a reasonably fine mesh is introduced. The purpose of the crude mesh is to illustrate the need for mesh control is solids undergoing mainly flexural stresses. To illustrate that point, Figure 5-12 presents the normal stress and shear stress, through the depth, at the L/4 and L/8 positions.

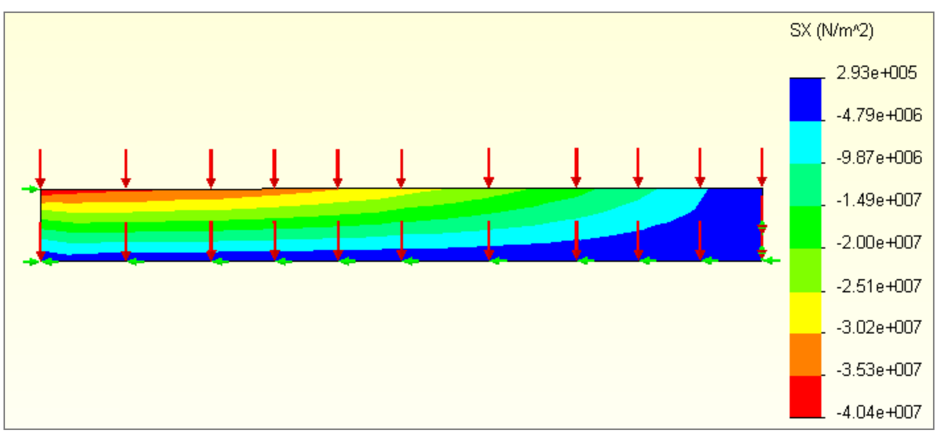
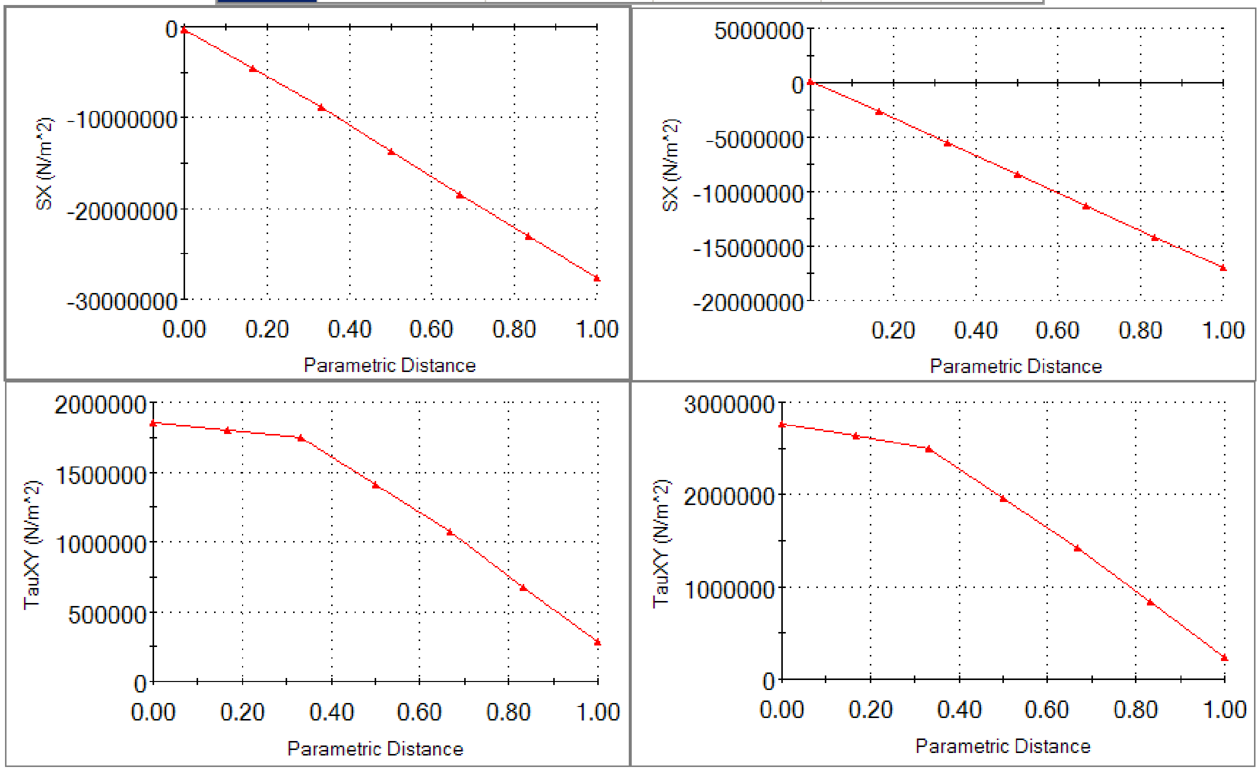


Figure 5-11 Horizontal stresses for the second loadcase

Beam theory says the normal stress is linear while the shear stress is parabolic. The beam theory shear stress should be zero at the top fiber and, for a rectangular cross-section, has a maximum value at the neutral axis of *1.88e6* N/m^2. The graph values in Figure 5-12 shows a plane stress maximum shear stress of *1.84e6* N/m^2 and a minimum of *0.3 e6* N/m^2. That is quite good agreement, but it took six quadratic elements through the full depth to capture the shear. The top parabolic segment is approximated by three linear segments.

Node	X (m)	Y (m)	Z (m)	SX (N/m^2)
74	-0.5	0.05	0.02	-4.03539e+007



L / 4   L / 8

Figure 5-12 Normal and shear stresses at two sections for the second loadcase

5.3.1 Alternate line load option

Before leaving this single load case the alternate method for applying the effect of the line load is mentioned. Basically it is applied as a pressure parallel to the flat face of the entire plane stress model (that is, it is actually a shear traction in the vertical direction). To compute the necessary shear stress the quarter model load (*2,500 N*) was divided by the part face area ($h\ L/2 = 0.025\ m^2$) and entered are the pressure value tangent to the reference geometry, as shown in Figure 5-13. This approach gave the exact same results as those in the previous section. This load approach as suppressed for the final combined loading.

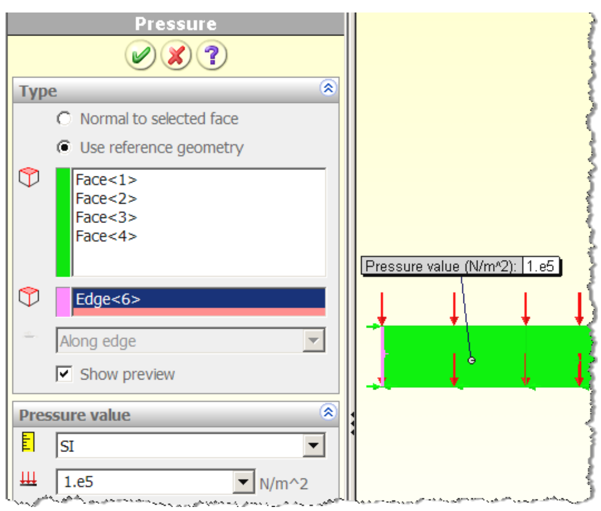
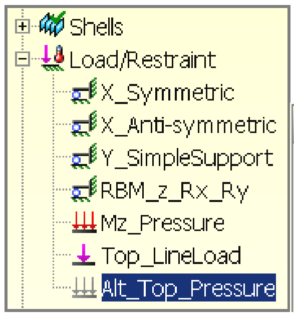


Figure 5-13 Applying a vertical load as a tangential shear stress

5.4 Combined load cases

Having validated each of the two load cases they are combined by un-suppressing the end moment condition (Figure 5-14 left) and running the study again with the same mesh. Here, the two sets of peak deflections and stresses simply add because it is a linear analysis. A quick spot check verifies the expected results. The reaction force components were verified (Figure 5-14 right) before listing the maximum deflection and fiber stress (Figure 5-15).



Reaction force (N)

Component	Selection	Entire Model
Sum X:	0.	18750
Sum Y:	0.	2498
Sum Z:	0.	0.00073855
Resultant:	0.	18916

Figure 5-14 Verifying the reaction for the shear stress load

Node	X (m)	Y (m)	Z (m)	URES (m)
57	-0.5	0	0.02	2.52903e-003

Node	X (m)	Y (m)	Z (m)	SX (N/m^2)
74	-0.5	0.05	0.02	-8.09971e+007

Figure 5-15 Probe values for maximum displacement and horizontal stress

What remains to be done is to examine the likely failure criteria that could be applied to this material. They include the Von Mises effective stress, the maximum principle shear stress, and the maximum principle normal stress. The Von Mises contour values are shown in Figure 5-16. Twice the maximum shear stress (the stress intensity) is given in the top of Figure 5-17, while the bottom portion displays the maximum principle stress (P3). Actually, P3 is compressive here but it corresponds to the mirror image tension on the bottom fiber of the actual beam. All three stress values need to be compared to the yield point stress of $2.8e7\ N/m^2$. The arrow in the figure highlights where that falls on the color bar. All of the criteria exceed that value, so the part will have to be revised. At this point failure is determined even before a Factor of Safety (FOS) has been assigned. For ductile materials, the common values for the FOS range from 1.3 to 5, or more [9, 12]. Assume a *FOS = 3*. The current design is a factor of about 3.3 over the yield stress. Combining that with the FOS means that the stresses need to be reduced by about a factor of 10.

The cross-sectional moment of inertia, $I = t\,h^3/12$, is proportional to the thickness, t, so doubling the thickness cuts the deflections and stresses in half. Changing the depth, h, is more effective for bending loads. It reduces the deflection by $1/h^3$ and the stresses by a factor of $1/(2\,h^2)$. The desired reduction of stresses could be obtained by increasing the depth by a factor of 2.25. The above discussion assumed that buckling has been eliminated by a buckling analysis. Since buckling is usually sudden and catastrophic it would require a much higher FOS.

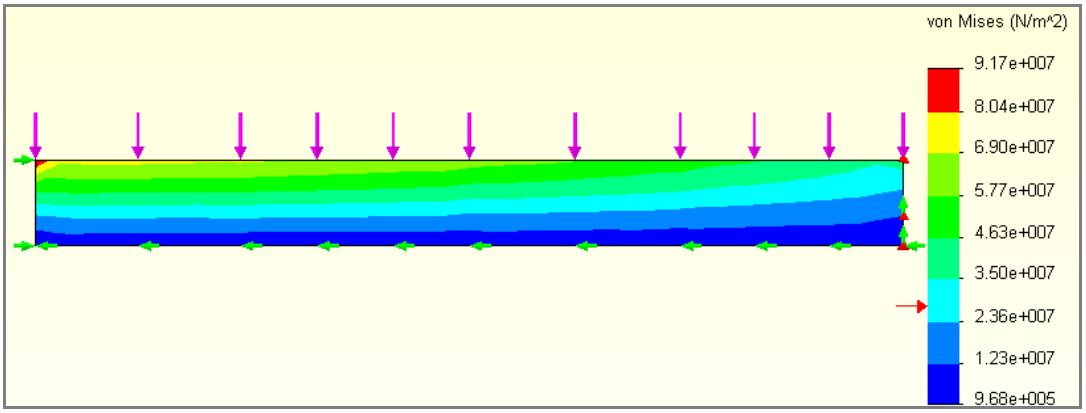
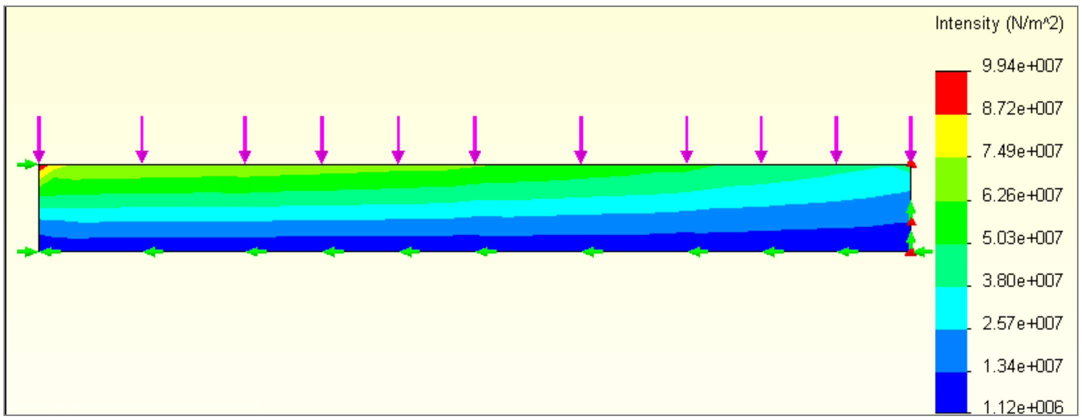


Figure 5-16 Von Mises stress in the beam with a line load.



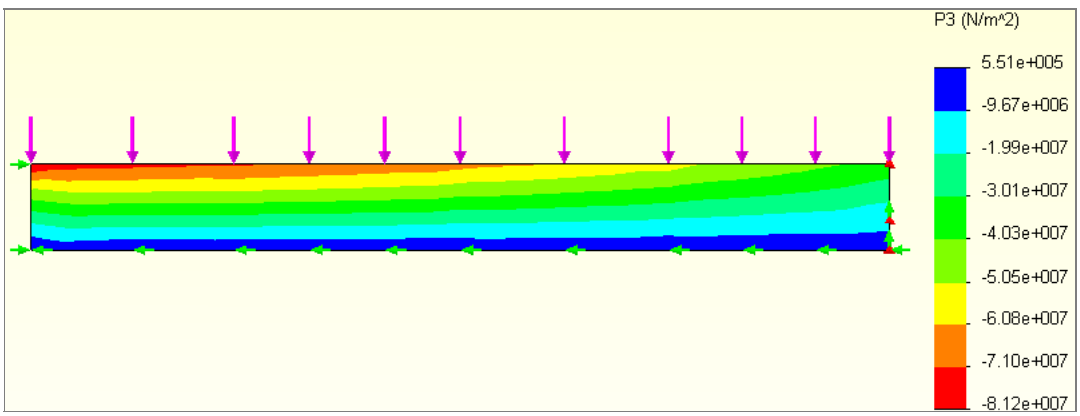


Figure 5-17 Beam principal stress and maximum shear contours

5.4.1 Advanced output options

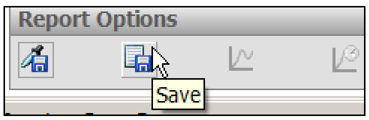
There are times when the software will not provide the graphical output you desire. For example, you may wish to graph the plane stress deflection against experimentally measured deflections. The CosmosWorks *List Selected* feature for any contoured value allows the data on selected edges, split lines, or surfaces to be saved to a file in a comma separated value format (*.csv). Such a file can be opened in an Excel spreadsheet, or Matlab, to be plotted and/or combined with other data. To illustrate the point, when the beam deflection values were contoured the bottom edge was selected to place its deflections in a table:

1. With a **displacement plot** showing right click on the **Plot name→List Selected**.
2. Select the four bottom lines of the beam. **Update**.
3. The bottom of the listing window has a **Summary** of the data.

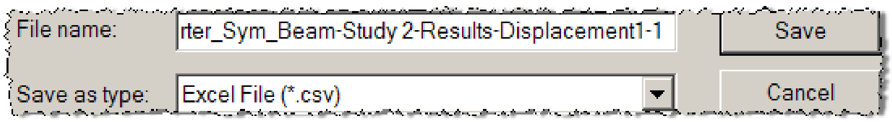
Summary

	Value	
Sum	0.31713	m
Avg	0.0019697	m
Max	0.002529	m
Min	1.1363e-007	m
RMS	0.0020401	m

4. The lower **Report Option** region does not include the Graph Icon, but does show a **Save icon**. That is because the path has multiple lines. Pick **Save** to have the listed data (node number, deflection value, and *x*-, *y*-, *z*-coordinates) to be output as a comma separated values (csv) file.



5. **Name and save** the data for use elsewhere.



Cosmos did not offer a plot option since could not identify which item to sort. You know that the multiple line segments should be sorted by the x-coordinate value. Therefore, the data were opened in Excel, sorted by *x*-coordinate, and graphed as deflection versus position (Figure 5-18). You could add experimental deflection values to the same file and add a second curve to the display for comparison purposes.

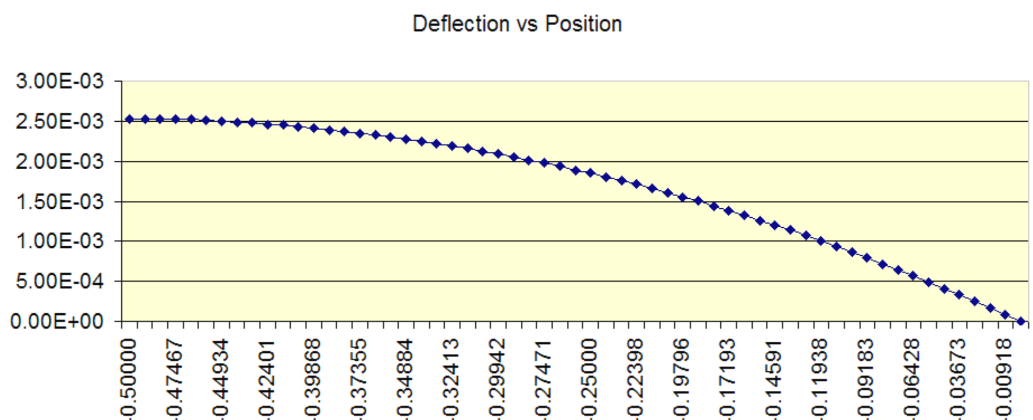


Figure 5-18 Excel displacement graph from saved CSV file

6 Centrifugal loads and angular accelerations

6.1 Introduction

This example will look at essentially planar objects subjected to centrifugal loads. That is, loads due to angular velocity and/or angular acceleration about an axis. The part under consideration is a spinning grid strainer that rotates about a center axis perpendicular to its plane. The part has five symmetrical segments, of 72 degrees each, and each segment has a set of slots that have mirror symmetry about a plane at 36 degrees. The questions are: 1. does a cyclic symmetric part, with respect to its spin axis, have a corresponding set of cyclic displacements and stresses when subject to an angular velocity, ω, about that axis? 2. does it have the same type of behavior when subjected to an angular acceleration?

To answer these questions you need to recall the acceleration kinematics of a point mass, dm = ρ dV, following a circular path of radius r. In the radial direction there are usually two terms, $r\ \omega^2$ that always acts toward the center and d^2r/dt^2 acting in the direction of change of the radial velocity. The latter term is zero when r is constant, as on a rigid body. In the tangential direction there are also two components in general: r α acting in the direction of **α** and a Coriolis term of 2 dr/dt ω, in the direction of **ω** if dr/dt is positive. Again the latter term is zero for a rigid body. The remaining radial acceleration ($r\ \omega^2$) always acts through the axis of rotation (as a purely radial load), thus, it will have full cyclic symmetry. Angular acceleration always acts in the tangential direction a rotating part, as it spins up or spins down. A part with angular acceleration is basically subjected to torsional or cantilever like loading about its axis of rotation. To illustrate these concepts consider a cantilever beam rotating about an axis outside its left end. The deformed shapes and stress levels for constant angular velocity and angular acceleration are shown in Figure 6-2 and Figure 6-2, respectively.

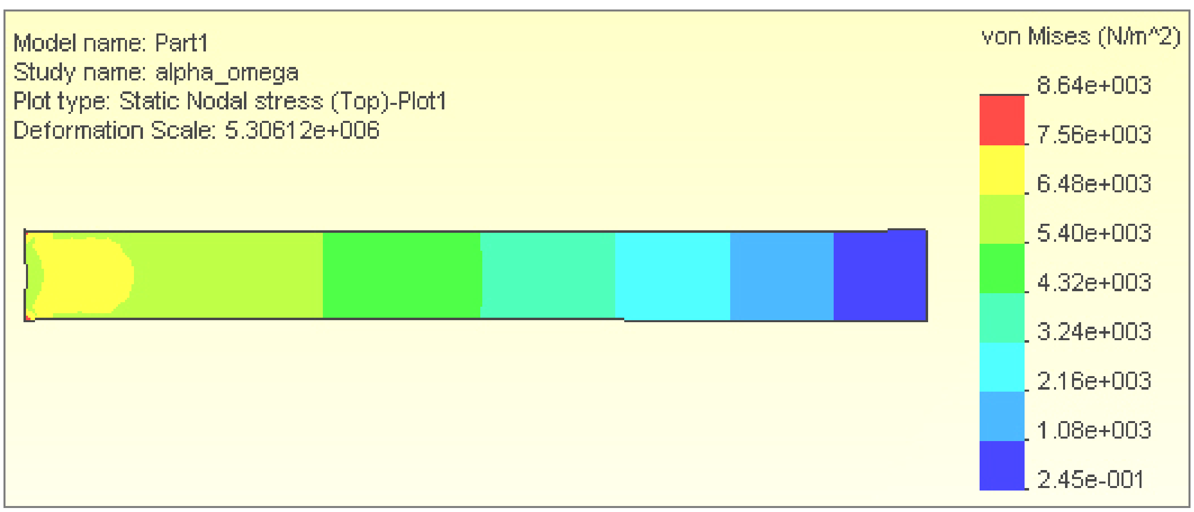


Figure 6-1 Beam with constant angular velocity about the left end

Note that any radial displacement induces hoop strains and stresses if some material is connected all around the circumference. For angular velocity loading only you can begin with the simplest symmetric segment. By way of comparison, for angular acceleration only, the body force load is transverse to the beam and increases with the distance from the left end (Figure 6-2). Of course, angular acceleration and angular velocity usually occur together in transient operations. The worst case is often at a sudden start or stop where $\boldsymbol{\alpha} \gg \mathbf{0}$, $\boldsymbol{\omega} = \mathbf{0}$.

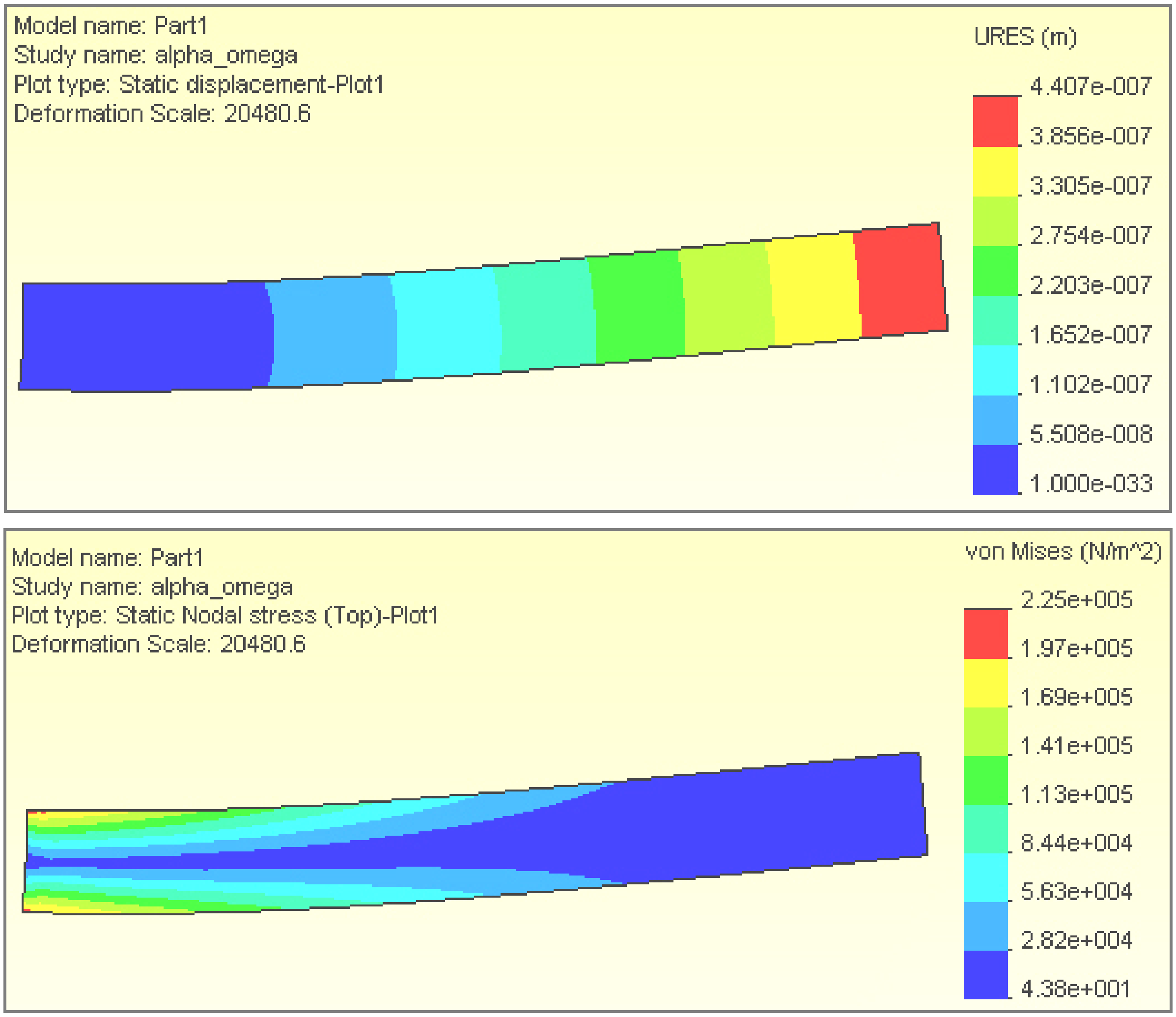


Figure 6-2 Beam with angular acceleration about the left end

6.2 Building a segment geometry

The desired rotating spin_grid is shown in Figure 6-3. The part has a inner shaft hole diameter, and outermost diameter of 1 inch and 4 inches, respectively. It has six curved slots ¼ inch wide, symmetric about the 36 degree line, and end in a semi-circular arc that is 1/8 inch from the 0 degree line. It is 0.20 inches thick.

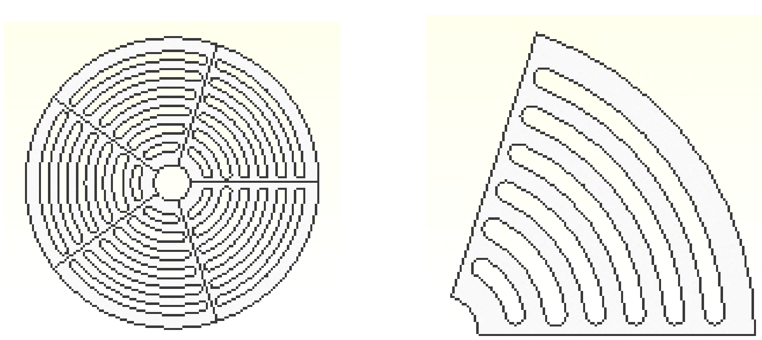
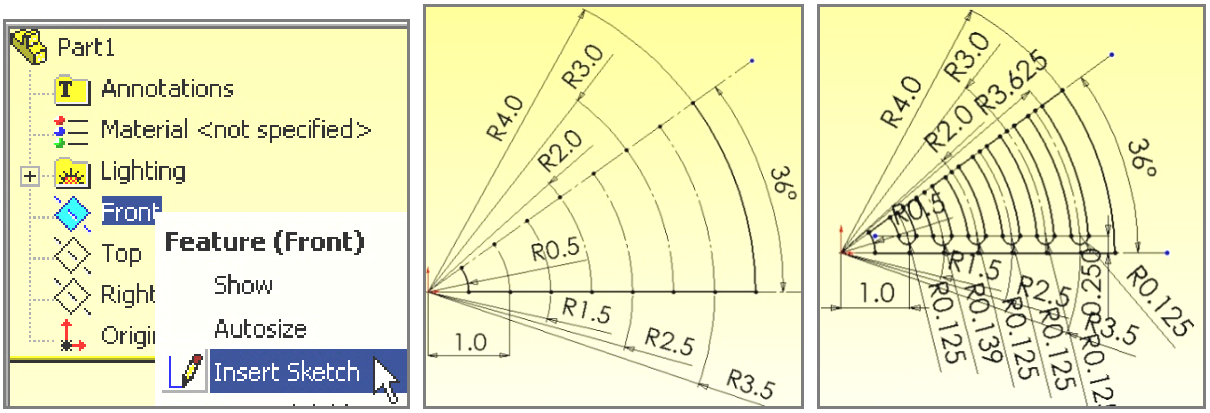


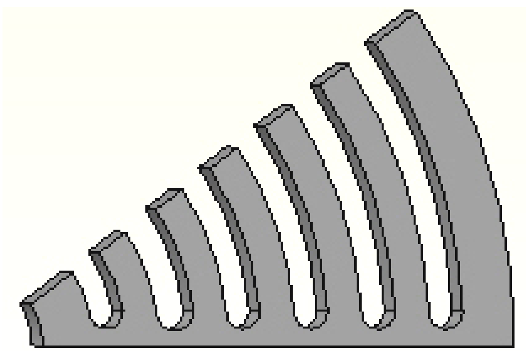
Figure 6-3 Full geometry and its one-fifth symmetry

Prepare a sketch in the top view with several radial and arc construction lines via:

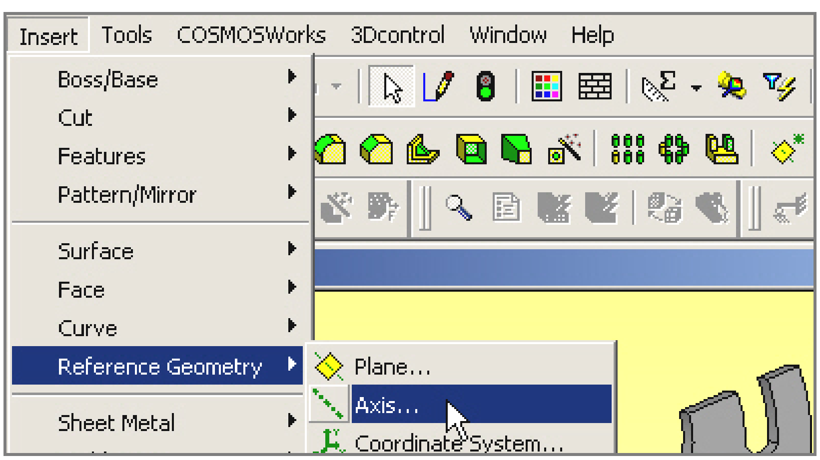
1. **Front→Insert Sketch**, build several construction lines for the center of the slots, the symmetry plane, and an off-set horizontal line for the fillet centers. Add arcs, and line segments to close the shape.



2. Extrude, about the mid-plane, to the specified thickness.



3. Define the axis of rotation. In this case it's the axis of the inner circular hole. Use **Insert→Reference Geometry→Axis** to open the **Axis panel.**



4. In the **Axis panel** check **Cylindrical/Conical Face** and select the inner-most cylindrical surface segment of the part and **Axis 1** will be defined as a reference geometry entity.

At this point the smallest geometrical region is complete and you can move on the **CosmosWorks Feature Manager** (CWManager) to conduct the deflection and stress analysis.

6.3 Initial CosmosWorks angular velocity model

6.3.1 Analysis type and material choice

To open a centrifugal analysis you have to decide if it is classified as static, vibration, or something else. Instead of applying Newton's second law, **F** = m **a**, for a dynamic formulation (where is **F** the resultant external force vector and is **a** the acceleration vector of mass m), D'Alembert's principle is invoked to use a static formulation of **F** – **F**$_I$ = **0**, where the inertia force magnitude is $F_I = m\ r\ \omega^2$ in the radial direction and/or m r α in the tangential direction. That is, we reverse the acceleration terms and treat it as a static problem. In Cosmos:

1. Right click on the **Part_name→Study, Name** the part (spin_grid here), pick **Static** analysis, and for the **Model Type** pick thin shell mid-surface. (A planar model is almost always the cheapest and fastest way to do initial studies of a constant thickness part.)
2. When the **Study Menu** appears right click on **Shells→Edit/Define Material**.
3. In the **Material panel** pick **Library→Steel→Cast Alloy**. Select **Units→English**. Note that the yield stress, SIGYLD, is about 35e3 psi. That material property will be compared to the von Mises Stress failure criterion later. (Property DENS is actually weight density, γ = ρ / g, and is mislabeled.)

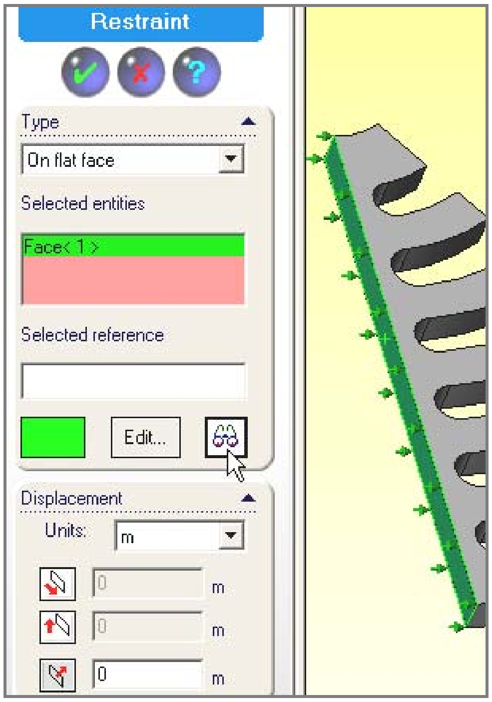
Material name: Cast Alloy Steel

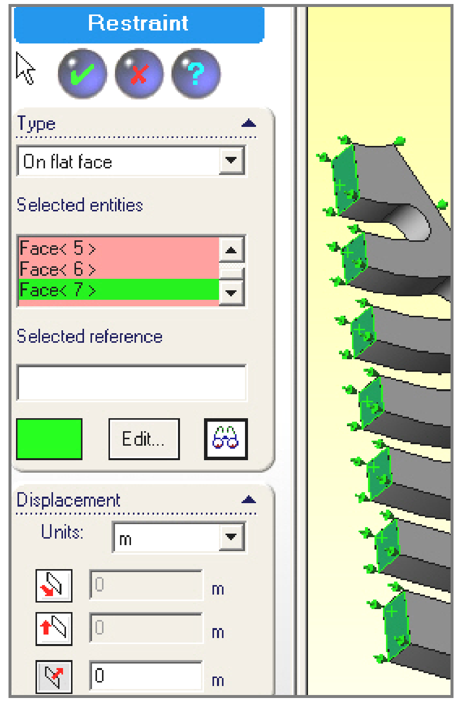
Property	Description	Value	Units
EX	Elastic modulus	27561888	psi
NUXY	Poisson's ratio	0.25999999	NA
GXY	Shear modulus	11314880	psi
DENS	Mass density	0.26372928	lb/in^3
SIGXT	Tensile strength	64999.996	psi
SIGXC	Compressive strength		psi
SIGYLD	Yield strength	34999.999	psi

6.3.2 Displacement restraints

For the centrifugal load due to the angular velocity only the spoke center plane and slots center plane always move in a radial direction. That means that they have no tangential displacement (that is, no displacement normal to those two flat radial planes). In the CW_Manager:

1. Select **Load/Restraints→Restraints** to activate the first **Restraint** panel. Pick **On flat face** as the **Type** and select the part face in the 0 degree plane. Set the **normal** displacement component to **zero** and **preview** the restraints, then click **OK**.





2. Repeat that process and select all seven of the part flat faces lying in the 36 degree plane (above).

At this point a total of eight surfaces are required to have only radial displacements. Either of the above two restraint operations eliminates a possible rigid body rotation about the axis of rotation. To prevent a rigid body translation of the part parallel to the axis you should also restrain the cylindrical shaft contact surface in that direction.

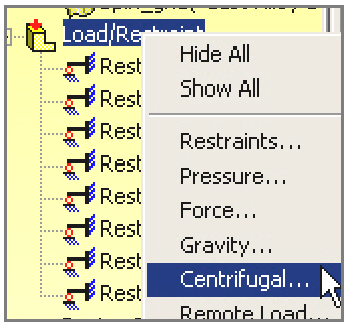
1. Select **Load/Restraint →Restraints** to activate the first **Restraint panel**.
2. **Type→On a cylindrical surface**, select the inter-most cylindrical surface.
3. **Displacement→Tangent** to the surface centerline.

The radial displacement component on that surface is not restrained so as to allow the (small) motion away from the shaft due to the angular velocity. Optionally, you could restrain the displacement around the circumference to prevent the rigid body rotation, but the restraints have already taken care of that.

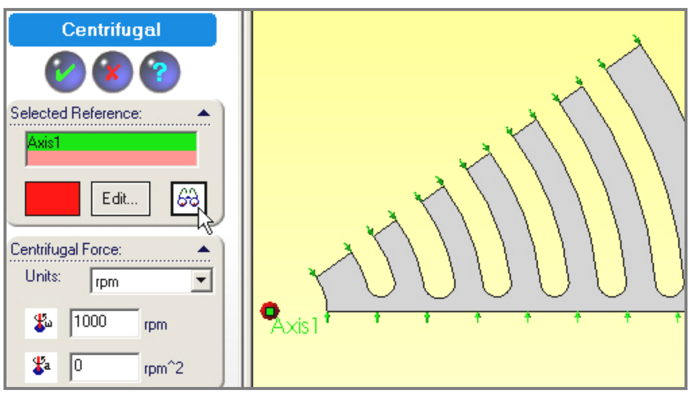
6.3.3 Angular velocity loading

Next you set the centrifugal body force load due to the angular velocity. That will require picking the rotational axis so:

1. Select **View→Axes.**
2. Right click **Load/Restraint→Centrifugal** to activate the **Centrifugal panel**.



3. There pick **Axis 1**, set rpm as the **Units** and type in 1000 for the **angular velocity.**

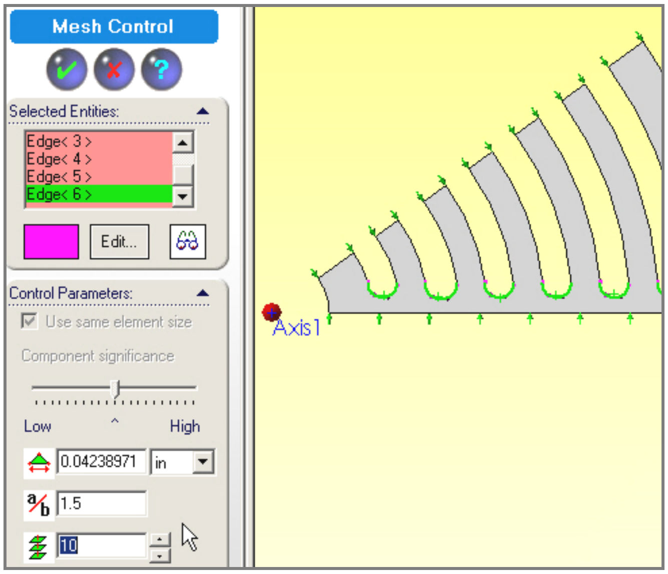


The radial acceleration, $a_r = r\,\omega^2$, varies linearly with distance from the axis, so expect the biggest loads to act on the outer rim. Also remember that the radial acceleration is also proportional to the square of the angular velocity. Thus, after this analysis if you want to reduce the stresses by a factor 4 you cut the angular velocity in half. (You would not have to repeat the analysis; just note the scaling in your written discussion. But you can re-run the study to make a pretty picture for the boss.)

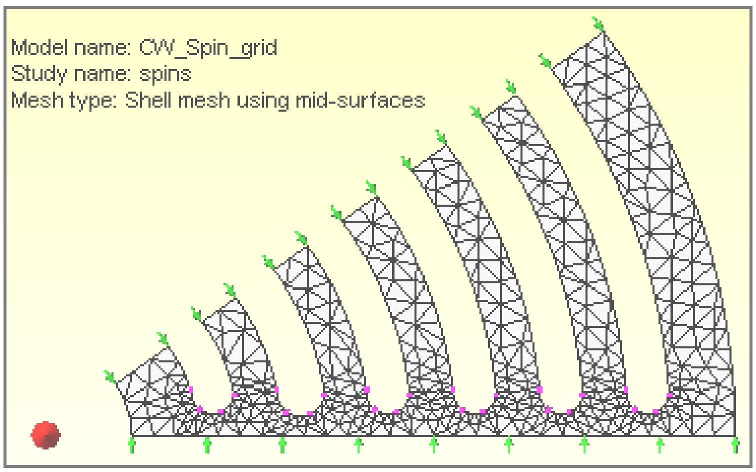
6.3.4 Mesh generation

For this preliminary study each curved ring segment will act similar to straight fixed-fixed beam of the same length under a transverse gravity load. (To know about what your answers should be from Cosmos, do that simple beam theory hand calculation to estimate the relative deflection at the center as well as the center and end section stresses.) Thus, bending stresses may concentrate near the ends so make the mesh smaller there:

1. Right click **Mesh→ Apply Control** to activate the **Mesh Control panel**.
2. There pick the six bottom arcs as the **Selected Entities** and size ten elements there.



3. Then right click **Mesh→Create.** The initial mesh looks a little coarse, but okay for a first analysis.



4. Start the equation solver by right clicking on the study name and selecting **Run.** When completed review the results.

6.3.5 Post-processing review

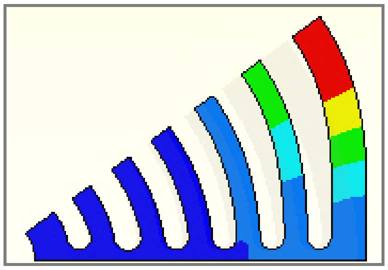
Both the displacements and stresses should always be checked for reasonableness. Sometimes the stresses depend only on the shape of the material. The deflections always depend on the material properties. Some tight tolerance mechanical designs (or building codes) place limits on the deflection values. CosmosWorks deflection values can be exported back to SolidWorks, along with the mesh model, so that an interference study can be done in SolidWorks.

6.3.5.1 Displacements

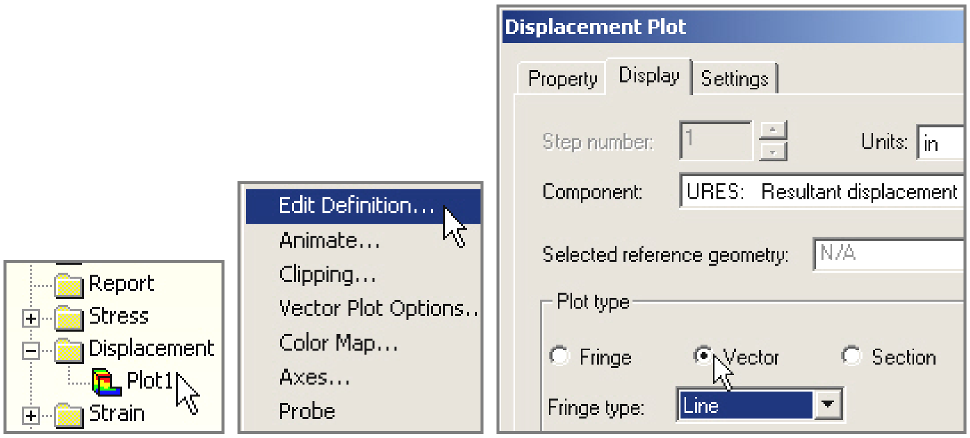
You should always see if the displacements look reasonable:

1. Double click on **Displacements→Plot 1** to see their default display, which is a continuous color contour format. Such a pretty picture is often handy to have in a report to the boss, but the author believes that more useful information is conveyed with the discrete band contours.

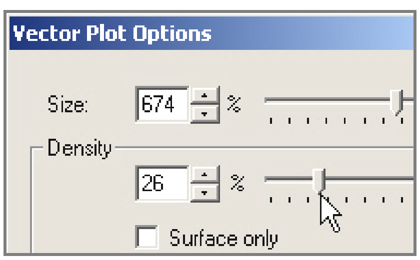
2. Right click on **Plot 1→Edit Definition→Fringe→Discrete**. The default deformed shape displacement contours and undeformed (gray) shape appears.

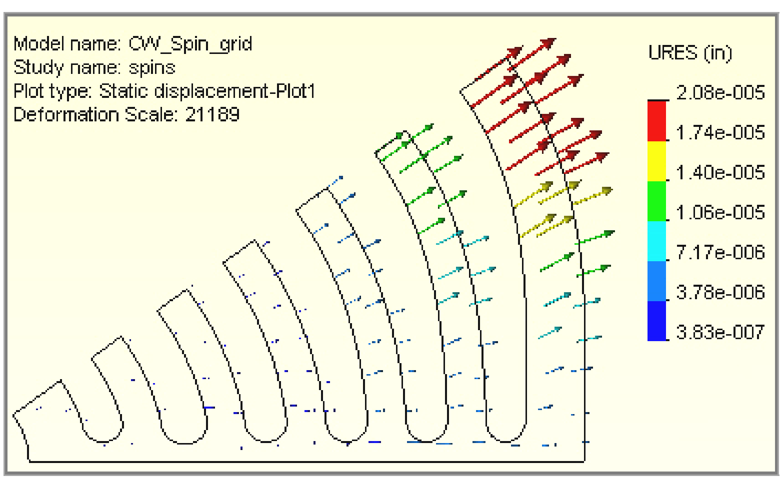


3. Since displacements are vector quantities you convey the most accurate information with vector plots. Right click **Plot1**; **Edit Definition→Displacement Plot → Display → Vector**.



4. When the vector plot appears, dynamically control it with **Vector Plot Options** and increase **Size**. Then dynamically vary the **Density** (of nodes displayed) to see different various nodes and their vectors displayed. Retain the one or two plots that are most informative.





As expected, the center of the outer-most rim has the largest displacement while the smallest displacement occurs along the radial “spoke” centered on the 0 degree plane. You may want to compare the relative displacement of the outer ring to a handbook approximation in order to validate the computed displacement results (and your knowledge of CosmosWorks). Localized information about the displacements at selected node or lines are also available:

1. Right click in the graphics area and select **Probe.** That lets you pick any set of nodes in the mesh and display their resultant displacement value and location (in gray) on the plot, as well as listing them.
2. The probe operation is usually easier if you display the mesh first with **Edit Definition→ Displacement Plot→Settings→Boundary Options→Mesh**. Using the support and center points in Figure 6-4 as probe points you find a relative displacement difference of about 2.01e-6 inches.

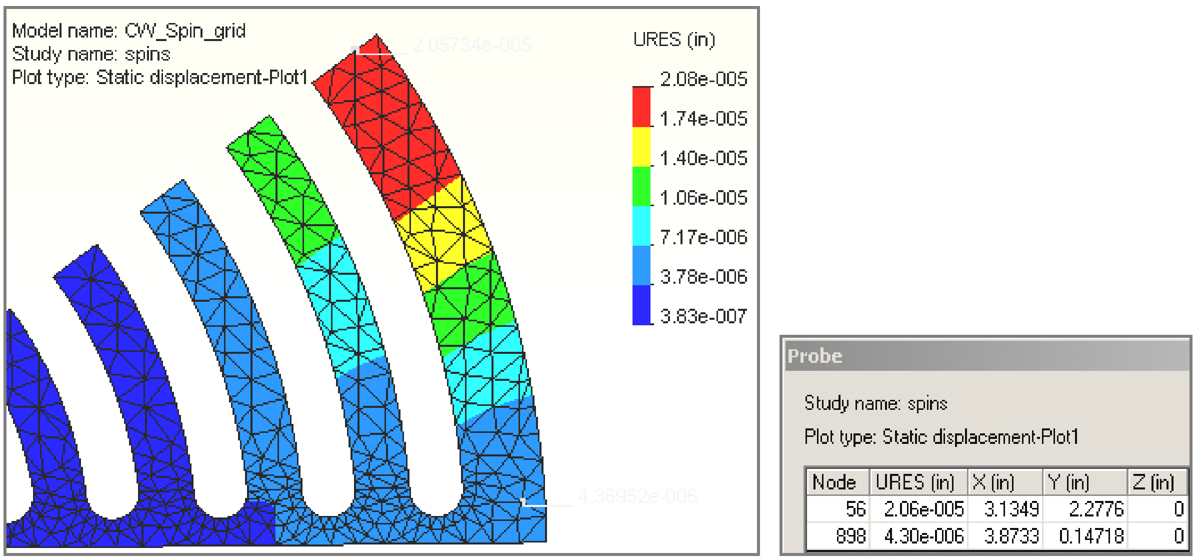


Figure 6-4 Outer ring displacements at the center and support

6.3.5.2 Stress results

All of the physical stress components are available for display, as well as various stress failure criteria. The proper choice of a failure criterion is material dependent, and it is the user’s responsibility to know which one is most valid for a particular material. The so called Effective Stress (actually distortional energy) value is often used for ductile materials.

6.3.5.3 Effective stress

The von Mises failure criterion (a scalar quantity) is superimposed on the deformed shape in Figure 6-5 using two formats: line contours, and discrete color contours. Also **Color Bar→ Color Map** was used to select only eight colors. The maximum effective stress is only about 260 psi compared to a yield stress of about 35,000 psi. That is a ratio of about 134. Since the centrifugal load varies with the square of the angular velocity you would have to increase the current ω by the square root of that ratio (about 11.5). In other words you should expect yielding to occur at ω= 11.5 (1000 rpm) = 11,500 rpm.

Since this is a linear analysis problem, it would not be necessary to repeat the run with that new angular velocity. You could simply scale both the displacements and the stresses by the appropriate constant. However, if you have a fast computer you may want to do so in order to include your most accurate plots in your written summary of the analysis.

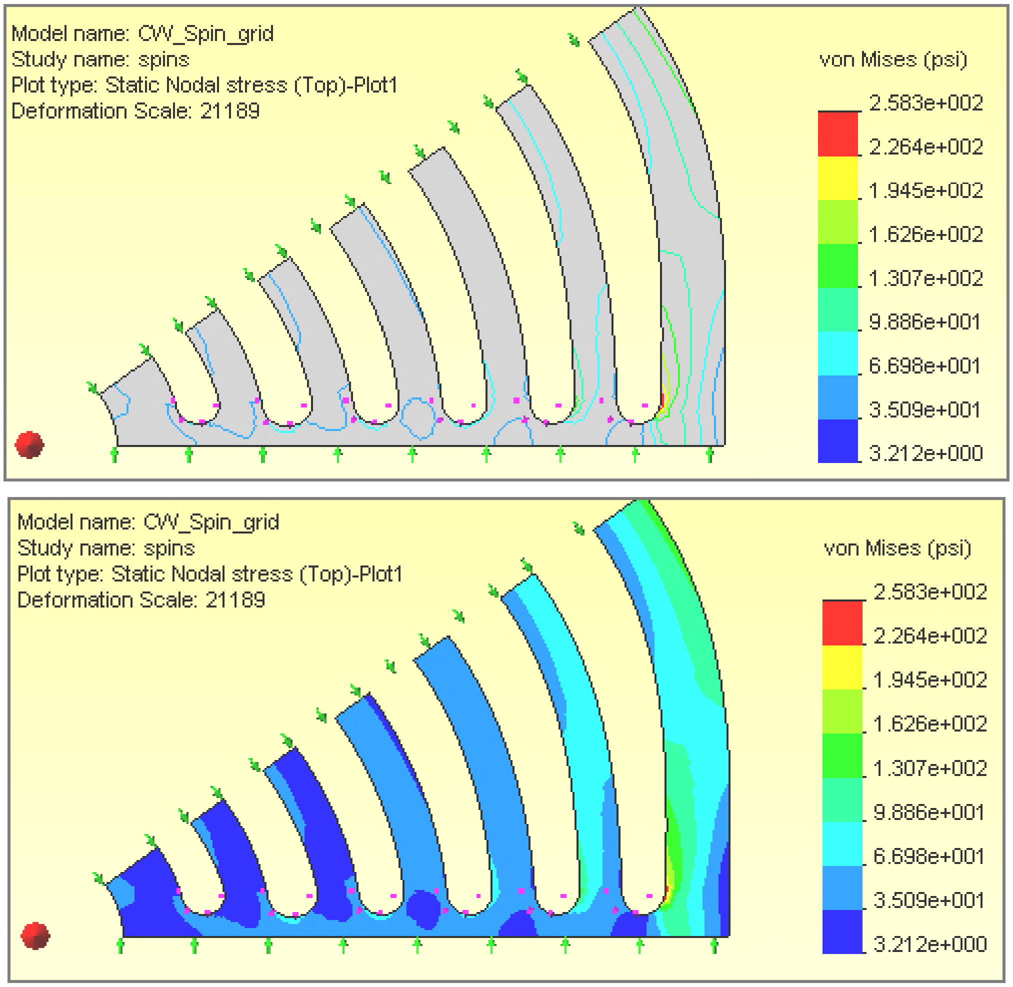


Figure 6-5 The von Mises failure criterion and deformed shape

6.3.5.4 Principal stress

The magnitude and direction of the maximum principal stress is informative (and critical for brittle materials). Since they are vector quantities they give a good visual check of the directions of the stress flow, especially in planar studies (they can be quite messy in 3-D):

1. Right click in the graphics area, select **Edit Description** then pick **P1 Maximum Principal Stress**, and **Vector** style and view the whole mesh again.
2. If the arrows are too small (look like dots) zoom in where they seem biggest and further enhance you plot with a right click in the graphics area, select **Vector Plot Options** and increase the vector size, and reduce the percentage of nodes used for the vector plot. A typical P1 plot, with the deformed shape, is given in Figure 6-6.

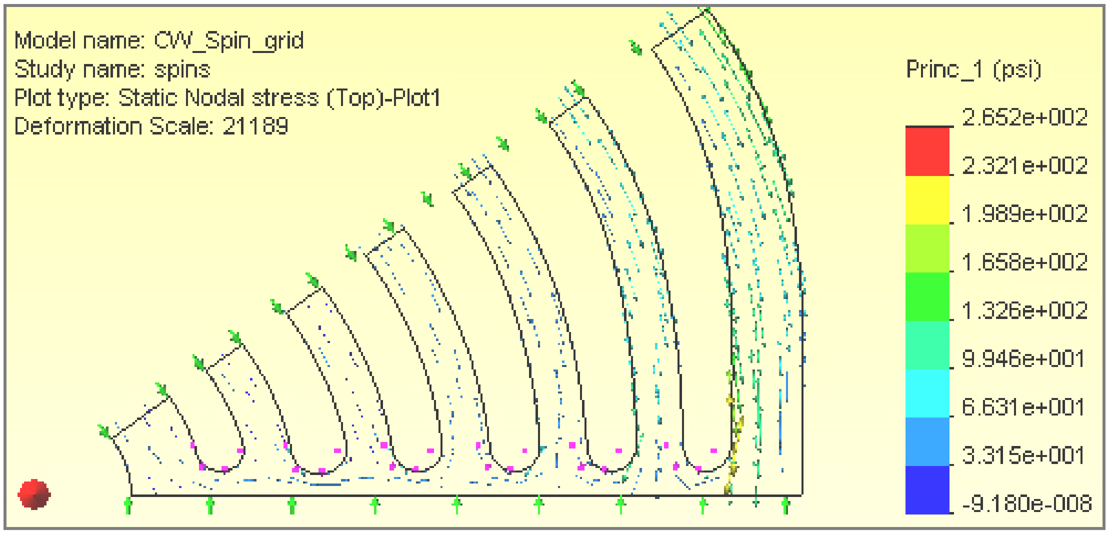


Figure 6-6 Principal stress vector due to angular velocity

6.3.6 Factor of safety

For this material you would use the von Mises effective stress failure criterion. That is, your factor of safety is defined as the yield stress divided by the maximum effective stress. It was noted above that the ratio is greater than ten in this preliminary study.

6.3.7 Expected and computed results

A handbook solution for a fixed-fixed beam is used for a validation estimate. Let the beam length be the arc length of the outer rib, $L = r\,(\Delta\theta) = (3.81 \text{ in.})(72/57.3 \text{ rad}) = 4.79$ inch. The load per unit length is the product of mass density, $\rho = \gamma/g$ with $\gamma = 0.264$ lb/in^3, area A = h t=0.0725 in^2, and the centripetal acceleration (for ω = 1000 rpm = 104.72 rad/sec), or $w = \rho A r \omega^2$. Here with $\gamma = 0.264$ lb/in^3 and $g = 386.4$ in/sec^2. Using the part material properties and geometrical data, w = (0.264 lb/in^3)(0.0725 in^2)(3.81 in.) (104.72 rad/sec)^2, so w = 2.07 lb/in, and the moment of inertia is $I = t\,h^3/12$ = 0.2"(0.3625")^3/12 = 2.88e-4 in^4. The maximum displacement (at the 36 degree plane relative to the 0 degree plane) is $\boldsymbol{\Delta_{max}}$ = 3.26e-6 inches.

This approximate displacement is too far from the computed value (found with the **Probe** feature) of 2.04e-5 inches. You want to be within a factor of ten, at the most, to validate that you have the correct material (Modulus of Elasticity, E = 27.6e6 psi). This would be an upper bound displacement since the full arc length was used, but the spoke has a thickness of about 0.25 inches. Thus, we could reduce the effective length to L= 4.24 inches. Since the beam theory deflection is proportional to L^4 the upper bound deflection estimate is $\boldsymbol{\Delta_{max}}$ = 3.26e-6 inches (4.24/4.79)^4 = 2.00e-5 inches, which is a very good agreement with the computed 2.04e-5 inches. (It is rare to get an estimate that agrees so well.)

The center moment is $M_1 = wL^2/24$ = 47.5 in-lb (for the first length assumption, and 37.2 in-lb for the second length). The maximum center bending stress is $\sigma_1 = M_1(h/2)/I$ = 2,062 psi (in the part's tangential direction), but the wall stress is twice that large (4,123 psi). This stress is in poor agreement with the computed value (found with the **Probe** feature) of about 260 psi. It now looks like a finer mesh is needed in the outer ring, but a coarser one could be used in the inner rings, for centrifugal loadings.

6.4 Angular acceleration model

The previous model considered only constant angular velocity, ω, so the angular acceleration was zero. During start and stop transitions both will present and the two effects can be superimposed because this is a linear analysis. Next consider the initial angular acceleration (where ω = 0 for an instant). You can always use the full model, but that takes a lot of computer resources. The most correct symmetric analysis would require using any 72 degree segment and invoking a special restraint know as multiple point constraints or repeated freedoms for nodes on those to edges. That means we know the two edges have the same displacement components normal and tangential to the edges, but they are still unknowns. Rather than doing that you can get an accurate approximation by assuming that the Poisson ratio effects can be neglected. Then symmetry planes of the model will have no radial displacement, just tangential displacement due to the tangential acceleration ($r\ \alpha$). Apply that approach with the previous 36 degree segment with to symmetry planes. Remember to prevent the rigid body motions of rotation about the axis and translation along the axis.

6.4.1 Restraints

First apply zero radial displacement on the two symmetry planes from the CWManager via **Load/Restraint→Restraints**. To prevent the two rigid body rotations select the inner cylindrical surface and impose displacement restraints in the axial dimension, and around the circumference, as shown in

Figure 6-7.

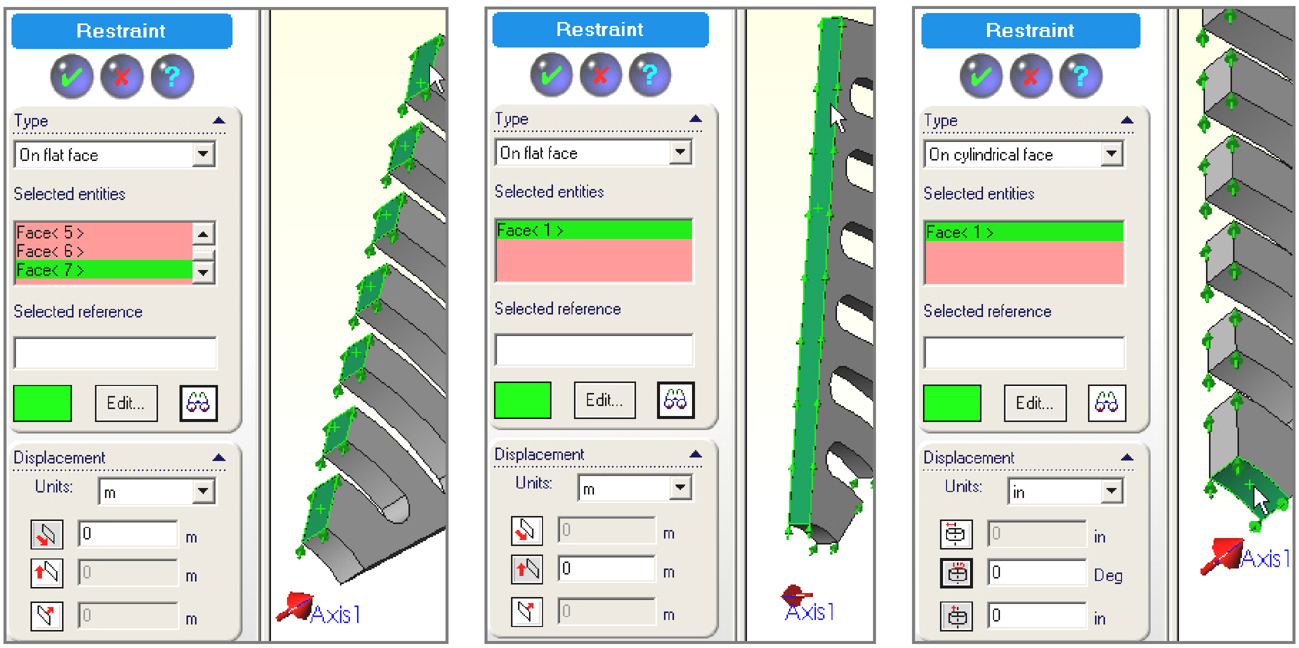


Figure 6-7 Invoking zero radial displacements and preventing rigid body motions

6.4.2 Angular acceleration Loads

You need to identify the axis and angular acceleration value. First, turn on the axes with **View→Axes.** Then select **Load/Restraint→Centrifugal** and pick **Axis 1** in the **Centrifugal panel** and type in the value of the angular acceleration (Figure 6-8).

6.4.3 Mesh and solution

Here you can simply use the same meshing process given above. That mesh creates slightly less than 9,000 equations to solve for the two displacement components at each node.

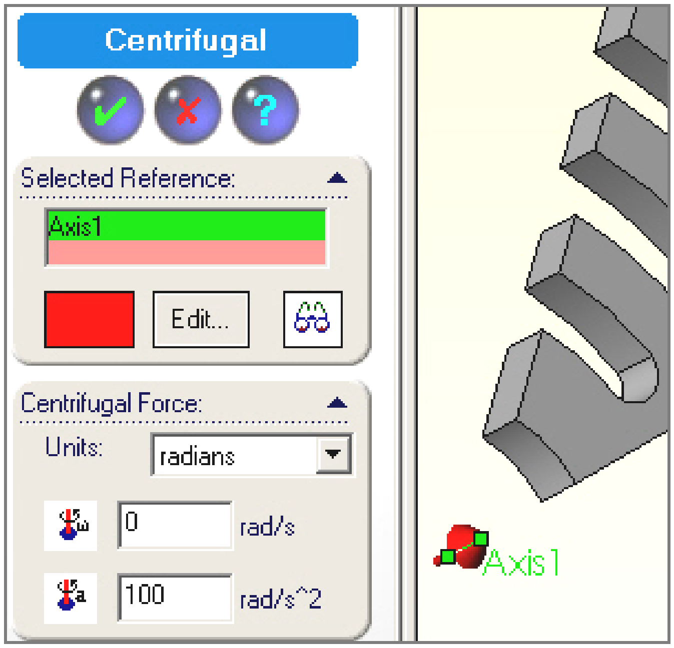


Figure 6-8 Angular acceleration (only) about the center axis

6.4.4 Post-processing

6.4.4.1 Displacements

The displacements, of

Figure 6-9, are mainly in the tangential direction (relative to the undeformed shape in light gray) as seen by the contour values and displacement vectors. Using the right click **Probe** the maximum displacement at the spoke was seen to be 2.1e-5 inches in the bottom table (and light gray in the middle).

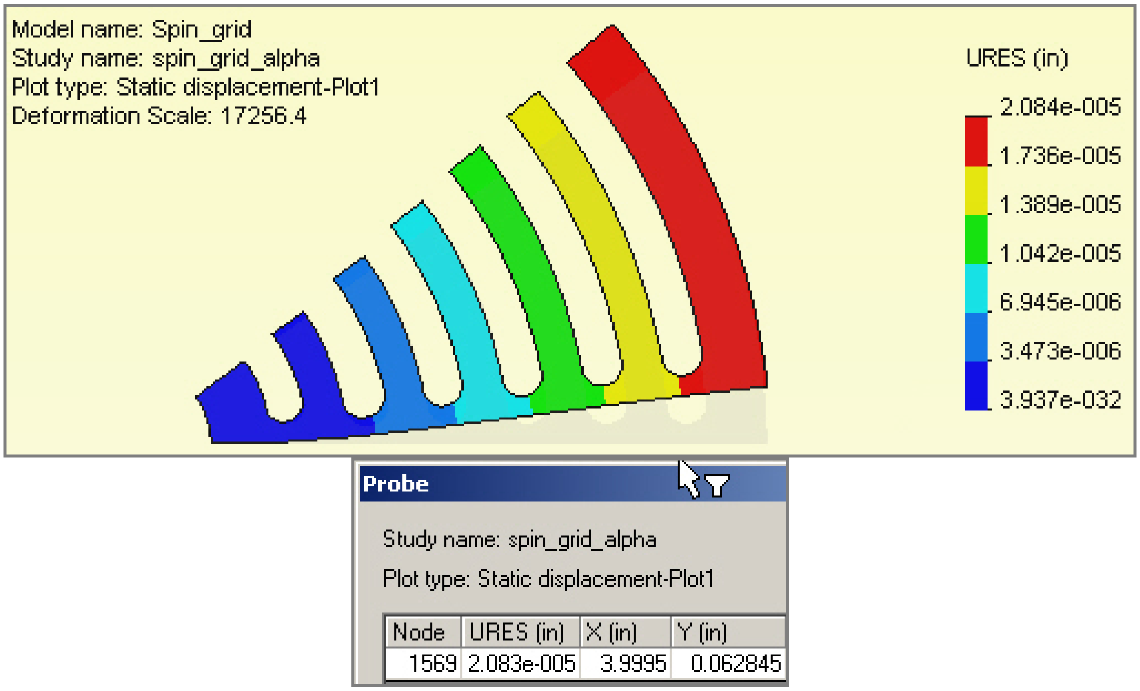


Figure 6-9 Maximum displacements from a vector plot and node probe

6.4.4.2 Stress results

The effective stresses (Figure 6-10) are quite low, for the current angular acceleration value. Increasing the angular acceleration by a factor of 500 would put the stress at about 34,000 psi, which is just below the yield point. The maximum stresses occur around the innermost slot. Figure 6-11 shows that the maximum tension occurs on the lower right edge of the first grove. Both the maximum compression and shear occur in the lower left (red) regions of Figure 6-10.

You should expect the maximum shear to occur there. The (reaction) torque necessary to accomplish the specified angular acceleration is transmitted radially by the spokes. The ribs can be viewed as similar to lumped masses attached to a cantilever spoke. As you move from the outer rib toward the rotation axis you are picking up more mass, and you get the maximum torque by the inner hole. That torque is transmitted by shear stresses here. The first open gap has the largest torque going through the spoke thickness.

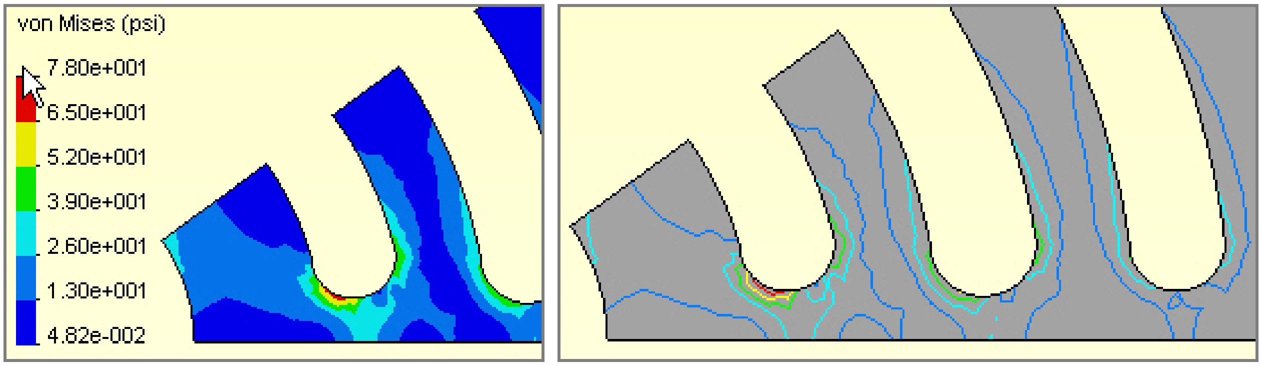


Figure 6-10 Region of maximum von Mises failure criterion

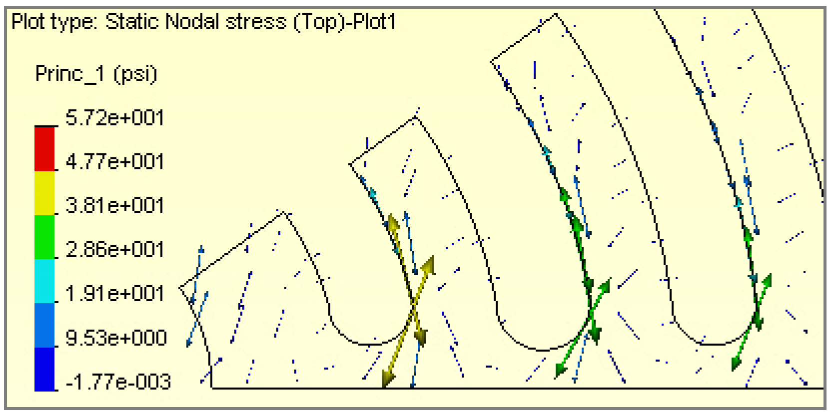


Figure 6-11 Maximum principal stress vectors around slot base

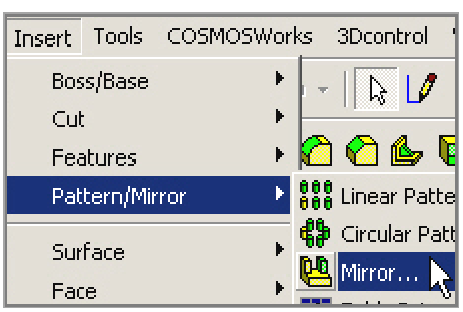
6.5 Full part model

If you failed to recognize the above symmetry cases and/or if you have both large angular velocity and angular acceleration acting at the same time (like braking from the maximum rotational speed) then you might be forced to employ a full part model. To do that you must first build the full part.

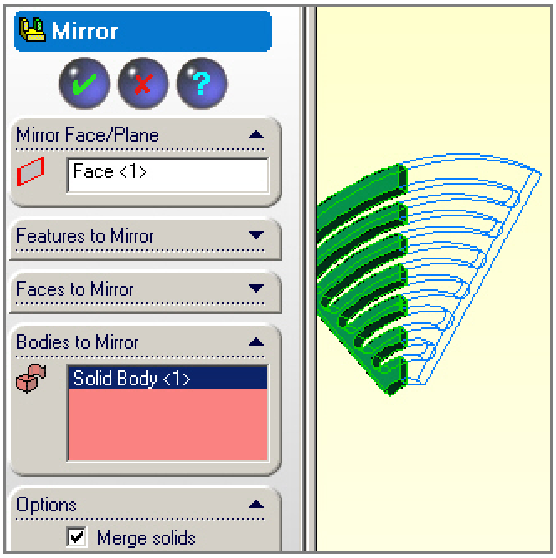
6.5.1 Building the part

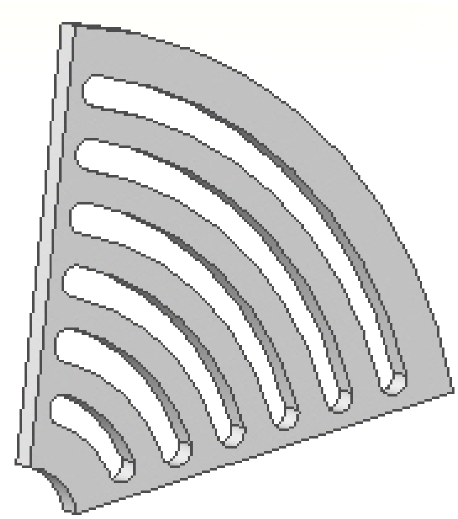
Return to the 36 degree segment geometry model. The full model geometry can be built from various mirrored and repeated uses of that segment. Open the segment geometry:

1. Select **Insert→Pattern/Mirror→Mirror**. Pick the original **Body**.



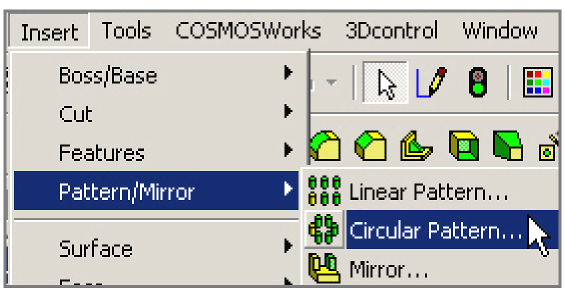
2. Chose any of the rib end faces as the **Mirror Face**, and **Merge** the two parts for a 72 degree segment.

3. 

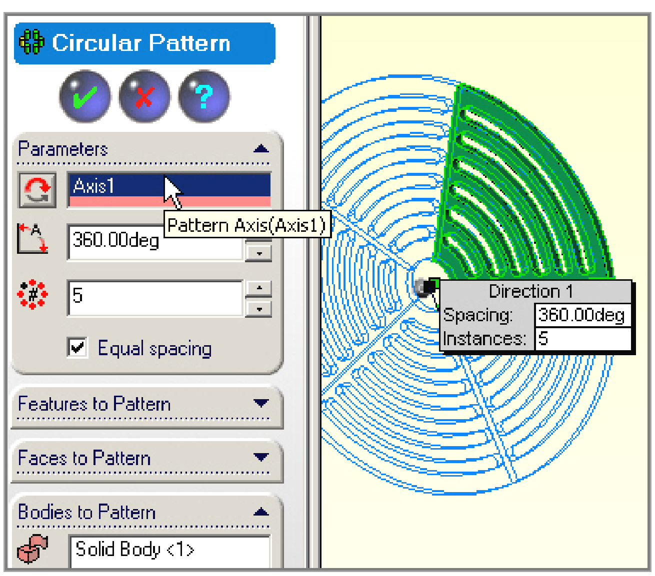


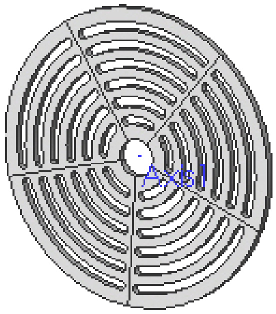
Five copies of that geometry will yield the full model. Construct them with a circular pattern copy:

1. **Insert→Pattern/Mirror→Circular Pattern**.

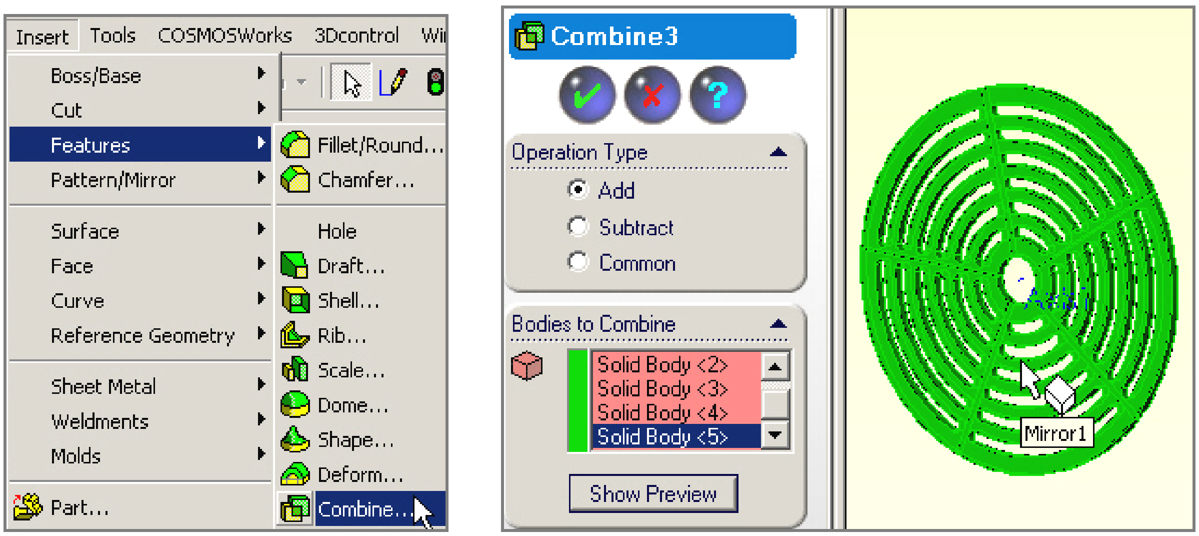


2. In the **Circular Pattern panel**, pick the new 72 degree segment as the **Body to Pattern**, preview the result and click **OK** to accept.





3. While this result may look okay, it is actually five independent parts (note the part lines every 72 degrees) and CosmosWorks will not accept it for analysis. They must be combined by a *Boolean union*. Use **Insert→Features→Combine** to bring up the **Combine panel**.



4. There the Boolean union **Operation Type** is named **Add**, and under **Bodies to Combine** you must select each of the five parts and click **OK**. Then the part lines vanish, as seen above, and you can now conduct an analysis in CosmosWorks.

6.5.2 Begin CosmosWorks study

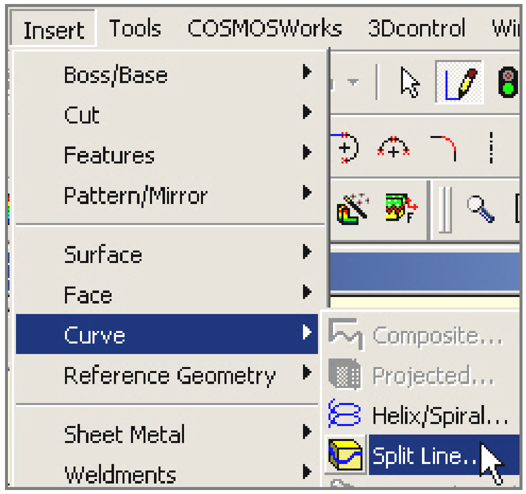
In the CWManager right click on the name and select **Study**, assign a name, select **Static** analysis, and use a **mid-surface shell** mesh. Use the same material properties as above.

6.5.3 Mesh generation

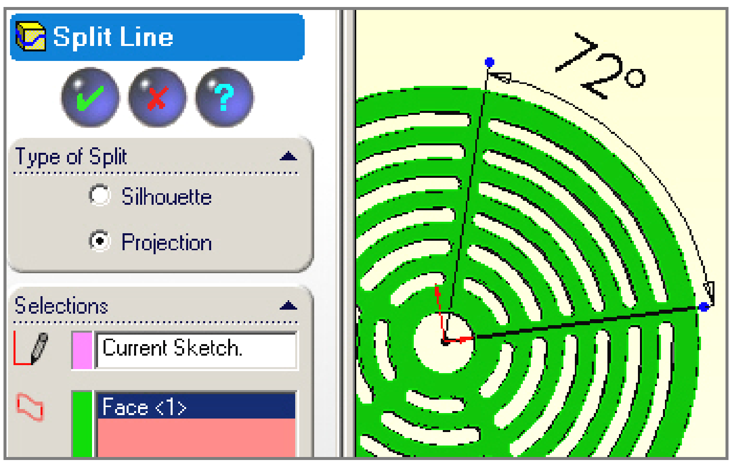
6.5.3.1 Split the surface

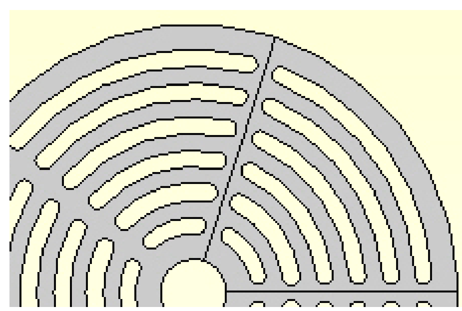
A full part model can require huge computer resources. You need to exercise care when generating the mesh. From the above preliminary studies (or engineering judgment) you only need a fine mesh in 36 degree segment, or the outer ring of that segment. To hold down the resources you should build a fine mesh in only one segment and specify large elements in the remaining 4/5 of the part. To do that you need to insert split lines to cut the front into a small and a large surface for assigning different element sizes.

1. Draw a radial straight line crossing over the 0 degree rib. Right click on the front face **Insert Sketch, Insert→Curve→Split Line.**



2. In the **Split line panel** use a **Projection Type of Split.** Repeat at the 72 degree spoke. When done you should see two lines creating the 1/5 and 4/5 surfaces.

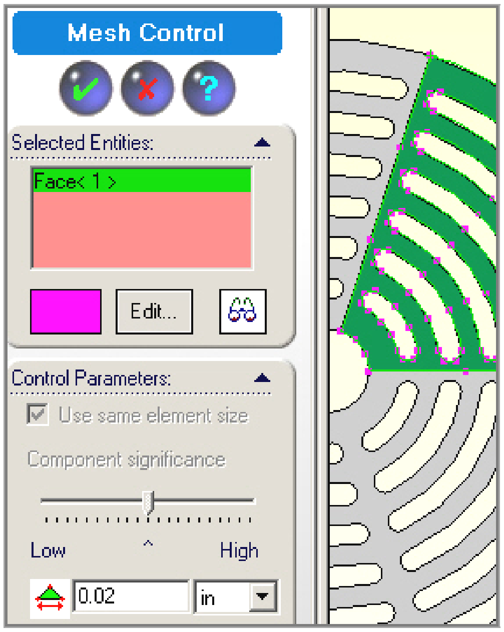


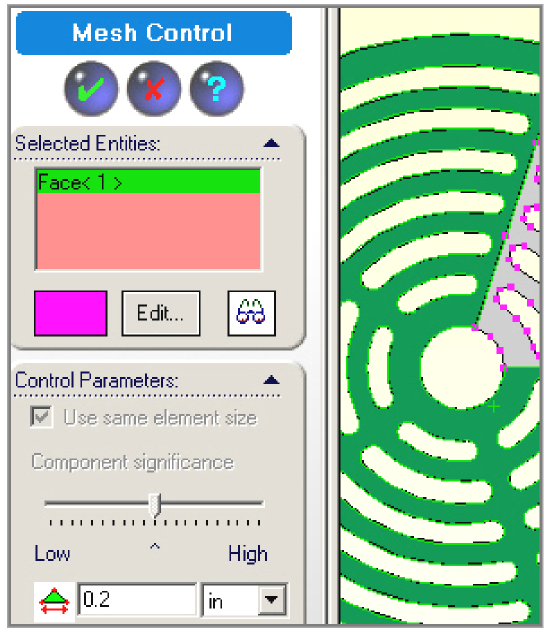


6.5.3.2 Controlling the mesh

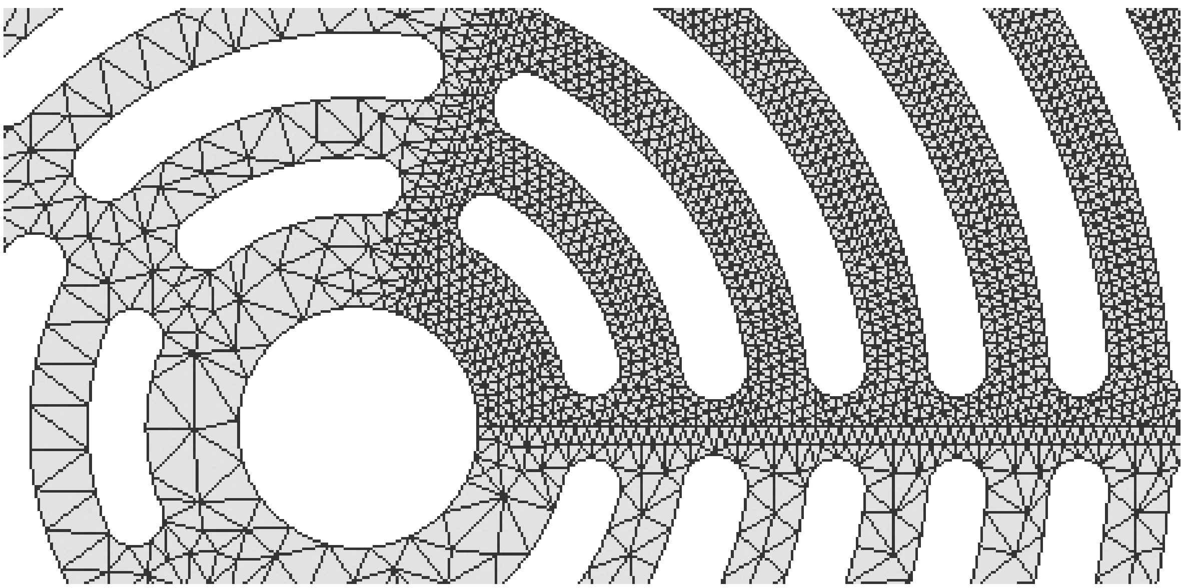
As noted above, you need a fine mesh only in one 72 degree segment (and maybe just in a 36 degree segment). In the CWManager:

1. Right click on **Mesh→Apply Control** to activate the **Mesh Control panel**. Note the default element size. Then pick the 1/5 region surface as the **Selected Entities** and change the element size to be smaller (0.02 inch here).





2. Next repeat the process for the lager surface area and use a much bigger element size (0.2 here), as shown above right.
3. Right click **Mesh→Create** to build the mesh (this will take some time).

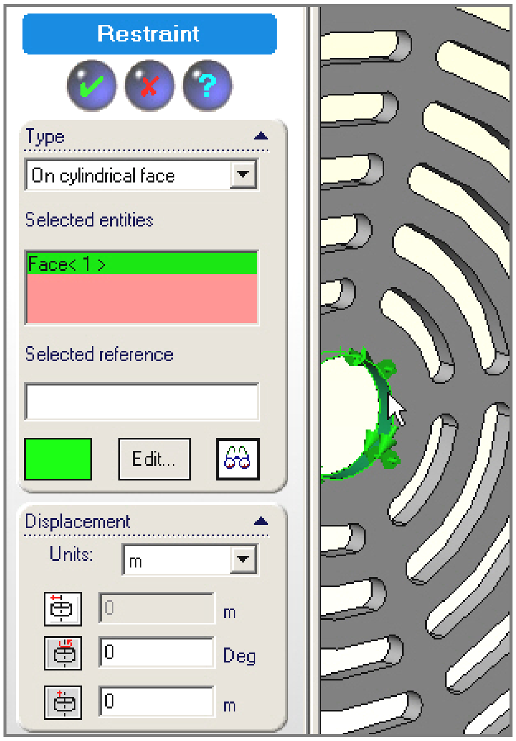


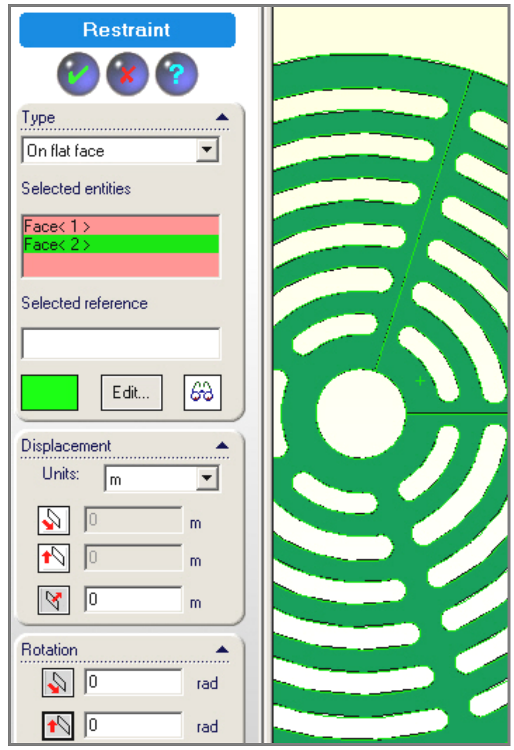
In this case the resulting mesh, includes almost 100,000 displacement components to compute (so you may run out of disk space). Note the transition region from the crude spoke mesh to the fine one.

6.5.4 Restraints

To eliminate the rigid body motion, along and about the axis:

1. Right click **Load/Restraint→Restraints** to open the **Restraints panel**. Select the cylindrical hole surface, set **Type** to **On cylindrical face.** Set the axial and circumferential displacement components to zero.



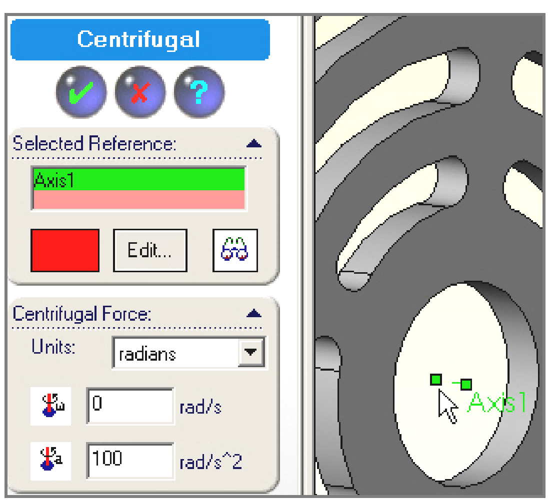


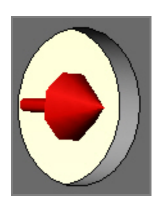
2. Since this is a quite large problem, it is wise to turn off the shell degrees of freedom (dof) that are zero (so they don't get computed). Pick the front face and set its normal displacement to zero, as well as the two in plane rotational components (illustrated above right).

6.5.5 Angular acceleration loading

As before, you just need to identify the axis and angular acceleration value:

1. Pick **View→Axes.**
2. Then right click **Load/Restraint→Centrifugal**
3. Pick **Axis 1** in the **Centrifugal panel** and type in the value of the acceleration. The direction of the angular acceleration vector is highlighted by a red arrow (according to the "right hand rule"). Next proceed to the solution.





6.5.6 Solution process

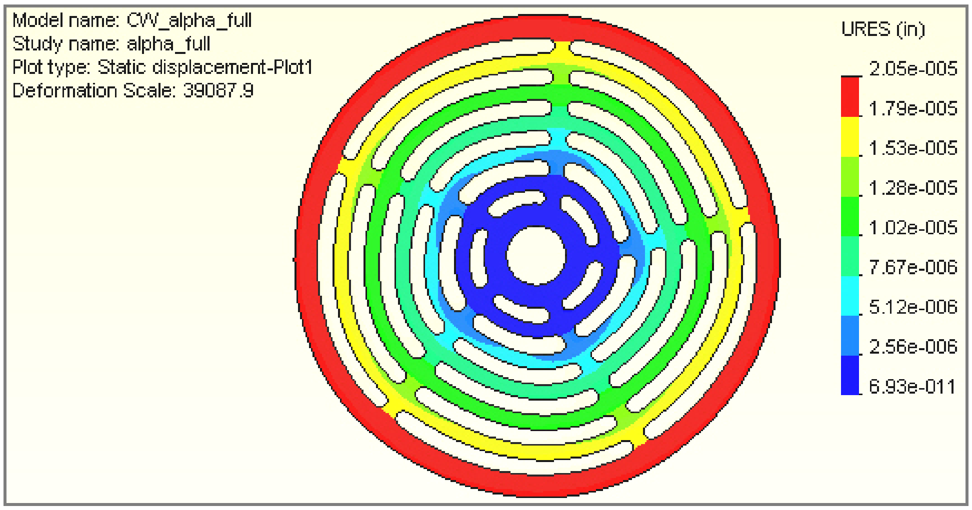
Even with this crude mesh over 4/5 of the part it has about 100,000 displacement components to compute. The results are summarized below. You will see that they compare well to the above approximations.

6.5.7 Post-processing

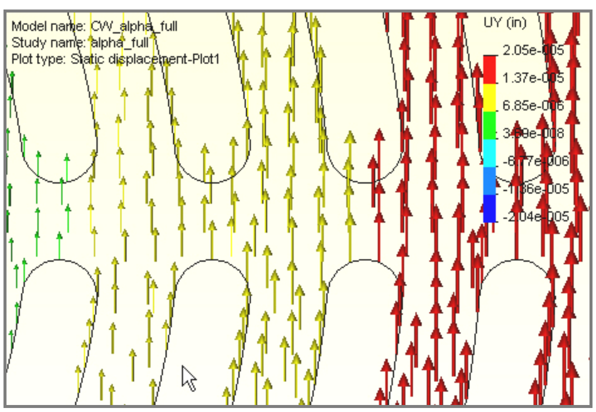
6.5.7.1 Displacements

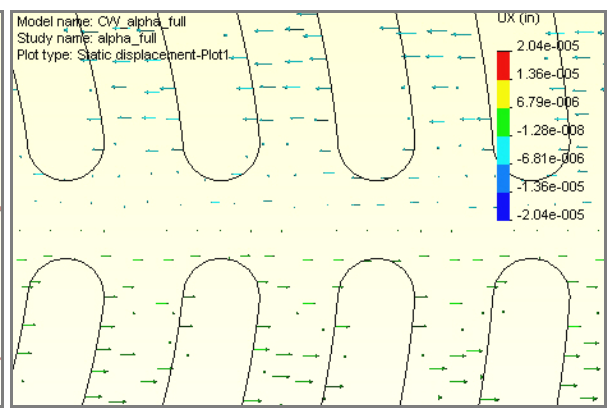
For the default displacement contour plot:

1. **Displacements→Plot1**. The displacement results are basically cyclically repeated every 72 degrees, as expected. The differences are due to the varying crudeness of the mesh that was used to reduce the computational effort.



2. To see the displacement vectors right click in the graphics area to **Edit Definition**.
3. In the **Edit panel** select the vertical component (**UY**), **Vector**, and **Line,** click **OK**. Also open **Vector Plot Options** to vary the arrow size and the percent of nodes displayed (while looking at the display).





Looking along the 0 degree line you see the displacement is mainly in the tangential (Y) direction and that the radial (X) displacement is basically zero, as assumed in 36 degree segment approximation. The contour of zero radial displacement (green line in Figure 6-12) continues through the coarse mesh region. That validates the assumptions used in the 36 degree model.

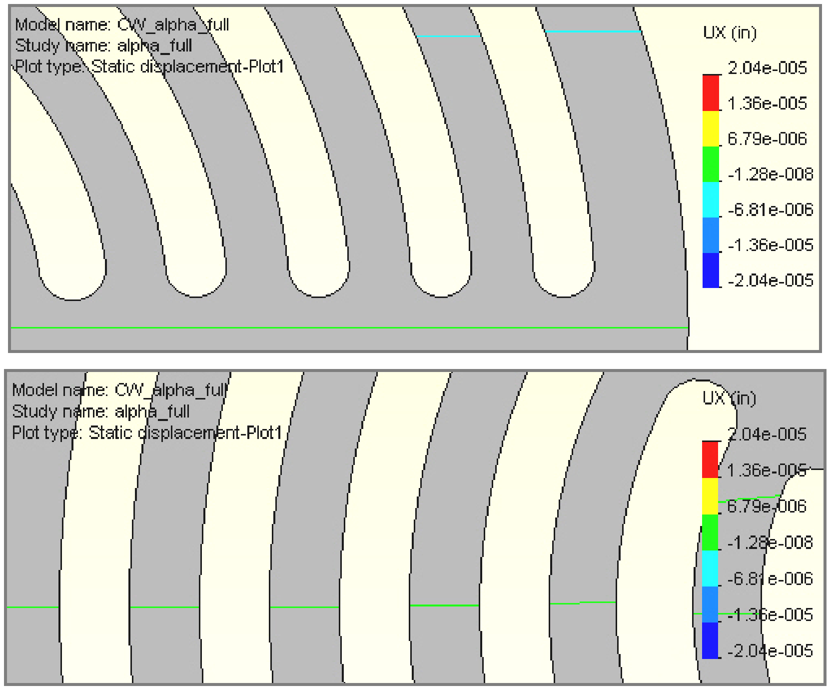


Figure 6-12 Checking radial displacement at spoke and arc centers

6.5.7.2 Stress results

The angular acceleration stress results are quite low, as before, and occur mainly around the innermost gap ends (Figure 6-13). That is to be expected since the angular acceleration of the outer mass (rings) causes the biggest shear stress in the first spoke segment.

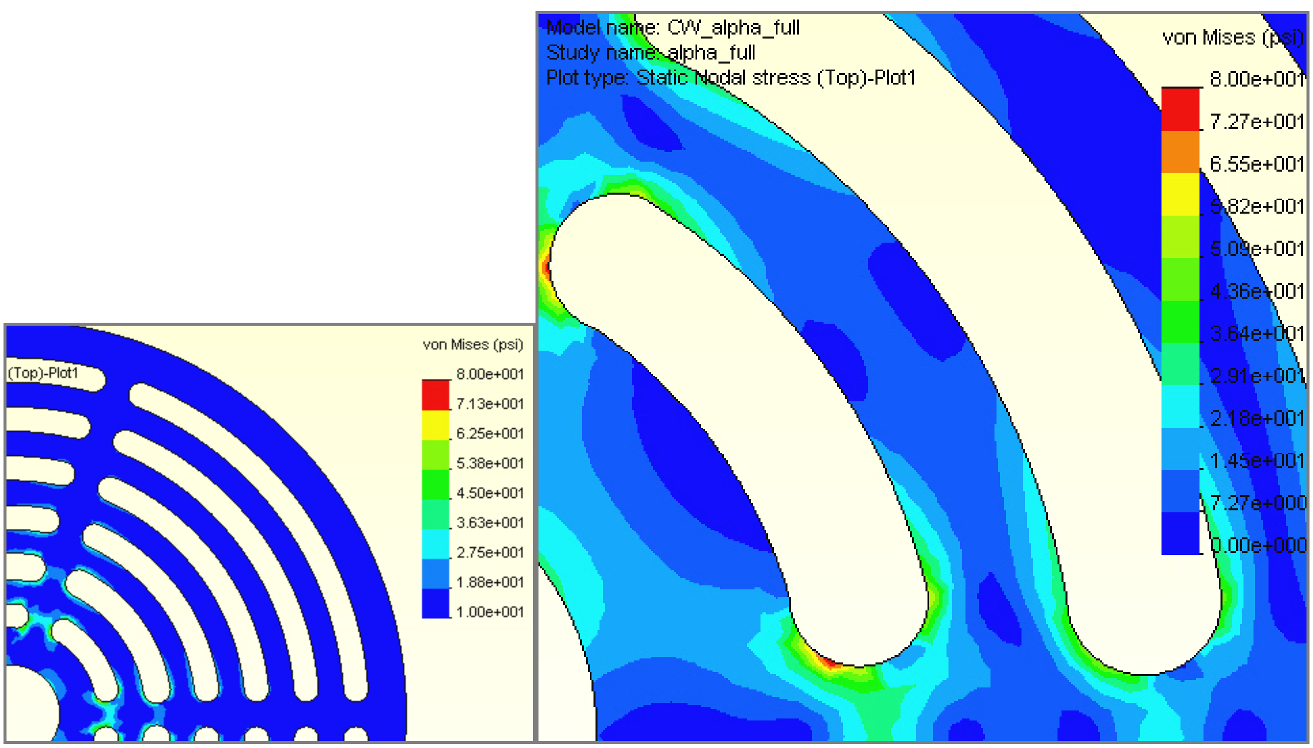


Figure 6-13 The von Mises failure criterion in the full part

6.5.8 Closure

These three studies show that you can often pick symmetry regions in rotational loadings and drastically reduce the computer resources required. Here it seems like you could refine the outer spoke mesh further to see angular velocity effects and refine the innermost gap ends to see the worst angular acceleration effects. Actually, the full model could have been replaced by the 72 degree segment using the Cosmos cyclic symmetry boundary condition.

The automation of a cyclic symmetry analysis requies that the software can express the degrees of freedom in terms of changing coordinate systems established tangential and normal to the repeated surface of cyclic symmetry. That is illustrated in Figure 6-14 where the top view of a cyclic symmetry impeller solid shows some of the pairs of tangential and normal displacements that have to be established and coupled by the analysis software. CosmosWorks includes this ability, but it is not demonstrated here.

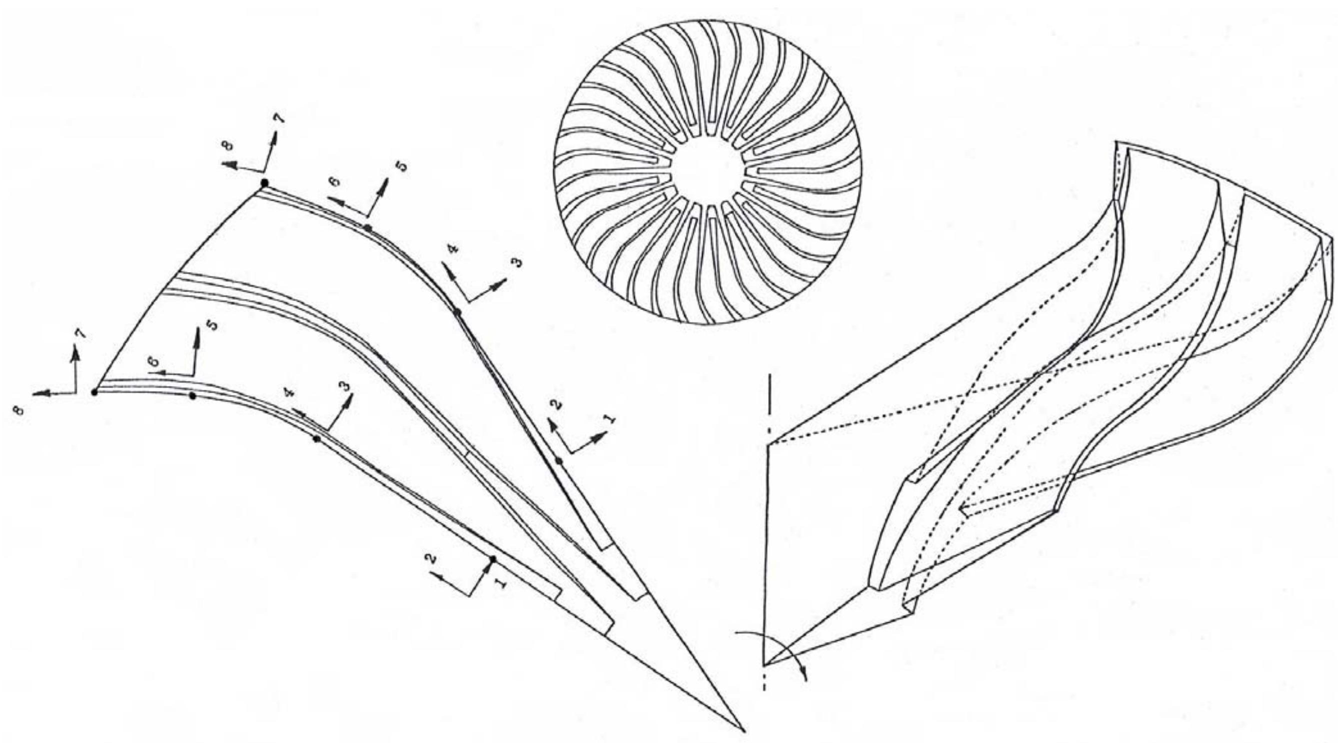


Figure 6-14 An impeller well suited for cyclic symmetry analysis

7 Flat Plate Analysis

7.1 Introduction

A flat plate is generally considered to be a thin flat component that is subjected to load conditions that cause deflections transverse of the plate. Therefore, the loads are transverse pressures, transverse forces and moment vectors lying in the plane. Those loads are resisted mainly by bending. It is assumed that in-plane membrane stresses are not present and that the transverse displacements are “small”. Generally, “small” is taken to mean a deflection that is less than half the thickness of the plate. If the deflection is larger than that and/or membrane forces are present you have to use a non-linear large deflection solution.

7.2 Rectangular plate

Figure 7-1 shows some of the boundary conditions that can be applied to the edges of a plate. A segment of a plate (as well as interior regions) can be fixed (encastre), free, or simply supported. A simply supported condition usually means that the transverse displacement is zero on that segment. A fixed supported condition usually means that the rotation vector tangent to the segment is also zero.

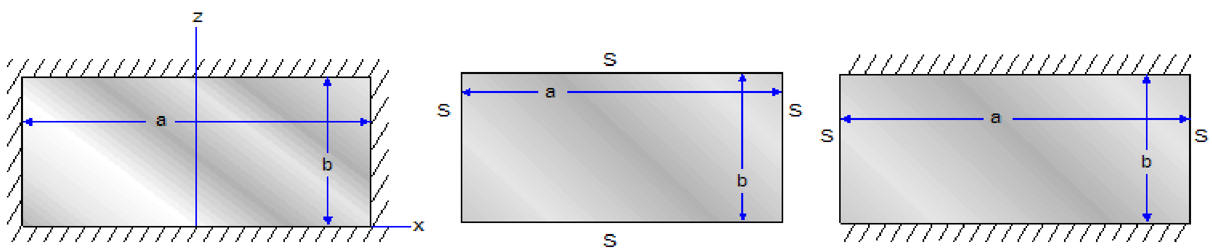


Figure 7-1 Some boundary condition options on rectangular plates

In this section the classic example of a simply supported plate subjected to a uniform transverse pressure will be illustrated. Quarter symmetry will be utilized to illustrate symmetry boundary conditions for an element with displacement and rotational degrees of freedom. A short story about this case will be noted at the end. The example plate is steel, with dimensions of 4.68 by 12.68 by 0.08 inches and is subjected to a uniform pressure of 100 psi. The total force is about 310 lb, so you expect the edge reactions to be equal and opposite. Since external edge effects are usually important, a finer mesh is employed along those edges. The plate is set to be of the “thin” type and the study is executed.

7.3 Post-processing

7.3.1 Displacements and rotations

For the two symmetry edges the restrained displacement components lie in the plane of the plate. The displacement perpendicular to the edge is zero and the rotation tangent to the edge is zero. These are seen along with the mesh in Figure 7-2. The plate deflections are given in Figure 7-3. The surface deflections are given as contours. The short symmetry edge deflection is graphed for more detail in the lower image. The graph starts at the outer (zero deflection) edge and goes to the maximum deflection at the center (zero rotation) point. It serves to verify that the restraints were properly applied. The center point deflection can also be compared to analytic estimates [14, 17]. Here the maximum computed deflection is less than half the thickness of the plate, therefore the small deflection assumption appears correct (and Cosmos did not issue a warning about the change in stiffness due to perceived large deflections).

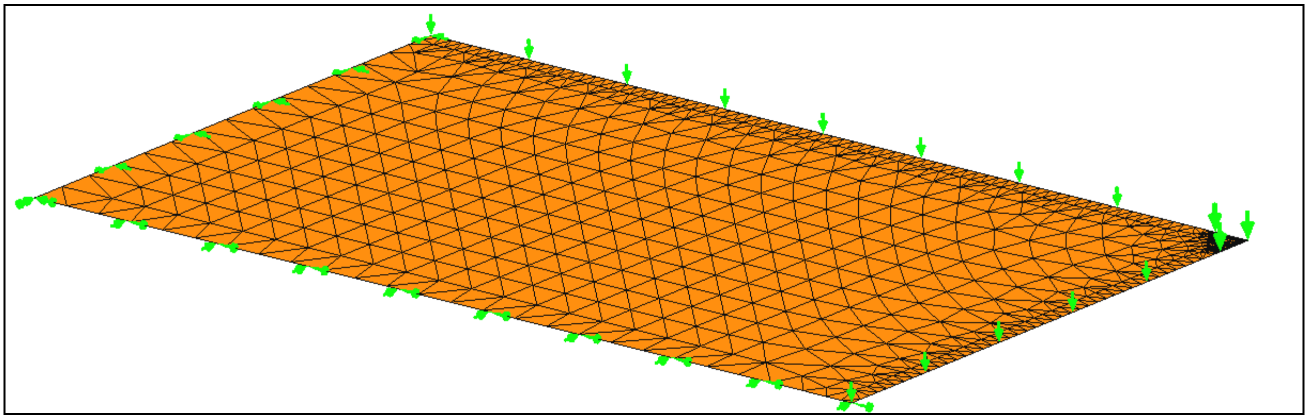


Figure 7-2 Boundary restraints and mesh for a quarter symmetry rectangular plate

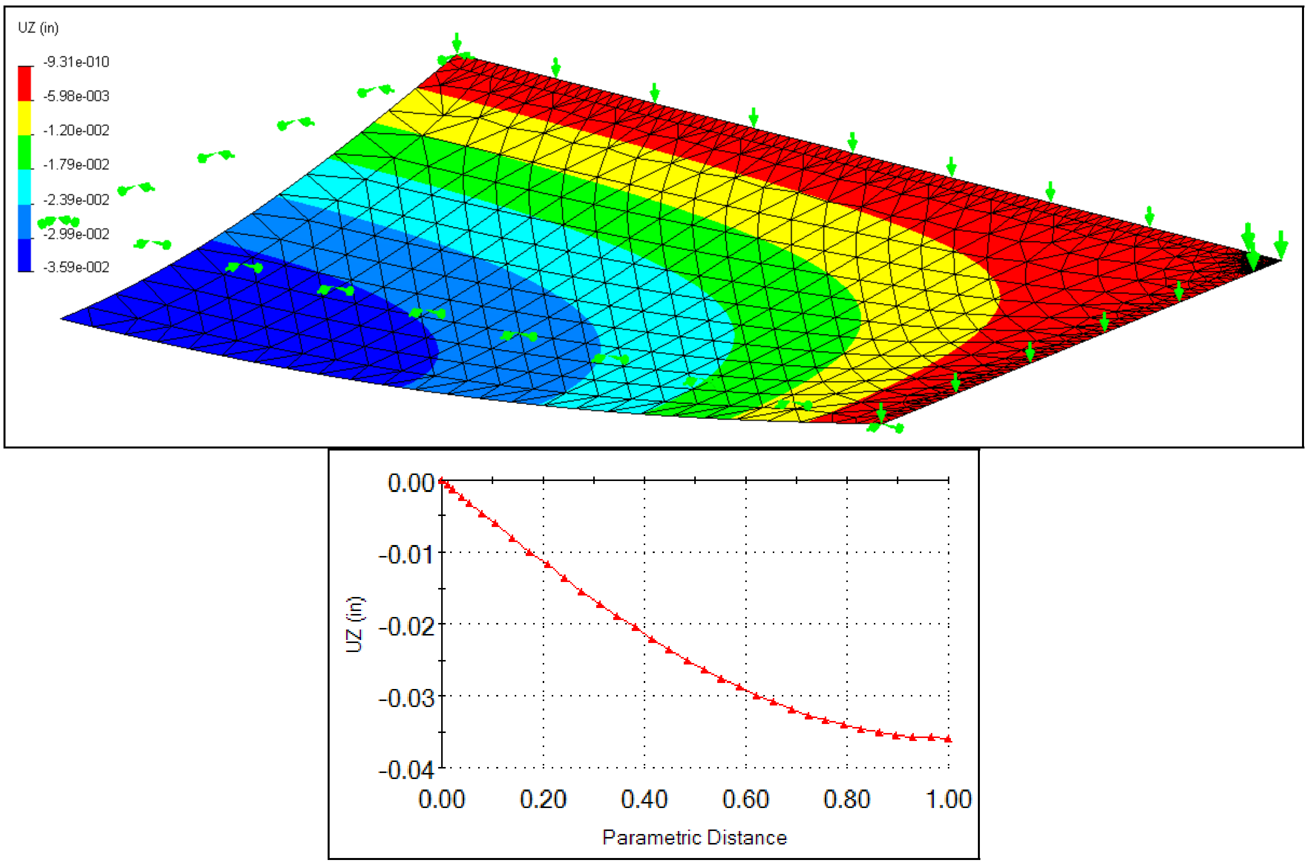
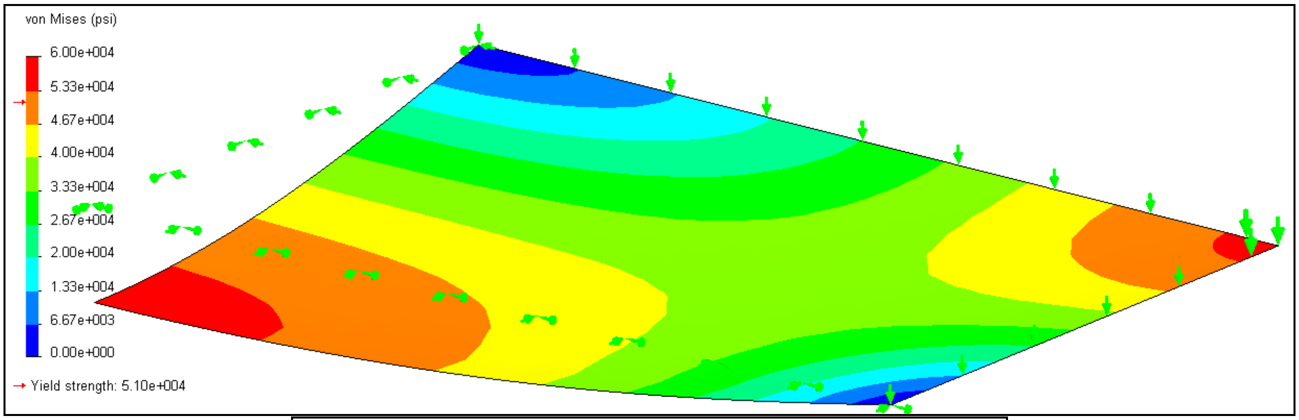


Figure 7-3 Deflections of the quarter symmetry plate, and its short edge

7.3.2 Stresses

Since plates and shells can be subjected to both bending and membrane (in-plane) stresses the stress results should be checked on the top, bottom, and middle surfaces. Here the membrane stress is zero (for small deflections). At a point on the plate it will be in tension on one side and have an equal amount of tension on the other. That is important when the material has different strengths in tension and compression (like concrete).

The von Mises effective stress is proportional to the square root of the sum of the squares of the differences in the principal stresses, so it is always positive. The surface, and short symmetry edge, values of the von Mises stress are given in Figure 7-4. Note that the peak values exceed the yield stress, and the factor of safety (FOS) is less than unity.



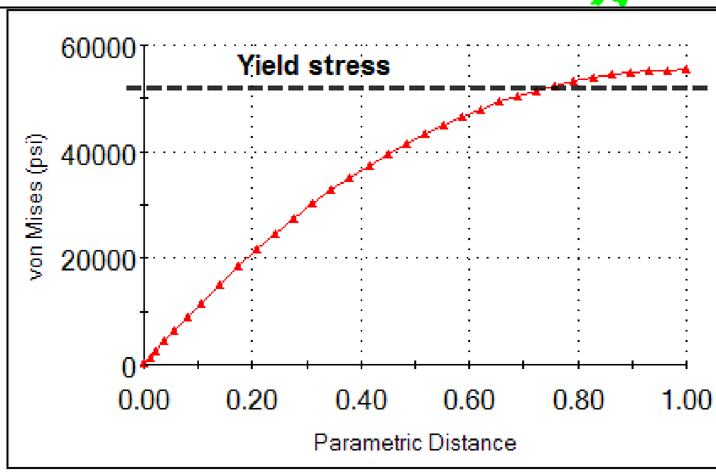
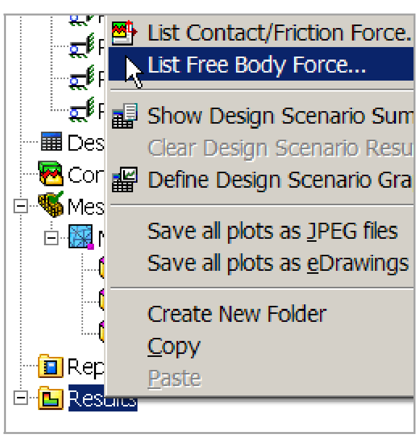


Figure 7-4 Von Mises stress in quarter symmetry plate and its short symmetry side

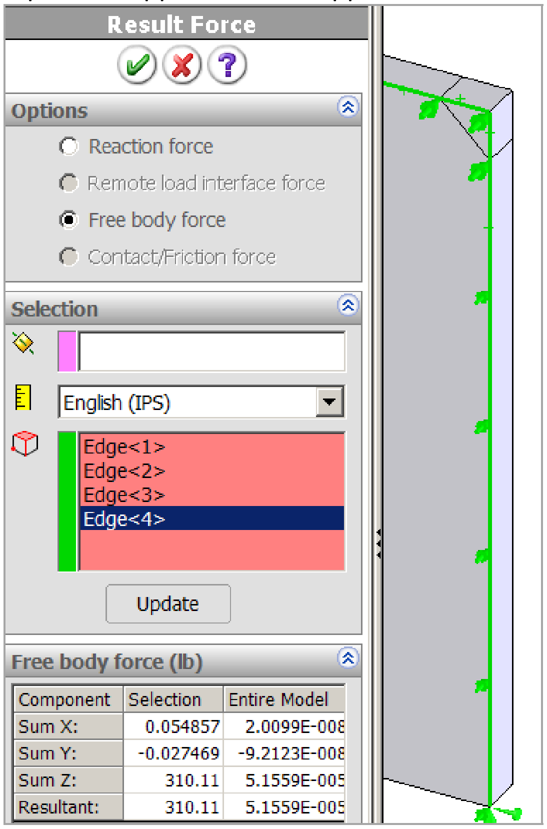
7.3.3 Surprising reactions

The reactions can be recovered in at least two ways. The approach using a free body diagram calculation is shown here. First the transverse reaction force on the full two supporting edges are recovered and found to be equal and opposite to the resultant applied force from the pressure.

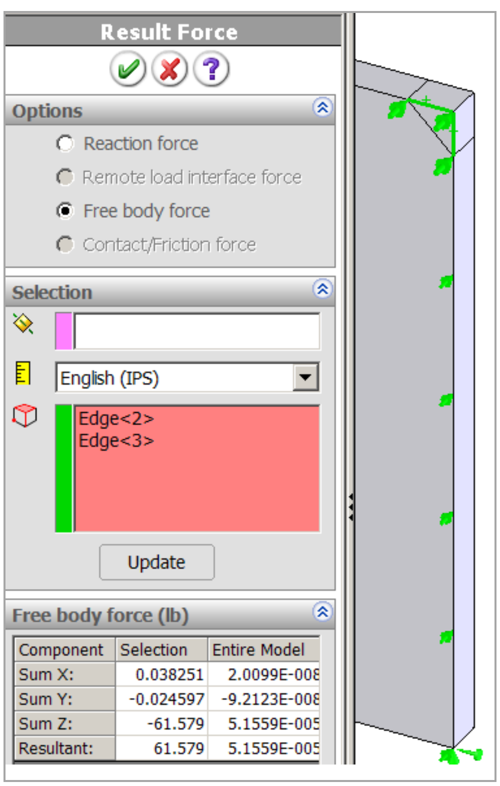
1. Right click on **Results→List Free Body Force** to open the **Reaction Force panel.**

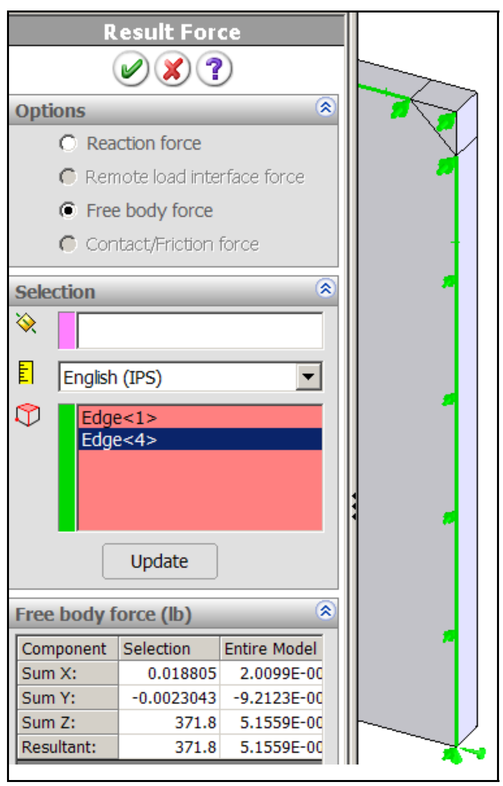


2. First select all the supporting edges to get the total reaction forces. They total about 310 lb., which is equal and opposite to the applied resultant force.



3. To see how much the small corner support region contributes select its small edges and then compare its reaction to the remainder of the two edges





The resultants on a free body diagram come from the integral of the reaction force per unit length of the edge restraints. The reaction per unit length is not constant and will vary in a complicated fashion. Knowing this, the support edges were split to introduce shorter edges at the corner. That lets you find the portion of the reactions coming from the small corner segments. There the corner reaction force has a different sign (it is in the same direction as the pressure) and a value of about 20 % of the applied force from the pressure.

7.3.4 Reaction discussion

Now for the related side story: A large analysis group had run the above problem to test a new finite element system that they had recently installed. I was called in as a consultant to fix an “error” they had found. Specifically, when a pressure load was applied downward to a flat plate some of its reactions were also found to be acting downward, just as noted above and in Figure 7-5. That seemed to them to be physically impossible. I stated that the software was giving the proper type of response since elementary plate and shell theory shows that the edge reactions per unit length must behave in that fashion. To help understand why, I had them plot the two non-zero top principal stresses (Figure 7-6) as well as the deflection and maximum (top) stress along the diagonal from the center point to the support corner, like Figure 7-7.

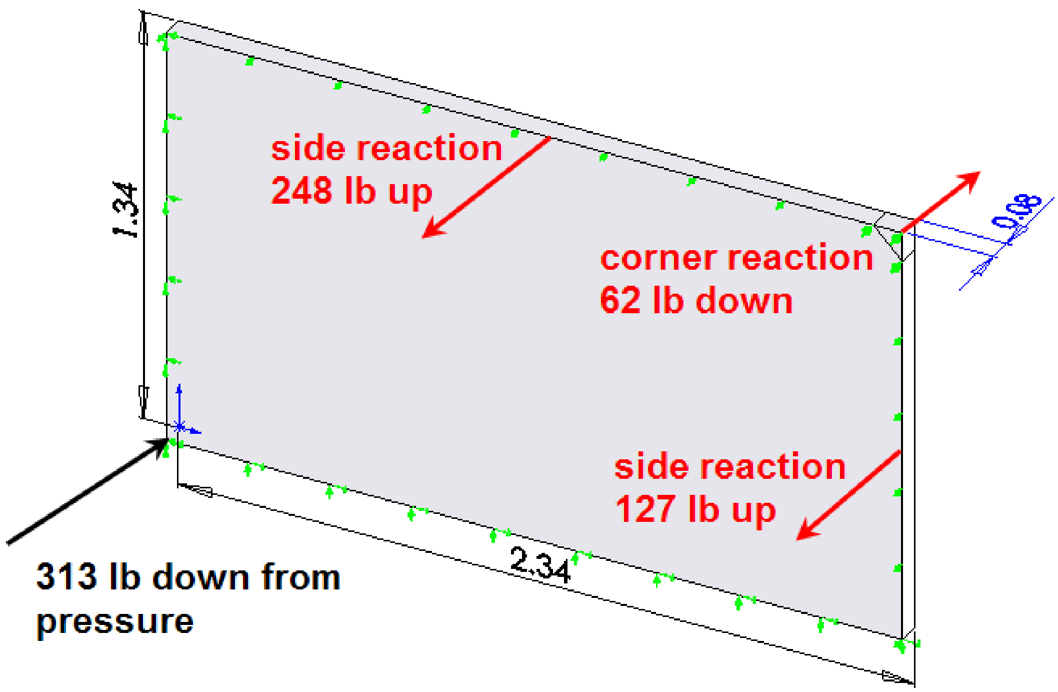


Figure 7-5 Approximate resultant applied force and reaction forces on the plate

The deflection plot in Figure 7-7 shows that the deflection curvature reverses its sign as it approaches the corner. The corner would lift up, but the *assumed* edge restraint requires that it not move. Therefore, tension forces must develop in the corner reactions to pull it down in the *assumed* restrained position. If the material along the restraint edge is capable of developing a resisting downward force, then you have the correct solution to the actual problem. Otherwise, you have the solution to the *assumed* problem. Unfortunately, many finite element studies give results for the assumed part behavior instead of the actual part behavior. Then, the plots are pretty, but wrong.

If the edges of the plate are simply sitting on top of to walls, then the wall could not pull down on the corner. An air gap would open; the corner would lift up off the wall, and all line reactions would be in compression where the plate remains on the wall. Sometimes you can actually see this corner lift off behavior in thin acoustical ceiling tiles. How much of the corner actually lifts off the wall must be computed from an iterative contact analysis.

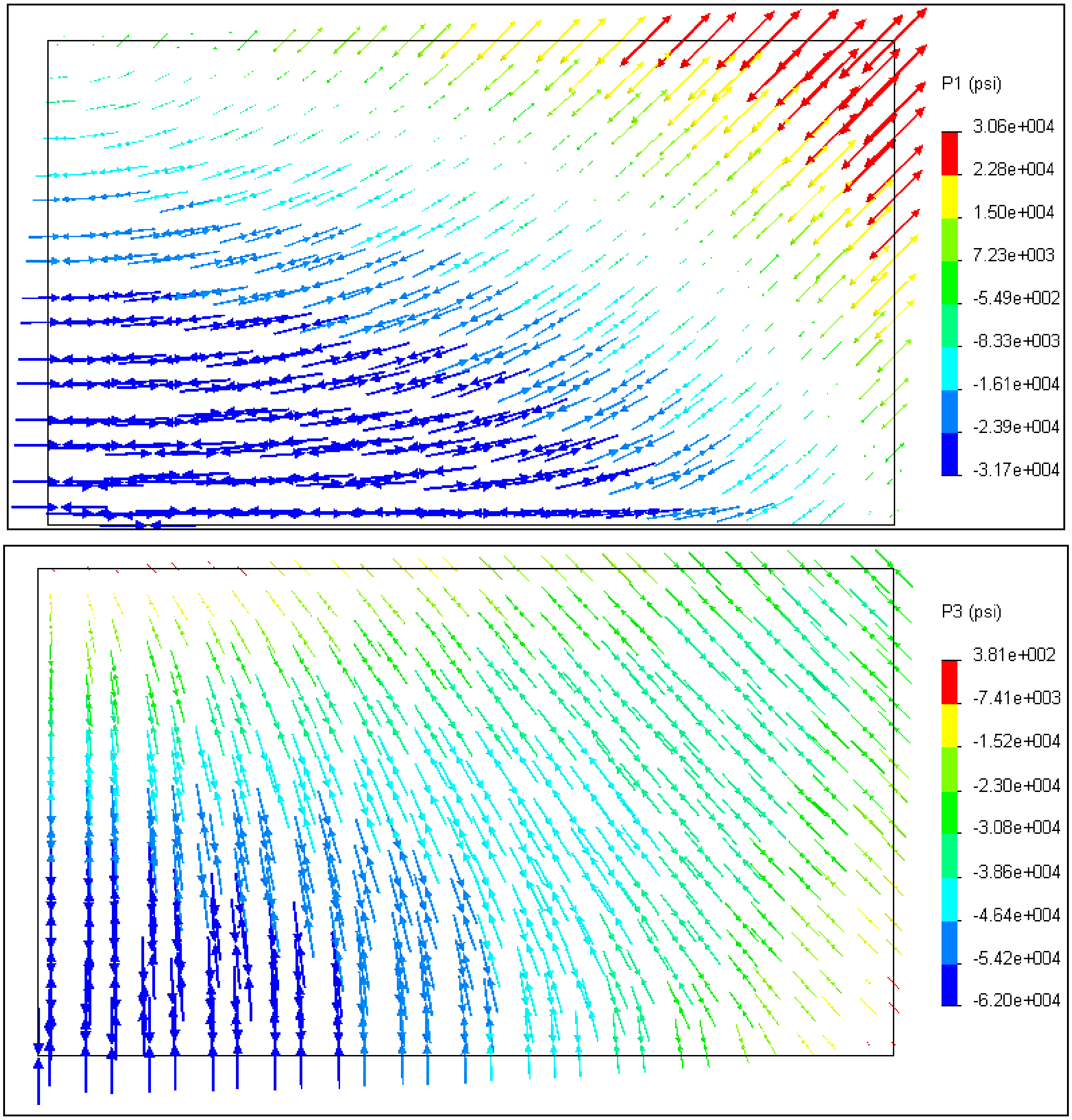


Figure 7-6 The principal stresses on the top of the quarter symmetry plate

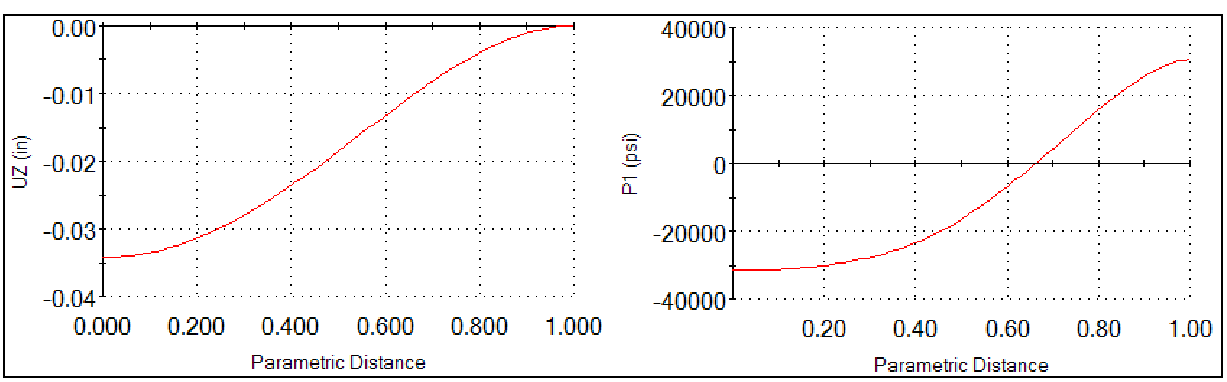


Figure 7-7 Transverse deflection and P1 stress along the center to corner diagonal

7.4 Edge support contact analysis

To illustrate the other result where the plate can lift off the support corner a contact analysis was executed with a relatively crude solid mesh. In Cosmos, a contact analysis requires the contact regions to be faces, not edges. That requires you to state the width of each supporting wall and its location relative to the plate. Here the wall exterior is assumed to match the original plate edges. The two new interior split lines in Figure 7-8 show the interior limits of the contact support areas.

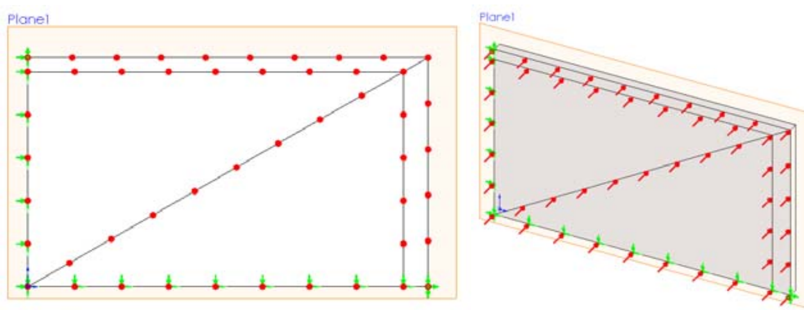
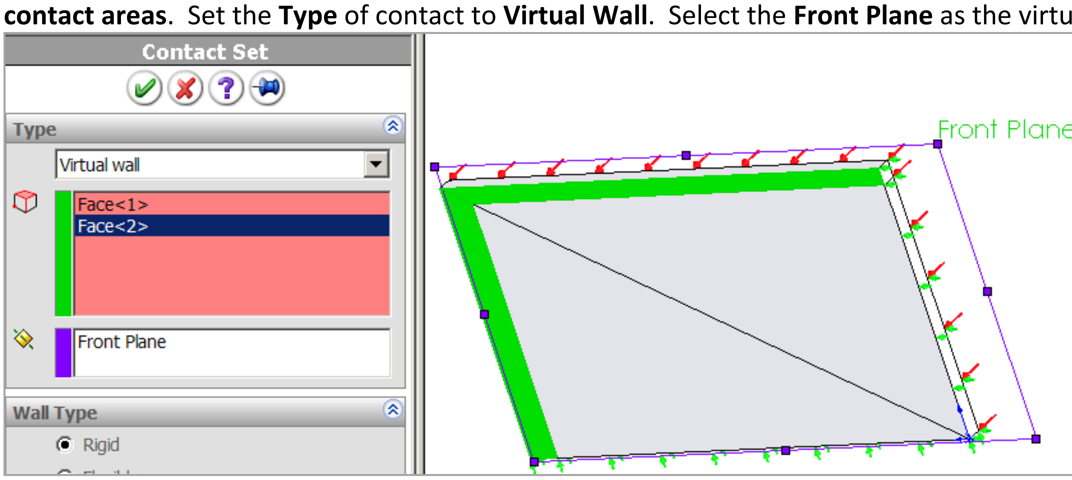


Figure 7-8 Solid plate approximation with two wall thickness split lines

To select the contact areas of the plate and their virtual mating with the Front plane:

1. Cosmos **Menu→Contact/Gaps** to open the **Contact Set panel**. Pick the **two faces** to be the plate **contact areas**. Set the **Type** of contact to **Virtual Wall**. Select the **Front Plane** as the virtual contact.



2. Create the **Mesh**, and **Run** the study, and observe the convergence of the iterations (Figure 7-9).

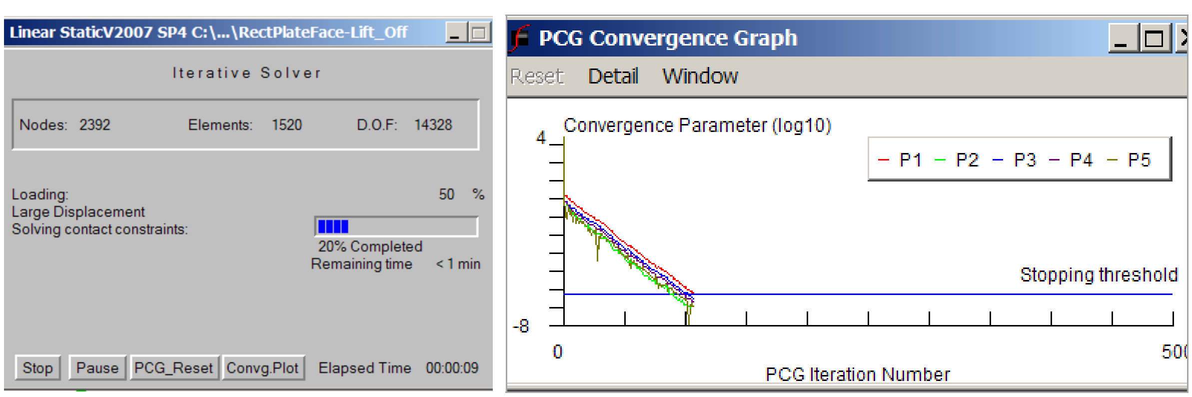


Figure 7-9 Viewing the non-linear contact solution iterations

7.4.1.1 Lift off displacements

Now, all of the reactions are compressive (acting against the applied pressure). The graphs of the plate deflections along the inside walls, in Figure 7-10, show that about 30% of the plate along the long wall lifts off, as it does along about 50% of the short wall. A graph of the transverse displacement along the diagonal from the center point to the corner point in Figure 7-11 clearly shows that the corner region lifts off the support walls (has a positive displacement).

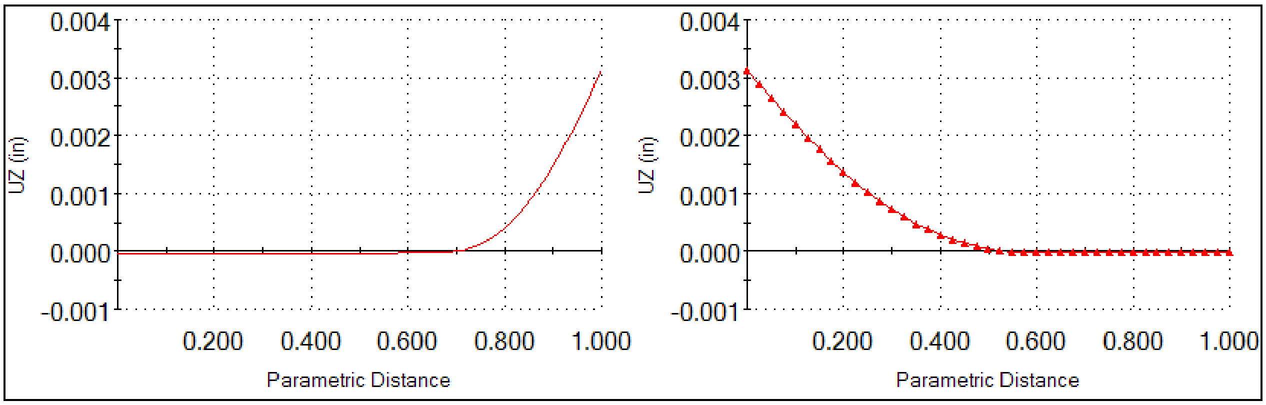


Figure 7-10 Plate deflections along the long and short support walls

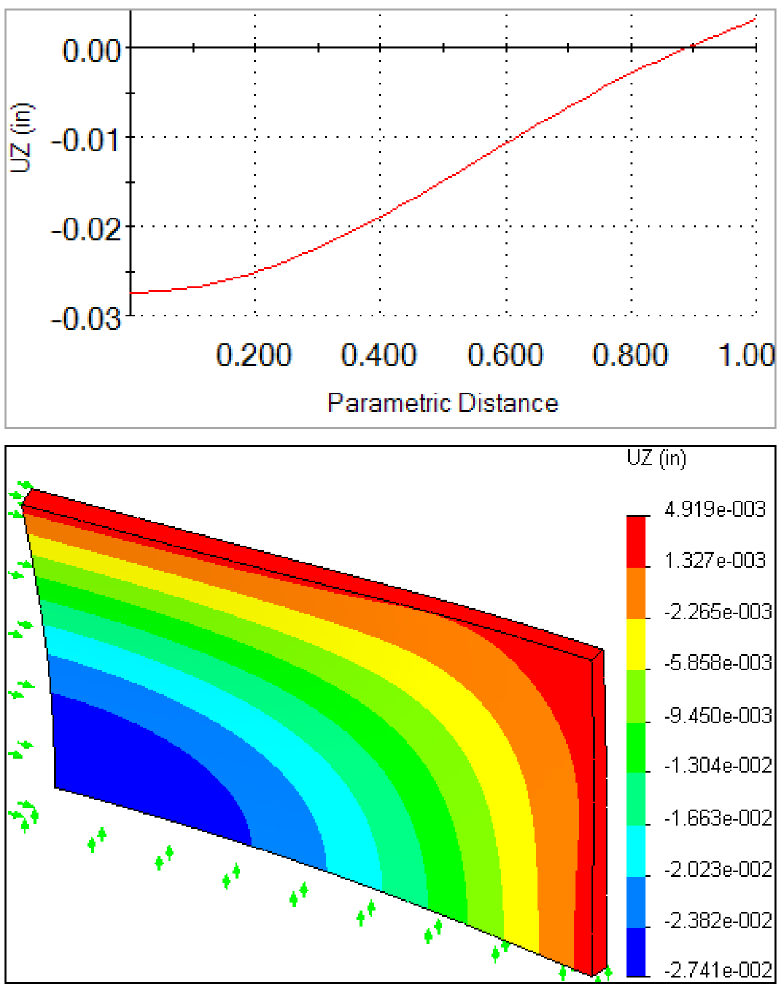


Figure 7-11 Center to corner and surface transverse displacements

7.4.1.2 Lift off stresses

Compared to Figure 7-4, the new (crude model) von Mises stresses in Figure 7-12 are quite low in the corner, when the plate is allowed to lift off. The peak value is now slightly less than the yield stress but the FOS is still too low.

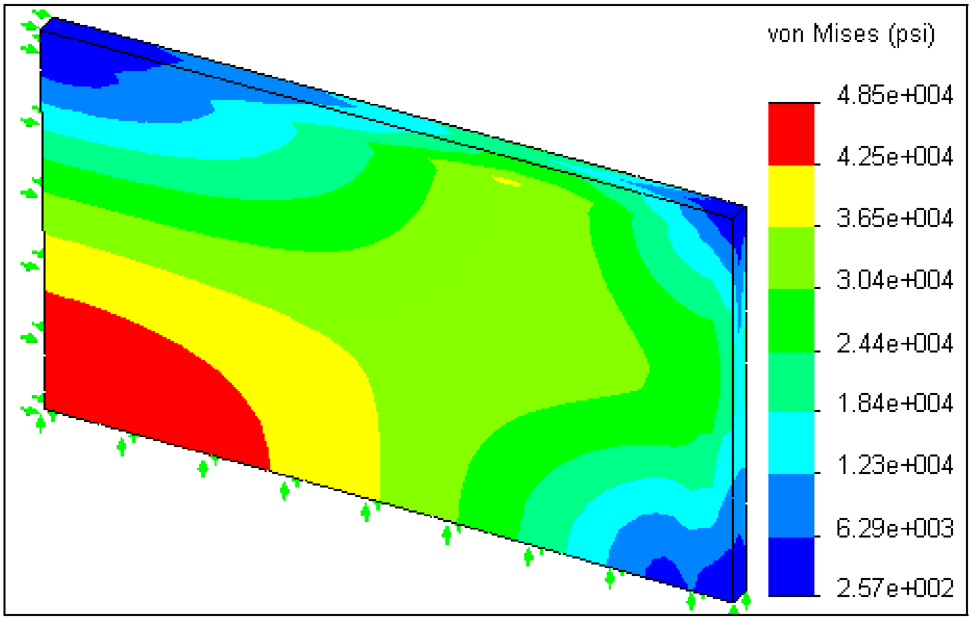


Figure 7-12 Plate von Mises stresses with corner lift off

7.4.2 Closure

If you did not have a contact analysis capability you could still get a reasonable answer to the lift off analysis. To do that you could introduce a split line on each edge near the corner (with parametric dimensions). Let the short end of each corner line be unsupported, solve the problem and check the reactions. If any negative reaction forces appear, then move the split line away from the corner and repeat the process. It may be a slow procedure, but it can lead you to the correct lift off regions.

8 Shell Analysis

8.1 Introduction

Within CosmosWorks there are two different options for creating shells: by part mid-surface, and by selecting specific surfaces. Creating shell surfaces by having them follow the mid-surfaces of connecting solid parts is usually the best ways to define shells. Cosmos does only a fair job of implementing this approach. Unlike other commercial codes it can not currently (2007) find the middle surface of solids that fork into two or more domains. Thus, Cosmos can not even automatically generate the mid-surface of a “T” shaped part and connect the three surfaces at their common junction. That can be a major short coming for injection molded parts which tend to have a large number of thin intersecting reinforcing ribs. The missing mid-surface shell regions near junctions can be manually constructed using the SolidWorks tools “extend surface”, “knit”, and “loft”, but it is a very slow process.

The two classes of shell selection offer the same restraints, but they are applied in different ways. Table 8-1lists the current restraint options for mid-surface shells within CosmosWorks. Restraints on an edge of a mid-surface shell are applied to the face of the solid that contains the edge. Note that they can involve components of the displacement and rotation vectors. The loading options for mid-surface shells are given in Table 8-2. They are also applied to the face of the solid that contains the edge.

When a part clearly has a mid-surface description most finite element systems will generate a variable thickness shell element by interpolating the thickness at each mesh node. Cosmos is an exception. The mid-surface shell option only works correctly if the part every where has a constant thickness. When you select that option, Cosmos issues a caution message (Figure 8-1) that you should view as a constant thickness requirement.

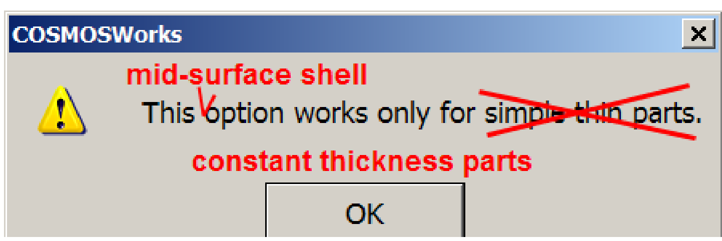


Figure 8-1 Mid-surface shell models require a constant thickness part

Piecewise constant thickness shell models, or 2.5D plane stress models, are available with the CosmosWorks selected surface shell mesh. Table 8-3 lists the current restraint options for selected surface shells within CosmosWorks. Restraints on an edge of a selected-surface shell must be applied by selecting that specific edge. They involve components of the displacement and rotation vectors. The loading options for selected surface shells are given in Table 8-4. They are also applied to the specific edge.

Table 8-1 Restraints for mid-surface shell stress analysis

Restraint Type	Mid-surface Shell Definition
Fixed	All translations and rotations are zero on an edge, or vertex. .
Hinge	On a cylindrical face, only the circumferential displacement is allowed.
Immovable	All three translations are zero on a face, edge or vertex.
On cylindrical face	The cylindrical coordinate displacements and rotations normal to and/or on the cylindrical surface are given.
On flat face	Displacements and rotations normal to and/or tangent to the flat face are specified.

On Spherical face	The spherical coordinate displacements and rotations normal to and/or on the spherical surface are given.
Roller/Sliding Symmetry	Two displacements tangent to a flat face and the rotation normal to the flat face are allowed.
Use reference geometry	A face, edge, or vertex can translate and or rotate a specified amount relative to a reference plane and axis.

Table 8-2 Load conditions for mid-surface shell stress analysis

Load Type	Mid-surface Shell Definition
Apply force	The total force on a mesh face is specified, or given on a side face or edge to define the mid-surface edge or vertex value.
Apply moment	The total moment on a mesh face is specified, or given on a side face or edge to define the mid-surface edge or vertex value.
Apply normal force	The total force normal to a face, at its centroid, is specified and converted to an equivalent pressure.
Apply torque	The total torque on a face is specified with respect to an axis and converted to an equivalent pressure.
Bearing Load	On a cylindrical surface give the total force in a Cartesian X or Y direction to convert to a sine distribution pressure.
Centrifugal	The angular acceleration and angular velocity are given about an axis, edge, or cylindrical surface.
Connectors	See CosmosWorks help files for bolts, pins, spot welds, etc.
Gravity	The gravitation acceleration value is given and oriented by an axis, edge, or a direction in or normal to a selected plane.
Remote load	See CosmosWorks help files.
Temperature	Not recommended. Transfer from thermal analysis.

Table 8-3 Restraints for picked surface shell stress analysis

Restraint Type	Picked-surface Shell Definition
Fixed	Translations and rotations are zero on surface edge or vertex.
Hinge	Circumferential displacement is allowed on a cylindrical face.
Immovable	All three translations are zero on surface, its edge, or vertex..
On cylindrical face	The cylindrical coordinate displacements and rotations normal to and/or on the cylindrical surface are given.
On flat face	Displacements and rotations normal to and/or tangent to the flat face are given.
On Spherical face	The spherical coordinate displacements and rotations normal to and/or on the spherical surface are given.
Roller/Sliding Symmetry	Two displacements tangent to a flat face and the rotation normal to the flat face are allowed.
Use reference geometry	A face, edge, or vertex can translate and or rotate a specified amount relative to a reference plane and axis.

Table 8-4 Load conditions for picked surface stress analysis

Load Type	Picked-surface Shell Definition
Apply force	The total force on a mesh face is specified. Or, given on the picked surface edge or vertex value.
Apply moment	The total moment on a mesh face is specified. Or, given on the picked surface edge or vertex value.
Apply normal force	The total centrodial force normal to a picked surface face is specified and

	converted to an equivalent pressure.
Apply torque	The total torque on a picked surface face is specified with respect to an axis and converted to an equivalent pressure.
Bearing Load	On a picked cylindrical surface give the total force in a Cartesian X or Y direction for a sine distribution pressure.
Centrifugal	The angular acceleration and angular velocity are given about an axis, edge, or cylindrical surface.
Connectors	See CosmosWorks help files for bolts, pins, spot welds, etc.
Gravity	The gravitation acceleration value is given and oriented by an axis, edge, or a direction in or normal to a selected plane.
Remote load	See CosmosWorks help files.
Temperature	Not recommended. Transfer from thermal analysis.

8.2 Quarter Symmetry Tank Stress

8.2.1 Introduction

You need to carry out the stress analysis of an outdoor water tank. Since it has quarter symmetry you can start by building only one-fourth of the geometry. Assume that the side wall has height of 72 inches and a small lip that extends below the tank bottom for 3 inches. The lower lip will give you more realistic options on how you may need to restrain the part. The dimensions of the tank bottom and the final quarter symmetry part are seen in Figure 8-2. The details of constructing this shell are given in Appendix A.

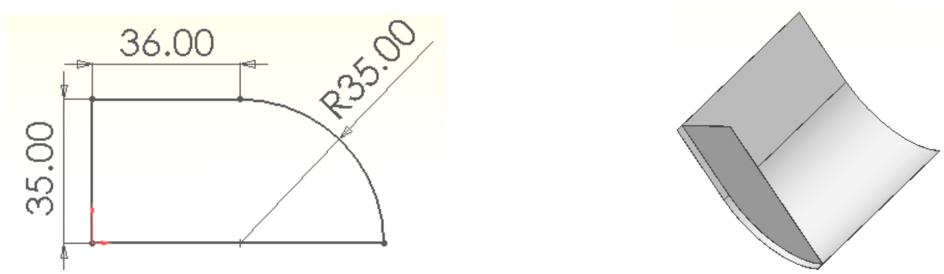


Figure 8-2 The final quarter symmetry tank part

8.2.2 CosmosWorks static studies

8.2.2.1 CosmosWorks Manager

At this point you are ready to move into the Cosmos finite element study interface by selecting the **CosmosWorks Manager** (CWManager) icon:

1. Right click on the top name to access **Study** which opens the **Study panel**.
2. Assign a **Study name**, choose **Static** for the **Analysis type.**
3. Define the **Mesh type** to be **mid-surface shells**.

8.2.2.2 Define the material

At this point **Mid-surface Shell** will appear in the CWManger menu:

1. Right click on it to apply material data to the shell. The tank is to be made of galvanized steel.
2. Pick **Apply Material to All→ Material panel→From library files button→Steel** and select galvanized steel, set the **Units** to English.

Material name: Galvanized Steel

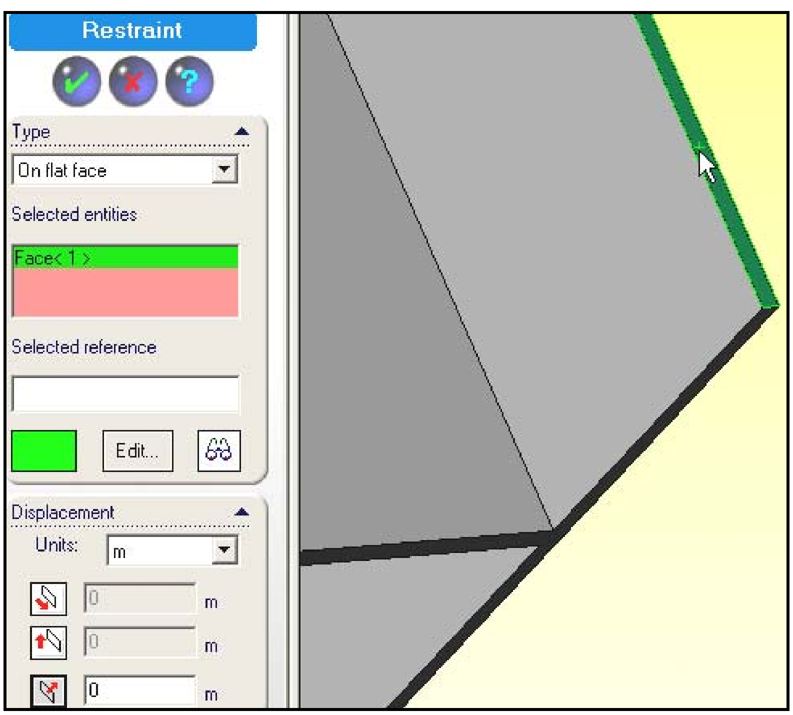
Property	Description	Value	Units
EX	Elastic modulus	29012512	psi
NUXY	Poisson's ratio	0.28999999	NA
GXY	Shear modulus		psi
DENS	Mass density	0.28432184	lb/in^3
SIGXT	Tensile strength	51772.927	psi
SIGXC	Compressive strength		psi
SIGYLD	Yield strength	29584.531	psi

Note that the yield strength (SIGYLD), taken from a uniaxial tension test, is about 29.6e3 psi. Since you selected a ductile material, that material yield property will later be compared to the von Mises, or effective, stress. Our Safety of Factor (for this material) will be this yield stress property divided by the von Mises stress.

8.2.2.3 Initial Restraints

Remember that the actual displacement supports (restraints) can be unclear and you usually need to check for a few possibilities. What looks like minor changes in the restraints of a part can cause large changes in the displacements and/or stresses. Also, remember that in a static analysis you must always provide enough restraints to prevent all of the six rigid body motions (RBM) possible in a three-dimensional part. In this example you will use an initial set of restraints, carry out the analysis, evaluate the study, and add new restraints for an additional analysis. Here, begin by supporting the bottom tank edge against vertical motion (only). That prevents three rigid body motions: motion in the vertical direction and rotation about the two horizontal axes. Eliminate the three RBM that remain:

1. Right click **Load/Restraint→ Restraints** to get the **Restraint panel**.
2. Zoom in on the bottom wall and pick the flat bottom face and enforce no displacement perpendicular to it. Pull down to **On flat face** to set the **Type** of entity being restrained, then in the zoomed graphics window you pick the flat bottom edge surface so it enters the **Selected entities** list.

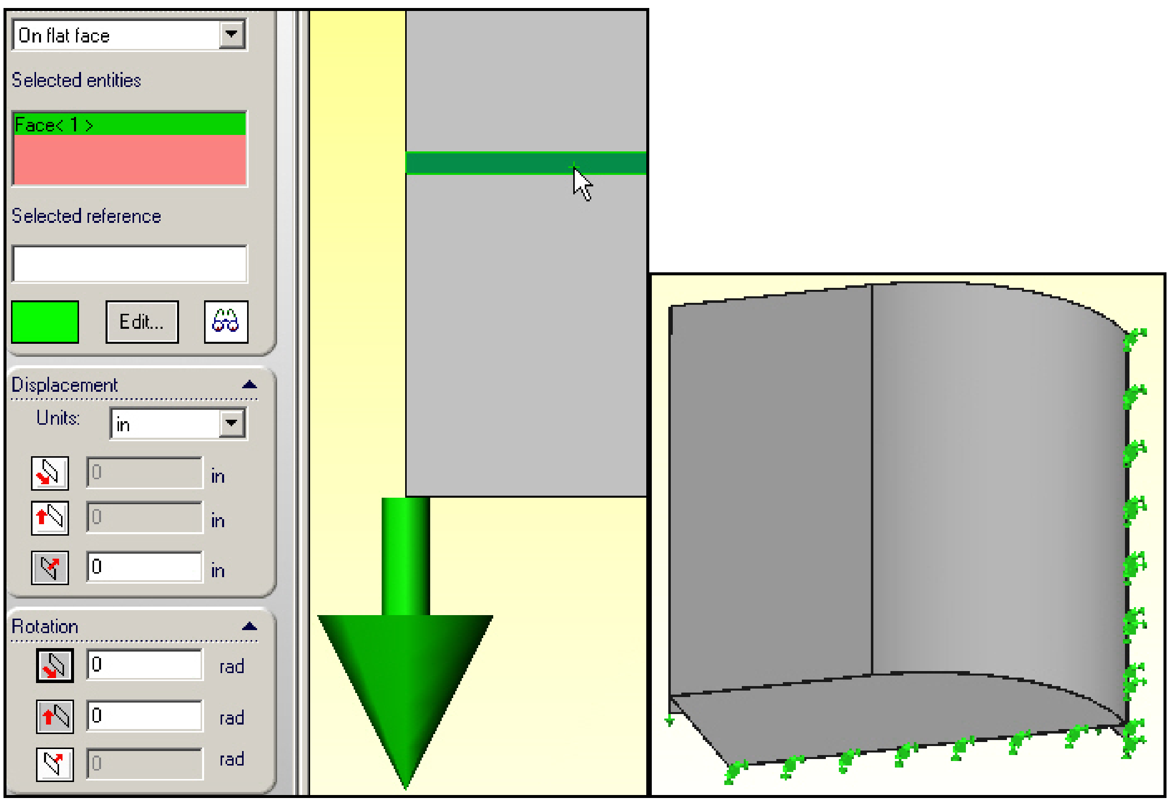


3. In the bottom **Displacement** list you pick the **normal-to** icon, and accept the default value of zero.
4. Clicking **OK** or **Preview** (eyeglass) icon yields a graphical verification of the restraints.

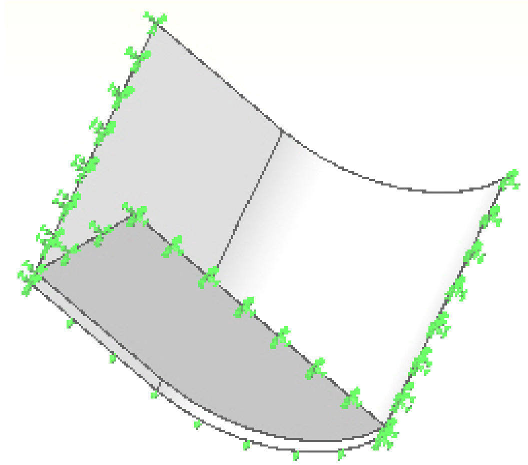
If the imposed displacements were not zero you would have to choose the correct **Units** before typing in the non-zero prescribed displacement normal to the face. This is a physical restraint. The next two restraints come from utilizing symmetry as a modeling tool. Since material has been removed at each of the symmetry cutting planes, you must impose displacement restraints that act like the removed material.

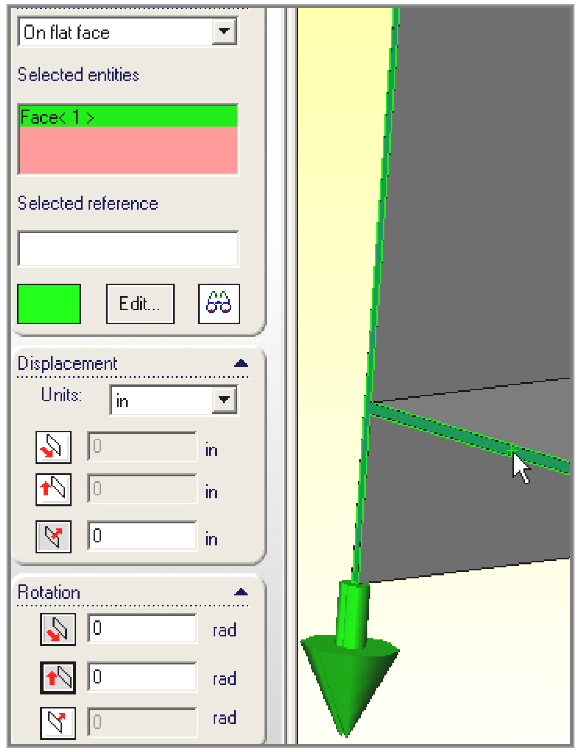
The symmetry planes are the front (x-z) and left (y-z) planes. To impose symmetry restraints:

1. Activate the **Restraint panel** and the **On flat face Type**. Zoom in on and pick the flat front plane surface. Set the symmetry condition of no displacement perpendicular to it. That prevents RBM to the back and RBM rotation about a vertical axis (so 1 RBM remains). Since a shell model will be invoked it, is also necessary to restrain the rotation vector tangent to that plane (i.e. its 2 components in that plane are identically zero). Note that we also get a visual check of the previous vertical support of the bottom-most face, and a graphical preview of the new displacement and rotation restraints.



2. The final (third) symmetry restraint will go on the left flat face. A zero normal displacement there eliminates the last RBM which was in the left-right direction. Again, for a shell the two in-plane rotation components are zero on a symmetry plane. In the **Restraint panel** of pick **On flat face Type** and zero the normal displacement and enplane rotations and preview the restraints.



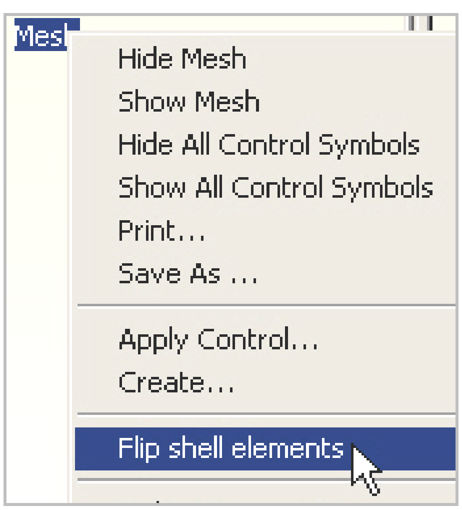


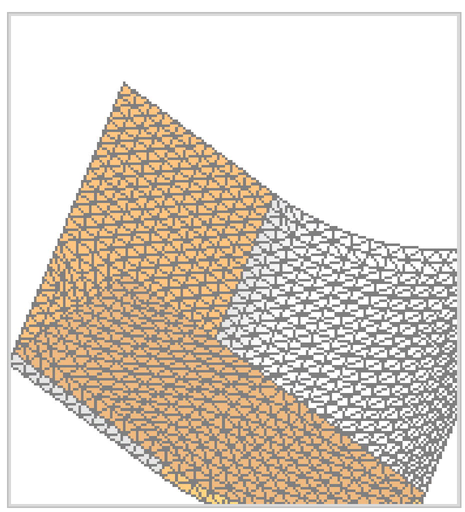
At this point a total of three sets of generalized displacement restraints exist and all rigid body motions have been eliminated. Remaining tasks include specifying the hydrostatic pressure loads, which will require defining a local coordinate system, and creating the shell mesh.

8.2.3 Shell mesh generation

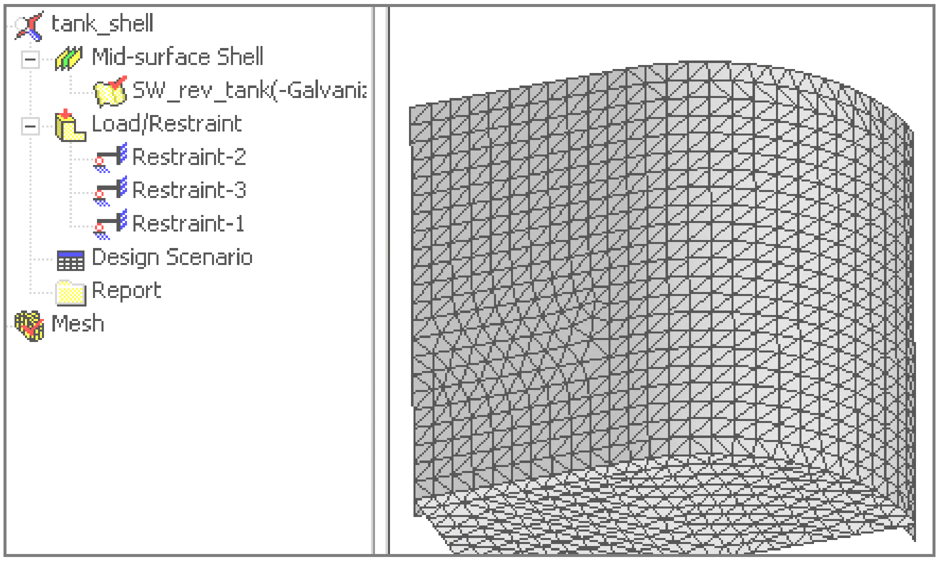
You should expect the highest bending stresses will be near the tank bottom-side wall junction region. Thus, we will eventually probably have to control the mesh to make the smallest elements occur there. However, for the first analysis you can accept the default mesh generation:

1. Right click on **Mesh→ Create** (and skip **Apply Control** for now).
2. In the **Mesh panel** select a coarse **Mesh Parameter→OK** for an initial crude mesh. The first mesh appears. The straight side wall and the curved lip wall need to be flipped (so the shell “top” will always be against the fluid).





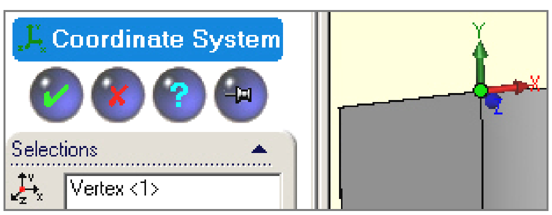
3. Control select those two segments and then right click **Mesh** and **Flip shell elements**. Now, all the (gray) shell tops should match in the fluid contact surfaces.



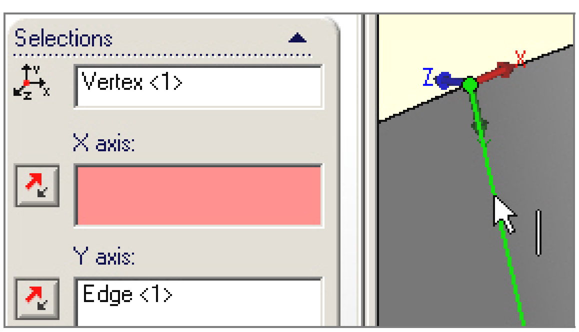
8.2.4 Variable pressure loading

In the CWManager portion above you can see there are three active restraint sets, but no loads. In order to apply a variable pressure you first need to create a local coordinate system. The worst case water load is when the tank is completely full. Thus, prescribe a hydrostatic pressure load increasing from the top edge by defining a new local coordinate system:

1. Go to **Insert→Reference Geometry→ Coordinate System** to open the **Coordinate System panel**.
2. First, locate the **origin** at the top.



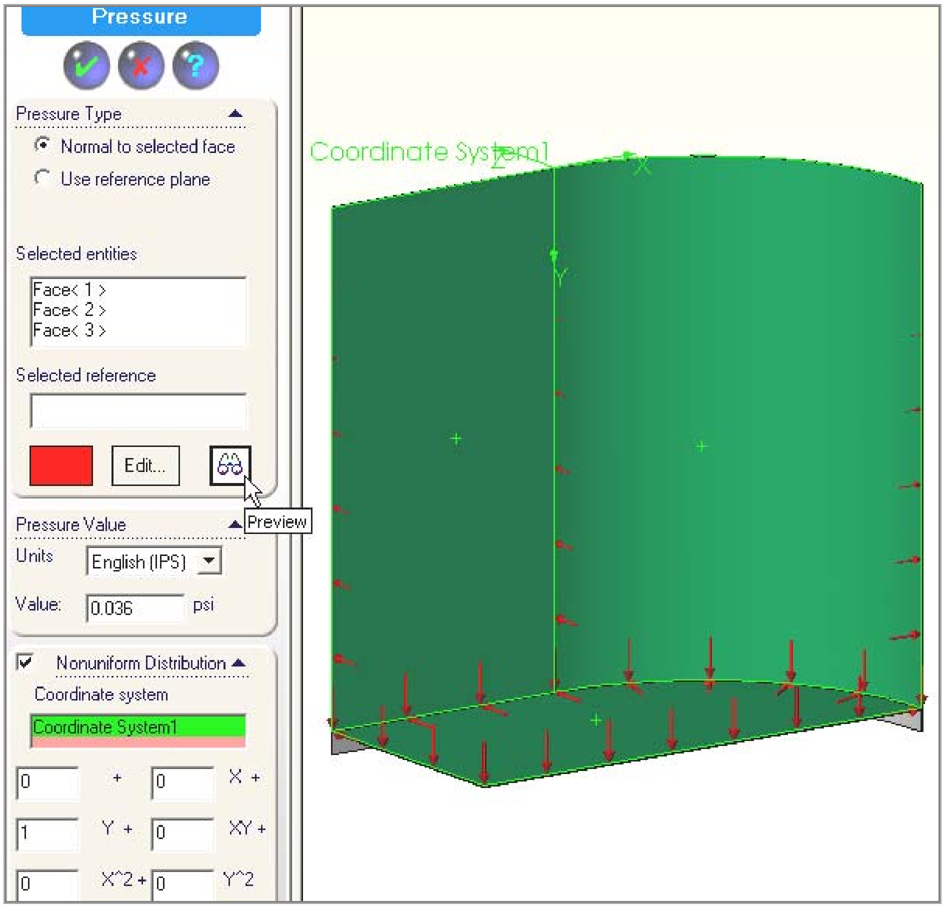
3. Put the local **Y-axis** downward along a vertical line (pressure increases directly with the local Y value).

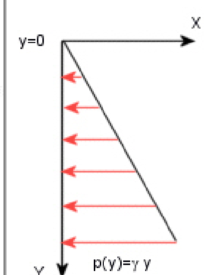


4. It is the only axis you will use so accept the default orientations for the other two local axes. There are several uses for local coordinate systems so remember this process. [***WARNING:*** *When a variable pressure load changes signs CosmosWorks expects a split line or split surface to be inserted into the model along the zero value contour*.]

Continue with the application of the pressure loading:

1. Turn on the **View→Coordinate System.**
2. Select **Load/Restraints→ Pressure** to impose the hydrostatic pressure load in the local Y-direction.
3. In the **Pressure panel** use normal to selected face as the **Pressure Type.**





4. Then pick the surfaces of the tank walls and bottom (but not the small outside bottom support edges).
5. Select the **Pressure Value Units** as psi (English).
6. Set the pressure dimensional scale **Value** to 0.036 psi (since the water density, γ, is 0.036 lb/in^3). That value is multiplied times the non-dimensional quadratic polynomial, in the local x-y coordinate directions, activated by checking a **Nonuniform Distribution**.
7. Set all the non-dimensional polynomial coefficients to zero except for the unity **Y** term (so as to create a linear pressure increase with vertical depth).
8. Picking **Preview** (the eyeglasses) gives you a visual check of the pressure distribution along the edges of the loaded faces.

8.2.5 Run the solution

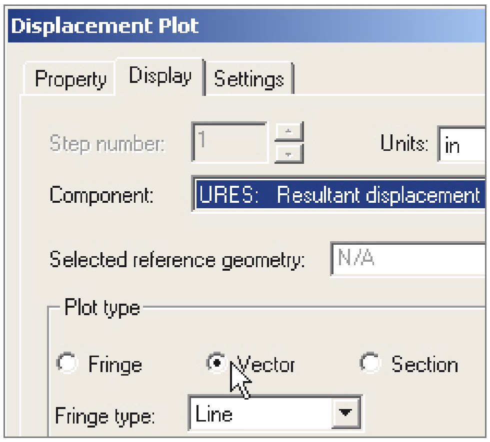
Now you can right click on the model name and select **Run** to start the first crude mesh analysis. Passing windows will keep you posted on the number of equations being solved and the status of the displacement solution process and post-processing. You should get a notice that the analysis was completed (not a failed message). Then you have access to the various CosmosWorks report and plot options needed to review the first analysis.

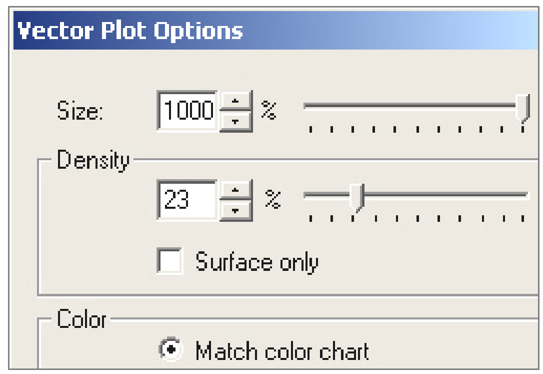
8.2.6 Post-processing

8.2.6.1 Displacement review

Start by double clicking the **Displacement Plot1** icon. The default plot is a smoothly filled (Gouraud) contour display of the resultant displacement magnitude and the deformed shape part. However, since displacements are vector quantities consider a vector plot first:

1. Access them from a right click, **Edit Definitions→Displacement Plot.**





2. **Edit Definitions→Vector Plot Options** (above) double click again on the plot icon to create the view shown in Figure 8-3.

Still, the contours values are useful at times. If you do not have a color printer and/or if you want a somewhat finer description you may want to change the default plot styles:

1. Click in the graphics window and select **Edit Definition →Displacement Plot panel→Display**.
2. Change from the default Gouraud filled image to a **Line** contour option.
3. Double click again on the **Plot** icon to get both the magnified deflected shape and the color contours of the displacement values shown in Figure 8-4. That alternate view may be easier to understand or to plot in grayscale.

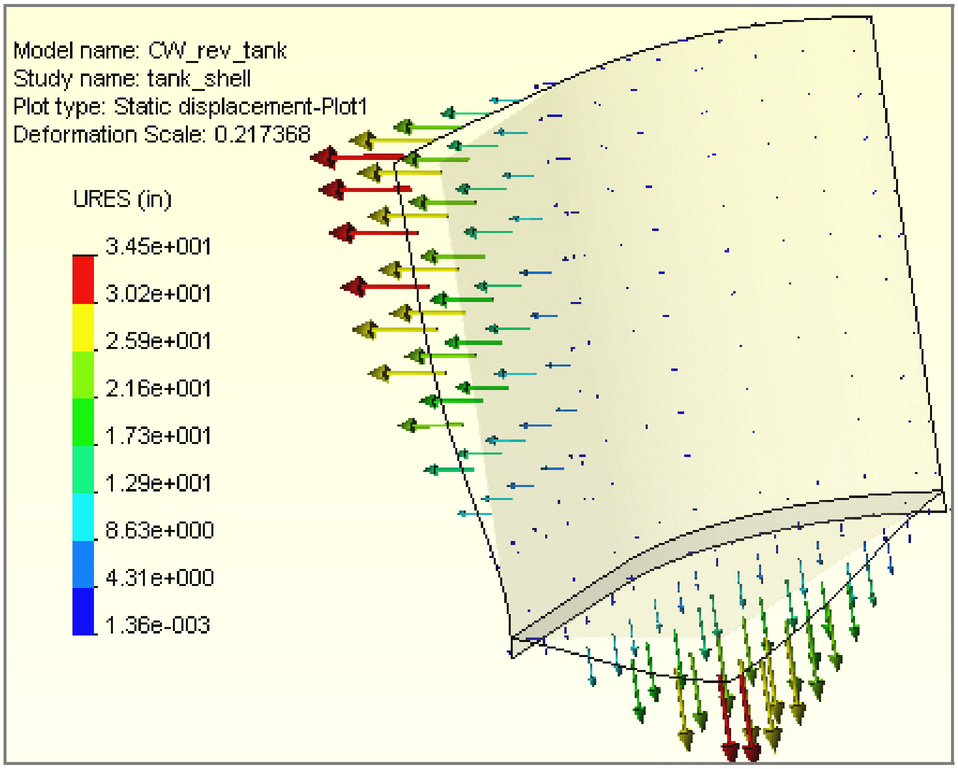


Figure 8-3 Resultant displacement vector plots

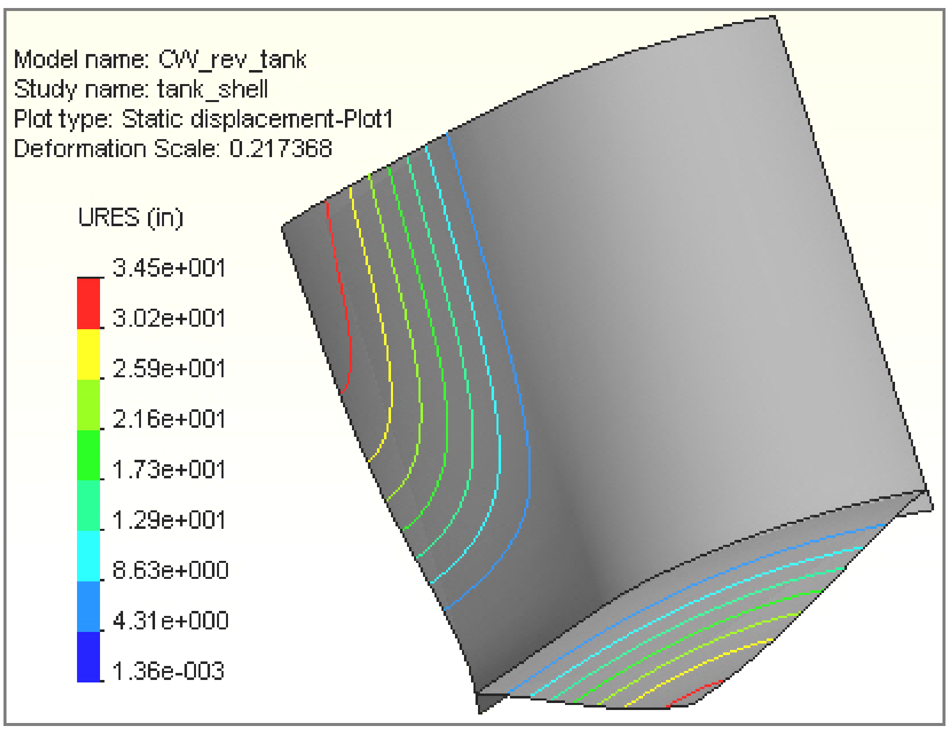


Figure 8-4 Displacement magnitude contours

8.2.6.2 Stress review

Next check the stress levels by double clicking on **Stress→ Plot** icon. There are many types of stress evaluations available. The default one is the scalar Von Mises (or Effective) stress. It is actually not a stress but a failure criterion for ductile materials. Since you picked a ductile material it should be examined and compared to the material yield stress (of about 4,000 psi).

Figure 8-5 shows that most of the tank is above the yield point, so you need to change the thickness, the material, and/or the restraint methods. As expected, in that plot, the maximum effective stress occurs near the junction of the tank wall and bottom. That suggests our next mesh should be controlled to give smaller elements in that region.

Next, examine another type of view of the stress results:

1. Pick **Edit Definition→Stress plot→Display.**
2. Chose **Vector** and **Component** P1.
3. Then use **Color Map** with **8 Thin** colors.
4. Set the color scale only down **7%** from the top of the graphics area.

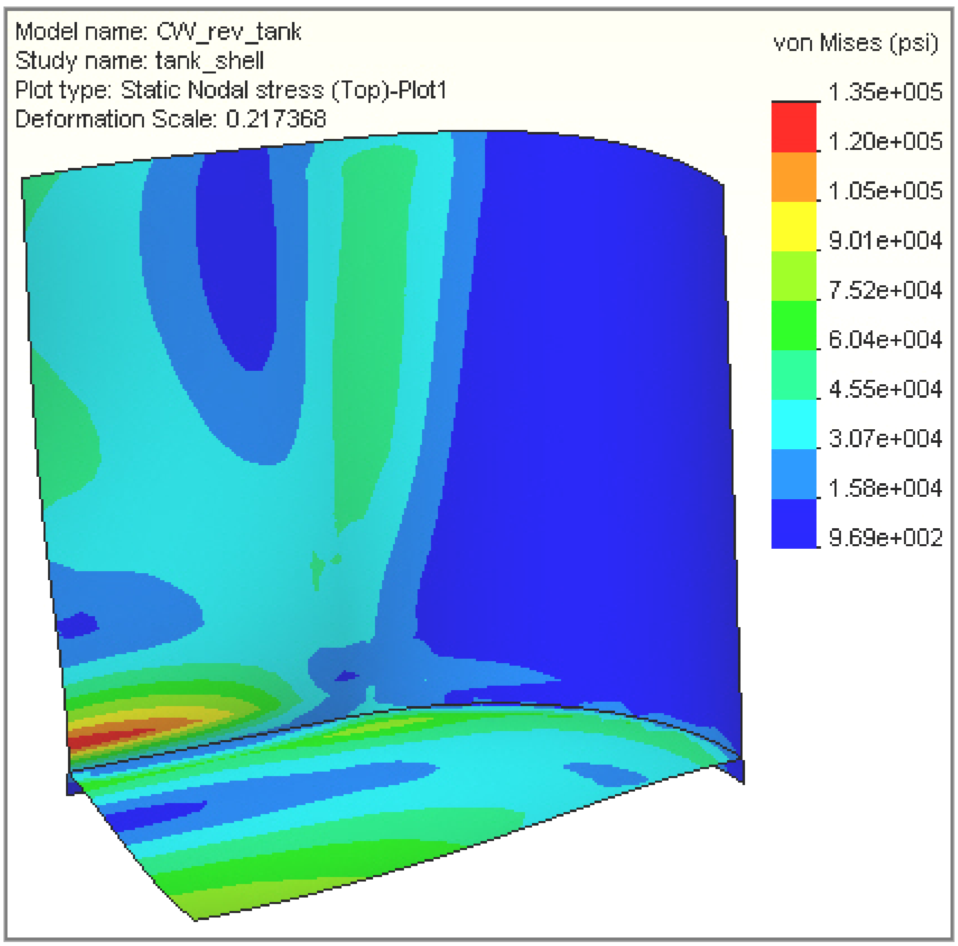


Figure 8-5 Effective stress distributions

Usually you should check the maximum principal stress vectors to see if the directions of the stresses look reasonable. Often you need to increase their visibility:

1. Pick **Vector Plot Options.**
2. Dynamically change the **Size** and **Density** of the vectors as you watch the plot.

As expected, the maximum tension (on the top face of the shell) in **Figure 8-6** occurs in the two segments by the wall-bottom junction. Zooming in on that region you get the principal stress vectors there. (The bottom faces would experience the maximum compression.)

8.2.7 Part revisions

This model could also be revised to look at other restrain conditions. For example, if the tank base sits on two 2" x 4" wooden boards (at the bottom edge) you should expect higher stresses. That study would require additional split lines on the current bottom edge lip surface to pick a smaller support surface.

Likewise, if you assumed that the tank sinks into the ground (or you eliminate the bottom edge) so the full tank bottom is supported in the vertical direction then you could use the existing geometry. You would just set a vertical (normal only) restraint there. The stresses and deflections would be much smaller in that case. To test that concept, you would only need to add one additional restraint set that provides vertical support to the tank bottom plate.

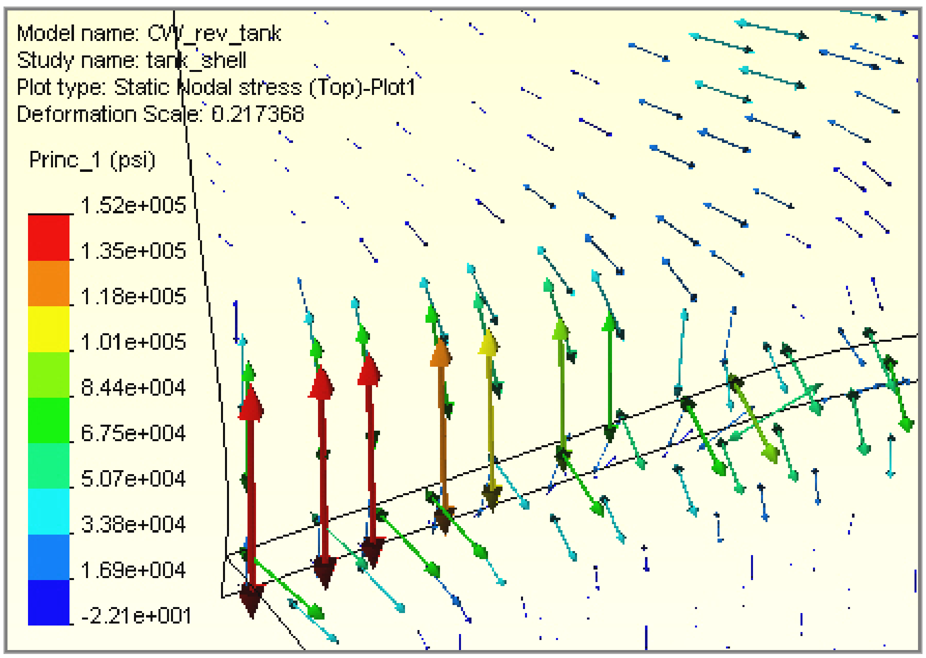


Figure 8-6 Principal stress vectors near the wall-bottom junction

8.3 2.5D Solid analysis

8.3.1 Introduction

There are many 3D parts that can be represented with a 2D drawing of regions noted as having different constant thicknesses. Components of that sort are commonly referred to as 2.5D solids. They can be analyzed with shell models, for any loading states, as a way to validate full 3D solid studies and/or to help plan the mesh controls needed to make the initial 3D study economical. As an example, consider a ship bulkhead that is subject to an in-plane constant bi-axial stress state, with $\sigma_x = \sigma$ and $\sigma_y = \sigma/2$. The bulkhead contains two symmetrical portals (Figure 8-7) that are 1 m wide. The openings will cause a local stress concentration, say σ_{max}, at their edge (to be shown below). Since this is a linear analysis, the results can be directly scaled for any value of σ.

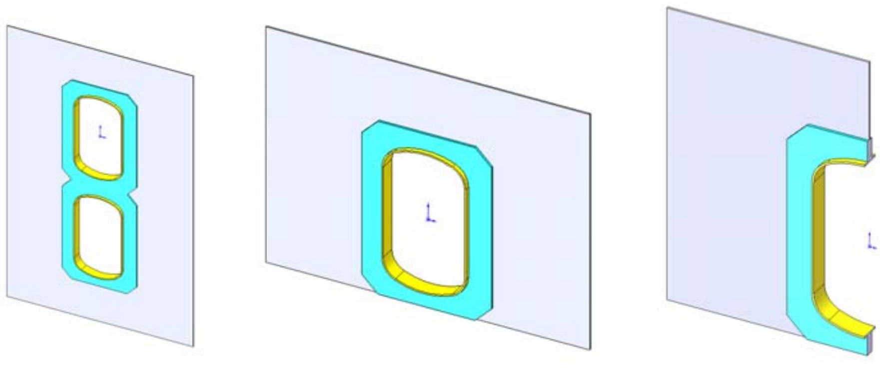


Figure 8-7 Portals through a ship bulkhead

The stress concentration factor, K_t, for bi-axial tension around an elliptical hole in an infinite plate is [11]:

$$K_t = 1 + \frac{2a}{b} - \frac{\sigma_y}{\sigma_x} = 1 + \frac{2(0.875)}{0.5} - \frac{\sigma/2}{\sigma} = 4.0,$$

where *a* and *b* are the major and minor axes of a similar ellipse. To reduce the stress concentration factor around the opening, the wall thickness is to be increased in two stages. Employ thickness ratios of 1:4:10 relative to the standard thickness of 0.02 m. Note that these wall thicknesses could be parameters in a weight optimization study. The dimensions on the two regions of increased wall thickness are seen in Figure 8-8.

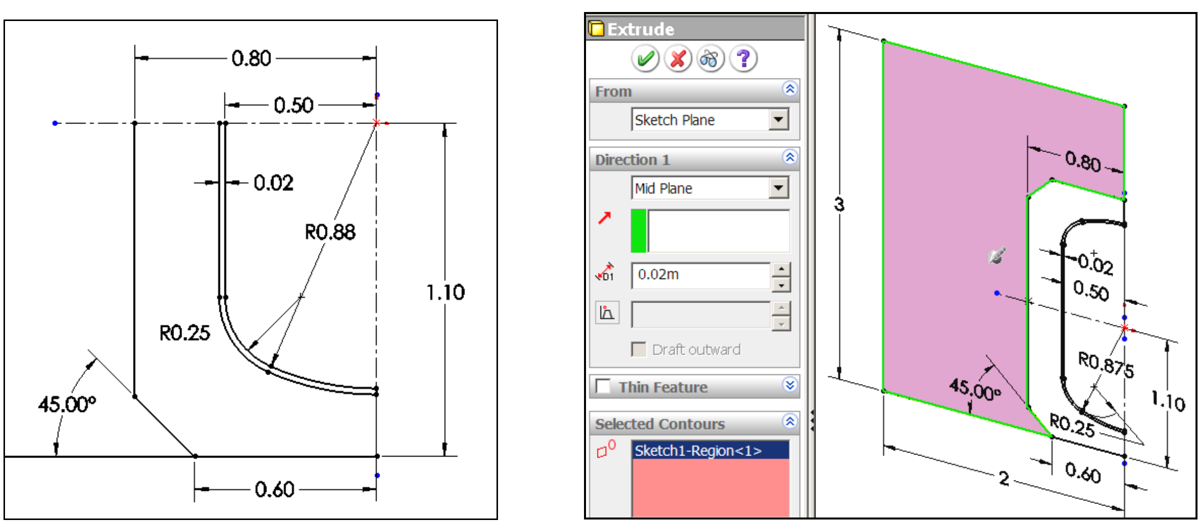


Figure 8-8 Regions of increased wall thickness

The sketch of the three regions is extruded relative to the mid-plane of the bulkhead, with merge results checked, to form a 3D solid. The quarter symmetry model with loads and restraints and its solid mesh is given in Figure 8-9. There is one quadratic element through the thickness of the main bulkhead. That is sufficient, since there are no transverse loads to cause bending. Otherwise, mesh control would be required to force more solid elements into the thickness. Before continuing on to the structural solid results, the creation of the 2.5D shell validation model will be introduced.

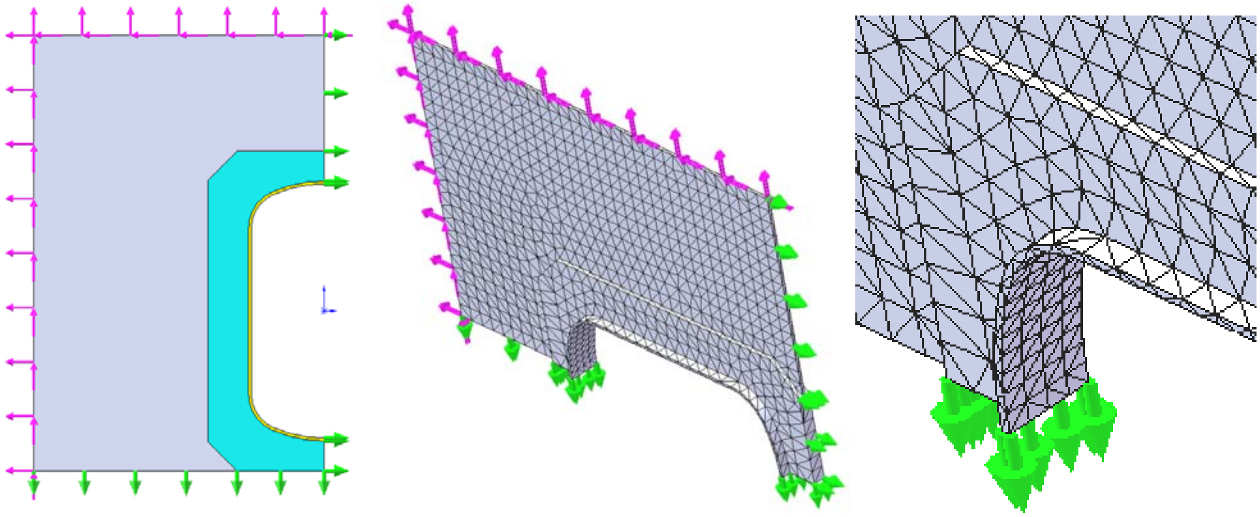


Figure 8-9 A quarter symmetry solid model and solid mesh

8.3.2 Piecewise constant thickness shell model

The original solid was extruded as three merged constant thickness regions, about a common mid-plane. Several commercial finite element systems could mesh such a solid with mid-surface shell elements and automatically assign the correct thickness to each element. Cosmos does not do that. It incorrectly creates a constant thickness flat plate, in the plane of the bulkhead, connected to a curved shell around the edge of the opening that extends away from the plate on both of the plate faces. Therefore, you can not use a mid-surface shell model to validate this solid. You can however employ an assembly of the three regions, bonded together, each consisting of a selected-surface shell having a specified thickness. Here, you need at least two of those three surfaces to be in the same plane. Extrude the first region just like it is shown in Figure 8-8. Save that body as "Thin" and suppress it so it does not merge with the next extrusion. Extrude the next body with the same thickness, name it "Mid" and suppress it. Extrude the third region with the same thickness and save it with the name "Thick". Import the three bodies into an assembly and mate them together. As shown in Figure 8-10, you can set each body to have a different constant thickness. The last two regions were defined as thick, although that was probably not necessary for this in-plane loading state. If the edge of the opening had been much thicker it would have also been re-run as an out of plane shell as another validation bound estimate.

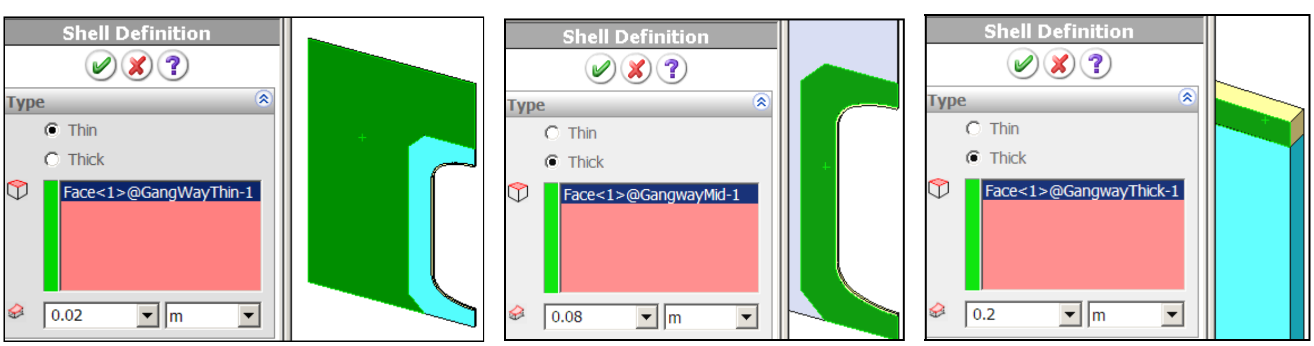


Figure 8-10 Setting constant thickness shells in three assembled bodies

Figure 8-11 shows the three imported bodies in the SolidWorks assembly, before mating, and the created shell meshes after bonding in CosmosWorks. This assembly was loaded and given symmetric restraints like in Figure 8-9 (but with additional rotational symmetry restraints for the shell edges).

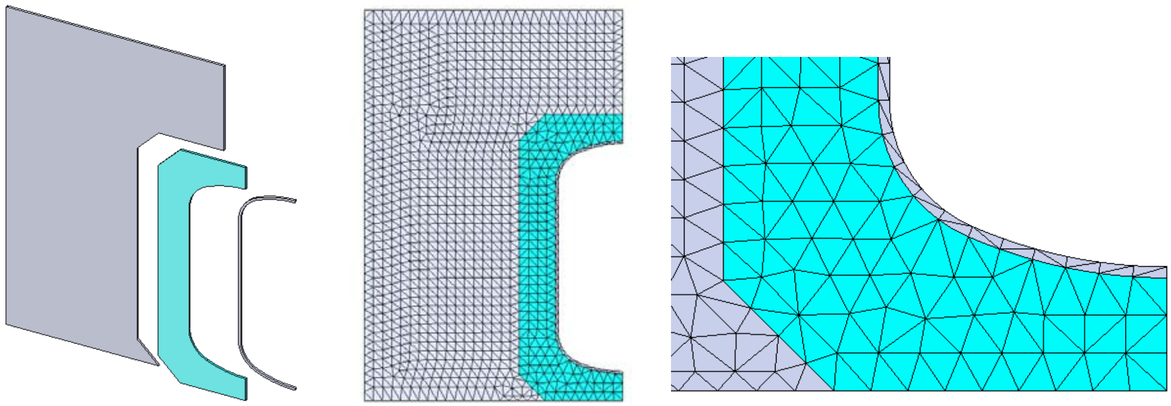


Figure 8-11 The flat shell, three thickness, bounded body assembly and mesh

8.3.3 The un-reinforced component

To illustrate the stress concentration around the opening of the single thickness model, the results for the displacement, von Mises stress, and (twice) the maximum shear stress are given in Figure 8-12. The above stress concentration factor approximation assumed the opening was in the center of a symmetrical region, which is not the case. Being offset from the center increases the stress level at the bottom of the modeled opening.

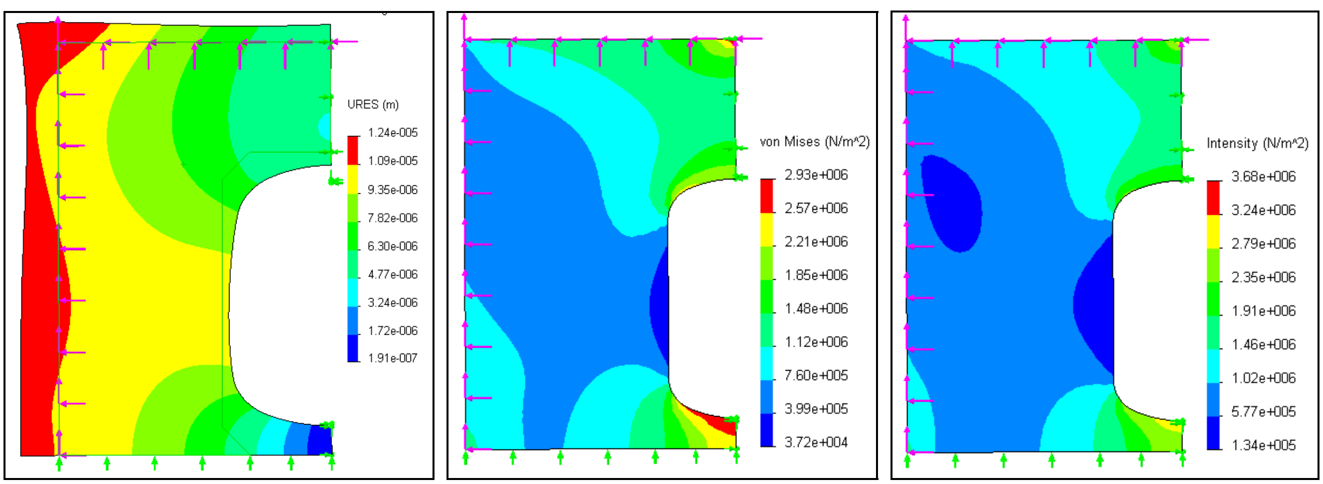
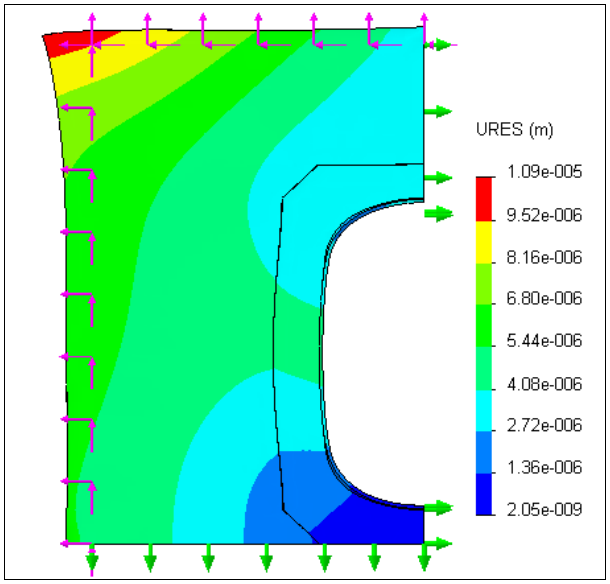


Figure 8-12 The un-reinforced opening results

8.3.4 Comparison of solid and 2.5D results

The 2.5D model gives a very good validation of the solid model results, with a lot less computational resources. The contour plot comparisons are set to have the same contour ranges. The contour plots have the solid on the left of the figure and the 2.5D (piecewise constant selected surface shell) model on the right. The displacements are illustrated in Figure 8-13. The von Mises stress comparison is seen in Figure 8-14. Compared to Figure 8-12, the peak stress has been reduced by about a factor of 7.5. That is seen more clearly in Figure 8-15 which gives the graph of the von Mises stress along the vertical line from the bottom of the opening to the bottom of the model. The intensity (twice the maximum shear stress) from the solid and 2.5D models are given in Figure 8-16. The validations are in good agreement.



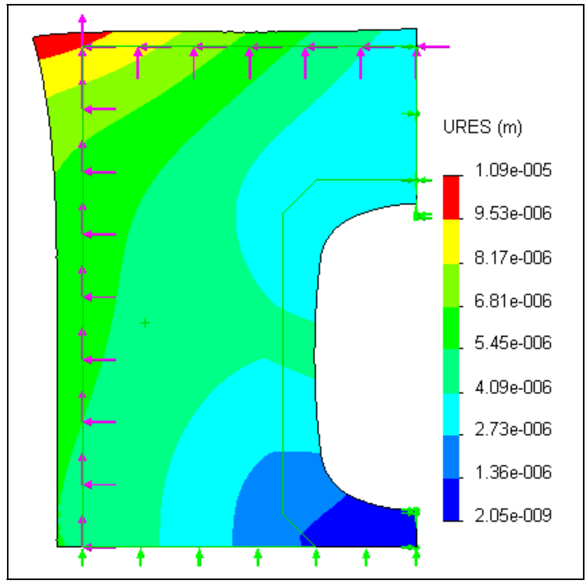
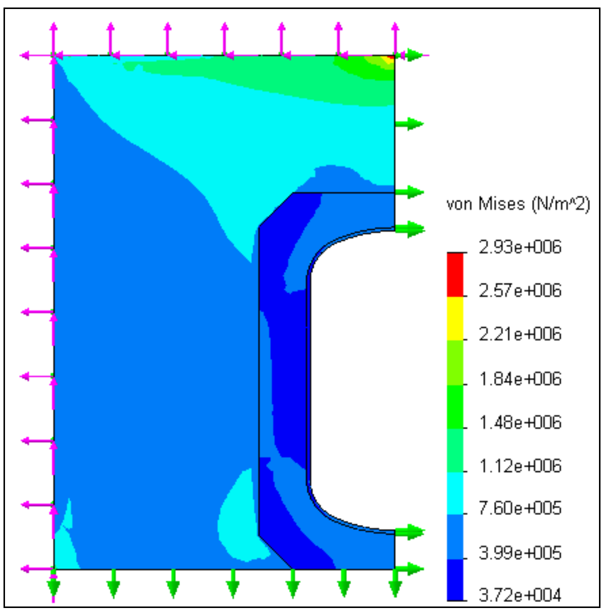


Figure 8-13 Displacement results: solid (left), 2.5D (right)



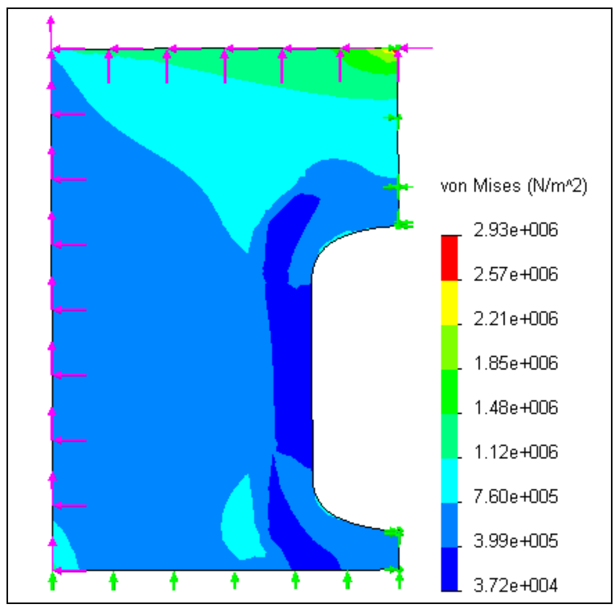
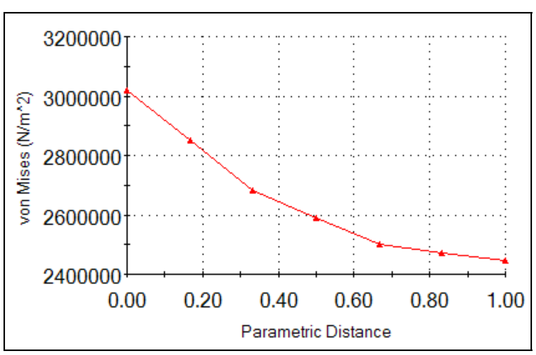


Figure 8-14 Von Mises stress results: solid (left), 2.5D (right)



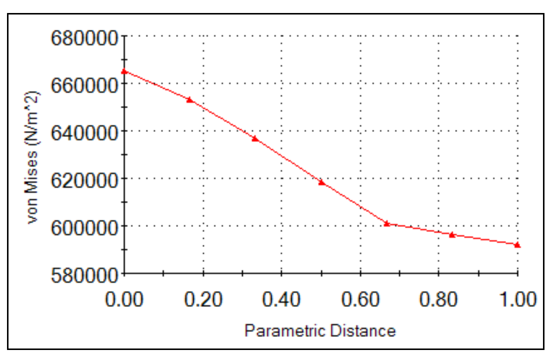


Figure 8-15 Peak stress reduction from single (left) to three thickness models (right)

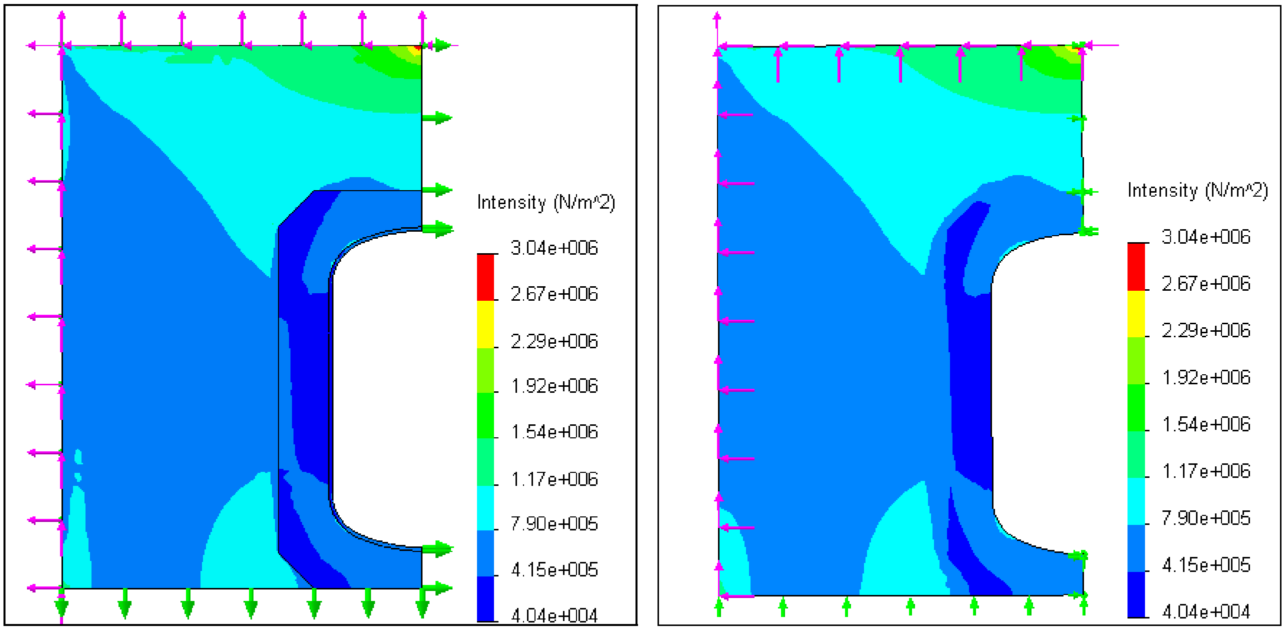


Figure 8-16 Twice the maximum shear stress: solid (left), 2.5D (right)

8.3.5 Closure

The above three thickness shell model did not catch some of the 3D response of the material adjacent to the hole. The flanges of the curved region around the hole of the solid model did not have constant displacements. The mid-plane moved the most, while the outer edges of the flange were seen to move less. That is, there was a relative, symmetric, slight curving into the opening. A graph of the flange displacement, from front to back, is given in Figure 8-17. The current 2.5D model missed that very small feature, but a second out of plane shell model would have shown a similar result.

It is always important to consider ways to validate your finite element calculations, even if that requires a different class of finite element model. It has been said that you should use two different models, as above, and then throw them both away and build a better model based on the insight gained from carrying out the first study and its validation.

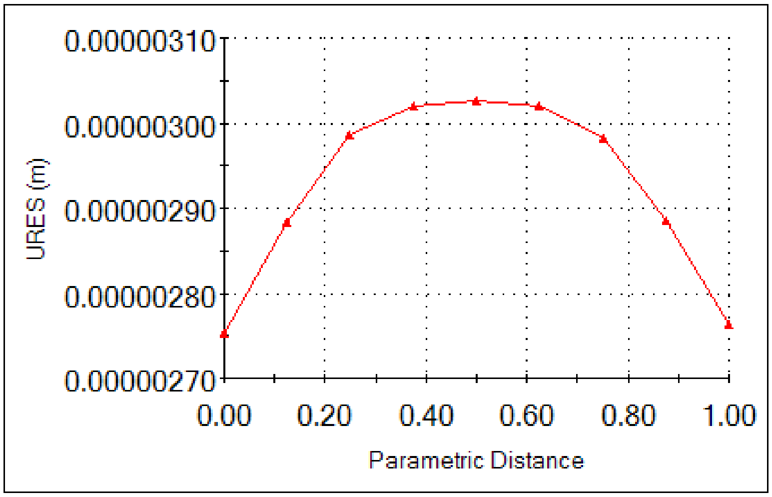


Figure 8-17 Deflection along the thick flange of the solid model

9 Space Truss and Space Frame Analysis

9.1 Introduction

One-dimensional models can be very accurate and very cost effective in the proper applications. For example, a hollow tube may require many thousands of elements to match its geometry, even though you expect its stresses to be constant. A truss (bar) or frame (beam) element can account for the geometry exactly and give "exact" stress results and deflections with just a handful of equations to solve. In Cosmos truss and frame elements are available only for static (constant acceleration), natural frequency, and buckling studies. It is recommended that you review the SolidWorks weldments tutorial before using trusses or frames.

The truss element is a very common structural member. A truss element is a "two force member". That is, it is loaded by two equal and opposite collinear forces. These two forces act along the line through the two connection points of the member. The connection points (nodes) are a concurrent force system. That is, the joints transmit only forces. Moments are not present at the joints of a truss. The truss elements in Cosmos are all space truss elements. There are three displacement dof at each node (see the right side of Figure 2-8), and up to three reaction forces at a restrained joint. A space truss has six rigid body motions, all of which must be restrained in an analysis. The space truss and space frame models are created in Cosmos by 3D line sketches. For a truss the lines must exactly meet at common points (joints). The lines of the space frame models can meet at common points and/or terminate as an intersection of two lines. To avoid numerical ill-conditioning, it is best if a space frame does not have two joints very close (say, the width of the cross-section) to each other. If that is necessary Cosmos takes special action to build the finite element model there.

The line that represents a truss or frame member has to be located relative to the cross-section of the member. Where it intersects the cross-section is called the pierce point. For trusses it is important that the pierce point be at the centroid of the cross-section. That happens automatically when you use a built in library shape. If you construct a cross-section make sure that the truss pierce point is at the centroid. If the pierce point is not at the centroid, as in the right of

Figure 9-1, then an axial load will cause bending stresses to develop and to be superimposed on the axial stress. That is allowed in frame elements but not truss elements.

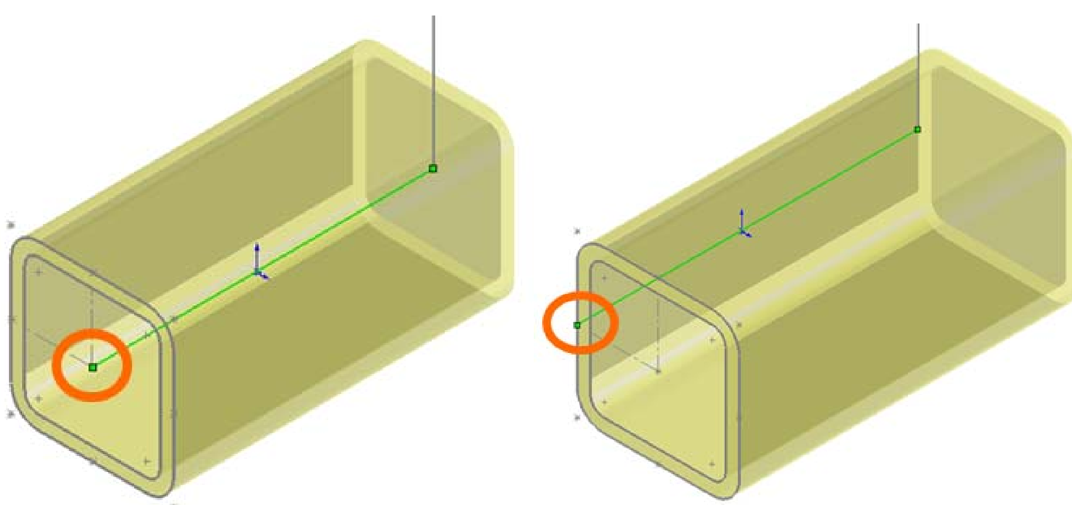


Figure 9-1 Centroidal and eccentric cross-section pierce points

Clearly, the elastic bar is a special form of a truss member. To extend the stiffness matrix of a bar to include trusses in two- or three-dimensions basically requires some analytic geometry. Consider a space truss segment in global space going from point 1 at (x_1, y_1, z_1) to point 2 at (x_2, y_2, z_2). The length of the element between the two points has components parallel to the axes of $L_x = x_2 - x_1$, $L_y = y_2 - y_1$, $L_z = z_2 - z_1$ and the total length is $L^2 = (L^2x + L^2y + L^2z)$. The direction cosines are defined as the ratio of the component length increments divided by the total length of the element. They are used to transform a bar stiffness matrix to the space truss stiffness matrix. For 2D problems only one angle is required to describe the member direction. A truss element stiffness requires only the material elastic modulus, E, the cross-sectional area, A, and the member length, L. A space frame element also requires the three geometric moments of inertia of the cross-section. Two inertias are needed for the transverse bending, and the third is needed for torsional effects. The Cosmos frame element also utilizes the material's Poisson's ratio. The mass density, ρ, is needed for gravity (acceleration) loads, or natural frequency computations.

If you combine the bar member, which carries only loads parallel to its axis, and a beam which carries only loads transverse to its axis you get the so-called beam-column element. Adding the ability to carry torsion moments along the element extends the behavior to a space frame. In other words, a space frame is a combination of individual beam-column elements that resists loadings by a combination of bending, axial member forces, and transverse (shear) forces, and axial torsion. Therefore, it is a more efficient structure than a space truss element.

9.2 Statically determinate space truss

Consider the simple symmetric space truss shown in Figure 9-2. It has two horizontal members, denoted by *a*, and an inclined member, *b*, in the vertical mid-plane. The truss has three immovable restraints (at the dashed circles) and a vertical point load, P = 1,000 lb at the free node. The dimensions are shown in the figure. The members are square hollow tubes, with the horizontal pair being 2 *x* 2 *x* 0.25 inches, and the other 4 *x* 4 *x* 0.25 inches. All three members are made of ASTM A36 steel.

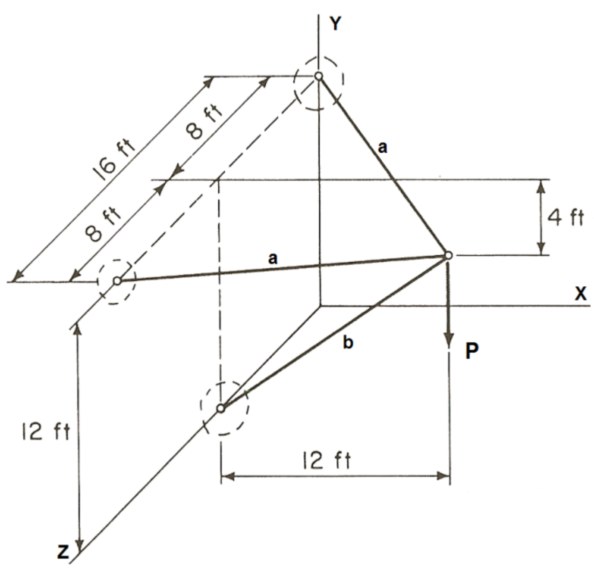
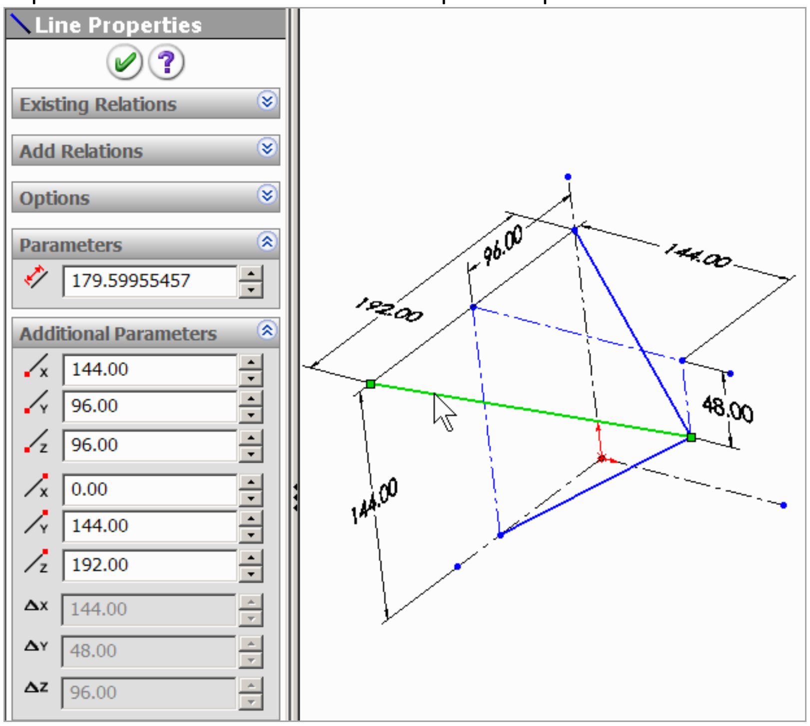


Figure 9-2 The simplest space truss

9.2.1 Construct the space truss

The construction of the 3D line models is done in SolidWorks by means of a **3D Sketch**:

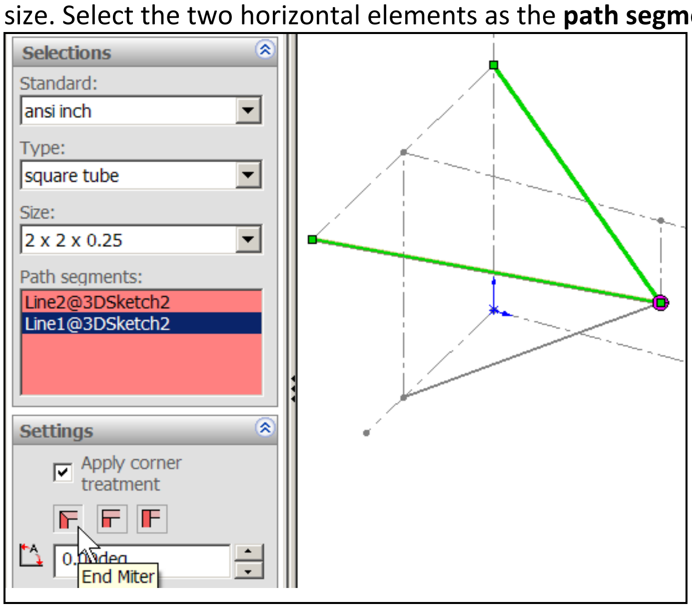
1. **Insert→3D Sketch**. Insert construction lines along each of the axes to help locate nodes. Add and dimension additional construction lines in each coordinate plane.
2. In the **3D Sketch→Lines**. Draw one line to open the **Line Properties panel**.
3. Expand the **Additional Parameters** option to provide access to the end points of the first element.

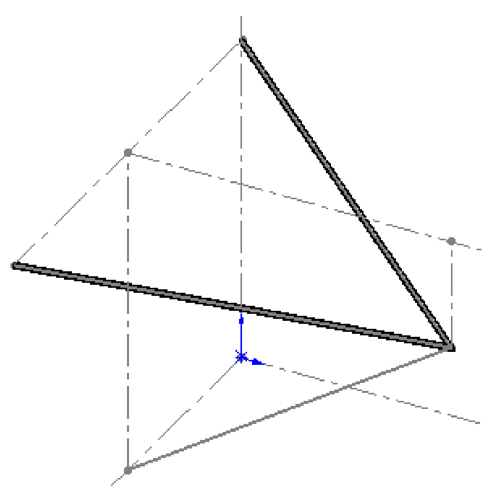


4. Specify the coordinates of the **starting point** and/or the **ending point** and/or the i**ncrements** in coordinates from one end. Spot check the element **length** value. Click **OK**.
5. Repeat for the other elements (or review the 3D sketch tutorial for other approaches).

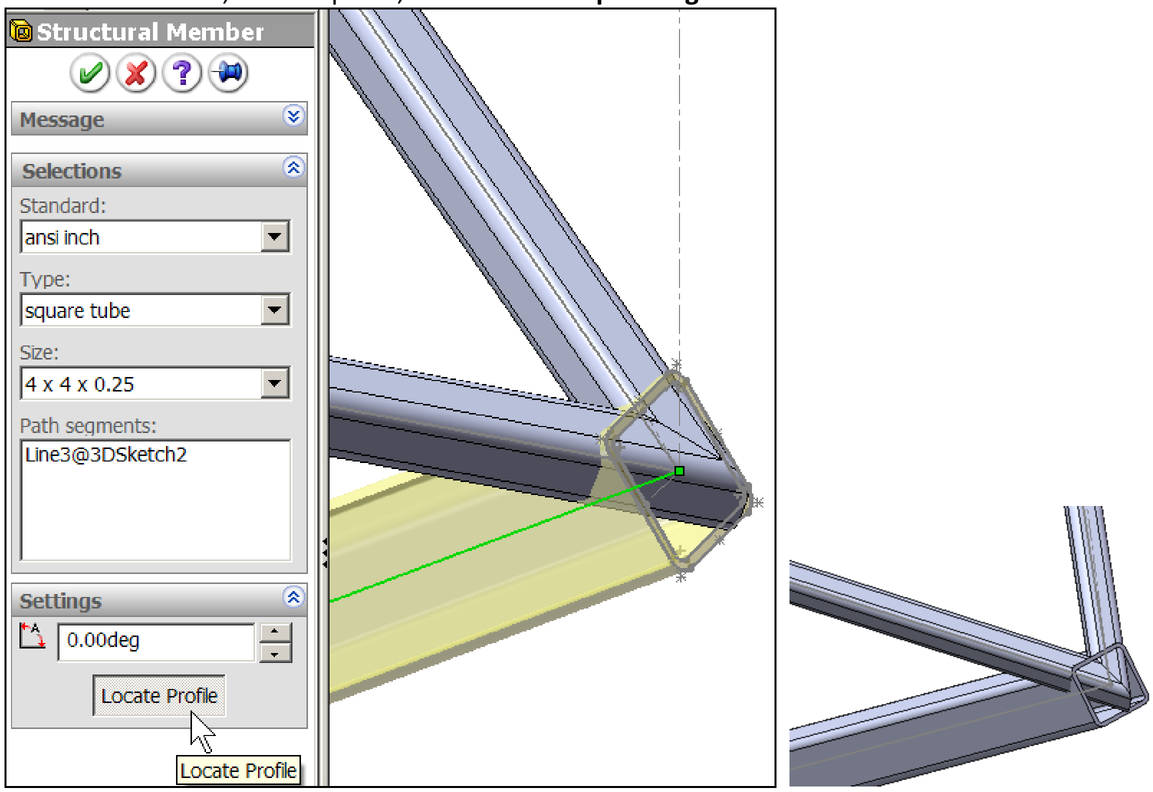
After all the space truss line elements have been located, and saved, the next task is to look up or construct each member's cross-section:

1. **View→Toolbars→Weldments** and click **Structural Member**.
2. In the **Structural Member panel, Selections→Type→Square tube**. Pull down to the 2 x 2 x 0.25 inches size. Select the two horizontal elements as the **path segments**.





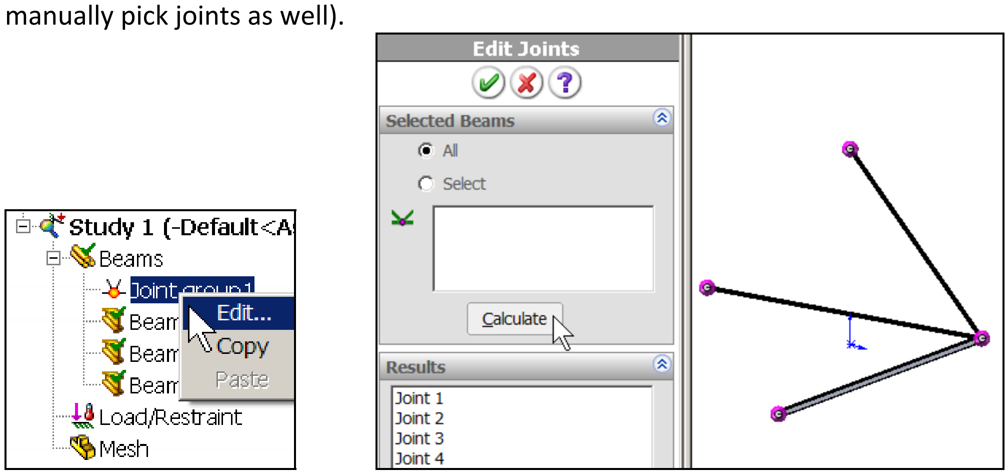
3. Under **Settings→Apply corner treatment** and click on **end miter** and **OK**.
4. In the **Structural Member panel Selections→Type→Square tube**. Pull down to the 4 x 4 x 0.25 inches size for the compression member.
5. Select the inclined, vertical plane, element as the **path segment**. Click **OK**.



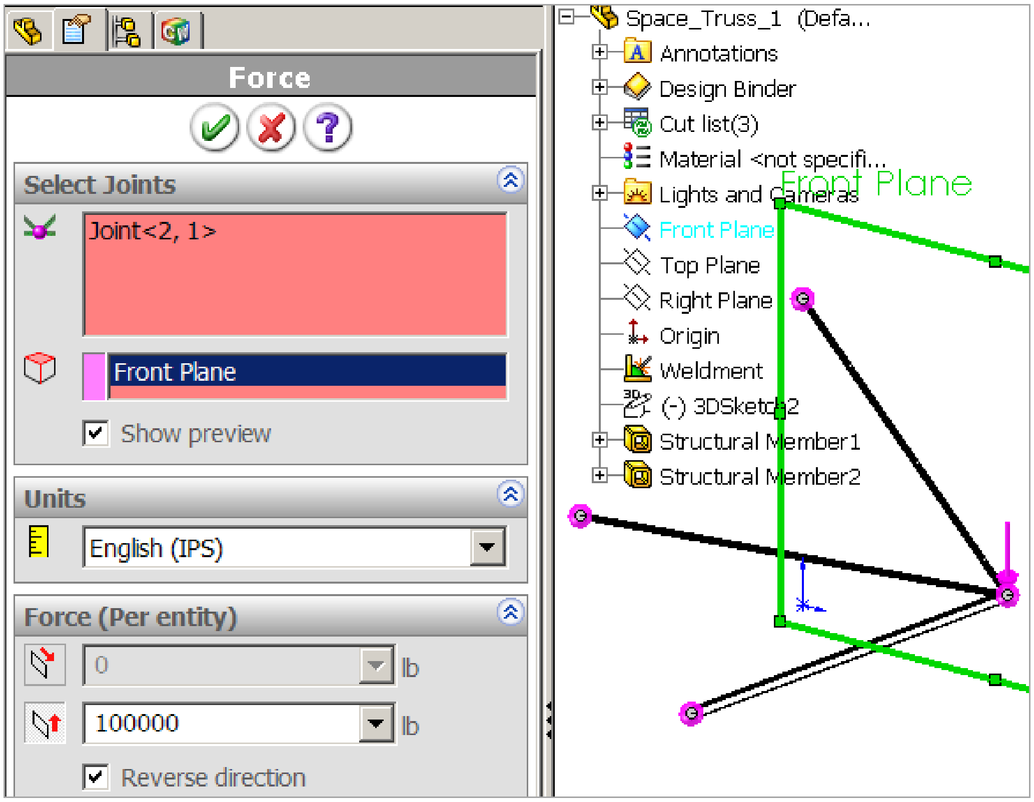
9.2.2 Create the study

In the CosmosWorks menu:

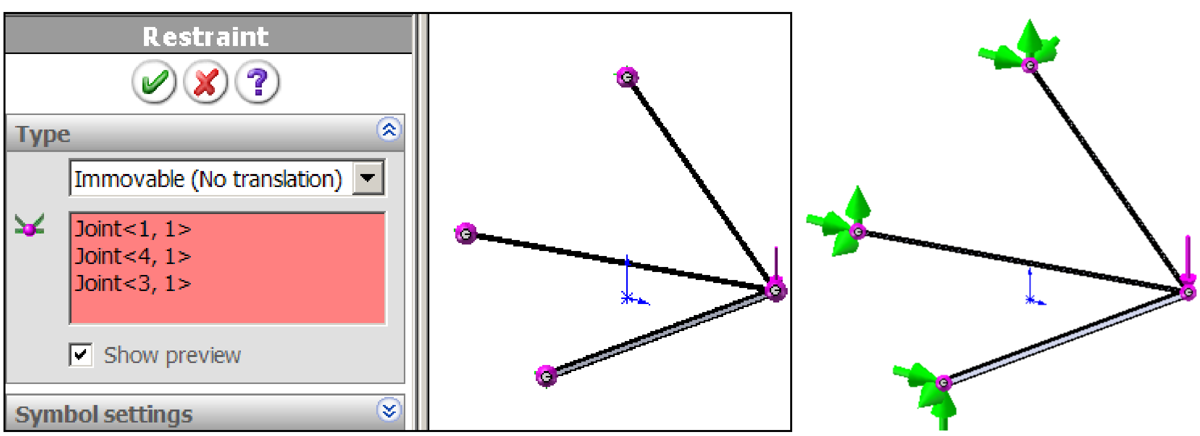
1. Right click on the **study name→Study→Static→Beam mesh**.
2. In the new study, right click on **Beams→Treat all structural members as beams**. The three space truss members appear as a beams list.
3. Select all three beams, **Edit definition→Truss**. Click **OK**.
4. Right click on **Beams→Apply Material→Library→Steel→ASTM A36**, click **OK**.
5. Right click on **Joints→Edit→All→Calculate**. Cosmos calculates and displays all four joints. (You can manually pick joints as well).



6. **Load/Restraint→Force**. In the **Force panel→Select Joints**. Pick the one free node and select the **Front plane** to set the direction. Click **OK**.



7. Select lb **units**, and set a **value** of 1,000 lb as the vertical component.
8. **Load/Restraint→Restraint→Immovable**, select the three wall joints. They prevent the three rigid body translations and three rotations. Click **OK**.



9. **Mesh→Create Mesh**. There are no mesh control options for trusses. The additional created nodes allow more displacement vector displays.
10. **Run.** Cosmos calculates the space truss joint locations and their displacements.

9.2.3 Post-processing

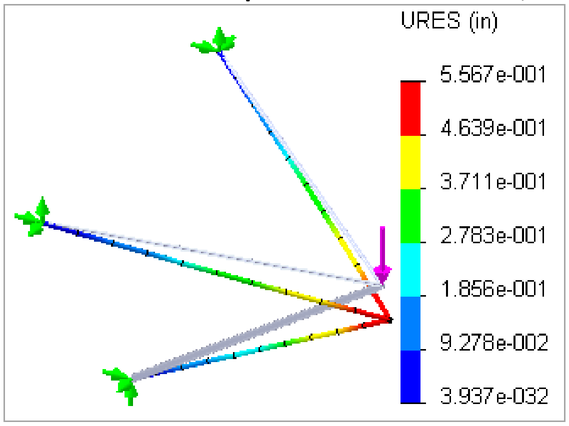
9.2.3.1 Displacements

In the Cosmos Menu:

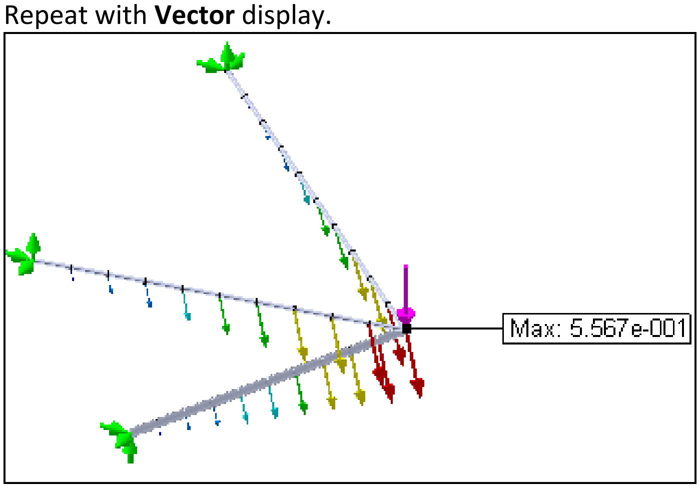
1. Right click **Results→List Displacements** for options. (Or, use Probe.)
2. **List options→Extremes**, 1 percent (there are more than 4 nodes now).
3. **List set→Displacement,** and click **OK**. View the list results.

Node	UX	UY	UZ	URES
1	0.00000e+000	0.00000e+000	0.00000e+000	3.93701e-032
2	1.21993e-003	-5.43165e-003	0.00000e+000	5.56697e-003
3	0.00000e+000	0.00000e+000	0.00000e+000	3.93701e-032
4	0.00000e+000	0.00000e+000	0.00000e+000	3.93701e-032

4. **Results→Define Displacement Plot→URES**, select **Fringe** display.



5. Repeat with **Vector** display.



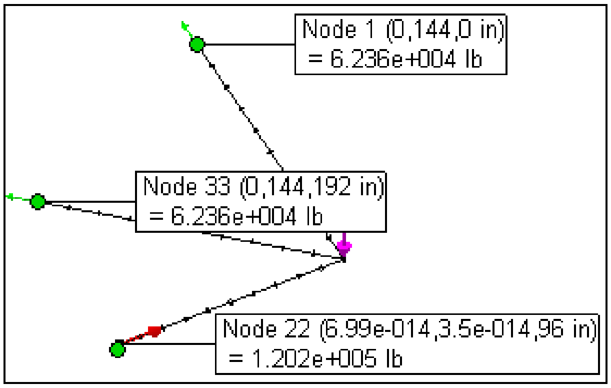
9.2.3.2 Reactions

In the Cosmos Menu:

1. Right click **List Displacements** for options.
2. **List options→From** node 1 to node 34 (the mesh generator created additional nodes).
3. **List set→Reaction forces**, click **OK**. Verify the sum is 1e5 lb, upward.

Node	RFX	RFY	RFZ	RFRES
1	-5.00000e+004	1.66669e+004	-3.33312e+004	6.23600e+004
22	9.99999e+004	6.66662e+004	-3.60947e-004	1.20185e+005
33	-5.00000e+004	1.66669e+004	3.33312e+004	6.23600e+004
Sum:	0.00000e+000	9.99999e+004	-3.60947e-004	9.99999e+004

4. **Results→Define Displacement Plot→RFRES** select **Vector** display.



9.2.3.3 Space truss member stresses

In the Cosmos Menu: Double click on **Stress1** to show the default plot. Select different view points, as in Figure 9-3. Note that each truss member has a constant axial stress level, even though it has multiple elements created by the mesh generation to allow better displacement plots. The maximum tension stress is about 3.9e4 psi and the maximum compression stress is about -3.4e4 psi. The yield stress is about 3.6e4 psi so the results are beyond the elastic limit (FOS < 1), but below the 5.8e4 psi tensile strength. The high compression stress in a "slender" member suggests that you should really worry about buckling. Buckling of this system will be checked in a later section.

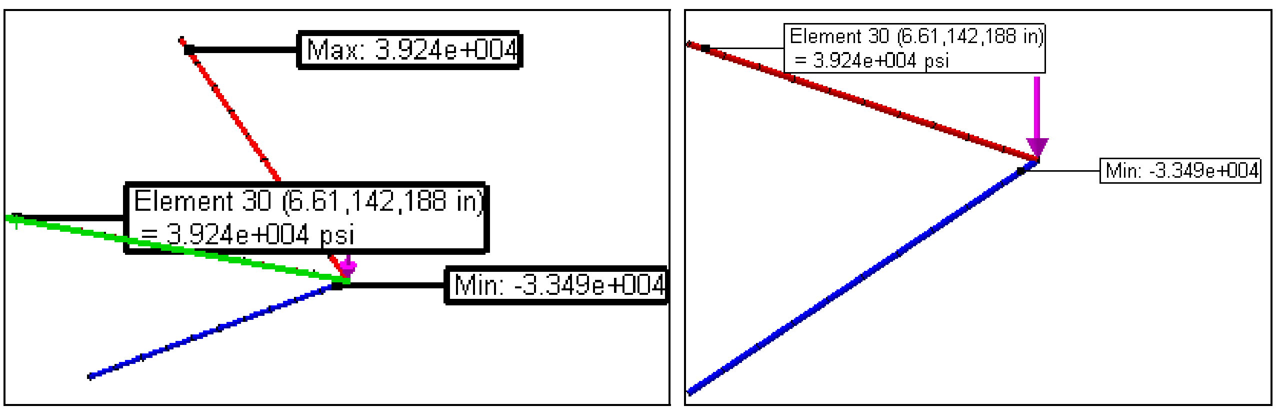


Figure 9-3 Trimetric and front views of truss member's axial stress

Here, the member sizes should be increased, and gravity loads should be included in a revised analysis. It would be difficult to construct the common joint as a pinned connection. For this load state and the small deflections obtained, the system does not appear to merit re-analysis as a space frame.

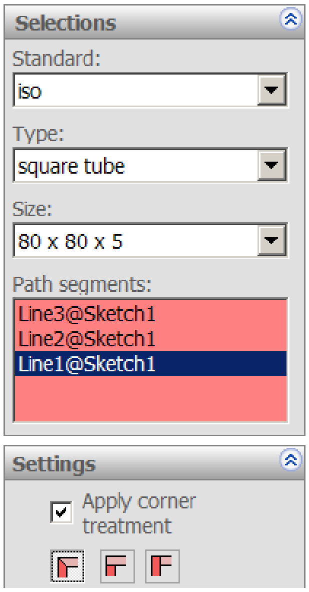
9.3 Statically indeterminate space frame

9.3.1 Support settlement load case

Consider an unequal leg planar frame that is to be subject to a transverse force. The cross-section is an ISO 80 *x* 80 *x* 5 mm square tube. The tall vertical leg is 15 ft, the short one 10 ft and the top member 12 ft long. A survey shows that the support for the shorter leg has settled and imposed a vertical downward displacement of 0.45 inch and a clockwise rotation of 0.02 radians normal to the plane of the frame. Before considering the transverse load a study needs to be run to establish the deflections and stresses introduced by the non-rigid support displacement.

Draw and dimension the three lines defining the frame:

1. **View→Toolbars→Weldments** and click **Structural Member**.
2. In the **Structural Member panel, Selections→Standards→ISO.**

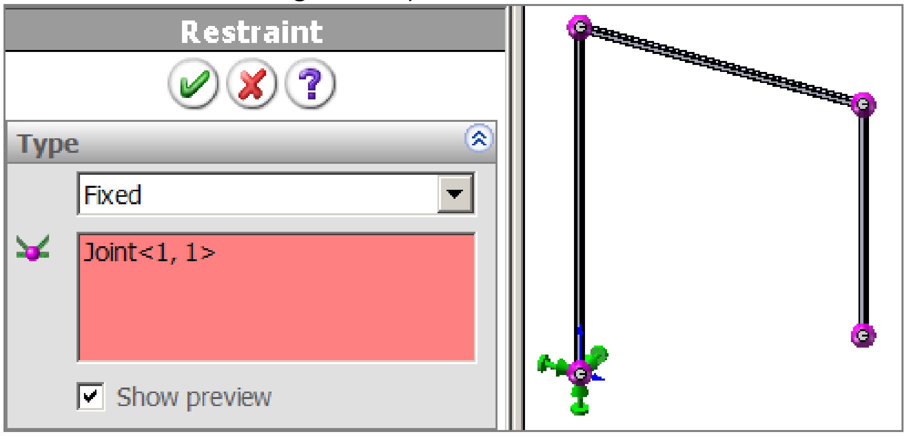


3. In **Selections→Type→Square tube**. Pull down to the 80 *x* 80 *x* 5 mm size.
4. Select all three line segments the as the **path segments**.
5. Under **Settings→Apply corner treatment** and click on **end miter** and **OK**.

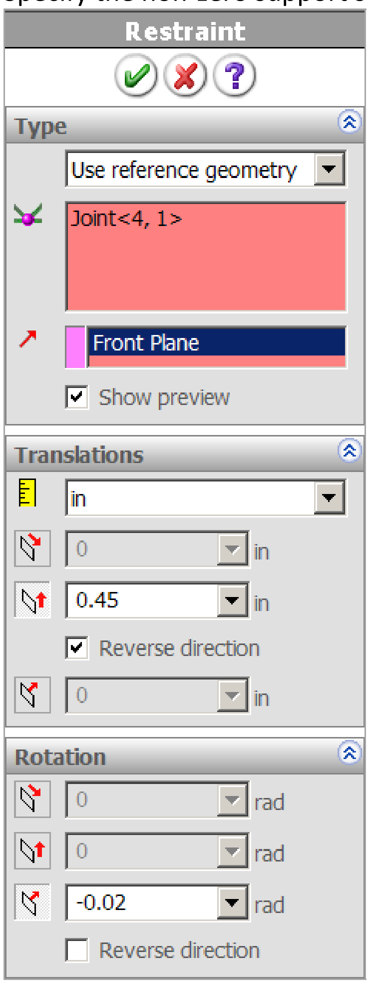
In the CosmosWorks menu:

1. Right click on the **study name→Study→Static→Beam mesh**.
2. In the new study, right click on **Beams→Add beam**. Select one of the frame lines. It appears in the Beams panel list of members.
3. Right click on **Beams→Treat all structural members as beams**. All three space frame members appear now in the beams list.
4. Select all three beams, **Edit definition→Beam**. Click **OK**.
5. Right click on **Beams→Apply Material→Library→Steel→ASTM A36**, click **OK**.
6. Right click on **Joints→Edit→All→Calculate**. Cosmos calculates and displays all four joints (you can manually pick joints as well).
7. **Load/Restraint→Restraint→Fixed.** Pick the lower left support as an encastre (fixed support). This prevents all three rigid body translations and three rotations. Click **OK**. Change the restraint name (in

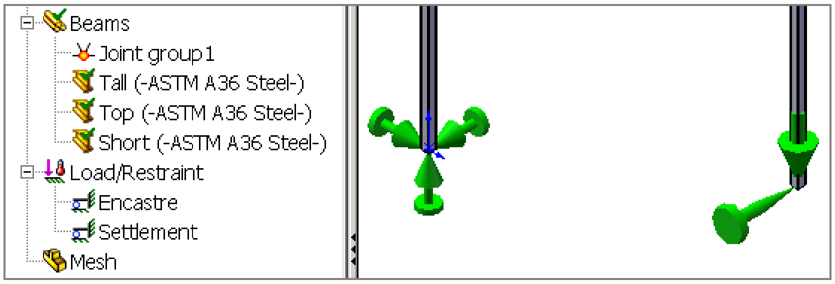
the CosmosWorks manager menu) to Encastre.



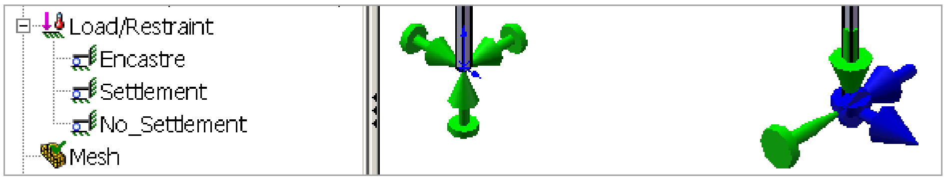
8. Specify the non-zero support settlements. **Load/Restraint→Restraint→Use Reference Geometry.**



9. Pick the lower right joint, selected the **Front Plane** as the reference geometry (from the SolidWorks menu, or by expanding the part tree in the graphics area).
10. Set the **Translations** as downward 0.45 inch, and the **Rotation** normal to the front plane (Z-axis) as 0.02 radians clockwise. Green symbols appear at the two settlements. **Name** the restraint Settlement.



11. **Load/Restraint→Restraint→Use Reference Geometry**. Select the **Front Plane.** Set the zero values for the other four dof at that joint, change their **color** to blue. **Name** the restraint No_Settlement.

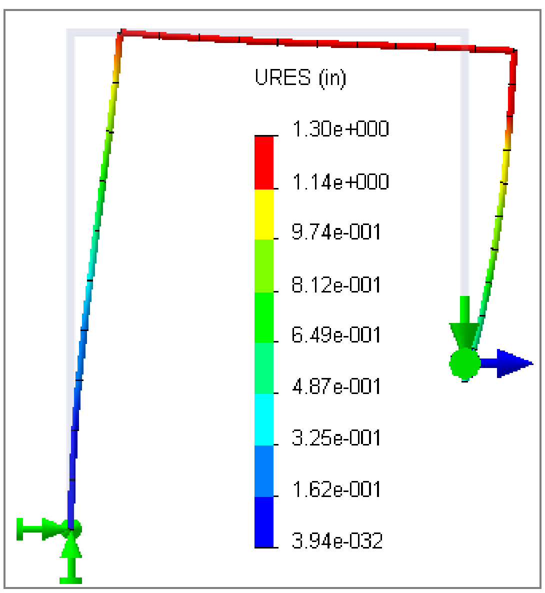


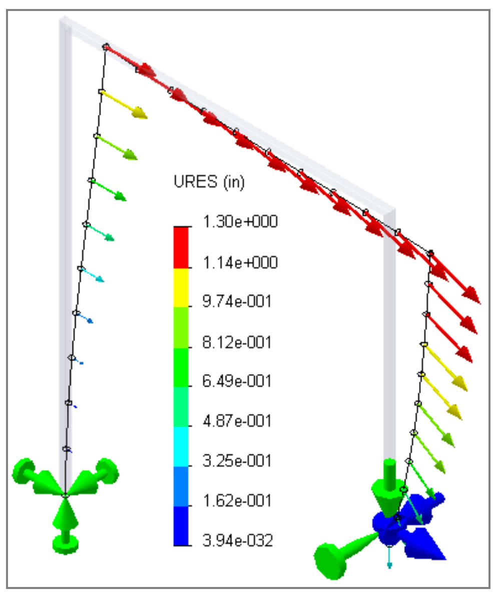
12. **Mesh→Create Mesh**. There are no mesh control options for frames. The additional created nodes allow more displacement vector displays, and better varying member stress displays.
13. **Run.** Cosmos calculates the space frame joint locations and computes the displacements.

9.3.2 Settlement load case post-processing

9.3.2.1 Displacements

There are numerous tabulated frame results given in [11]In the Cosmos menu select **Results→Define Displacement Plot→URES**, select **Fringe**, and repeat with **Vector** selected. In this case all translational displacements are in the plane of the frame:

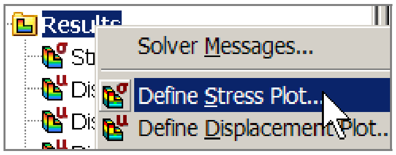




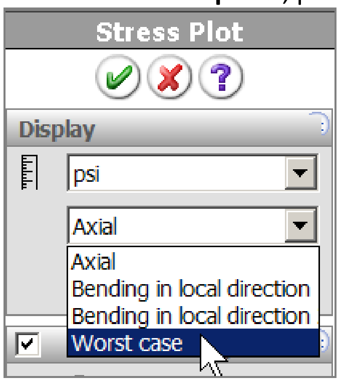
9.3.2.2 Stresses

For frame members you can have axial stress, torsional stress, and two flexural stresses combined. Therefore, the failure criterion varies over the cross-sectional area. It can also vary over the length of each element. At each end of an element Cosmos computes the stress at four (or more) locations. Cosmos can display the stress components, like axial, and/or the worst stress value:

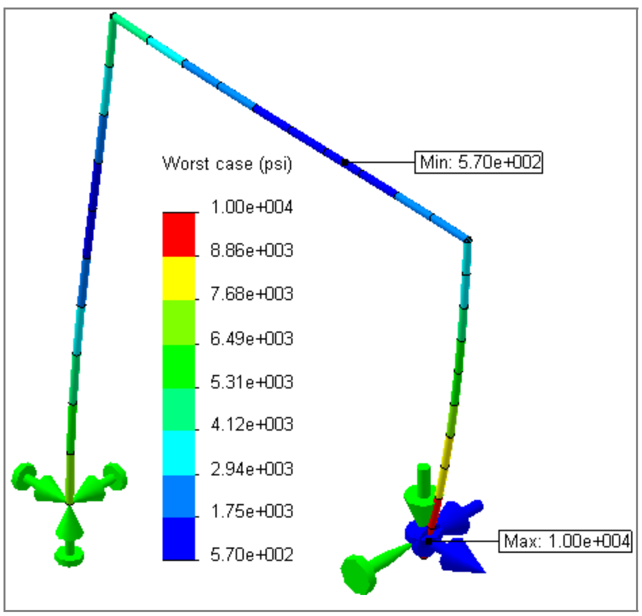
1. **Results→Define Stress Plot**



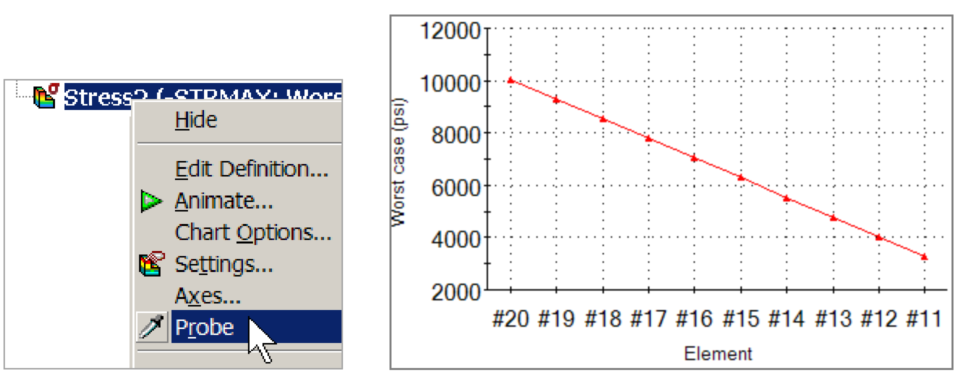
2. In the **Stress Plot panel**, pull down **Worst case**, set **units** to psi. Click **OK**.



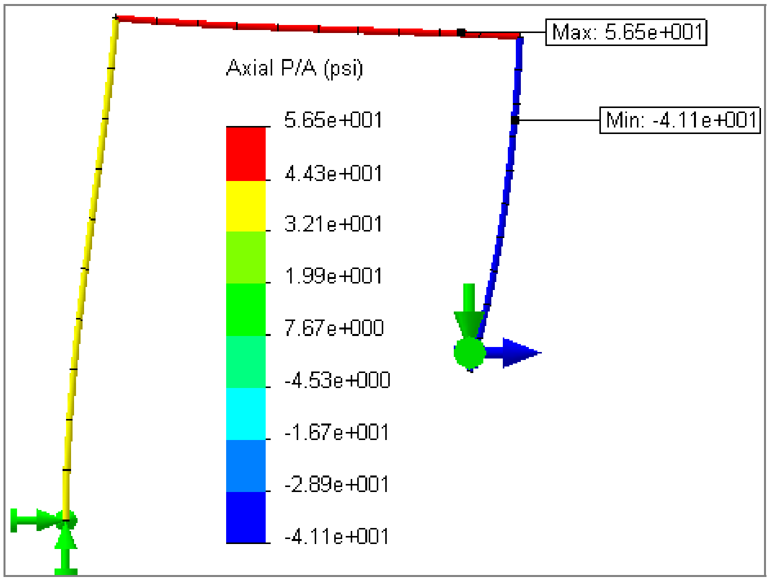
3. Review the default plot.



4. Right click on the **Stress Plot name→Probe**. Select each node on the right leg from support to corner. Pick **graph icon**, click **OK**.



5. In the **Stress Plot panel**, pull down **Axial**. Click **OK**.



9.3.2.3 Reactions

In the Cosmos Menu:

5. Right click **List Displacements** for options.
6. **List options→From** node 1 to node 23 (the mesh generator created additional nodes). **List set→Reaction forces**, click **OK**.

Results R_x

Node	Value (lb)	X (in)	Y (in)	Z (in)
1	-1.257e+002	0	0	0
23	1.257e+002	144	60	0

Results R_y

Node	Value (lb)
1	-9.155e+001
23	9.155e+001

Results R_z

Node	Value (lb)
1	2.097e-015
23	-1.664e-015

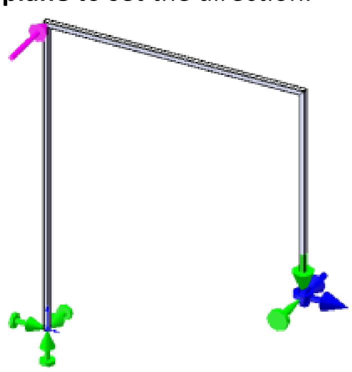
Results M_z

Node	Value (lb-in)	X (in)	Y (in)
1	1.443e+004	0	0
23	-2.007e+004	144	60

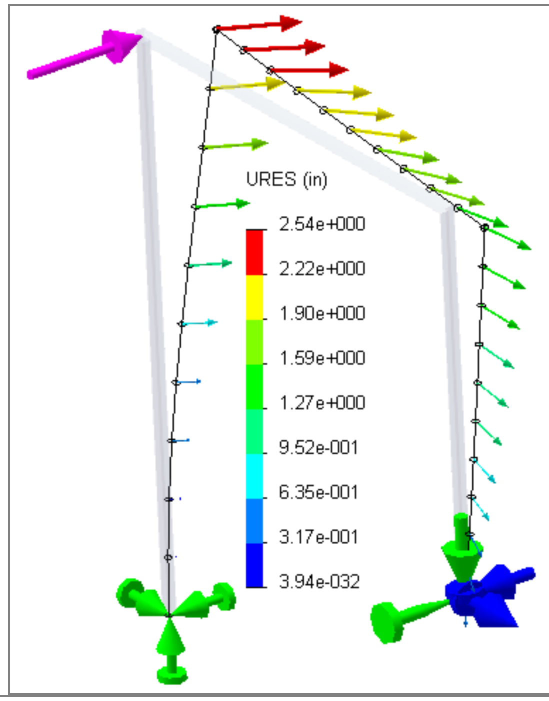
9.3.3 Settlement with transverse joint force load case

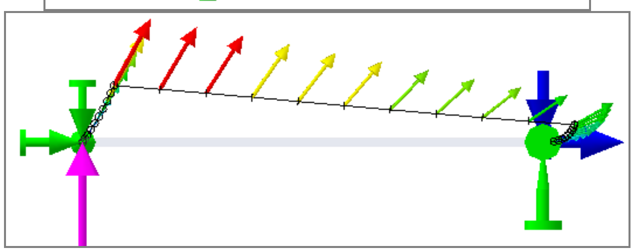
The total reaction force above was about 200 lb. Now, a normal joint force of 200 lb. is superimposed on the previous to see the out of plane displacements of this space frame:

1. **Load/Restraint→Force**. In the **Force panel→Select Joints**. Pick the top left joint and select the **Front plane** to set the direction.



2. Pick the direction as **Normal to**, and set the **Force** to 200 lb. Click **OK**.
3. **Mesh→Create Mesh**. There are no mesh control options for frames.
4. **Run**
5. **Results→Define Displacement Plot→URES**, select **Vector**. Look at the **trimetric** and **top** views.

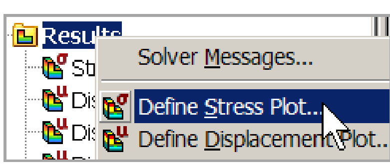


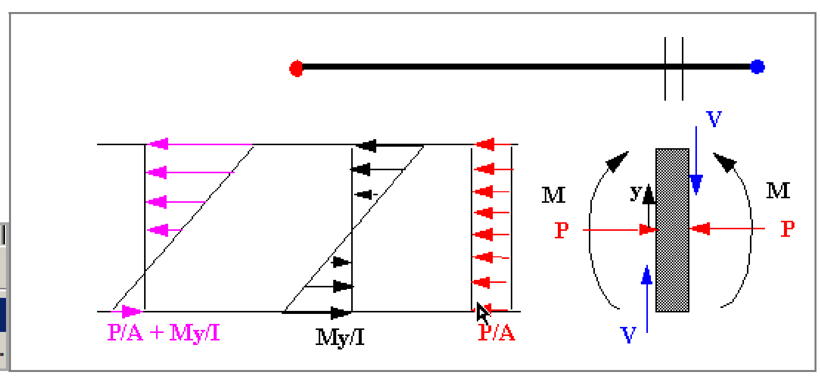


9.3.3.1 Stresses

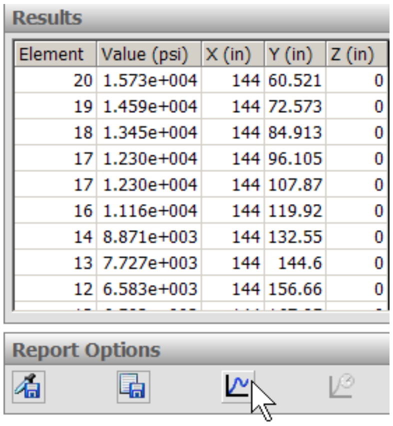
The out of plane stresses are superimposed on the previous ones above. Therefore the worst stresses will vary more around the perimeter of the cross-section. Some regions will have higher failure criterion while others have reduced values. Re-check the shorter right leg stress values:

1. **Results→Define Stress Plot**

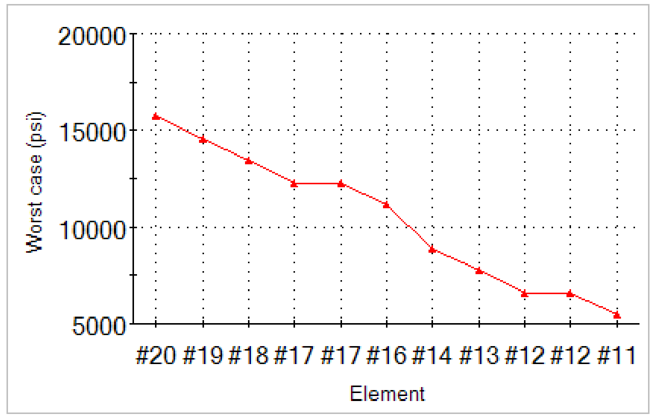




2. In the **Stress Plot panel**, pull down **Worst case**, set **units** to psi. Click **OK.**
3. Right click on the **Stress Plot name→Probe**. Select each node on the right leg from support to corner. Pick **graph icon**, click **OK**. The values have increased by about 50% are vary more with location.



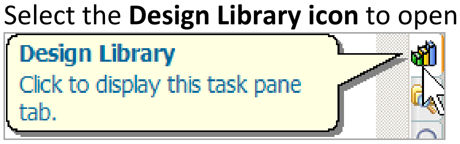
Element	Value (psi)	X (in)	Y (in)	Z (in)
20	1.573e+004	144	60.521	0
19	1.459e+004	144	72.573	0
18	1.345e+004	144	84.913	0
17	1.230e+004	144	96.105	0
17	1.230e+004	144	107.87	0
16	1.116e+004	144	119.92	0
14	8.871e+003	144	132.55	0
13	7.727e+003	144	144.6	0
12	6.583e+003	144	156.66	0



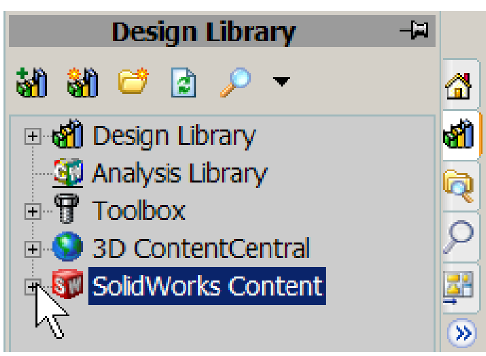
9.4 Extensive structural members library

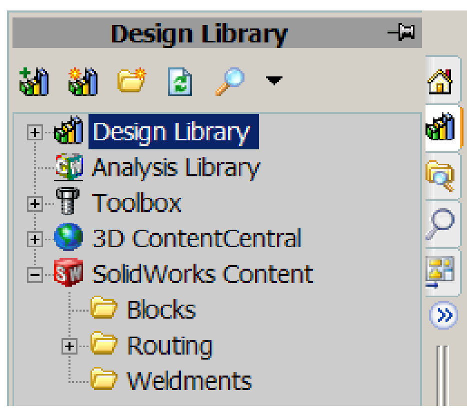
The above examples utilized only a couple of common cross-sections for the structural shapes. There is a huge number of structural shapes that have been standardized across the world, in many national and professional standards and units. They are too numerous to include in a code like Cosmos. However, they are readily available for selection and importing into Cosmos from the SolidWorks supporting web site. Therefore, they can be utilized as beams, or members of frame or truss systems. To examine the available resources:

1. Select the **Design Library icon** to open the **Design Library panel**.

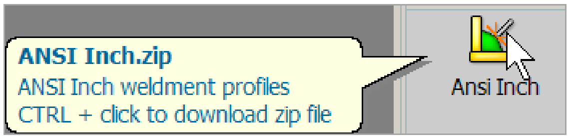


2. In the **Design Library panel** select the plus sign by **SolidWorks Content**.

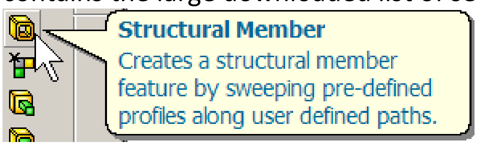




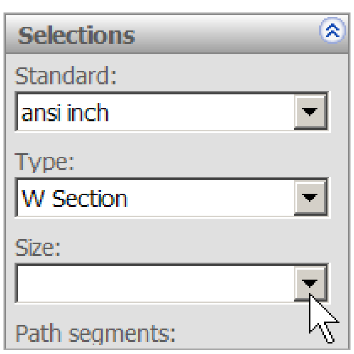
3. When the web connection is complete select **Weldments**. The list of standards appears: Ansi (American National Standards Institute) Inch, BSI (British Standards Institute), CISC (Construction Industry Standards Committee), DIN (Deutsches Institut fur Normung), GB (China Standard), ISO (International Standards Organization), JIS (Japanese Industrial Standards), Unistrut (Unistrut.com). Pick the standard you desire, in this case **Ansi Inch**. Click **OK**.

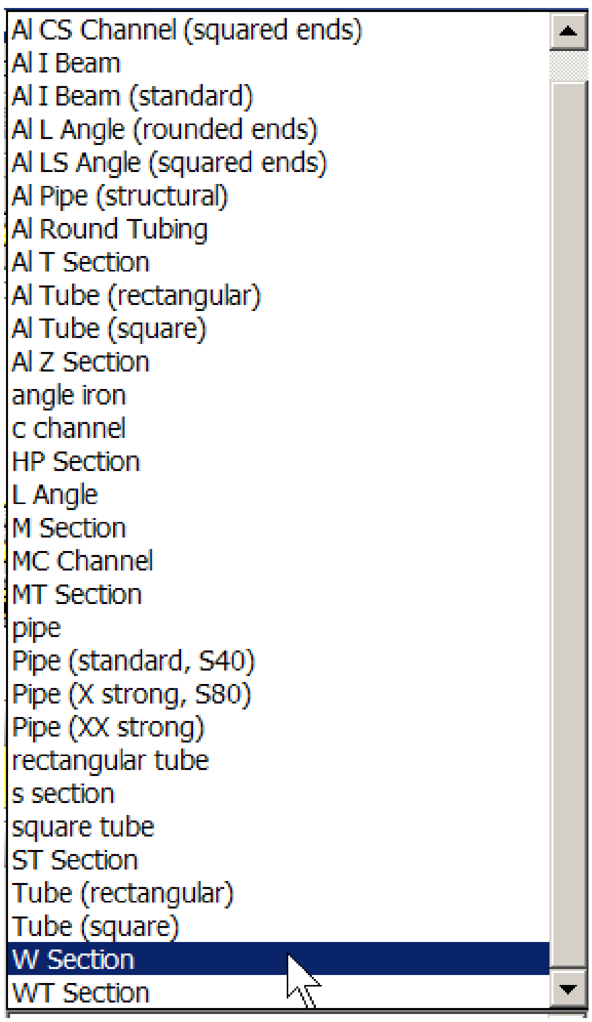


4. In SolidWorks select the **Structural Member icon** to get the **Structural Member panel**, which now contains the large downloaded list of selections.

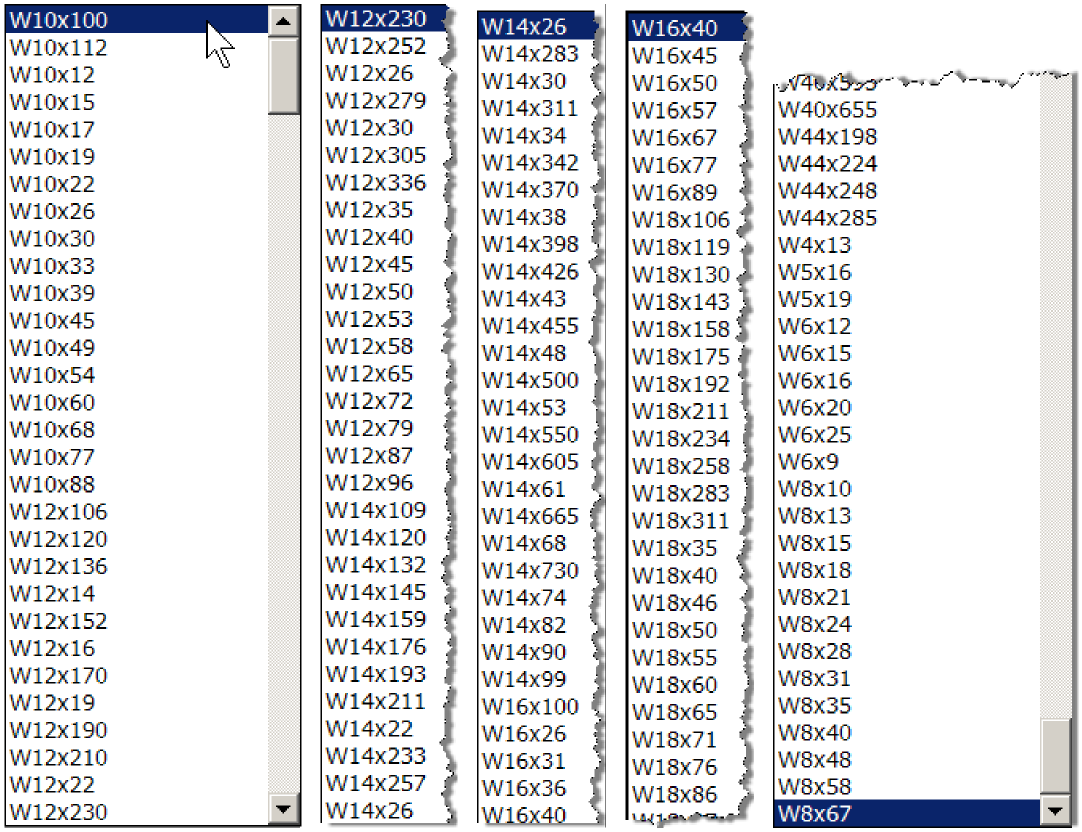


5. In the Selctions section of that panel pick the ansi inch Standard. Review the many Types now available. Chose the shape such as a wide flange (W Section) member.

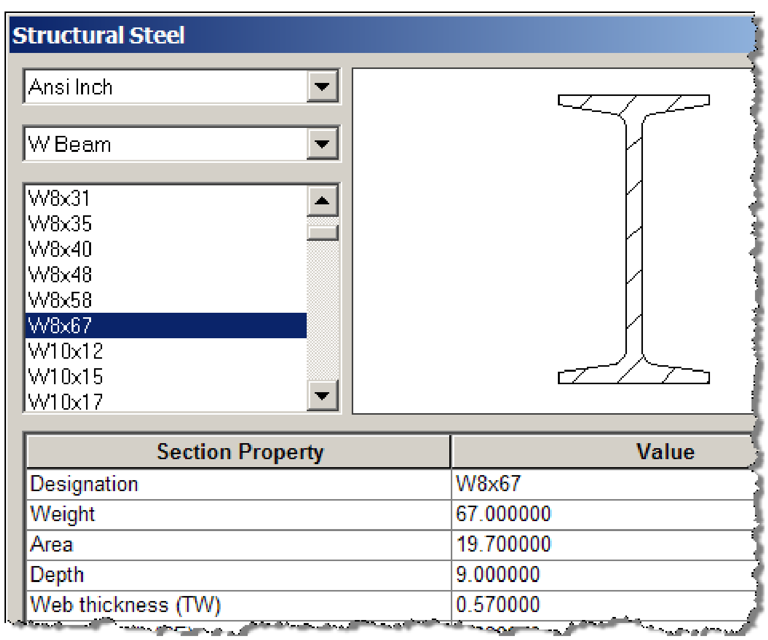




6. Pull down the **Size list** to see the many standard sizes available for that shape.



7. Select the size you want, say W8x67. Double check its numerous properties; **Toolbox→Structural Steel**. Set the data and review the cross-sectional shape drawing and the many properties.



8. Proceed with selecting the model line segments and continue your analysis as illustrated above for trusses and frames.

10 Vibration Analysis

10.1 Introduction

Consider the single degree of freedom (dof) system in Figure 10-1 that is usually introduced in a first course in physics or ordinary differential equations. There, k is the spring constant, or stiffness, and m is the mass, and c is a viscous damper. If the system is subjected to a horizontal force, say f(t), then Newton's law of motion leads to the differential equation of motion in terms of the displacement as a function of time, x(t):

$$m\, d^2x / dt^2 + c\, dx / dt + k\, x(t) = f(t)$$

which requires the initial conditions on the displacement, x(0), and velocity, v(0) = dx / dt(0). When there is no external force and no damping, then it is called free, undamped motion, or simple harmonic motion (SHM):

$$m\, d^2x / dt^2 + k\, x(t) = 0.$$

The usual simple harmonic motion assumption is $x(t) = a \sin(\omega t)$ where a is the amplitude of motion and ω is the circular frequency of the motion. Then the motion is described by

$$[k - \omega^2 m]\, a \sin(\omega t) = 0, \text{ or } [k - \omega^2 m] = 0.$$

This is solved for the circular frequency, ω, which is related to the so called natural frequency, F_n, by $F_n = \omega / 2\pi$.

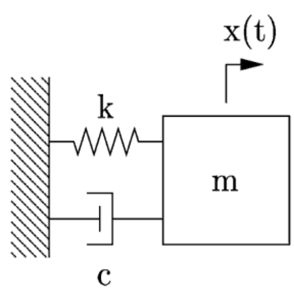


Figure 10-1 A spring-mass-damper system

10.2 Natural Frequencies

A spring and a mass interact with one another to form a system that resonates at their characteristic natural frequency. If energy is applied to a spring-mass system, it will vibrate at its natural frequency. The level of a general vibration depends on the strength of the energy source as well as the damping inherent in the system. The natural frequency of an undamped (c = 0), free (f=0), single spring-mass system is given by the following equation:

$$\omega = 2\pi F_n = \sqrt{(k/m)}$$

where F_n is the natural frequency. From this, it is seen that if the stiffness increases, the natural frequency also increases, and if the mass increases, the natural frequency decreases. If the system has damping, which all physical systems do, its frequency of response is a little lower, and depends on the amount of damping. Numerous tabulated solutions for natural frequencies and mode shape can be found in [3]. They can be useful in validating finite element calculations.

10.3 Finite element vibrations

Any physical structure vibration can be modeled by springs (stiffnesses), masses, and dampers. In elementary models you use line springs and dampers, and point masses. In finite element models, the continuous nature of the stiffness and mass leads to the use of square matrices for stiffness, mass, and damping. They can still contain special cases of line element springs and dampers, as well as point masses. Dampers dissipate energy, but springs and masses do not.

If you have a finite element system with many dof then the above single dof system generalizes to a displacement vector, **X**(t) interacting with a square mass matrix, **M**, stiffness matrix, **K**, damping matrix **C**, and externally applied force vector, **F**(t), but retains the same general form:

$$\mathbf{M}\, d^2\mathbf{X} / dt^2 + \mathbf{C}\, d\mathbf{X} / dt + \mathbf{K}\, \mathbf{X}(t) = \mathbf{F}(t)$$

plus the initial conditions on the displacement, **X**(0), and velocity, **v**(0) = d**X** / dt(0). Integrating these equations in time gives a *time history solution*. The solution concepts are basically the same, they just have to be done using matrix algebra. The corresponding SHM, or free vibration mode (**C** = **0**, **F** = **0**) for a finite element system is

$$\mathbf{M}\, d^2\mathbf{X} / dt^2 + \mathbf{K}\, \mathbf{X}(t) = \mathbf{0}.$$

The SHM assumption generalizes to **X**(t) = **A** sin (ωt) where the amplitude, **A**, is usually called the mode shape vector at circular frequency ω. This leads to the general matrix *eigenvalue problem*

$$|\, \mathbf{K} - \omega^2\, \mathbf{M}\, | = 0.$$

There is a frequency, say ω_k, and mode shape vector, $\mathbf{A}_k$, for each degree of freedom, k. A matrix eigenvalue-eigenvector solution is much more computationally expensive that a matrix time history solution. Therefore most finite element systems usually solve for the first few natural frequencies. Depending on the available computer power, that may mean 10 to 100 frequencies. CosmosWorks includes natural frequency and mode shape calculations as well as time history solutions.

Usually you are interested only in the first few natural frequencies. A zero natural frequency corresponds to a rigid body motion. If a shell model is used the rotational dof exist and the mass matrix is generalized to include the mass moments of inertia. For every natural frequency there is a corresponding vibration mode shape. Most mode shapes can generally be described as being an axial mode, torsional mode, bending mode, or general mode

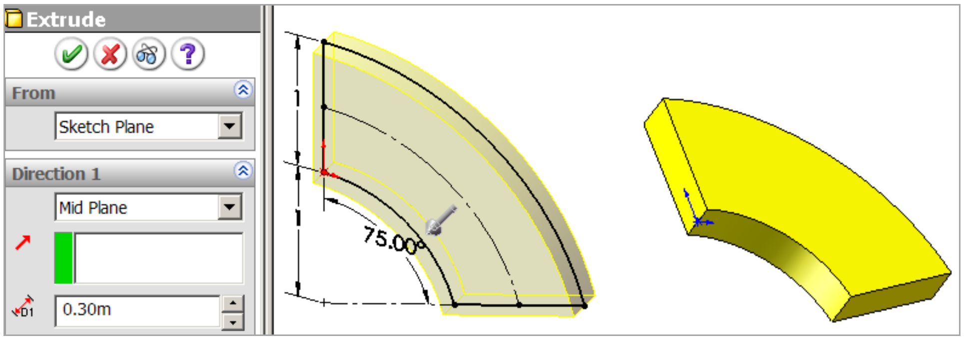
Like stress analysis models, probably the most challenging part of getting accurate finite element natural frequencies and mode shapes is to get the type and locations of the restraints correct. A crude mesh will give accurate frequency values, but not accurate stress values. TK Solver contains equations for most known analytic solutions for the frequencies of mechanical systems. They can be quite useful in validating the finite element frequency results.

10.4 Frequencies of a curved solid

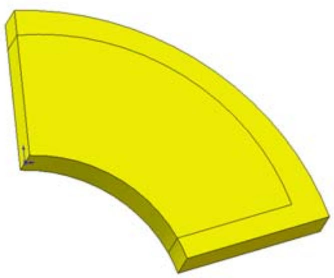
To illustrate a typical natural frequency problem consider a brass, 75 degree segment of an annulus solid having a thickness of 0.3 m, an average radius of 1.5 m, and a width of 1 m. The component is encastred at one rectangular face. The thickness to width ratio is 0.3. That suggests that the study should be conducted with either a solid model or a thick shell model. Since there is no analytic estimate to validate the study, both types will be used to indicate the range of uncertainty.

Generally, the displacement degrees of freedom are more important in getting natural frequencies and mode shapes than are rotational dof. Therefore, the solid study is probably best here. In vibration problems, the material located farthest from the supports are more important. You should use mesh control to create small elements in such regions. The modeling process is:

1. Sketch and dimension the area. **Extrude** it to a thickness of 0.3 m.



2. Click on a curved face, **Insert Sketch.**
3. Add a line and arc near the free edges farthest from the support, for later mesh control.
4. **Insert→Curve→Split Line**



10.4.1 CosmosWorks frequency studies

10.4.1.1 CosmosWorks Manager

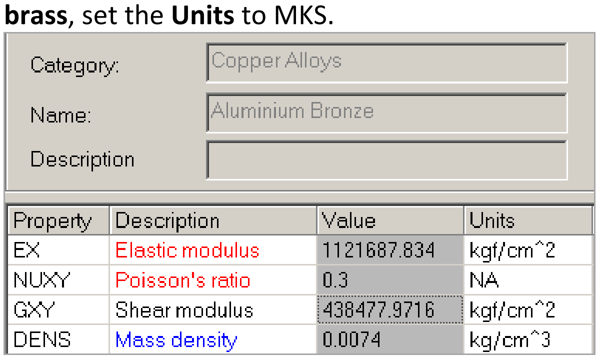
Selecting the **CosmosWorks Manager** (CWManager) icon:

4. Right click on the top name to access **Study** which opens the **Study panel**.
5. Assign a **Study name**, choose **Frequency** for the **Analysis type.**
6. Define the **Mesh type** to be **solid,** click **OK**.

10.4.1.2 Define the material

At this point **Solids** will appear in the CWManager menu:

1. Right click on it to apply material data. The component is to be made of brass.
2. Pick **Apply Material to All→ Material panel→From library files button→Copper Alloys** and select **brass**, set the **Units** to MKS.



Category: Copper Alloys

Name: Aluminium Bronze

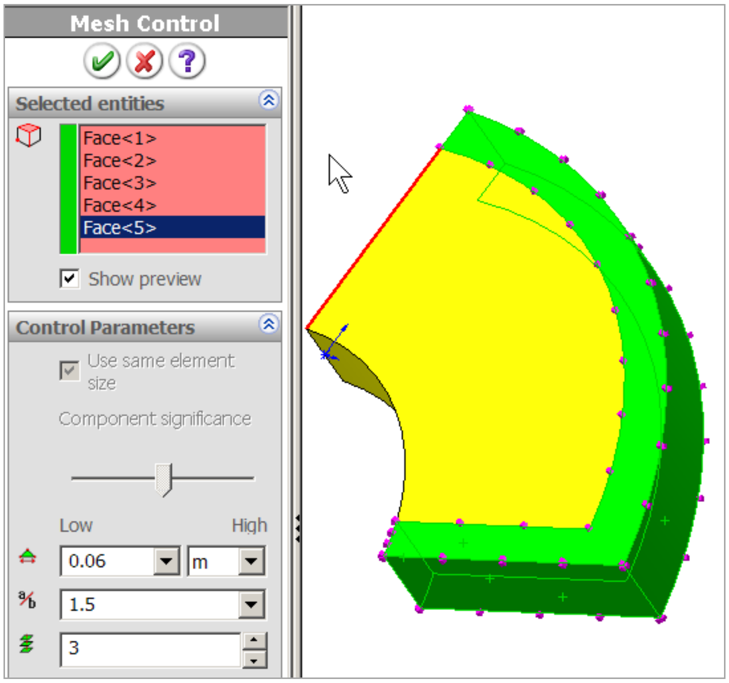
Description:

Property	Description	Value	Units
EX	Elastic modulus	1121687.834	kgf/cm^2
NUXY	Poisson's ratio	0.3	NA
GXY	Shear modulus	438477.9716	kgf/cm^2
DENS	Mass density	0.0074	kg/cm^3

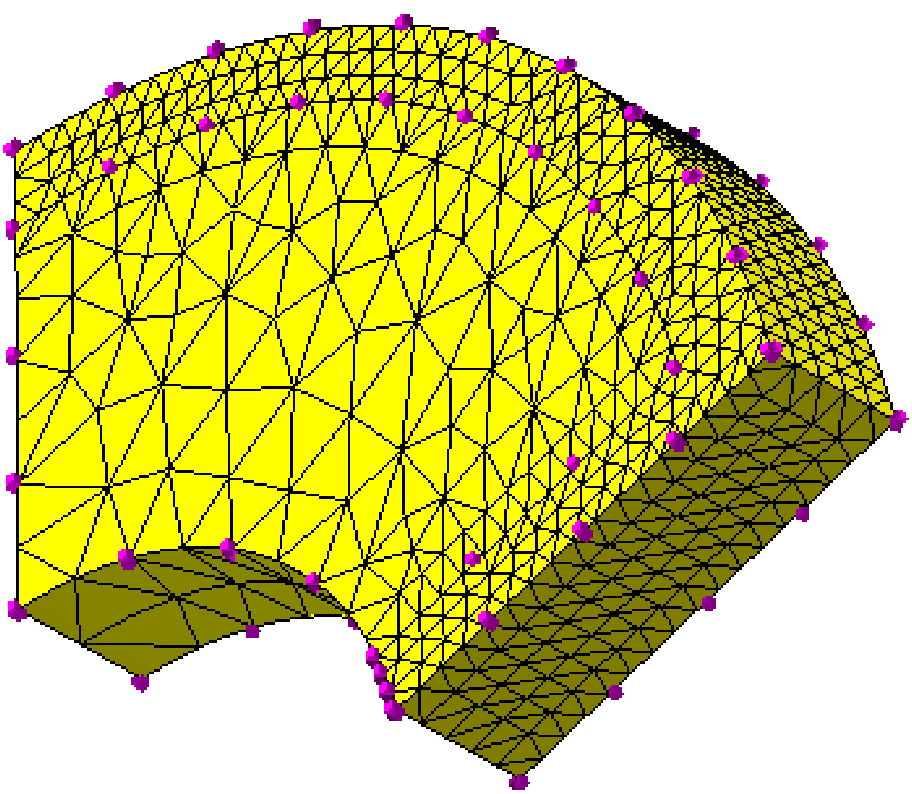
10.4.1.3 Meshing

Specify a finer mesh away from the support, and a crude mesh near the support:

1. **Mesh→Mesh Control**, select small outer faces, set size to 0.06 m.

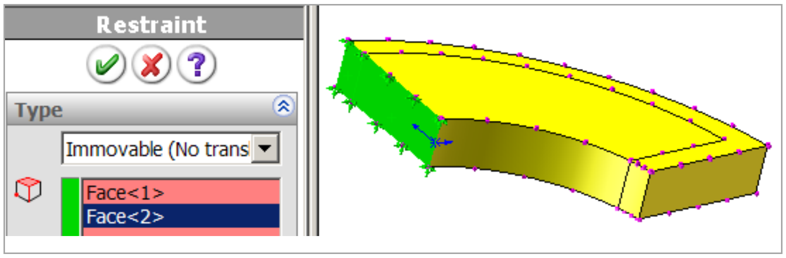


2. **Mesh→Mesh Control**, select other faces, set size to 0.3 m.
3. **Mesh→Create Mesh**



10.4.1.4 Restrain the system

4. Select **Load/Restraint→Restraints→Immovable** and pick the support rectangles. Click **OK**. **Run**.



10.4.1.5 Post-process the frequencies and mode shapes

The Run properties were set to compute five modes and frequencies, but only the first three are summarized here. Select Results and display each mode in turn. Change views for better understanding as in Figure 10-2. Mode one is like that of a cantilever beam, with the outer edge moving perpendicular to the original plane. Mode two is a vibration in the original plane. Mode three seems to be mainly a twisting vibration. The frequencies are shown in the figure text. You can also have Cosmos list them. The first three modes are also given in Table 10-1, along with the corresponding values from a thick shell model presented below. There is about a 10% difference in the computed frequencies.

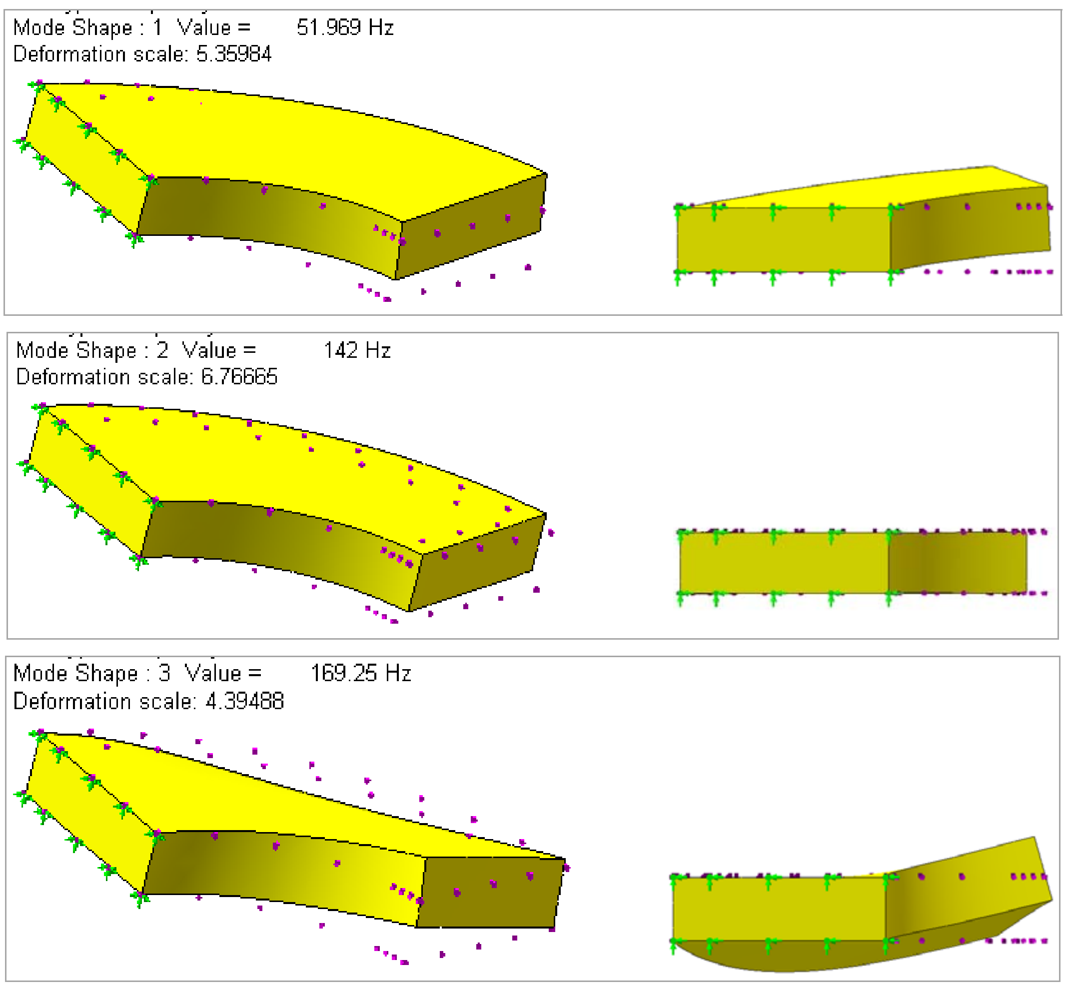


Figure 10-2 First three solid studies modes and frequencies

Table 10-1 Natural frequencies (Hz) from solids and thick shells

Model	Mode 1	Mode 2	Mode 3
Solid	52	142	169
Thick shell	46	126	155

10.4.2 Thick shell version

The above study was repeated with a thick shell and the same mesh controls. Some results are in Figure 10-3

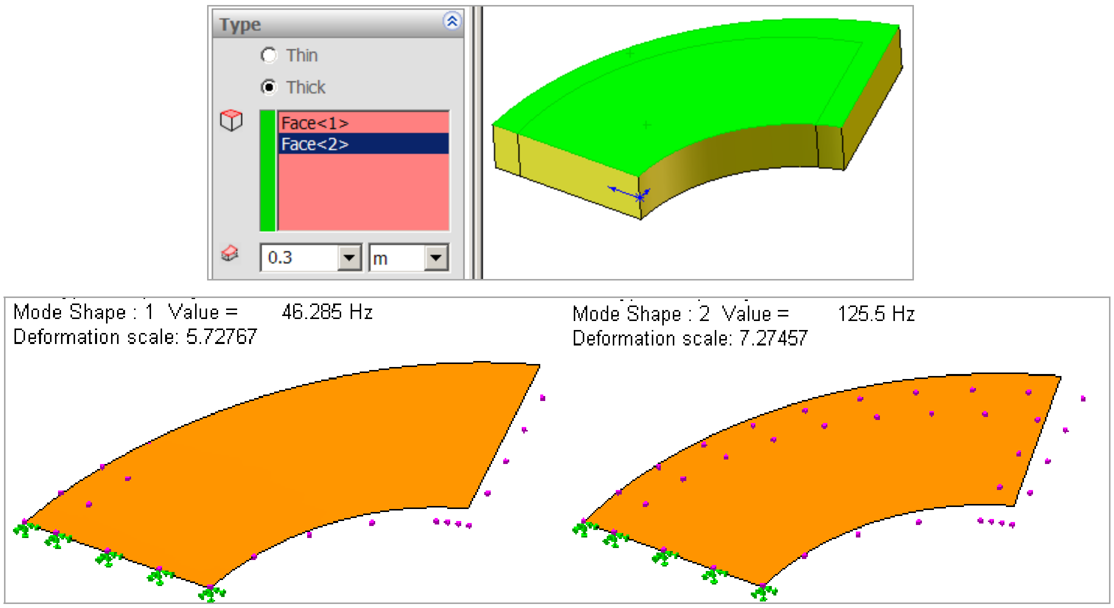


Figure 10-3 First two thick shell frequencies

10.5 Influencing the natural frequency

If you wish to influence the natural frequency you can automate the process by employing the Cosmos optimization ability to vary the part geometric design parameters. You can also get a feel for the controlling factors by noting the fact that the natural frequencies (in Hz) of plates can generally be expressed as

$$f_j = \frac{\lambda_j}{2\pi L^2}\left[\frac{E\,h^3}{12\,\rho(1-\nu^2)}\right]^{1/2}$$

where f_j is the natural frequency, E is the elastic modulus, ν is Poisson's ratio, ρ is the mass density, h is the (thin) plate thickness, L is a characteristic length of the plate. The remaining factor, λ_j, is dependent on the support conditions and geometric shape of the plate. It is often a tabulated feature in standard handbooks like [3, 11]. Usually, the thickness and length are the easiest features to change. The quantity in square brackets reduces to [E I/ρ] for a straight beam.

11 Buckling Analysis

11.1 Introduction

There are two major categories leading to the failure of a mechanical component: material failure and structural instability, which is often called buckling. For material failures you need to consider the yield stress for ductile materials and the ultimate stress for brittle materials.

Those material properties are determined by axial tension tests and axial compression tests of short columns of the material (see Figure 11-1). The geometry of such test specimens has been standardized. Thus, geometry is not specifically addressed in defining material properties, such as yield stress. Geometry enters the problem of determining material failure only indirectly as the stresses are calculated by analytic or numerical methods.

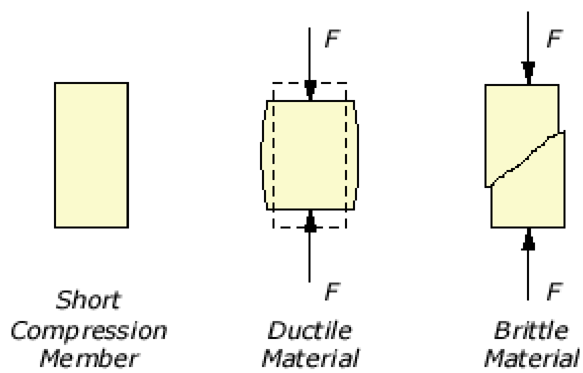


Figure 11-1 Short columns fail due to material failure

Predicting material failure may be accomplished using linear finite element analysis. That is, by solving a linear algebraic system for the unknown displacements, $\boldsymbol{K}\,\boldsymbol{\delta} = \boldsymbol{F}$. The strains and corresponding stresses obtained from this analysis are compared to design stress (or strain) allowables everywhere within the component. If the finite element solution indicates regions where these allowables are exceeded, it is assumed that material failure has occurred.

The load at which buckling occurs depends on the stiffness of a component, not upon the strength of its materials. Buckling refers to the loss of stability of a component and is usually independent of material strength. This loss of stability usually occurs within the elastic range of the material. The two phenomenon are governed by different differential equations [18]. Buckling failure is primarily characterized by a loss of structural stiffness and is not modeled by the usual linear finite element analysis, but by a finite element eigenvalue-eigenvector solution, $|\boldsymbol{K} + \lambda_m \boldsymbol{K_F}|\ \boldsymbol{\delta}_m = 0$, where λ_m is the buckling load factor (BLF) for the m-th mode, $\boldsymbol{K_F}$ is the additional "geometric stiffness" due to the stresses caused by the loading, $\boldsymbol{F}$, and $\boldsymbol{\delta}_m$ is the associated buckling displacement shape for the m-th mode. The spatial distribution of the load is important, but its relative magnitude is not. The buckling calculation gives a multiplier that scales the magnitude of the load (up or down) to that required to cause buckling.

Slender or thin-walled components under compressive stress are susceptible to buckling. Most people have observed what is called "Euler buckling" where a long slender member subject to a compressive force moves lateral to the direction of that force, as illustrated in Figure 11-2. The force, F, necessary to cause such a buckling motion will vary by a factor of four depending only on how the two ends are restrained. Therefore, buckling studies are much more sensitive to the component restraints that in a normal stress analysis. The theoretical Euler solution will lead to infinite forces in very short columns, and that clearly exceeds the allowed material stress. Thus in practice, Euler column buckling can only be applied in certain regions and empirical transition equations are required for intermediate length columns. For very long columns the loss of stiffness occurs at stresses far below the material failure.

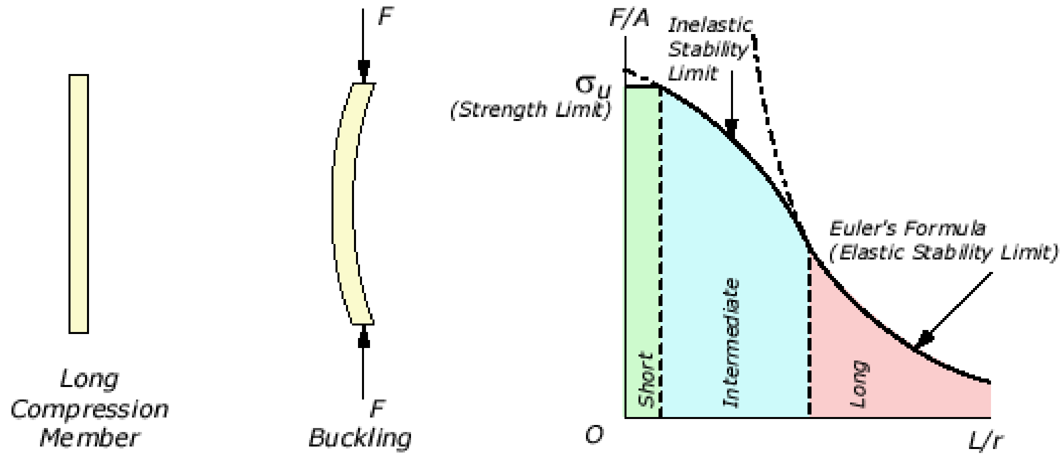


Figure 11-2 Long columns fail due to instability

There are many analytic solutions for idealized components having elastic instability. About 75 of the most common cases are tabulated in the classic reference "Roark's Formulas for Stress and Strain" [15-17], and in the handbook by Pilkey [11].

11.2 Buckling terminology

The topic of buckling is still unclear because the keywords of "stiffness", "long" and "slender" have not been quantified. Most of those concepts were developed historically from 1D studies. You need to understand those terms even though finite element analysis lets you conduct buckling studies in 1D, 2D, and 3D. For a material, stiffness refers to either its elastic modulus, E, or to its shear modulus, $G = E / (2 + 2 v)$ where v is Poisson's ratio.

Slender is a geometric concept of a two-dimensional area that is quantified by the radius of gyration. The radius of gyration, r, has the units of length and describes the way in which the area of a cross-section is distributed around its centroidal axis. If the area is concentrated far from the centroidal axis it will have a greater value of r and a greater resistance to buckling. A non-circular cross-section will have two values for its radius of gyration. The section tends to buckle around the axis with the smallest value. The radius of gyration, r, is defined as:

$$r = (I / A)^{1/2},$$

where I and A are the area moment of inertia, and area of the cross-section. For a circle of radius R, you obtain $r = R / 2$. For a rectangle of large length R and small length b you obtain $r_{max} = R / 2\sqrt{3} = 0.29\ R$ and $r_{min} = 0.29\ b$. Solids can have regions that are slender, and if they carry compressive stresses a buckling study is justified.

Long is also a geometric concept that is quantified by the non-dimensional "slenderness ratio" L / r, where L denotes the length of the component. The slenderness ratio is defined to be long when it obeys the inequality

$$L / r > (\pi / k)\ (2E / \sigma_y)^{1/2}$$

where k is a constant that depends on the restraints of the two ends of the column. A long slenderness ratio is typically in the range of >120. The above equation is the dividing point between long (Euler) columns and intermediate (empirical) columns. The critical compressive stress that will cause buckling always decreases as the slenderness ratio increases.

Euler long column buckling is quite sensitive to the end restraints. Figure 11-3 shows five of several cases of end restraints and the associated *k* value used in both the limiting slenderness ratio and the buckling load or stress. The critical buckling force is

$$F_{Euler} = k\,\pi^2 E\,I / L^2 = k\,\pi^2 E\,A / (L / r)^2$$

So the critical Euler buckling stress is

$$\sigma_{Euler} = F_{Euler} / A = k\,\pi^2 E / (L / r)^2 .$$

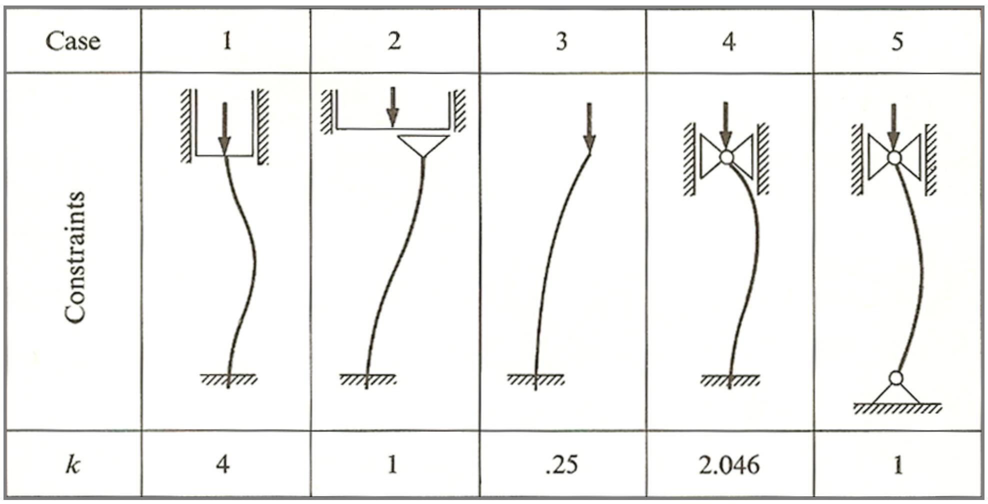


Figure 11-3 Restraints have a large influence on the critical buckling load

11.3 Buckling Load Factor

The buckling load factor (BLF) is an indicator of the factor of safety against buckling or the ratio of the buckling loads to the currently applied loads. Table 11-1 Interpretation of the Buckling Load Factor (BLF) illustrates the interpretation of possible BLF values returned by CosmosWorks. Since buckling often leads to bad or even catastrophic results, you should utilize a high factor of safety (FOS) for buckling loads. That is, the value of unity in Table 11-1 Interpretation of the Buckling Load Factor (BLF) should be replaced with the FOS value.

Table 11-1 Interpretation of the Buckling Load Factor (BLF)

BLF Value	Buckling Status	Remarks
>1	Buckling not predicted	The applied loads are less than the estimated critical loads.
= 1	Buckling predicted	The applied loads are exactly equal to the critical loads. Buckling is expected.
< 1	Buckling predicted	The applied loads exceed the estimated critical loads. Buckling will occur.
-1 < BLF < 0	Buckling possible	Buckling is predicted if you reverse the load directions.
-1	Buckling possible	Buckling is expected if you reverse the load directions.
< -1	Buckling not predicted	The applied loads are less than the estimated critical loads, even if you reverse their directions.

11.4 General buckling concepts

Other 1D concepts that relate to stiffness are: axial stiffness, E A / L, flexural (bending) stiffness, E I / L, and torsional stiffness, G J / L, where J is the polar moment of inertia of the cross-sectional area (J = Iz = Ix + Iy). Today, stiffness usually refers to the finite element stiffness matrix, which can include all of the above stiffness terms plus general solid or shell stiffness contributions. Analytic buckling studies identify additional classes of instability besides Euler buckling (see

Figure 11-4 and Figure 11-5). They include lateral buckling, torsional buckling, and other buckling modes. A finite element buckling study determines the lowest buckling factors and their corresponding displacement modes. The amplitude of a buckling displacement mode, $|\delta_m|$, is arbitrary and not useful, but the shape of the mode can suggest whether lateral, torsional, or other behavior is governing the buckling response of a design

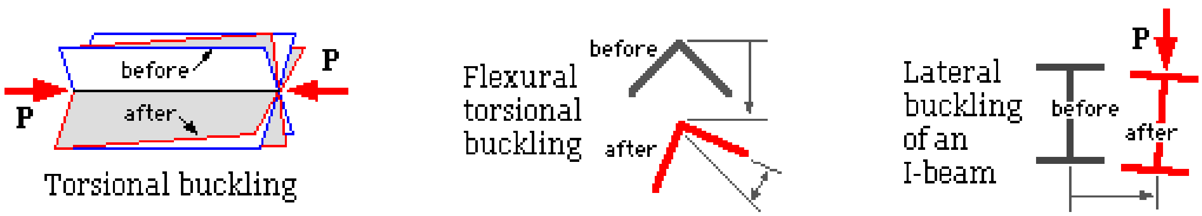


Figure 11-4 Some sample buckling shapes

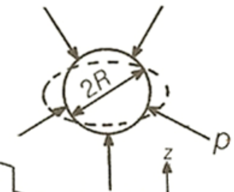
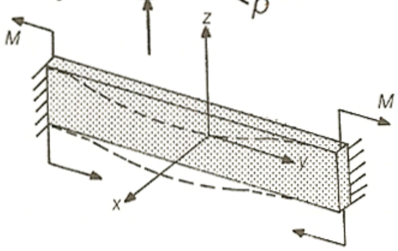
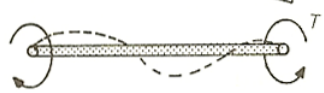
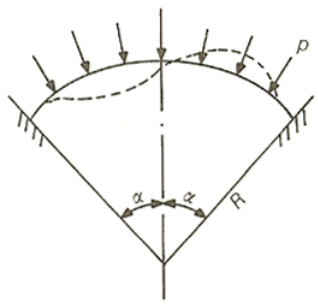
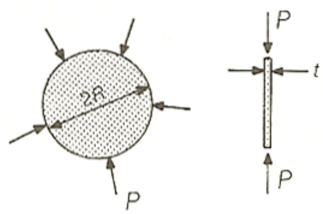
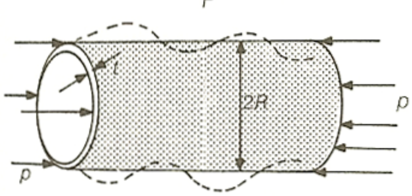
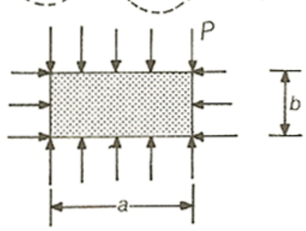
Geometry	Description	Critical load
	Buckling of a ring or tube of thickness t under external pressure p_{cr}	$p_{cr} = 3EI/R^3$ $I = t^3/12$
	Lateral buckling of a thin beam under a moment on its plane $I_p = I_x + l_z$, Length L	Hinged ends: $M_{cr} = \frac{\pi}{L}(EI_z GI_p)^{\frac{1}{2}}$ Fixed ends: $M_{cr} = \frac{2\pi}{L}(EI_z GI_p)^{\frac{1}{2}}$
	Torsional buckling of a thin rod in torsion	$T_{cr} = 2\pi EI_p/L$
	Arch or shell clamped at both ends	$p_{cr} = EI(k^2 - 1)/R^3$ $k \tan\alpha \cot k\alpha = 1$
	Column on elastic foundation constant $\beta = \bar{\beta} L^4/\pi^4 EI$ $\bar{\beta}$: soil modulus (Table 9.6)	B.C. — $P_{cr} =$ pinned–pinned — $2(\beta EI)^{\frac{1}{2}}$ free–free — $(\beta EI)^{\frac{1}{2}}$
	Circular plate force P per unit arc length	Boundary — $P_{cr} =$ free — $2.88t^2E/R^2$ fixed — $9.79t^3E/R^2(1 - \nu^2)$
	Long tube under axial thrust pressure p	$p_{cr} = \frac{2tE}{R[3(1-\nu^2)]^{\frac{1}{2}}} \cdot \frac{t}{R}$
	Rectangular plate of thickness t under uniform force per unit length P	$P_{cr} = \frac{\pi^2 Eh^2}{3(1-\nu^2)}\left(\frac{1}{a^2} + \frac{1}{b^2}\right)$

Figure 11-5 Critical buckling states for common components

11.5 Local Buckling of a Cantilever

11.5.1 Background

You previously went through the analysis of a horizontal tapered cantilever subject to a transverse load distributed over its free end face. The fixed support at the wall included a semi-circular section of the supporting vertical section. The member was *L = 50 inch* long, *t = 2 inch* thick, and the depth, *d*, tapered from 3 inch at the load, to 9 inch at the support. A complete plane stress analysis was conducted. The computed stresses were relatively low. It was decided to save material costs by reducing the thickness of the beam.

11.5.2 Factor of Safety

For the ductile material used here the factor of safety (FOS) is defined as the material yield stress divided by the von Mises' effective stress. To view its distribution the default results plot is opened with **Design check→Plot 1**, which is shown in Figure 11-6.

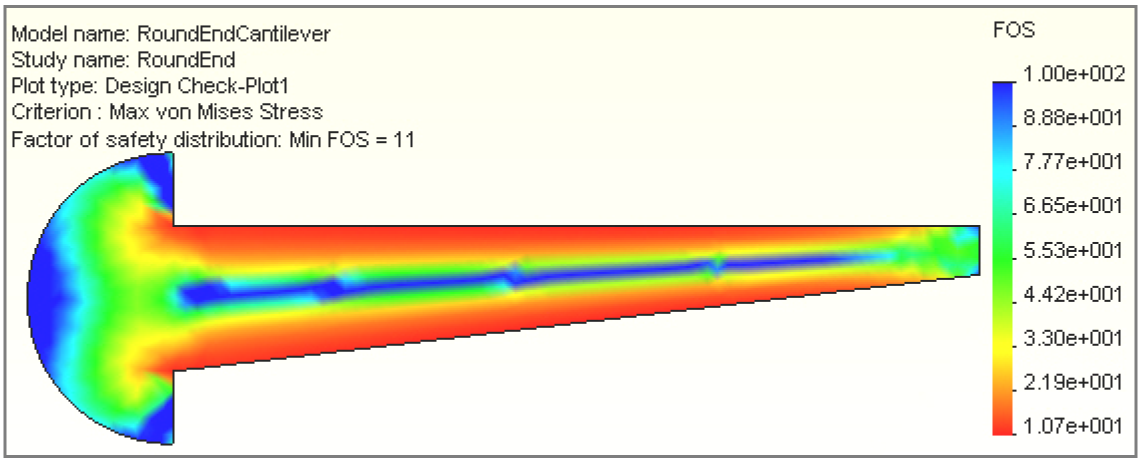


Figure 11-6 Original factor of safety in bending

The FOS is also quite high, ranging from a low value of about 10 to a high value of about 100. This probably suggests (incorrectly) that a simple redesign will save material, and thus money. The load carrying capacity of a beam is directly proportional to its geometric moment of inertia, $I_z = t\, d^3 / 12$. Thus, it also is proportional to its thickness, *t*. Therefore, it appears that you could simply reduce the thickness from t = 2 to 0.2 inches and your FOS would still be above unity. If you did that then the "thickness to depth ratio" would vary from *0.2 / 3 = 0.067* at the load to *0.2 / 9 = 0.022* at the wall, a range of about 1/15 to 1/45. The edited extrusion feature is given in Figure 11-7 employs that thickness.

11.5.3 Local buckling

If component has a region where the relative thickness to depth ratio of less than 1/10 you should consider the possibility of "*local buckling*". It usually is a rare occurrence, but when it does occur the results can be sudden and catastrophic. To double check the safety of reducing the thickness you should add a second study that utilizes the CosmosWorks buckling feature to determine the lowest buckling load. To do that:

1. Right click on the **Part name→Study** to open the **Study panel**.
2. Assign a new **Study name**, select Buckling as the **Type of analysis**, and use the thin shell as the **Model type**, click **OK**.
3. To use the same loads and restraints **drag** the **Load/Restraints** from the first study and **drop** them into the second one.

4. Likewise, **drag and drop** the first shell **Materials** into the second study.
5. Create a new finer mesh, or drag and drop the first mesh.
6. Right click on the **Part name→Run**.

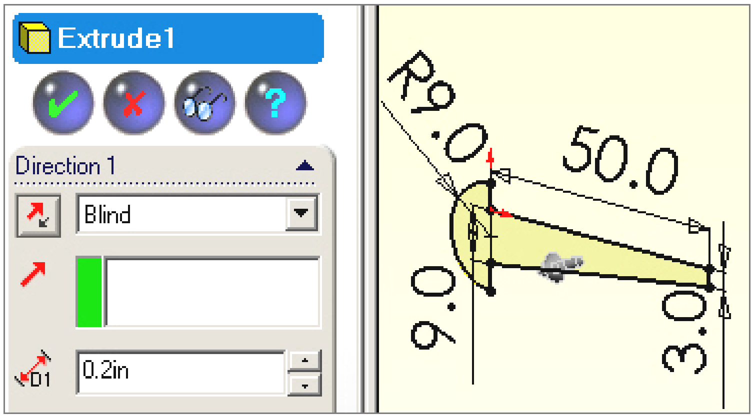


Figure 11-7 The reduced thickness extrusion

11.5.4 Buckling mode

A buckling, or stability, analysis is an eigen-problem. The magnitude of the scalar eigen- value is called the "*buckling load factor*", BLF. The computed displacement eigen-vector is referred to as the "*buckling mode*" or mode shape. They are only relative displacements. Usually they are presented in a non-dimensional fashion where the displacements range from zero to ±1. In other words, the actual value or units of a buckling mode shape are not important. Still, it is wise to carry out a visual check of the first buckling mode:

1. When the solution completes, pick **Displacements→Plot1** and examine the resultant displacement URES. Note that the displacement contour curves in Figure 11-8 are inclined to the long axis of the beam instead of being vertical as before.
2. Use **Edit Definition→Vector→Line** to get a plot of the displacement vectors, and rotate to an out-of-plane view, as shown in Figure 11-9.

From Figure 11-9 you see that under the vertical load the (very thin) beam deflected mainly sideways (perpendicular to the load) rather than downward. This is an example of lateral buckling. That is typical of what can happen to very thin regions. Next, the question is: how large must the end load be to cause such motion, and failure?

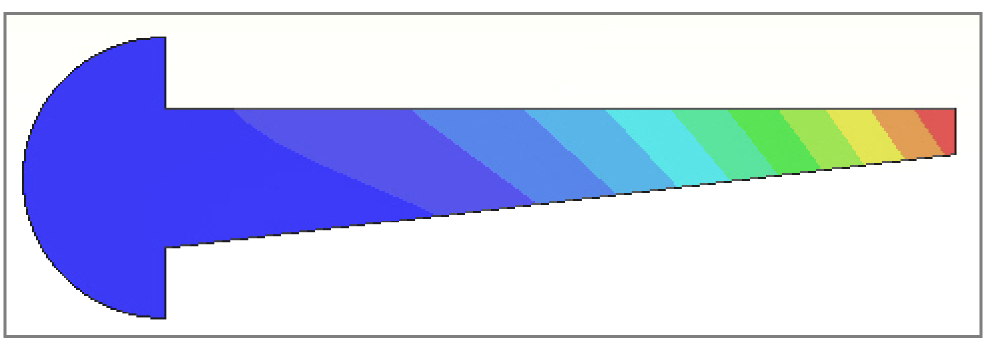


Figure 11-8 Relative buckling mode displacement values

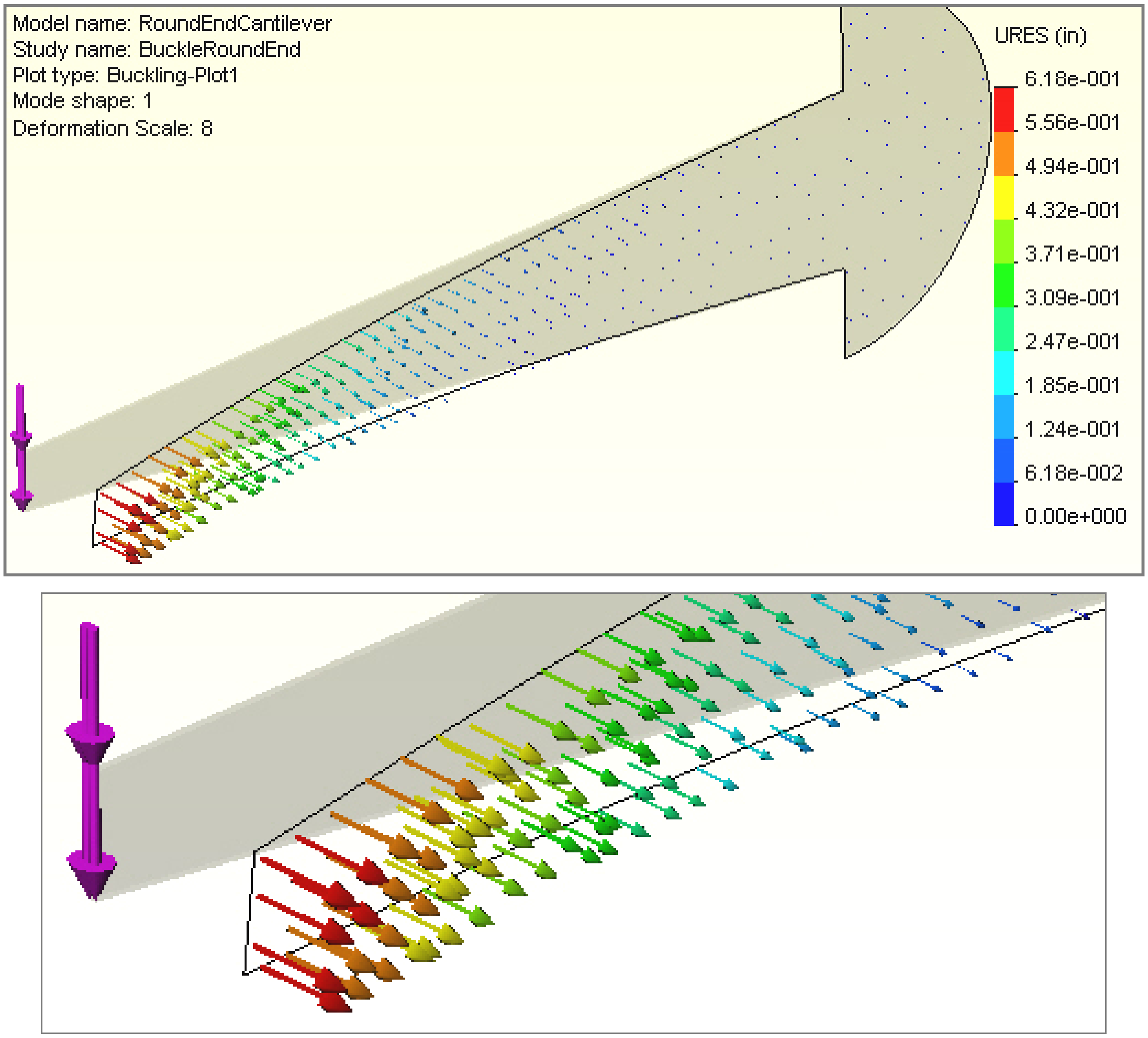


Figure 11-9 Relative lateral buckling mode displacement vectors

11.5.5 Buckling Load Factor

To see the magnitude of the BLF (eigenvalue):

1. Right click on **Deformation→List Mode Shape**.
2. In the **Mode Shape panel**, Figure 11-10, read the BLF value of about 0.03.

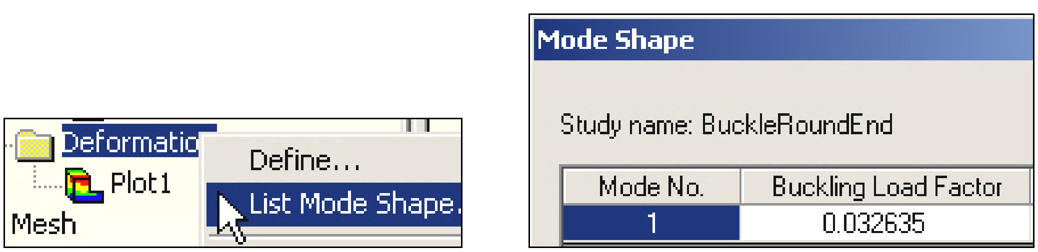


Figure 11-10 First buckling mode load factor

You want the BLF to be quite a bit higher than unity. Instead, the study shows that only about 3% of the planned load will cause this member to fail by lateral buckling due to loss of stiffness in the out of plane direction. Thus, you must re-consider the thickness reduction. Remember that the geometric moment of inertia about the vertical (y) axis is $I_y = d\, t^3 / 12$. It is a measure of the lateral bending resistance. By reducing the thickness, *t*, by a factor of 10 the original I_z (and the in-plane bending resistance) went down by the same factor of 10, but I_y (and the out-of-plane bending resistance) went down by a factor of 1,000.

The buckling load factor is an indicator of the factor of safety against buckling or the ratio of the buckling loads to the currently applied loads. Since buckling often leads to bad or even catastrophic results, you should utilize a high factor of safety (>3) for buckling loads.

12 Concepts of Thermal Analysis

12.1 Introduction

There are three different types of **heat transfer**: conduction, convection, and radiation. A temperature difference must exist for heat transfer to occur. Heat is always transferred in the *direction* of decreasing temperature. Temperature is a scalar, but heat flux is a vector quantity.

Conduction takes place within the boundaries of a body by the diffusion of its internal energy. The temperature within the body, *T*, is given in units of degrees Celsius [C], Fahrenheit [F], Kelvin [K], or Rankin [R]. Its variation in space defines the temperature gradient vector, ∇T, with units of [K/m] say. The heat flux vector, ***q***, is define by Fourier's Conduction Law, as the thermal conductivity, *k*, times the negative of the temperature gradient, $\boldsymbol{q} = - k \nabla T$. Thermal conductivity has the units of [W/m-K] while the heat flux has units of [W/m^2]. The conductivity, *k*, is usually only known to two or three significant figures. For solids it ranges from about 417 [W/m-K] for silver down to 0.76 [W/m-K] for glass.

A perfect insulator material ($k \equiv 0$) will not conduct heat; therefore the heat flux vector must be parallel to the insulator surface. A plane of symmetry (where the geometry, *k* values, and heat sources are mirror images) acts as a perfect insulator. In finite element analysis, all surfaces default to perfect insulators unless you give a specified temperature, a known heat influx, a convection condition, or a radiation condition.

Convection occurs in a fluid by mixing. Here we will consider only *free convection* from the surface of a body to the surrounding fluid. *Forced convection*, which requires a coupled mass transfer, will not be considered. The magnitude of the heat flux normal to a solid surface by free convection is $q_n = h A_h (T_h - T_f)$ where *h* is the convection coefficient, A_h is the surface area contacting the fluid, T_h is the convecting surface temperature, and T_f is the surrounding fluid temperature, respectively. The units of *h* are [W/m^2-K]. Its value varies widely and is usually known only from one to four significant figures. Typical values for convection to air and water are 5-25 and 500-1000 [W/m^2-K], respectively.

Radiation heat transfer occurs by electromagnetic radiation between the surfaces of a body and the surrounding medium. It is a highly nonlinear function of the absolute temperatures of the body and medium. The magnitude of the heat flux normal to a solid surface by radiation is $q_r = \varepsilon \sigma A_r (T_r^{\,4} - T_m^{\,4})$. Here T_r is the absolute temperature of the body surface, T_m is the absolute temperature of the surrounding medium, A_r is the body surface area subjected to radiation, σ = 5.67 x 10^8 [W/m^2 – K^4] is the Stefan-Boltzmann constant, and ε is a surface factor (ε = 1 for a perfect black body).

Transient, or unsteady, heat transfer in time also requires the material properties of specific heat at constant pressure, c_p in [kJ/kg-K], and the mass density, *ρ* in [kg/m^3]. The specific heat is typically known to 2 or 3 significant figures, while the mass density is probably the most accurately known material property with 4 to 5 significant figures.

Useful conversions are: Energy, 1 J = 1 N-m, 1 BTU = 1.055 kJ. Power, 1 W = 1 J/s, 1 BTU/s = 1055 W. Temperature, K = 5/9 R, C = K – 273.15, F = 9/5 C + 32. Temperature difference, Δ C = Δ K, Δ R = 5/9 Δ K, Δ F = 5/9 Δ C.

12.1.1 One-dimensional thermal-structural analogy

The one-dimensional governing differential equation for transient heat transfer is

$$\partial(A(x)\, k_x(x)\, \partial T / \partial x) / \partial x + Q(x)\, A(x) = \rho\, c_p\, A(x)\, \partial T / \partial t,$$

or for the common case of a constant area, A:

$$\partial(k_x \partial T / \partial x) / \partial x + Q(x) = \rho\, c_p \partial T / \partial t,$$

for $0 \le x \le L$ and time $t \ge 0$. With steady state ($\partial T/\partial t = 0$) boundary conditions of:

1. T prescribed at 0 and L, or
2. T prescribed at one end and a heat source at the other, or
3. T prescribed at one end and a convection condition at the other, or
4. A convection condition at one end and a heat source at the other, or
5. A convection condition at both ends.

These thermal conditions, in 1D, are related to the displacements and stress in an axial bar as summarized in Table 12-1. (The structural version is: $\partial(A(x)\, E(x)\, \partial u / \partial x) / \partial x + Q(x)\, A(x) = \rho\, A(x)\, \partial^2 u / \partial t^2$, E = elastic modulus, or $\partial(E\, \partial u / \partial x) / \partial x + Q(x) = \rho\, \partial^2 u / \partial t^2$ for constant properties.)

Table 12-1 Terms of the 1D thermal-structural analogy

Thermal Analysis Item, [units], symbol	**Structural Analysis Item, [units], symbol**
Temperature [K], T	Displacements [m], u
Temperature Gradient [K/m], ∇ T	Strains [m/m], ε
Heat flux [W/m^2], q	Stresses [N/m^2], σ
Heat Source for point, line, surface, volume [W],[W/m],[W/m^2], [W/m^3], Q	Load for point, line, surface, volume [N],[N/m],[N/m^2],[N/m^3], Q
Restraint Prescribed temperature value [K], T	Restraint Prescribed displacement component [m], u
Reaction Heat flow resultant [W], Q	Reaction Force component [N], Q
Material Property Thermal conductivity [W/m-K], k	Material Property Elastic modulus [N/m^2], E
Material Law Fourier's law	Material Law Hooke's law

12.1.2 Three-dimensional formulation

In the 3D case the differential equation becomes the orthotropic Poisson Equation (see Chapter 16). That is, the above diffusion term (second derivatives in space) is expanded to include derivatives with respect to y and z, times their corresponding thermal conductivity values.

12.2 Thermal analysis input properties

The thermal material properties available in Cosmos are listed in Table 12-2 and Table 12-3. Only the conductivities are theoretically needed for a steady state study, but Cosmos always requests the mass density. Any transient (time dependent) thermal analysis involves the product of the mass density and specific heat, as seen in the above equation.

Table 12-2 Isotropic thermal properties

Symbol	Label	Item	Application
ρ	DENS	Mass density	Transient
k	KX	Thermal conductivity	Steady state and transient
c	C	Specific heat, at constant pressure	Transient

Table 12-3 Orthotropic thermal properties in principal material directions

Symbol	Label	Item
ρ	DENS	Mass density
k_x	KX	Thermal conductivity in material X direction
k_y	KY	Thermal conductivity in material Y direction
k_z	KZ	Thermal conductivity in material Z direction
c	C	Specific heat, at constant pressure

12.3 Finite Element Thermal Analysis

12.3.1 Thermal rod element

From the above analogy the matrix equations of a single element (from sections 2.3 and 2.4) is

$$k \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} \Delta_1 \\ \Delta_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix}$$

where $k \equiv k_x A / L$ may be referred to sa the thermal stiffness of the rod of length, L, area ,A, and thermal conductivity k_x. In this case, Δ corresponds to a nodal temperature, and *F* corresponds to the resultant nodal heat power from the various heat sources. The typical units of the above three matrices are W/C, C, and W.

12.3.2 Algebraic equations

The finite element method creates a set of algebraic equations by using an equivalent governing integral form that is integrated over a mesh that approximates the volume and surface of the body of interest. The mesh consists of elements connected to nodes. In a thermal analysis, there will be one simultaneous equation for each node. The unknown at each node is the temperature. Today, a typical thermal mesh involves 20,000 to 100,000 nodes and thus temperature equations. The restraints are specified temperatures (or a convection condition since it includes a specified fluid temperature). The reactions are is the resultant heat power necessary to maintain a specified temperature. All other conditions add load or source terms. The default surface condition is an insulated boundary, which results in a zero source (load) term.

The assembled matrix equations for thermal equilibrium have exactly the same partitioned form as the structural systems of section 2.5:

$$\begin{bmatrix} K_{uu} & K_{ug} \\ K_{gu} & K_{gg} \end{bmatrix} \begin{Bmatrix} \Delta_u \\ \Delta_g \end{Bmatrix} = \begin{Bmatrix} F_g \\ F_u \end{Bmatrix}$$

where now Δ_g represents the given (restrained) nodal temperatures, F_g represents the known resultant nodal heat power. This system of equations are solved just as described in section 2.5.

The thermal restraints items for steady state analysis are given in Table 12-4. Most programs offer only a temperature restraint. Cosmos also offers the ability to define a non-ideal material interface, as illustrated in

Figure 12-1. This is often needed in practice and is referred to as a contact resistance. It basically defines a temperature jump across an interface for a given heat flux through the interface. The necessary resistance input, R, depends on various factors. The R value is the same concept used is specifying home insulation.

Table 12-5 gives typical R values, while

Table 12-6 cites values of its reciprocal, the conductance.

Table 12-4 Restraints in steady state thermal analysis

Restraint Type	Geometric Entities	Required Input
Temperature	Vertexes, edges, faces and parts	Temperature value and units
Contact resistance	Two contacting faces	Total thermal resistance or unit thermal resistance. See discussion.

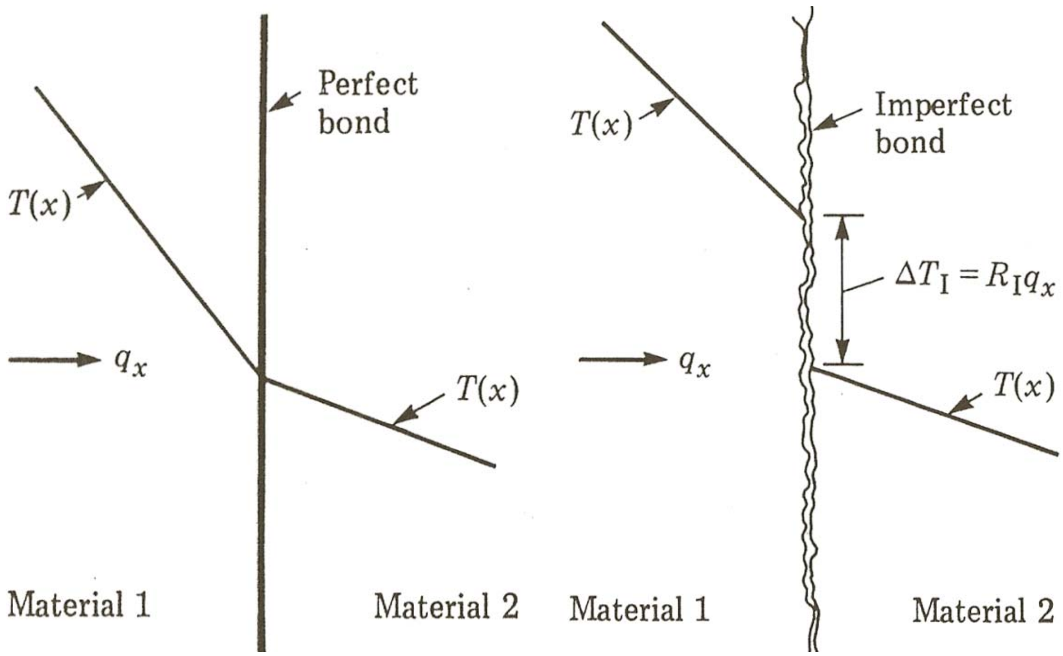


Figure 12-1 Ideal and thermal contact resistance interfaces

Table 12-5 Typical contact resistance values, R x e4, [m^2 K/W]

Contact Pressure	"Moderate"	100 kN/m2	10,000 kN/m2
Aluminum/aluminum/air	0.5	1.5-5.0	0.2-0.4
Copper/copper/air	0.1	1-10	0.1-0.5
Magnesium/magnesium/air		1.5-3.5	0.2-0.4
Stainless steel/stainless steel/ air	3	6-25	0.7-4.0

Table 12-6 Typical contact conductance values, C, [W/m^2 K]

Contacting Faces (pressure unknown)	Conductance
Aluminum / aluminum / air	2200 - 12000
Ceramic / ceramic / air	500 - 3000
Copper / copper / air	10,000 - 25,000
Iron / aluminum / air	45,000
Stainless steel / stainless steel / air	2000 - 3700
Stainless steel / stainless steel / vacuum	200 - 1100

The thermal load (source) items for steady state analysis are given in Table 12-7. Both convection and radiation require inputs of the estimated surface conditions. Typical convection coefficients are given in

Table 12-8. Note that there is a wide range in such data. Therefore, you will often find it necessary to run more that one study to determine the range of answers that can be developed in your thermal study. Having supplied all the restraints, loads, and material properties you can run a thermal analysis and continue on to post-processing and documenting the results.

Table 12-7 Loads for steady state thermal analysis

Load Type	Geometric Entities	Required Input
Convection	Faces	Film coefficient and bulk temperature in the desired units
Radiation	Faces	Surrounding temperature, emissivity values and units, and view factor for surface to ambient radiation
Heat Flux	Faces	Heat flux (heat power/unit area) value in the desired units
Insulated (Adiabatic)	Faces	None. This is the *default condition* for any face not subject to one of the three above conditions
Heat Power	Vertexes, edges, faces and parts	Total heat power value and units (rate of heat generation per unit volume times the part volume)

Table 12-8 Typical heat convection coefficient values, h, [W/m^2 K]

Fluid Medium	h
Air (natural convection)	5-25
Air / superheated steam (forced convection)	10-500
Oil (forced convection)	60-1800
Steam (condensing)	5000-120,000
Water (boiling)	2500-60,000
Water (forced convection)	300-6000

12.3.3 Post-processing

The temperature often depends only on geometry. The heat flux, and the thermal reaction, always depends on the material thermal conductivity. Therefore, it is always necessary to examine both the temperatures and heat flux to assure a correct solution. The heat flux is determined by the gradient (derivative) of the approximated temperatures. Therefore, it is less accurate than the temperatures. The user must make the mesh finer in regions where the heat flux vector is expected to rapidly change its value or direction. The heat flux should be plotted both as magnitude contours, and as vectors. The heat flux vectors should be parallel to insulated surfaces. They should be nearly perpendicular to surfaces with a specified constant temperature.

The temperatures should be plotted as discrete color bands or as contour lines. The temperature contours should be perpendicular to insulated boundaries. Near surfaces with specified temperatures, the contours should be nearly parallel to the surfaces. The exact temperature gradient is discontinuous at an interface between different materials because their thermal conductivities will be different. Pretty continuous color contours (the default) tend to prevent these important engineering checks. The temperature and temperature gradient vector can depend only on the geometry in some problems. However, the heat flux vector will always depend on the material properties. The items available for output after a thermal analysis run are given in Table 12-9.

In CosmosWorks it is possible to list, sum, average, and graph results along selected edges, lines, or curves (or surfaces, but that tends to be less helpful). Thus, you should plan ahead and add "split lines" to the mesh where you expect to find such graphs informative. Written results should not be given with more significant figures than the material input data. For heat transfer problems that is typically two or three significant figures.

Table 12-9 Thermal analysis output options

Symbol	Label	Item
T	TEMP	Temperature
$\partial T/\partial x$	GRADX	Temperature gradient in the selected reference X-direction
$\partial T/\partial y$	GRADY	Temperature gradient in the selected reference Y-direction
$\partial T/\partial z$	GRADZ	Temperature gradient in the selected reference Z-direction
$\lvert \nabla T \rvert$	GRADN	Resultant temperature gradient magnitude
q_x	HFLUXX	Heat flux in the X-direction of the selected reference geometry
q_y	HFLUXY	Heat flux in the X-direction of the selected reference geometry
q_z	HFLUXZ	Heat flux in the X-direction of the selected reference geometry
q	HFLUXN	Resultant heat flux magnitude

CosmosWorks also offers p-adaptive elements (p is for polynomial). Keeping the mesh unchanged, it can automatically run a series of cases where it uses complete second, third, fourth, and finally fifth order polynomial interpolations. It allows the user to specify the allowable amount of error. That is, it can solve a given problem quite accurately. However, you still must define the geometry, materials, load and restraint locations, and load and restraint values as well as interpret the results properly. You still have the age old problem of garbage-in garbage-out, so avoid computer aided stupidity.

12.3.4 Classical Solutions

There are a few well know thermal problems that have known simple solutions that give you some insight into the phenomenon and are easily verified with a CosmosWorks analysis. The first of these is a planar wall with a temperature difference on each side. This is often approximated as a semi-infinite wall, which reduces the problem to a one dimensional study. The solution [5] shows that the temperature through the wall is linear in space. Therefore, the heat flux, per unit area, will be constant. Any finite element model should give the exact result everywhere [2].

12.4 Planar wall

The heat transfer through a wall will be illustrated by a CosmosWorks model. It could be solved with a single layer of elements through the wall. Here it is assumed that the analytic solution is not known, so several thousand unknowns are used to clearly illustrate the response. The wall in this case is five inches thick and made of alloy steel. A unit cross-sectional area is used. The outer (left) side is kept at 100 F while the inner side is at 0 F. Those two restraints must be explicitly applied. The other four faces of the body are planes of symmetry and are automatically treated as insulated. The mesh is shown along with the resulting linear temperature drop distribution. The linear temperature change with position is clearly seen in Figure 12-2.

Note that at a position 40 % through the wall the temperature difference has dropped 40 % to 60 F. This result will be compared to a cylindrical wall later. The heat flux should be constant. Constant values do not contour well so the contour bounds must be set to give a reasonable plot. The flux values at the inlet and outlet faces are selected and listed in tables shown in Figure 12-3. It shows that each square inch of the outer wall requires about 0.0134 BTU/s of power to maintain the outer temperature. For a planar wall made up of constant thickness layers of different materials the heat flux must still remain constant, but the temperature difference will occur as linear changes from one interface to the next. The linear distribution of temperature is more easily seen with a graph along one edge of the mesh.

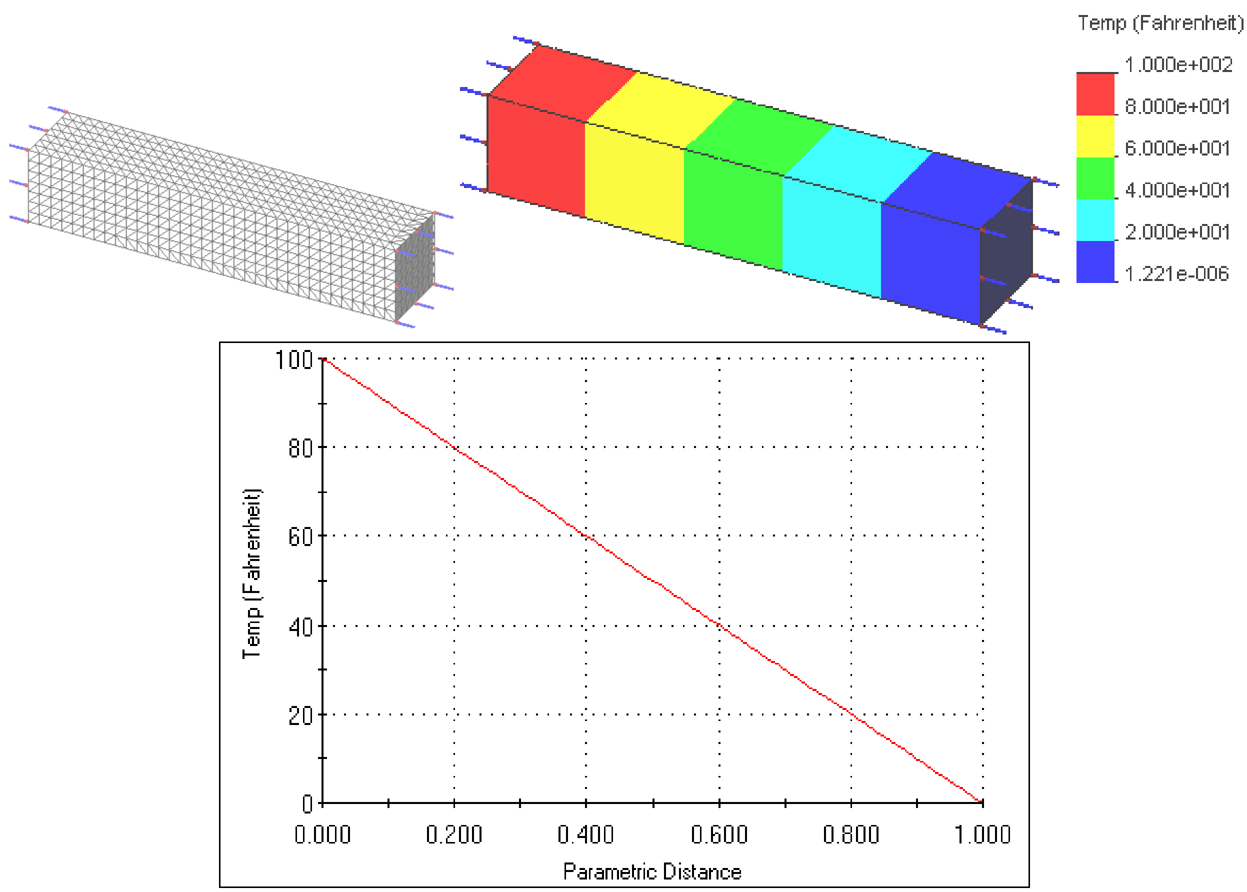
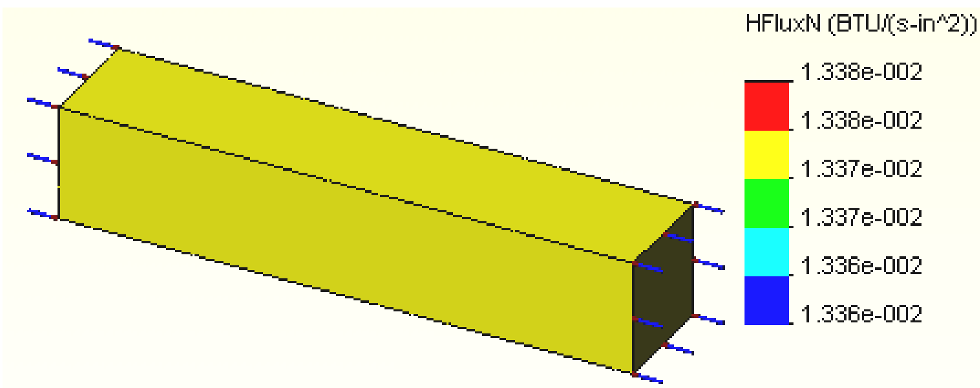


Figure 12-2 Temperatures of a homogeneous wall

12.5 Cylindrical walls or pipes

Another well known heat transfer problem with a simple analytic solution is that of radial conduction through an infinite pipe, or curved wall. In that case, the temperature difference varies in a logarithmic manner through the wall thickness. That means that the heat flux must also vary through the wall, since it passes through more material as the radius increases.

The example here [4], will be for an alloy steel pipe with an inner radius of 10 inches and with a thickness of 5 inches. Thus it is very similar to the previous example having inner and outer temperatures of 100 F and 0 F, respectively. In this case, each of those restraints are applied to cylindrical faces. The other four faces are insulated and do not require specific action. The geometry, a very fine mesh, the resulting temperature contours, and the radial variation of the temperature are given in Figure 12-4. The contour plot there might appear to again be linear, but the graph of the temperature along a radial edge is actually logarithmic. Compared to Figure 12-2, you see that at a distance of 40 % through the wall the temperature has dropped more than 40 % to about 56.4 F. The non-constant nature of the corresponding heat flux is seen in the contour plot and in the radial edge heat flux graph of Figure 12-5.



Summary

	Value	Units
Total Heat	0.013375	BTU/s
Avg	0.013375	BTU/(s-in^2
Max	0.013375	BTU/(s-in^2
Min	0.013375	BTU/(s-in^2
RMS	0.013375	BTU/(s-in^2

Summary

	Value	Units
Total Heat	-0.013375	BTU/s
Avg	0.013375	BTU/(s-in^2
Max	0.013375	BTU/(s-in^2
Min	0.013375	BTU/(s-in^2
RMS	0.013375	BTU/(s-in^2

Figure 12-3 Constant heat flux through a wall

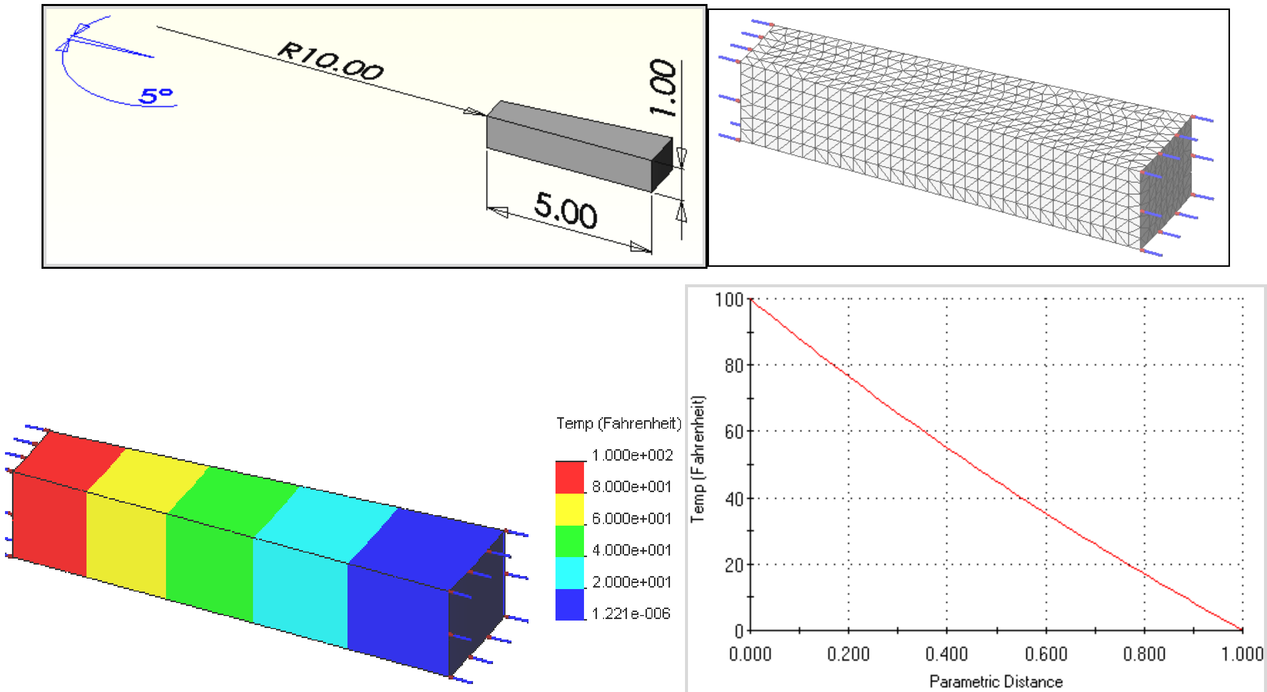


Figure 12-4 Radial temperature through a cylindrical wall

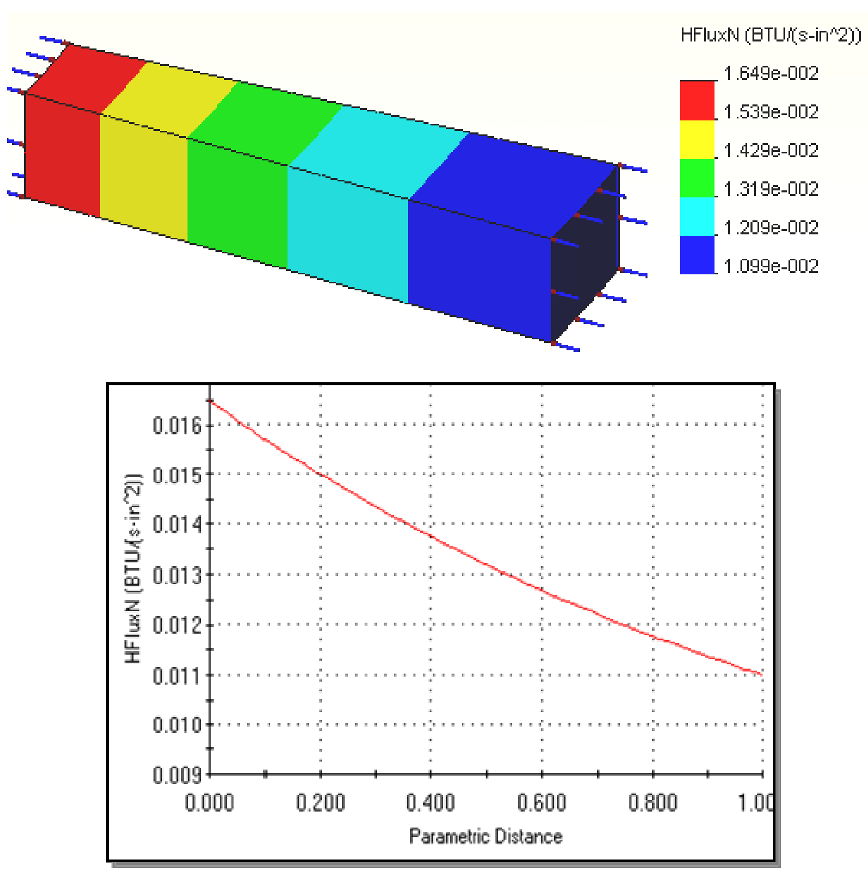


Figure 12-5 Contours and graph of radial heat flux in a cylindrical wall

12.6 Heat transfer with an orthotropic material

12.6.1 Introduction

It is becoming more common to encounter materials which have properties that are directionally dependent (anisotropic). A common case is that of orthotropic materials that have their properties completely defined in terms of three perpendicular directions. Those three principal material directions are usually defined by a user defined coordinate system or a user defined reference plane. Cosmos employs the reference plane approach. The input reference system provides the data necessary to compute the direction cosines between the material directions and the global x-y-z-axes. That defines a coordinate transformation matrix, say $\mathbf{T}$, that converts the principal properties, say $\mathbf{K}_{123}$, to the corresponding global properties as $\mathbf{K}_{xyz} = \mathbf{T}^T \mathbf{K}_{123} \mathbf{T}$. For the common isotropic case this reduces to $\mathbf{K}_{xyz} = k\,\mathbf{I}$, where $\mathbf{I}$ is the identity matrix.

It can be confusing to input orthotropic properties into commercial software so it is wise to begin with a problem with a known solution. There are few such problems but [4] presents the exact solution for temperatures in an orthotropic rectangular block with a constant internal heat generation rate, Q. The block is 2 by 1 m by 0.1 m thick and its outer edge faces are held at a constant temperature of 0 C. The thermal conductivity in the long direction is 2 W/(m K) while that short direction is 1.2337 W/(mK). It is assumed that no heat transfer takes place through the thickness. Cosmos expects the total heat power to be input as HP = Q V = 50 W, where V is the volume of the block.

12.6.2 Material reference directions

Since the geometry, boundary conditions, heat generation source and material properties all have double symmetry you can utilize a quarter symmetry part. For a single part Cosmos expects the material directions to be defined by a reference plane associated with the part. For multiple parts in an assembly the reference plane for each different orthotropic material must be defined within the assembly. Here you could simply select the Front plane to define the material directions. However, for arbitrary material directions you need to define a particular reference plane, so that approach will be used. Typically such planes are defined by a preplanned split line and a point on the surface. Here you can use one edge of the block as a line to start defining a plane:

1. Select **Insert→Reference geometry→Plane** to open the **Plane** panel.
2. Pick **Edge<1>**, to serve as material direction 1, or X as seen on the left in Figure 12-6.
3. Pick the back **Vertex<1>** to set the plane and the material 2, or Y. The third material direction is defined by the right hand rule from the first two material directions.
4. Pick **Through Lines/Points**, click **OK**.

The constructed plane is automatically named Plane1 in the Feature Menu. Use a slow double click on that name to change it to Plane1_ortho for later reference in the Materials panel. Note that the material directions differ from the global model directions. You will input the material K_x and K_z relative to this plane.

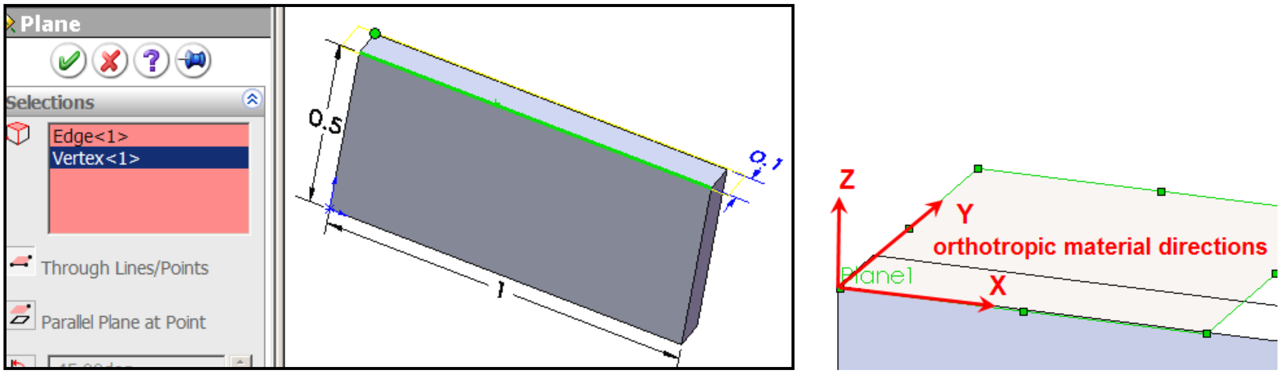
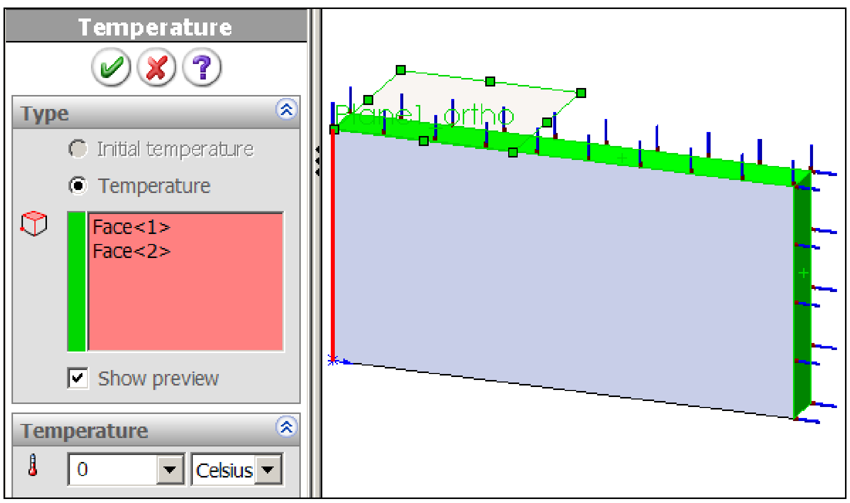


Figure 12-6 Rectangular block and material reference plane

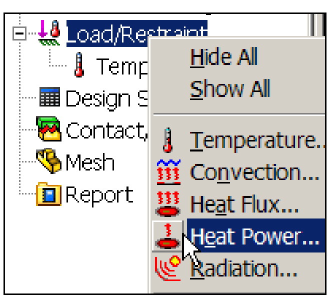
12.6.3 Cosmos analysis

Start a 2D thermal analysis by clicking on the Cosmos icon:

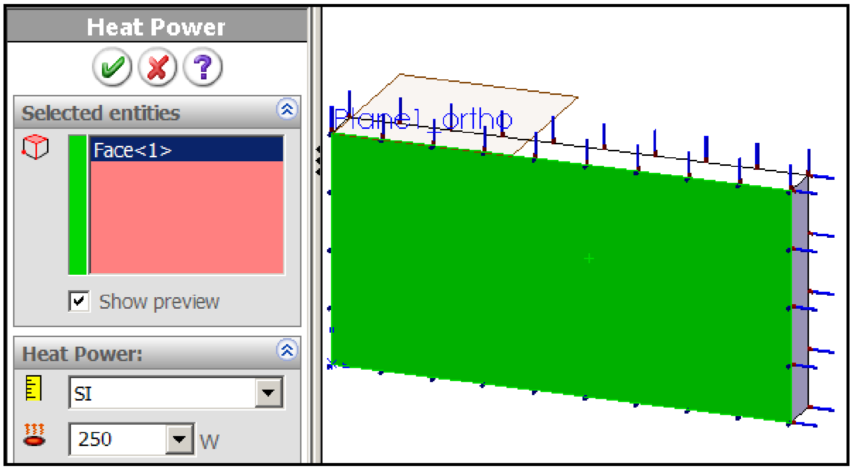
1. Right click on the part name and select **Study**. Pick **Thermal** and **Mid-surface shell** analysis.
2. Change the study name to Carslaw_orthotropic, click **OK**.
3. Right click on **Load/Restraint→Temperature**, pick the two outer edges and set the value to 0 C.



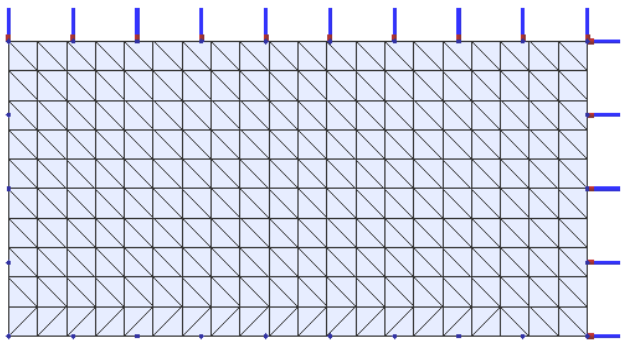
4. Right click on **Load/Restraint→Heat Power**, pick the front face.



5. Enter 50 W in the **Heat Power panel**, click **OK**.

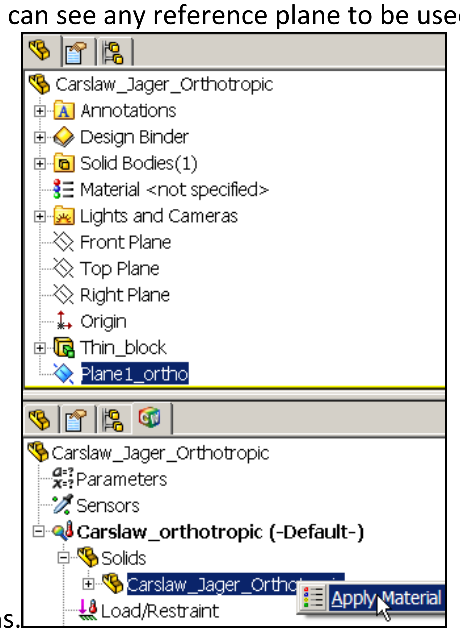


6. Right click on **Mesh→Create** and accept the default element size.



Since no boundary restraints have been applied to the two symmetry planes they default to the insulated (no normal heat flux) condition. Next it is necessary to input the orthotropic material data. To do that you must be able to see both the CosmosWorks and SolidWorks menus at the same time:

1. Set the mouse on the topmost horizontal bar above the Cosmos menu and pull down the SolidWorks until you can see any reference plane to be used in defining the material directions.



2. Click on the name of the **orthotropic part** in the Cosmos Menu.

3. Click on the name of the **reference plane** in the SolidWorks menu.
4. Right click on the **orthotropic part→Apply Material** to open the **Material panel**.
5. There set the **Model Type** to **Linear Elastic Orthotropic**.
6. Verify that the **Reference geometry** is displayed as **Plane1_ortho** activated above.
7. Enter **KX** = 2, **KY** = 0, and **KZ** = 1.2337 W/(m K). Set other properties to unity or zero.

Material Properties

Model Type: Linear Elastic Orthotropic

Reference geometry: Plane1_ortho

Units: SI

Category: custom made

Name: User Defined

Description: Carslaw test material

Property	Description	Value	Units	Temp Dependency
SIGXT	Tensile strength	1	N/m^2	Constant
SIGXC	Compressive strength	1	N/m^2	Constant
SIGYLD	Yield strength	1	N/m^2	Constant
ALPX	Thermal expansion coef in x	1	/Kelvin	Constant
ALPY	Thermal expansion coef in y	1	/Kelvin	Constant
ALPZ	Thermal expansion coef in z	1	/Kelvin	Constant
KX	Thermal conductivity in x	2	W/(m.K)	Constant
KY	Thermal conductivity in y	0	W/(m.K)	Constant
KZ	Thermal conductivity in z	1.2337	W/(m.K)	Constant

8. Click **OK**, and verify that the proper reference geometry is active in response to the system message.

Please verify that you selected a proper reference geometry for defining the orthotropic material directions. Refer to the online help for details.

Those properties will be used to computer the temperature and heat flux distributions. Then two different material sets will be created in a similar manner. The first will reverse the two conductivities and the third will use the average of the two. That is, the third example is an isotropic material that has been input as if it were orthotropic, as shown in Figure 12-7.

12.6.4 Analysis results and validation

Carslaw and Jaeger [4] list the exact temperatures at the center point as 83.72, 60.13, and 70.31 C, respectively for the three different orthotropic examples. The three temperature contours and center point probe values from Cosmos agree within less that 1% difference. That should assure you that you now know how to properly input orthotropic material properties. The three sets of temperature distributions are displayed in Figure 12-8. Note that the same contour ranges have been used in those temperature plots. The three maximum

KX	Thermal conductivity i	1.2337	W/(m.K)
KY	Thermal conductivity i	0	W/(m.K)
KZ	Thermal conductivity i	2.0	W/(m.K)

KX	Thermal conductivity i	1.6114	W/(m.K)
KY	Thermal conductivity i	0	W/(m.K)
KZ	Thermal conductivity i	1.6114	W/(m.K)

Figure 12-7 Second and third orthotropic thermal conductivity versions

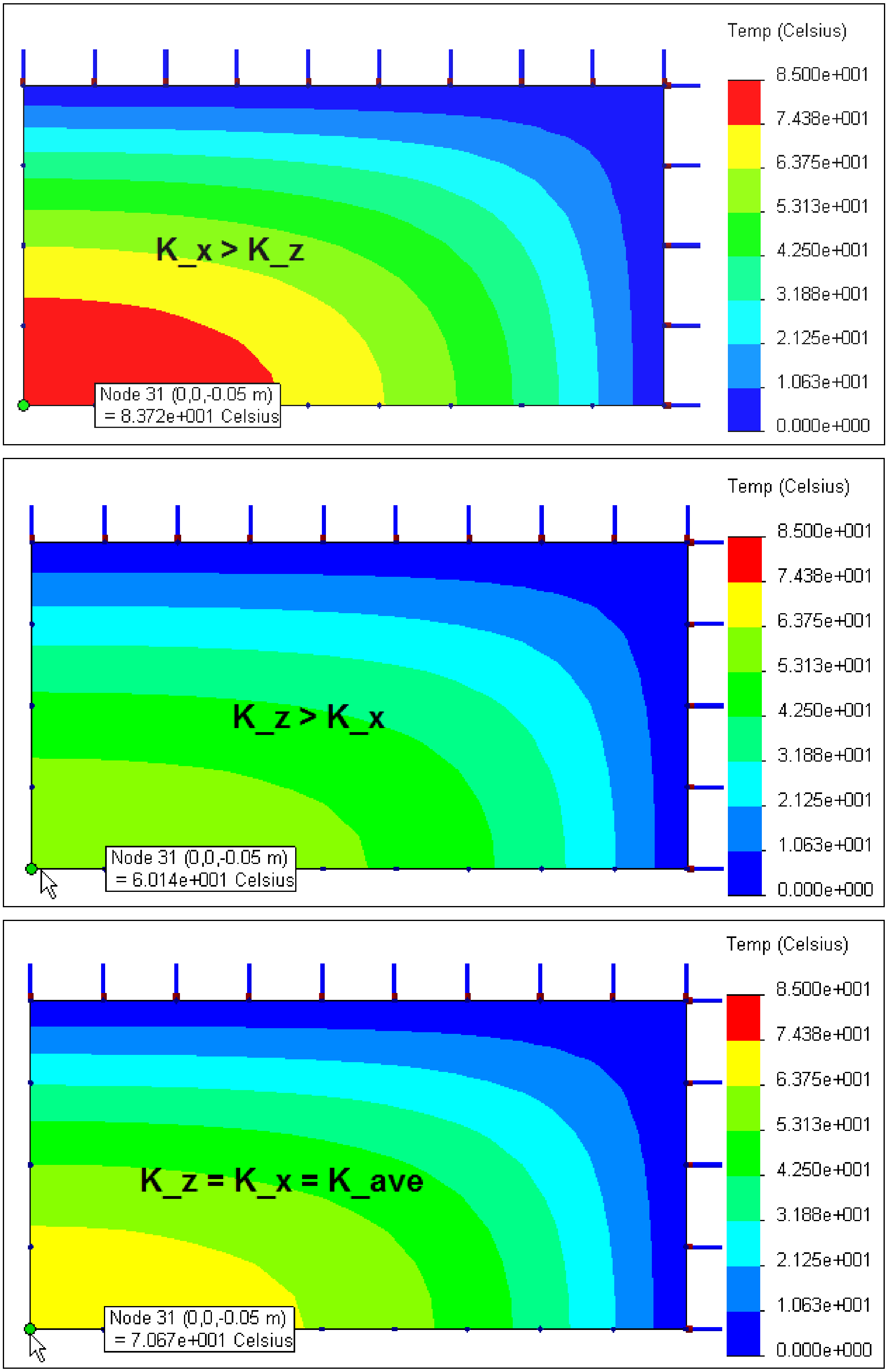


Figure 12-8 Effect of orthotropic conductivities on temperature

(center point) temperatures differ by as much as 60%. The spatial distribution of the associated heat flux values is noticeably different, as is illustrated in Figure 12-9. Their peak values differ only by about 10%. If you also plotted the heat flux vectors you would see that they are parallel to the two insulated sides, as they should be.

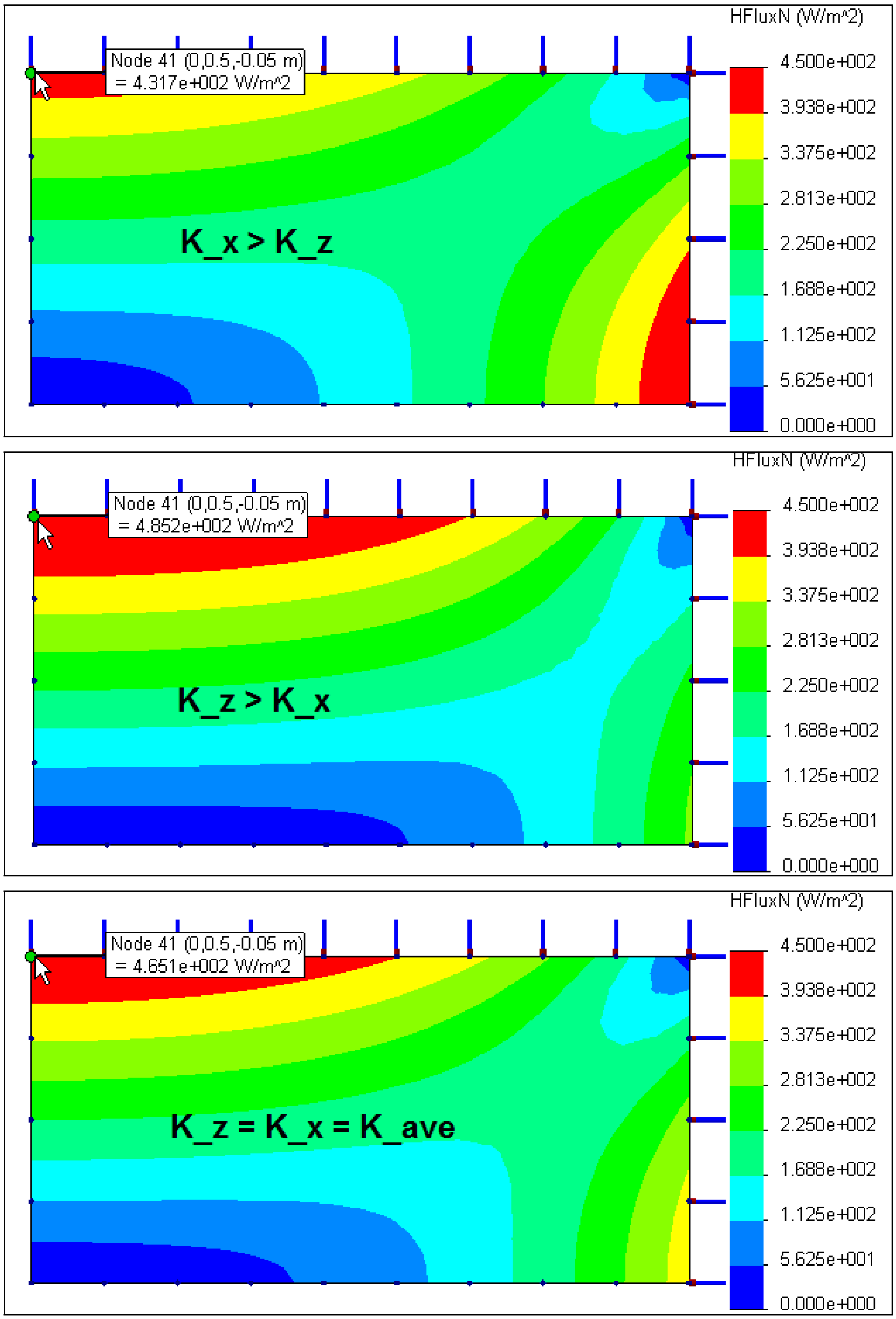


Figure 12-9 Effect of orthotropic conductivities on heat flux

12.7 Shell thermal model

The last example could have also been solved by using the CosmosWorks mid-surface shell element, which has one temperature unknown per mesh node. When the 5 degree solid segment of the cylinder (top) is meshed as a mid-surface shell (in the circumferential direction) the mesh is placed in the middle of a plane of constant thickness. Here the mesh is generated in a constant axial (z) plane. Clearly, it has only a few percent as many equations as the solid mesh above. The two temperature restraints are applied to the two circular arc edges. The two straight edges and the shell face(s) are insulated. The temperature results agree very closely with the much more expensive solid computations. That is easily seen by examining the temperature results given in Figure 12-10. Likewise, the heat flux contours and radial graph values in Figure 12-11 are also in close agreement with the solid model (and the analytic solution).

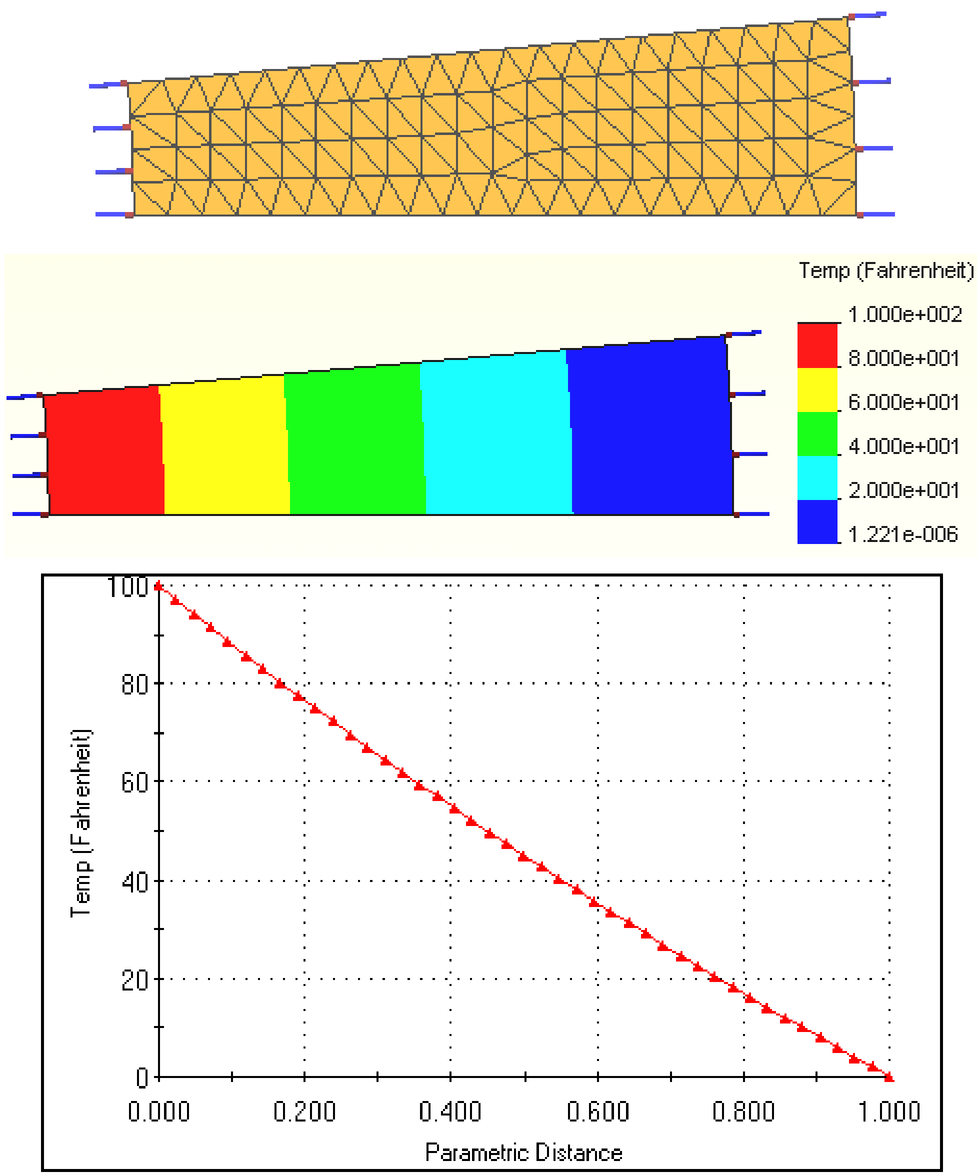


Figure 12-10 Pipe segment temperatures from mid-surface shell mesh

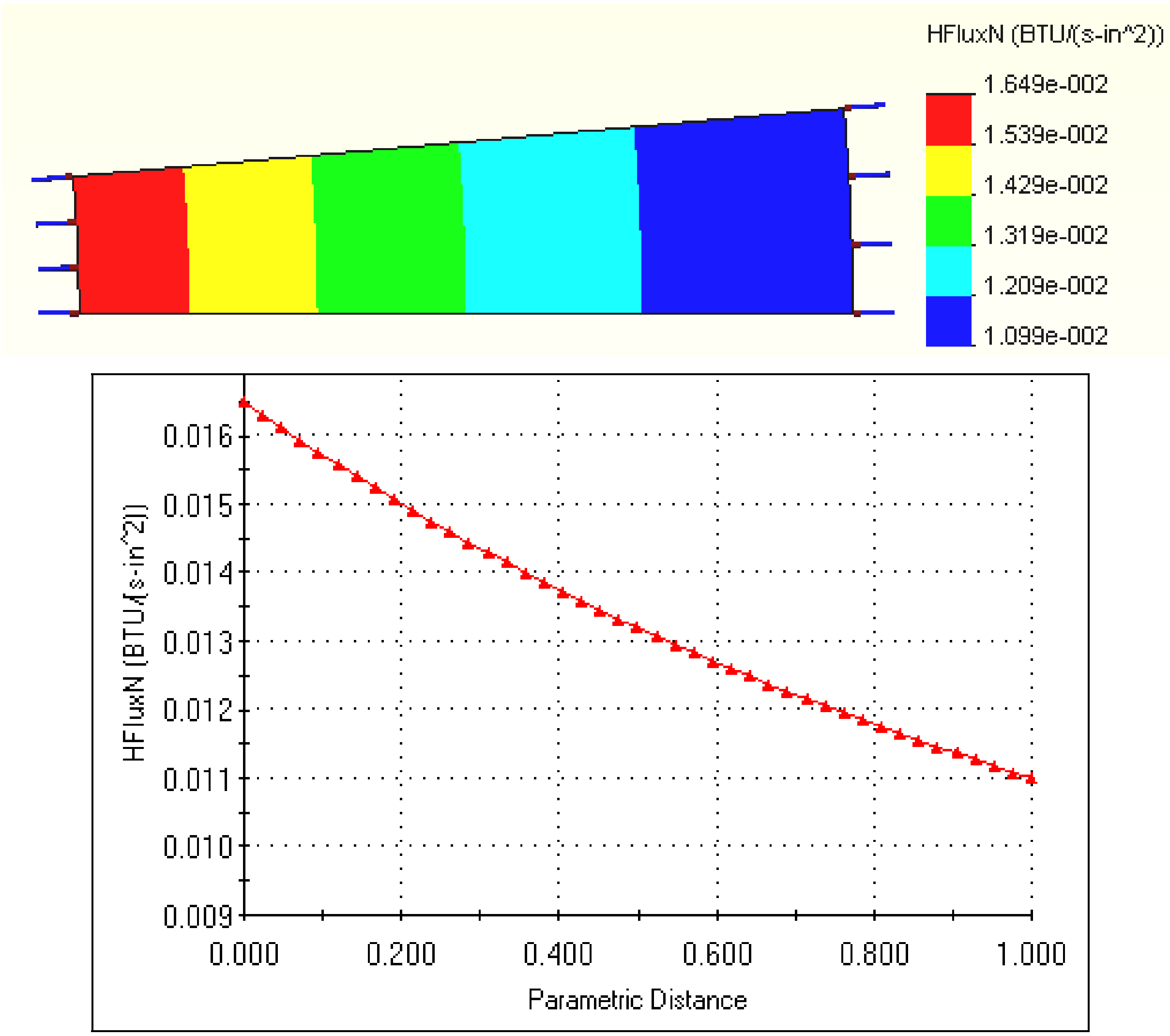


Figure 12-11 Mid-surface shell heat flux result for the pipe

12.8 Conducting rod with convection

Most conduction problems also involve free convection. That usually gives a steeper change in temperature over a region. Here a segment of a circular rod (Figure 12-12) is examined where the length is only two times the diameter. That is near the lower limit where you might want to expect a one-dimensional approximation to be accurate. Convection occurs on the outer surface while one end is kept at 100 F. The other three symmetry surfaces in the model are insulated. Any wedge angle could have been used, but a value of 30 degrees was picked to give good element aspect ratios.

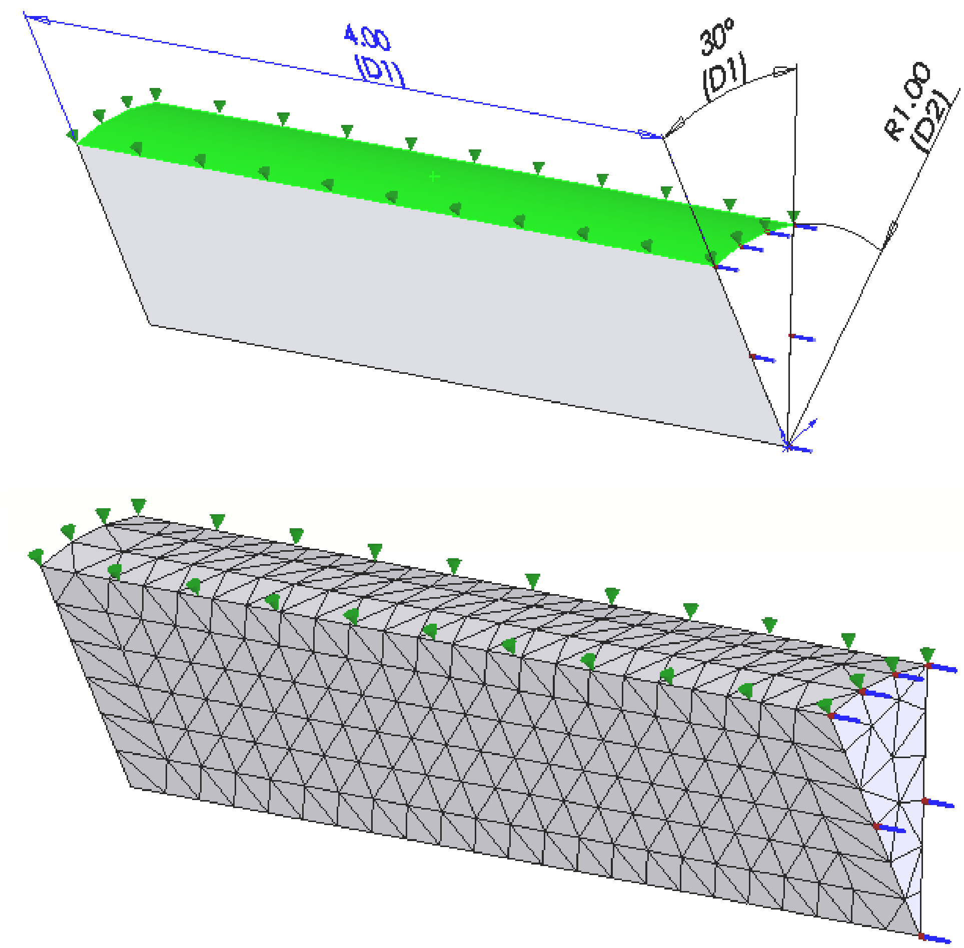
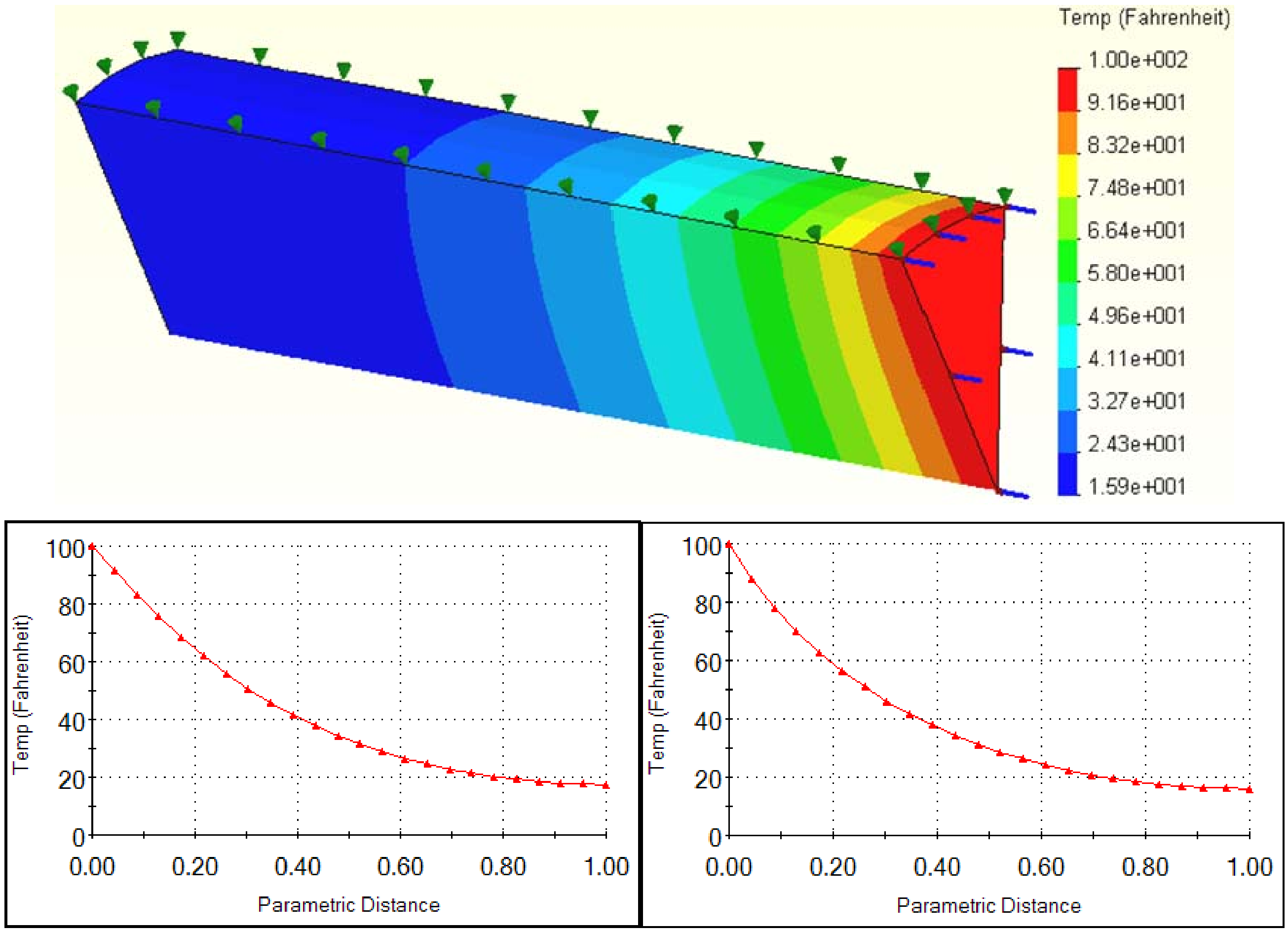


Figure 12-12 Circular rod segment with an end temperature and convection

Myers [8] gives the one-dimensional solution for a rod conducting heat along its interior and convection that heat away at its surface. The temperature is shown to change with axial position, *x*, as a hyperbolic cosine of *mx*, where $m^2 = h\,P\,L\,/\,k\,A\,L$, is a ratio of convection strength to conduction strength. It involves the surface convection coefficient, *h*, the perimeter, *P*, of the conducting area, *A*, over the length *L*, and the material thermal conductivity, *k*. Typical temperature distributions, for a low value of *m* are seen in Figure 12-13.

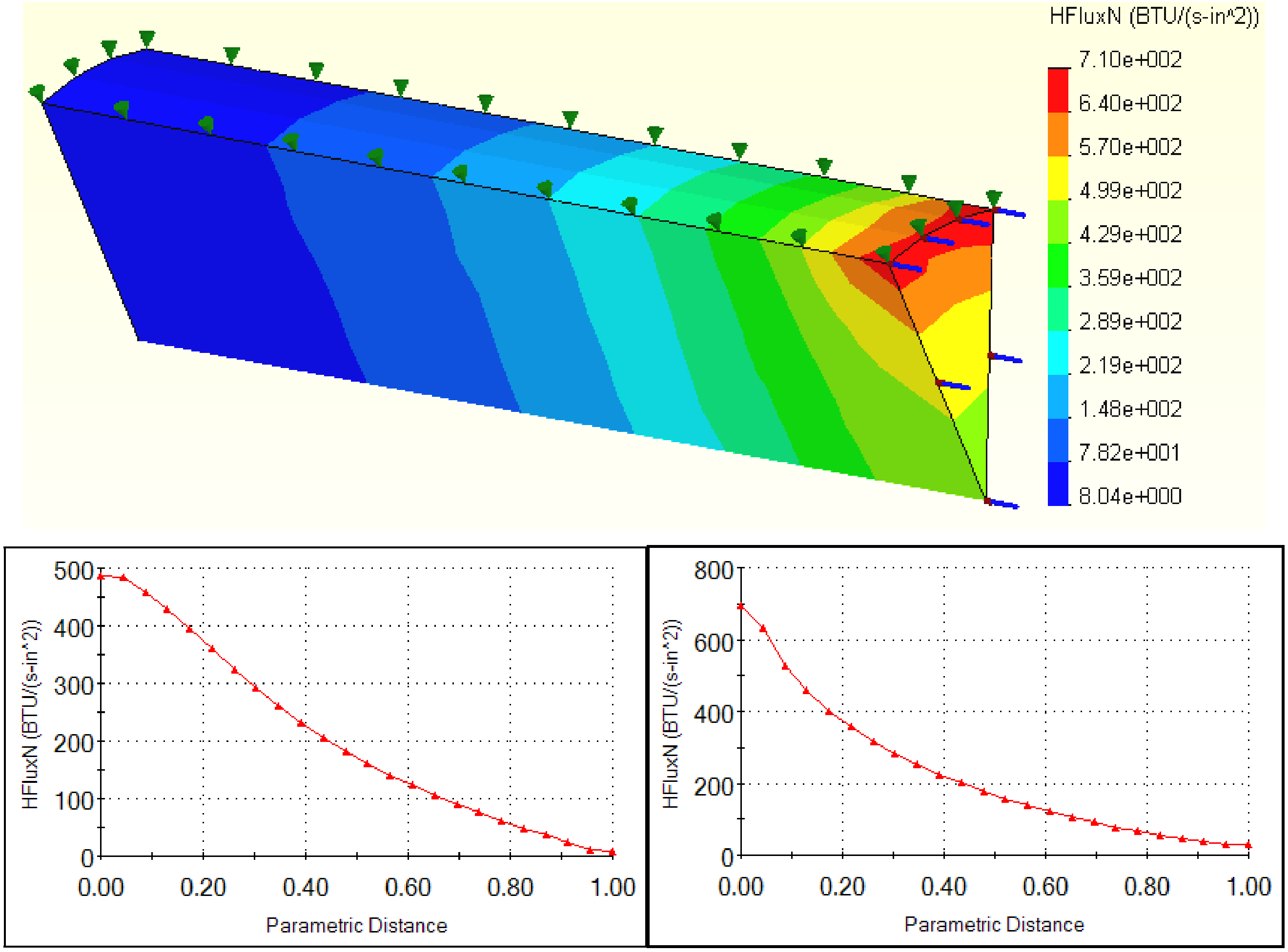


Centerline Surface

Figure 12-13 Rod temperature distributions, for a small *m* value

The surrounding free convection air is assumed to be at 0 F. Comparing the centerline and surface graphs of the temperature there is very little difference and they both follow the one-dimensional approximation given by Myers. Notice that the far end plane temperature does not match that of the surrounding air. A similar comparison of the heat flux magnitude is given in Figure 12-14. That figure shows a much larger difference between the centerline and surface heat flux. But the average of the two graphs is still quite close to the analytic approximation given by Myers.

It is not uncommon for the user supplied convection coefficient to be in error due to measurement errors or errors occurring in a units conversion. As an example, the above study was re-run with the convection coefficient increased by a factor of 10. That is, the convection heat transfer mode was increased relative to the conduction mode (*m* was approximately tripled). The new temperatures, in Figure 12-15, are significantly different from those of Figure 12-13. The surface and centerline temperature graphs are still about the same and still follow the hyperbolic cosine change given by Myers. However, the temperatures in the distal half of the bar have dropped to rapidly approach, or match the temperature of the surrounding air.



Centerline Surface

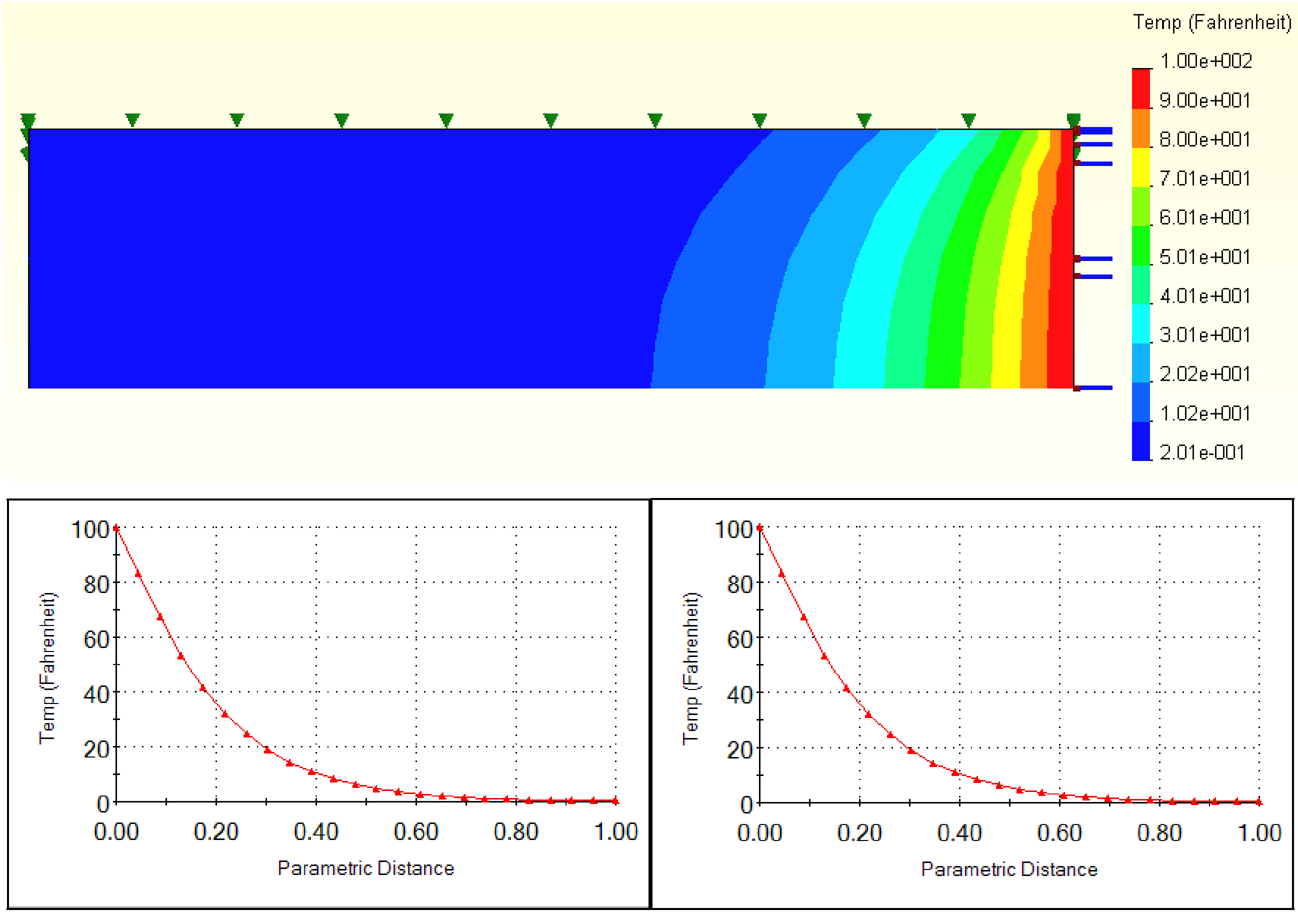
Figure 12-14 Typical heat flux magnitude results, for a small *m*

12.8.1 Limiting values of the convection coefficient

The convection coefficient has lower and upper bounds, of 0 and ∞, respectively. They have different physical effects in a study. A low value of *h* causes the surface to approach an insulated state, while a high value causes the surface to approach a restraint of a specified temperature. The latter state is what is seen in Figure 12-15. The distal end of the part is responding as if it had a restraint temperature of 0 F applied to it.

These two limits on *h* are also reflected in terms of the temperature contour lines. The lower limit causes the contour lines to approach being perpendicular to the surface as they do for insulated boundaries. Likewise, the upper limit causes the temperature contours to approach being parallel to the surface as they would if it was subjected to a constant temperature restraint.

These examples illustrate the value of using analytic approximations to estimate and validate the results from a finite element study. The first example also shows that if an analytic solution is not available for validating a solid study sometimes an independent two-dimensional finite element study can be useful. You always should estimate the expected results before you start a study and to validate the study results when finished.



Centerline Surface

Figure 12-15 Temperatures when *h* is increased by a factor of 10

12.9 Thermal analysis of a plate with a circular hole

12.9.1 Introduction

A vertical square steel plate is 30.48 cm on each side, has a 1.27 cm radius center hole, and is 1.00 cm thick. The steel is measured to have a thermal conductivity of 25.95 W/m C. The center hole surface is at a temperature of 398.9 C and the left and right faces of the square plate have natural convection to air at 21.26 C and a convection coefficient of 408.9 W/m^2 C. The top and bottom faces of the plate are insulated. Prepare a report on the temperature and heat flux distributions, note the maximum temperature on the convection surface, and estimate the heat input necessary to maintain the center hole temperature

The problem geometry and material have one-eighth symmetry, but the restraints and thermal load have only one-fourth symmetry. Thus the best we can do from an efficiency point of view is to model one of the 90 degree segments (Figure 12-16). Note that by "cutting" the part with two symmetry planes, you will have to assign proper boundary conditions on those two planes to account for the removed material.

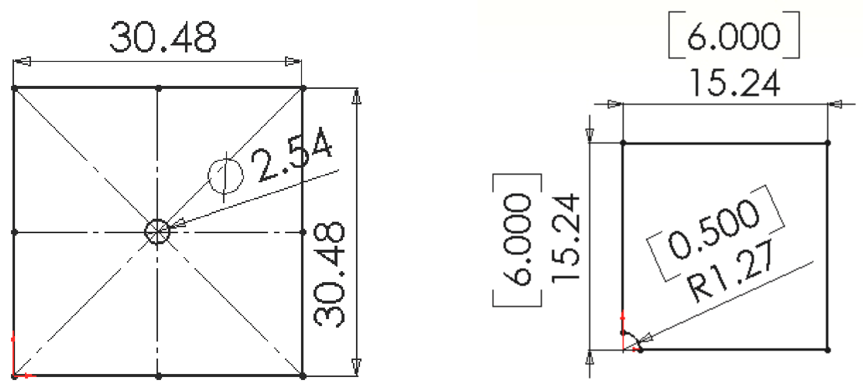


Figure 12-16 Identifying two planes of symmetry

12.9.2 Estimated solution

Before beginning the following finite element analysis you should estimate the temperature results and/or attempted to bound them. For a plane wall with a known inside temperature on one side and convection on the other the exact temperature solution is linear through the wall. The 1D analytic solution for that estimation gives the temperature of the convection surface as

$$T_s = (h_{air} T_{air} + T_{wall} k/L) / (k/L + h_{air}),$$

where L is the thickness of the wall. The temperature along a line of symmetry can often be modeled with a 1D model that has the same end conditions as the symmetry line. Here those end conditions are the same and mainly their lengths vary. The lower length is L_0 = 0.1524 m, and the vertical and top line combine into L_1 = 0.2921 m. Therefore, the estimated outside wall temperatures here are

$$T_0 = \frac{(408.8\ \mathrm{W/m^\wedge 2\ C} * 21.26\ \mathrm{C} + 398.9\ \mathrm{C} * 25.95\ \mathrm{W/m\ C} / 0.1524\ \mathrm{m})}{(25.95\ \mathrm{W/m\ C} / 0.1524\ \mathrm{m} + 408.8\ \mathrm{W/m^\wedge 2\ C})} = 132.2\ C,$$

and likewise the top end point is estimated to have a lower value of T_1 = .88.5 C. These two estimates mean we expect the temperature on the convection surface to decrease from bottom to top points. You can also estimate the heat flow through a 1D wall there (assuming parallel heat flux vectors) as $q_0 = K (T_{wall} - T_{air}) / L_0$, which gives estimates of q_0 = 18,900 W/m^2, and q_1 =.9,865 W/m^2 at the same lower and upper points.

You should anticipate some visual results that should appear in the post-processing. The temperature contours should be parallel to each surface with a given constant temperature (the central hole), and they should be perpendicular to any insulated surface (the top and bottom faces) and any symmetry plane. Neither case should occur at a convection boundary, except for the two special extreme cases of h = 0 so $T_s = T_{wall}$ and h = ∞ (or h >> k/L) which gives $T_s = h_{air}$. Those two special conditions can exist, but they usually occur because of user data errors. Finally, the temperature and heat flux contours should be smooth. Wiggles in a contour usually mean that the mesh is too crude there. If wiggles occur in an important region the mesh should be refined there and the analysis repeated. You can also visualize some of the heat flux vector results. First, they should be parallel to any insulated surface (or symmetry condition). They will change rapidly in magnitude and direction around the point of a re-entrant corner (or re-entrant edge in 3D).

12.9.3 Thermal study and material

To change from SolidWorks to CosmosWorks click on the *CosmosWorks icon*. In the **CosmosWorks Manager panel:**

1. Right click on the **Part name** and select **Study**
2. In the **Study panel** set the **Study name,** static **Analysis type**, and mid-surface shell **Mesh type**.
3. Under the part name right click **Mid-surface shell→Apply Material to all**. A review of the **Material panel** standard materials yields no match. Thus, turn on **Custom defined, in** Figure 12-17**,** and type in 25.95 W/m C for **Thermal conductivity,** click **OK.**

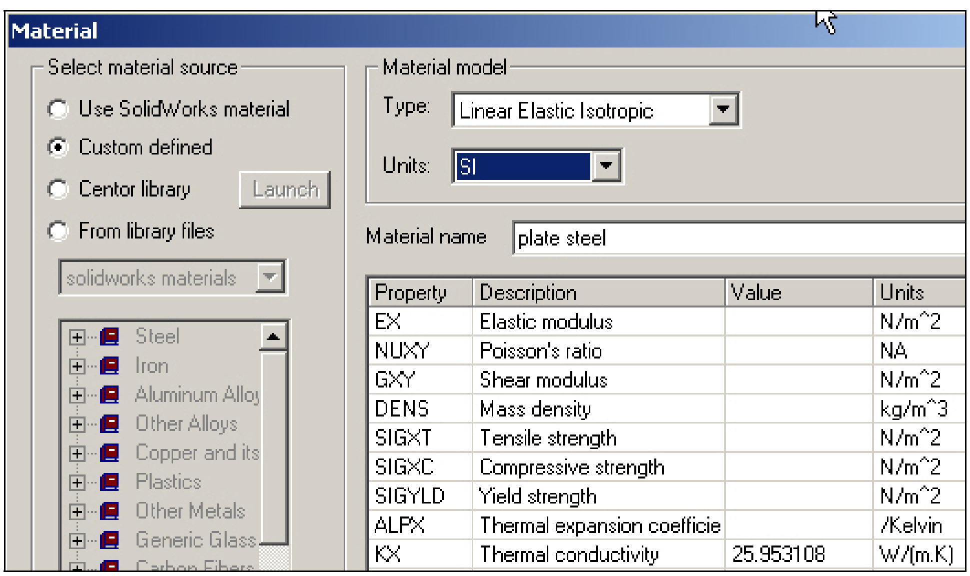
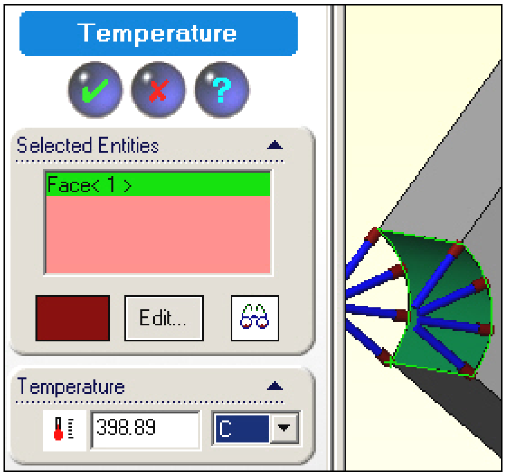


Figure 12-17 Define and apply a custom material property

12.9.4 Temperature restraint

Apply the only "*essential boundary condition*" (the known temperature) here. In the CosmosWorks **Manager menu**:

1. Use **Load/Restraint→ Temperature** to open the **Temperature panel**.
2. Set the **Temperature** to 398.9 C and pick the cylindrical face as the **Selected Entity**, in that panel, click **OK**.

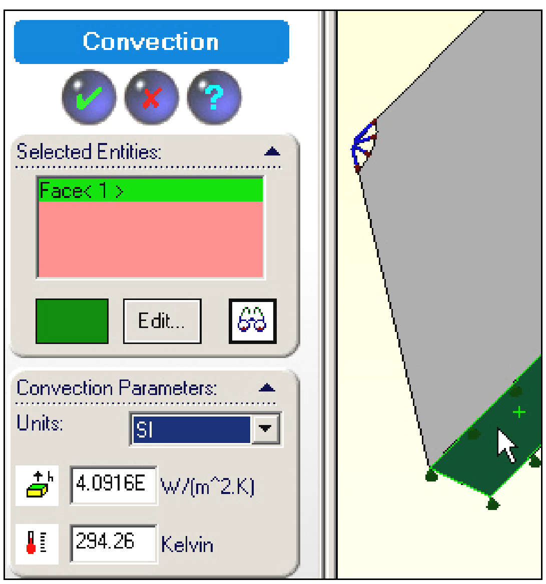


Later, you will want to know how much heat must flow into the system to maintain that temperature. That is the thermal reaction.

12.9.5 Convection load

Invoke the right side free convection as the only loading condition. In the **Manager menu**:

1. Use **Load/Restraint→Convection** to open the **Convection panel.**
2. Pick the flat convection face as the **Selected Entity**
3. In **Convection Parameters** set the convection coefficient, **h** = 408.8 W/(m^2 C)
4. Set the air **temperature** to 294.26 K (about 70 F), and note that Kelvin is the only allowed input unit.



5. Select **Preview** (eyeglasses icon), click **OK.**

Remember that when you create a mid-surface shell mesh it will be assigned the above extrusion thickness, and will use it to compute the effective surface area that is subjected to the air convection.

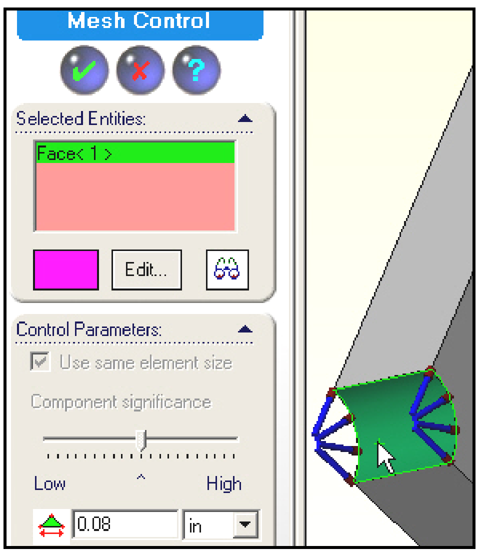
12.9.6 Insulated surfaces

The insulated surfaces, which correspond to the top plane and the symmetry planes, require no action. That is because in any finite element thermal analysis that state (of zero heat flux) is a "*natural boundary condition*". That is, it occurs automatically unless you actively prescribe a different condition on a boundary. This also means that the front and back of the extruded part (i.e., the "top and bottom" of your shell) are automatically insulated.

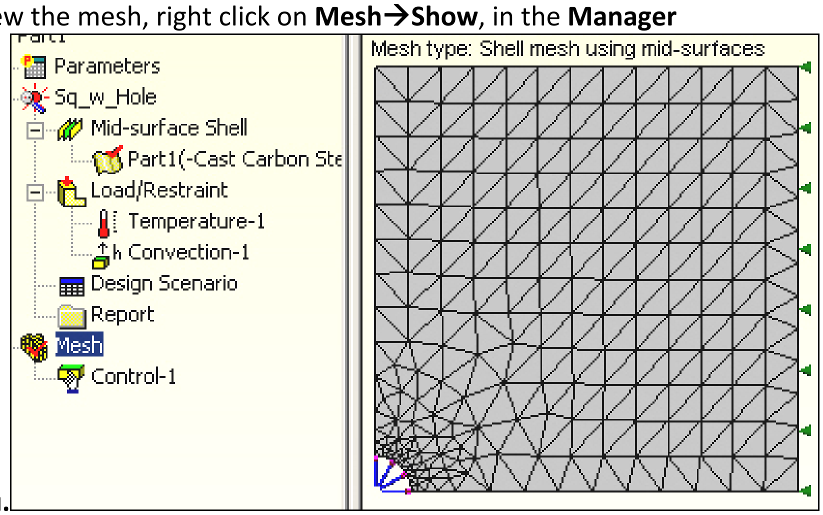
12.9.7 Mesh generation

The central hole is so small that you should expect to have high temperature gradients there and plan ahead to assure smaller elements there. In the **Manager menu**:

1. Use a right click **Mesh→Apply control.** In **Mesh Control** select the cylindrical surface as the **Selected entity**.
2. Observe the default element size and reduce its value in the **Control Parameter** to 0.08 inch, click **OK**.



3. In the **Manager menu** right click **Mesh→Create.**
4. In the **Mesh panel** click **OK** for mesh generation.
5. To view the mesh, right click on **Mesh→Show**, in the **Manager menu.**



12.9.8 Temperatures computation

Start the temperature solution with right click on the **Name→Run**. 

Usually you get a solution completed message. Sometimes the fast iterative solver might fail. That causes a "non-existent" data message when you try to review the results. If that happens you need to change to the sparse direct solver (under **Tools→Options**) which is slower, but more robust.

12.9.9 Post-processing

12.9.9.1 Temperature results

Begin the results review with a temperature plot. In the **Manager menu** under **Thermal:**

1. Double click on **Temperature→Plot 1**. The default contour plot of temperatures would appear as a smoothed (Gouraud) color image. Usually a stepped shaded image gives a better hint of a bad mesh.
2. To create one right click in the graphics window, **Edit Definition→Thermal Plot→Display.**
3. Set **Units** to Celsius and change **Fringe type** to discrete filled, click **OK**.
4. Right click in the graphics window, select **Color map.**
5. In the **Color Map** panel pick 8 **colors** of thin **Width** and 2 **Decimal** places, click **OK**. Such a typical temperature plot is seen in Figure 12-18, on the left side.

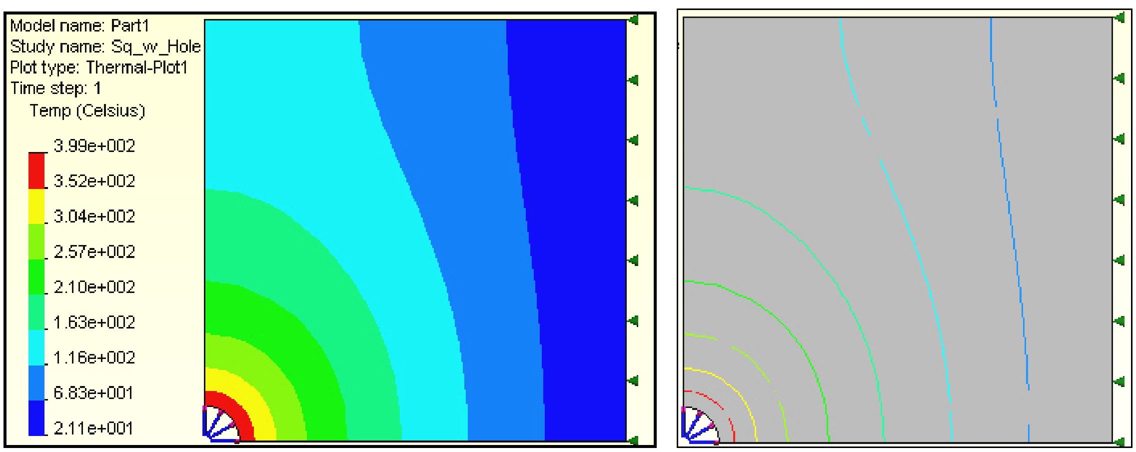


Figure 12-18 Temperature level contours

For another type of view generate a temperature line contour:

1. Right click in the graphics window, **Edit Definition→Thermal Plot→Display.**
2. Change **Fringe type** to **Line**, click **OK**, for the results in the right side of Figure 12-18.

Either form of the temperature contours display the visual features that were discussed above at the end of the Estimated Solution section. Also, you do not see wiggles in the temperature contour lines, so the mesh is not obviously poor. It is also possible to display and list the temperatures at any selected nodes. This is called probing the results. To do that you usually want to display the mesh (with or without the contours) and employ the probe command:

1. Right click in the graphics window, **Edit Definition→Thermal Plot**
2. Change the **display** from model to mesh, click **OK.**

3. Right click in the graphics window, pick **Probe** and use the cursor to pick the nodes whose temperatures are to be displayed and listed. The **Probe panel** lists the node, its temperature, and location coordinates as each one is picked.

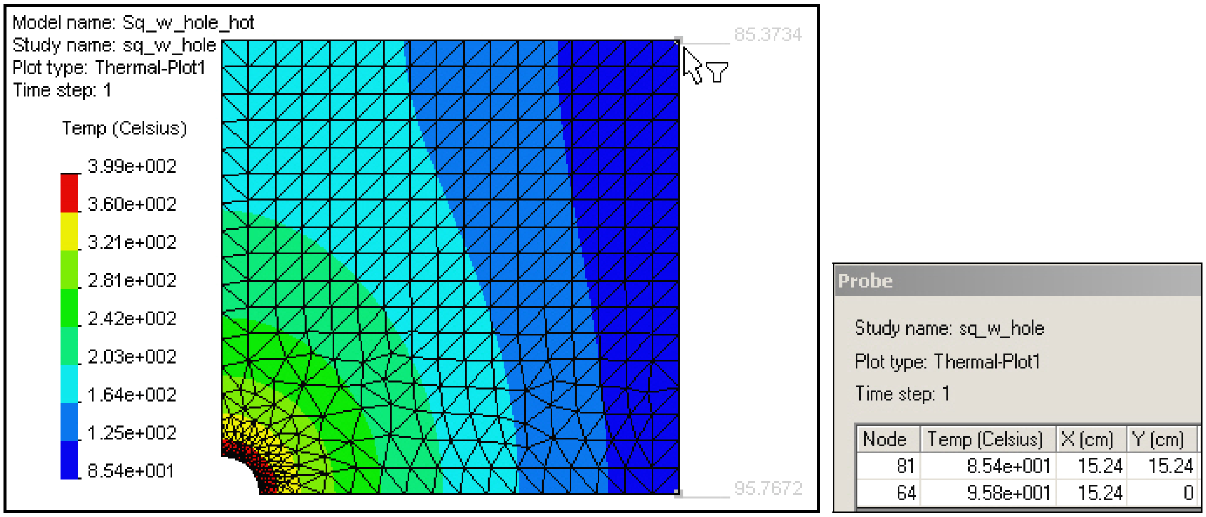
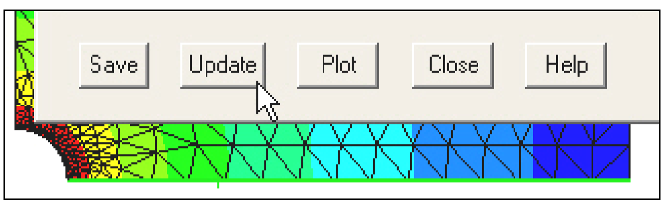


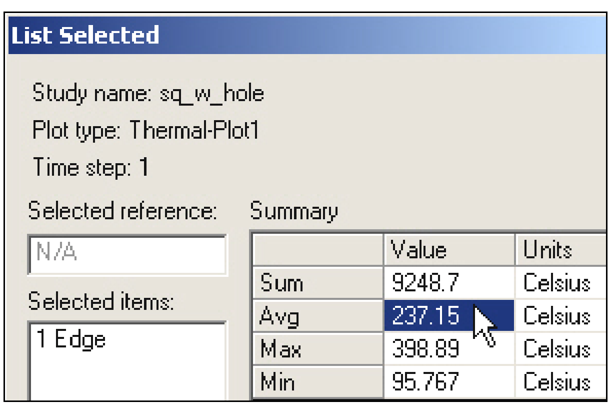
Figure 12-19 Probing the mesh for selected nodal temperatures (light gray)

You can also obtain graphs of selected results along the boundary of the part. To obtain a temperature graph:

1. Right click in the graphics area of a temperature plot, pick **List Selected…**
2. Pick the desired edge (lower straight green symmetry line) as the **Selected items**. Click on **Update**.



3. The summary of the temperature values along that path appear in the list at each node on that path. The **Avg, Max**, and **Min** temperatures appear in the **Value** column. In this case those values were 237, 399 and 96 C, respectively.



4. To see a graph of the temperature along that edge select **Plot**. That graph indicates a bad mesh if the graph is not smooth. Here it seems smooth, but it has a very sharp gradient at one end.

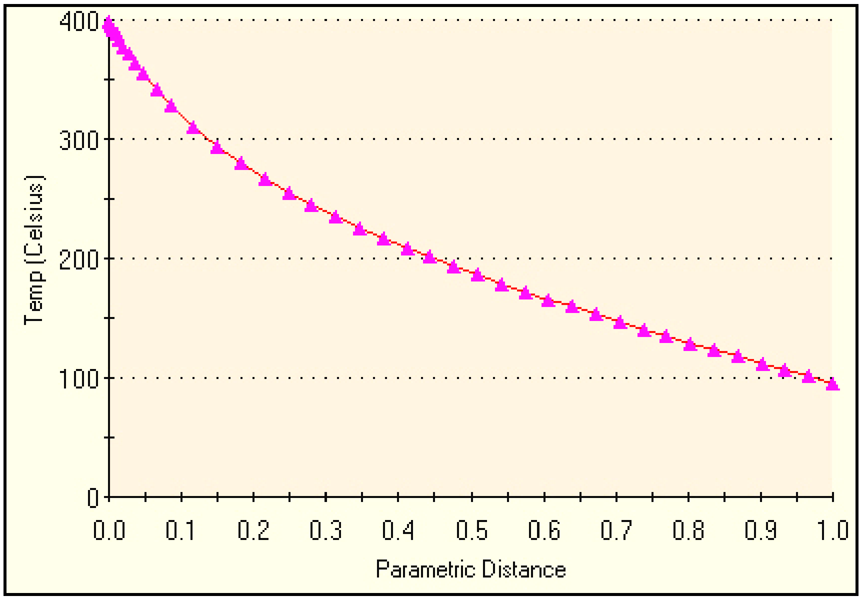


Figure 12-20 Graph of temperatures along lower symmetry line

Remember that the 1D approximation would give you a straight line between the max and min temperatures, so the actual temperature graph gives you some feel for how much to trust such a 1D estimate. Had you carried out this study with a 3D mesh you would still get the nodal summary data if you picked a surface before choosing Update, but you would not get the graph since the surface nodes occur in random order and are more difficult to represent in a parametric form (parametric surface) for arbitrary geometry.

The temperature distribution on the free convection surface is usually of specific interest. It also can some times be compared to known solution chosen to try to estimate a correct result, as is done here with a 1D estimate. Therefore it is desirable to supplement the above probe operation with an edge temperature summary and a graph. That is accomplished by repeating the last set of operations, but selecting the insulated edge as the selected item. The average temperature there is seen to be 90.4 C, in Figure 12-21. The corresponding temperature graph is seen in Figure 12-22, of a non-dimensional distance.

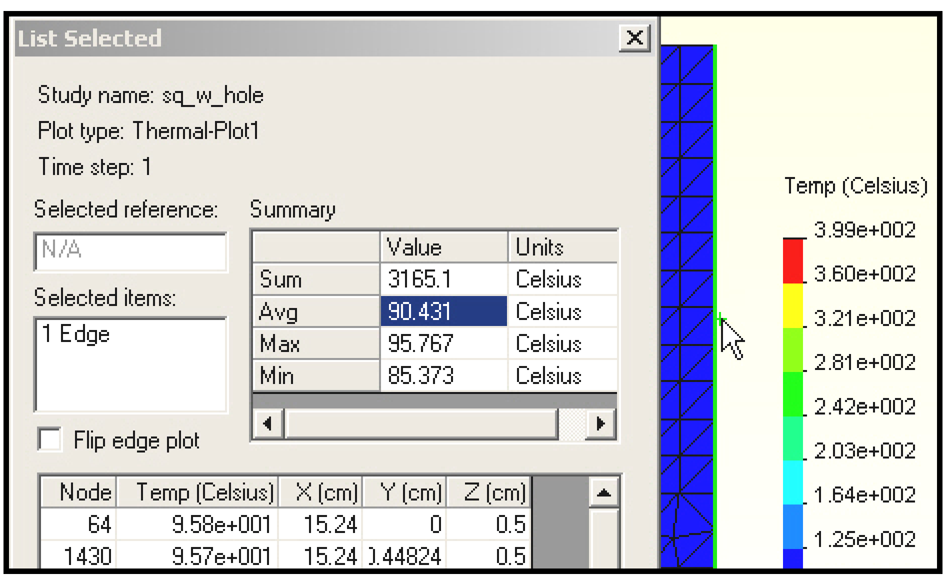


Figure 12-21 Nodal temperature summary for the convection edge

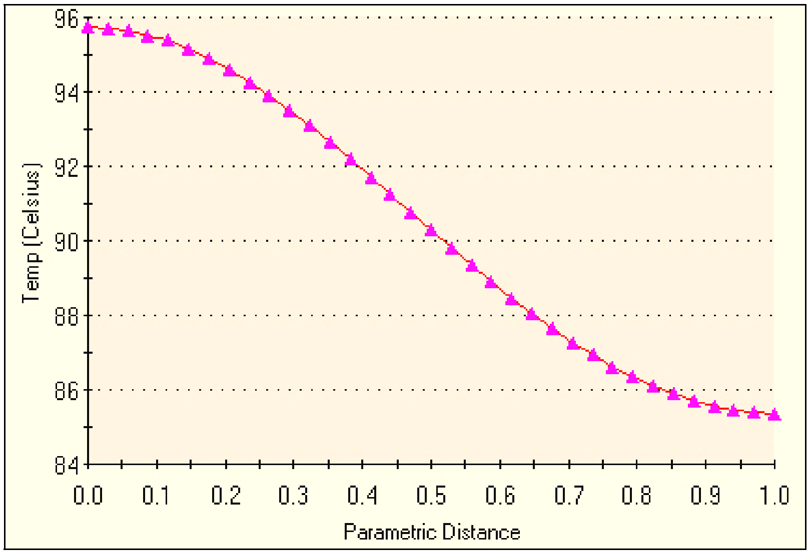


Figure 12-22 Graph of convection surface temperatures (y-increasing)

12.9.9.2 Temperature validation check

The 1D hand calculated T_0 temperature range along the convection surface ranged from 132 C down to 88 C. That agrees reasonably well with the computed range of 96 to 85 C for the actual 2D shape.

12.9.9.3 Heat flux

The heat flux is a vector quantity defined by Fourier's Law: **q** = -K ∇ T. Thus, it is best displayed as a vector plot:

1. Right click in the graphics window, **Edit Definition→Thermal Plot →Display.** Set **Units** to W/m^2.
2. Pick **Component** resultant heat flux.
3. **Plot type** vector. **Fringe type** line, click **OK.**
4. Dynamically control the plot with a right click in the graphics window, select **Vector Plot Options**
5. Vary the vector **Size** and **Density** (of the percent of elements displayed). The heat flux vector plot is shown on the left in Figure 12-23 along with the heat flux magnitude on the right.

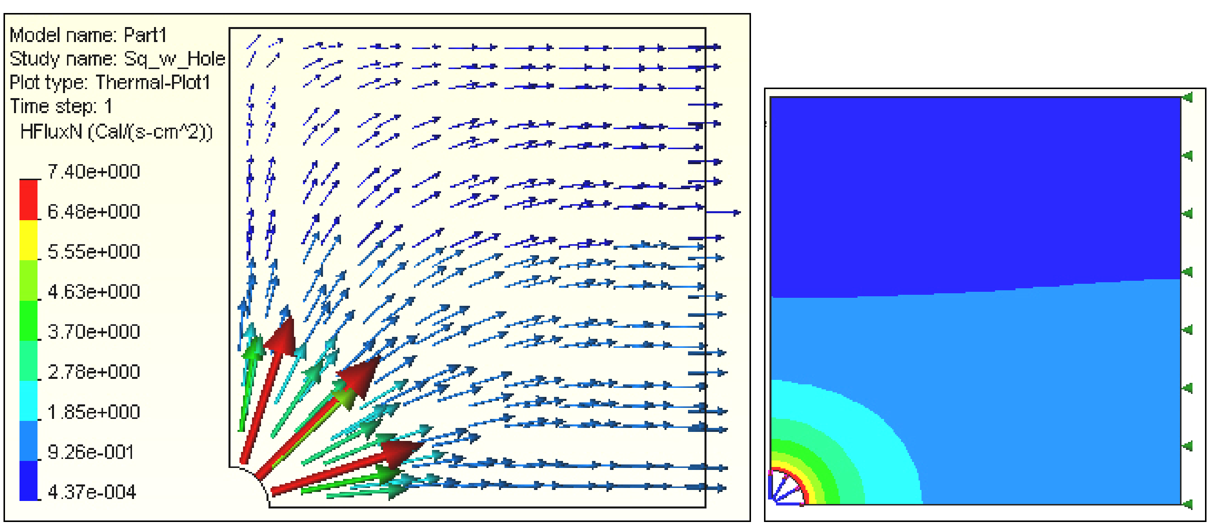
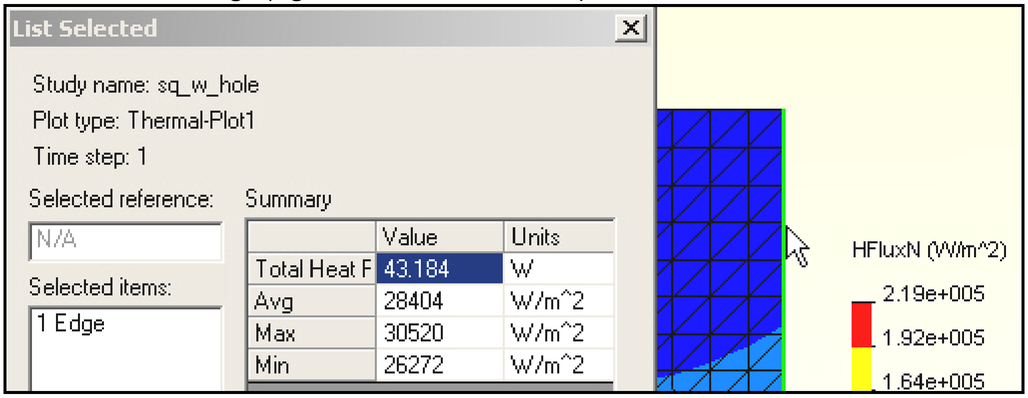


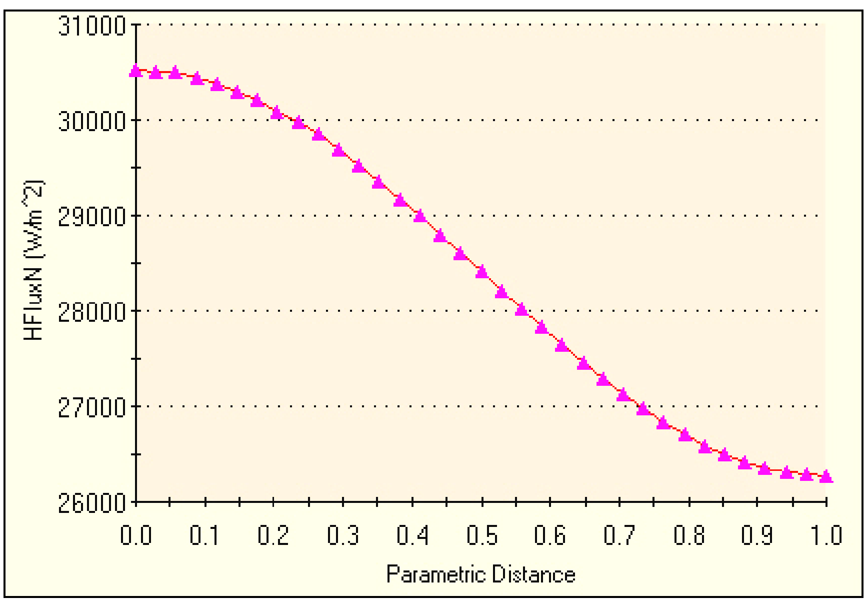
Figure 12-23 The heat flux (element centroid) vectors and (nodal) magnitudes

Like with the temperatures, one can obtain summary results for the heat flux on a surface or edge. To obtain a heat flux summary and graph for the convection edge:

1. Right click in the graphics area of a heat flux plot, pick **List Selected…**
2. Pick the desired edge (right convection line here) as the **Selected items**.



3. Click on **Update**. The summary of the normal heat flux values along that path appear in the list at each node on that path. The **Avg, Max**, and **Min** heat flux magnitudes appear in the **Value** column. In this case, the average heat flux magnitude was about 28,400 W/m^2.
4. Note that the **Value** column also contains the **Total Heat Flow** as 43.2 W (positive out, negative in) across the selected surface. It is the integral of the normal heat fluxes over the surface area selected.
5. To see a graph of the normal heat fluxes along that edge select **Plot**. That graph is smooth and indicates a good mesh in that region.



The above heat flow out of this system (given above) should be equal and opposite to the heat flow coming in at the circular hole (since there were no internal heat generation rate data). If they do not reasonably agree then the mesh should be revised. Such differences occur since they are calculated from the gradients of an approximate temperature solution. The temperatures are always more accurate that the heat flux, but you need to have acceptable accuracy for both. Repeating the above procedure for that arc gives the total heat flow (Figure 12-24) as -43.1 W, an acceptable difference of about 2%.

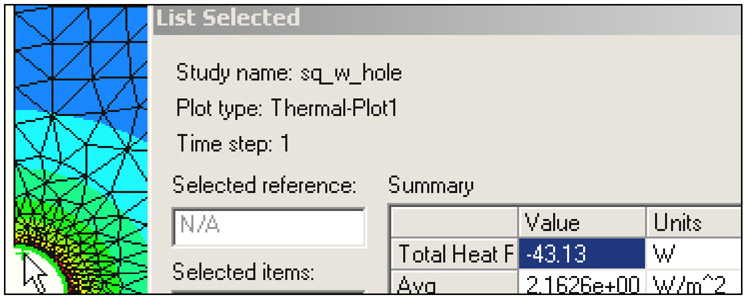


Figure 12-24 Checking the heat flow balance at the temperature restraint

Likewise, you can plot the heat flux crossing the circular arc (which is the required thermal reaction necessary to maintain the specified temperature). That graph is given in Figure 12-25. It is reasonably smooth but does show some wiggles. Since this is a region of high temperature gradients a slightly finer mesh should be considered around the center hole.

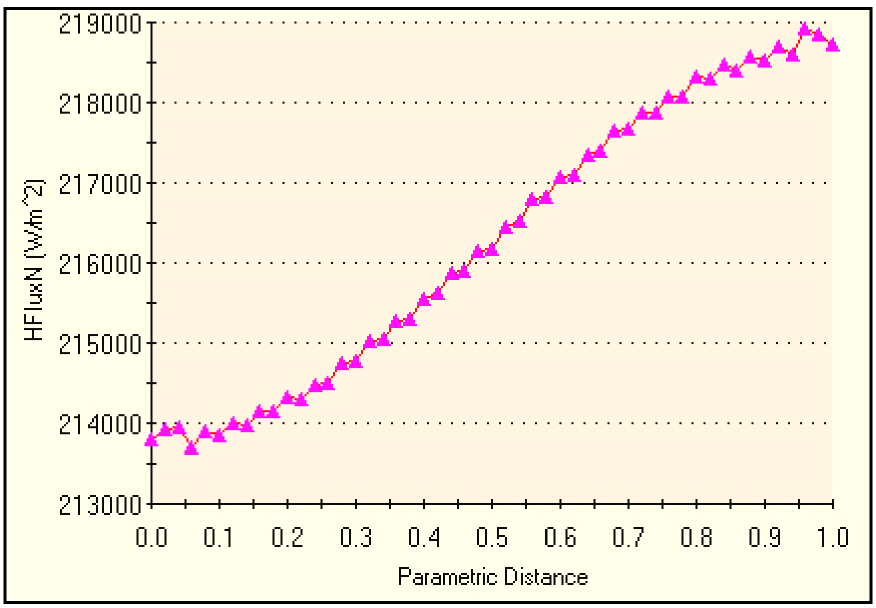


Figure 12-25 Graph of the normal heat flux into the system

12.9.9.4 Heat flow reaction validation

A heat flow in or out of the system must occur at every specified temperature node and at any convection nodes. If your finite element system provides those data it is good practice to review them. CosmosWorks does allow you to recover those data, as shown above, but if it did not basic engineering would give you an estimate of the total heat flow (per unit length assumed here for the thickness into the page). Such a validation check is important since it is not unusual for a user to enter incorrect values for the K and h values. The ratio of those values is important in convection calculations.

From the vector plot of heat flux in Figure 12-23 you see that at both the inner cylindrical surface and the outer insulated surface that the flow is basically normal to the surface. Integrating the normal heat flux passing through either surface gives the total heat flow lost.

To estimate the heat loss, manually change the color bar to more clearly give the range along the outer surface:

1. Right click, **Edit Definition→Thermal Plot→Settings**
2. Set the **Display legend** to defined
3. Assign a **minimum** value of 0, and **maximum** of 5,000 W/m^2, respectively (arrived at by trial and error), for the results in Figure 12-26.

Estimate the length of the outer surface associated with each solid color heat flux range. Multiply that distance by the thickness (1 cm) to get the heat outflow surface area, and multiply by the average contour value (for that color) for its heat loss, in Watts. The boundary length is 0.1524 m. The corresponding heat flux values (by averaging the color value ranges) is 2.81e4 W/m^2, respectively. The total model heat loss, per unit length, is estimated at Q =0.1524 m (2.81e4 W/m^2) 0.01 m, or Q = 42.8 Watts. Your computer model was only 1/4 of the total domain. Therefore, the true heat loss is about Q_{total} = 171 Watts.

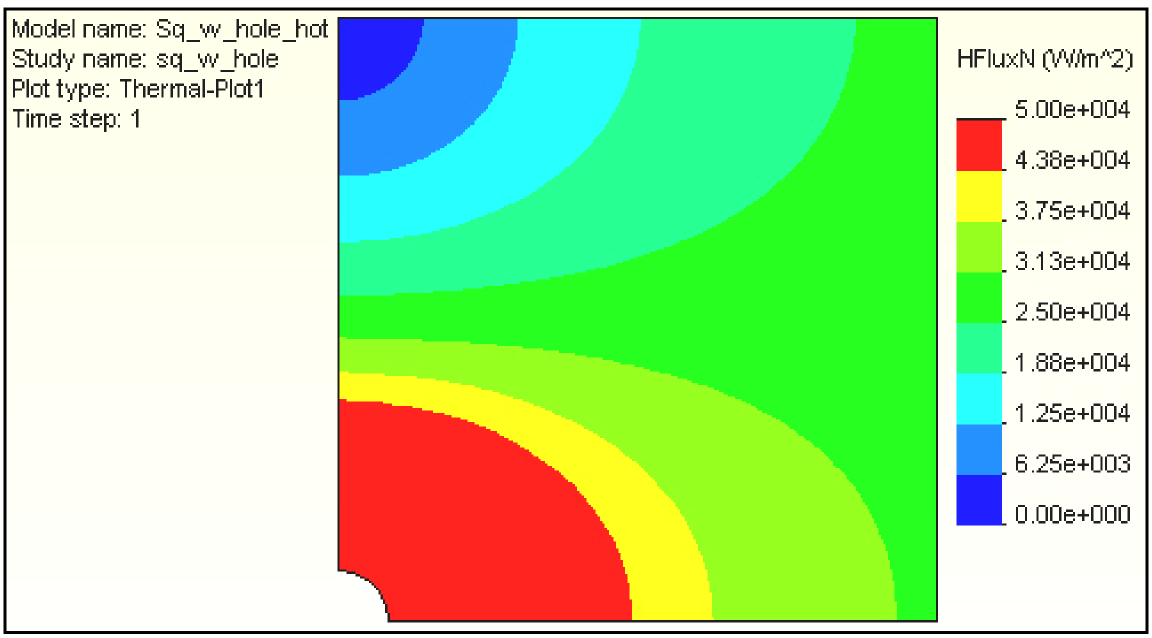


Figure 12-26 Adjusting the color bar to emphasize the convection surface

12.10 Local Singularity Conduction

12.10.1 Introduction

A hot duct work consists of thin aluminum square tube that is 1.2192 m (4 ft). on each side that contains a hot gas at 298.9 C (570 F) and is surrounded with a layer of insulating fiberglass that has an outer radius of 1.2192 m (4 ft). The insulation material has a thermal conductivity of K = 3.462e-2 W/m C (0.020 BTU/hr ft F). The duct is surrounded by free air convection having a convection coefficient of h = 408.8 W/m^2 C (72.0 BTU/hr ft^2 F) and the air is at 23.9 C (75 F). Assume the aluminum is so thin that it causes the same temperature on the inter-most square surface of the insulation. Use CosmosWorks to determine the maximum surface temperature of the duct outer surface and the distribution of the internal heat flux vectors. For a typical duct unit length of 1 m (3.281 ft) estimate how much heat flows into the air. Note that the square duct causes a re-entrant corner in the insulation. Thus, there will be a local singularity in the temperature gradient at that point.

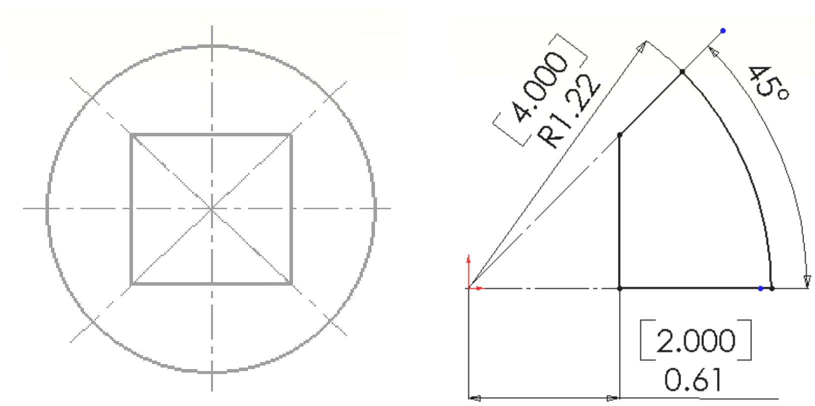


Figure 12-27 Sketch for the part extrusion

12.10.2 SolidWorks Geometric model

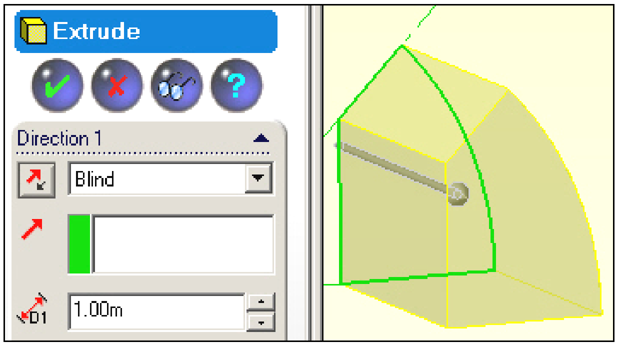
First select the working units (and dual dimension display):

1. Use **Tools→Options→Document Properties→Units**
2. Set **Unit System** to meters, and **Dual units** to feet, click **OK**
3. Use **Tools→Options→System Options→Detailing**
4. In **Dimensioning standard** turn on **Dual dimension display**, click **OK**

The problem has a one-eight symmetry, so model the first 45 degree segment.

1. Use **Top→Insert Sketch.**
2. Insert a 0 and 45 degree construction line. Place and dimension the (insulation-air) arc
3. Place and dimension the vertical (aluminum) line
4. Connect the remaining two symmetry lines, as in Figure 12-27.
5. Select **Extruded Boss/Base**.

6. In the **Extrude panel** set distance **D1** to 1 m, and method to **Blind,** click **OK**



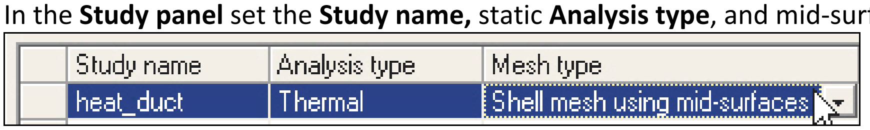
12.10.3 CosmosWorks model

After saving the extruded part you are ready to move to the CosmosWorks option. Pick the CosmosWorks icon to activate its manager panel.

12.10.3.1 Analysis type and material selection

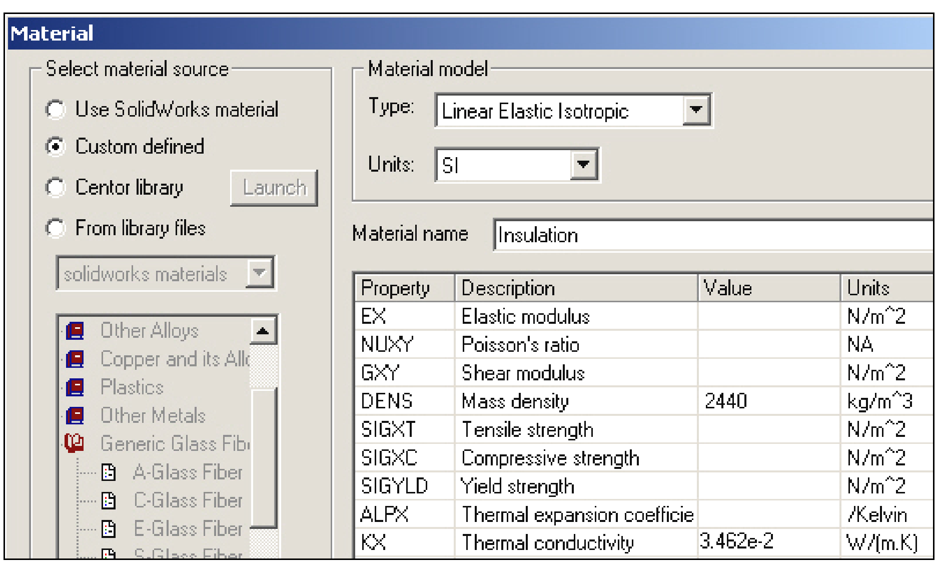
In the **Manager panel:**

1. Right click on the **Part name** and select **Study.**
2. In the **Study panel** set the **Study name,** static **Analysis type**, and mid-surface shell **Mesh type**.



3. Right click **Mid-surface shell→Apply Material to all**.

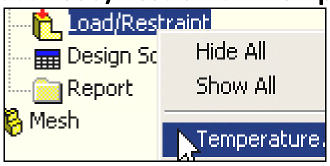
4. A review of the **Material panel** standard materials yields no match. Turn on **Custom defined** and type in 3.462e-2 for **Thermal conductivity,** click **OK**.



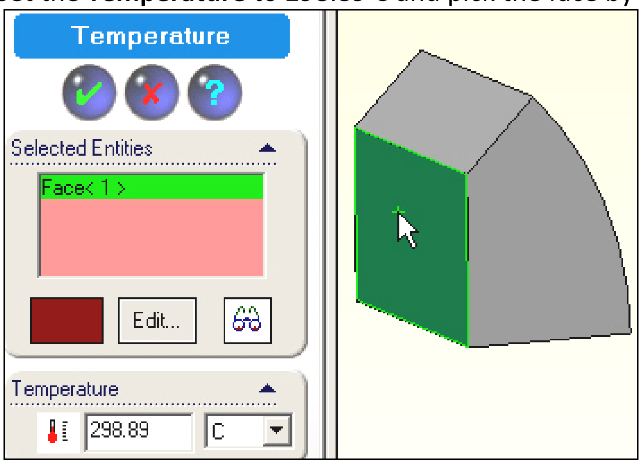
12.10.3.2 Temperature restraint

Apply the only "*essential boundary condition*" (the known temperature) here:

1. Use **Load/Restraint→ Temperature** to open the **Temperature panel**.



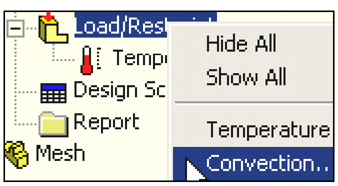
2. Set the **Temperature** to 298.89 C and pick the face by the aluminum as the **Selected Entity**, click **OK**.



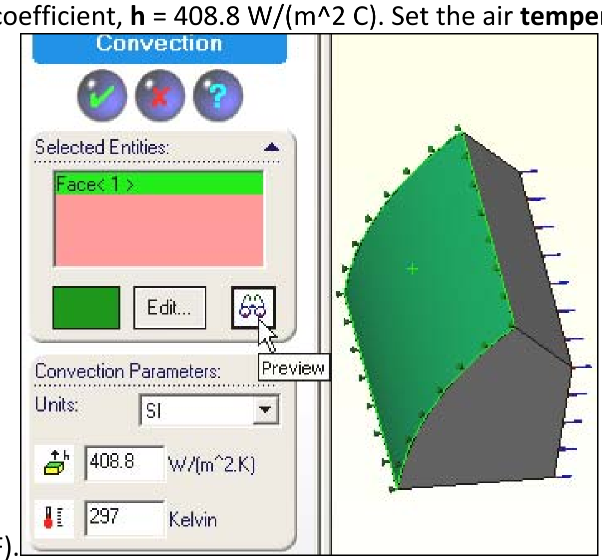
12.10.3.3 Convection load

Invoke the only loading condition:

1. Use **Load/Restraint→Convection** to open the **Convection panel.**



2. There pick the curved face as the **Selected Entity.** In **Convection Parameters** set the convection coefficient, **h** = 408.8 W/(m^2 C). Set the air **temperature** to 296.9 K (75



F).

3. Select **Preview,** click **OK.**

Remember that when you create a mid-surface shell mesh it will be assigned the above extrusion thickness and will use it to compute the effective surface area that is subjected to the air convection.

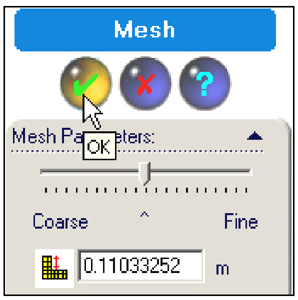
12.10.4 Insulated surfaces

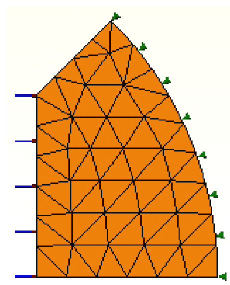
The insulated surfaces, which correspond to the two symmetry planes, require no action. That is because in any finite element thermal analysis that state (of zero heat flux) is a "*natural boundary condition*".

12.10.5 Mesh generation

For the preliminary analysis accept the default mesh size:

1. Use a right click **Mesh→Create**
2. In the **Mesh panel** click **OK** for a default mesh (which is rather crude).





12.10.6 Temperatures computation

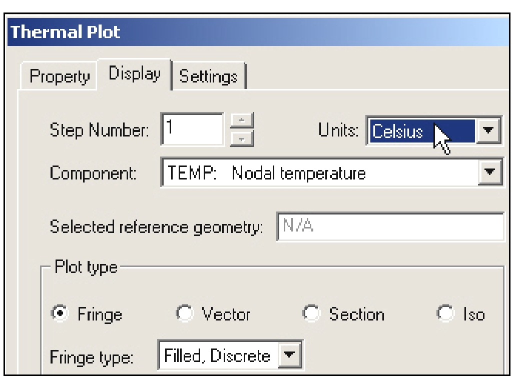
Start the temperature solution with right click on the **Name→Run**.

12.10.7 Post-processing

12.10.7.1 Temperatures

Under **Thermal** double click on **Temperature→Plot 1**. The default contour plot of temperatures would appear as a smoothed (Gouraud) color image. Usually a stepped shaded image gives a better hint of a bad mesh. Create that with:

1. A right click in the graphics window, **Edit Definition→Thermal Plot→Display.** Set **Units** to Celsius.

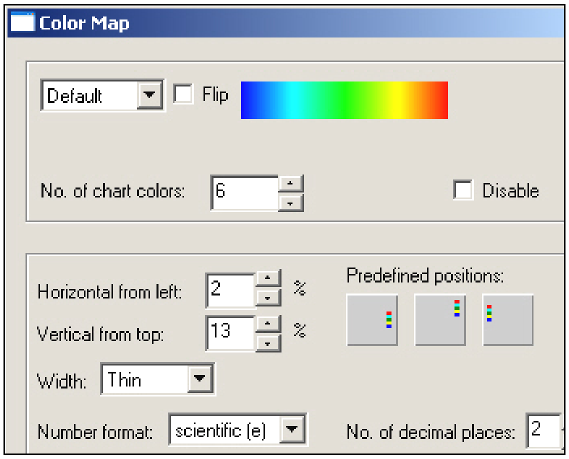


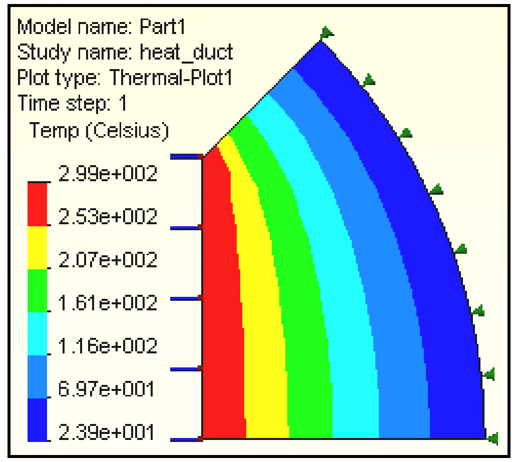
2. Change **Fringe type** to discrete filled, click **OK**.

3. Right click in the graphics window, select **Color map.**

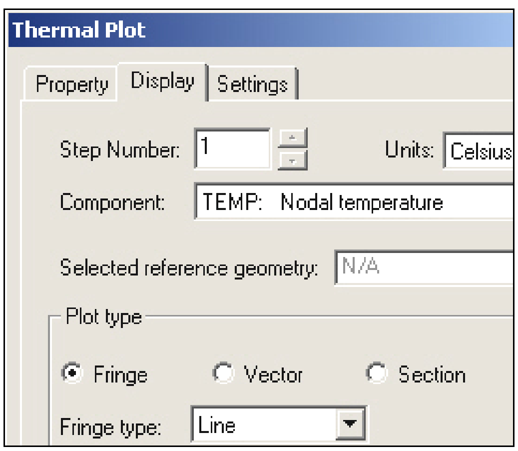
4. In the **Color Map** panel pick 6 **colors** of thin **Width** and 2 **Decimal** places, click **OK**.

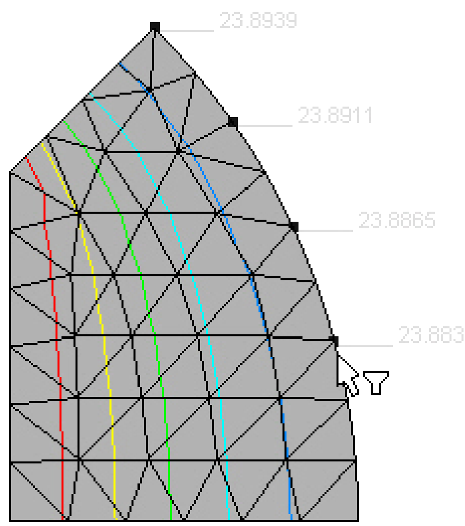




Note that the highest temperature contour, near the upper symmetry plane is wiggly. That indicates that this mesh is too crude in the high temperature gradient (and high hest flux) region. For another type of view generate a temperature line contour:

1. Right click in the graphics window, **Edit Definition→Thermal Plot→Display.**
2. Change **Fringe type** to Line, click **OK**.



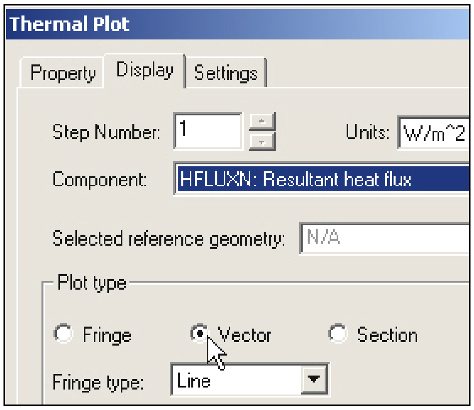


It also shows the warning wiggles that the default smoothed plot tends to hide (try one). Observe that the maximum surface temperature is barely above that of the air.

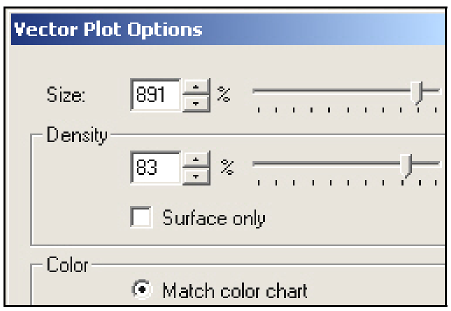
12.10.7.2 Heat flux

The heat flux is a vector quantity defined by Fourier's Law: **q** = -K **Δ**T. Thus, it is best displayed as a vector plot using typical controls:

1. Right click in the graphics window, **Edit Definition→Thermal Plot →Display.**
2. Set **Units** to W/m^2.
3. Pick **Component** resultant heat flux.
4. **Plot type** vector
5. **Fringe type** line, click **OK.**



6. Control the plot with right click in the graphics window, **Vector Plot Options.**
7. Vary the **Size** and **Density** (of the percent of vectors displayed).



The heat flux vector plot is shown in Figure 12-28 along with the heat flux magnitude. The latter line contour plot of the magnitude of the heat flux may also identify where the mesh is too crude and needs to be re-built with engineering judgment (via **Mesh→Apply Control**).

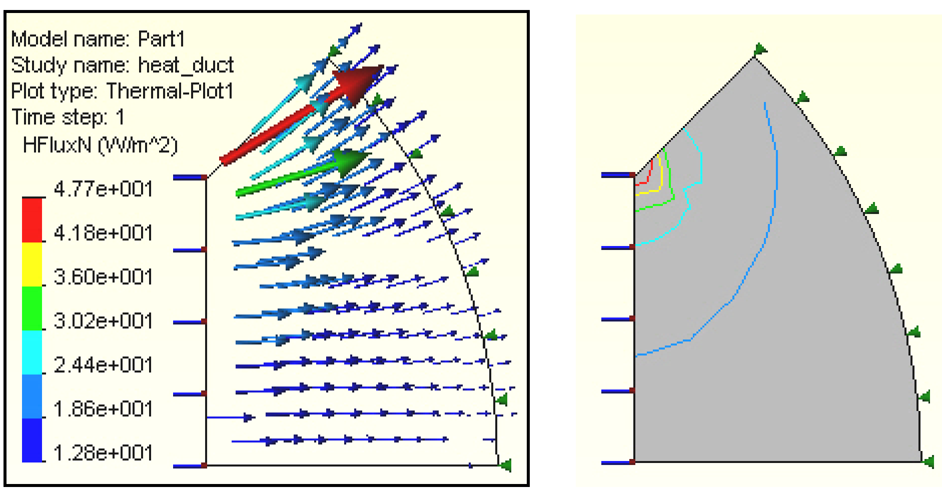


Figure 12-28 Heat flux vectors and magnitudes

12.10.8 Thermal reactions

A heat flow in or out of the system must occur at every specified temperature in order to maintain that desired temperature. If your finite element system provides those data it is good practice to review them. CosmosWorks provides a way to obtain the thermal reactions.

In this case heat is flowing in along the straight inner side, and an equal amount of heat must flow out of the curved outer side (since there was no internal heat generation). To obtain the reaction heat flow:

1. Right click in the graphics area of a heat flux plot, pick **List Selected…**
2. Pick the desired edge (straight side here) as the **Selected items**.
3. Click on **Update**. The reaction flux values appear in the list at each node, and the **Total Heat Flow** appears in the **Value** column. In this case, as seen in Figure 12-29, the total is listed as 13.75 W flowing into the part.
4. To see a graph of the heat flux along the edge, select **Plot**. That graph, in the same figure, indicates a bad mesh if the graph is not smooth. Here it seems smooth, but it has a very sharp gradient at one end.

The heat flow is the area under the curve. In other words, it is the integral of the normal component of the heat flux along the part edge. The sharp gradient occurs at the re-entrant corner of the full part, as seen in Figure 12-28. That always causes an infinite heat flux to develop at that point, in theory. Here, the outlet side is smooth and should have an equal and opposite value of the reaction heat flow. If not, that shows that the mesh needs to be refined.

Check that side with:

1. Right click in the graphics area of a heat flux plot, pick **List Selected…**
2. Pick the desired curved outer side as the **Selected items**.
3. Click on **Update**. The reaction flux values appear in the list at each node, and the **Total Heat Flow** appears in the **Value** column. In this case, as seen in Figure 12-29, the total is listed as 14.25 W flowing out of the part.
4. To see a graph of the heat flux along that edge, select **Plot**.

Your model was only 1/8 of the total domain. The true heat loss is Q_{total} = 113.6 Watts.

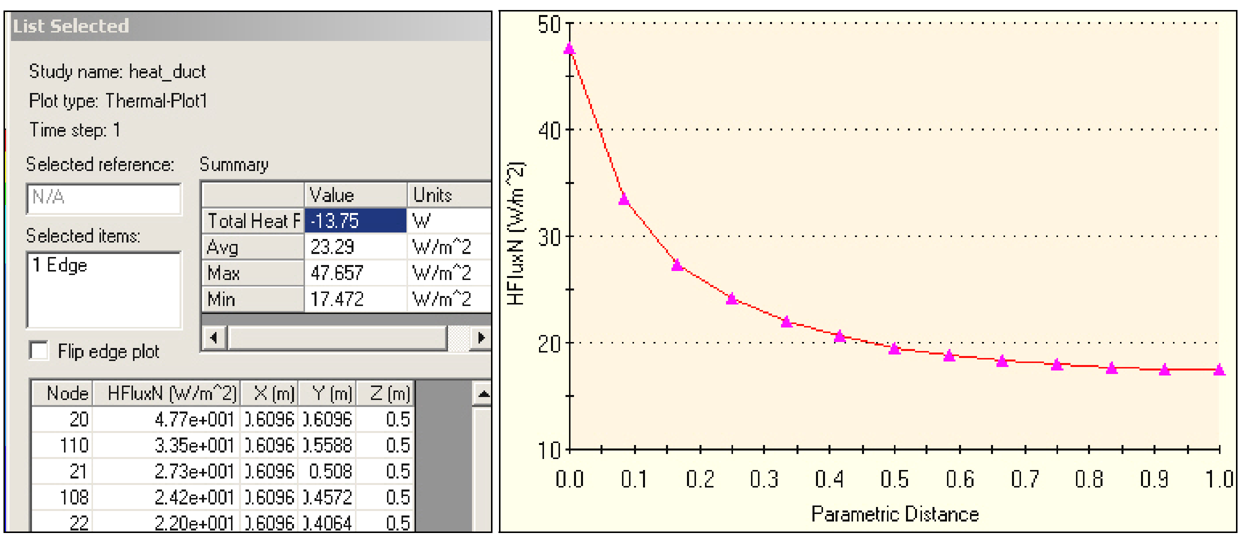


Figure 12-29 Inner straight edge heat flow data

The lack of balance between the heat flows in and out is only about 4%. Even so, a small refinement of the mesh should give smoother (more accurate) contours and a better heat flow balance. A new mesh is shown in Figure 12-31. The new values of the heat flow are in better balance, having only a 1% difference. The new values are 14.2 W, as seen in Figure 12-32.

The corresponding new graphs of the normal heat flux along the inlet and outlet are seen in Figure 12-33. They are both smoother than before. The maximum value on the outlet side (< 18 W/m^2) has barely changed. However, there has been a large change on the inlet side near the re-entrant corner. Its peak value has doubled, jumping from about 45 to about 90 W/m^2 at the "corner". Comparing the two graphs clearly shows the effect of the weak local singularity. Reasonable mesh refinement helps, but you will never reach the infinite value at the corner point (because the element smears it out, except for special singularity elements).

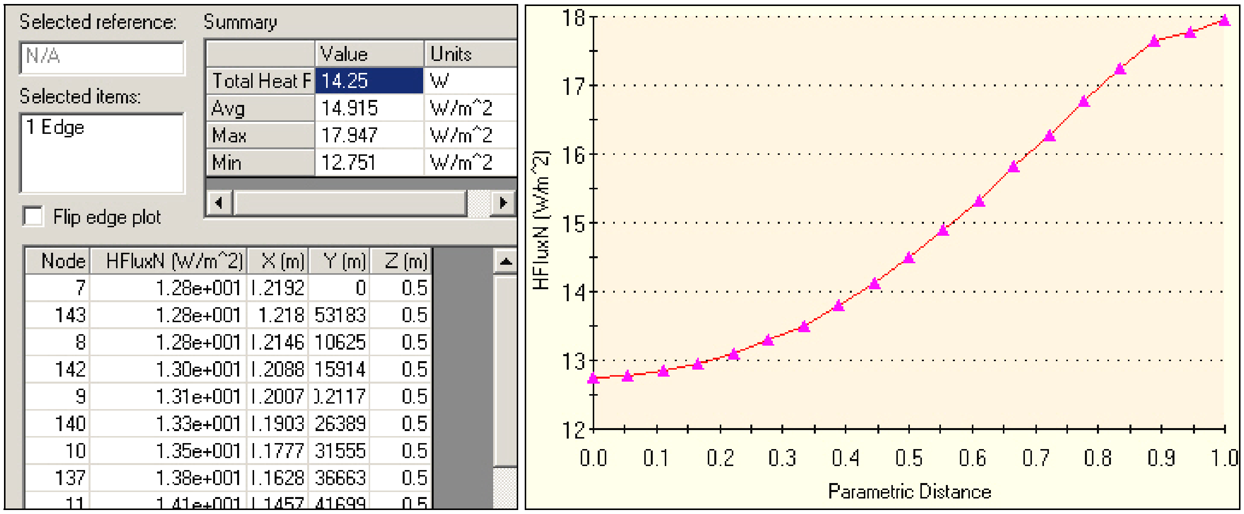


Figure 12-30 The heat flux data on the outer edge

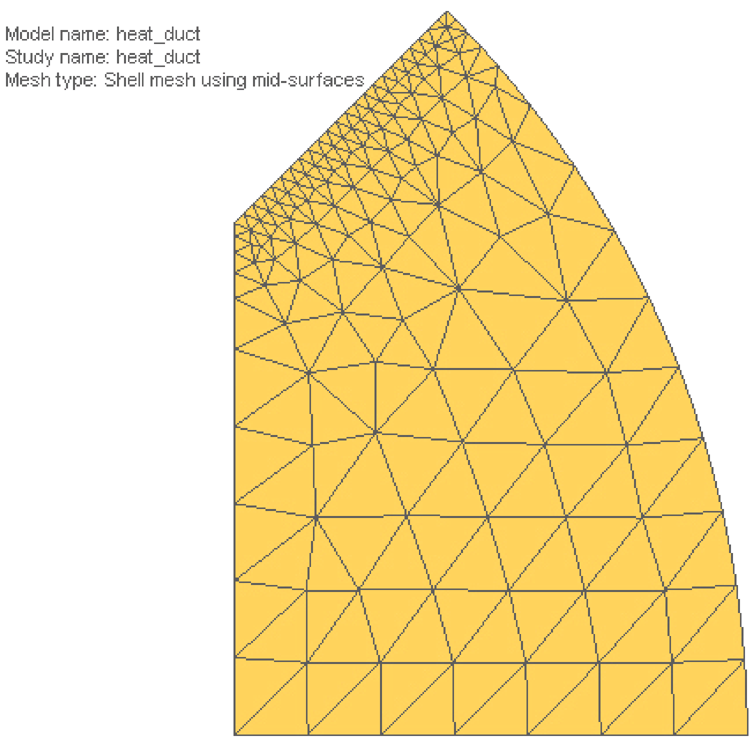


Figure 12-31 Refined mesh for better heat flow balance

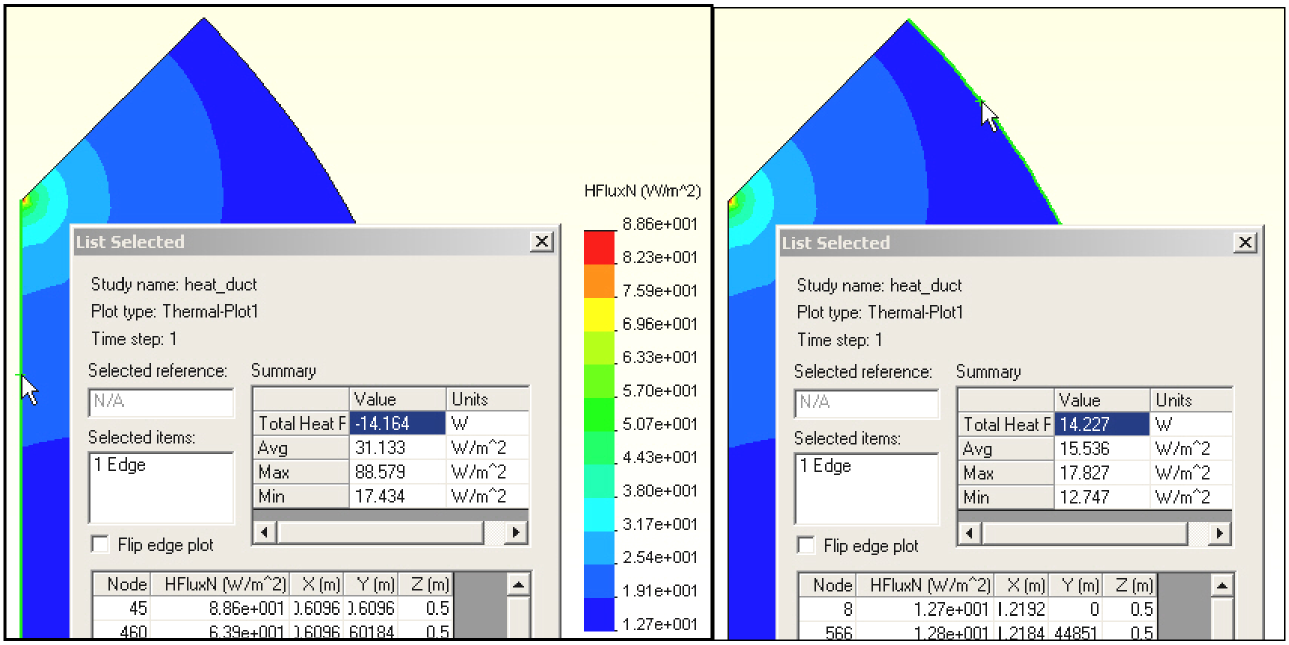


Figure 12-32 New heat flow values in and out (14.2 W)

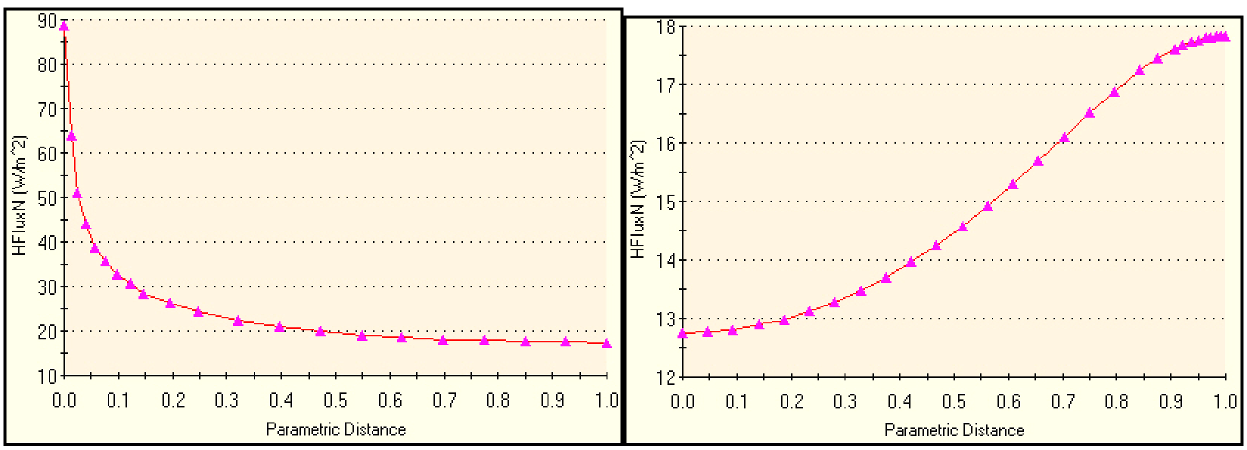


Figure 12-33 Heat flux graphs in (left) and out for the refined mesh

12.10.9 Validation of computed results

12.10.9.1 Temperatures

Before beginning the above analysis you should have estimated the temperature result and/or attempted to bound it. For a plane wall with a known inside temperature on one side and convection on the other the exact temperature solution is linear through the wall. The analytic solution for the temperature of the convection surface is

$$T_s = (h_{air} T_{air} + T_{wall}\, k/L) / (k/L + h_{air}),$$

where L is the thickness of the wall. The temperature along a line of symmetry can often be modeled with a 1D model that has the same end conditions as the symmetry line. Here those end conditions are the same and it appears that only the lengths vary. The lower length is L_0 = 0.6096 m (2 ft). Therefore, the estimated outside wall temperatures there is

T_0 = (408.8 W/m^2 C * 23.9 C + 298.9 C * 3.462e-2 W/m C / 0.6096 m)

____ = 23.9 C.

(3.462e-2 W/m C / 0.6096 m + 408.8 W/m^2 C)

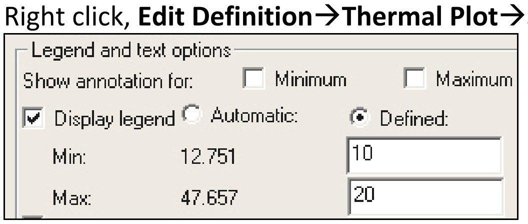
The T_0 temperature agrees well with the computed result, and suggests a user error in the given data. The convection coefficient is so relatively high (compared to k/L) that it acts almost like an essential boundary condition requiring the outer wall to be at the air temperature of 23.9 C. Most likely the convection coefficient is too high or the conductivity is too low. One of those data items probably has a decimal point error (was off by a factor of 10 or 100). Even so, you can estimate the heat flow through the wall there as $q_0 = K (T_{wall} – T_{air}) / L_0$, which gives q_0 = 9.52 W/m^2. The average value is about 14.2 W/m^2 so the estimated value is reasonable. You might be tempted to also use the above formulas for the upper wall. However, the diagonal corner is a re-entrant corner. Thus, it causes a mild local singularity and a rapid change in the temperature gradient there (which the crude mesh has mainly missed). So you can not use a uniform wall estimate.

12.10.9.2 Heat flow validation

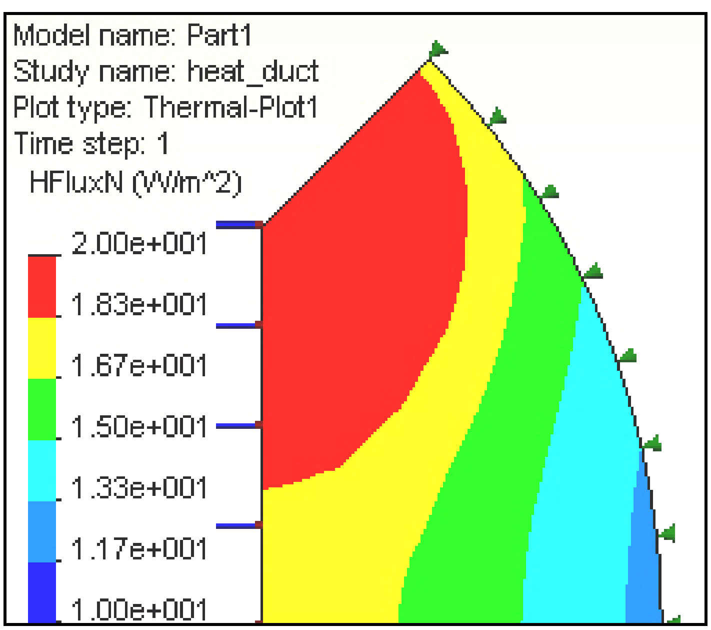
Some finite element systems do not make it easy to recover those data, but basic engineering will give you an estimate of the total heat flow (per unit length assumed here for the thickness into the page). From the vector plot of heat flow you see that at the outer surface that the flow is basically normal to the surface. (That is approximately true on the inner surface too.)

Integrating the normal heat flux passing through the outer surface gives the total heat lost. To estimate the heat loss, manually change the color bar to more clearly give the range along the outer surface:

1. Right click, **Edit Definition→Thermal Plot→Settings.** Set the **Display legend** to defined.



2. Assign a **minimum** value of 10, and **maximum** of 20 W/m^2, respectively (arrived at by trial and error), for the results below.



Estimate the arc length of the outer surface associated with each solid color heat flux range. Multiply that distance by the thickness (1 m) to get the heat outflow surface area, and multiply by the average contour value (for that color) for its heat loss, in Watts. The approximate angles from the bottom are 12.1, 13.0, 7.7 and 12.2 degrees. The correspond arc lengths are 0.258, 0.277, 0.164, and 0.260 m, respectively. The corresponding heat flux values (by averaging the color value ranges) are 17.5, 15.9, 14.2, and 12.5 W/m^2, respectively. The total heat loss, per unit length, is estimated at Q = (17.5*0.258 + 15.9*0.277 + 14.2*0.164 + 12.5*0.260)*1.0, or Q = 14.5 W. This agrees very well with the Cosmos value of 14.2 W.

Your computer model was only 1/8 of the total domain. Therefore, the true heat loss is Q_{total} = 116 Watts. Since the heat flow out from the cylindrical insulating surface must equal the heat flow in over the square duct surface you could employ a similar process on the inner edge as a double check (try it).

12.11 Crossing Pipes Analysis

Chapter 7 of [7] outlines a steady state thermal analysis of an assumed pipeline junction. Here alternate points of view and additional post-processing features will be presented. The first difference is to recognize that the geometry, material properties and boundary conditions have a plane of symmetry. Therefore, a half model can be employed. That lets the full mesh be efficiently applied to non-redundant results. The half model is seen in Figure 12-34, where each pipe hot end surface has been color coded light red, the original interior is green, and the material exposed on the symmetry plane is in yellow. This is one of those problems where the temperature solution depends only on the geometry and is independent of the material used. The actual material is brass and its thermal properties from the material library files are listed in Figure 12-35. Of course, the heat flux vectors and the thermal reactions will always depend on the thermal conductivity, *k*.

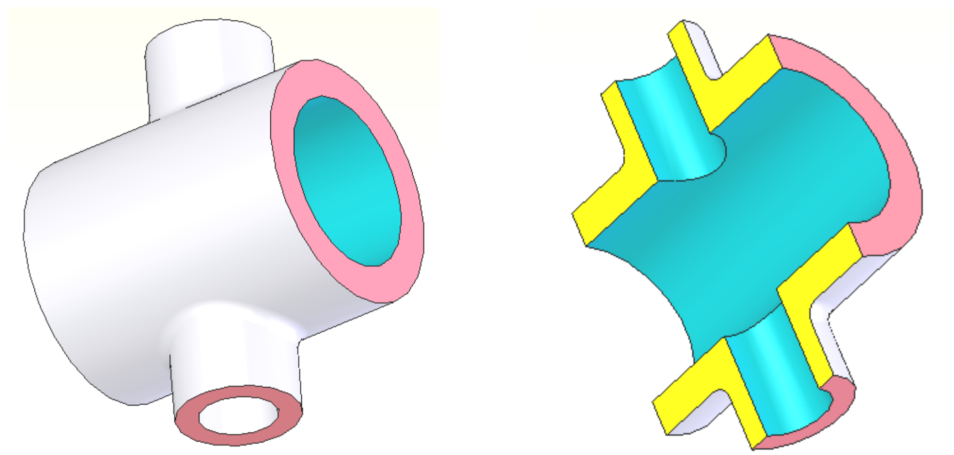
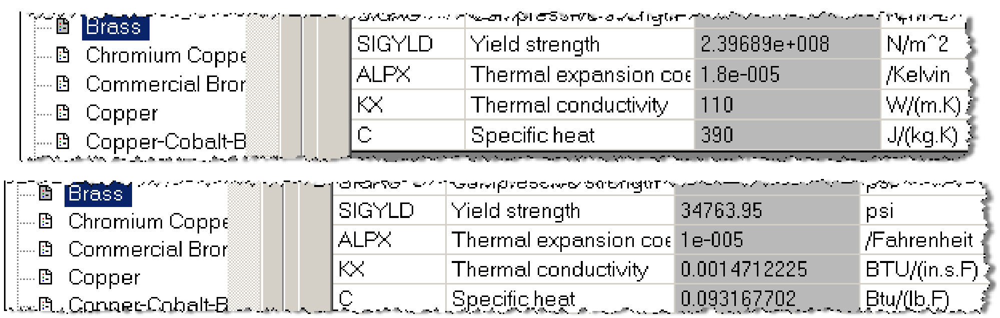


Figure 12-34 External and section view of crossing pipes



SIGYLD	Yield strength	2.39689e+008	N/m^2
ALPX	Thermal expansion coe	1.8e-005	/Kelvin
KX	Thermal conductivity	110	W/(m.K)
C	Specific heat	390	J/(kg.K)

SIGYLD	Yield strength	34763.95	psi
ALPX	Thermal expansion coe	1e-005	/Fahrenheit
KX	Thermal conductivity	0.0014712225	BTU/(in.s.F)
C	Specific heat	0.093167702	Btu/(lb.F)

Figure 12-35 Switching units for property displays (note significant figures)

Most experimental properties are known only to two or three significant figures. However, the library table values can be misleading because the properties sometimes were measured in a different set of units and multiplied by a conversion factor and incorrectly displayed to 7 or 8 significant figures. Here the conductivity of brass is displayed (Figure 12-35, top) in its experimentally measured units as k = 110 W/m-K, but had you used English units it would be converted and displayed (Figure 12-35, bottom) to 8 significant figures instead of a realistic value of about k = 1.47e-3 BTU/in-s-F.

This pipe junction has four sets of restraints, or "essential boundary conditions", where the temperature is specified at each pipe end ring surface. The first is applied by selecting the hottest end area and assigning it a given value of 400 C. The other three ends are treated in a similar way. The first restraint is illustrated in Figure 12-36 (left). That figure also shows that the default restraint names have been replaced with more meaningful ones. That practice often saves time later when a problem has to be reviewed.

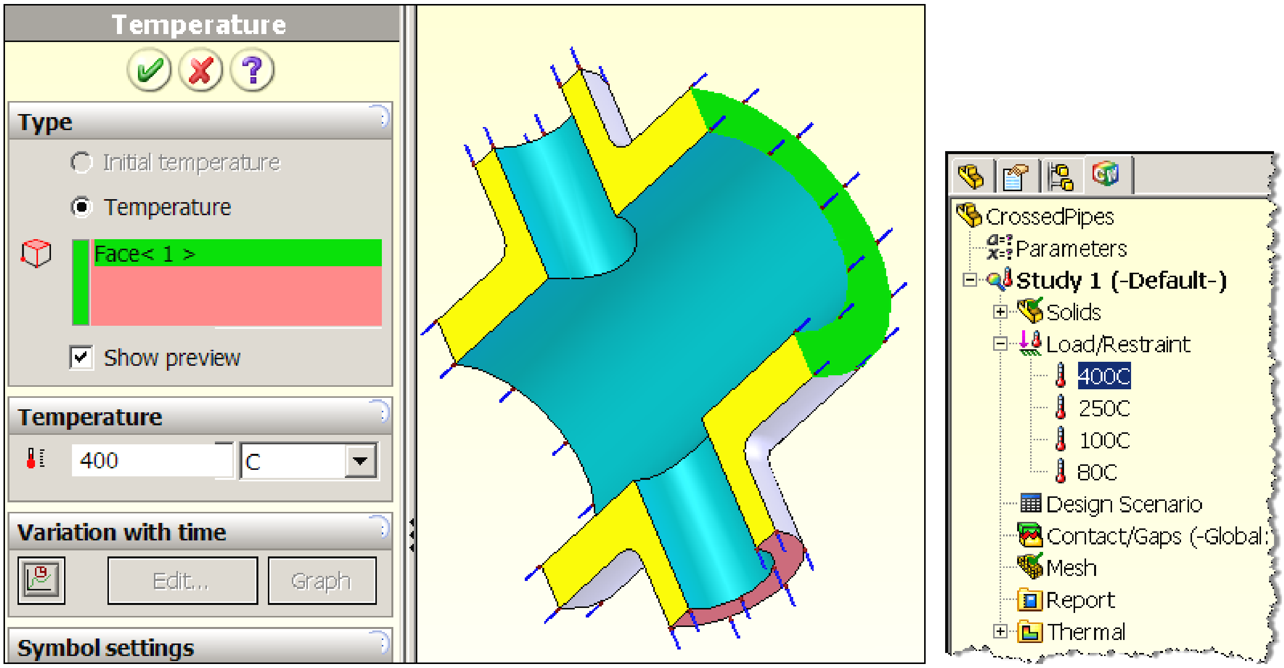


Figure 12-36 Inlet temperature at large pipe

The boundary condition on the (yellow) symmetry plane must be introduced to account for the removed material. Since it is a plane of symmetry it acts as a perfect insulator. That is, there is no heat flow normal to the plane ($q_n = 0$). That is the "natural boundary condition" in a finite element analysis and is automatically satisfied. The original interior surface also has not yet been specifically addressed. Neither has the remaining exterior (gray) surface. They both also default to insulated (or adiabatic) surfaces having no heat flow across them. That is probably not realistic and convection conditions there will be considered later in the appendix. All that remains is to generate a mesh, compute the temperatures and post-process them.

The automatic mesh generator does a good job. When the solution is run the default temperature plot (in Figure 12-37) shows two hot pipe regions and two cooler ones. However, in the author's opinion, the default continuous color plots hide some useful engineering checks of the temperatures. Therefore, in Figure 12-38, the plot settings are changed to first show discrete color bands. The new inner and outer surface temperature displays are seen in Figure 12-39. They make it a little easier to check that the contours are perpendicular to the flat symmetry plane, and nearly parallel to surfaces having constant temperatures. Another alternative is to change the settings to line contours to obtain the results of Figure 12-40. You can also change the number of color segments to reduce or increase the number of contours lines displayed in either mode.

It is common to make technical reports more specific by graphing the results along selected lines or curves. That is done by utilizing the "*List Selected*" option after the temperature has been plotted. Then you select an edge, of model split line, pick *update* and then *plot*. The selected line, at an inner edge adjacent to the hottest inlet (and seen previously in Figure 12-39), is seen in Figure 12-41 . The resulting temperature graph versus the non-dimensional position along the line is given on the left in Figure 12-42, while its continuation along the curved cylinder intersection curve is shown on the right.

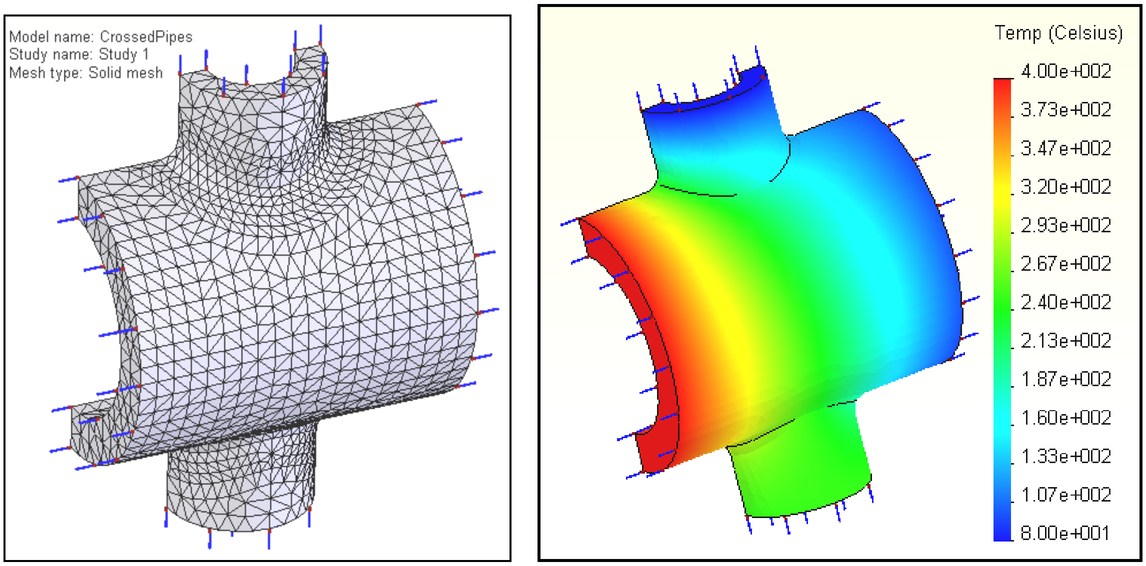


Figure 12-37 Half model mesh and temperature result

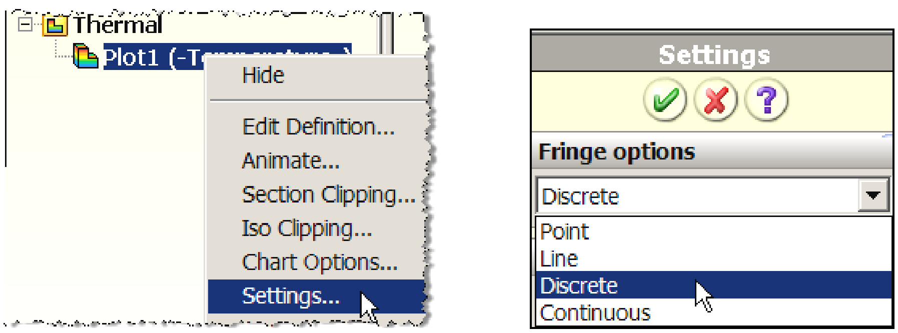


Figure 12-38 Switching from continuous to discrete contours

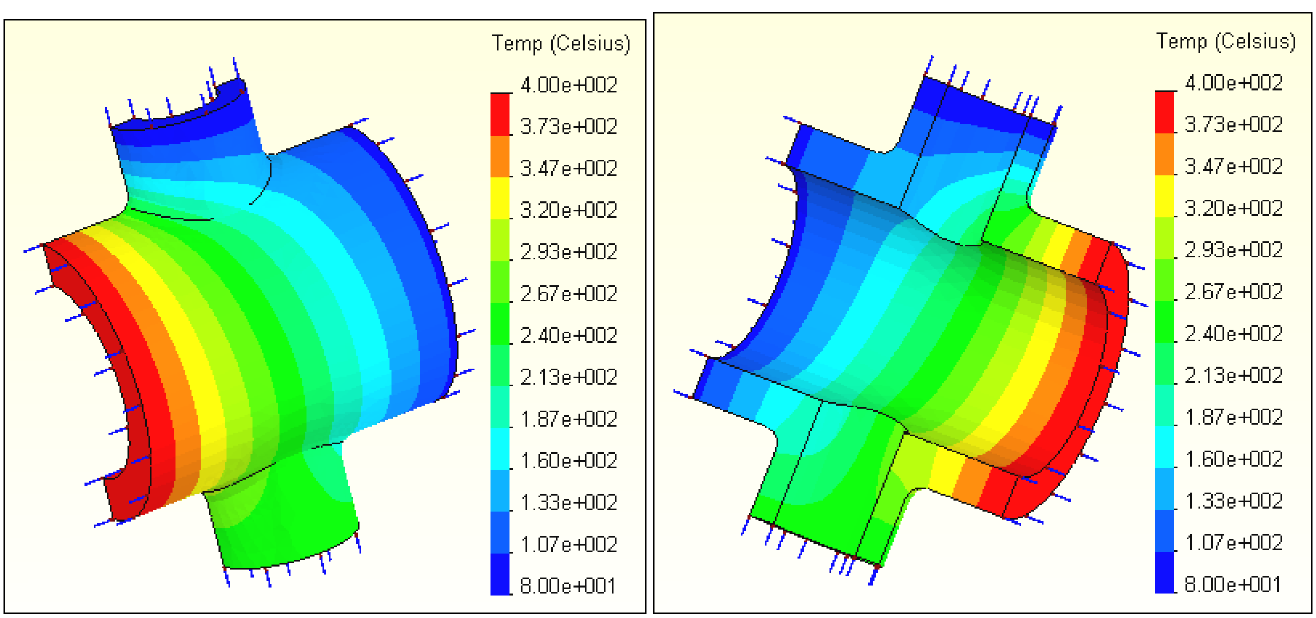


Figure 12-39 External and internal temperature result

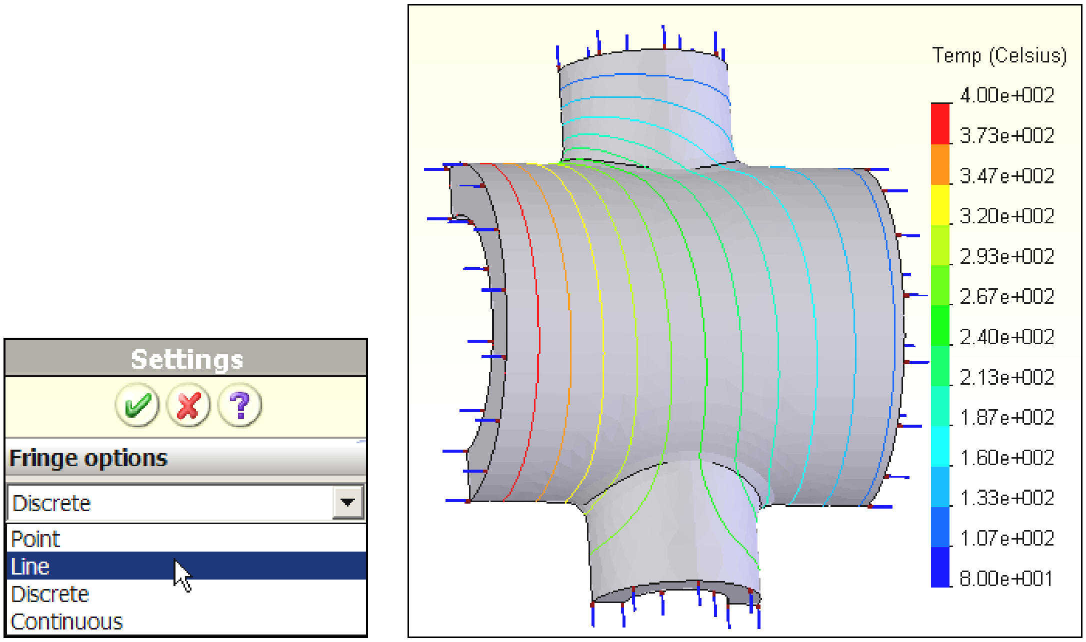


Figure 12-40 Line contour temperature result

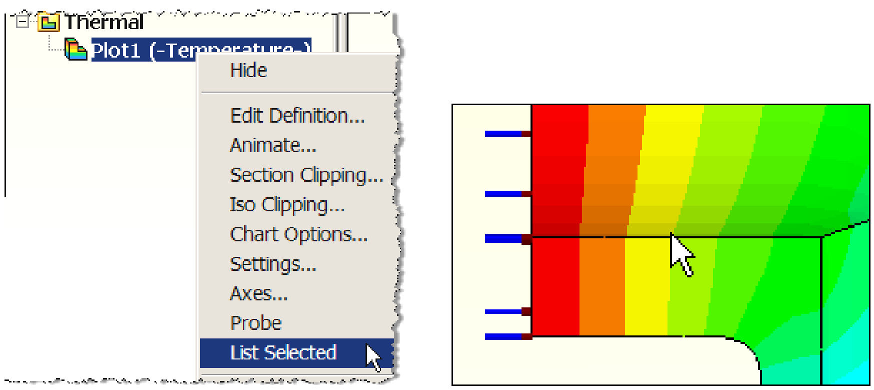


Figure 12-41 Selecting model lines for graphing temperature probe lists

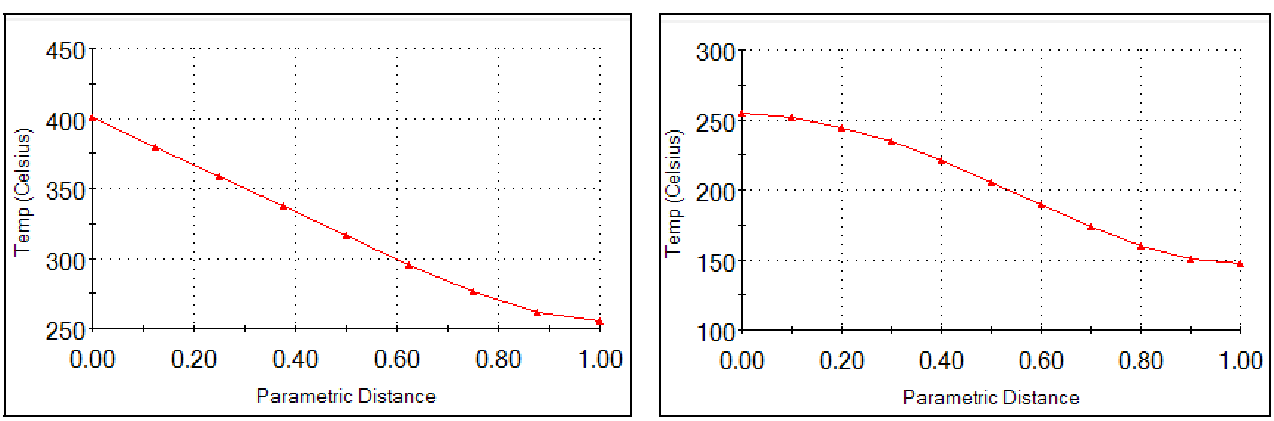


Figure 12-42 Temperature graphs along two corner lines

Since there is no convection, heat generation, or non-zero heat flux conditions the temperature results and contours are the same for all materials. However, the heat flux does change magnitude with different materials. The heat flux magnitude contours are given in Figure 12-43 (left). The contour lines are less smooth than the previous ones for the temperature because the heat flux is always less accurate than the temperatures. Had these contours shown larger wiggles it would be a signal that a finer mesh should be utilized. Since the heat flux is a vector quantity it should also be plotted as a vector. That setting change is also shown on the right in Figure 12-44. The resulting vector plot is shown in Figure 12-44 (after changes with “Vector Plot Options”). The heat flux vectors show that heat flows into the junction at the 400 C surface and out at the 80 and 100 C ends. The 250 C end has some inflow and some heat outflow.

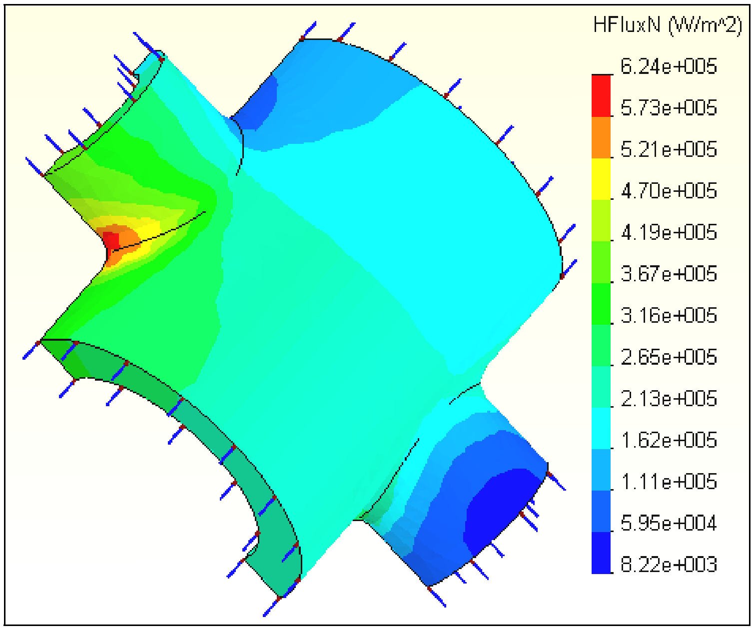


Figure 12-43 Crossing pipes heat flux contours

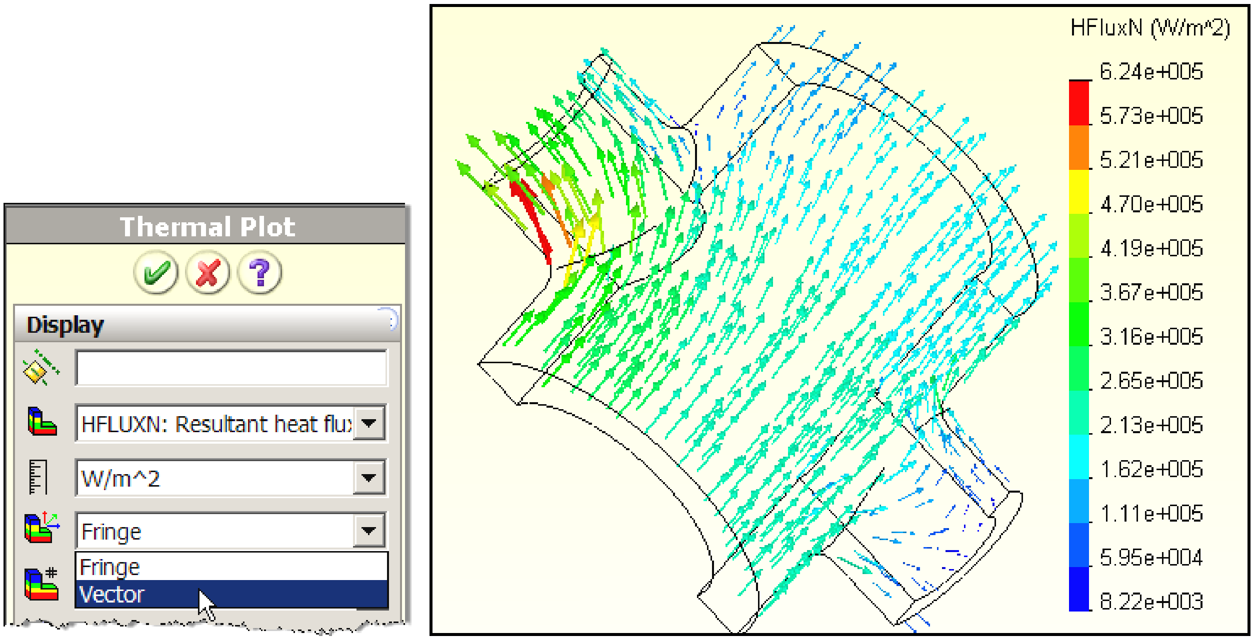


Figure 12-44 Crossing pipes internal heat flux vector distribution

Here you can also have CosmosWorks compute the thermal reactions necessary to maintain the specified temperatures. To do that use the "*List selected*" option and select each of the four pipe ends in order. At each one, you pick *update* to list and sum all the individual nodal heat flux values. Figure 12-45 shows those four total heat flows. They show that about 850 W of power in at the two hottest surfaces and out the two coolest pipe ends. Since a half symmetry model was used here, that figure needs to be doubled to determine the required input power of about 1700 W to maintain the specified temperature restraints. If the current example were changed to steel with a conductivity of about k = 51.9 W/m-K then the heat flux magnitudes and required power would drop by about a factor of two while the temperature distribution would be unchanged.

Selected reference: N/A
Selected items: 1 Face

Summary	Value	Units
Total Heat F	842.21	W
Avg	2.6094e+00!	W/m^2
Max	3.2453e+00!	W/m^2
Min	2.301e+005	W/m^2
RMS	2.6291e+00!	W/m^2

400 C End

Selected reference: N/A
Selected items: 1 Face

Summary	Value	Units
Total Heat F	8.3492	W
Avg	64909	W/m^2
Max	1.0775e+00!	W/m^2
Min	8220.5	W/m^2
RMS	72212	W/m^2

280 C End

Selected reference: N/A
Selected items: 1 Face

Summary	Value	Units
Total Heat F	-570.03	W
Avg	1.7628e+00!	W/m^2
Max	2.1351e+00!	W/m^2
Min	1.1956e+00!	W/m^2
RMS	1.7906e+00!	W/m^2

100 C End

Selected reference: N/A
Selected items: 1 Face

Summary	Value	Units
Total Heat Flow	-280	W
Avg	2.9518e+	W/m^2
Max	4.0065e+	W/m^2
Min	2.0079e+	W/m^2
RMS	3.0356e+	W/m^2

80 C End

Figure 12-45 Crossing pipes reaction total heat flow at inlets and outlets

12.11.1 Adding convection

Convection conditions or known heat flow on the boundaries will change the temperature distribution in the original problem. To illustrate this point, assume the outer surface convects to air at a temperature of 30 C (303 K) with a convection coefficient of about h = 5 W/m^2. Also let the pipe interiors convect to oil at 70 C (343 K) with an assumed convection coefficient of about 600 W/m^2. You simply have to apply two convection conditions (seen in Figure 12-46), and re-compute the solution.

The original graph of temperatures in Figure 12-42 shows less temperature drop than the new graph in Figure 12-47. The resulting discrete temperatures are shown in Figure 12-48. There are steeper temperature gradients than seen in Figure 12-39 of the original problem. The new heat flux vectors are given in Figure 12-49. There you note that the location of the maximum heat flux has changed. If you again compute the reaction heat flow (given in Figure 12-50) you need 2 * 2070 W/s = 4140 w/s.

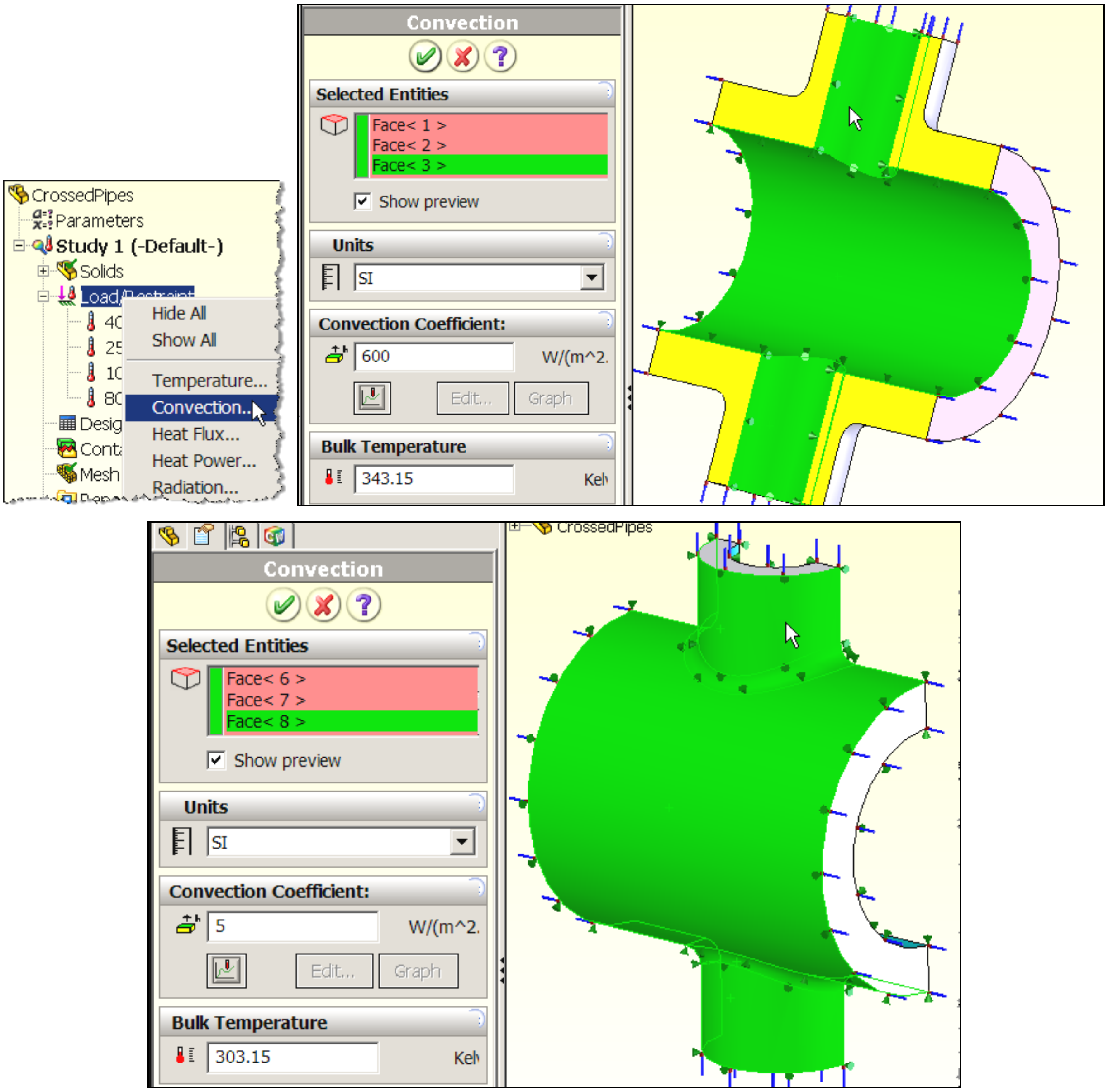


Figure 12-46 Adding internal and external convection to the crossing pipes

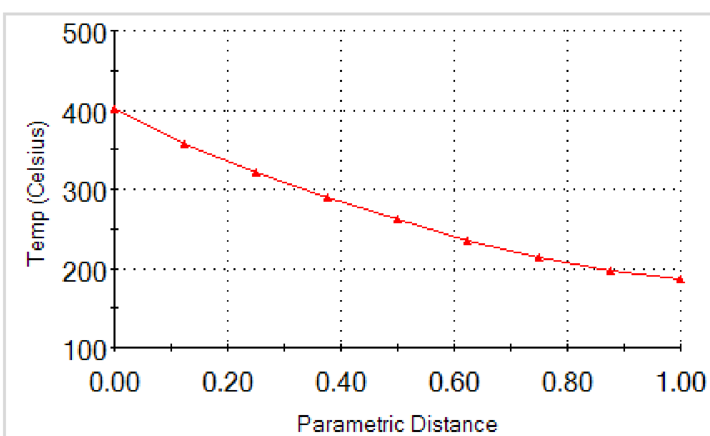


Figure 12-47 Revised edge temperature graph

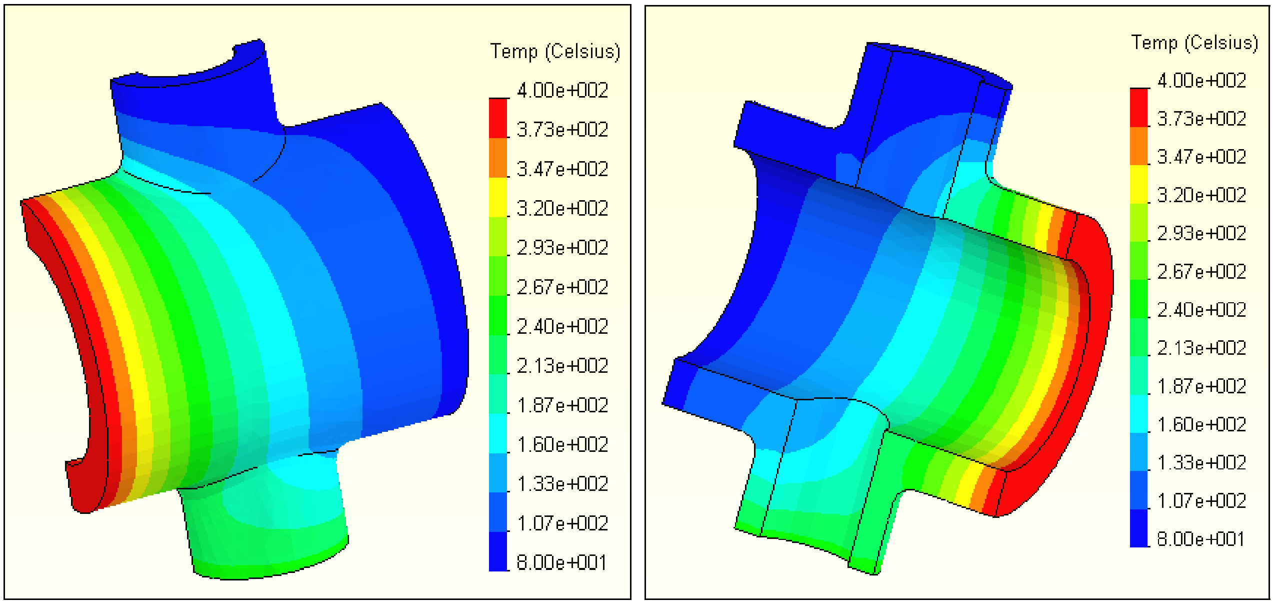


Figure 12-48 Crossing pipe temperatures for convection changes

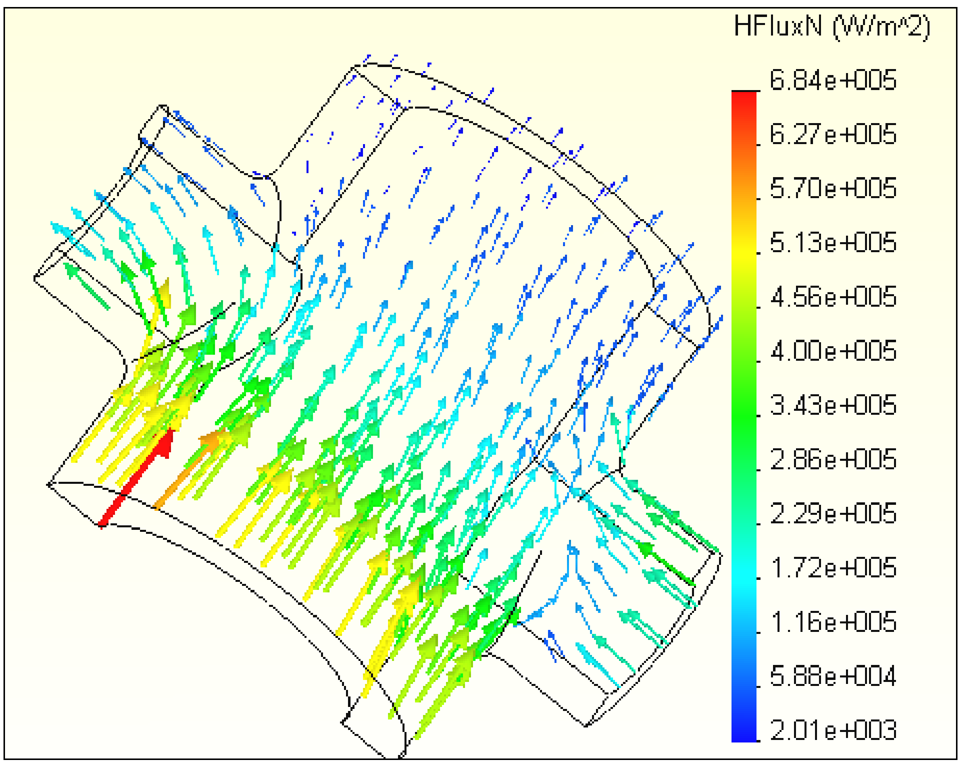


Figure 12-49 Crossing pipes internal heat flux vectors with convection effects

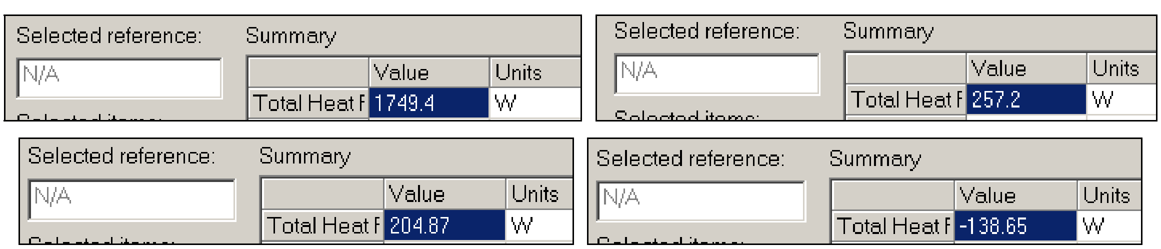


Figure 12-50 Thermal reaction total heat flow for crossing pipes with convection

12.12 Thermal Analysis of a Pressure Block

12.12.1 Introduction

You previously went through the stress analysis of a block, shown in Figure 12-51, with an internal pressure. The cylindrical internal passage carried a fluid at 400 F, while the exterior surface was cooled by natural convection by surrounding cool water (at 45 F). The convection coefficient there is typical for water, h = 2.89e-4 BTU/s in^2 F. A uniform temperature rise of an unrestrained body will not cause thermal stresses to develop. However, here you should expect non-uniform temperatures that will cause thermal stresses. Here you will get the temperature distribution, using a conductivity of k = 1.736e-4 BTU/s in F, but delay the thermal stress load case until later. (Review the CosmosWorks tutorial on thermal stresses.)

The oval hole will be assumed to be filed with another solid surrounded by an insulating material. Later, you could repeat this study in an assembly that has those materials. That would give a more accurate temperature distribution around the oval opening (which is the main location of concern).

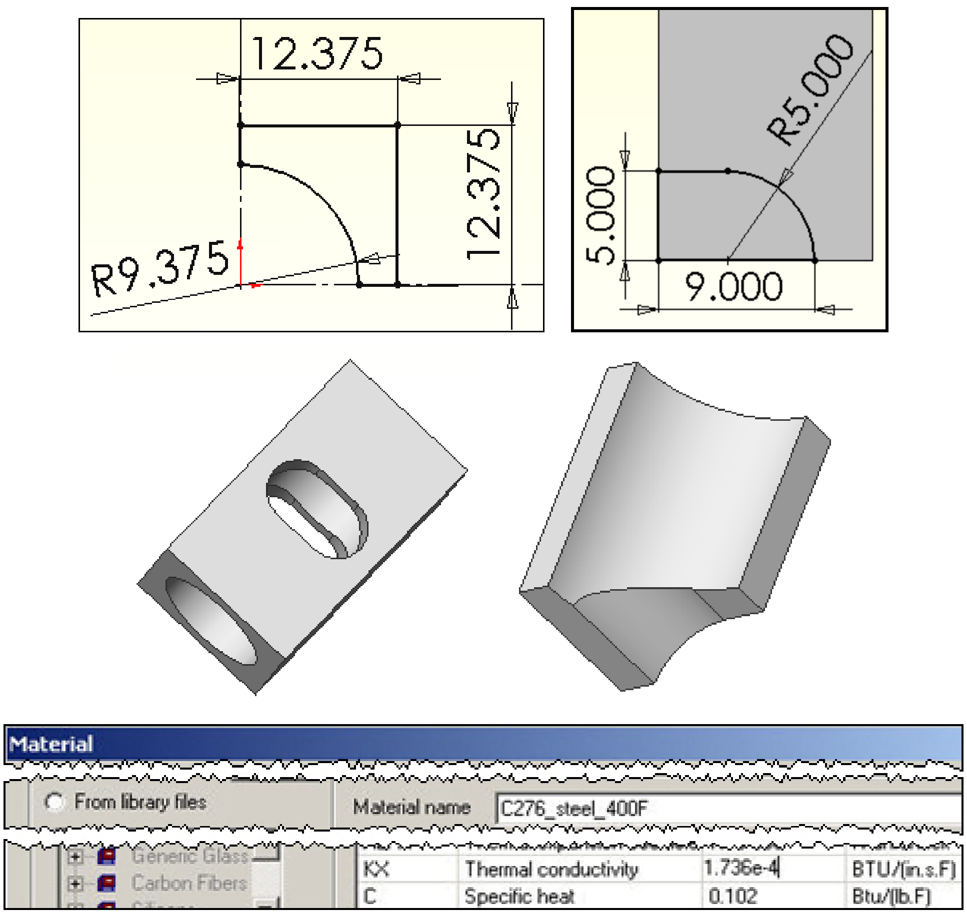


Figure 12-51 Solid block, its symmetric corner, and material

12.12.2 Estimating a solution

Before beginning a 3D study it is wise to estimate some aspect of the answer in advance. The estimate might be analytic, 1D or 2D finite element, or a combination of those. For plane walls the exact temperature varies linearly through the wall and linear finite elements can give an exact solution, with a single element through each material. But this part is closer to a thick wall cylinder, which generally has a logarithmic temperature distribution. Thus, a typical 1D finite element solution would need more than one axisymmetric element to get a reasonable approximate solution.

There is an exact solution for a thick cylinder with the inside temperature given and convection on the outer surface. Here you could use that to estimate the unknown part temperature at the convecting water surface. You could get bounds by using the minimum and maximum (corner) wall thickness. Jiji [6] gives the analytic solution for the temperature through the cylinder. Let *a* and *b* denote the inner and outer radii, respectively. For a conductivity of *k* and convection to water at a temperature of T_w with a convection coefficient of h_w the logarithmic temperature distribution is:

$$\mathbf{T(r) = T_a - \mathit{ln}\,(r/a)\,(T_a - T_w) \,/\, [k \,/\, b\, h_w + \mathit{ln}\,(b/a)].}$$

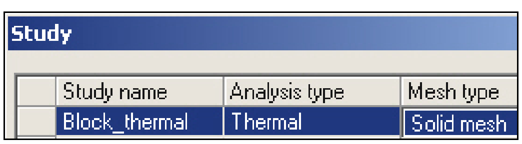
The radial heat flow (W/s) per unit length of the cylinder is constant, but the heat flux, q_r, is not because the cylindrical area of flux flow, $A = 2\pi\, r$, is not. The heat flux (W/s in^2) is

$$q_r = \{(T_a - T_w) \,/\, [\,\mathit{ln}\,(b/a) \,/\, k + 1/(b\, h_w)]\} \,/\, r.$$

To estimate the body temperature at the water simply set r = b. From the given data a = 9.375”, for this part 12.375” ≤ b ≤ 14.5” so say b = 13.4”. Here k = 1.736e-4 BTU /s in F, T_w = 45 F, h = 2.89e-4 BTU/s in^2 F so that T (b) = T_b = 84.6 F is a reasonable average estimate, and rising to about 98 F at the thinnest section. The heat flux is about q_r = 0.020 BTU/ s in^2 at the inner radius, and drops to about 0.015 BTU/s in^2 at the outer radius.

12.12.3 Begin the CosmosWorks thermal study:

The part had one-eighth symmetry, so that relatively simple geometry will be employed again here in a thermal study:

1. First copy the geometry file and give it a new name, say Block_thermal.
2. **New→Part→OK.** Right click on the **Part name→Study** to get the **Study panel**.
3. 

4. Enter a **Study name**, pick thermal as the **Analysis type**, use a solid mesh for the **Mesh type,** click **OK**.

12.12.4 Material property

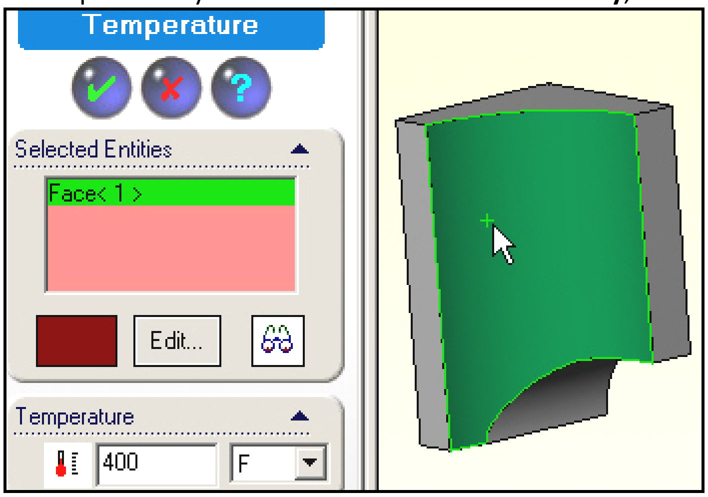
Use the same material properties for C276 steel as before (or simply just the thermal conductivity value of K = 1.736e-4 BTU/in s F).

12.12.5 Temperature restraint

Only the inner circular surface needs a restraint:

1. **Load/Restraints→Temperature** opens the **Temperature panel.**

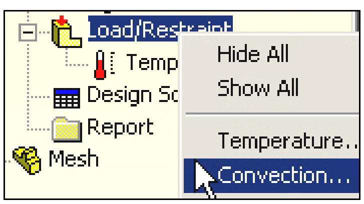
2. There pick the cylindrical face as the **Selected entity**, enter 400 F as the **Temperature,** click **OK.**



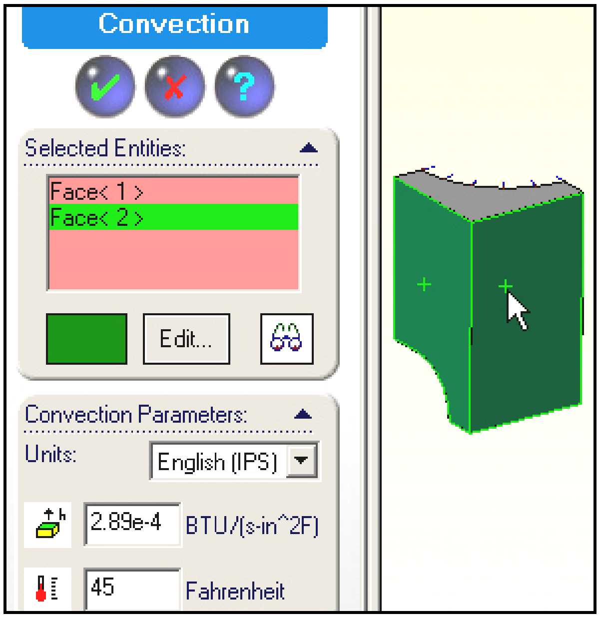
12.12.6 Convection load

Only the two flat outer walls have convection. Apply it via:

1. **Load/Restraints→Convection** to open the **Convection panel**.



2. There pick the two faces as the **Selected Entities**. Enter 2.89e-4 BTU/s in^2 F for **h**, and 45 F for the **temperature** under the **Convection Parameters,** click **OK.**



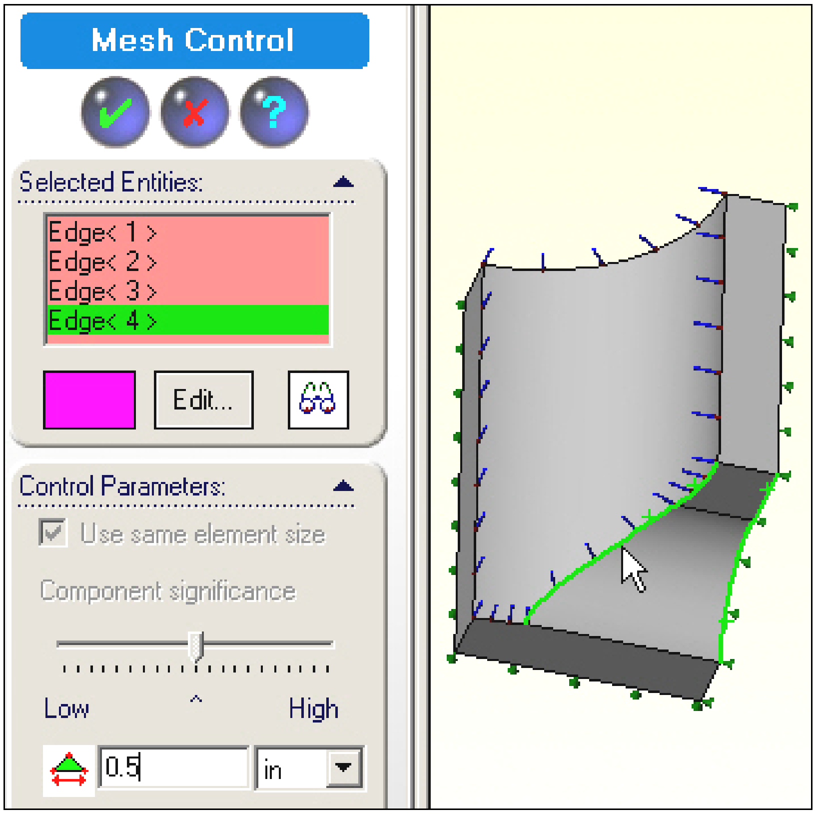
12.12.7 Insulated surfaces

The remaining five surfaces are insulated. They include the three flat rectangular symmetry planes, and by assumptions the oval access surface and the top most surface. An insulated surface (zero normal heat flow) is a natural boundary condition in finite element formulations. They require no action or input by the user.

12.12.8 Mesh control and generation

Were the oval hole not present you would expect the temperature contours to be very close to those in a thick cylinder. Handbook 1D solutions are available for thick wall cylinder. That relatively uniform temperature distribution will be disrupted by the (insulated) oval hole. Therefore, you should control the mesh there:

1. Right click on **Mesh→Apply Control** to open the **Mesh Control panel**.
2. There pick all the edge curves on the oval surface to be the **Selected Entities**, just to be safe.
3. Under **Control Parameters** reduce the element **size** from the default displayed value to 0.5 inch, click **OK**. The resulting mesh given in Figure 12-52 is reasonable fine.



12.12.9 Temperature calculations

Pick **Study name→Run** to generate the solution

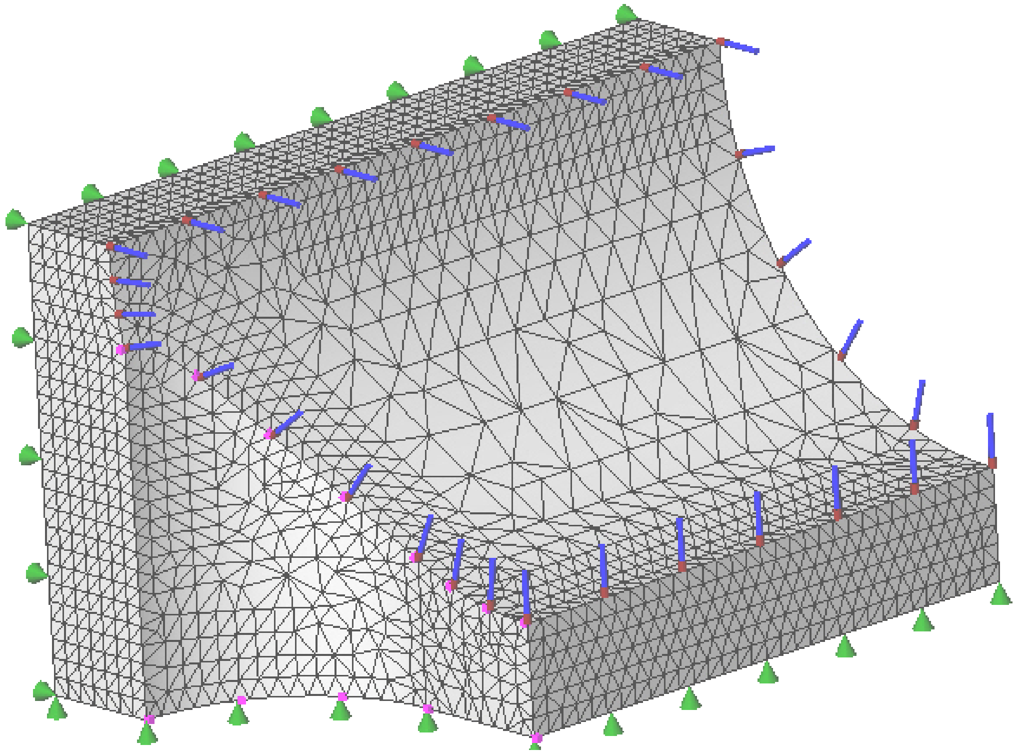


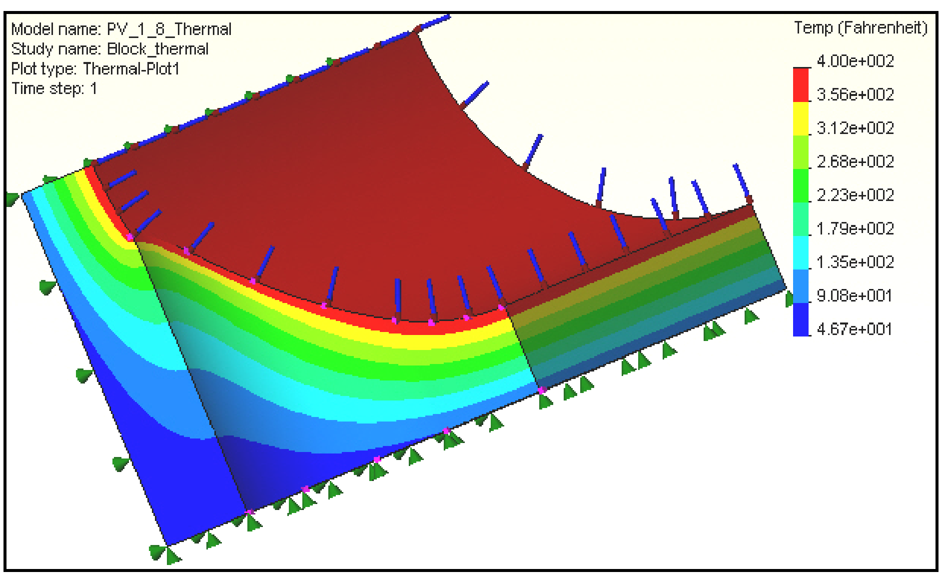
Figure 12-52 Initial solid mesh with thermal symbols

12.12.10 Post-processing

12.12.10.1 Temperatures

The only plots and probes are under the thermal report. Start with the default temperature plot:

1. **Thermal→Plot 1** gives the surface temperature plots. They will include the extreme values since no internal volumetric heat generation was active here.
2. **Rotate** the temperature and search for the region of closest contour lines (highest temperature gradient) since that region would cause the highest thermal stresses. As expected, that occurs on the internal intersection line between the oval and the cylindrical holes.



3. **Examine** that region in more detail in the bottom model view of Figure 12-53.

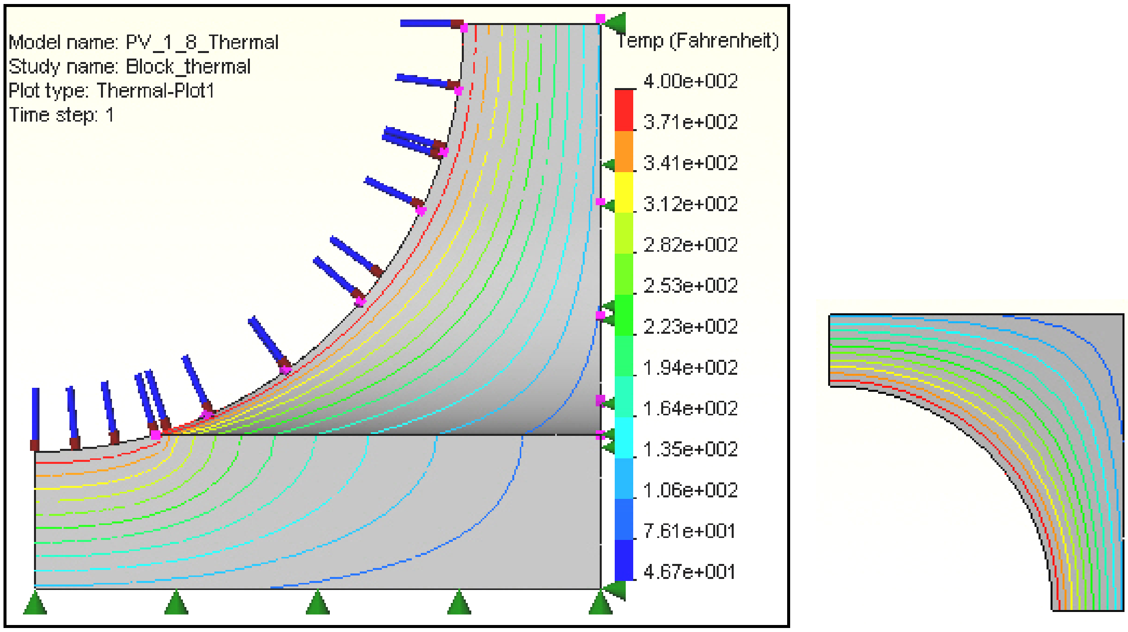


Figure 12-53 Temperature transition at middle intersection edge, and top surface

Even though the highest temperature will be on the surfaces, it may be useful to also plot an interior section so as to see how close most of the body would be to the 1D axisymmetric analytic solution:

1. Right click in the graphics area, **Edit Definition**.
2. **Select Section** and **filled fringe**, click **OK**.
3. Right click in the graphics area, pick **Clipping**. Select the **Y-axis plane** and move the **slider** to see various sections.

As expected, Figure 12-54 shows that the internal contours are almost nested cylindrical isosurfaces like (the logarithmic distribution) of a thick walled cylinder.

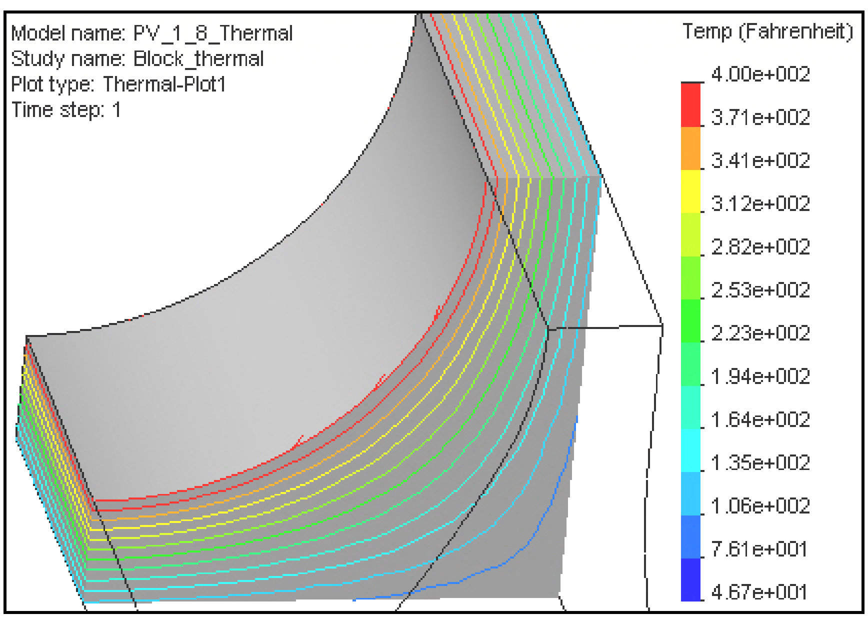


Figure 12-54 Nearly cylindrical temperature contours in an upper section

12.12.10.2 Heat flux

Regions of high thermal gradient, and thus high heat flux, may be missed with a smoothed temperature contour plot. Thus, also display them with:

1. Right click in the graphics area, **Edit Definition**.
2. Select resultant heat flux (**HFLUXN**) and the desired **units**, click **OK** (see Figure 12-55).

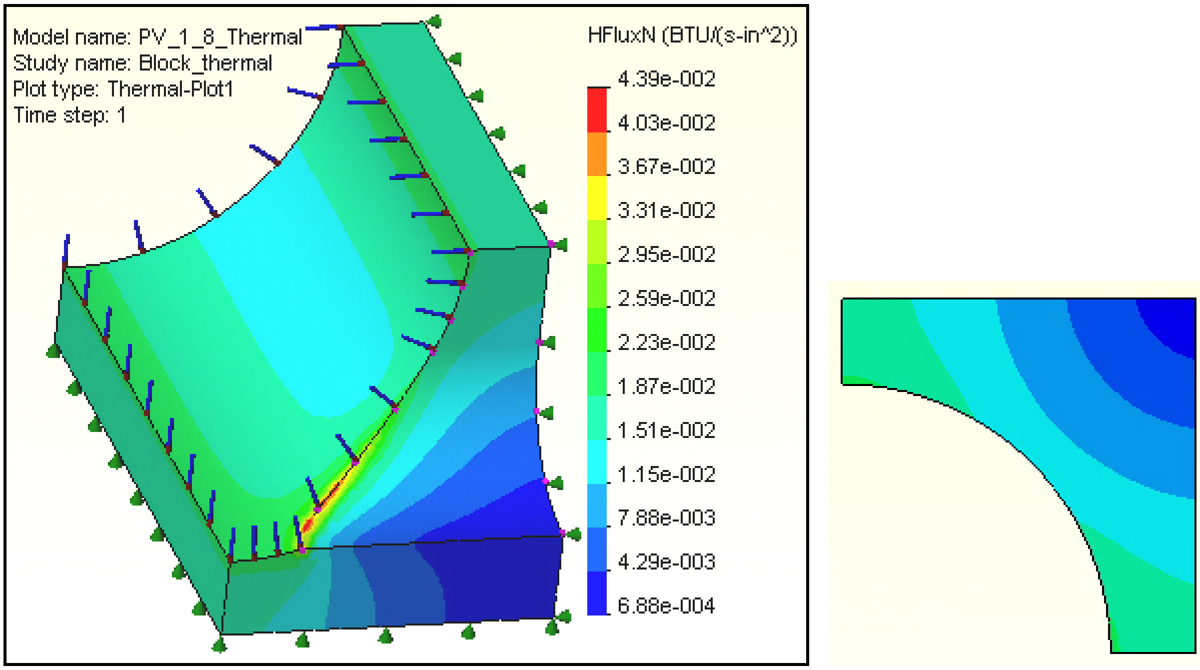


Figure 12-55 Magnitude of the heat flux peaks at the inner intersection, and top surface

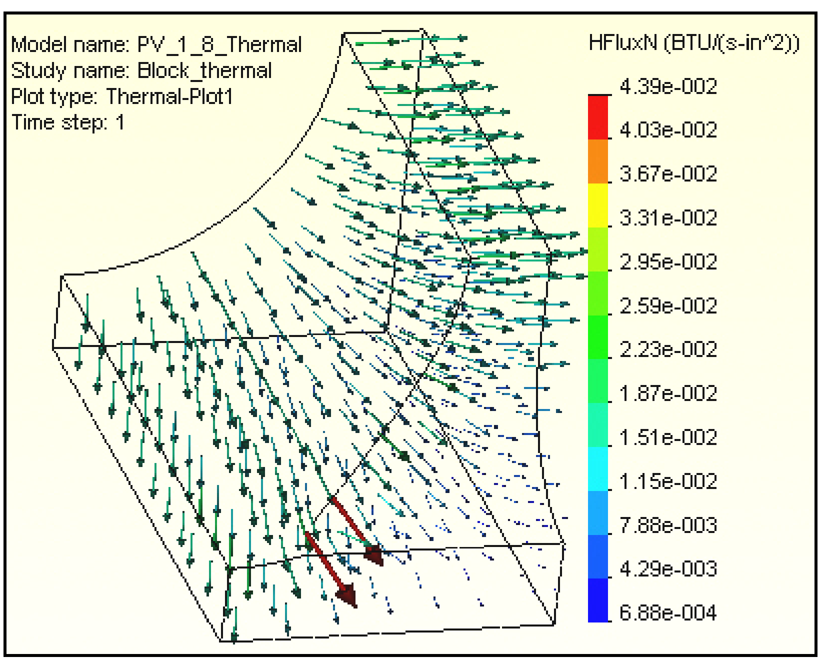


Figure 12-56 Wireframe view with heat flux vectors

The magnitude display of Figure 12-55 confirms the location of concern for future thermal stress studies. Complete that check by looking at the heat flux vectors:

1. Right click in the graphics area, **Edit Definition**.
2. Select **Vector, Line,** click **OK**.

The display in Figure 12-56 (above) yields a final verification of the concern about the intersection curve area having high temperature gradients.

12.12.11 Compare estimated and computed values

The estimated thin wall surface temperature of about 84.4 F agrees very well with the temperature contours seen in Figure 12-53. A plot of the estimated 1D temperature is in Figure 12-57. The approximate average radial heat flux of about 1.8e-2 BTU/s in^2 is also in good agreement with Figure 12-55 and Figure 12-56. A plot of estimated heat flux is given in Figure 12-58.

If you were not aware of the analytic approximation, or know how to derive it you could find a 1D (axisymmetric) finite element solution to estimate the expected results. The summary of such a solution is based on the typical linear element described in Akin [2] for approximating radially symmetric heat problems. A single element could give an analytic estimate of the surface temperature. Such an element has a constant heat flux, so several should actually be utilized in a 1D validation check.

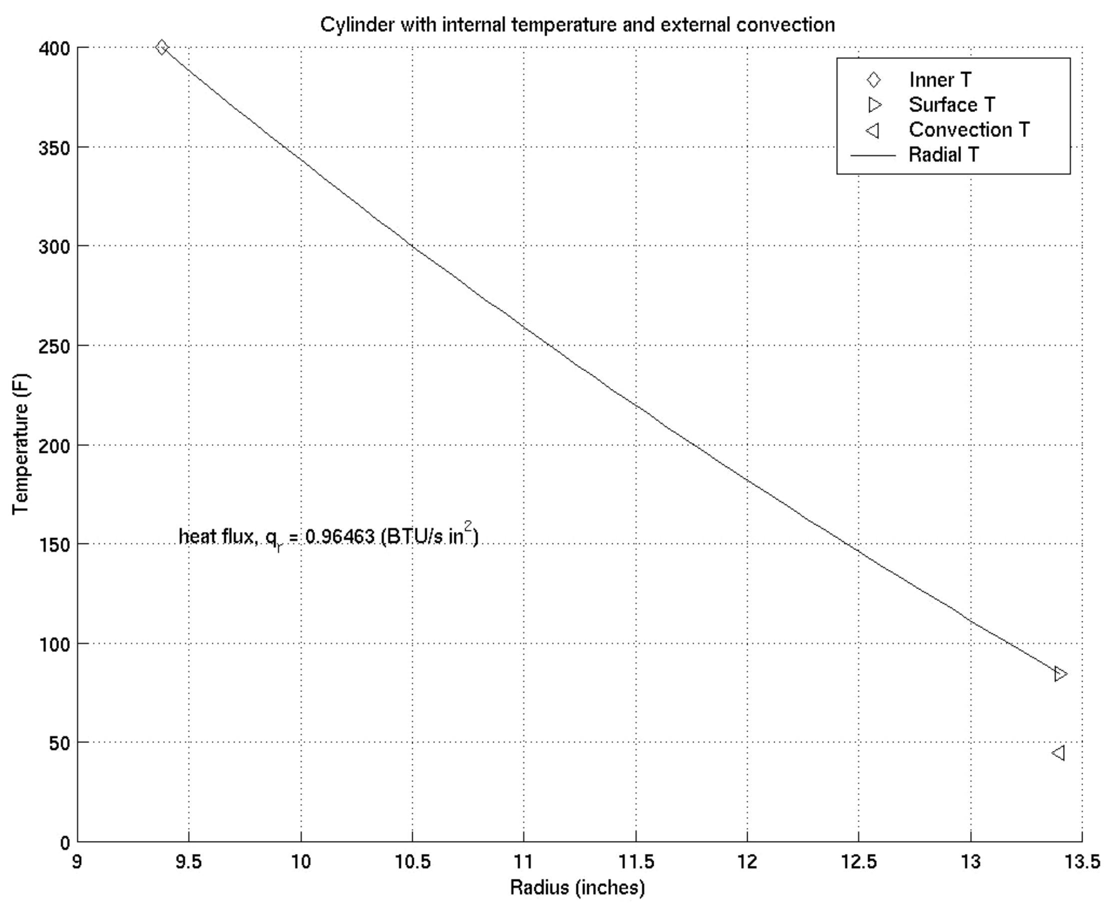


Figure 12-57 Typical analytic temperature estimate

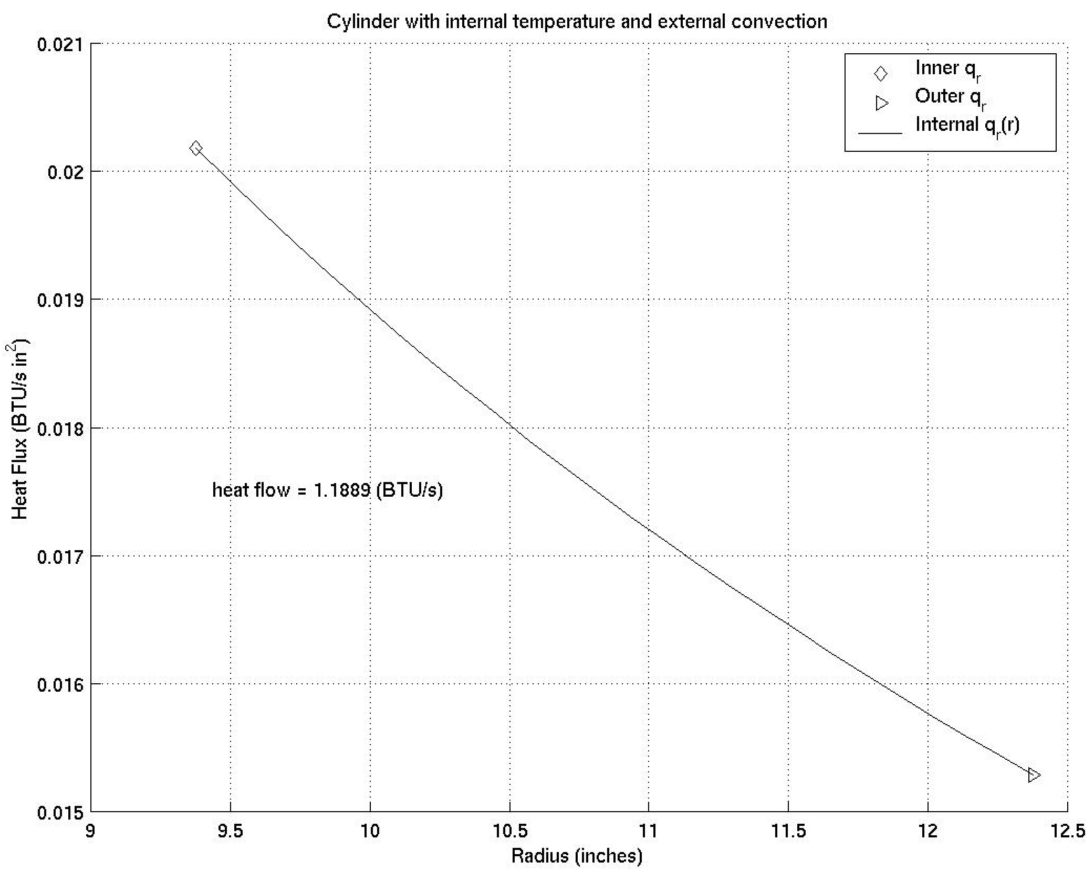


Figure 12-58 Typical analytic heat flux estimate

12.13 Three Material Thermal Study

12.13.1 Introduction

When you are learning a powerful system like SolidWorks and CosmosWorks it is wise to reproduce problems with known solutions to verify that you have properly mastered the interface. The purpose of this study is to demonstrate how to combine parts with different material properties in a thermal analysis. The basic outline is that you must create a SolidWorks part file for each single material part, combine them into a SolidWorks assembly, and mate the touching surfaces. Then you can open the assembly in CosmosWorks, specify the material properties of each part, bond surfaces where needed, apply loads and restraints and solve the equations. Usually you will also need to post-process the results for items of interest.

Here you will utilize a known 1D solution to verify your use of a 3D code. Then you should have no trouble mastering parts of arbitrary 3D geometry. The popular heat transfer text by Chapman [5] provides several analytic and numerical studies that can be used a validation exercises. One such application (Example 2.2, [5]) gives the analytic solution for steady temperature and heat flow through a flat wall made of three layers of bonded materials. The outside of the wall has natural convection to air at 35 C with a convection coefficient of h_1 = 30.0 W/m^2 C. The outer facing brick has a thermal conductivity of k_1 = 1.32 W/m C and a thickness of 10 cm. It bonds to a 15 cm layer of common brick with k_2 = 0.69 W/m C. It in turn is covered with plaster that is 1.25 cm thick having k_3 = 0.48 W/m C. Finally, the plaster convects to inside air at 22 C and has a convection coefficient of h_2 = 8 W/m^2 C. You want to know the inner and outer surface temperatures, and the heat flux (per unit area) through the wall.

In a 1D solution the temperature through a flat wall (without internal heat generation) will wary linearly with position in each region of constant conductivity, k. It will be continuous at the bonded material interface but the slope is discontinuous where k changes. Likewise, the heat flux should be constant and the same in each material.

You need to determine the two wall surface temperatures at the air, and the heat flow through the wall. At the end of this study the analytic 1D temperatures and head flux will be compared to the computed results.

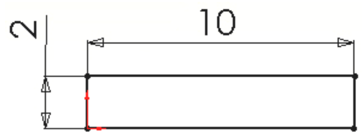
12.13.1.1 Face brick part

Since CosmosWorks defaults to 3D solids you will need to build three different solid parts, one for each segment of the wall. The respective lengths are specified above. The thickness and width of each brick shape (to define the wall area) needs to be the same. Their values can be arbitrary and are usually picked to give good aspect ratios for the elements. Here, the vertical height is assumed to be 2 cm, and the horizontal width is 1 cm. Start SolidWorks and begin by specifying the units to be utilized in building the first part:

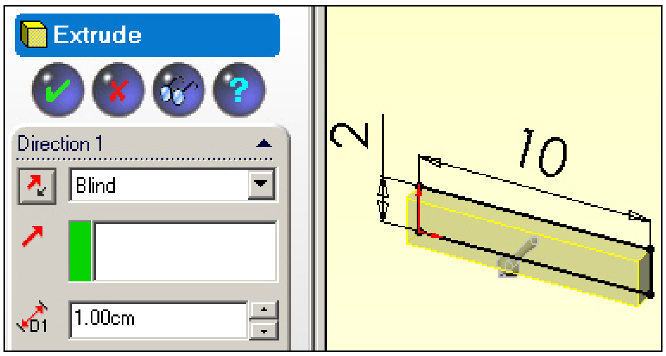
1. Use **Tools→Options→Document Properties→Units**.
2. In the **Units panel** check **CGS** (cm, gram, sec) as your **Unit system**.
3. If you wish, select feet as the optional **Dual units** for dimensions. These units' choices are seen in Figure 12-59.

The first part (facing brick) will simply be an extruded rectangle. Build it with:

1. Right click on **Front→Insert Sketch→Rectangle**. Start one corner at the origin, drag out the rectangle.
2. **Smart dimension** the length to 10 cm, and the (arbitrary) height to 2cm.



3. Pick **Extrude Bass/Boss** icon to get the **Extrude panel**.
4. Use a blind extrusion in **Direction 1** of (arbitrary) width of 1 cm, click **OK**.



5. **File→Save as** filename Face_Brick.sldprt.

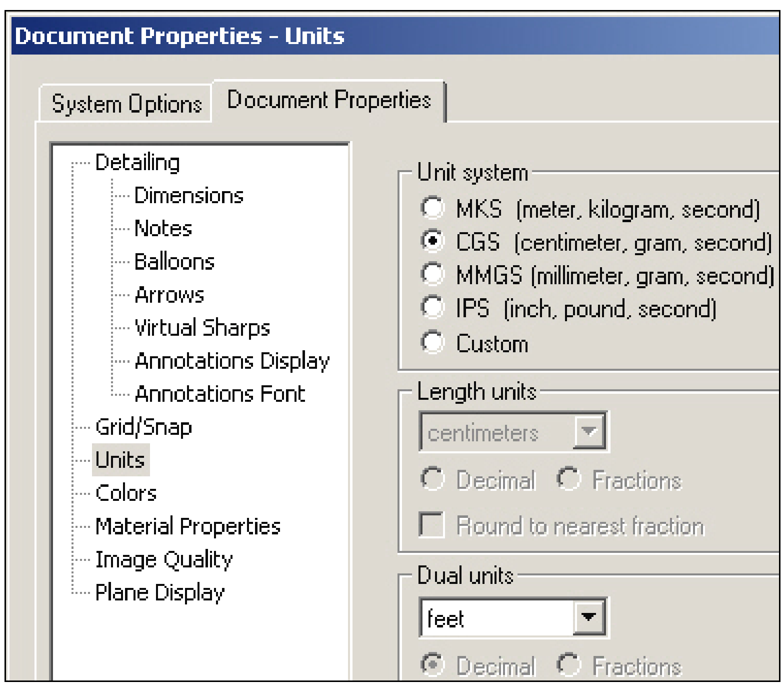


Figure 12-59 Selecting the units for the first part

12.13.1.2 Common brick and plaster parts

Follow a similar process to create the Common_Brick part, and the Plaster part, as summarized in Figure 12-60. Remember to start with **Tools→Options→Document Properties→ Units.**

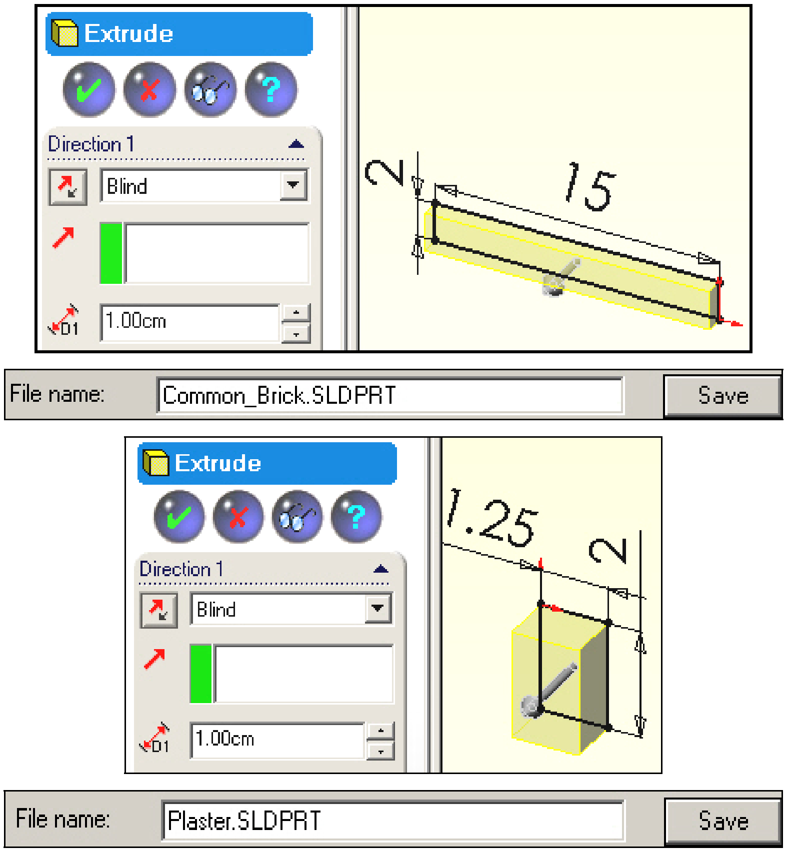


Figure 12-60 Forming the next two wall layers

12.13.2 Building and mating the assembly

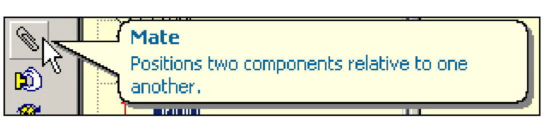
Next you must move all the parts of different materials into an assembly and mate them as needed:

1. **New→Assembly→OK**. Right click on **Window→Tile Horizontally** to see the original three parts and the assembly insertion progress (see Figure 12-61).
2. In the **Insert Component panel** highlight the Face_Brick part to start the assembly and to locate all other parts relative to it.
3. Click on the Common_Brick part and drag it into the assembly.
4. Finally, click on the Plaster part and drag it into the assembly.
5. Click on the **Assembly name bar→Window→Cascade** to visually check the parts, as in Figure 12-62.
6. Use the **Move Component** icon to grasp the last two parts and move them so the surfaces intended for mating are close to each other, click **OK**.

12.13.2.1 Mate the touching surfaces

For each of the three material interfaces you can require them to by making two pairs of perpendicular edges be coincident:

1. Select the paper clip **Mate** icon (just above the **Move Component** icon).



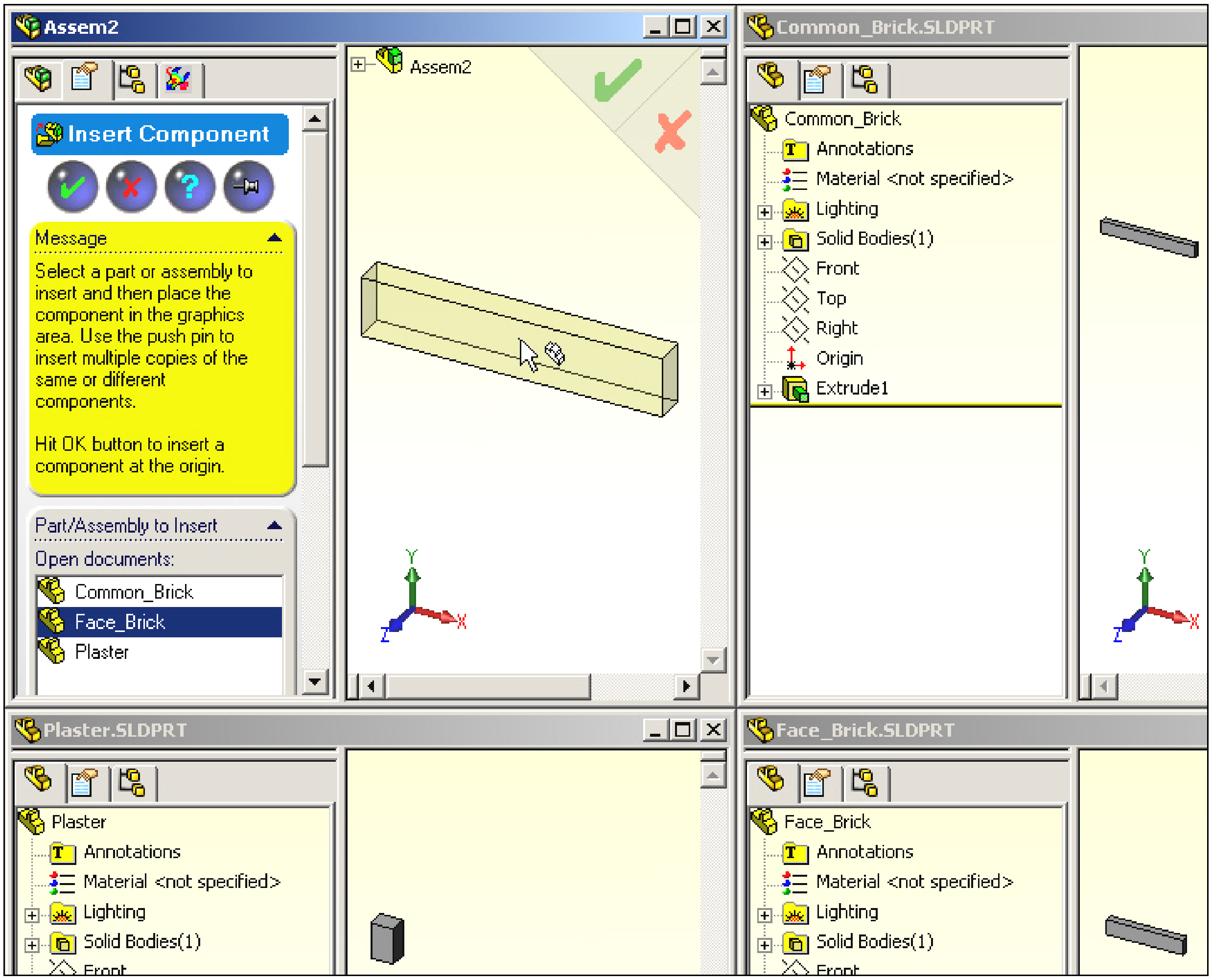


Figure 12-61 Starting the assembly process

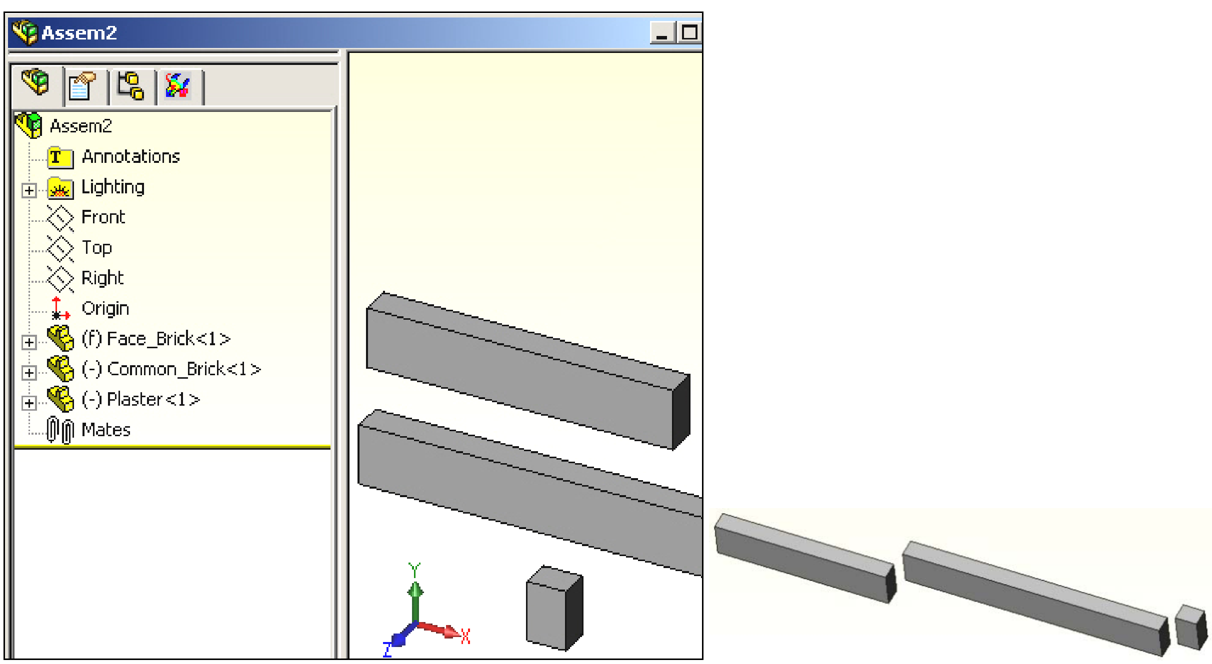
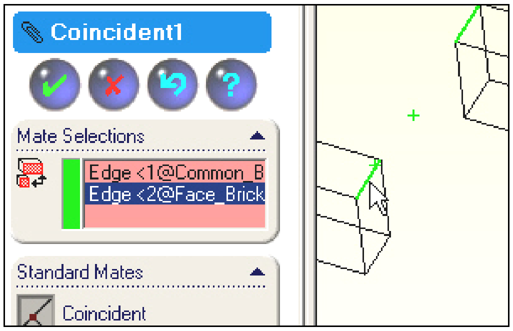
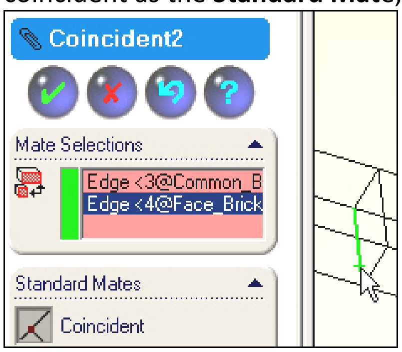


Figure 12-62 Check and rearrange the assembly of three parts

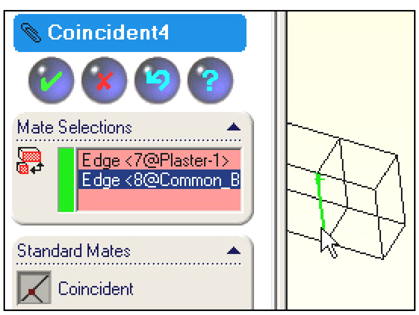
2. Pick the inside (1 cm) top horizontal edge of the face brick and the corresponding top outer edge of the common brick.
3. In the **Coincident panel** select coincident as the **Standard Mate**, click **OK**.

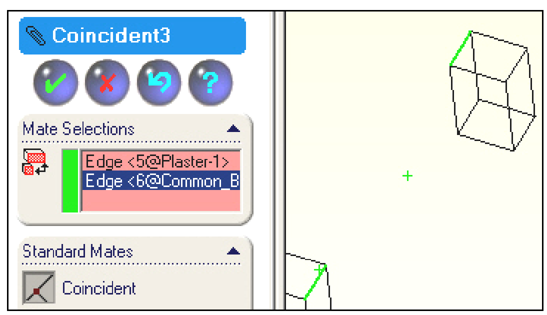


1. Use the vertical front edges, intersecting the above two edges, and in the **Coincident panel** select coincident as the **Standard Mat**e, click **OK**.

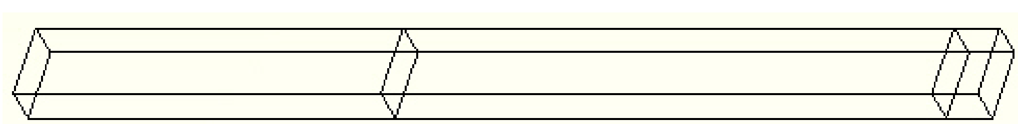


2. Use the same pair of coincident mates at the common brick-plaster interface.





3. Verify the mated assembly of the three different materials.



4. Use **File→Save as** to create an **Assembly name** of House_wall, click **OK**.

Now you have built a solid assembly which is equivalent to the 1D verification problem. This was clearly a lot more work (just to get started) than the full 1D analytic solution. But after mastering these concepts the finite element approach readily extends to arbitrary three-dimensional domains, while the corresponding analytic approaches bog down and become very difficult, and impractical.

12.13.3 Beginning the CosmosWorks Model

Pass the completed and saved assembly to the CosmosWorks analysis system:

1. Pick the **CosmosWorks icon** (fourth from the left).

2. Right click on the **Assembly name→Study** to get the **Study panel**.

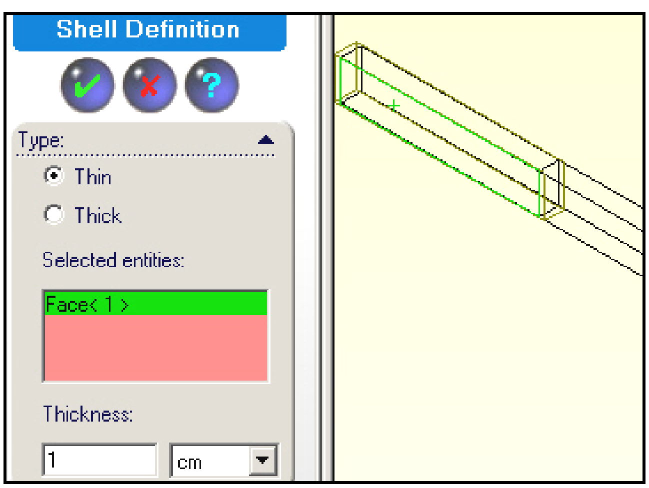
3. There, enter a **Study name**, pick thermal as the Analysis type, and select a shell as the **Mesh type** (try repeating this later with a solid mesh), click **OK**.

12.13.3.1 Select the shells and assign materials

Had you picked a solid mesh type you could next directly define the three sets of material properties. For the chosen shell type of mesh, using surfaces, you must first select the three (connected) surfaces to be meshed with shells. Since the problem is really 1D you could select any of the four sets of three connected faces running through the wall thickness. Here the front face of the assembly will be employed. (Now you are using a 2D model to solve the 1D problem). Start with the face brick:

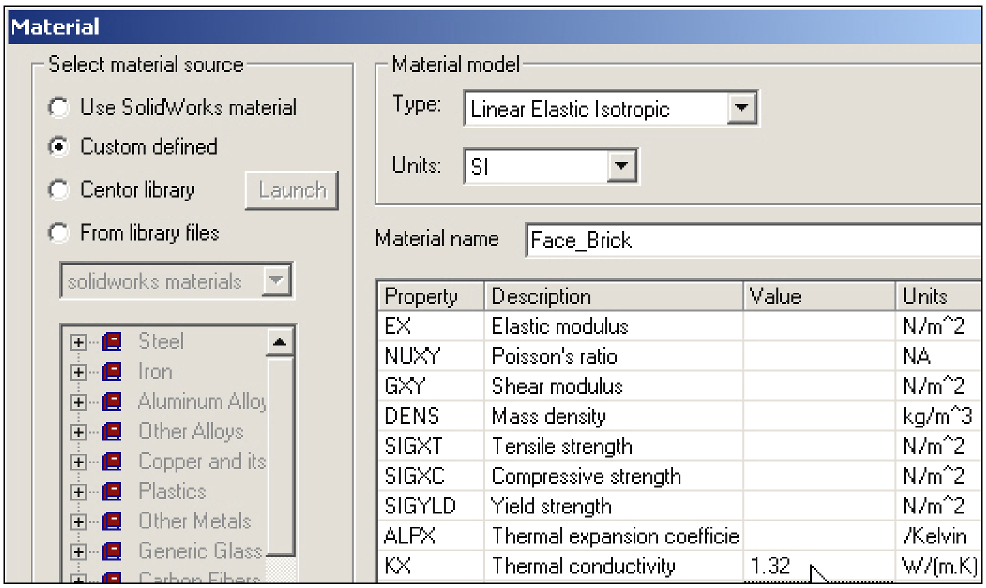
1. Right click on **Shells→Define by Selected Surfaces** to get the **Shell Definition panel**.

2. In the **Shell Definition panel**, set the **Type** to thin, pick the model front surface of the facing brick as the **Selected entity.**



3. Set the **Thickness** to 1 cm, click **OK**. Especially note this step, since the extrusion thickness would automatically be assigned in a mid-surface shell. However, that option is not available here. (What would happen if you inconsistently type in 2 cm here, but use 1 cm for the next two sets of shells?)
4. In the manager panel **Shell 1→Apply/Edit materials** to get the **Material panel**.
5. 

6. Check **Custom defined**, enter Face_Brick as the **Material name**, type the thermal conductivity value (**KX**) as 1.32 W/m^2 C, click **OK**.



7. Repeat this process for the next two materials (as in Figure 12-63 and Figure 12-64).

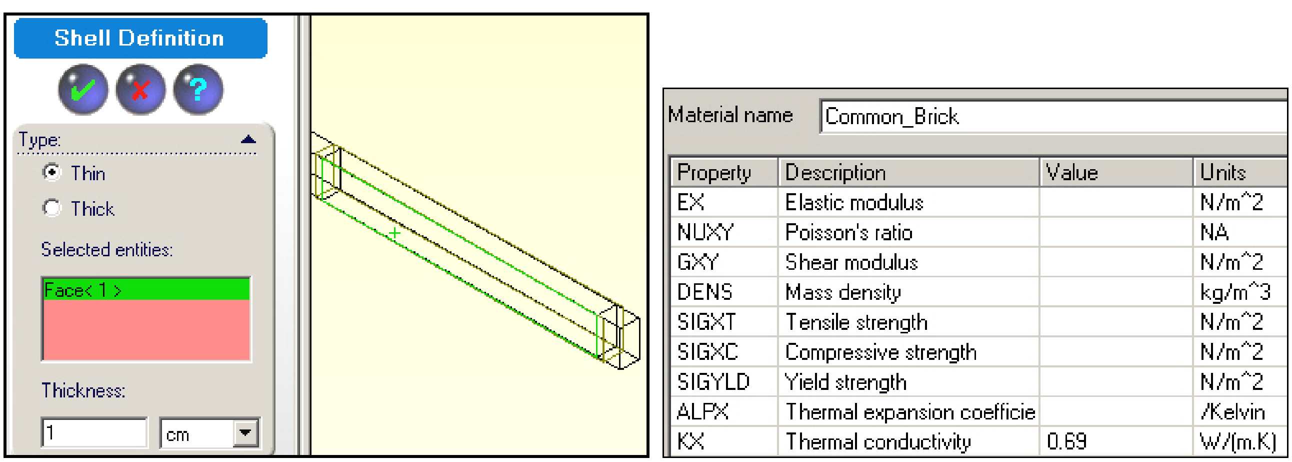
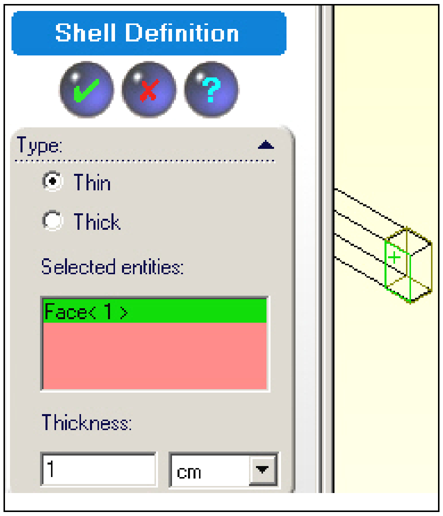


Figure 12-63   Set the shell surface, thickness, and property for the common brick



Material name: Plaster

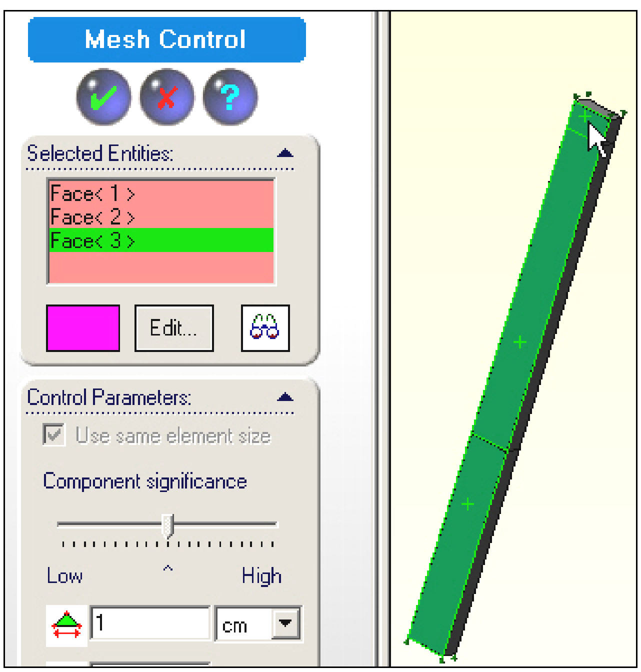
Property	Description	Value	Units
EX	Elastic modulus		N/m^2
NUXY	Poisson's ratio		NA
GXY	Shear modulus		N/m^2
DENS	Mass density		kg/m^3
SIGXT	Tensile strength		N/m^2
SIGXC	Compressive strength		N/m^2
SIGYLD	Yield strength		N/m^2
ALPX	Thermal expansion coefficie		/Kelvin
KX	Thermal conductivity	0.48	W/(m.K)

Figure 12-64 Set the shell surface, thickness and property for the plaster

12.13.3.2 Control and create the mesh

Knowing the analytic result, each layer of the wall could be solved exactly with one rectangular element or two triangular elements (forming the rectangle). However, you will generate a general 2D (or 3D) mesh with a few coarse elements to avoid long skinny (high aspect ratio) elements which usually adversely affect the accuracy:

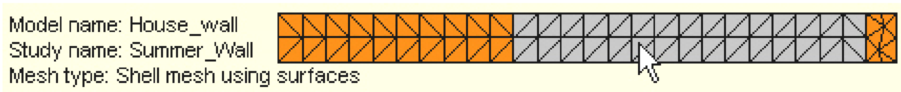
1. In the manager panel, right click on **Mesh→Apply Control**.
2. In the **Mesh Control panel**, select the three faces (or three bodies if 3D) as the **Selected Entities** and set **Control Parameter** (element size) to a relatively large number like 1 cm, click **OK**.



3. **Mesh→Create→Mesh panel** enter similar coarse element sizes, click **OK**.

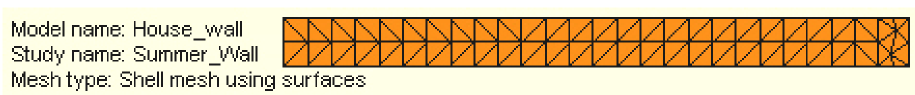
After the mesh generation phase is created you need to visually verify the mesh (use **Mesh→Show mesh**). Mainly you need to look for bad aspect ratios. For shell elements assure that the color coded top (or bottom) sides of all elements (i.e., their normal vectors) are the same. (That is very important in shell bending structural models.). Here, they should be all gray or orange (bottom side) color coded. Note that the common brick region of elements needs to be "flipped" To fix that:

1. Place the cursor on the gray region of the shell mesh, **control click** the region.



2. **Mesh→Flip shell elements**, click **OK**.

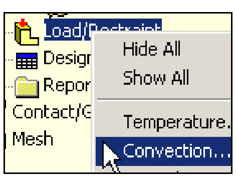
3. Double check the mesh.



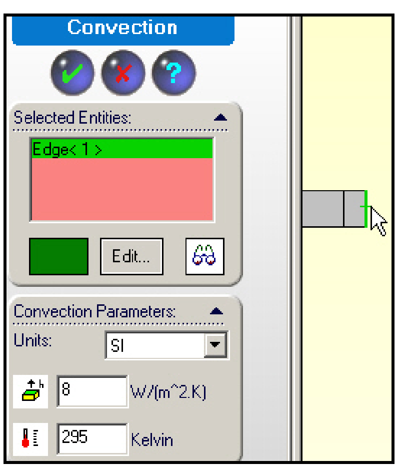
12.13.4 Impose convection loadings

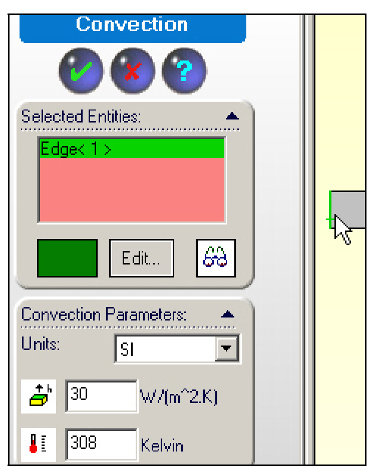
Apply the two loads (at the two face convection surfaces) from the manager panel:

1. **Load/Restraints→Convection** to open the **Convection panel**.



2. At the inside wall pick the edge (or face in 3D) as the **Selected Entity**. Set **Convection Parameters** to **h** = 8 W/m^2 C, and 295 K as the air **temperature,** click **OK**. Note the inconsistent need to convert the air to degrees Kelvin. While it is an easy external unit's conversion, it introduces the possibility of a user error (garbage in, garbage out).





3. Repeat step 2 for the outside convection edge (face in 3D), as shown above.

12.13.5 Insulated segments

You have taken the wall segments to be horizontal. Thus, you have assumed that the section through the wall to be insulated on its top and bottom surfaces. This does not require any user action because that is the "*natural boundary condition*" in finite element analysis if no other boundary condition is applied to a surface.

12.13.6 Essential boundary condition

A thermal analysis usually requires an "*essential boundary condition*" which is specifying at least one known temperature point (via **Load/Restraints→Temperature**). Here you have done that indirectly because of the nature of any convection boundary condition. Thus, no additional action is needed to have an algebraic equation system that can be solved for the unknown temperatures.

12.13.7 Solve for the temperatures

Invoke the equation solver in the manager: right click **Assembly name→Run**. If you wish, you can rename the mesh regions and/or loads and restraints so your study will mean more to you if you have to come back to it in a few weeks or years. The results of such renaming can be seen in Figure 12-65.

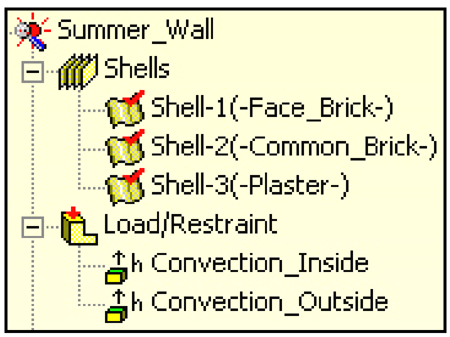


Figure 12-65 Rename the material regions and convection loads

12.13.8 Post-processing

After all the temperatures are obtained you usually want to post-process them for your analysis study report. You also want to check other items computed from the temperatures. Remember, in any finite element analysis the nodal temperatures are the most accurate quantities. The temperature gradients and heat flux are the least accurate. You should always check the temperatures and heat flux since one may depend simply on the part shape while the other also involves the material properties.

12.13.8.1 Temperature results

As expected, the temperature varies linearly through each material layer and has a change of slope at the interfaces. That is a little hard to see in a standard contour plot (Figure 12-66). A slightly more informative set of information is to probe for the convection surface (and possibly the interface temperatures):

1. Right click in the graphics area and select **Probe**.
2. Pick points at the outside air (they are about 33.9 C).
3. Pick points at the inside air (they are about 25.3 C), as in Figure 12-67.

Note that these values also appear in a **Probe panel** along with the point position and the part name where the probe occurred.

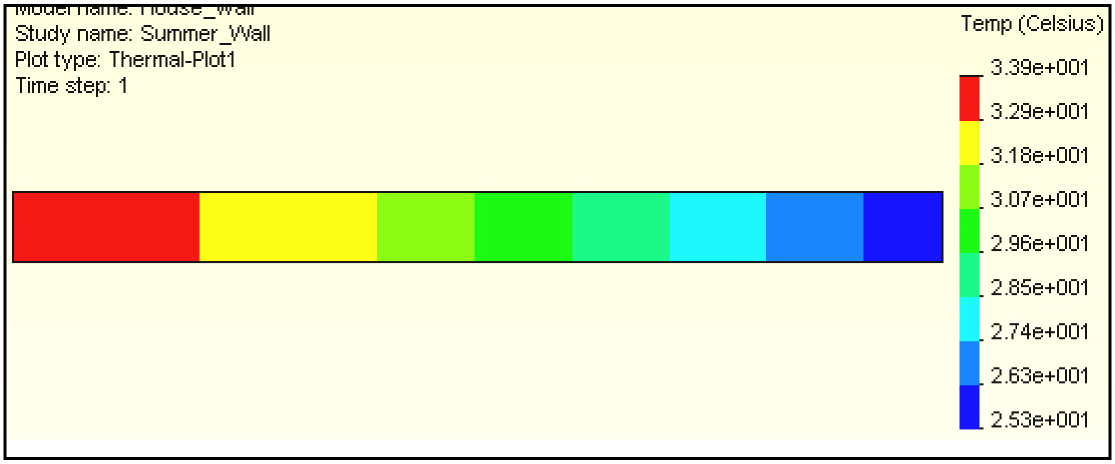
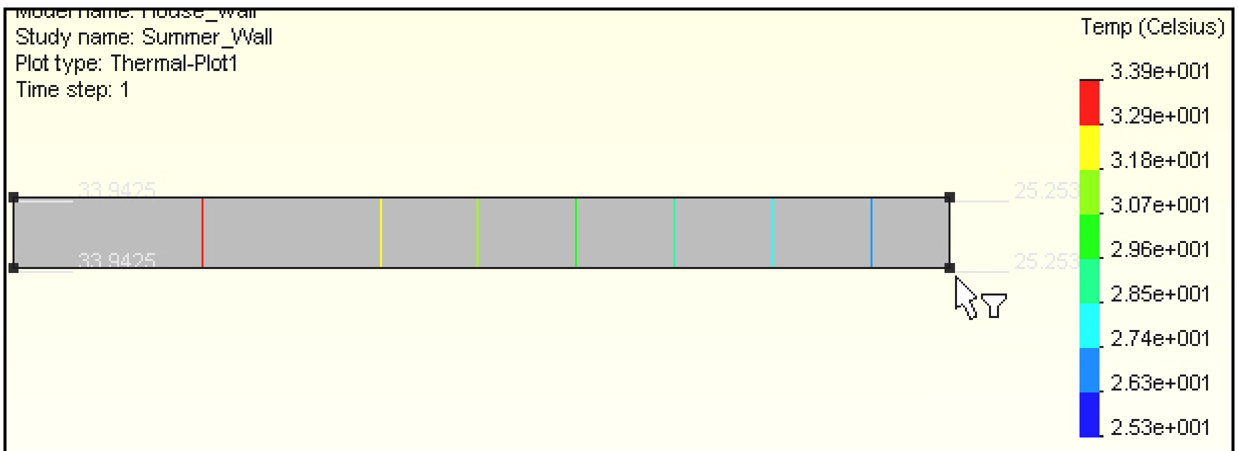


Figure 12-66 Linear temperature drops through each material



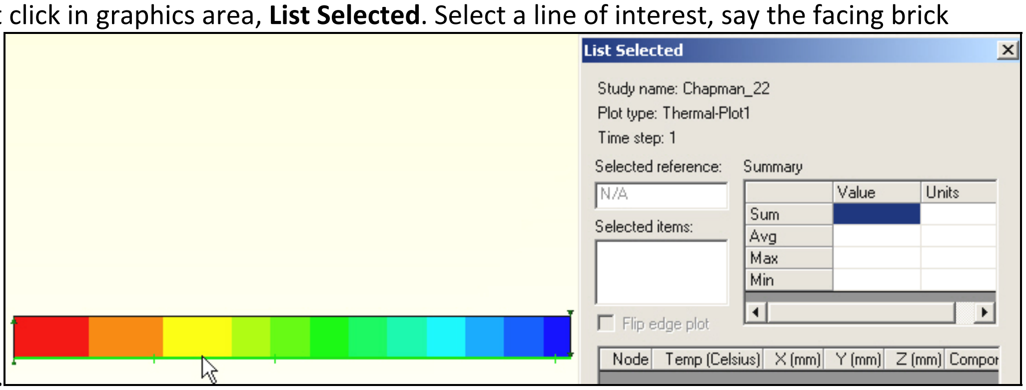
Node	Temp (Celsius)	X (mm)	Y (mm)	Z (mm)	Components
175	3.39e+001	-38.504	1.3343	29.412	Face_Brick-1
156	3.39e+001	-38.504	21.334	29.412	Face_Brick-1
257	2.53e+001	224	1.3343	29.412	Plaster-1
259	2.53e+001	224	21.334	29.412	Plaster-1

Figure 12-67 Probing the temperature extremes

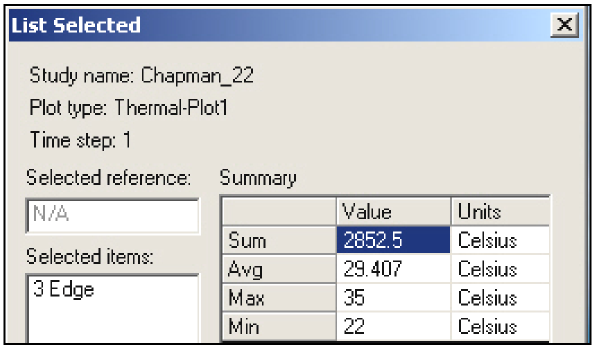
12.13.8.2 Graphing Temperatures

You can produce graphs or lists of the temperature values on selected lines or surfaces with:

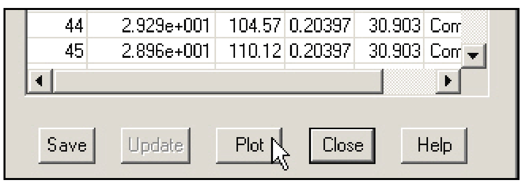
1. Right click in graphics area, **List Selected**. Select a line of interest, say the facing brick base.

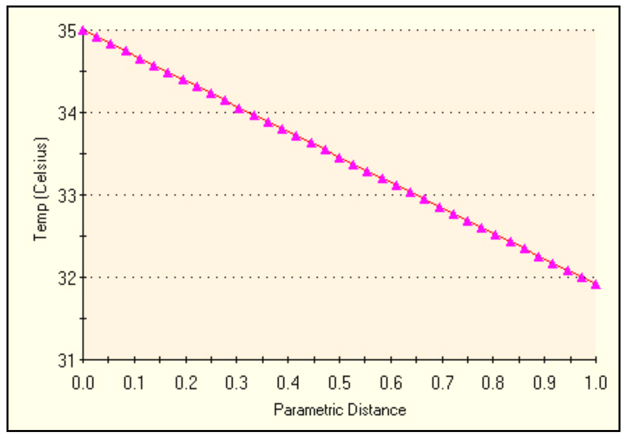


2. In the **List Selected panel** click on **Update**. The top on that panel will give the nodal summary for the line. Each nodal value is listed in the panel.

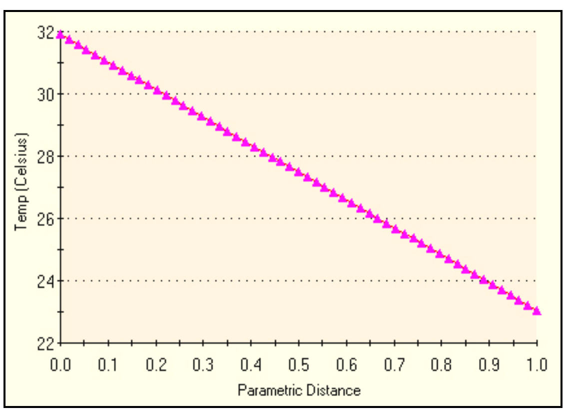


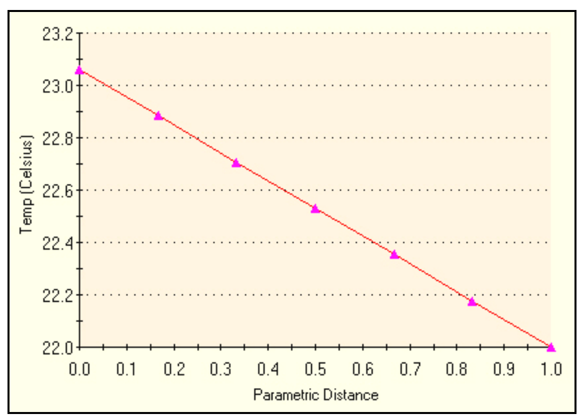
3. In the **List Selected panel** click on **Plot** to get a graph of temperatures vs. relative position on the line along the face brick (from 35 to about 31.9 C).





4. Pick the base of the interior brick and the plaster to graph their temperatures, from about 31.9 to about 23.1 to 22 C.



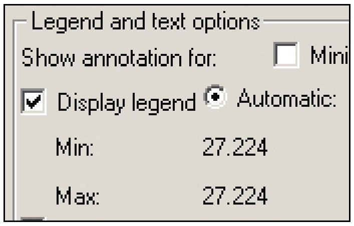


Note that at the material interfaces there is no discontinuity in the temperatures. That is because the automatic interface condition in a finite element analysis is an ideal one, as shown on the left of Figure 12-1.

12.13.8.3 Heat flux results

The heat flux should be constant. The default contour plot looks variable until you notice that the numbers are the same except for some round-off errors. Check the plot ranges:

1. Right click in the graphics area, **Edit Definition→Settings**
2. Examine the **Legend and text options** for the automatic maximum and minimum contour range levels. Since they are both the same (27.2 W/m^2) a contour plot is not needed.

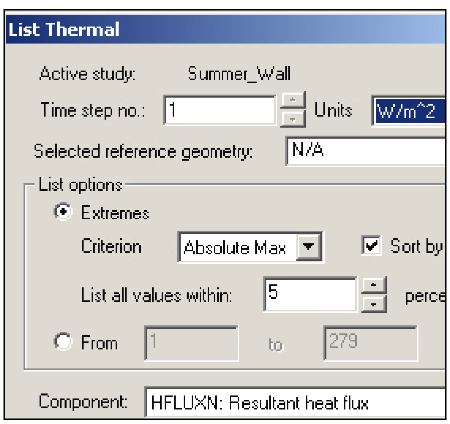


3. To double check this, click in the graphics area and select **Probe** .
4. Picking nodes on the plaster and facing brick verifies the expected constant heat flux result.

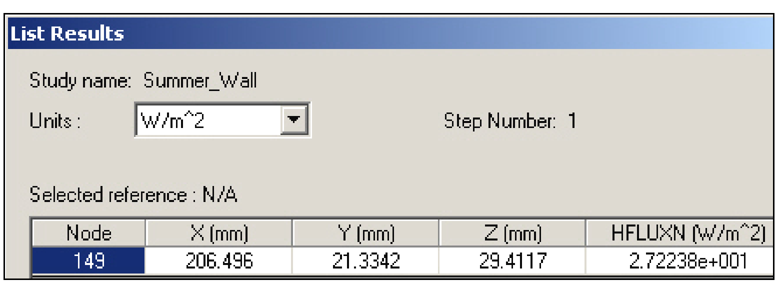
Node	HFluxN (W/m^2)	X (mm)	Y (mm)	Z (mm)	Components
259	2.72e+001	224	21.334	29.412	Plaster-1
257	2.72e+001	224	1.3343	29.412	Plaster-1
156	2.72e+001	-38.504	21.334	29.412	Face_Brick-1
175	2.72e+001	-38.504	1.3343	29.412	Face_Brick-1

You can also create a similar list of the maximum nodal values found in the mesh. The list can be saved to place in your analysis report. Use

1. Under the study name, **Thermal→List→List Thermal panel**.
2. Set **Units** to W/m^2, **Option** extreme absolute max, and resultant heat flux (HFLUXN) as the **Component**, click **OK**.



3. The **List Results panel** will appear.



12.13.9 Validation

These computed results agree very well with the analytic solution [5]. That gives the inside plaster surface temperature to be 25.4 C (77.7 F), and the constant heat flux (per unit area) to be 27.2 W/m^2 (8.63 Btu/h ft^2). This problem was also solved with a 1-D finite element solution using three conduction elements and two convection elements. The temperature plot is shown in Figure 12-68.

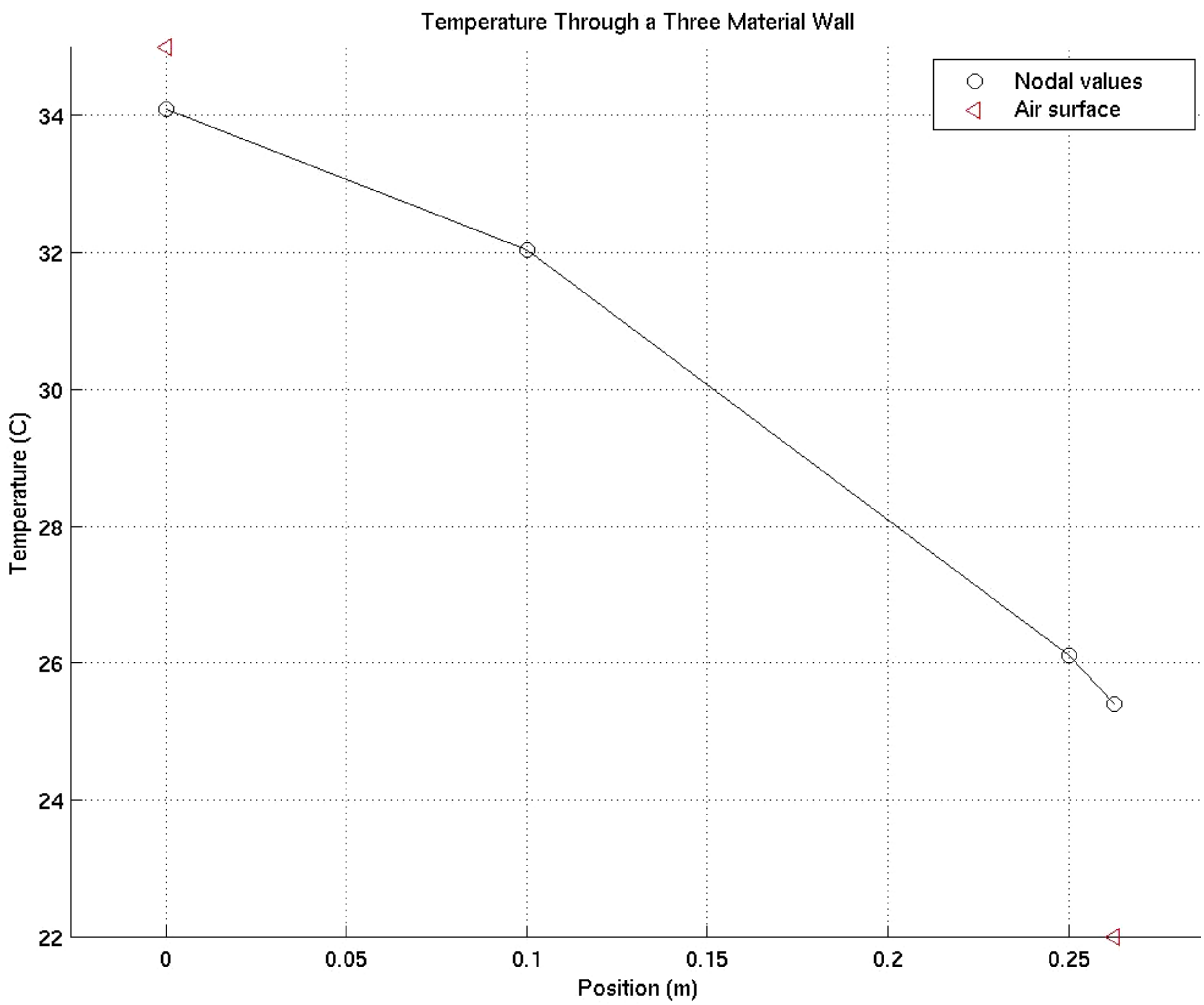


Figure 12-68 One-dimensional validation plot

Observe that at the left and right ends there is a discontinuity between the air and solid temperatures. That is always true for convection, unless the convection coefficient is so high that acts as a specified temperature restraint, (which is usually a user input error). The interior temperatures are everywhere continuous for ideal material interfaces.

13 Axisymmetric Solid Analysis

13.1 Cylinder with given temperature and convection

13.1.1 Introduction

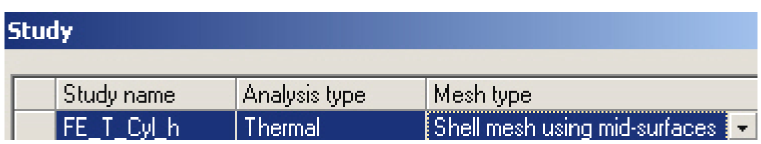
Consider a thick walled cylinder with a given internal temperature and convection at its external radius. The analysis of heat transfer in a thick wall infinite cylinder is a one-dimensional problem, and the analytic solution is known [1]. The cylinder has inner and outer radii of 9.375 and 13.40 inches, respectively, and is made of a material with a thermal conductivity of 1.736e-4 BTU/in-s-F. The inner and outer temperatures are 500 and 45 degree F, respectively, with and outer convection coefficient of 2.89e-4 BTU/F-s-in^2. The goal is to determine the inner temperature distribution, and the required heat flow through the wall. This problem provides a chance to verify ones knowledge of CosmosWorks, and to illustrate some optional features. To access some optional features you must have a part open, or prepare to open the first part with **File→New→Part→OK**.

13.1.2 CosmosWorks thermal study

Next, enter CosmosWorks by selecting its managed icon, . In CosmosWorks:

1. **Right click** on the **Part name→Study** to open the **Study panel**.

2. Insert a **Study name** (say FE_T_Cyl_h), select **Thermal** from the pull down **Analysis type** list, and pull down the **Mesh type** list options.

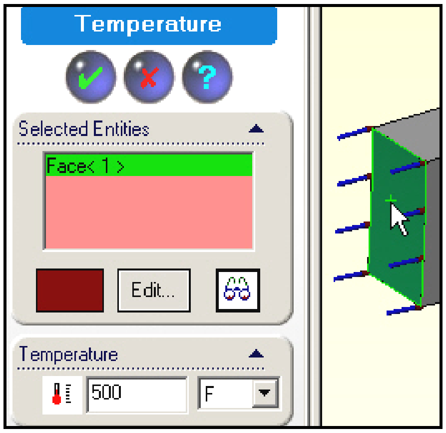


3. Any of the three mesh types (solid or two shell options) can be used. For a faster 2D solution select **Shell mesh using mid-surfaces**. (At this point a new user will not know which mid-plane will be selected: an axial slice (yes) or a radial slice.)

13.1.3 Boundary conditions

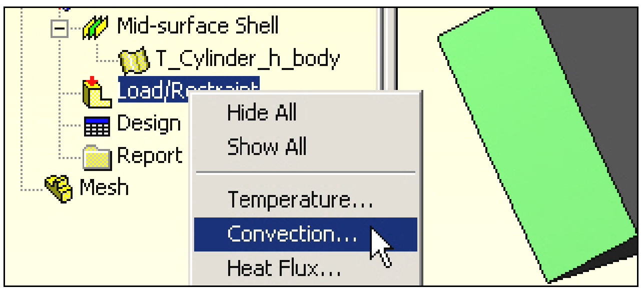
The above choice of a **Shell mesh using mid-surfaces** defines the methods used to apply the boundary conditions. Since part faces are used to locate the mid-surface, the boundary conditions **must be specified on the faces of the part**. By way of comparison, if the alternate shell mesh by defined surfaces had been selected the boundary conditions would have to be specified on the edges of the faces picked to define the mesh. Begin the mid-surface mesh boundary conditions with the inner cylindrical (red) face:

1. In the CosmosWorks manager right click on **Load/Restraints→Temperature**.
2. When the **Temperature panel** appears **right click** on the inner face to insert it as the **Selected Entities**.
3. Under **Temperature** set the **units** as degrees **F**ahrenheit and insert a **value** of 500.
4. Hit **Preview** (eyeglass icon) to verify your assignment, click **OK**.

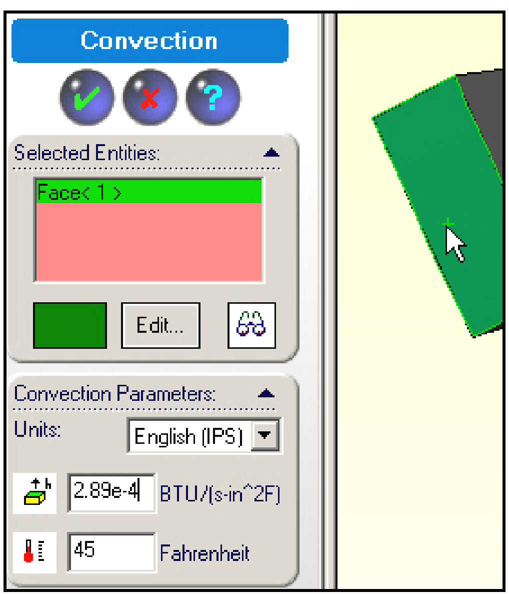


Next apply the convection conditions of the outer cylindrical (light green) face:

1. In the CosmosWorks manager right click on **Load/Restraints→Convection**.



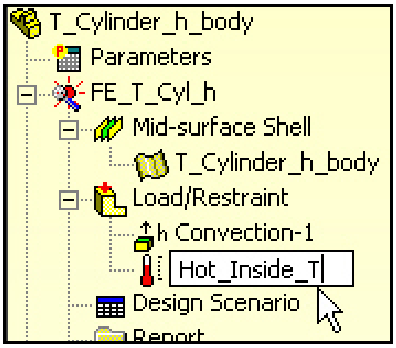
2. When the **Temperature panel** appears rotate the part until you can see the light green face.
3. **Right click** on the outer face to insert it as the **Selected Entities**.
4. Under **Convection Parameters** select **English units**, set the temperature to 45 **F**, and the surface free convection coefficient (**h**) to 2.89e-4 BTU/F-s-in^2.

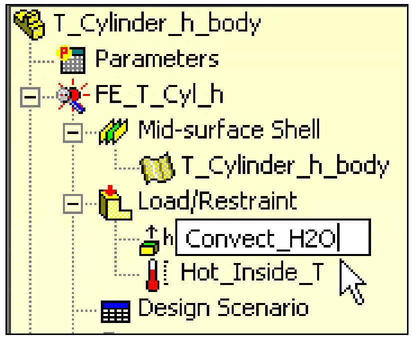


5. Click **Preview,** and then **OK**.

Another very useful option is the ability to **assign names** to any item in the **construction tree**. This helps you remember your thought process when you need to come back at a later time to review your analysis or design. (For legal purposes many engineering designs must be stored for seven years typically.) In the manager panel:

1. ***Slow* double click** on **Temperature-1** under Load/Restraint.
2. Enter a descriptive term, say Hot_Inside_T.
3. Likewise, change **Convection-1** to Convect_H2O.

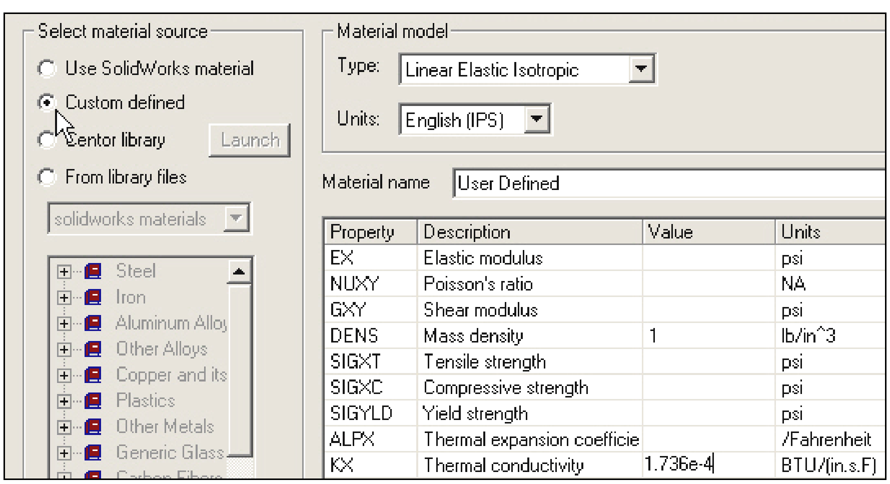




13.2 Define the material

The homogeneous (location independent) conducting material is not in the standard materials library, so its properties must be user defined:

1. In the **manager tree**, under **Mid-surface Shell**, right click on the **Part name→Apply/Edit Material** to open the **Material panel**.
2. In the **Material panel**, **Source→Custom defined**, **Units→English**, **Type→Linear Elastic Isotropic** (direction independent).
3. Assign a **Material name**, say User Defined.
4. In the **Thermal conductivity** row enter a value of 1.736e-4 BTU/in-s-F. That should be sufficient for this study. However, CosmosWorks also demands the value of the density in case you do a transient study. Since the density is unknown enter a fake value, say unity.

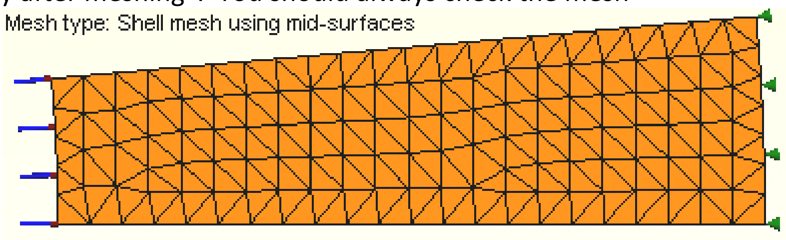


Here the material property is known to four significant figures. That is high for experimental material property measurements. To be consistent, the results of the analysis should not be reported with a larger number of significant digits (although they frequently are).

13.3 Meshing

For this 1D problem only a few elements are needed in the radial direction. Thus, the default mesh will be generated:

1. In the **manager tree**, right click **Mesh→Create** to open the **Mesh panel**.
2. In the **Mesh panel** accept the default element size and transition controls, click **OK**. Do not check “Run study after meshing”. You should always check the mesh first.



3. Visually **inspect** the mesh. Having about 20 elements in the radial direction should be fine. The solution in the circumferential direction should be constant, so the number of elements in the second does not matter (but they do cost). **Save** the CosmosWorks files.

13.4 Temperature solution

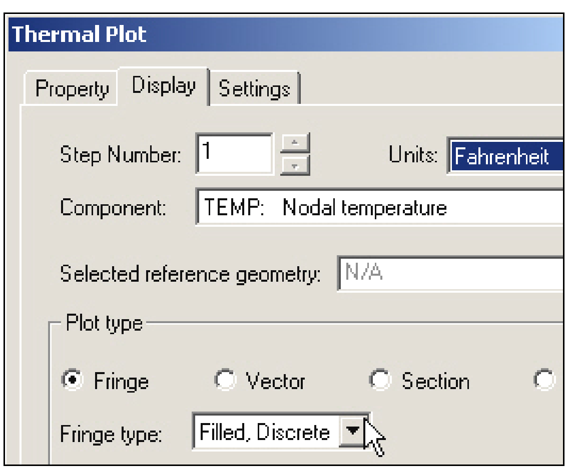
The thermal analysis is ready for solution. Right click on the **Study name→Run**. After the equation solver reports a successful calculation the post-processing results can be reviewed and plotted.

13.5 Post processing

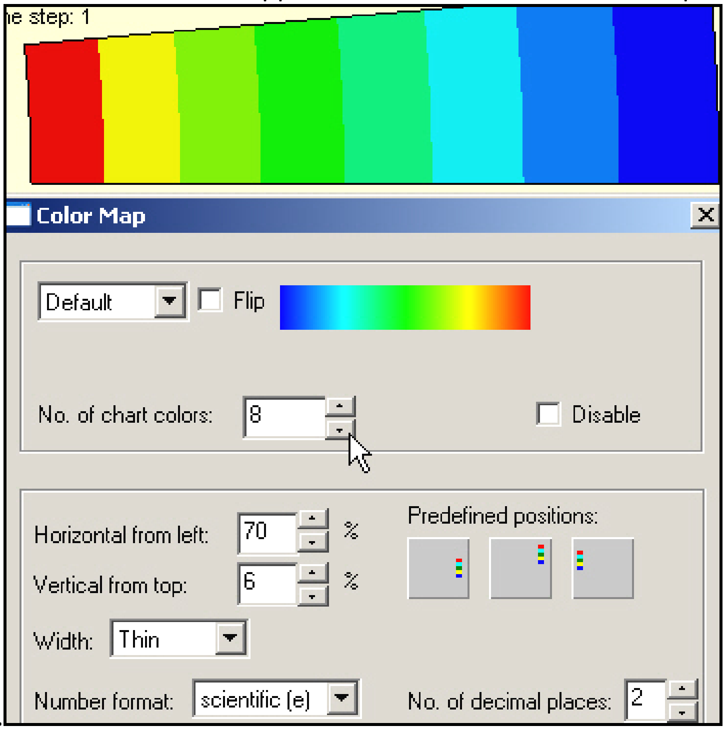
13.5.1.1 Temperatures

Begin the study review by examining the temperature distribution:

1. In the **Cosmos manager tree** click on **Temperature** and the double click **Plot-1**. The default plot is a continuous color contour.
2. For an alternate format right click in the graphics area and pick **Edit Definition**. In the **Thermal Plot** panel pick a **Filled, Discrete** for the **Fringe Type**, click **OK**.



3. The discrete contours appear. The convection surface temperature is about 96

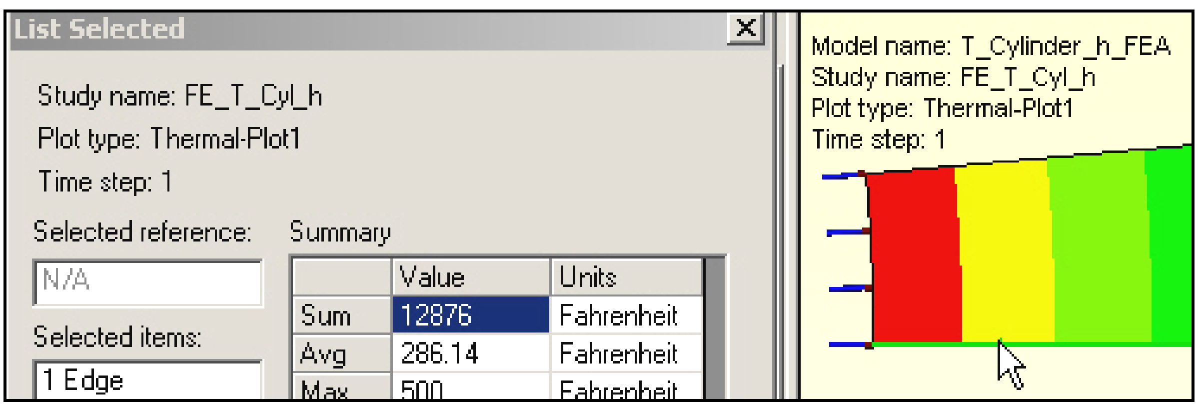


F.

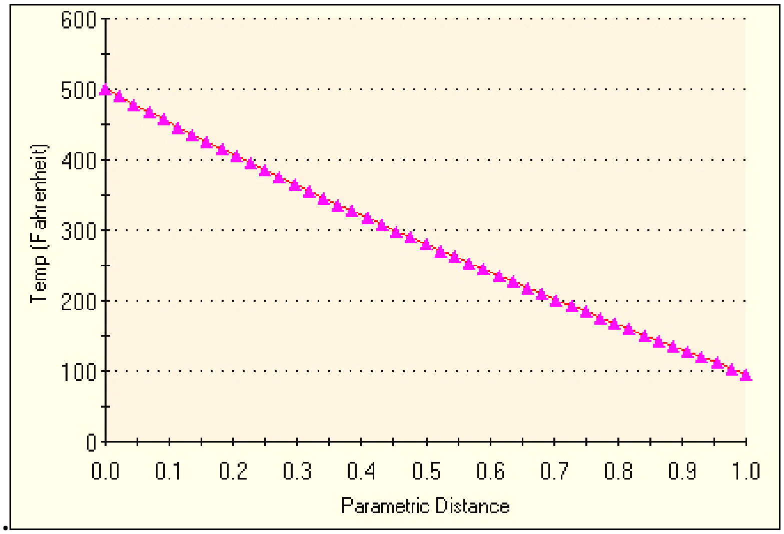
4.

Graphs can provide more detail in selected regions. Graph the radial temperature first:

1. Right click in the graphics window, pick **List Selected**.
2. Select the **bottom edge**, click **Update** to see max, min, and average values



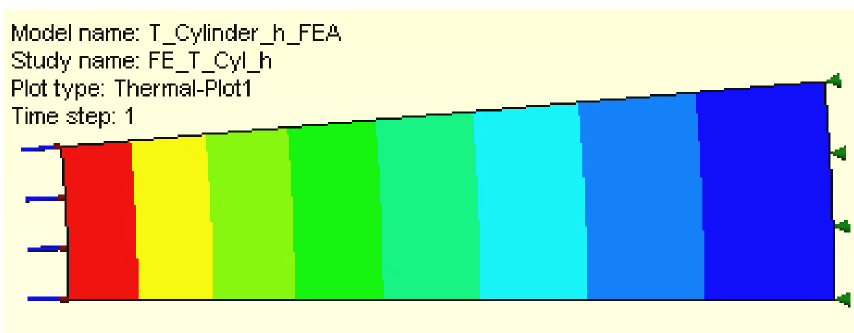
3. In **List Selected panel**, pick **Plot** to open the **Edge Plot**

4. 

The graph agrees very well with the logarithmic analytic solution. A reasonable estimate could have been obtained with a single element hand solution to help validate the temperature result.

13.5.1.2 Heat flux

The heat flux is a vector quantity obtained from the scalar temperature. In this case it must be in the radial direction (plot the vector form to see that) so just the values are shown here in Figure 13-1.



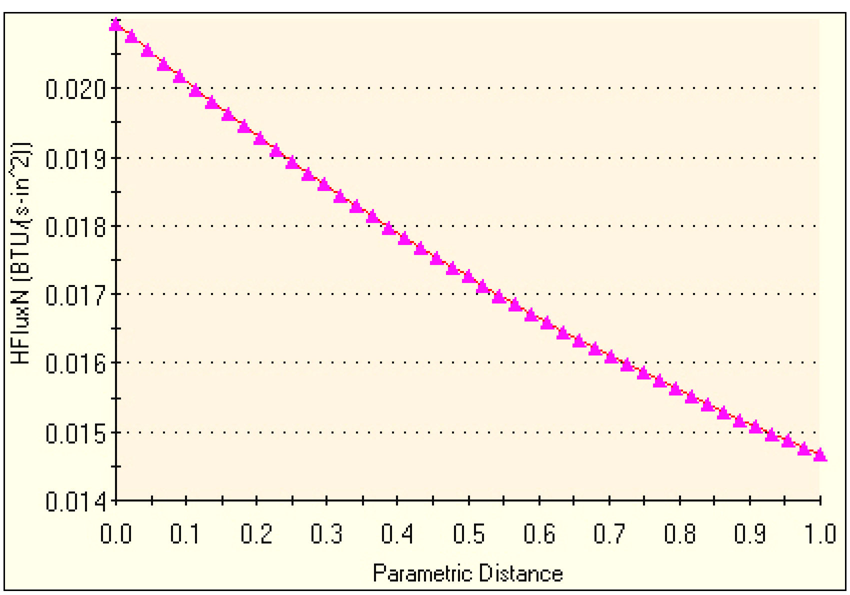


Figure 13-1 Radial heat flux magnitudes

14 Thermal Stress Analysis

Non-uniform temperature distributions in a component cause deflections and stresses in the part. Such “thermal loads” are difficult to visualize and usually need to be determined by a thermal analysis. The output temperatures from that analysis can be used as input data for a stress analysis. In this section a common physics experiment, involving a constant temperature change in the component, will be studied.

14.1 Two Material Thermal Stress Model

14.1.1 Introduction

Design tasks often involve parts made of more than one material. Sometimes the materials are bonded together and at other times they are not. Assembled parts may look like they consist of bonded parts simply because appear to be touching. However, analysis software assumes that adjacent parts are not bonded or even in contact unless the user specifically establishes such relations. To illustrate these concepts consider the common elementary physics experiment of uniformly heating two bonded beams made of materials having different coefficients of thermal expansion, α. You probably recall that when subject to a uniform temperature change, ΔT, the bonded beam takes on a state of constant curvature, of radius r, even though there are no externally applied forces. Denote the unstressed free length as L, and the final end deflection as δ (see Figure 14-1).

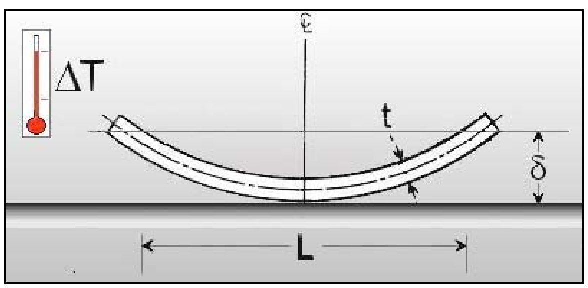


Figure 14-1 A bimatellic strip

Such devices are commonly used in mechanical switches that are temperature activated.

To illustrate this type of device, make a relatively long beam of rectangular cross-section from steel, and one with identical geometry of copper. You will save the two parts and then bring them together in an assembly.

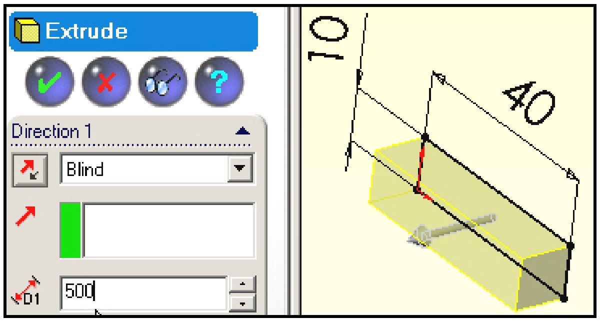
14.1.2 Two beam parts of different materials

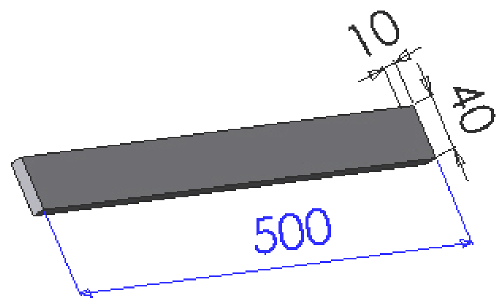
Let the length be L = 1000 mm, width w = 80 mm, and depth d = 10 mm for each material. When assembled, the two members will have a total thickness of t = 2d and two planes of symmetry of the geometry, material properties, and the uniform temperature change. Thus, you will use a one-quarter symmetry model. One symmetry plane will be at the middle of the length and transverse to it. The second will pass through the long axis of the beam and divide it into two equal sizes. Therefore, each of the material parts will be 500 mm by 40 mm by 10 mm.

14.1.3 Construct the steel first beam:

1. Right click **Front→Edit Sketch→ Insert rectangle**. Set the width to 40 mm and the depth to 10 mm.

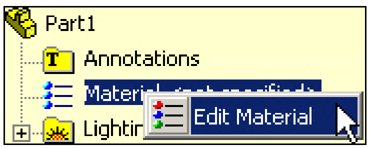
2. **Extruded Boss/Base→Extrude panel.** Pick **Blind** and a distance of **D1** = 500 mm, click **OK**.





Now that you have built the first beam body, assign it the material property of steel.

1. Right click on **Materials <none selected> →Edit Material**.



2. In the **Materials Editor** select the material desired, click **OK**.

3. Verify that the part now shows the selected material name. **File→Save as** filename steel_beam.sldprt.

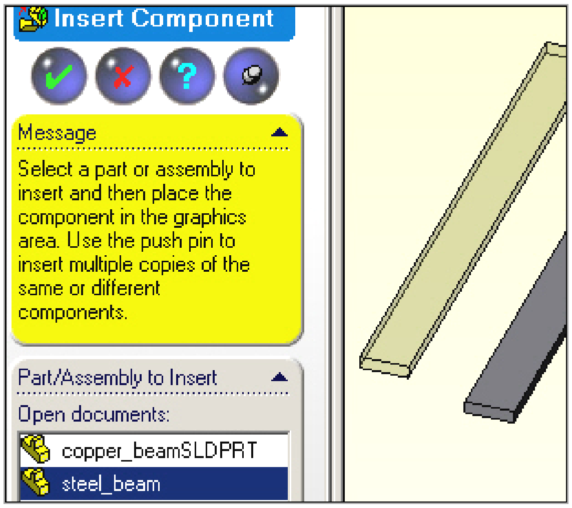
14.1.4 Copper beam

Open a new (second) part. Repeat the processes above except select copper as the material and save it with the filename copper_beam. Use **Window→Tile Horizontally** to see both parts, such as seen in Figure 14-2

14.1.5 Part assembly and geometric mating

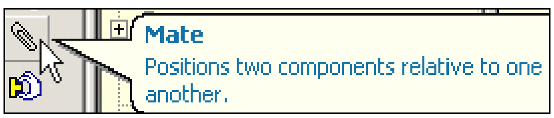
Now open a new assembly:

1. Use **New→Assembly→OK**.
2. In the **Insert Component** panel click on the steel beam in its window and drag it to the assembly window. As the first part it will be fixed in the view and other parts will move relative to it.
3. Click on the copper beam part and drag it into the assembly window.

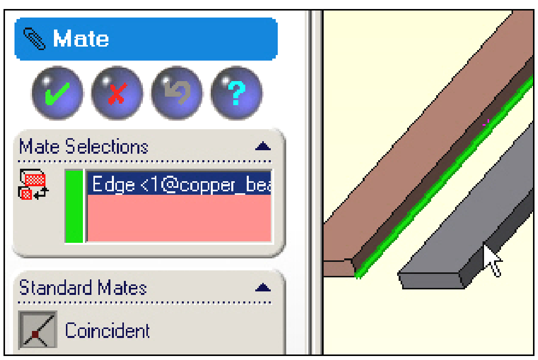


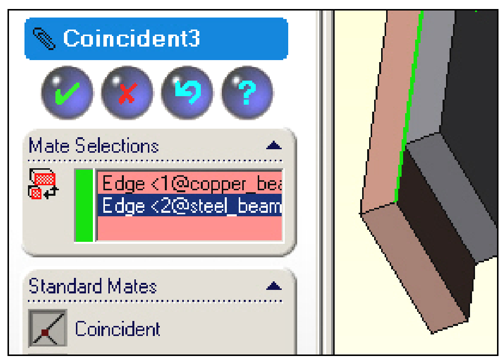
Next you need to carry out the geometric mating of the different materials so that the surfaces that may be bonded in the finite element analysis are touching in the assembly:

1. Select the **Mate** icon.

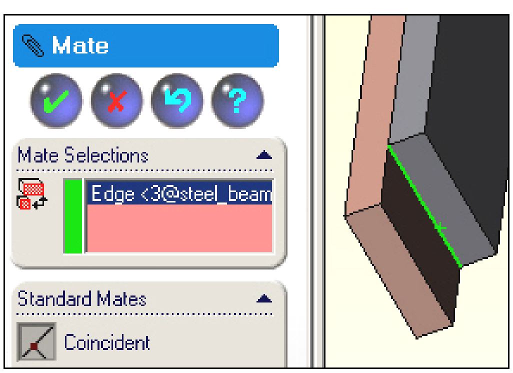


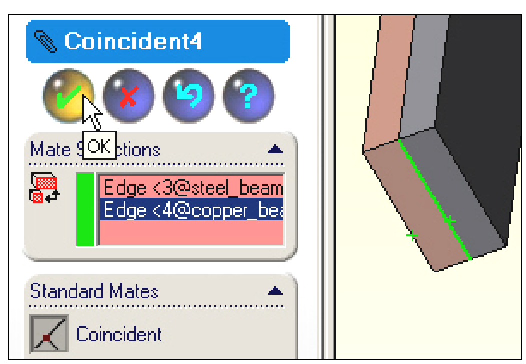
2. In the **Mate panel** pick **concentric** as the **Standard Mate**.
3. Select the long bottom edge of the copper and the long top edge of the steel as the **Mate selections**.





4. In the **Mate panel** pick **concentric** as the **Standard Mate**.
5. Select the short bottom front edge of the copper and the short top front edge of the steel as the **Mate selections**, click **OK**. Finally, **File→Save as** file name copper_steel.sldasm.





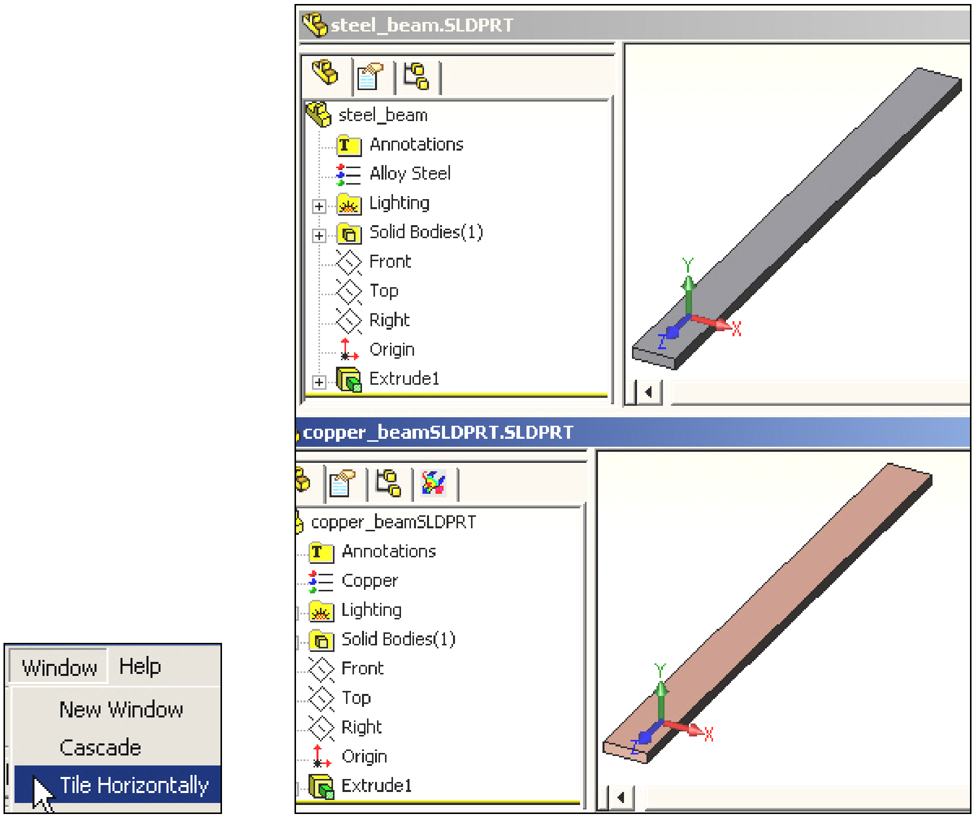


Figure 14-2 View both parts in tiled windows

Before leaving the assembly, note in Figure 14-3, that the copper part was placed on top of the steel part. That should give you a hint on the deflection directions that will be computed in the CosmosWorks study.

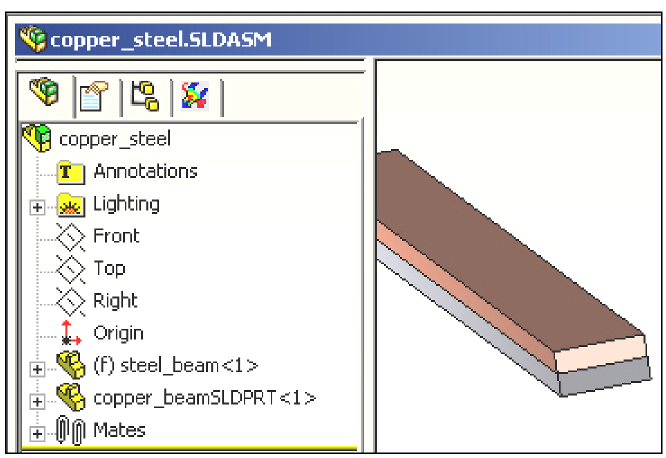


Figure 14-3 The mated assembly before entering CosmosWorks

14.2 Estimated deflection results

Before building a full computer model you should try to estimate what to expect as an answer. As the assembly was built, the copper was on top. Since it has a higher coefficient of thermal expansion the top will get longer that the bottom. If they are bonded then beam should bend away from the copper. If bonded there will be a common axial extension, which is less than the free thermal expansion of the copper, but more than the free expansion of the steel. An equal and opposite set of internal bonding forces are developed at the common (equilibrium) extension. The classic handbook “Roark’s Formulas for Stress and Strain” [16, 17] gives the beam theory solution for the deflections and stresses for a bimetallic strip (neglecting Poisson's ratio included here). The maximum deflection for equal thickness material layers reduces to:

$$\delta_Y = 6(\alpha_s - \alpha_c)(T\text{-}T_0)(L^2) / (t^2 K_1), \quad K_1 = 14 + (E_s/ E_c) + (E_c/ E_s).$$

For the given data: K_1 = 16.433, $(T\text{-}T_0)$ = 280 C, $(\alpha_s - \alpha_c)$ = - 1.1e-5 1/C, L = 0.5 m, and t = 0.01 m which yields a deflection estimate of δ_Y = 0.0281 m.

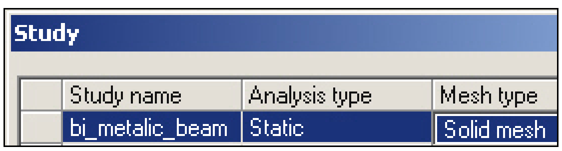
14.3 CosmosWorks thermal stress model

14.3.1 Beginning a study

Assume that the mated assembly was stress free at a temperature of 20 C and is to be heated to a uniform temperature of 300 C. Still in the assembly process, begin a CosmosWorks study.

1. Select the **CosmosWorks icon** to open its manager panel. Because the assembly has touching faces the **Contact/Gaps** feature appears.

2. Right click on it and select **Bonded** touching faces. (Later, you will try **Free** for comparison purposes).
3. Right click on the **Part Name→Study**. In the **Study panel** give a **Study name**, pick the static **Analysis type**, and select a solid (or shell) **Mesh type,** click **OK** .

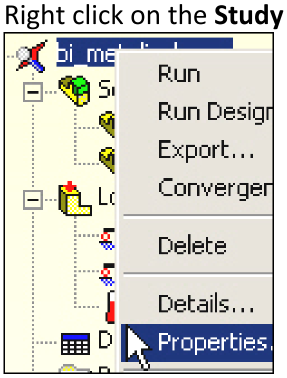


Having given a study name and a mesh type you would usually click on the mesh type and assign material properties. However, that assignment was made within SolidWorks while constructing the two individual beam parts. You could edit the imported properties at this point if you have decided on a different material.

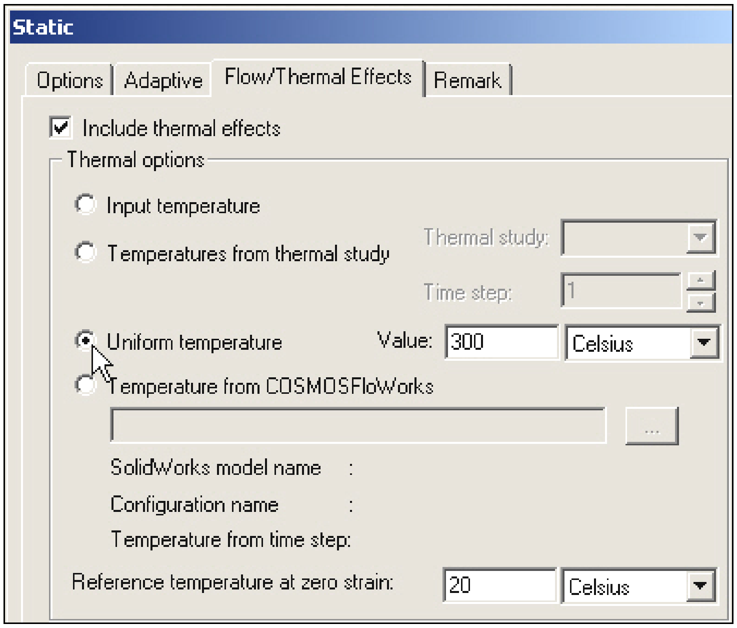
14.3.2 Material selections and thermal loading

Remember that the material properties for each beam were selected within SolidWorks. They have been imported into CosmosWorks and may need to be reviewed or edited. In this problem you have bonded materials subjected to the same temperature change. Therefore the coefficients of thermal expansion (actually their difference) are very important and should be double checked. Assume that has been done. Identify the thermal loading early in the process:

1. Right click on the **Study name→Properties** (rather than the **Temperature** panel).



2. In the **Static panel** pick **Flow/Thermal Effects**.
3. Check the **Include thermal effects**, and pick **Uniform temperature**.
4. Enter the final temperature **value and units** and the **stress free** (zero strain) temperature, click **OK**.



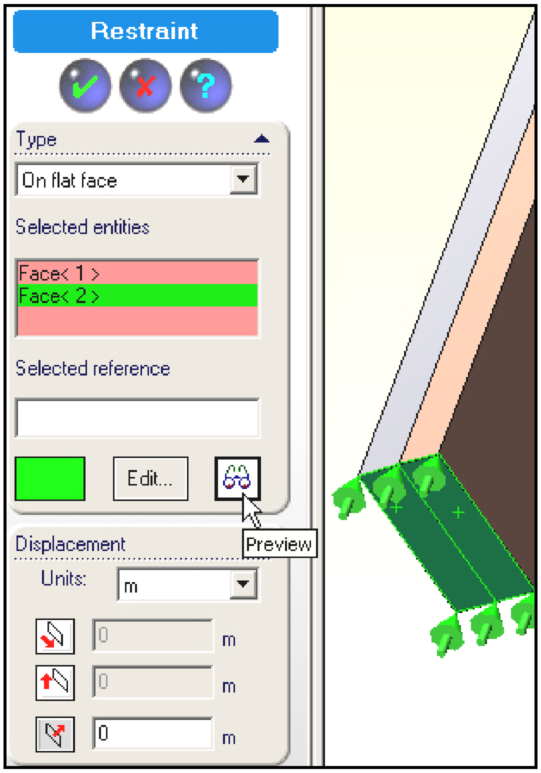
For future reference you should note that the **Static panel** provides for non-uniform temperature distributions to be input, or imported from external files.

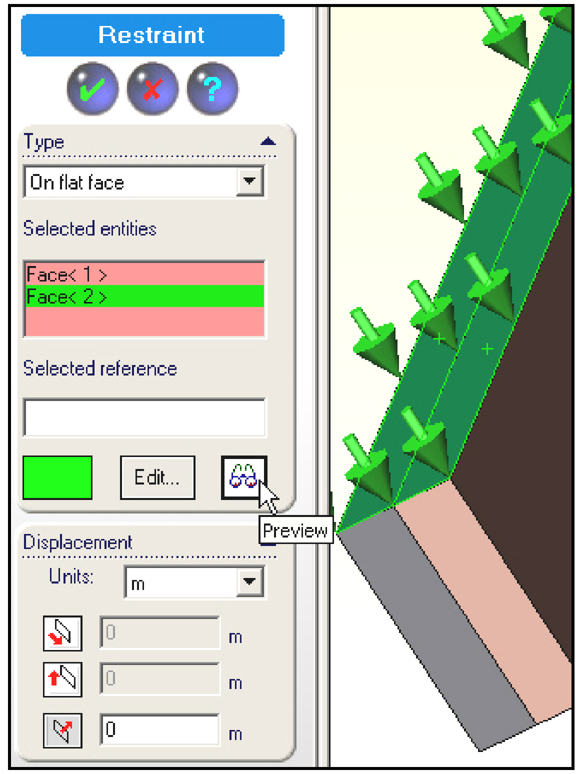
14.3.3 Restraints

As with any problem, you must prevent the six possible rigid body motions (RBM). That must be done in a way that does not restrict the thermal motion, unless the true support system does that. You also need to invoke the two symmetry planes used to reduce the computational costs. The first (short) symmetry plane restrains rigid body motion in the direction of the long axis of the beam (i.e. normal to the plane). Since there are multiple points on that restraint plane it will also prevent RBM about the two coordinate axes lying in that plane (so 3 RBM remain to be addressed). The second (long) symmetry plane must also prevent displacement normal to it, and thus also prevents the third remaining RBM rotation (1 RMB translation remains). Finally, one point must be restrained to eliminate the remaining possible RBM (in the direction parallel to the corner formed by the two symmetry planes). Any point in the assembly could be picked and all computed motions in this last direction will be computed *relative to that chosen point*. The most logical point to chose is in the corner junction of the first two restraint planes, and at the interface of the two materials at the center of the beam.

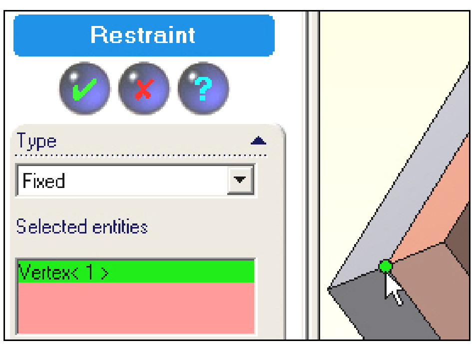
The basic restraint steps are:

1. Right click on **Loads/Restraints→Restraints→Restraint pane**l.
2. Pick the on a flat face **Type** with zero normal **Displacement** and use the symmetry plane as the **Selected Entities,** click **OK**.





3. Right click on **Loads/Restraints→Restraints→Restraint panel**. Select the adjacent faces (above).
4. Use the corner vertex as the **Selected Entities**, and fixed as the **Type**. (Actually, you only needed to fix the displacement parallel to the intersection line of the first two restraint planes because two of the point restraints are already done.)

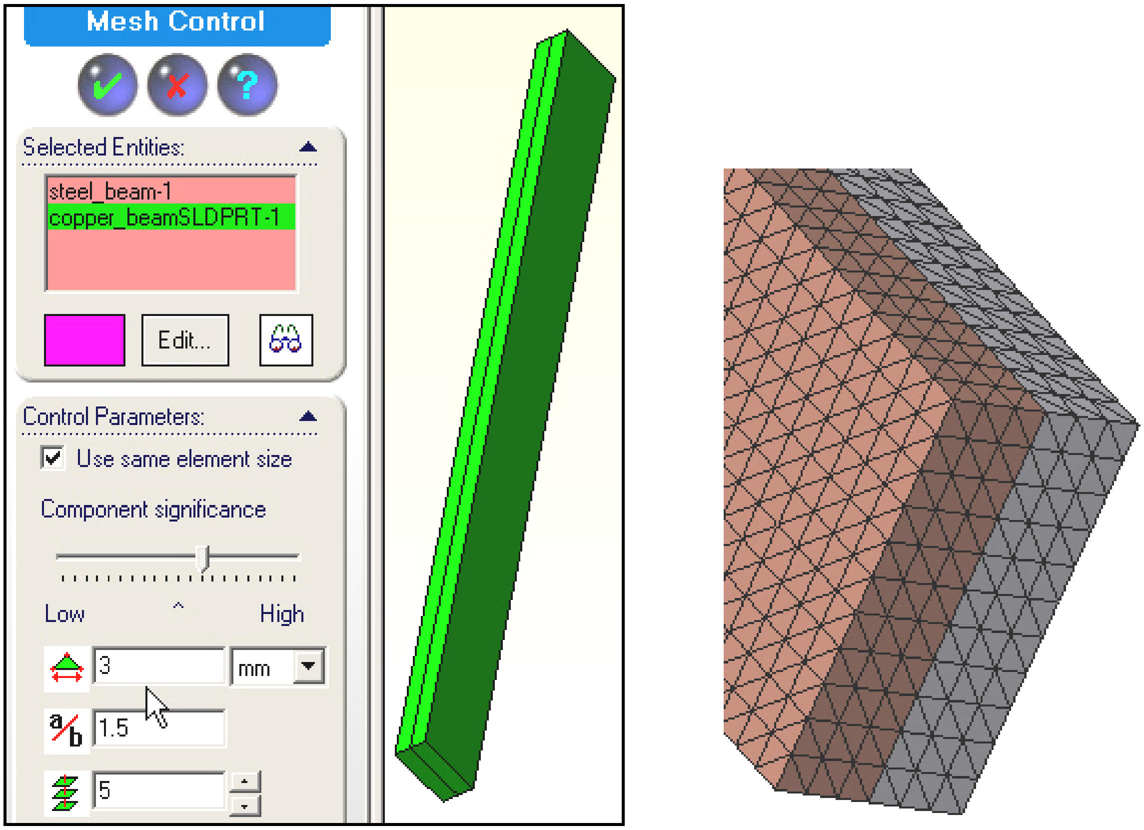


14.4 Mesh control

You should expect the temperature change to cause a bonded beam to curve. Therefore bending stresses will develop. A crude mesh will probably give a reasonably accurate displacement calculation. However, to get accurate bending stress recovery you need at least 5 to 6 layers of elements through the thickness of each material. That requires mesh control on each of the four faces sharing the thickness dimension. Bending fiber stresses are most likely to be large on the top and bottom layer of each material. Thus they need a good surface mesh too. You can either lower the average element size to get that and/or manually control the element sizes on those three surfaces:

1. Right click on **Mesh→Mesh Control**.

2. In the **Mesh Control panel** select both parts in the assembly as the **Selected Entities** and check **use same element size.** Examine the default element size and reduce it 3 mm for an initial mesh.



3. Right click on **Mesh→Create**, then **Mesh→Show mesh** (above).

14.4.1 Mechanical Loading

Remember that there are no external mechanical loads to be applied here, and the uniform temperature change has all ready been addressed.

14.5 Displacements solution

There are many displacement degrees of freedom in this mess and most are wasted since there will be little change in displacements through the width, w. While you should get good deflection results the three elements through each material thickness may not give good stress estimates. (A shell model, in plane stress, probably would have been more accurate and had much fewer equations to solve.) Now start the equation solver with **Study name→ Run** in the manager panel.

14.6 Post-processing

14.6.1 Displacement results

A look at the deformed shape shows the shape that was expected due to the higher thermal expansion coefficient of the copper. The overall displacement size (un-magnified) is large compared to the total beam thickness. That would usually mean that you should consider a geometrically non-linear "*large deflection analysis*", but it might not be justified for this thermal loading. If the materials had not been bonded you would have just axial (UZ) displacements. Looking at them in detail, as seen in Figure 14-4, you see that component is always positive, but decreasing from top to bottom (from copper to steel).

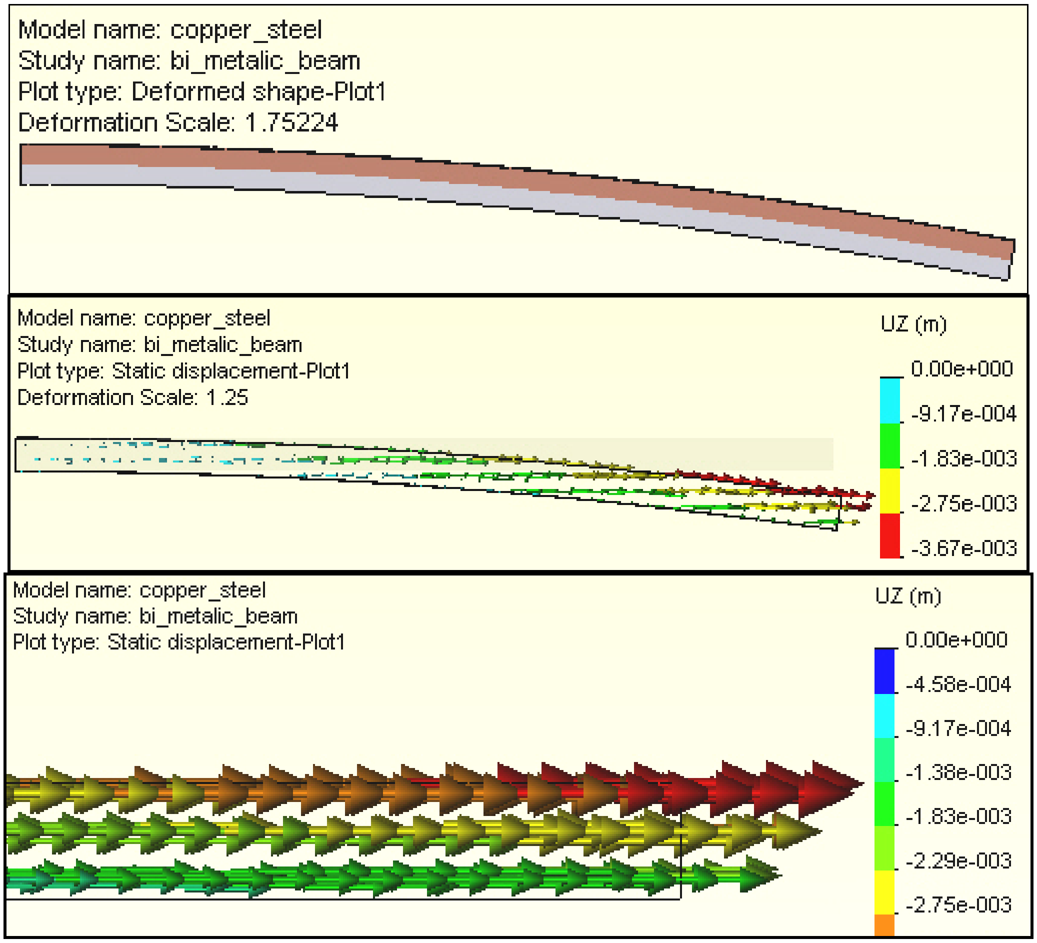


Figure 14-4 The deformed shape and its axial components

The total displacements (URES), and the transverse (UY) components, given in Figure 14-5, are much larger that the previous axial components. You can create a list of the largest UY components by zooming in on the region and picking nodes:

1. Right click in the graphics area and select **Probe**.
2. Select the desired nodes and the component values will be added to the plot (usually this step is easier if you use **Mesh→Show mesh** before **Probe**), and placed in a list that appears in the **Probe panel**.

Probe

Study name: bi_metalic_beam

Plot type: Static displacement-Plot1

Node	UY (m)	X (mm)	Y (mm)	Z (mm)	Components
22405	85e-002	112.34	114.81	-116.17	steel_beam-1
276	84e-002	112.34	134.81	-116.17	copper_beam

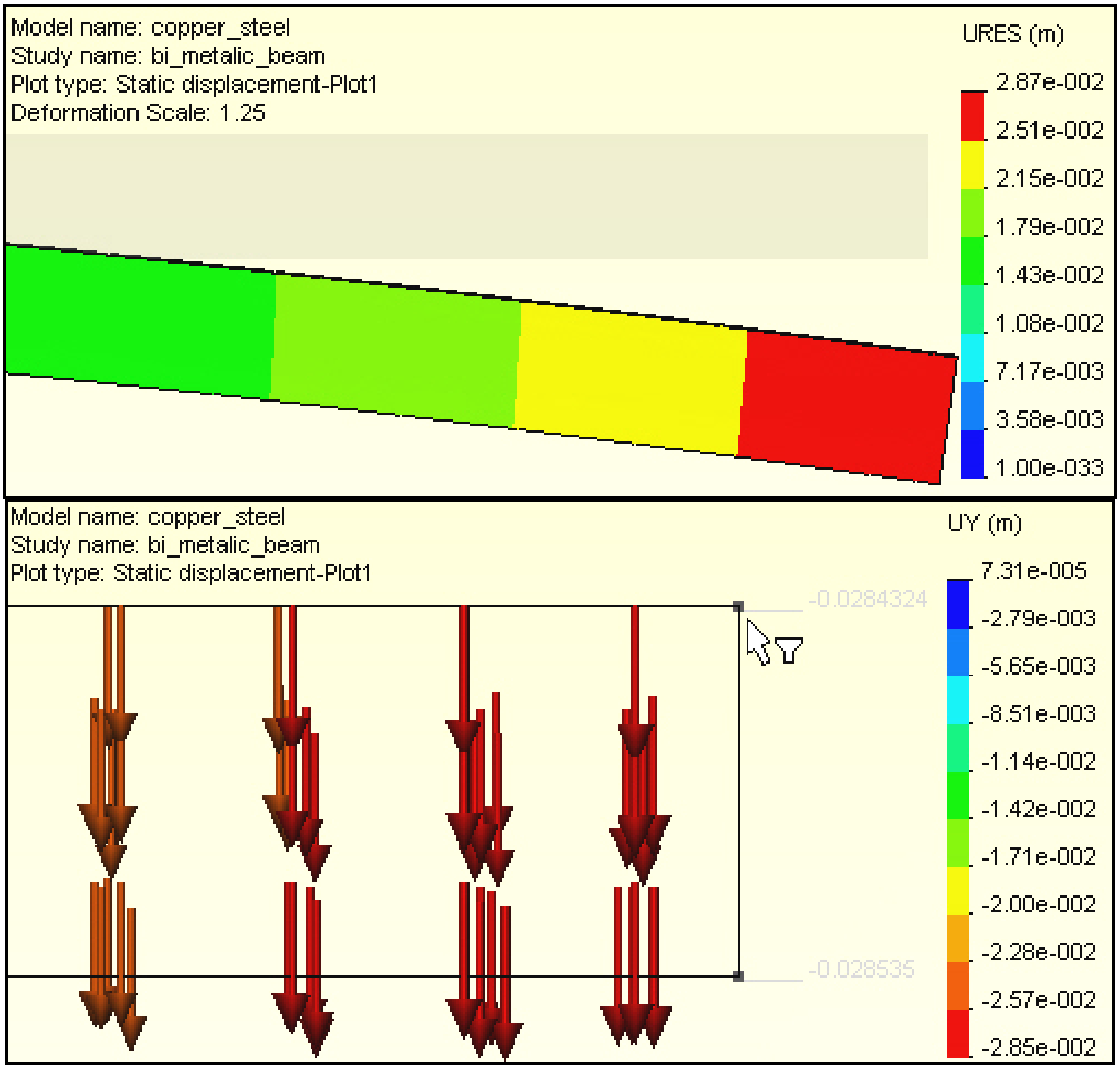
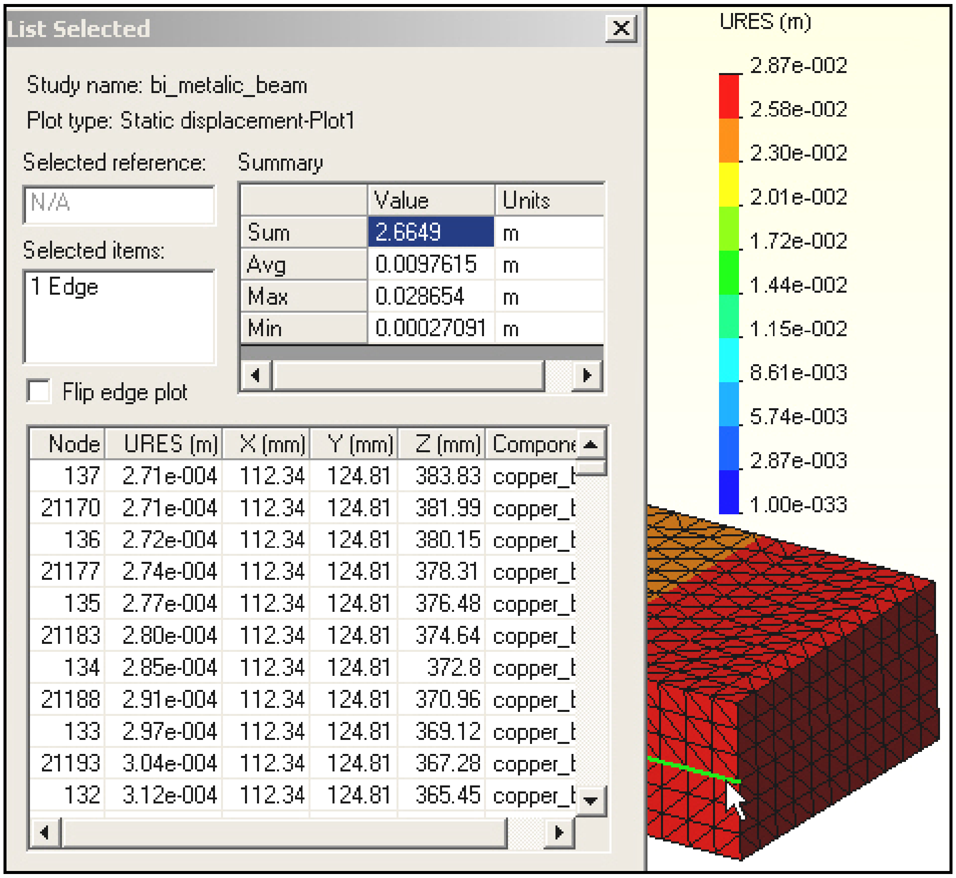


Figure 14-5 The undeformed assembly, resultant and UY displacement components

14.6.2 Listing and graphing displacements

CosmosWorks allows the creation of tables and graphs of any displacement component. Graphics usually show more detail than a contour plot. To display graphs you:

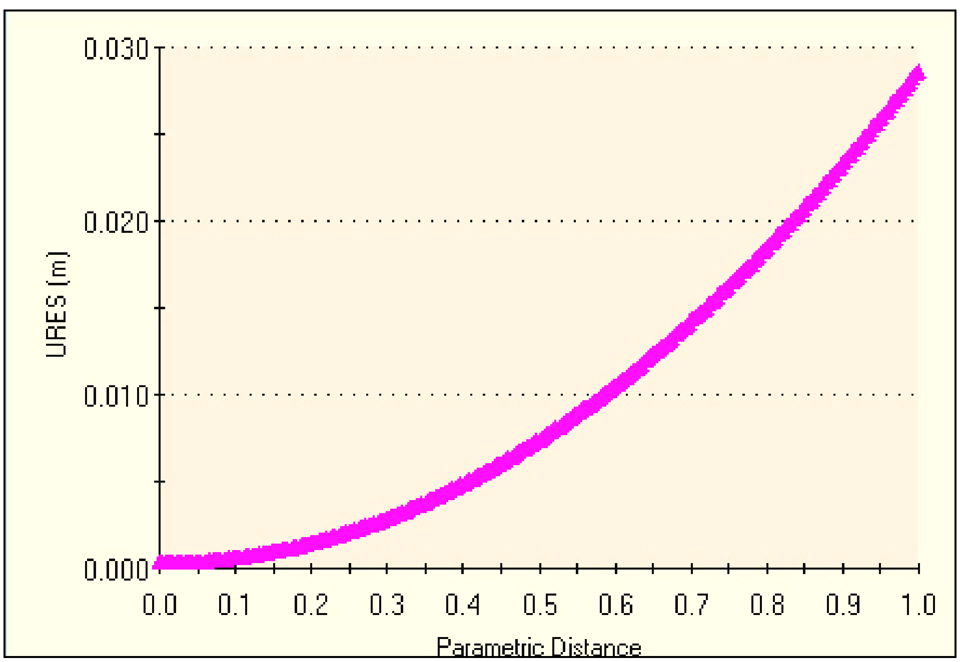
1. Display the displacement component of interest.
2. Right click in graphics area → **List Selected.** Pick the surface or (split line) curve.
3. Click on the **Update** button on **List Selected panel**. Note that the **Sum**, **Av**erage, **Max**imum, and **Min**imum values are listed along with all the nodal values selected.



	Value	Units
Sum	2.6649	m
Avg	0.0097615	m
Max	0.028654	m
Min	0.00027091	m

Node	URES (m)	X (mm)	Y (mm)	Z (mm)	Compone
137	2.71e-004	112.34	124.81	383.83	copper_t
21170	2.71e-004	112.34	124.81	381.99	copper_t
136	2.72e-004	112.34	124.81	380.15	copper_t
21177	2.74e-004	112.34	124.81	378.31	copper_t
135	2.77e-004	112.34	124.81	376.48	copper_t
21183	2.80e-004	112.34	124.81	374.64	copper_t
134	2.85e-004	112.34	124.81	372.8	copper_t
21188	2.91e-004	112.34	124.81	370.96	copper_t
133	2.97e-004	112.34	124.81	369.12	copper_t
21193	3.04e-004	112.34	124.81	367.28	copper_t
132	3.12e-004	112.34	124.81	365.45	copper_t

4. To display a graph of the selected items click on the **Graph** icon.



The temperature difference not only causes a curvature in the long direction of the assembly, but also in the narrow direction. To see that effect (Figure 14-6), repeat the above steps on a line in the narrow direction.

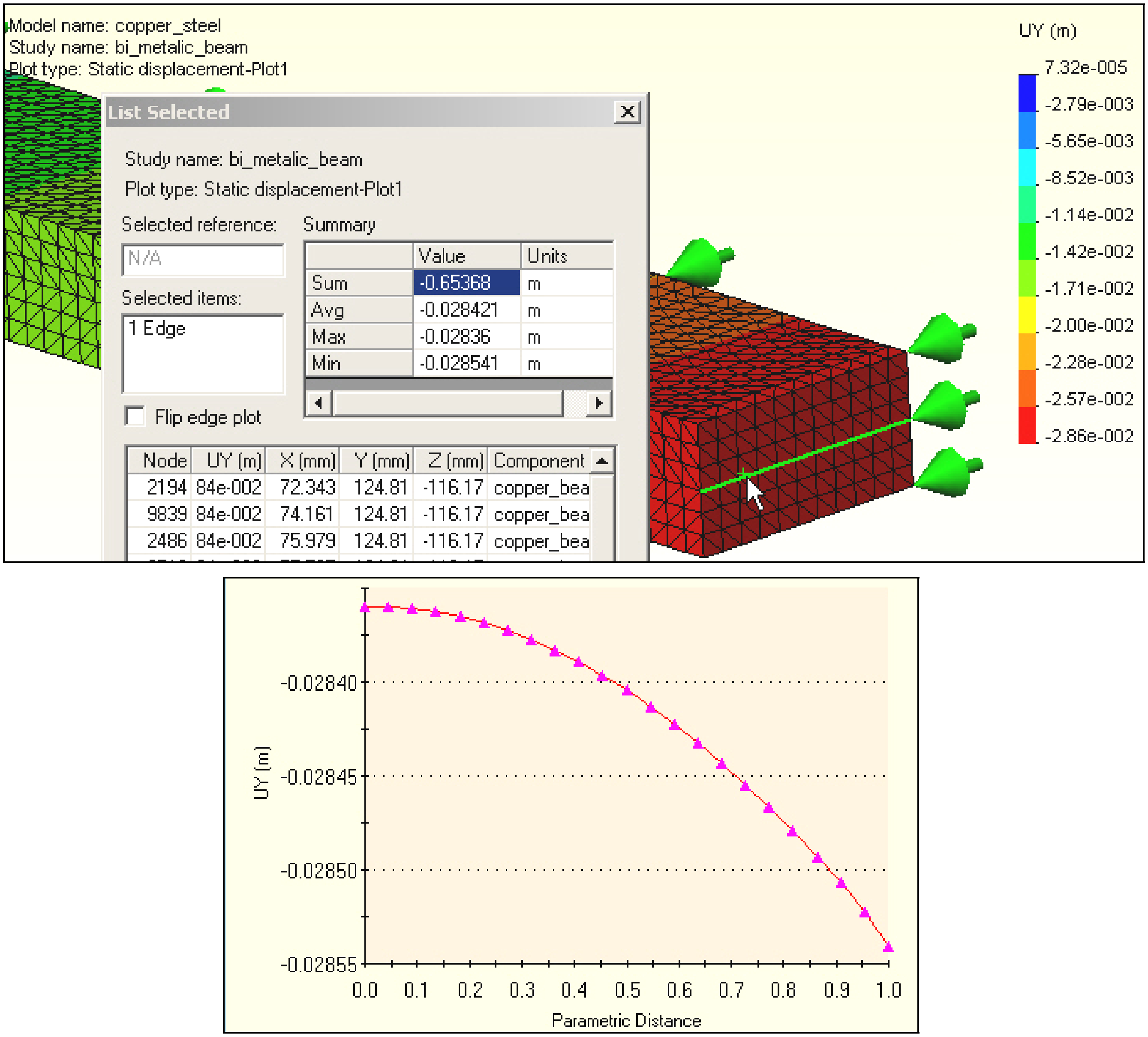


Figure 14-6 Transverse curving at interface plane

14.6.3 Thermal stresses

The stresses in the bimetallic beam should be essentially constant except very close to the free end. Thus, you need only display the two ends in detail. The mid-span symmetry plane normal Z-stress component (beam fiber stress) is given in the top of Figure 14-7, while the bottom of that figure shows the corresponding results at the free end. The wiggles in the contours imply that the mesh is too crude near the center bonding plane (which is an important region for this load state.)

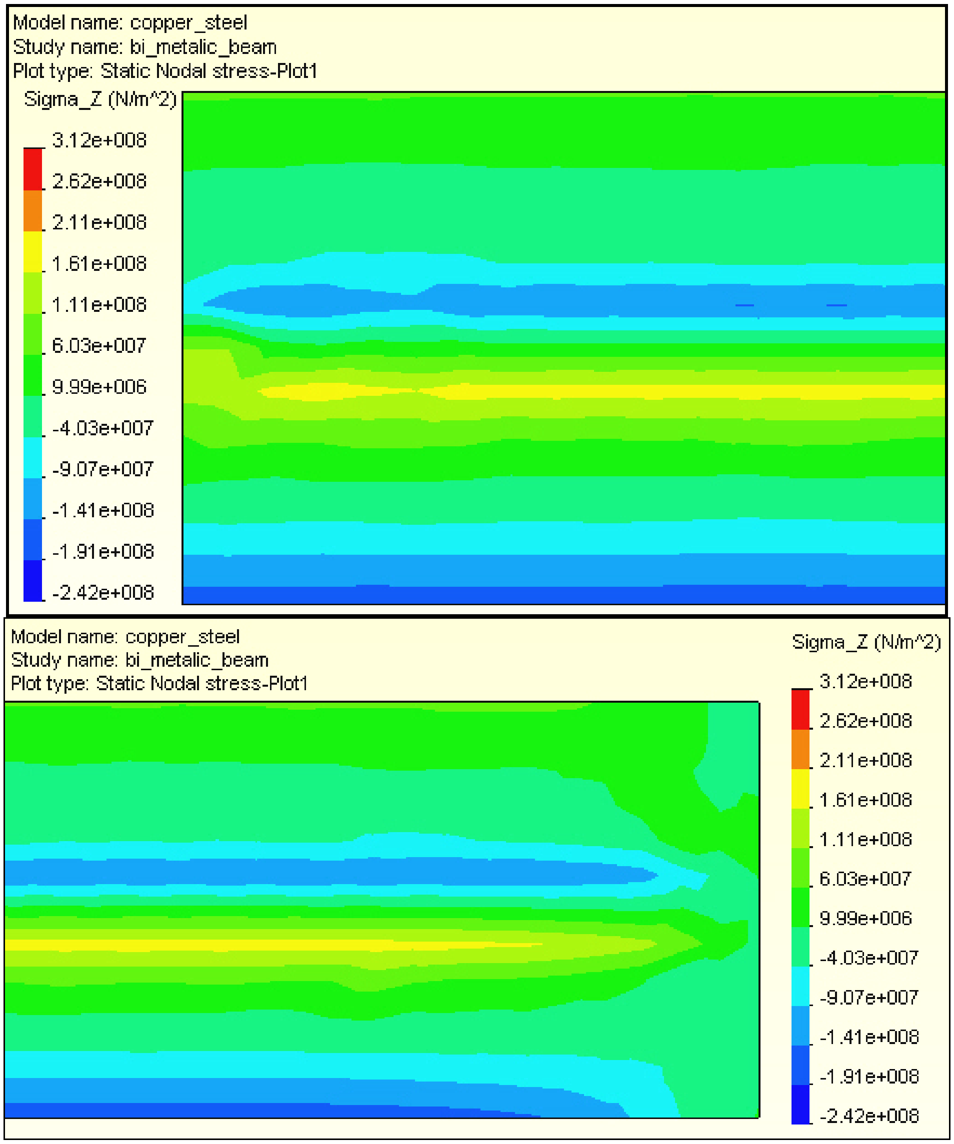


Figure 14-7 The fixed and free end axial fiber stress magnitudes

To view these data in another way display a vector plot on this component (at each end of the beam):

1. Right click in graphics area, **Edit Definition→Vector→Line**.
2. Right click in graphics area, **Vector plot options**, and dynamically control the maximum vector size, and the percentage of nodes shown (see Figure 14-8). Observe that this type of plot does not make obvious the presence of a crude mess.

At this point (because of Figure 14-7) you should already expect that this mesh is too crude to rely upon for stress results. To verify that, consider the free end region von Mises failure criterion plot of Figure 14-9. Clearly, the contour lines have many wiggles and the study should be repeated with a finer mesh and/or a plane stress (shell) mesh. If you look at the deflection and stress plots in all three coordinate planes (not shown here) you see that this 3D model has mainly a 2D response. Very little is changing through the 40 mm half width. That also suggests using a shell model for a refined model. After doing that you could more wisely control a 3D mesh for a final study.

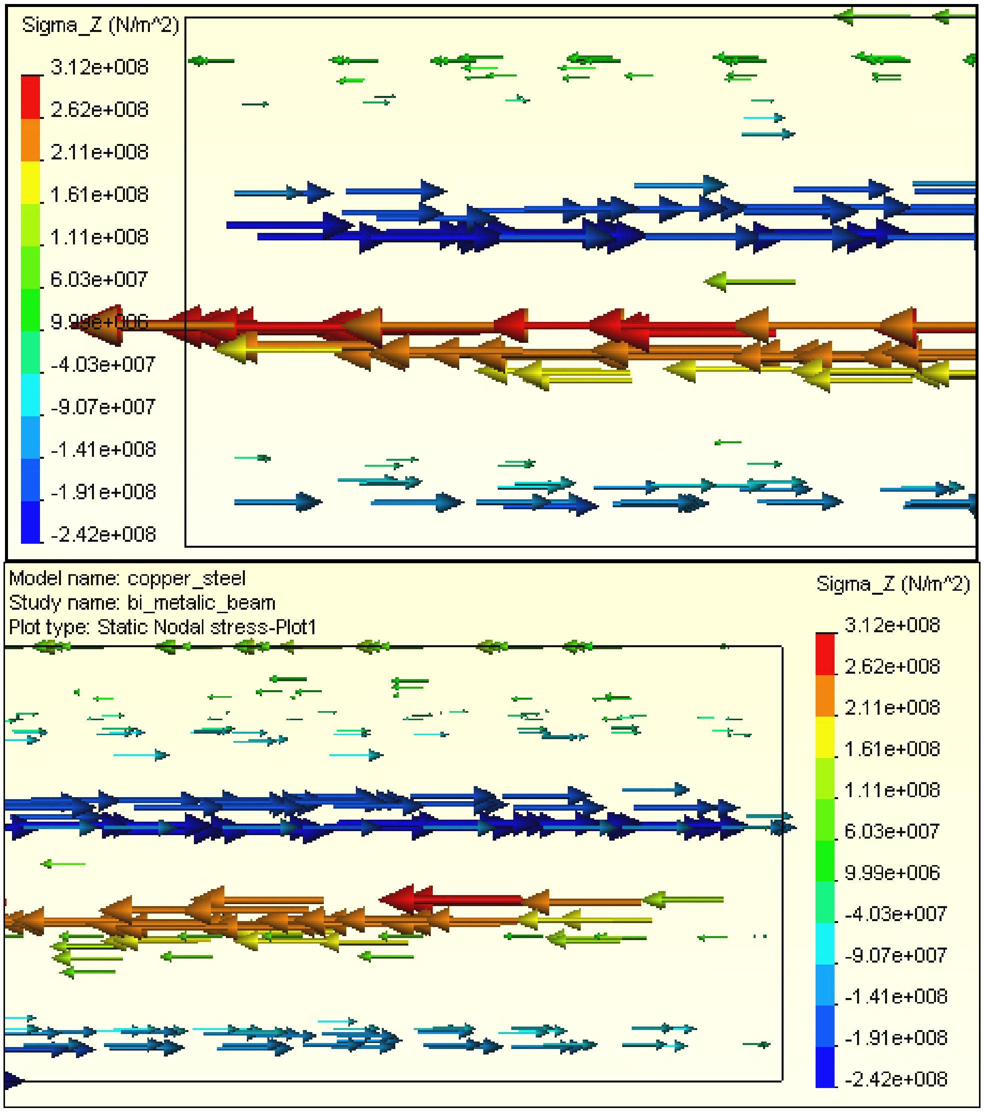


Figure 14-8 The mid-section and free end axial fiber stress component vectors

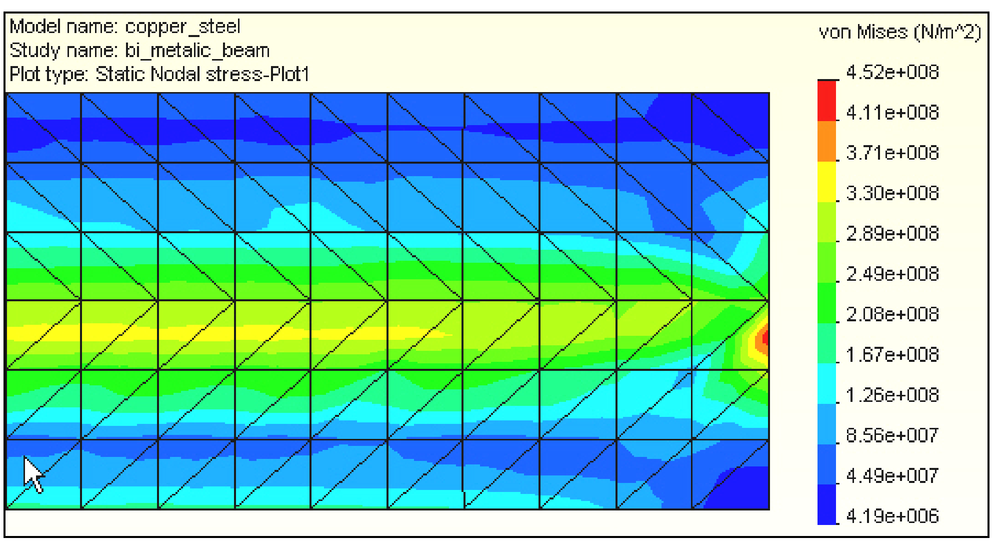


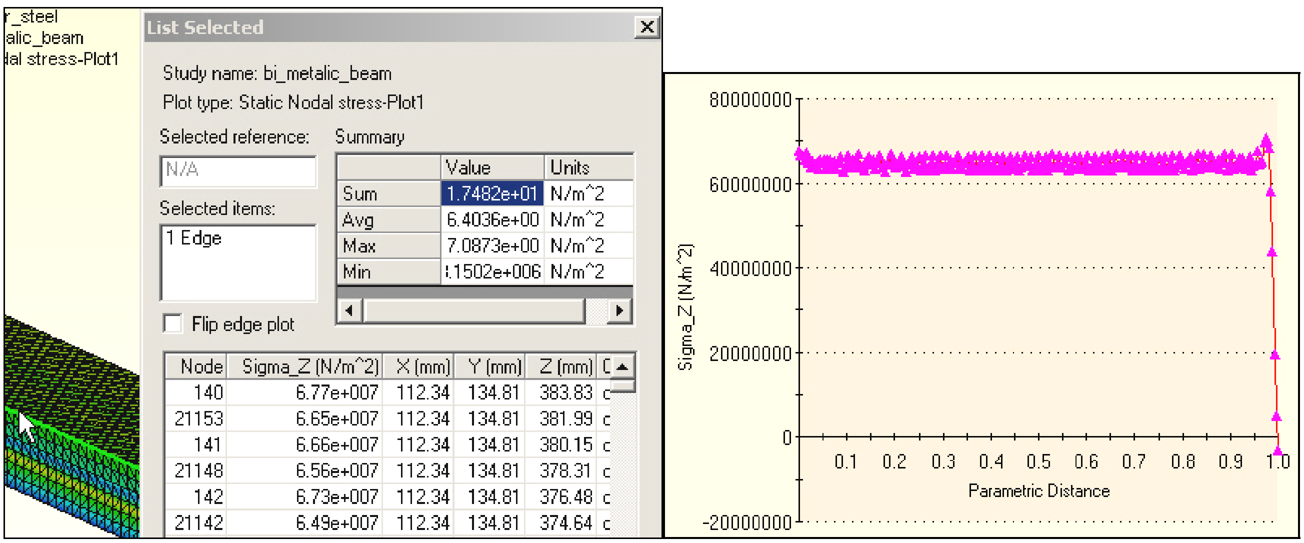
Figure 14-9 Crude von Mises contours at the free end region

Wiggles is stress contours can also be caused by using the default small deflection theory (as done here) to obtain a result that seems to be a large deflection result (greater than half the smallest material or part thickness). The above results suggest that this study should be rechecked, after the final part revisions, with a large displacement analysis option.

14.6.4 Listing and graphing stresses

CosmosWorks allows the creation of tables and graphs of any component available in the post-processing graphics options. To display such items you:

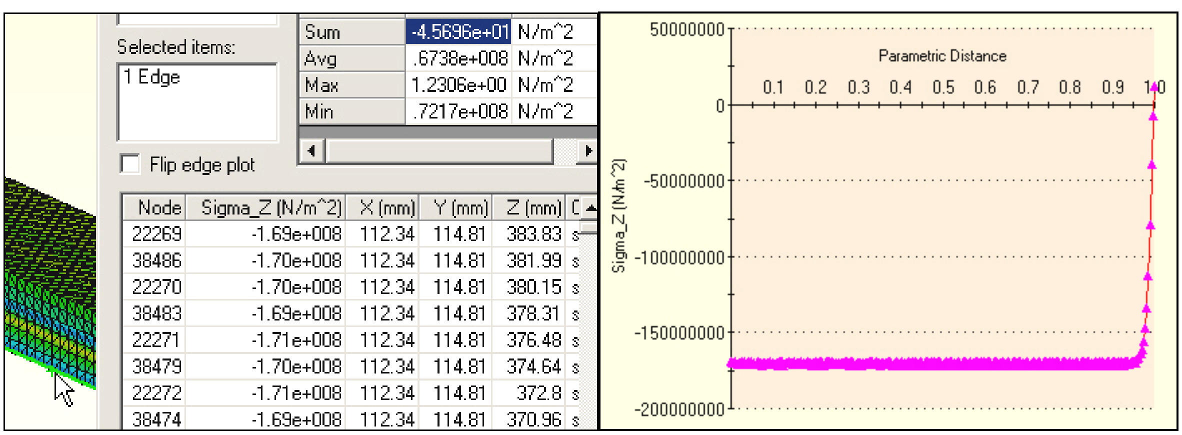
1. Display component of interest. Right click in graphics area → **List Selected.**
2. Pick the surface or (split line) curve.
3. Click on the **Update** button on **List Selected panel**. Note that the **Sum**, **Av**erage, **Max**imum, and **Min**imum values are listed along with all the nodal values selected.
4. To display a graph of the selected items click on the **Plot** button. For a surface you get a graph (in more or less random order) by node positions. For a line or curve selection you get a graph versus the non-dimensional position along the curve. You should plan ahead for such displays by adding split lines to the model where the model edges are not where you wish to plot the results.



	Value	Units
Sum	1.7482e+01	N/m^2
Avg	6.4036e+00	N/m^2
Max	7.0873e+00	N/m^2
Min	:.1502e+006	N/m^2

Node	Sigma_Z (N/m^2)	X (mm)	Y (mm)	Z (mm)
140	6.77e+007	112.34	134.81	383.83
21153	6.65e+007	112.34	134.81	381.99
141	6.66e+007	112.34	134.81	380.15
21148	6.56e+007	112.34	134.81	378.31
142	6.73e+007	112.34	134.81	376.48
21142	6.49e+007	112.34	134.81	374.64

Figure 14-10 Axial stress along top edge



Sum	-4.5696e+01	N/m^2
Avg	.6738e+008	N/m^2
Max	1.2306e+00	N/m^2
Min	.7217e+008	N/m^2

Node	Sigma_Z (N/m^2)	X (mm)	Y (mm)	Z (mm)
22269	-1.69e+008	112.34	114.81	383.83
38486	-1.70e+008	112.34	114.81	381.99
22270	-1.70e+008	112.34	114.81	380.15
38483	-1.69e+008	112.34	114.81	378.31
22271	-1.71e+008	112.34	114.81	376.48
38479	-1.70e+008	112.34	114.81	374.64
22272	-1.71e+008	112.34	114.81	372.8
38474	-1.69e+008	112.34	114.81	370.96

Figure 14-11 Axial stress along bottom beam edge

14.6.5 Validation calculations

While you may not trust the current stress results, you should still compare the computed displacements (which are always more accurate that the computed stress) to the initial engineering estimate of the maximum deflection. The computed maximum magnitude, in Figure 14-5 , was about 2.87 mm. The original deflection estimate was δ_Y = 2.81 mm, so the agreement was surprisingly good.

14.6.6 Large deflection check

CosmosWorks allows a large deflection calculation for solid meshes only. A large deflection analysis applies an iterative loading and displacement solution loop. In each loop a small part of the (thermal) loading is added to form a new resultant load. The corresponding displacements are computed and added to the original positions of the elements' nodes. The updated element positions are employed to re-compute the element stiffness matrices. The looping is repeated about 10 to 20 times to obtain the final displacements and the last set of element stresses. That is usually called a "*geometrically non-linear analysis*" (as contrasted to a "*materially non-linear analysis*" which is much more complicated).

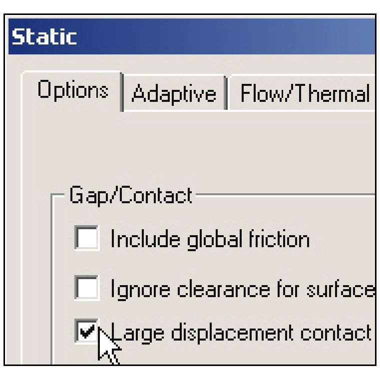
While some finite element systems have such a free standing option, in CosmosWorks you must employ a little trick you activate that feature since it was designed to be used with the large deflection contact analysis. The trick is to select two surfaces, on different components of the assembly, and have CosmosWorks check for them contacting (an additional and relatively computationally expensive process). Time is spent checking for a contact you know is extremely unlikely to occur, but you still get the desired large deflection analysis.

14.6.7 Start a new study

Such an analysis runs about 50-60 times longer that the standard analysis presented above. This was done, but the process is only outlined here since the changes were quite small and it takes a long time to run. To use this process you need a new study:

1. Right click on the **Assembly name→Study** to get to the **Study panel.**
2. In the **Study panel** assign a new **Study name**, **Analysis type**, and **Mesh type**. Before closing that panel, also pick **Properties** to get the **Static panel**.

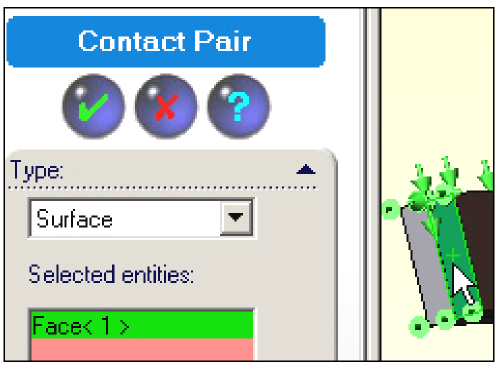
3. In the **Static panel→Options** check **Large displacement contact**, and next select **Flow/Thermal Effects** and set the same conditions as before.

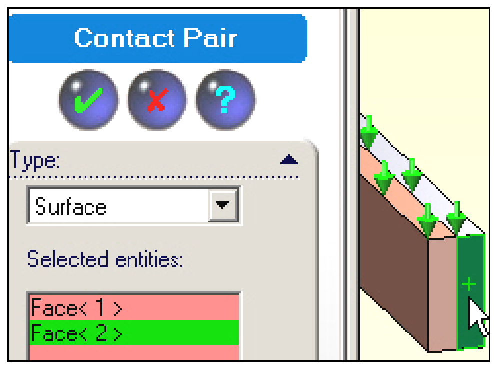


When the assembly opens, you will see that the **Contact/Gaps** icon has appeared above the usual Mesh icon location. The new icon must be used to both bond the touching faces, as before, but also to pick to potential contact surfaces as a trick to activate the large deflection iterative solver. To do that:

1. Right click **Contact/Gaps**, turn on **Touching Faces Bonded**, and select **Define Contact Pair**

2. When the **Contact Pair panel** appears, select a surface **Type**, pick the symmetry end surface on the copper component (actually any copper surface for a fake contact), rotate the assembly and pick the free end surface of the steel, respectively as the **Selected Entities→OK**.





14.6.8 New large deflection solution results

Using the usually solution process, **Assembly name→Run** (and a slightly cruder mesh) you eventually get new results for the displacements and stresses. However, they are nearly the same despite a much longer solution time. The new results are summarized in Figure 14-12. The bottom part of that figure shows the cruder mesh used here superimposed on the von Mises contours, which seem even less useful that from the original study. Thus, this study itself needs to be redone, with a much finer mesh through the thickness. Make that change, restart the iterative solution, and go out for a snack break.

14.6.9 Closure

The bonding makes a great difference in the response of any assembly. To see that, you could reload the assembly and not bond the touching surfaces. The resulting stresses should be zero and the displacements will only be in the UZ direction (try it).

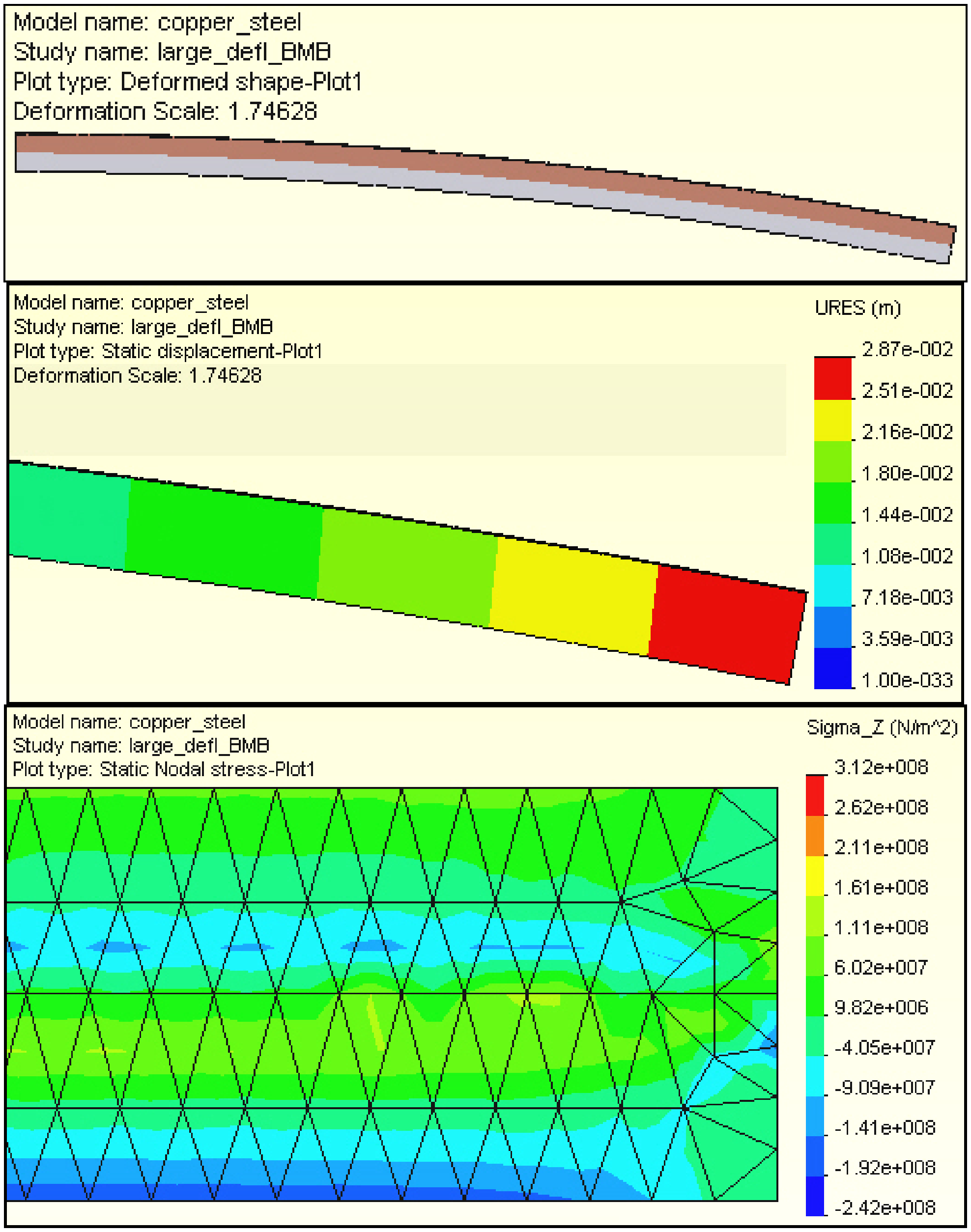


Figure 14-12 Selected large deflection results

15 CosmosWorks design resources

15.1 Introduction

This is simply a restatement of the CosmosWorks online design scenarios tutorial with a little more visual detail supplied on the various menu picks and thought processes that are covered in the version supplied with the software. In this lesson, you use design scenarios to investigate the effect of changing the applied loads and some dimensions of the part on the static analysis results. The part is supported and loaded as shown in Figure 15-1. The total force will be specified and it will be divided between various regions of the hanger. The force on the central hanger (FCenter) is twice the force on each of the side hangers (FSide).   You will study the effect of changing the total applied force, thickness and height of hangers, and the thickness of the back plate on the stress distribution in the part.

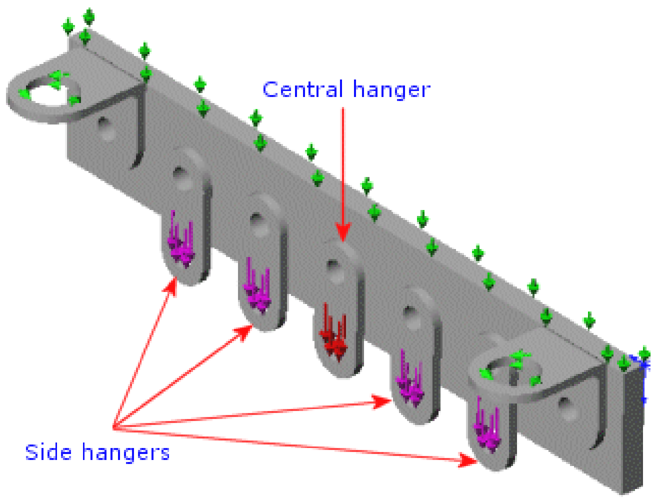


Figure 15-1 The assembly of interest

First you need to switch to the CosmosWorks Manager by clicking its icon, . Start a new study by:

1. Double click on the **Part Name → Study**
2. In the **Study panel,** assign a **Study name.**
3. Pick **Analysis type→Static**
4. Select the **Mesh Type→Solid Mesh**.

When the analysis menu expands you will see that the second item is associated with **parameters**. That is where you will have access to the definition of the parameters chosen by you to control, or define, the design scenarios.

15.2 Design parameter selection

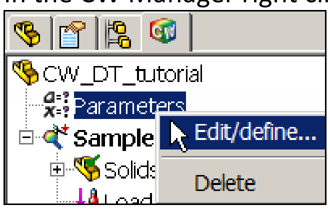
To begin that process you should have planned ahead and identified the items (geometry, loads, restraints, etc.) that you want to see varied. Then you have to input those design parameters. The following parameters, in Table 15-1, have been chosen to define the design scenarios of interest here:

Table 15-1 Descriptions of the design parameters

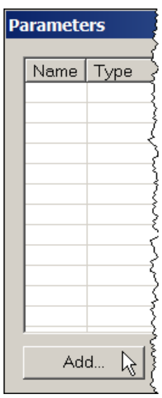
Parameter Name	Type	Description
FTotal	Force	Total force on the hangers
FCenter	Force	Force on the Central hanger. FCenter=FTotal/3.
FSide	Force	Force on a side hanger. FCenter=FTotal/6.
Height	Linear Dimension	Hanger height
Thickness	Linear Dimension	Hanger thickness
Back_T	Linear Dimension	Back plate thickness

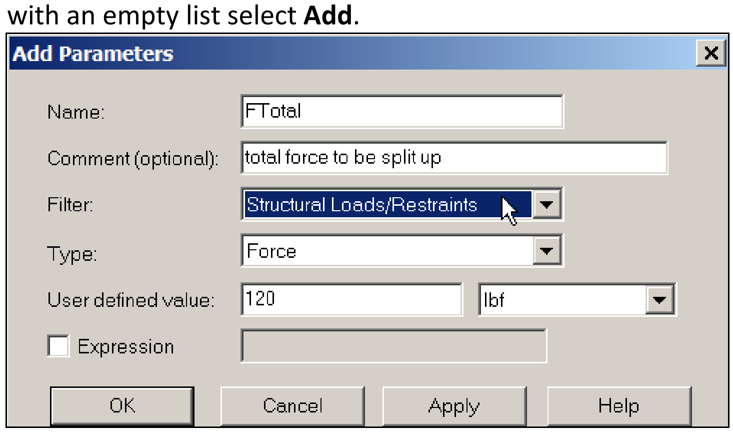
You can begin adding design parameters by defining the total force of interest:

1. In the CW Manager right click **Parameters→Edit/Define** to open the **Parameters panel**.



2. Since you are beginning with an empty list select **Add**.

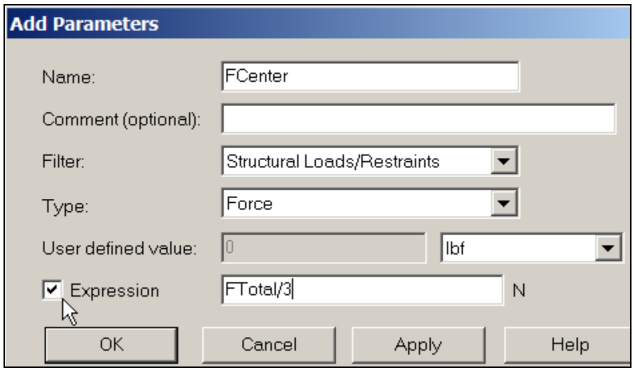


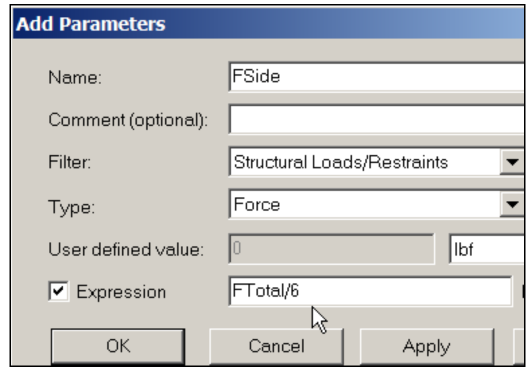


3. The **Add Parameters panel** appears (above). Enter the **Name → FTotal** and add a comment.
4. Select the **Filter→Structural Loads**, and pick **Type→Force**.
5. Finally set the initial **Value** and **Units** (say 120 lbs), click **OK**.

Then the entered data will appear in the top line of the Parameters panel. Continue defining force parameters in a similar fashion. The next one will be given in terms of a parametric expression of the now know total force:

1. In the **Parameters** panel select **Add** again.
2. The **Add Parameters panel** appears. Enter the **Name → FCenter** and add a comment.
3. Select the **Filter→Structural Loads**, and pick **Type→Force** and desired **Units**.
4. This time you want a parametric equation so check **Expression**.
5. Enter the relation, (FCenter =) **FTotal/3,** click **OK**. The second parameter appears in the **Parameters panel**.

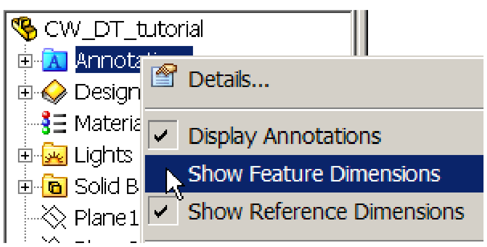




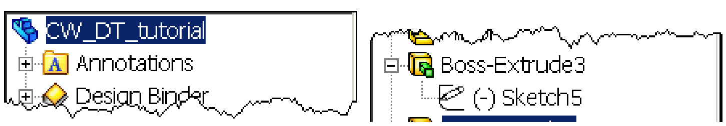
6. Carry out the same sequence of steps (above) for the last force parameter, FSide. The only difference is the **Expression** which is (FSide=) **FTotal/6**. The results for the last two force definitions as the next two lines in the **Parameters panel**.

The next design parameters will be part dimensions picked from the graphics display. To assure that they are visible to pick:

1. **Select** the **SolidWorks** icon to open that Manager panel.
2. Right click on **Annotations→Show Feature Dimensions**.



3. Knowing that we are interested in **Boss-Extrude3**, expand it and open **Sketch5** so the dimensions you want will be available to pick with the cursor. (Here is an example where renaming Boss-Extrude3 to something related to the part would have been helpful.)



4. While it is not required for the design scenarios, remember that it may save time in later design reviews to change the dimension name to match the design parameter name that you are getting ready to assign. In the graphics area **select dimension** 0.308472 (D1) **→Properties→Dimension Properties→Name→Height→OK**. This is shown in Figure 15-2.

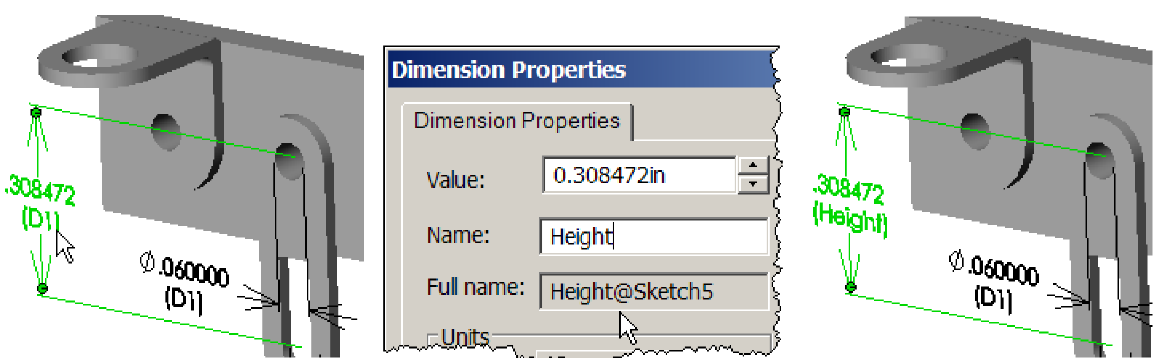
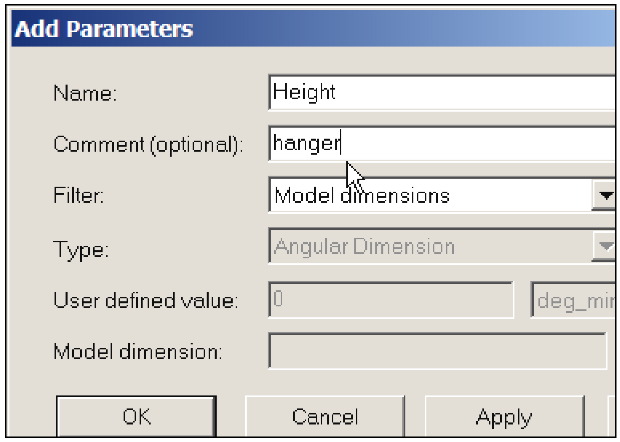


Figure 15-2 Renaming the hanger height dimension

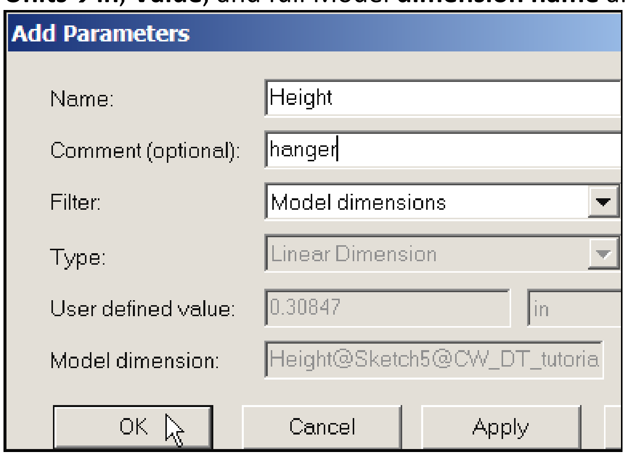
Now you are ready to continue defining the class of design parameters that are part dimensions:

1. **Parameters panel→Add**. In **Add Parameters panel→Name→Height**. Add a comment.

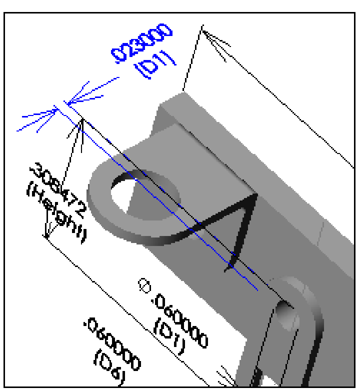


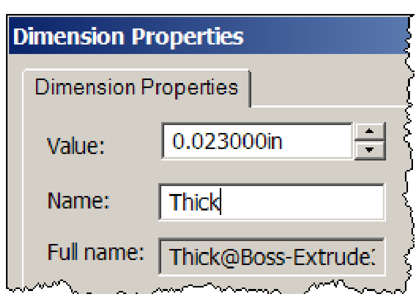
2. **Filter→Model dimensions.**

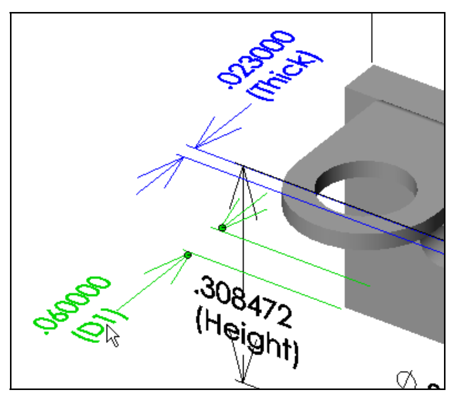
3. In the graphics area pick the above dimension (0.308472 in). Note that the **Type →Linear Dimensions, Units→in**, **Value**, and full Model **dimension name** are automatically inserted. Check them, click **OK**.



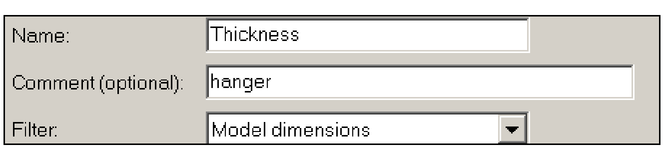
4. Repeat this process, including renaming the dimension, for the hanger thickness of 0.023000.







5. In **Add Parameters→Name→Thickness.**



6. The final dimension parameter is the back plate thickness of 0.060000. Simply pick it (without renaming) and give it the **parameter name→Back_T,** click **OK**.
7. Now the initial design parameters are complete as seen in the **Parameter panel** of Figure 15-3.

Parameters

Name	Type	Unit	User define...	Current value	Expression	Comment
FTotal	Force	lb	120	120		total force
FCenter	Force	lb	40	40	FTotal/3	
FSide	Force	lb	20	20	FTotal/6	
Height	Length/Displace...	in	0.30847	0.30847	Height@Sketch5@CW...	hanger
Thickness	Length/Displace...	in	0.023	0.023	Thick@Boss-Extrude3...	hanger
Back_T	Length/Displace...	in	0.06	0.06	D1@Base-Extrude@C...	back plate

Figure 15-3 The completed geometric and force design parameter definitions

15.3 Linking force or restraint parameters to the analysis model

The three geometric parameters are now related to the solid model, but we have not yet linked the two free force parameters (FCenter and FSide) to the finite element model. Before doing that the basic material properties and loads/restraints for the analysis, Figure 15-4, will be defined.

1. In the **CosmosWorks Manager** menu, **Solid→Edit/Define Materials→Steel→Allow steel**.
2. Right click on **Loads/Restraints→Restraints→On cylindrical face**. Select the top two cylindrical holes and set **Circumferential component** = 0. Either of those will eliminate a rigid body motion (**RBM**) about the vertical (z-) axis. The zero circumferential displacement around the full circle includes two points where the x-displacement will also be zero and two points with zero y-displacements. Thus, rigid body translations in x- and y- directions are prevented (now takes care of 3 of 6 RBM).
3. Right click on **Loads/Restraints→Restraints→On flat face**. Select the top surface of the back plate.
4. Set the **normal (z-) component** to zero, click **OK**. That will eliminate the possible RBM of a z-displacement, x-rotation, and y-rotation (6 of 6).

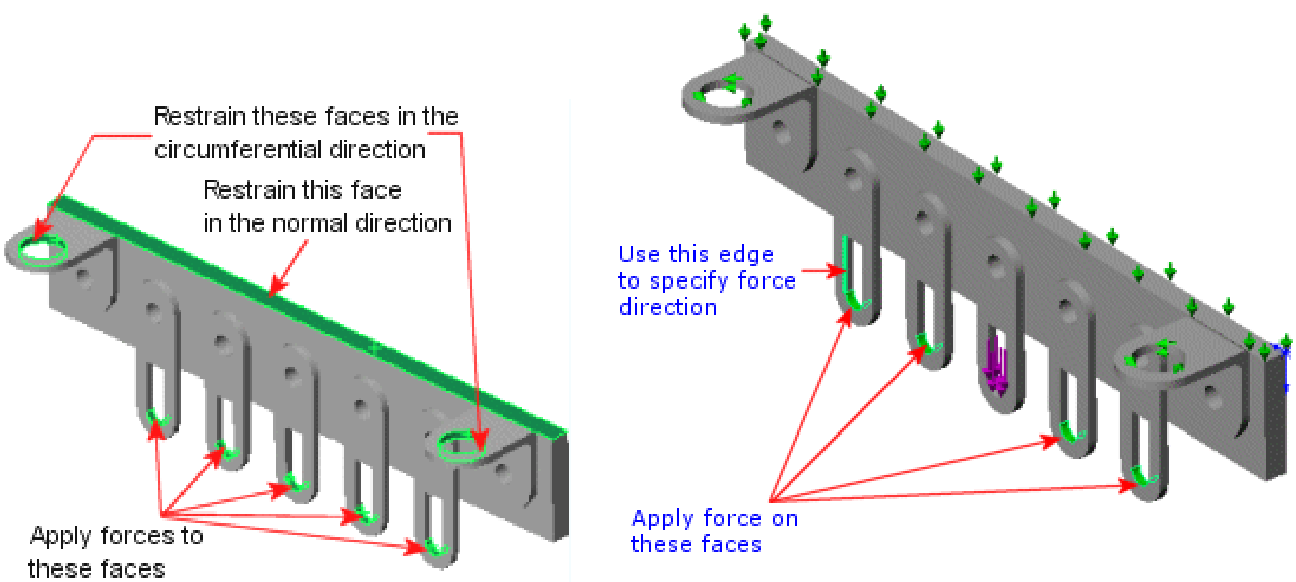
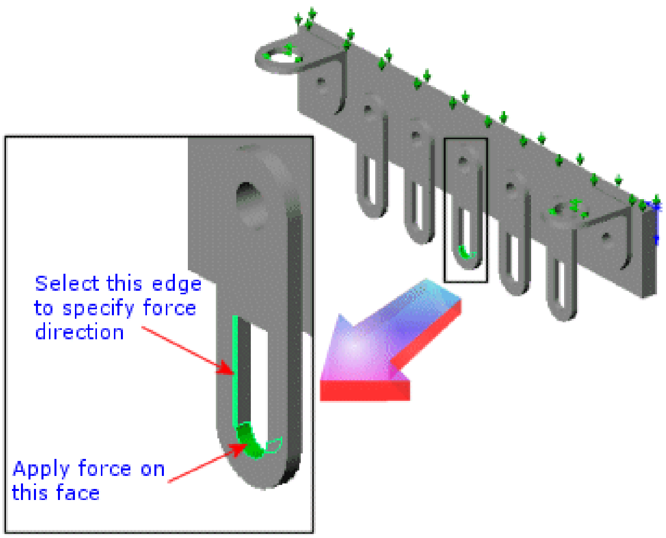


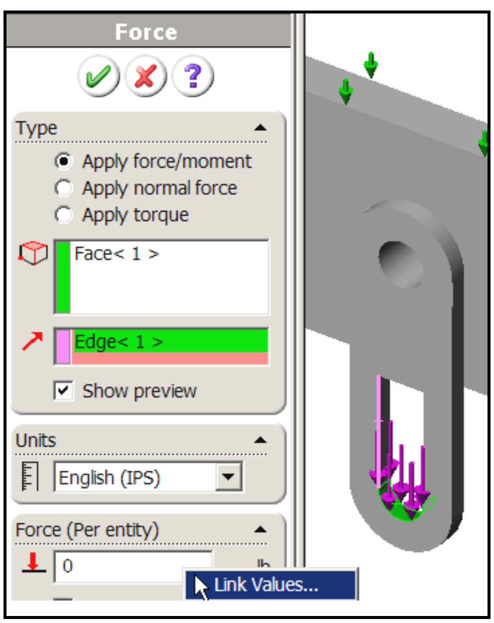
Figure 15-4 Loading and restraint regions

Now you are ready to specify the loadings and select some or all of them to be linked to the force design parameters. Begin with the center hanger:

1. Right click on **Loads/Restraints→Force** to open the **Force panel.**
2. **Type→Apply force/moment.**
3. Click inside the **Faces, Edges, Vertices for Force** box, then click the **cylindrical face** of the **central hanger**.

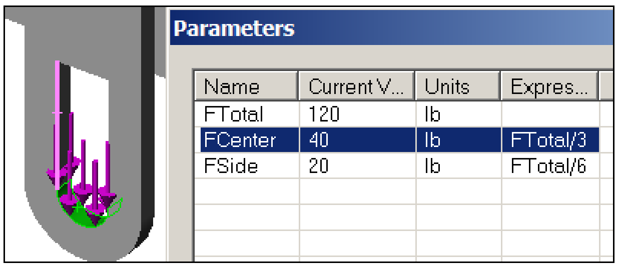


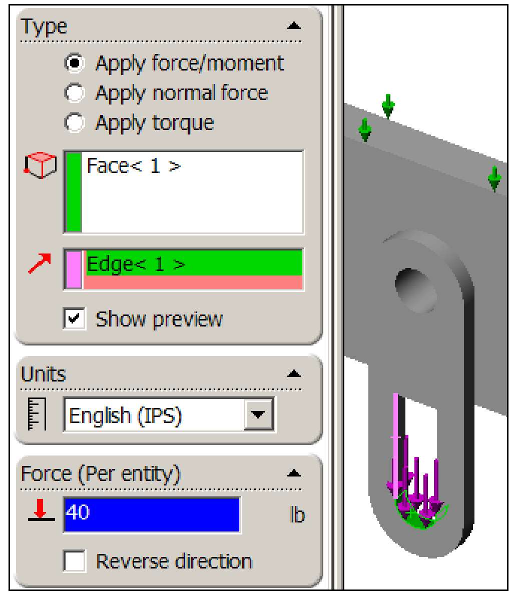
4. Click inside the **Face, Edge, Plane, Axis for Direction** box and select the **vertical edge** to indicate the direction (downward) desired for the force. Switch directions if the **preview** is up instead of down.
5. At this point you usually enter a numerical value for the force Value. Instead right click inside the **Value box** and select **Link Values**. The **Parameters panel** automatically appears.



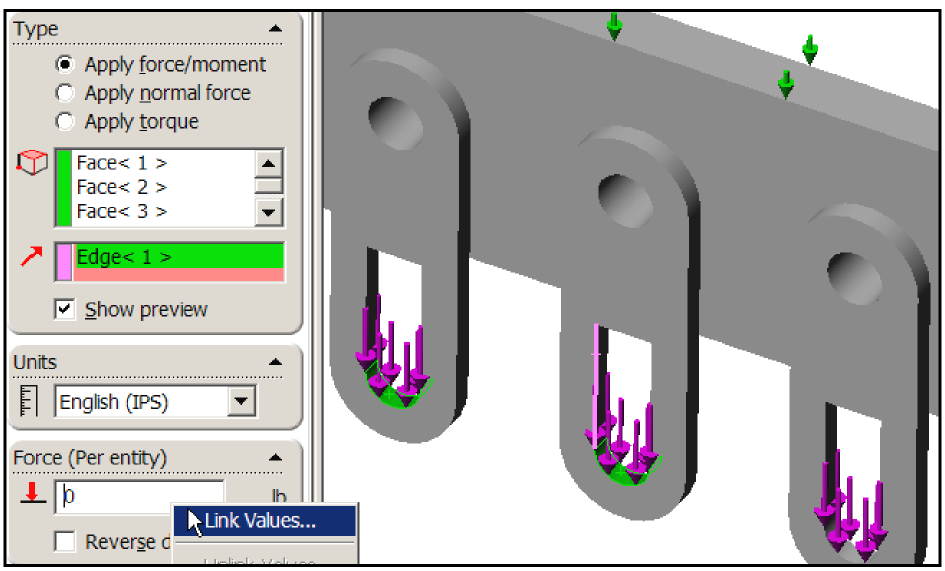
6. Select **FCenter** (second line in the list of parameter names). The background color of the **Value box** changes to **blue**, indicating that this field is **linked** to the **FCenter parameter** that you originally

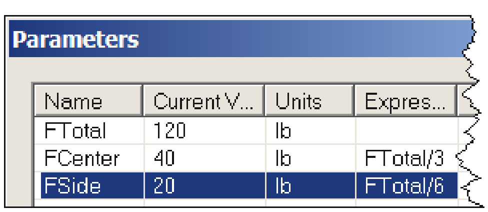
established. Note also that the current value of the **FCenter** parameter appears in the force **Value box**. Click **OK** to apply the force at the selected location.

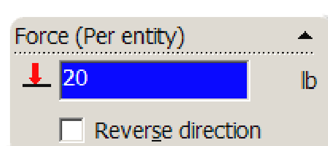




7. Repeat this process for the two pairs of brackets on each side of the center one. Right click inside the **Value box** and select **Link Values.** Select **FSide** and click **OK**. The **FSide** force is applied to each of the four side hangers.





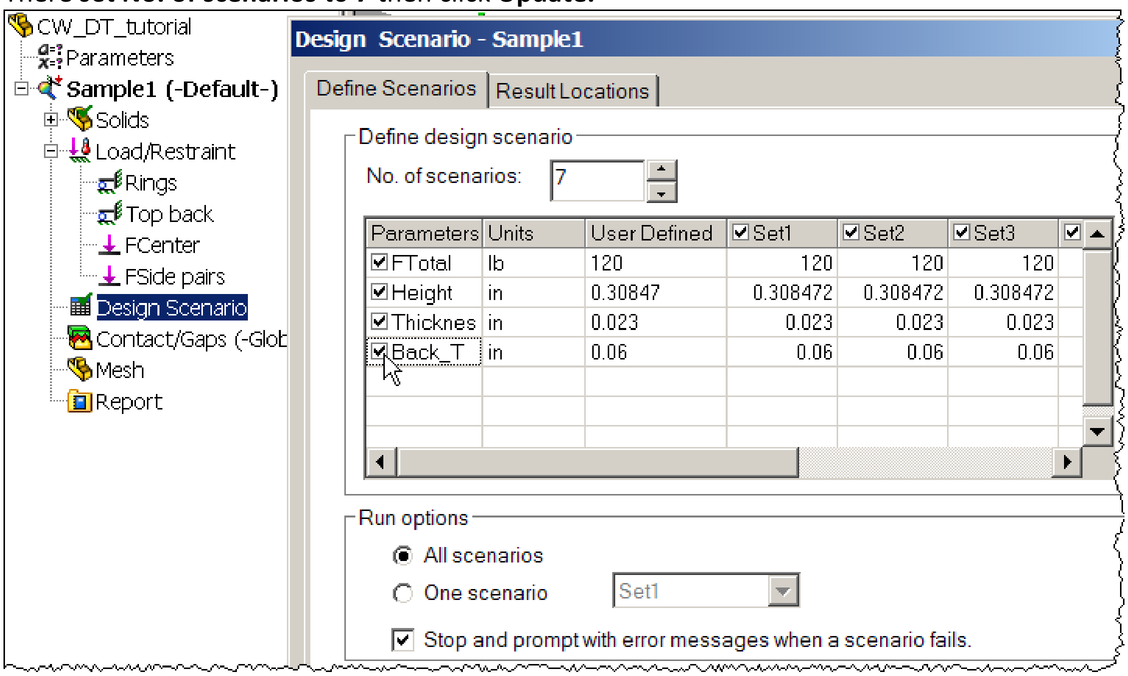


Now three parameters have been linked to the solid model (Height, Thickness, Base_T), two have been linked to the analysis model (FCenter, FSide), and the last, FTotal has been assigned an initial value in the Parameters panel. At this point you can specify all the different combinations that you want to examine in a series of design scenarios.

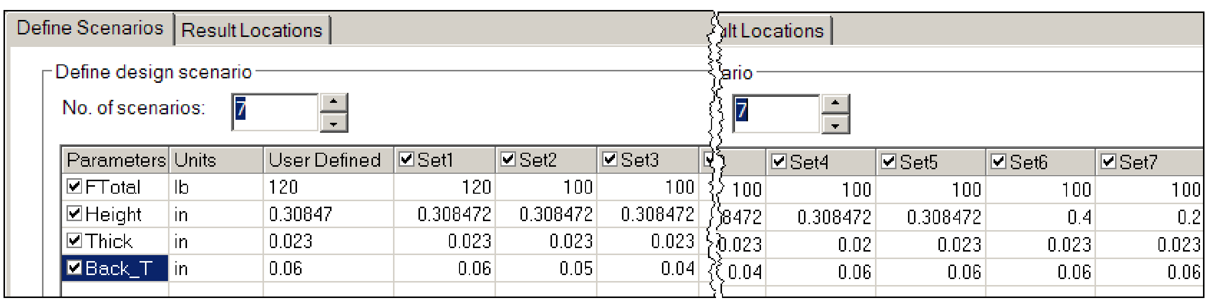
15.4 Defining design scenarios

In this procedure you define seven design scenarios. Each design scenario is a set of values given to the parameters you defined above; FTotal, Height, Thickness, and Back_T:

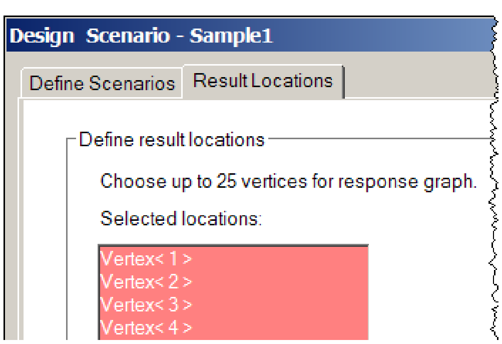
1. Double-click the **Design Scenario** to open the **Design Scenario panel**.
2. There set **No. of scenarios** to **7** then click **Update.**

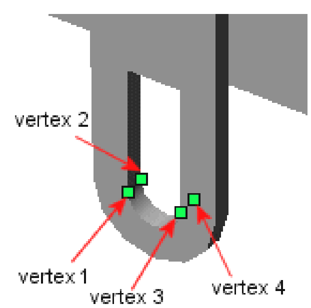


3. In the parametric set table, select all 4 parameters by **check**ing the box by their **names**. Enter parametric values for each of the seven scenarios you wish to define.



4. At **Run options**, pick **All scenarios** and **Stop and prompt with error messages when a scenario fails.**
5. Open the **Result Locations** tab to select the locations for result outputs. In the graphics area, click the four vertices of the central hanger. They appear in the **Selected locations** list box. Click **OK**.





Central Hanger result locations

15.5 Running the design scenarios

Each of the design scenarios will require a mesh. Thus you should select your mesh control options and create an initial mesh. An initial mesh is not requires but it can serve as a check on the initial element sizes of the mesh you may need. Clearly the results will depend on the fineness of the mesh in important regions. Also, the run time goes up with the mesh fineness. A typical mesh is seen in Figure 15-5.

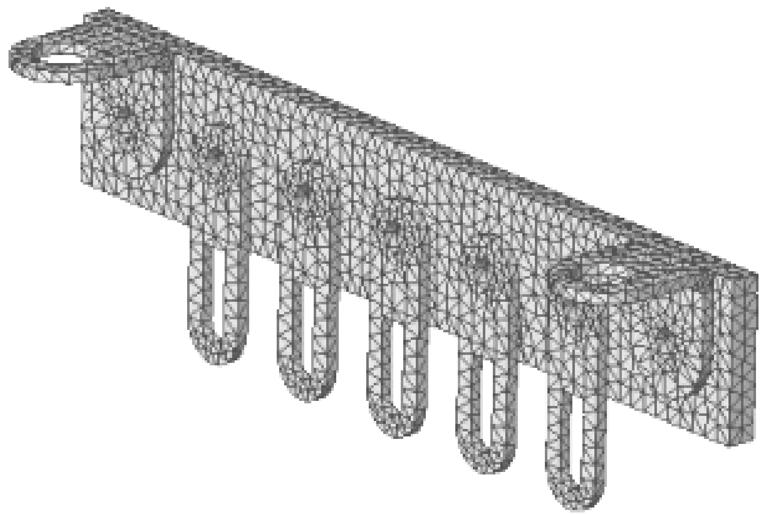
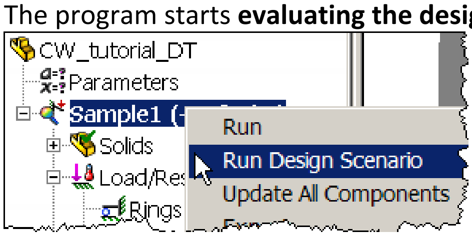


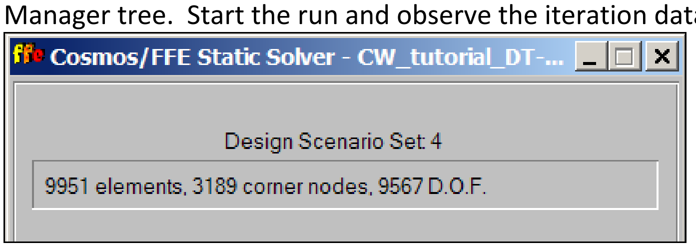
Figure 15-5 A typical mesh of the assembly

At this point a check mark appears on the **Design Scenario** icon to indicate that all data required for a design scenario have been completed. To run these design scenarios:

1. In the COSMOSWorks Manager menu, right-click the **Basic Study** icon and select **Run Design Scenario**.
2. The program starts **evaluating the design** scenarios one at a time



3. When analysis is completed, a **Design Scenario Results** folder is created in the COSMOSWorks Manager tree. Start the run and observe the iteration data for each scenario.

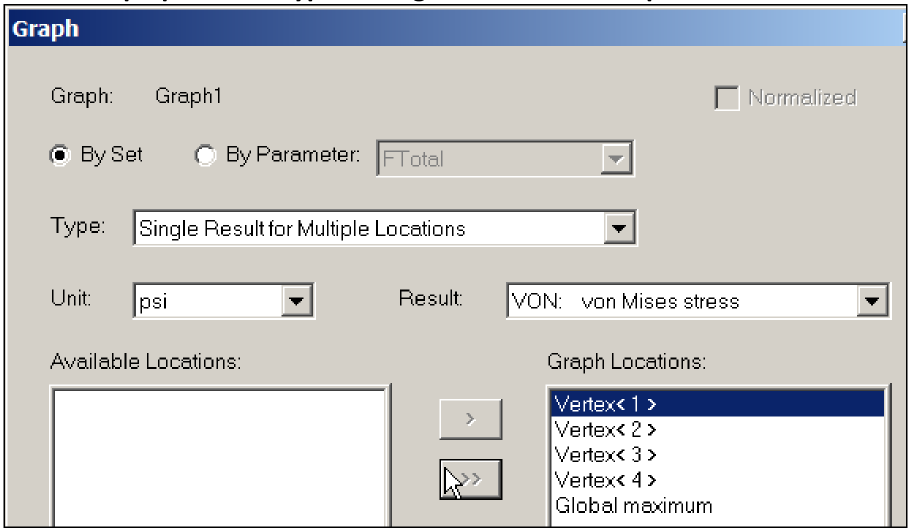


15.6 Post-processing the scenario results

Now you can graph and/or list the results from each scenario at any of the selected output locations. For example, you can graph the von Mises stress at all specified locations for all design scenarios:

1. In the COSMOSWorks **Manager tree**, click the plus sign beside the **Design Scenario Results** folder. Graph1 appears.
2. Right-click **Graph1** icon and select **Edit Definition**. The **Graph** dialog box appears.

3. In the **Graph panel** set **Type** to **Single Result for Multiple Locations**.



4. Set **Units** to **psi**. Set **Result** to **VON: von Mises stress**.
5. Click the double arrow (>>) to move all items from **Available Locations** list box on the left to the **Graph Locations** list box on the right.
6. Click **OK.** At this point the Von Mises graph appears as given in Figure 15-6.

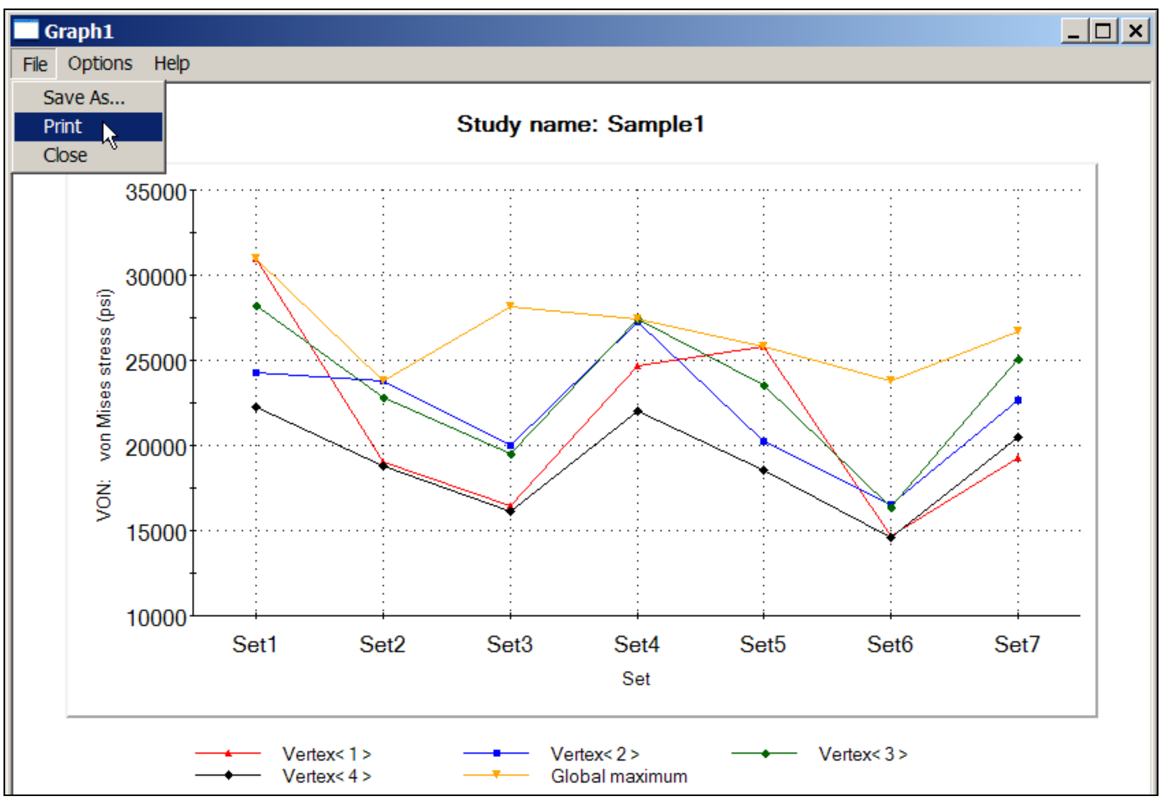


Figure 15-6 Von Mises failure criterion for each set location

A graph can be enlarged or reshaped by grabbing a corner and moving it around. There are options to print the graph or save it as a file, etc. It is wise to check the principle stresses as well (and required for brittle materials). Therefore the **Graph** panel is edited to select principle stresses P1, P2, P3, move them to the right under graph results (see Figure 15-7) and get the additional plot shown in Figure 15-8.

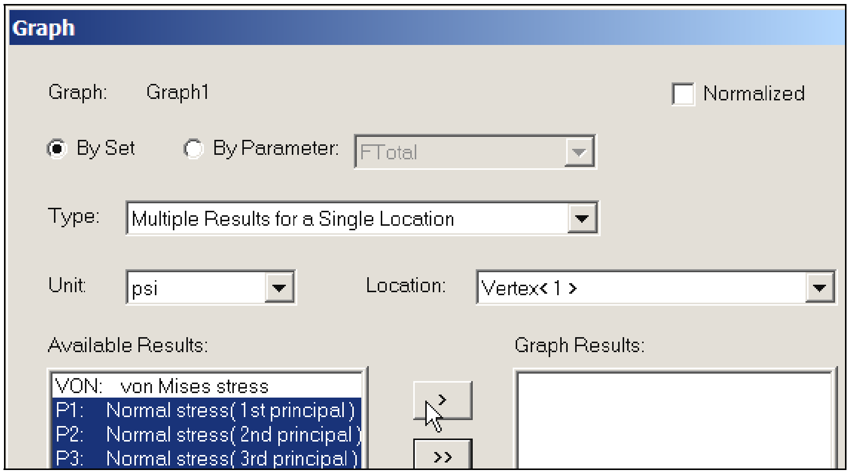


Figure 15-7 Picking the principle stresses from available results

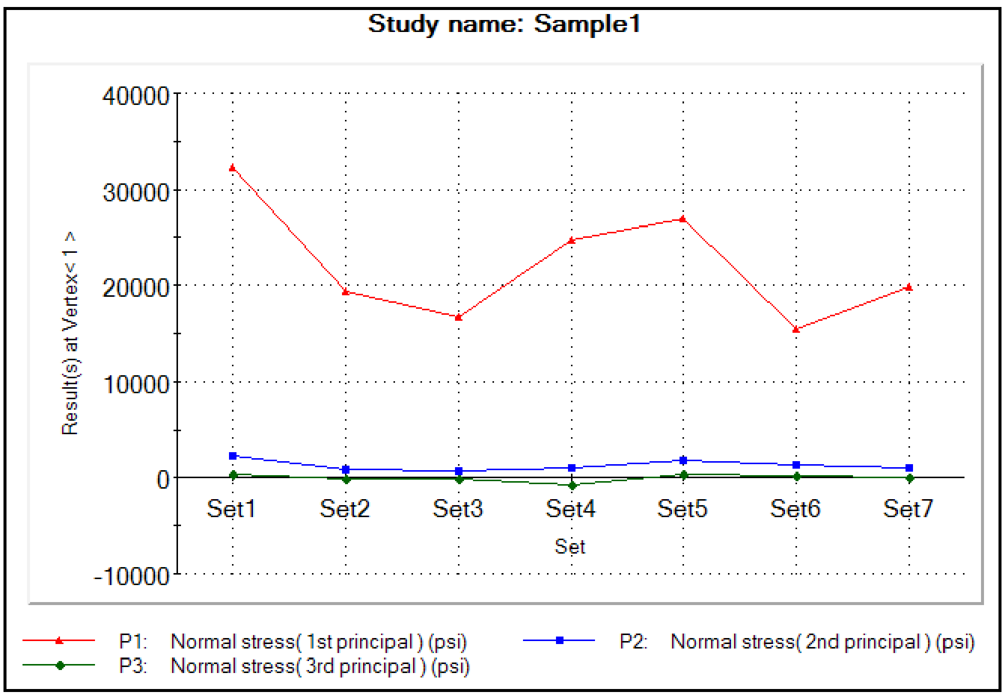


Figure 15-8 Principle normal stress results

15.7 Listing results

You can also get a listing of these results in the CosmosWorks Manager tree, by right-clicking the **Design Scenario Results → Show Summary.** A summary of results appears as illustrated in Figure 15-9. The summary includes the following sections:

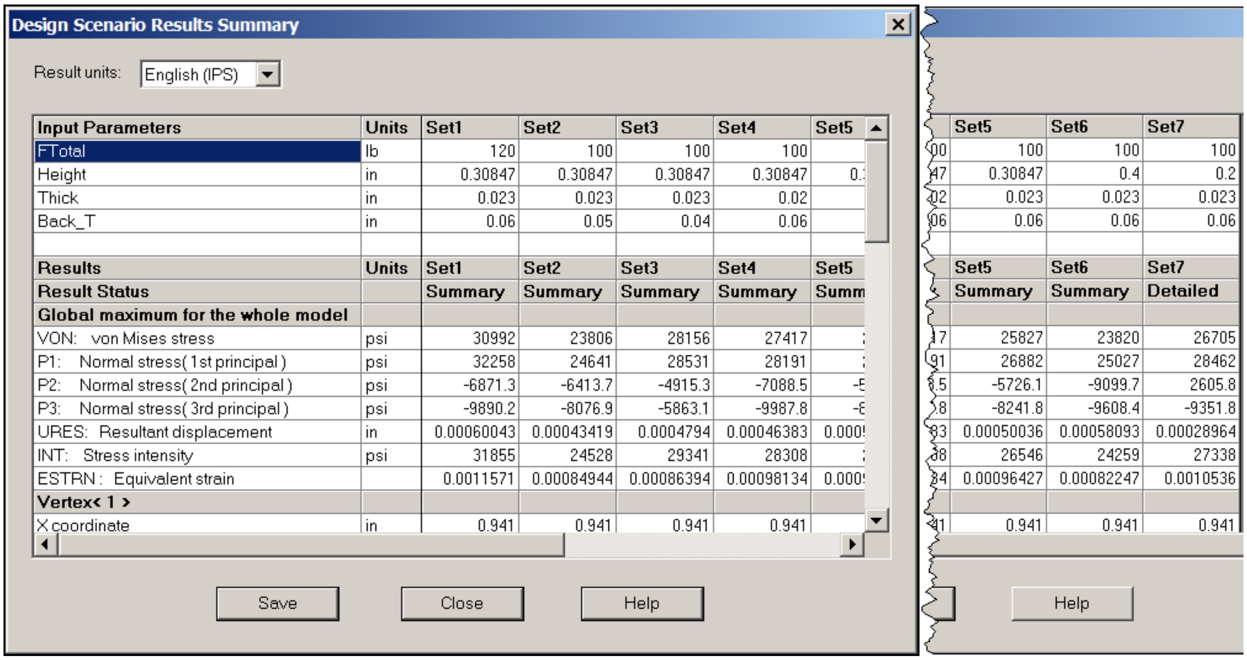
Input Parameters: lists the definition of design scenarios.

Result Status: lists the status of the results. Sets 1 through 6 show **Summary** indicating that only summary results are currently available for these sets. Set7 shows **Detailed** indicating that detailed results are available for this set since it is the last that the program evaluated. The results folders in the CosmosWorks Manager tree correspond to set 7.

Global Maximum for Whole Model: Lists the extreme values of of von Mises and principal stresses, resultant displacement, and the equivalent strain.

Vertex (i): Lists the global coordinates of the vertex and the values of von Mises and principal stresses, resultant displacement, and the equivalent strain at location (i). Four locations are listed since you selected four vertices for result locations.

The format for the graphs in Figure 15-6 and Figure 15-8 gives a feel for how the maximum and minimum stresses vary, but not how sensitive they were to the parametric geometry, or loads. Of course, for any linear problem the displacements and stresses are directly proportional to the loads. Therefore, you can usually get more insight about what is most important in a design by seeing how changes in the geometric parameters, or support parameters affect the results. That is usually referred to as a sensitivity study.



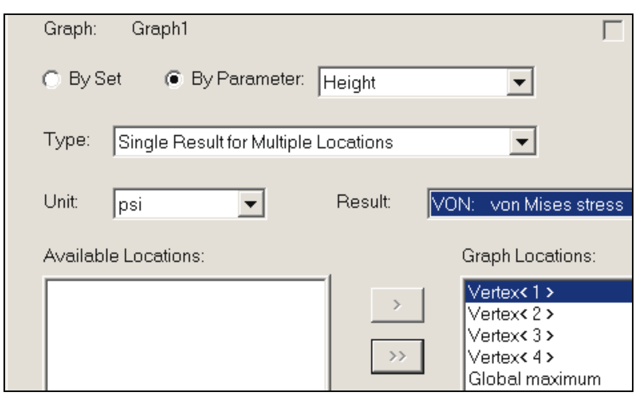
Input Parameters	Units	Set1	Set2	Set3	Set4	Set5	Set6	Set7
FTotal	lb	120	100	100	100	100	100	100
Height	in	0.30847	0.30847	0.30847	0.30847	0.30847	0.4	0.2
Thick	in	0.023	0.023	0.023	0.02	0.023	0.023	0.023
Back_T	in	0.06	0.05	0.04	0.06	0.06	0.06	0.06
Results	**Units**	**Set1**	**Set2**	**Set3**	**Set4**	**Set5**	**Set6**	**Set7**
Result Status		Summary	Summary	Summary	Summary	Summary	Summary	Detailed
Global maximum for the whole model								
VON: von Mises stress	psi	30992	23806	28156	27417	25827	23820	26705
P1: Normal stress(1st principal)	psi	32258	24641	28531	28191	26882	25027	28462
P2: Normal stress(2nd principal)	psi	-6871.3	-6413.7	-4915.3	-7088.5	-5726.1	-9099.7	2605.8
P3: Normal stress(3rd principal)	psi	-9890.2	-8076.9	-5863.1	-9987.8	-8241.8	-9608.4	-9351.8
URES: Resultant displacement	in	0.00060043	0.00043419	0.0004794	0.00046383	0.00050036	0.00058093	0.00028964
INT: Stress intensity	psi	31855	24528	29341	28308	26546	24259	27338
ESTRN : Equivalent strain		0.0011571	0.00084944	0.00086394	0.00098134	0.00096427	0.00082247	0.0010536
Vertex< 1 >								
X coordinate	in	0.941	0.941	0.941	0.941	0.941	0.941	0.941

Figure 15-9 Tabulated summaries of all design scenario results are available

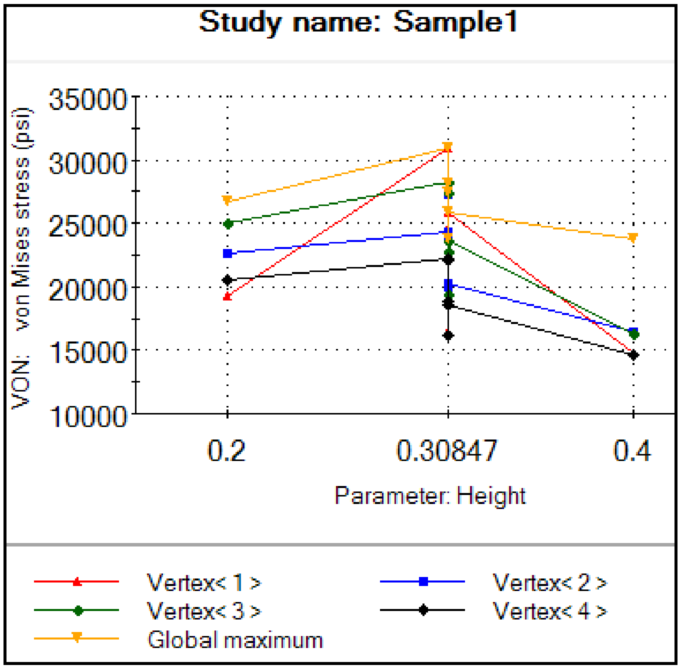
15.8 Sensitivity review

To see how the geometric height and thickness affect some of the stress results you need to change the graph mode to let those design parameters vary along the horizontal axis of the graph:

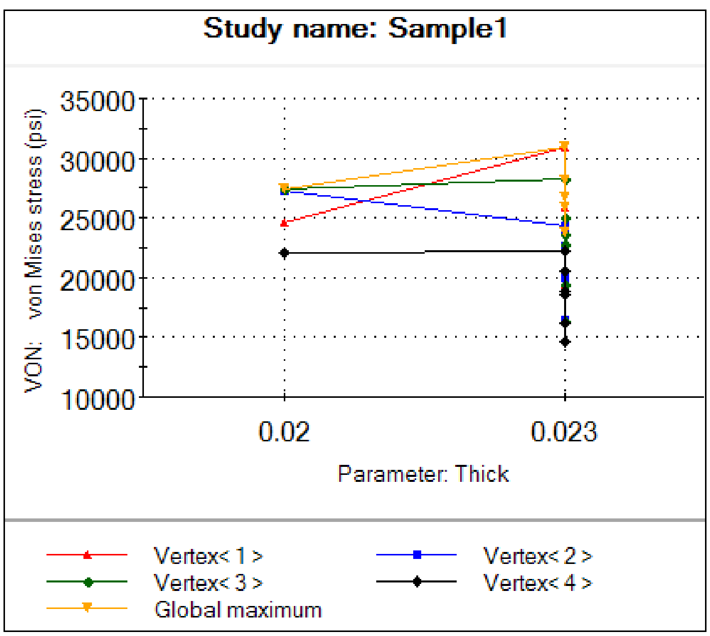
1. **Right-click Graph1** icon and select **Edit Definition**. The **Graph** dialog box appears.



2. In the **Graph** panel set **Type** to **Single Result for Multiple Locations**.
3. Set **Units** to **psi**. Set **Result** to **VON: von Mises stress**.
4. Uncheck the By Set button and check the **By Parameter** button.



5. Scroll down and select **Height** for the independent variable to plot.
6. Click the double arrow (**>>**) to move all items from **Available Locations** list on the left to the **Graph Locations** list on the right. Click **OK.** The **new graph** appears



The small range of height values occurs because that was the range cited in the seven trials. The slope of the lines indicates which locations have von Mises stress values that are sensitive to changing the height. Carrying out a similar process using the hanger thickness as a design parameter yields the stress resultsin the last figure above.

15.9 Discussion

The last two graphs are just shown to illustrate the process. Here the loads on the hanger are in the plane of the hanger. Therefore they cause mainly axial stress like σ = Force/Area. In other words, you expect stress locations near the hanger to depend mainly on the hanger thickness and not its height. If the loads had been perpendicular to the hanger you would get more complicated bending stresses where the hanger length is the lever arm for the bending moment. Likewise, the local maximum bending stress would be inversely proportional to the hanger thickness squared.

Also in this problem the meaning of hanger thickness is unclear because the hanger is fully welded to the back plate. So, in some places by hanger thickness you mean parameter “Thick” while in others you mean “Thick + Back_T”. We know the analysis *assumed* the plate and hanger are welded because the assembly fully bonded touching surfaces as the default gap condition. Also, no restraint condition (like a bolt connection) was given at the top of the hangers even though their geometry might imply that they would be bolted instead of welded. You should always note such implied assumptions in your reports so that you and others can check or revise them.

16 Related Analogies

16.1 Basic Concepts

The differential equation used in a finite element study in one discipline often appears in a different discipline, but with a different physical meaning for the unknown and the coefficients in the equation. That is particularly true for the diffusion equation (heat transfer here) and the biharmonic equation (flat plate deflection here). They are the most common second order and fourth order differential equations in engineering. Consider the slightly generalized 2D field equation, in the solution domain:

$$k_x \frac{\partial^2 \varphi}{\partial x^2} + k_y \frac{\partial^2 \varphi}{\partial y^2} + P = 0$$

Subject to a Dirichlet boundary condition on boundary segment Γ_D of

$$\varphi = \varphi_{given}$$

or a Neumann boundary condition (known normal flux) on boundary segment Γ_N of

$$k_n \frac{\partial \varphi}{\partial n} = f_{given}$$

or a Convection (Robin) boundary condition on boundary segment Γ_R of

$$k_n \frac{\partial \varphi}{\partial n} = h(\varphi - \varphi_\infty) = h\varphi + g$$

where the boundaries do not overlap. The meanings of the above symbols, in a few disciplines, are listed in Table 16-1. These analogies allow you to use Cosmos to solve problems in such fields by replacing the Cosmos inputs with corresponding values (and units) for the field of interest. You should also edit the graphic outputs to show the desired terminology (as done here with the SnagIt software).

Table 16-1 Some general field equation terms

Field	φ	k	P	h	φ_∞	f
Heat Transfer	Temperature	Thermal Conductivity	Heat Power	Convection Coefficient	Convection Temperature	Boundary Flux
Electric Conduction	Voltage	Electrical Conductivity	Current Source	0	0	Boundary Current
Porous Media Flow	Hydraulic Head	Hydraulic Conductivity	Flow Source	0	0	Boundary Flow
Irrotational Flow	Velocity Potential	1	Flow Source	0	0	Boundary Velocity

16.2 Seepage under a dam

Water in a reservoir will almost always seep under and/or around a dam. It needs to be controlled to have a low velocity so that the region around the dam will not be eroded away. This seepage, or porous media flow, field will be illustrated for a dam resting on layered, and thus orthotropic, soils [13]. The dimensions of the soil regions, dam, and toe wall are given in Figure 16-1. The left side of the dam holds water 30 m deep, while the right side is 1 m deep. Split lines locate the impervious dam interface at the soil top. The far boundaries of the soil are also impervious (no normal flow, f = 0, the natural). The layer soil permeabilities (or hydraulic conductivities) are 20 m/day and 15 m/day in the horizontal and vertical directions, respectively. Those two orthotropic properties are specified with respect to the Front Plane and input as seen in Figure 16-2. Usually layered soils are inclined and require the use of a reference plane to define the principal material directions.

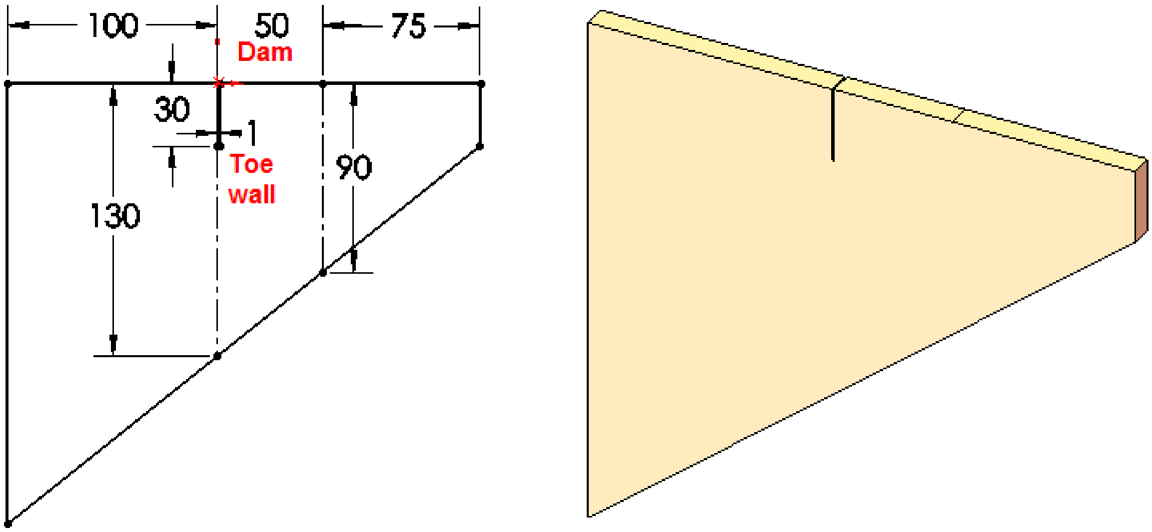


Figure 16-1 Orthotropic soil beneath a dam

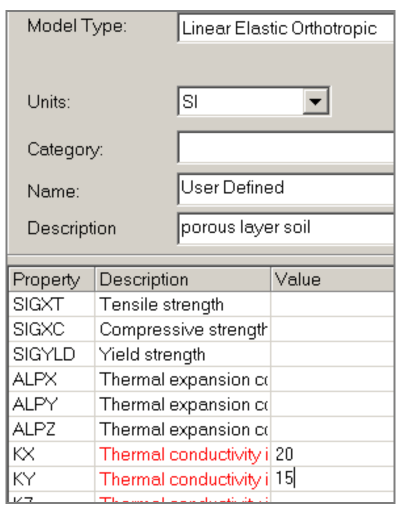


Figure 16-2 Input soil permeabilities in the Front Plane

16.2.1.1 Restraints

The constant hydraulic head on either side of the dam are like specified temperatures. Of course, the chosen unit of *Celsius* here represents *m* (meters of water). Those two essential restraints are seen in Figure 16-3.

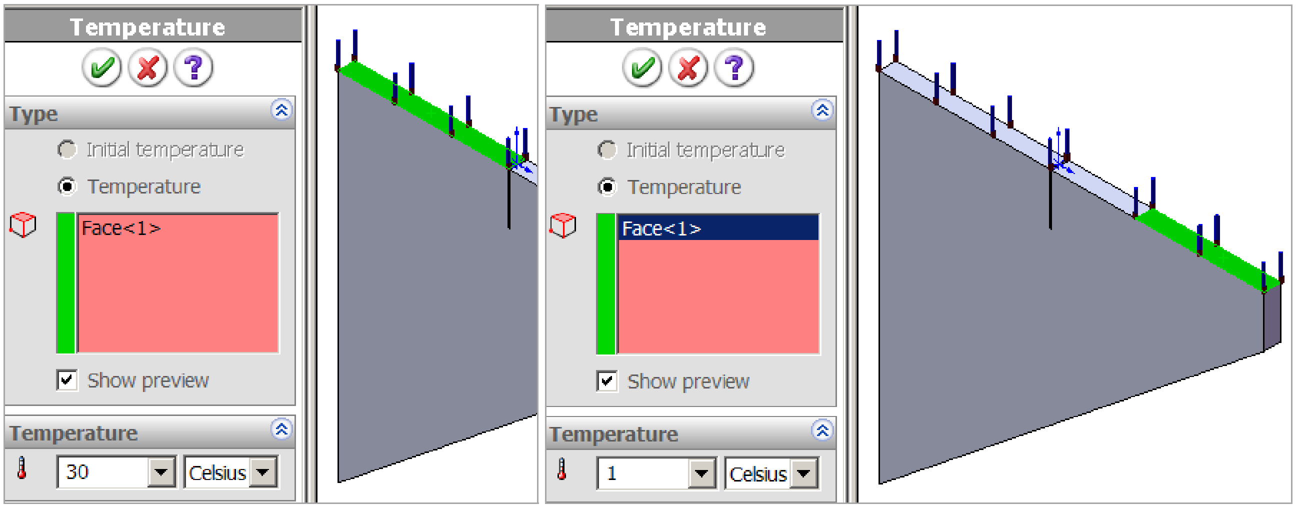


Figure 16-3 Set water pressure boundary values

16.2.1.2 Mesh and execute

The very narrow toe wall constructed at the front of the dam causes a sharp re-entrant corner in the soil (almost a crack). That means very high gradients (velocities) will occur there. Therefore, it is necessary to invoke mesh control there to force small elements at the base of that wall. A portion of the soil mesh is given in Figure 16-4. Now you can **Run** the study. The resulting soil pressures are given in Figure 16-5.

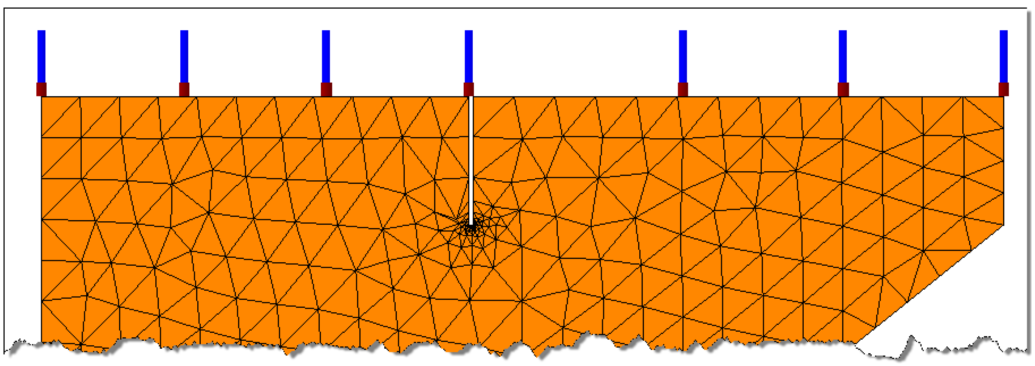


Figure 16-4 Refined mesh around the toe wall tip

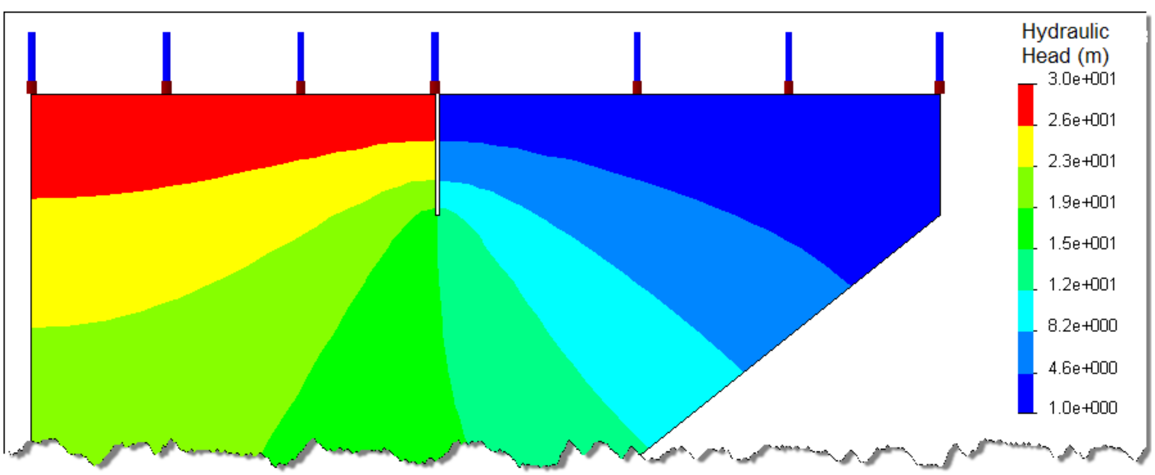


Figure 16-5 Hydraulic head (water pressure) in the soil, with edited color bar

16.2.1.3 Seepage velocity vectors

To see the velocity vectors, just select the Cosmos heat flux vector plot and re-label the color bar to display the units of m/day (with SnagIt, etc.). As desired, the velocities are quite small through the soil. The largest vales occur where the water changes directions from down to up around the toe wall tip (see Figure 16-6).

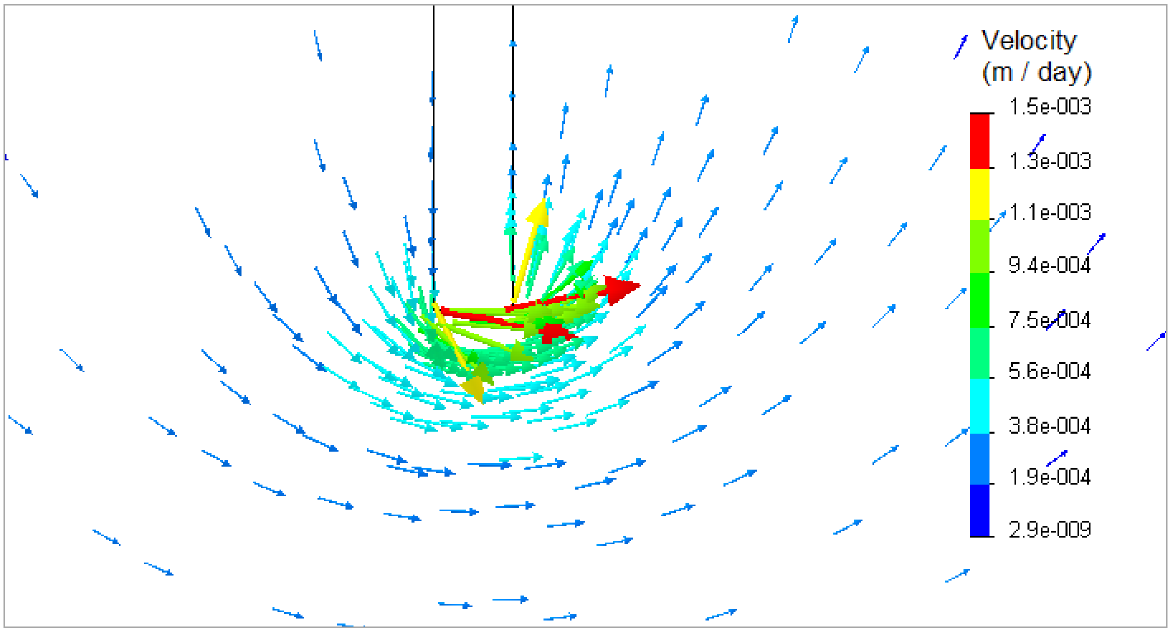


Figure 16-6 Seepage velocities at the toe wall tip

16.3 Potential flow around a cylinder

Consider the Irrotational flow of an ideal fluid around solid cylinder within a rectangular channel dimensioned as shown in Figure 16-7. Its properties are unity as given in Figure 16-1. The fluid enters at the left with a constant normal velocity of 5 cm/sec, and exits at the right with the same speed in order to conserve mass. That means that only Neumann boundary conditions are required in theory to determine the value of the velocity potential to within an arbitrary constant. In practice, due to machine accuracy slightly violating mass conservation, you should pick one point to assign an arbitrary value to the potential. Here, use the top wall point centered over the cylinder (Figure 16-8).

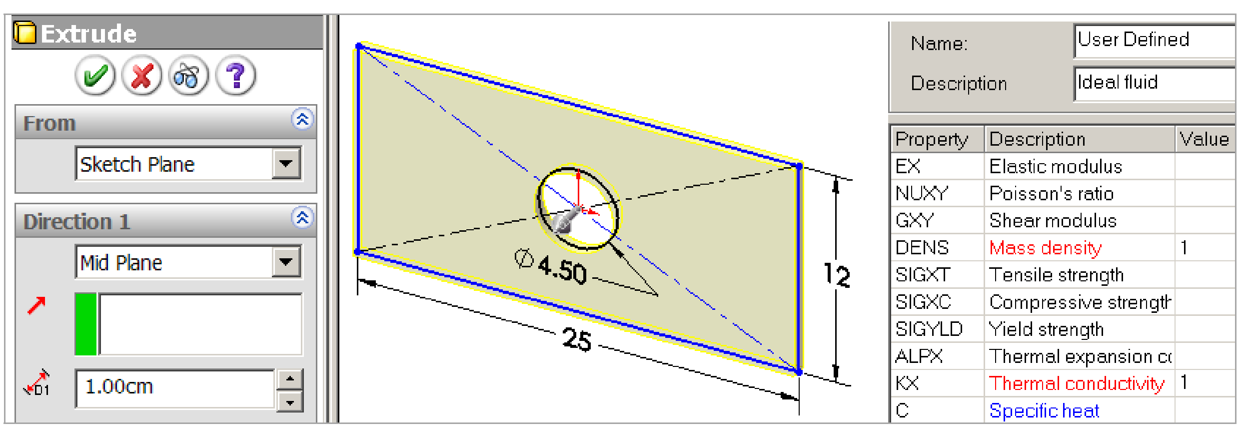


Figure 16-7 Fluid around a solid cylinder in a rectangular channel

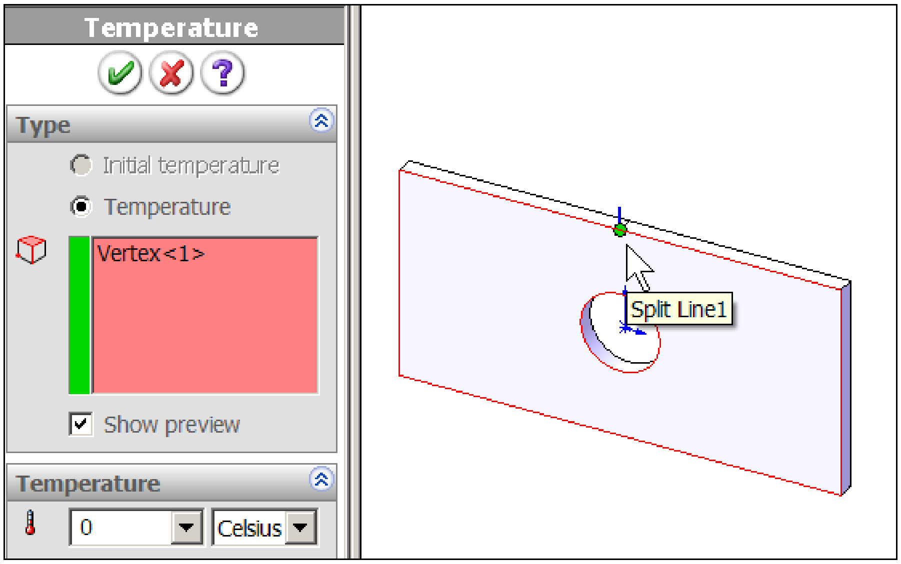


Figure 16-8 Specify the potential at an arbitrary point

16.3.1 Inflow and outflow boundary sources

For potential flow, a velocity inward across a boundary is negative. The sources must satisfy mass conservation. Since the length of the outlet is the same as the inlet only the sign changes at the right end outflow. Those two flow loads are illustrated in Figure 16-9. The mesh, in Figure 16-10, was controlled to be finer where the velocities are expected to change rapidly around the cylinder.

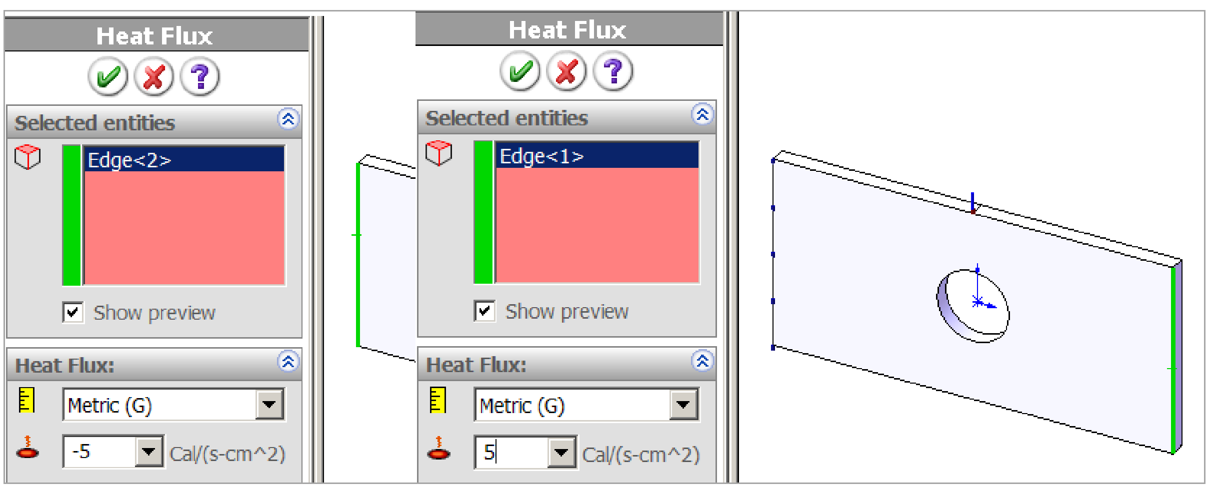


Figure 16-9 Inflow and outflow boundary restraint (cm/sec)

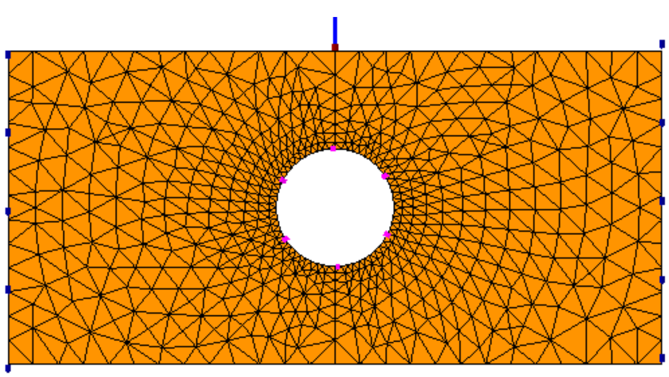


Figure 16-10 Graded mesh around the cylinder

16.3.2 Analysis results

The primary unknown, velocity potential, does not have a physical meaning but its value shown in Figure 16-11 confirms the expected anti-symmetric distribution. The velocity magnitudes and vectors are given in Figure 16-12 and Figure 16-13, respectively.

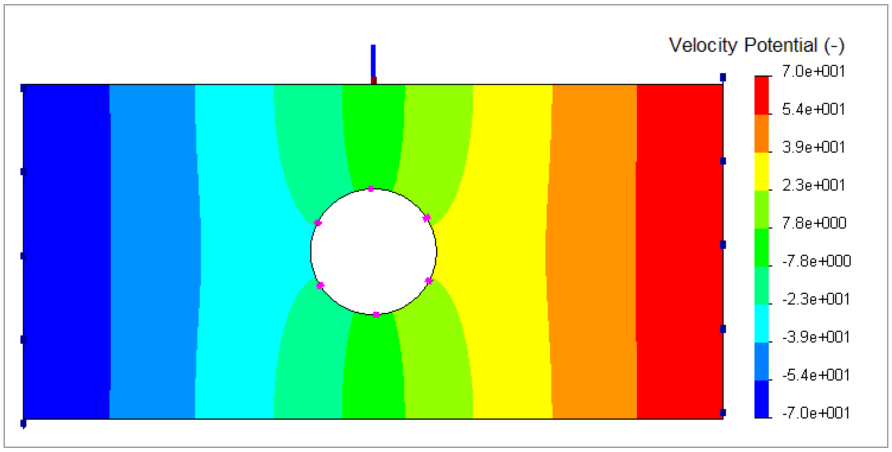


Figure 16-11 Anti-symmetric velocity potential around the cylinder

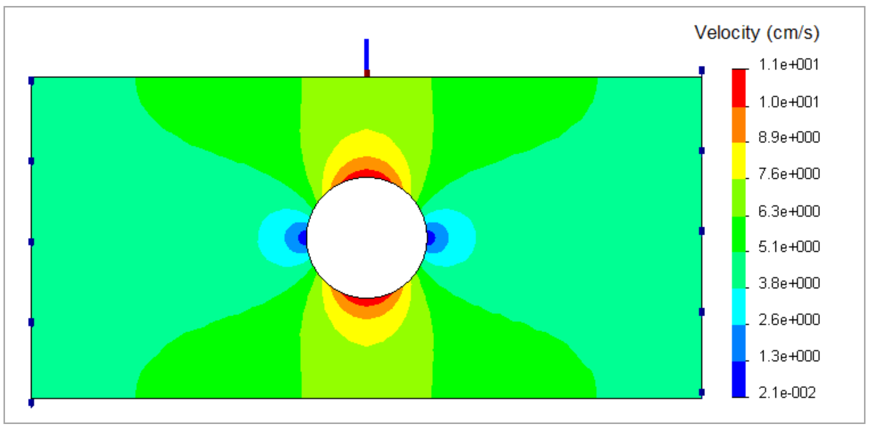


Figure 16-12 Fluid speed with anti-symmetric boundary flows

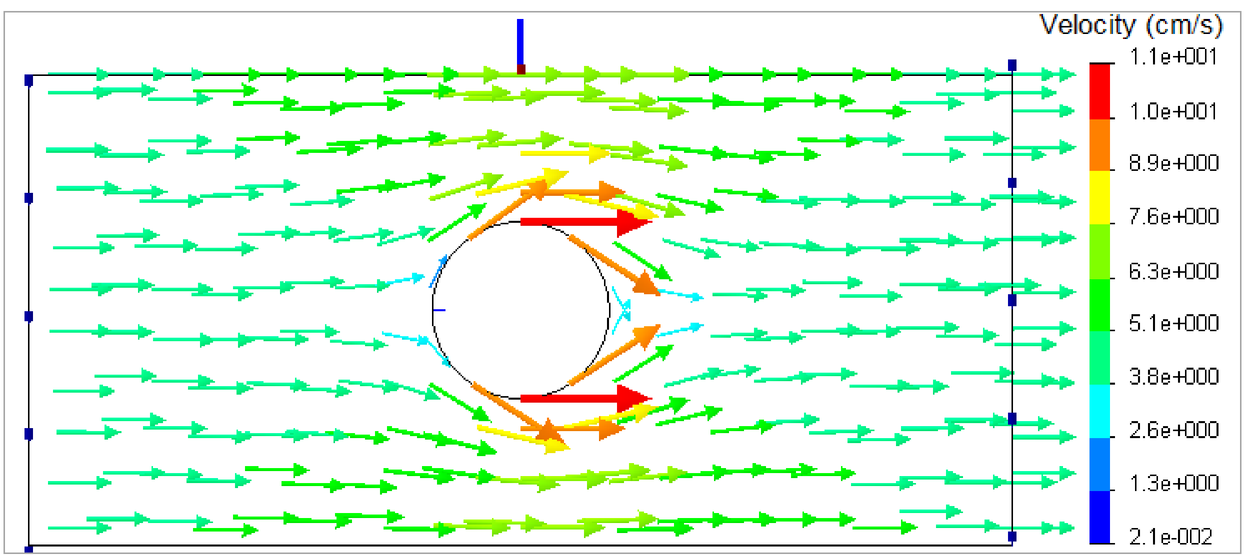


Figure 16-13 Velocity vectors with anti-symmetric boundary flows

16.3.3 Alternate outflow region

A considerably different result is obtained for other outlets. Here, half of the top channel edge is utilized. To conserve mass the normal flow component must be reduced, in Figure 16-14. That produces the new velocity vectors of Figure 16-15.

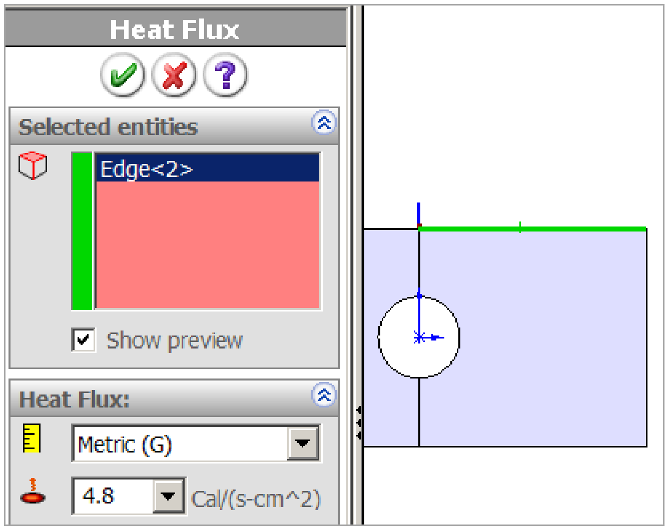


Figure 16-14 Modify the normal outflow direction and value, cm/sec

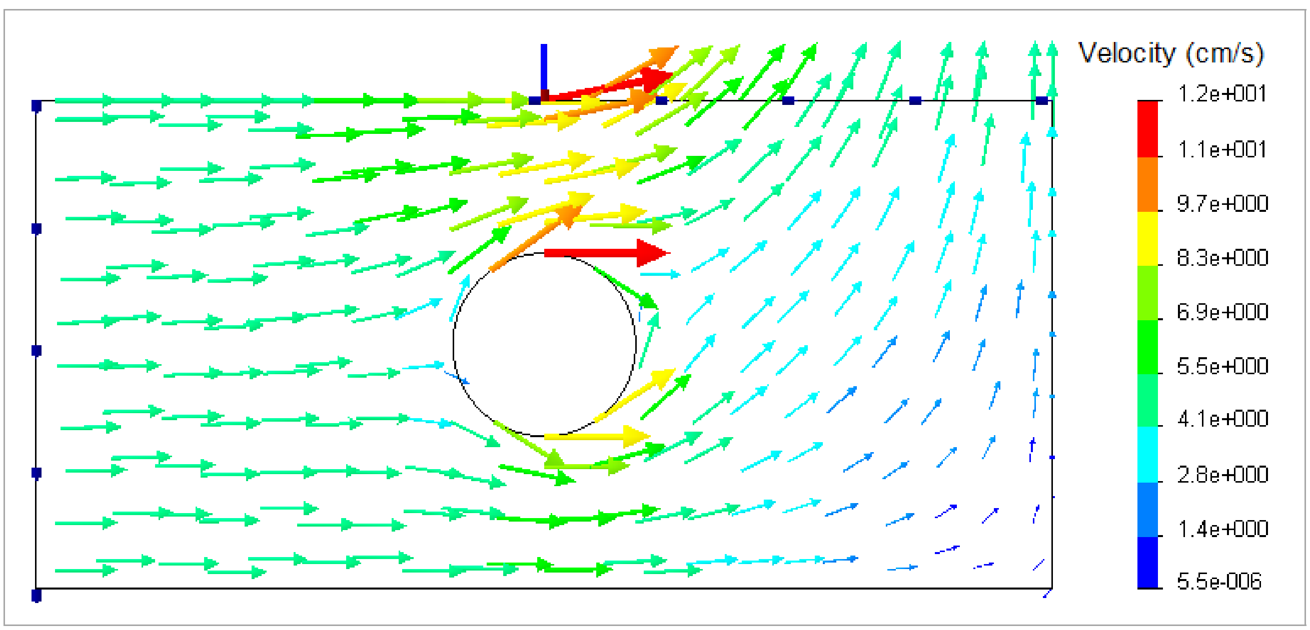


Figure 16-15 Velocity vectors for side outflow

16.4 Closure

Be alert for analogies that can extend the power and usefulness of you finite element software. Many commercial systems offer specific input and output interfaces for the alternate disciplines, but the underlying numerical calculations are basically the same. Minor exceptions are the torsional analogy and the pressurized membrane analogy which both utilize the integral of the solution additional as partial output.

17 Appendix A: Example Parts Construction

17.1 Construction of the Zee-beam solid model

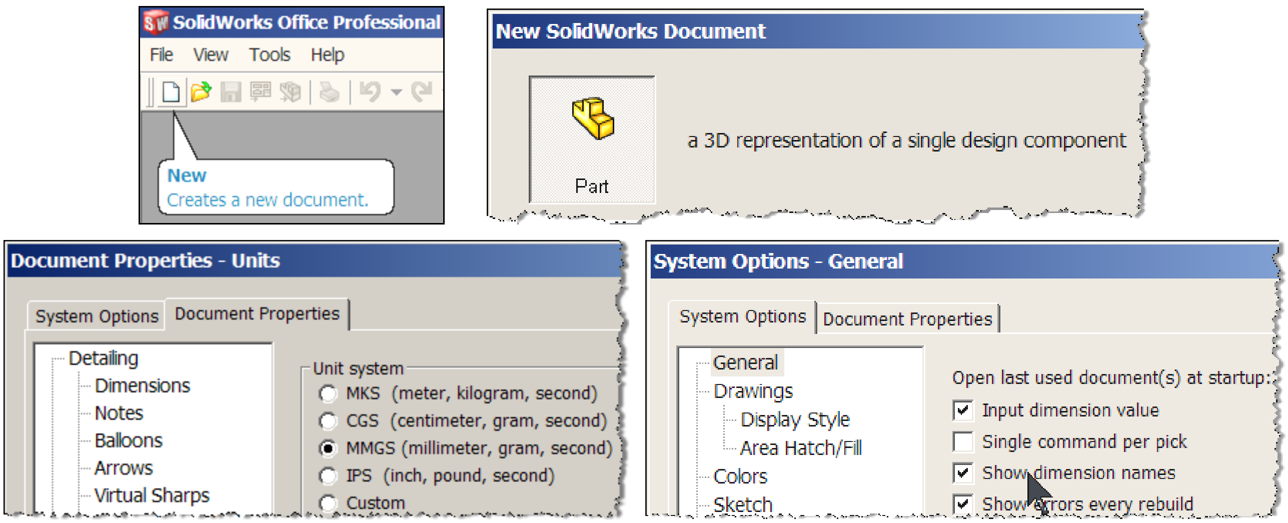


Figure 17-1 Selecting model units and dimension name display

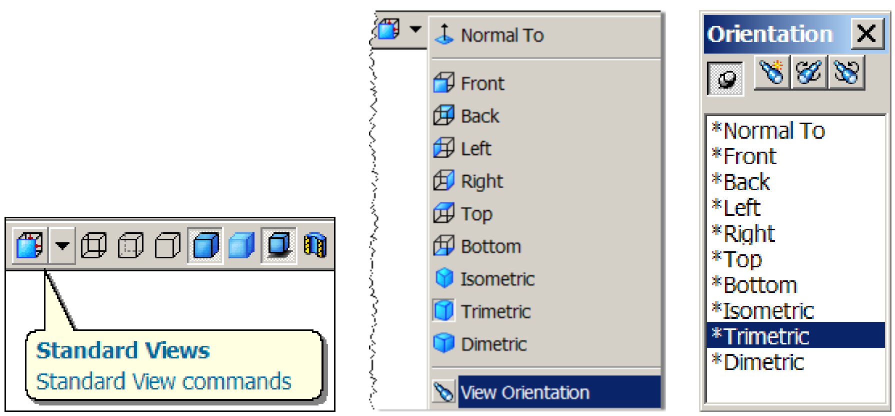
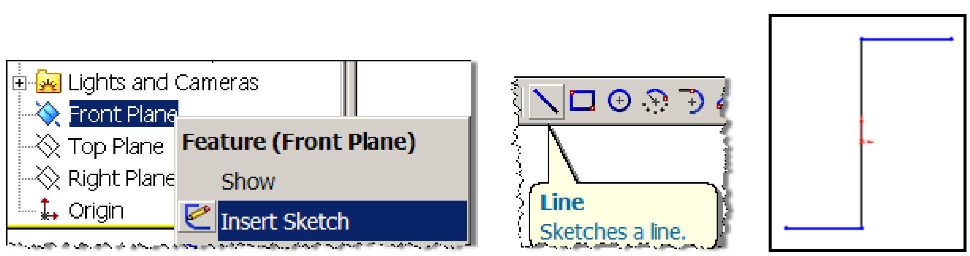


Figure 17-2 Pinning the view orientation panel in the graphics window



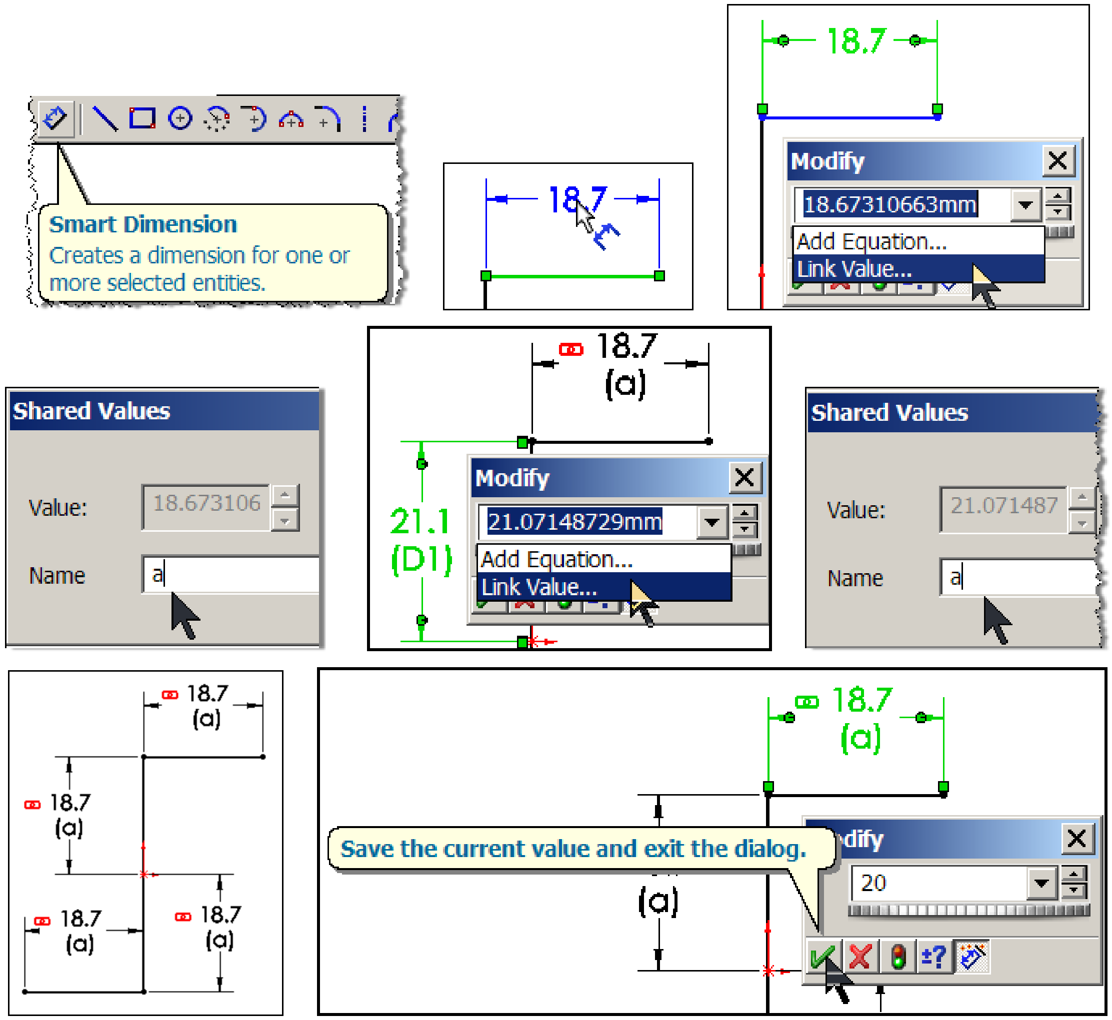


Figure 17-3 Linking shared values for dimensions

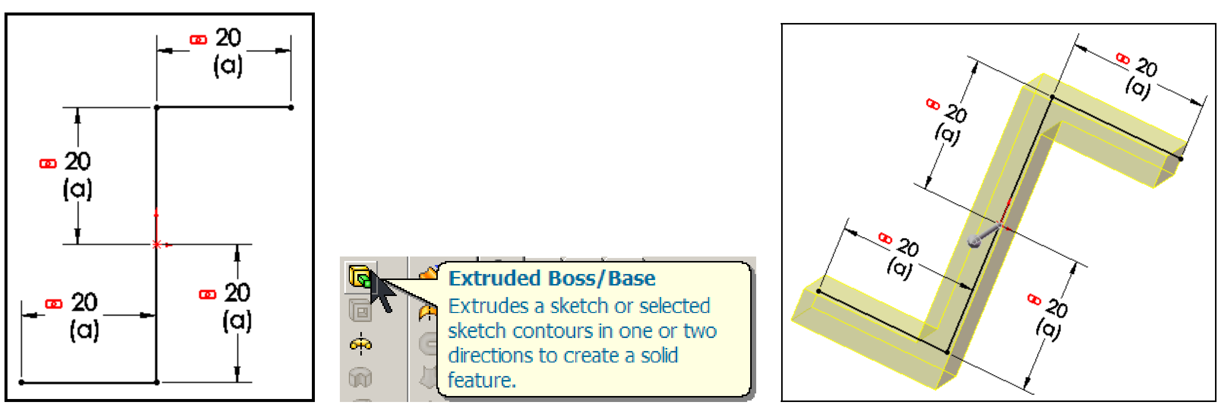
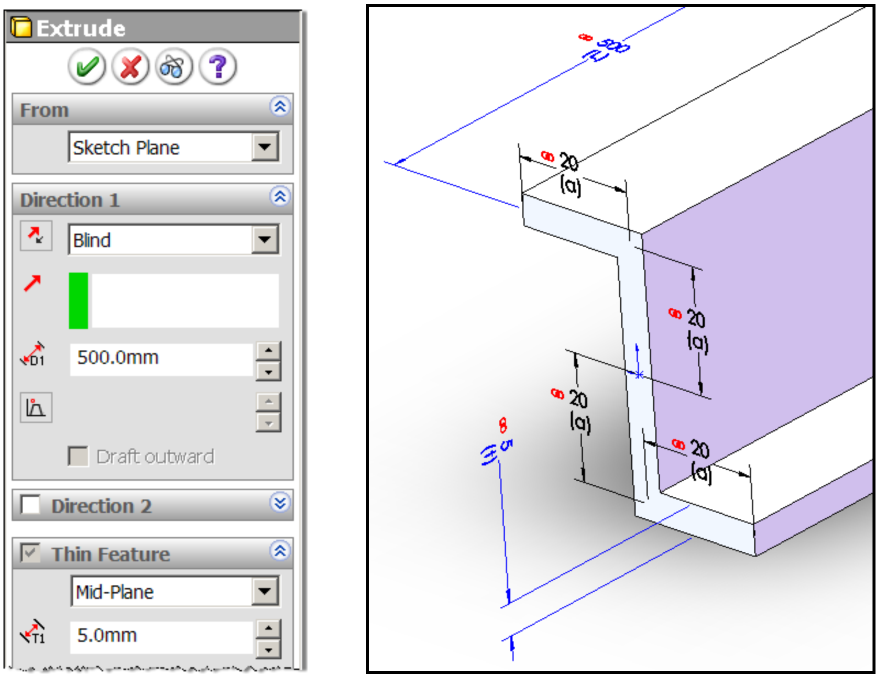


Figure 17-4 Thickening the Zee centerline



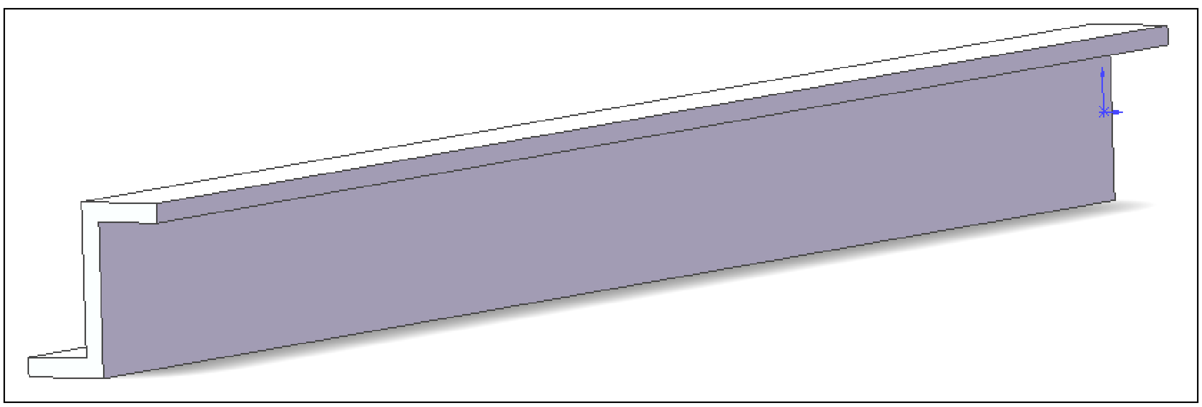


Figure 17-5 The extruded linled value solid

17.2 Construct a quarter symmetry tank

Begin by selecting English units, but also plan to activate dual units in millimeters so you have the option to display English or both units in the dimensions:

1. Pick **Tools→Options→Document Properties→Units** and select the **IPS Unit System**, select the number of decimal places to display and pick millimeters as the optional dual system (if you wish), as illustrated in Figure 17-6**.**

2. If you decide to use dual units then turn them on in the same panel by checking **dual dimension display** under **Document Properties→Detailing,** as given in Figure 17-7**.**

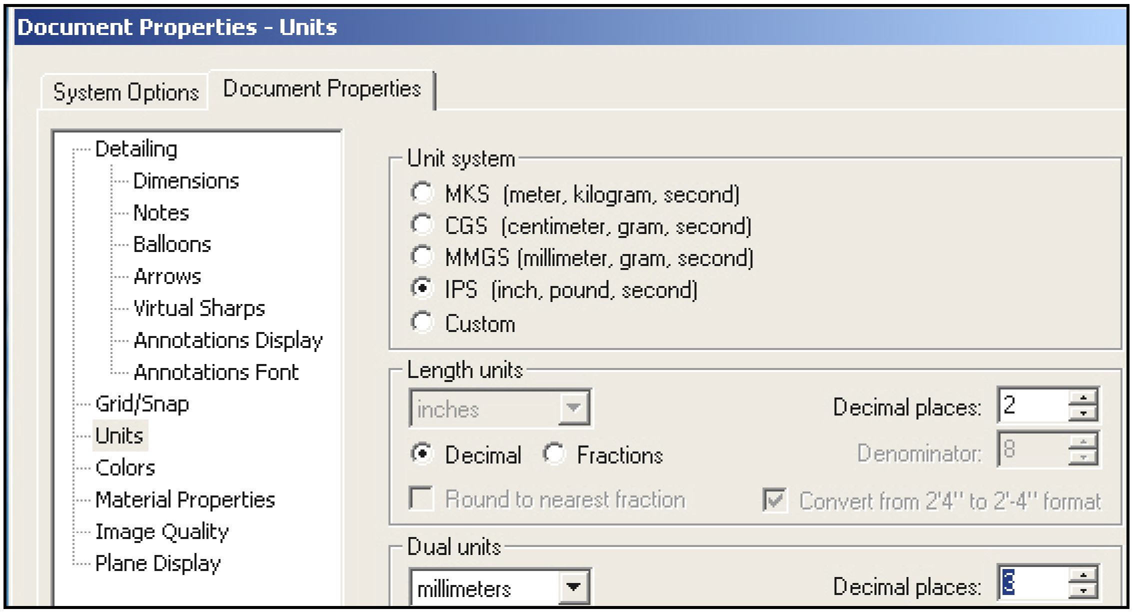


Figure 17-6 Selecting drawing units

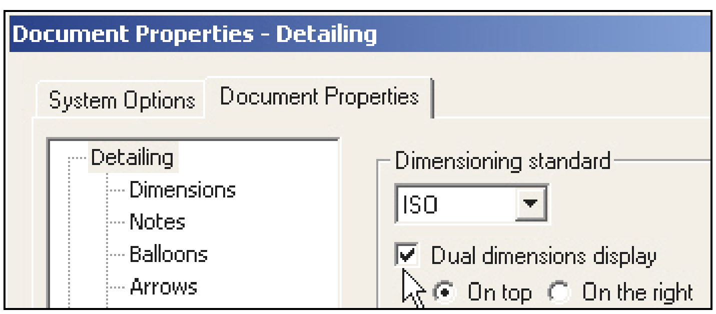


Figure 17-7 Option for dual units dimension display

Begin the part construction by inserting a sketch in the top plane with a true shape view (Figure 17-8). Specify the dimensions shown for a quarter of the base plate area:

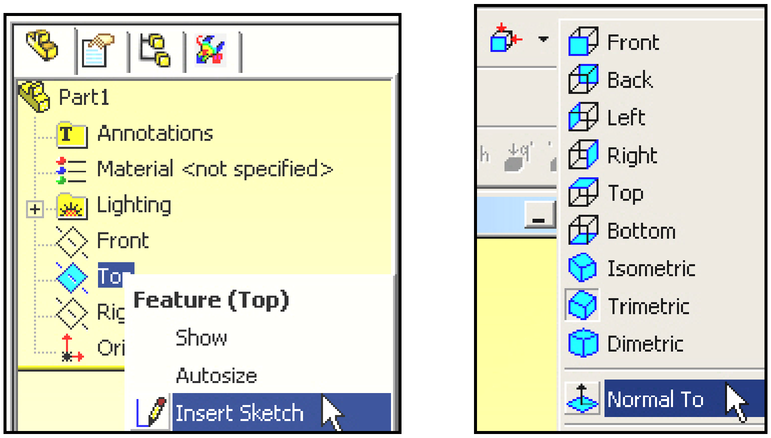


Figure 17-8 Sketching true shape in the top view

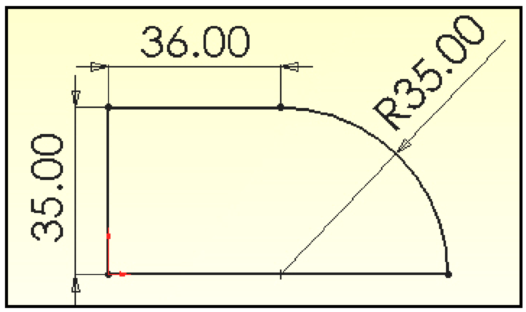


Figure 17-9 Base plate sizes

Figure 17-10 Beginning the extrusion

Next you need to select the thickness of the base plate. That is needed both for the extrusion to form the (thin) solid, but also to later set the structural stiffness. The stretching stiffness of a surface is proportional to its thickness, but its bending stiffness is proportional to the cube of the thickness. Choose a thickness of 0.20 inches for that component. After the stress analysis is completed it may be necessary to change this parametric dimension during a design revision. To extrude the bottom thickness:

1. Begin with **Extruded Boss/Base** in the utility menu (Figure 5).
2. That brings up the **Extrude panel** of
3. Figure 17-11 where you type in the chosen base thickness and click **OK** (the green arrow button) to execute the extrude process. This is a thin walled tank, so the extrusion of the bottom is hard to see (
4. Figure 17-12).

The outer wall of the tank will also be another thin layer of material. Its thickness (and thus its stiffness) does not have to be the same as the base, so start with a thickness of 0.15 inches. Use that value as the only new parametric dimension on the outer edge of the bottom:

1. Right click **Top→Insert Sketch**, in Figure 17-12, to prepare to sketch a thin edge to extrude in both directions away from the base (for the wall and supporting lip).
2. Sketch the 4 lines and 2 arcs adjacent to the curved edge so they are easy to see.
3. Then set the actual dimension of the wall thickness as 0.15 to uniquely define the wall area to be extruded (Figure 17-13).

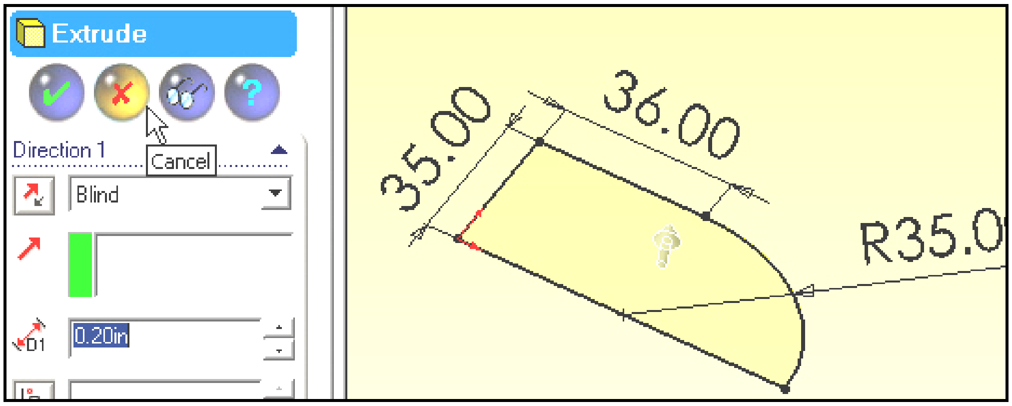


Figure 17-11 Single direction extrude of the base area

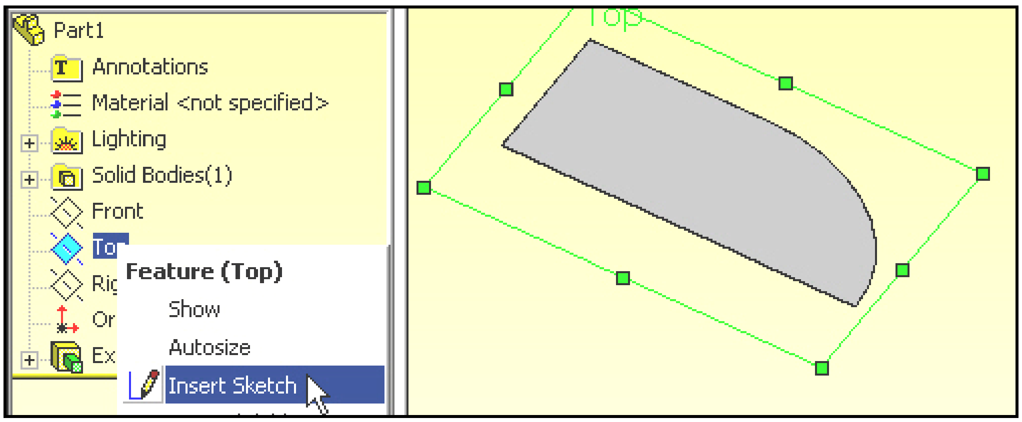


Figure 17-12 Preparing to sketch the thin wall region

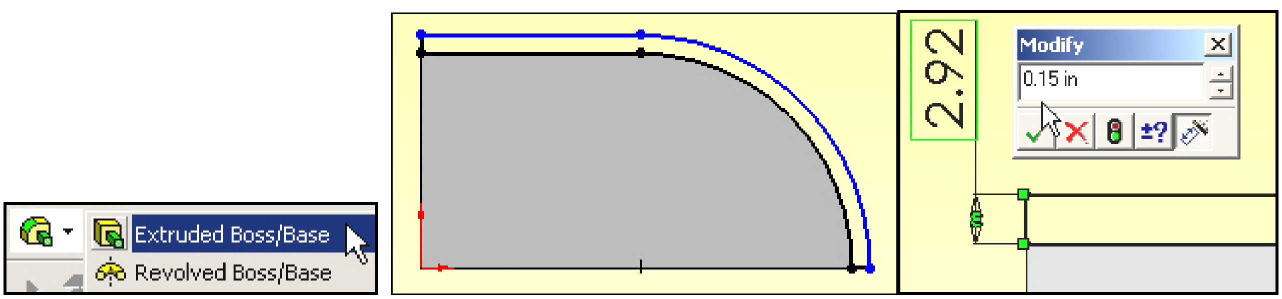


Figure 17-13 Sketch and then dimension the thin wall

The tank wall will be 72 inches (6 feet) above the bottom, while the support lip will be about 3 inches below. Thus, to construct these segments:

1. Select the **Extrude Boss/Base** option.
2. In the **Extrude panel** set **Direction 1** for the upper (water confining) wall and **Direction 2** for the support (lower) wall, as in Figure 17-14. Figure 8-2 shows the final part, as viewed from below.

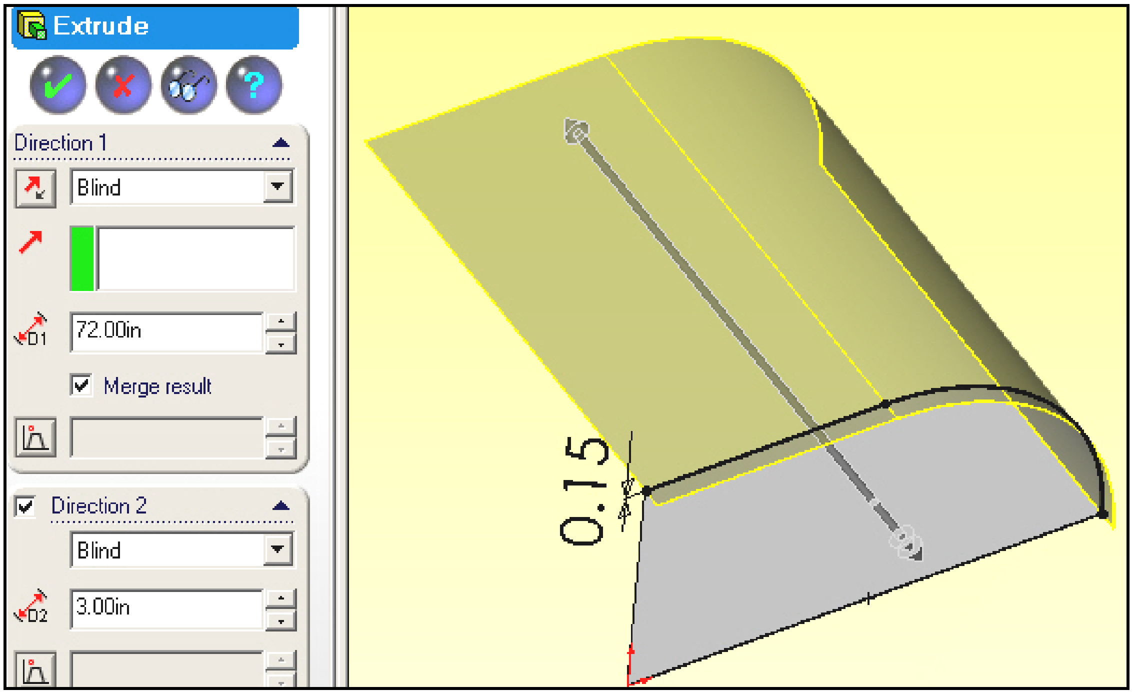


Figure 17-14 Two direction extrusion of the wall

17.3 Cylinder with external convection

17.3.1 Options for units and parametric dimensions

You should plan ahead and select the most likely units system that you or others working on the part may wish to display. Also, if this part is likely to be created in various different sizes you should plan for parametric design and assign names to the important dimensions so they could be automatically changed later by a design table and/or by the native equation system in SolidWorks. Create these useful options via:

1. **Tools→Options→Document Properties tab→Units→IPS→OK**. While there you can control the number of decimal places displayed.
2. Next selecting the **System Properties tab→General→Show dimension names** give you the power to set parametric dimensions for future use (Figure 17-15).

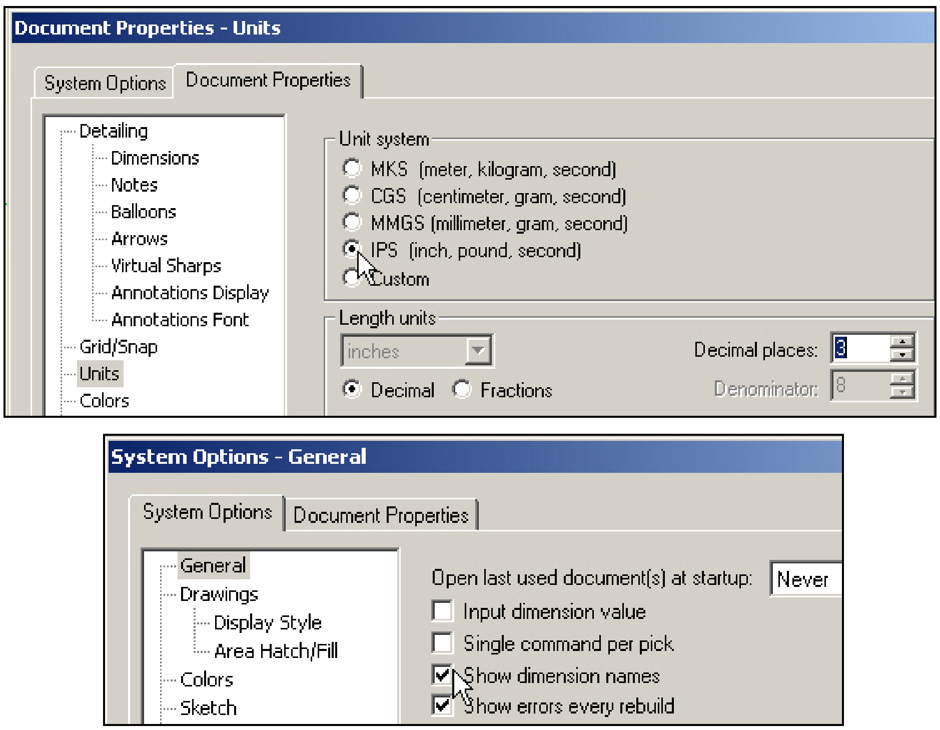


Figure 17-15 Selecting units and dimension name display

17.3.2 Geometric model

Since the SolidWorks database is three-dimensional you must at least start by defining the region of interest as a solid. The radial distances are clearly known and provide a starting point for the solid. As an axially infinite cylinder you can pick an arbitrary slice in the axial direction. Just use a length that will help keep the element aspect ratios reasonable. Likewise, as with any axisymmetric body, you simply need to define the planar (radial-axial) shape and rotate it through an arbitrary angle. Use 5 degrees here for reasonable aspect ratios (if you use a 3-D mesh). Here the area to rotate is just a rectangle. Draw it true shape in the front plane with:

1. **Front→Insert Sketch** and use **Front** or **Normal to** view if necessary.
2. Select the two point **Rectangle** construction (as in Figure 17-16).
3. Begin the rectangle relative to the **origin** (and thus the axis of revolution).
4. Click and drag out an arbitrary rectangle.

5. Right click in the graphics area and pick **Smart Dimensions.**
6. Select the **origin** point and the outer radial vertical **line** and drag the dimension (as initially sketched) below the part and click to place it there. The sketch dimension appears (here 2.4 inches) along with the current **default name** (here D1) as seen in Figure 17-17. The name appears because of our initial option choice.
7. To change the dimension value and/or name, double click on that dimension (when you see the cursor change to the **dimension icon**). The **Dimension panel** appears.
8. Since you wish to change both select **More Properties** in the **Dimension panel** to get access to the name as well as the value.
9. When the **Dimension Properties** appears type in the new **parametric name** (R_outer) and its actual **value** (13.4 inches), **OK**. (See Figure 17-18)
10. Repeat that process for the inner radius (**R_inner** and 9.375 inches) and the arbitrary height (**Z_high** and 0.5 inches). The results are summarized in Figure 17-19.

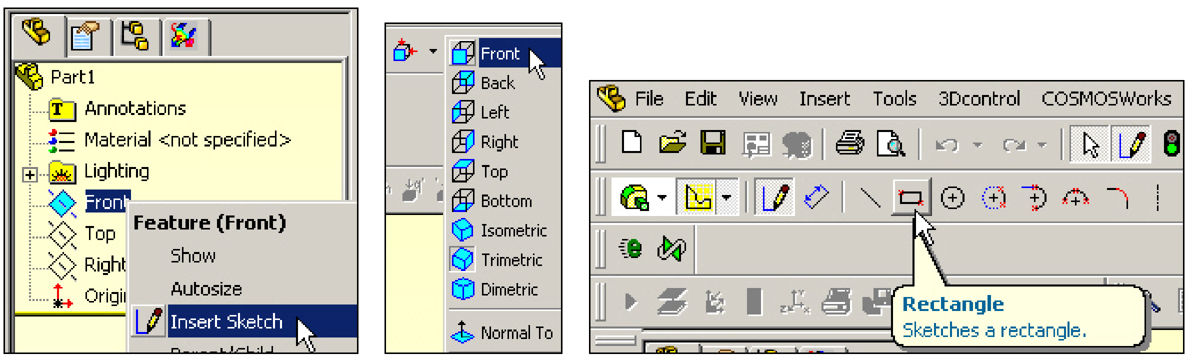


Figure 17-16 Construct the inner and outer radii and vertical height

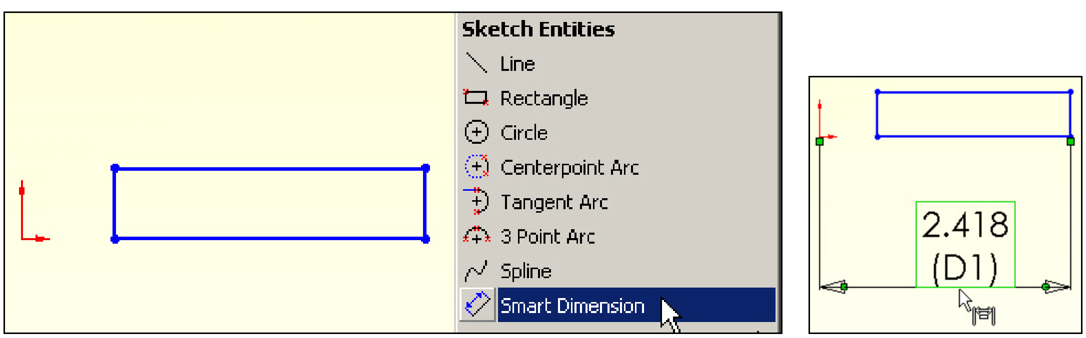
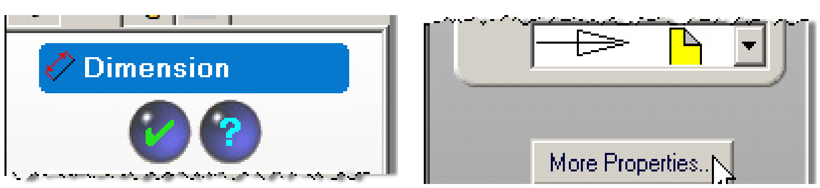


Figure 17-17 Sketching and dimensioning the default rectangle



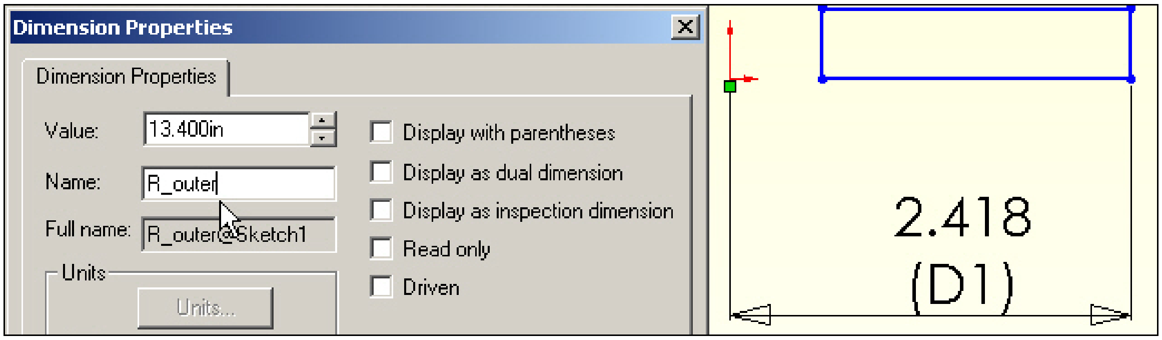


Figure 17-18 Assigning a name and value to the outer radius of the cylinder

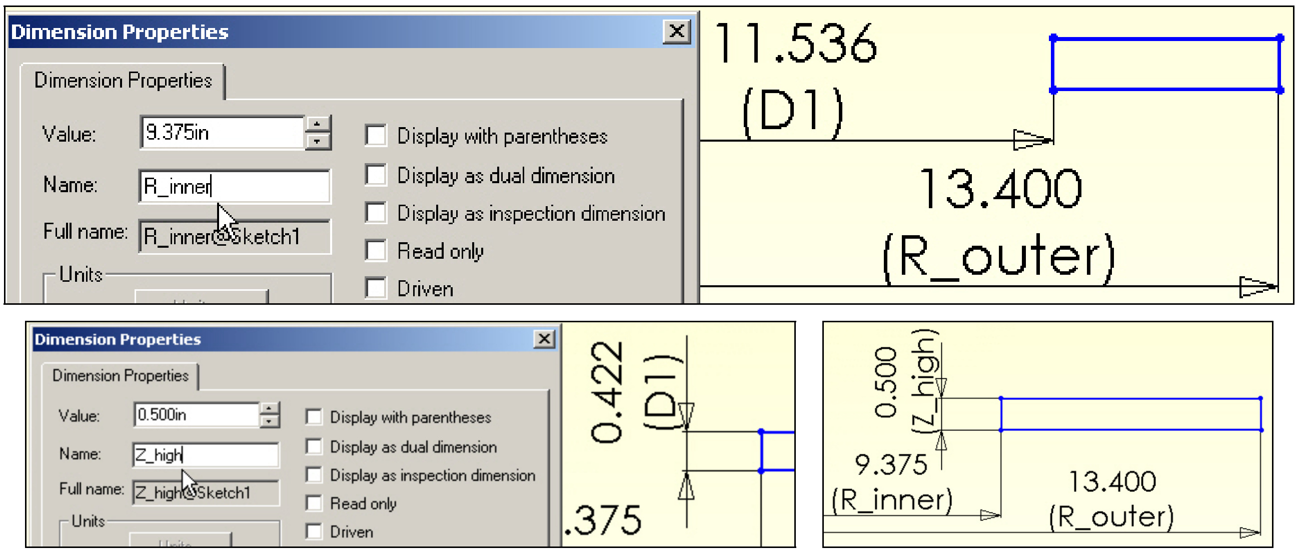
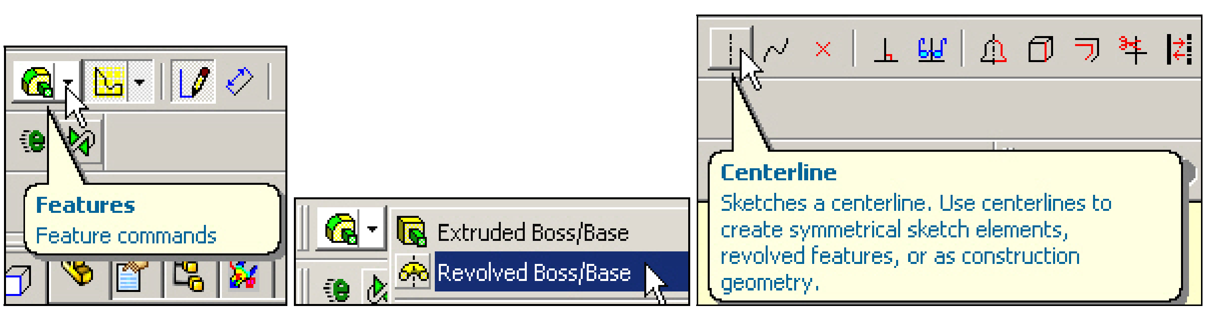


Figure 17-19 Completing the radial slice dimensions and names

17.3.3 Form an axisymmetric body

With the 2D sketch completed it is converted to a #D body by extruding the 2D sketch about the vertical (Z) axis of the cylinder. If the boundary conditions were not axisymmetric the full (360 degree) body could be formed. But here any slide will do, so a 5 degree segment is randomly picked to give reasonable element aspect ratios for the dimensions of the 2D sketch:

1. Pick **Features→Revolved Boss** for the current sketch to open the **Revolve panel**.
2. Insert a vertical **centerline** at the origin to guide the revolution.
3. In the **Revolve panel** select the new centerline as the **axis**. See Figure 17-20.
4. Enter 5.0 for the **angle** to revolve through.
5. Optionally, pick **Reverse direction** so the 3D part will be behind the front plane of the 2D sketch. That will give a prettier picture when the results are later viewed in the front plane.



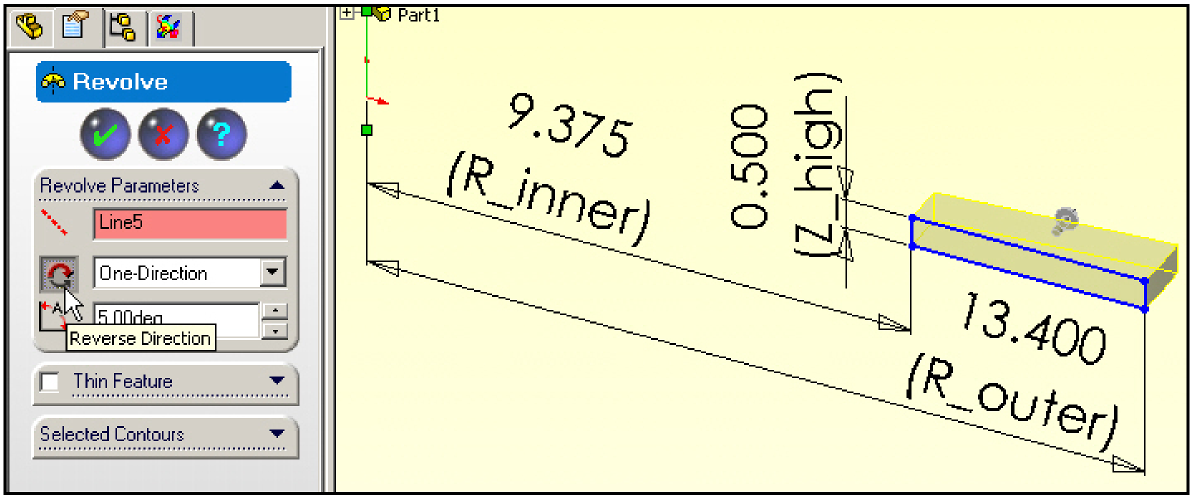


Figure 17-20 Form a wedge segment of the infinite cylinder.

17.3.4 Color code boundary surfaces

The required solid body is now finished. At this stage it sometimes helps to assign different colors to various surfaces of the part. In this example surface colors could remind you of where various boundary conditions are applied. For example:

1. Assign green for convection at the outer radius. **Right click** on the **outer face**.
2. Select **Face→Appearance→Color** to open the **color panel**.
3. In the **Color panel** pick the desire **green** color, click **OK**. (Top of Figure 17-21.)
4. Repeat for the inner face and pick **red** for a known temperature (bottom Figure 17-21).

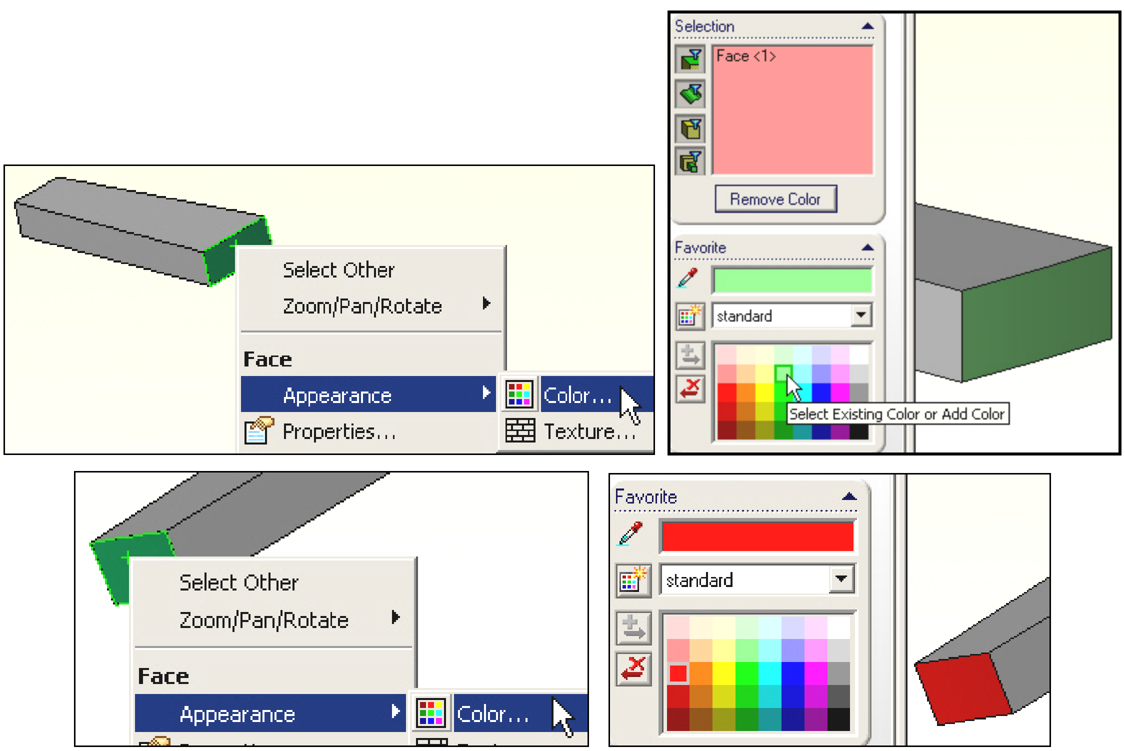


Figure 17-21 Assigning optional surface colors

Another useful option is being able to turn on (or off) the dimensions of a body or sketch:

1. In the **Feature Manager** right click on **Annotations**.
2. Select **Show Feature Dimensions**. **Save as** T_Cylinder_h_body in SolidWorks.

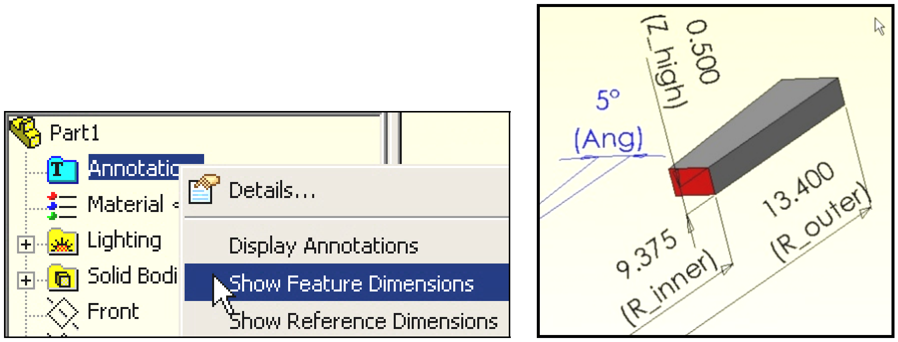


Figure 17-22 Optional display of the segment dimensions

17.4 Thermal study of a hole in a square plate

17.4.1 Geometry

First select the working units (and a dual dimension display):

1. Select **Tools→Options→Document Properties→Units**
2. Set **Unit System** to centimeters, and **Dual units** to inches, click **OK**
3. Use **Tools→Options→System Options→Detailing**
4. In **Dimensioning standard** turn on dual dimension display, click **OK**

Build the segment in the first quadrant of the x-y coordinate system:

1. Select **Top→Insert Sketch.**
2. Insert a 0 and 90 degree construction line through the origin
3. Place and dimension the central arc
4. Place and dimension the four lines
5. Select **Extruded Boss/Base** (see Figure 17-23**).**
6. **I**n the **Extrude panel** set distance **D1** to 1 m, and method to **Blind, click OK**

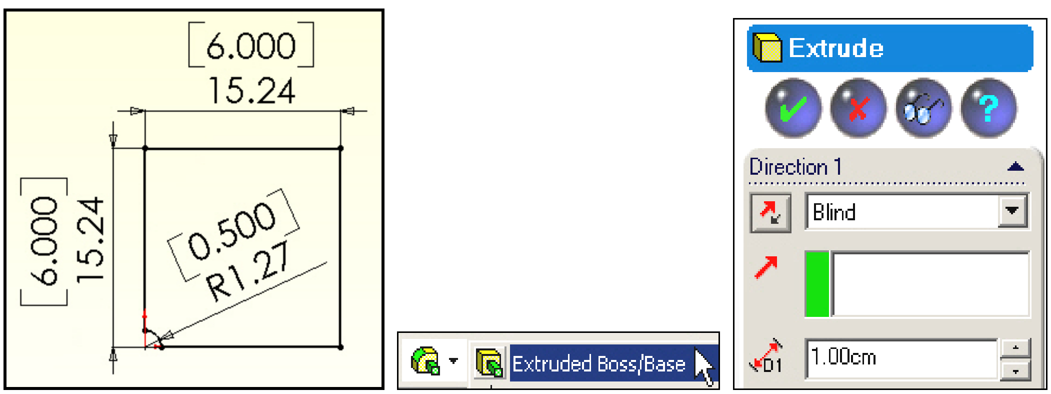


Figure 17-23 Use the first quadrant to extrude a model

18 Appendix B: Typical unit conversions

18.1 Angular velocity

UNIT	Symbol	rad/s	rps	rpm
1 radian per second	rad/s	1	0.159155	9.54930
1 revolution per second	rps	6.28319	1	60
1 revolution per minute	rpm	0.104720	0.0166667	1

18.2 Convection coefficient and thermal conductance

UNIT	Symbol	W/m^2 K	kcal/h m^2 K	Btu/h ft^2 R
1 watt per sq. meter Kelvin	W/m^2 K	1	0.859845	0.17611
1 kilocalorie per hour sq. meter Kelvin	kcal/h m^2 K	1.163	1	0.204816
1 Btu per hour sq. foot Rankine	Btu/h ft^2 R	5.67826	4.88243	1

18.3 Force

UNIT	Symbol	N	lb_f	oz_f
1 Newton = 1 kilogram meter per second^2	N = kg m/s^2	1	0.224809	3.59694
1 pound-force	lb_f	4.44822	1	16
1 ounce-force	oz_f	0.278014	0.0625	1

18.4 Heat flux

UNIT	Symbol	W/m^2	kW/cm^2	$kcal/m^2h$	Btu/ft^2h
1 watt per sq. meter	W/m^2	1	$1e^{-7}$	0.859845	0.316998
1 kilowatt per sq. centimeter	kW/cm^2	$1e^7$	1	$8.59845e^6$	$3.16998e^6$
1 kilocalorie per sq. meter hour	$kcal/m^2h$	1.163	$1.16300e^{-7}$	1	0.368669
1 Btu per sq. foot hour	Btu/ft^2h	3.15459	$3.15459e^{-7}$	2.71246	1

18.5 Elastic modulii or Pressure or Stress

UNIT	Symbol	Pa $=N/m^2$	atm	lb_f/in^2	lb_f/ft^2	ft. H_2O
1 Pascal = 1 Newton per sq. meter	Pa = N/m^2	1	$9.8692e^{-6}$	$1.4504e^{-4}$	$2.08854e^{-2}$	$3.3455e^{-4}$
1 atmosphere	atm	$1.0133e^5$	1	14.696	2116.22	33.8985
1 pound-force per sq. inch	lb_f/in^2	$6.8948e^3$	0.068046	1	144	2.30666
1 pound-force per sq. foot	lb_f/ft^2	47.8803	$4.7254e^{-4}$	$6.9444e^{-3}$	1	0.0160185
1 foot of water	ft. H_2O	$29.891e^2$	0.0295008	0.433527	62.4280	1

18.6 Power

UNIT	Symbol	W = J/s	kcal/h	ft lbf/s	hp	Btu/h
1 watt = 1 joule per second	W = J/s	1	0.859845	0.737562	1.34102e-3	3.41214
1 kilocalorie per hour	kcal/h	1.163	1	0.857783	1.55961e-3	3.96832
1 foot pound-force per second	ft lbf /s	1.35582	1.16580	1	1.81818e-3	4.62625
1 horsepower	hp	745.700	641.186	550	1	2544.43
1 Btu per hour	Btu/h	0.293071	0.251996	0.216158	3.93015e-4	1

18.7 Specific heat

UNIT	Symbol	kJ/kg K	cal/kg K	W h/kg K	Btu/lbmR
1 kilojoules per kilogram Kelvin	kJ/kg K	1	2.38846e2	0.277778	0.238846
1 calorie per kilogram Kelvin	cal/kg K	4.1868e-3	1	1.1630e-3	1e-3
1 watt hour per kilogram Kelvin	W h/kg K	3.600e-3	859.845	1	0.859845
1 Btu per pound-mass Rankine	Btu/lbmR	4.1868	1e3	1.16300	1

18.8 Thermal conductivity

UNIT	Symbol	W/m K	kcal/h m K	Btu/h ft R
1 watt per meter Kelvin	W/m K	1	$85.9845e^{-2}$	0.577789
1 kilocalorie per hour meter Kelvin	kcal/h m K	1.163	1	0.671969
1 Btu per hour foot Rankine	Btu/h ft R	1.73073	1.48816	1

18.9 Torque

UNIT	Symbol	J =N m	kW h	ft lb_f	hp h
1 joule = 1 Newton meter	J = N m	1	$2.77778e^{-7}$	0.737562	$3.72506e^{-7}$
1 kilowatt hour	kW h	$3{,}600e^{3}$	1	$2.65522e^{6}$	1.34102
1 foot pound-force	ft lb_f	1.35582	$3.76616e^{-7}$	1	$5.05051e^{-7}$
1 horsepower hour	hp h	$2.6845e^{6}$	$745.700e^{-3}$	$1.98e^{6}$	1

19 References

1. Adams, V. and A. Askenazi, *Building Better Products with Finite Element Analysis*. 1999, Santa Fe: Onword Press.
2. Akin, J.E., *Finite element analysis with error estimators*. 2005, Amsterdam: Elsevier/Butterworth-Heinemann.
3. Blevins, R.D., *Formulas for natural frequency and mode shape*. 1979, Malabar, FL: Krieger.
4. Carslaw, H.S. and J.C. Jaeger, *Conduction of Heat in Solids*. 1959, Oxford: Oxford Press.
5. Chapman, A.J., *Fundamentals of Heat Transfer*. 1987: Collier Macmillan.
6. Jiji, L.M., *Heat Transfer Essentials: A Textbook*. 1998: Begell House Publishers, Inc.
7. Kurowski, P.M., *Engineering Analysis with CosmosWorks Professional 2006*. 2006: SDC Publishing.
8. Myers, G.E., *Analytical Methods in Conduction Heat Transfer*. 1971, McGraw-Hill: New York.
9. Norton, R.L., *Machine design: an integrated approach*. 2006: Prentice-Hall.
10. Oden, J.T. and E.A. Ripperger, *Mechanics of Elastic Structures, 2nd Edition*. 1981, New York: McGraw-Hill.
11. Pilkey, W.D., *Formulas for Stress, Strain, and Structural Matrices*. 1993: John Wiley & Sons, Inc. New York, NY, USA.
12. Popov, E.P., *Engineering Mechanics of Solids*. 1990: Prentice-Hall, Inc. 752.
13. Segerlind, L.J., *Applied Finite Element Analysis*. 1984, New York: John Wiley.
14. Timoshenko, S. and S. Woinowsky-Krieger, *Theory of Plates and Shells*. 1987, McGraw-Hill, New York.
15. Young, W.C. and R.G. Budynas, *Roark and Young on TK*. 2002, Universal Technical Systems Rockford, IL.
16. Young, W.C., R.G. Budynas, and R.J. Roark, *Roark's formulas for stress and strain*. 2003: McGraw Hill.
17. Young, W.C. and R.G. Budynas, *Roark’s Formulas on Excel*. 2005, Universal Technical Systems, Rockford, IL.
18. Ziegler, H., *Principles of structural stability*. 1968: Blaisdell.

20 Index