

2D Stress Analysis (Draft 1, 10/10/06)

Plane stress analysis

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. This will be illustrated with a simple rectangular beam plane stress analysis. Consider a beam of rectangular cross-section with a thickness of $t = 2 \text{ cm}$, a depth of $h = 10 \text{ cm}$, and a length of $L = 100 \text{ cm}$. Let a uniformly distributed downward vertical load of $w = 100 \text{ N/cm}$ 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 $M = 1.25e3 \text{ N-m}$. 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.

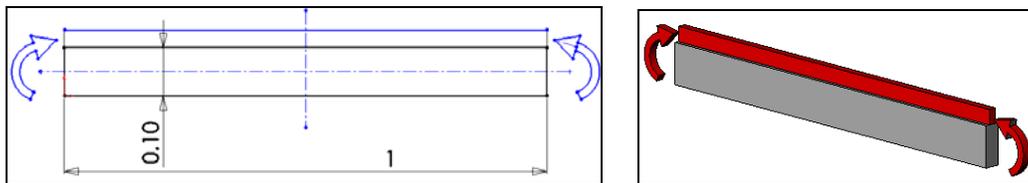


Figure 1 A simply supported beam with line load and end moments

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 or anti-symmetry.

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

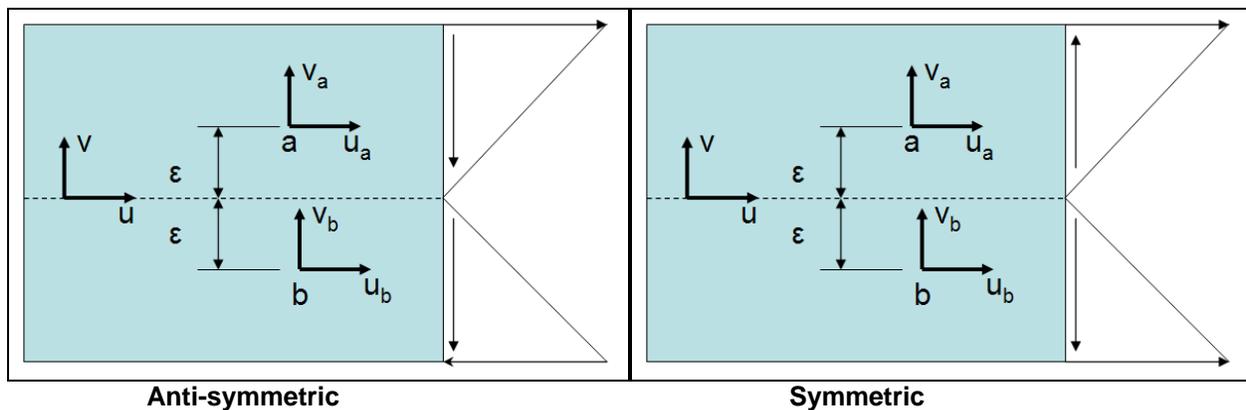


Figure 2 Anti-symmetric ($u = 0, v = ?$), and symmetric ($u = ?, v = 0$) displacement states

dashed centerline of the anti-symmetric image at the left in Figure 2. 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 ($\epsilon \rightarrow 0$). 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 wish 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 2 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.***

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 forced 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 called 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

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

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, CosmosWorks non-uniform pressure data requires a pressure scale, 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: $p(y) = p_{scale} * y$ (with y non-dimensional). This must match p_{max} at $y = h / 2$, so

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

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, $p_{max} = 3.75e7 \text{ N/m}^2$, $p_{scale} = 7.5e8$,

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 $v_{max} = 1.36e-3 \text{ m}$. 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 3, will be built, restrained, and loaded. A new static study is opened using a shell mesh defined by selecting the front face of the model, and its thickness is defined to be 0.02 m (Figure 4).

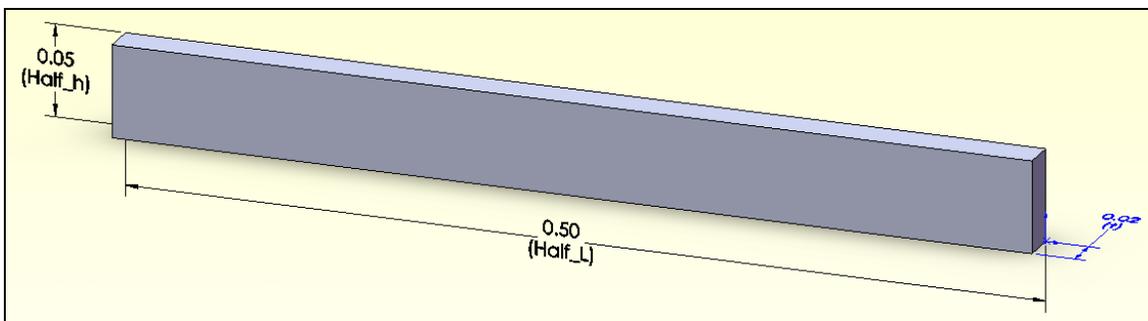


Figure 3

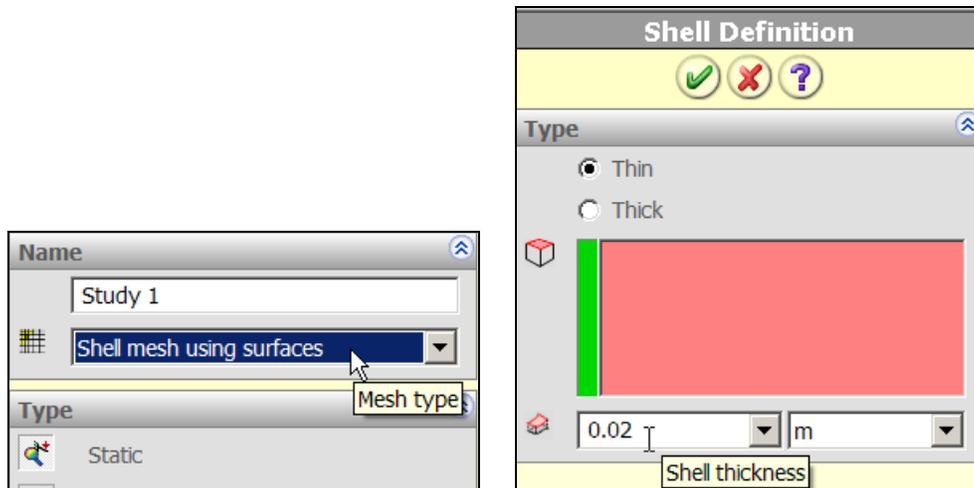


Figure 4

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. Here splits were inserted near the interior quarter points (Figure 5) so, including the end lines five graphing sections will be available. Having finished the geometric model the restraints and loads will be applied.

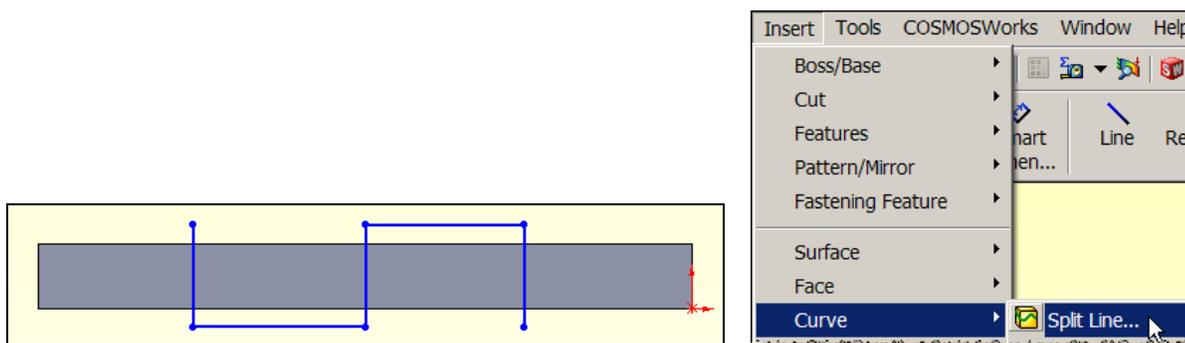


Figure 5

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. In Figure 6, the zero horizontal (x) deflection is applied as a symmetry condition on the edge corresponding to the beam vertical centerline, and then as the anti-symmetry condition along the edge of the neutral axis.

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, in Figure 7, and about one-third of that end was

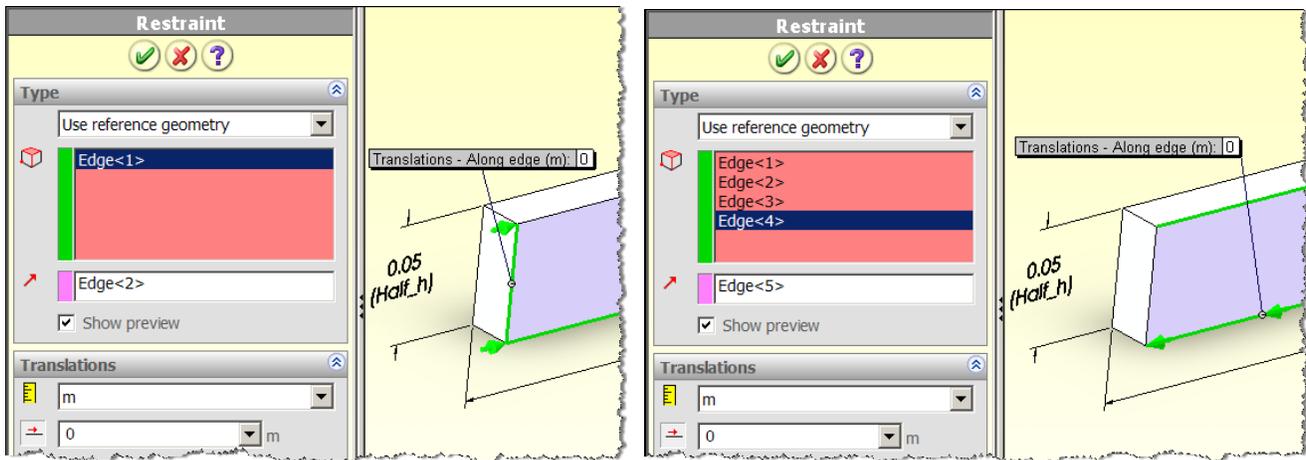


Figure 6

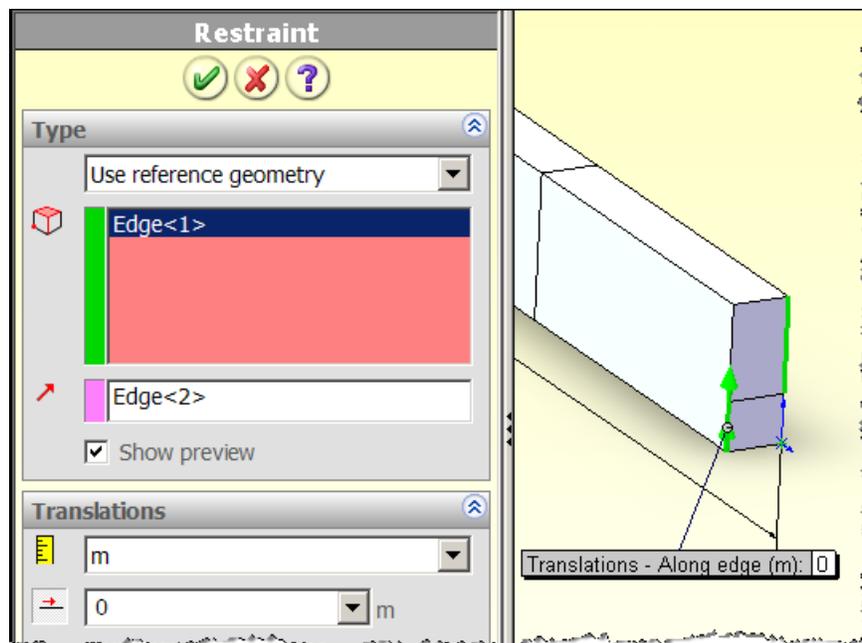


Figure 7

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.

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. That is done simply

by zeroing the z-translation along the top edge of the beam, as in Figure 8. The combination of all the plane stress supports and the rigid body motion restraint is seen in Figure 9.

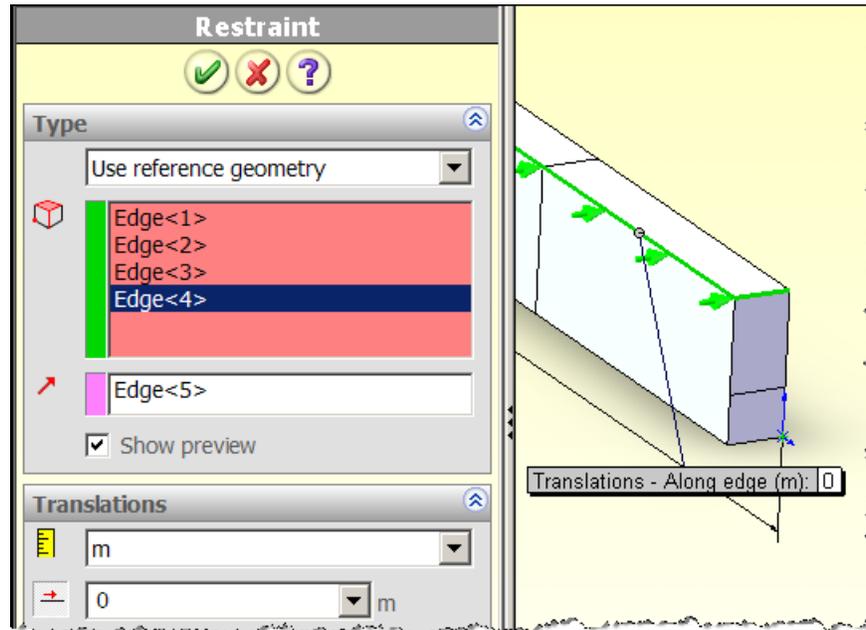


Figure 8

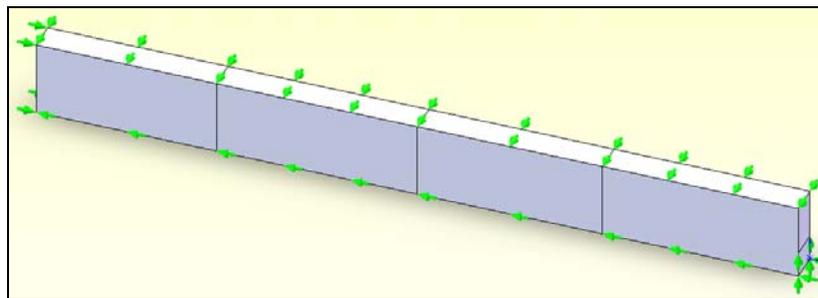


Figure 9

Moment application as a non-uniform 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 pick **OK**, else pick part edges to orientate the y-axis (Figure 10).

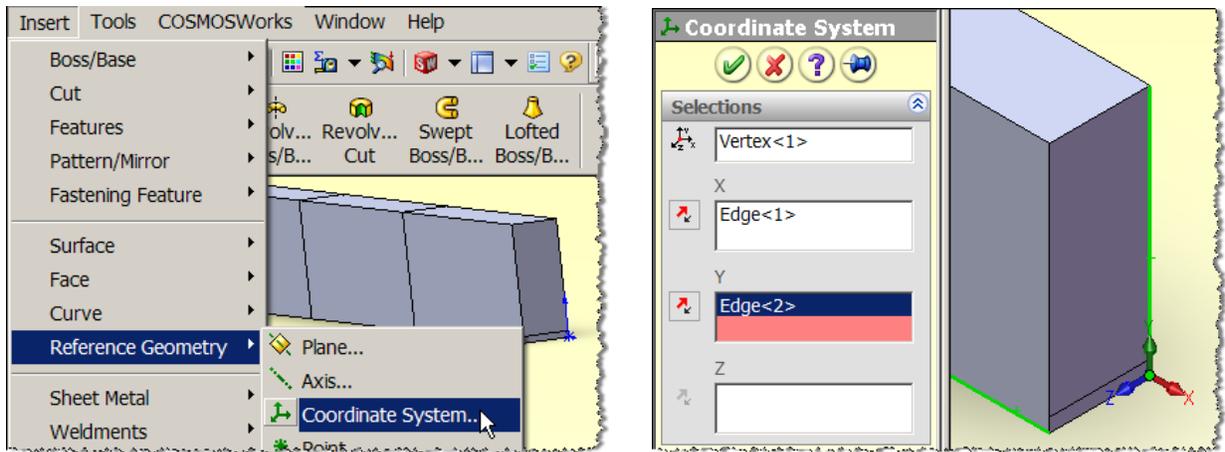


Figure 10

The application of the non-uniform pressure is applied at the front vertical edge at the simple support in the **Pressure** panel of Figure 11. 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.

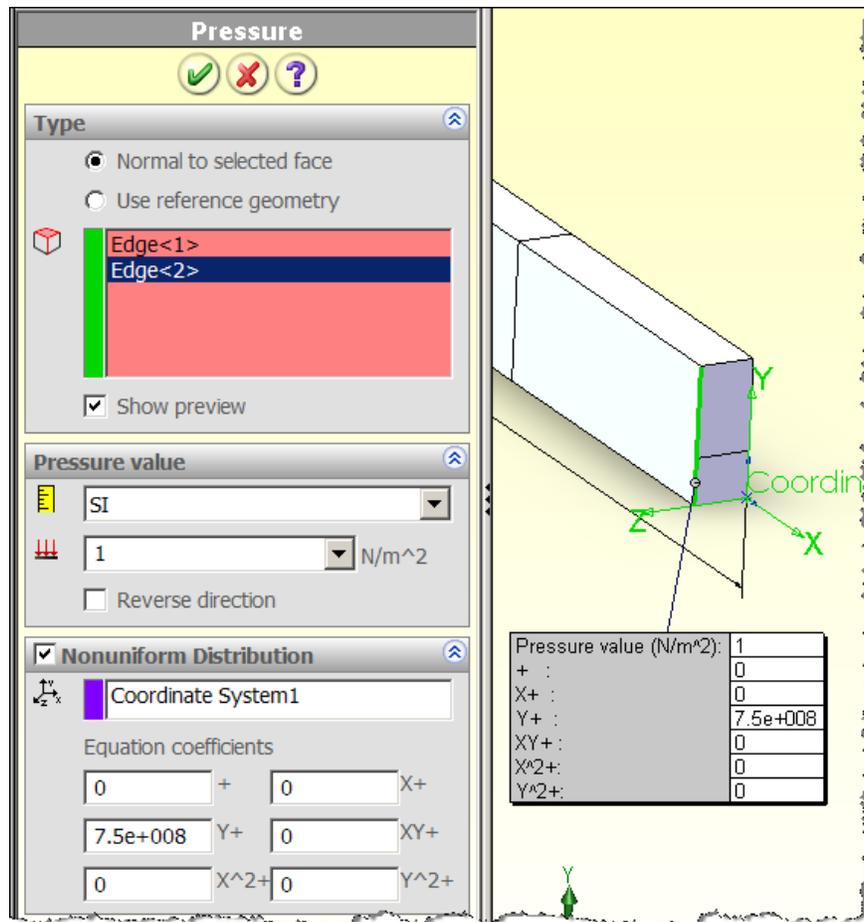


Figure 11

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 12). 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.

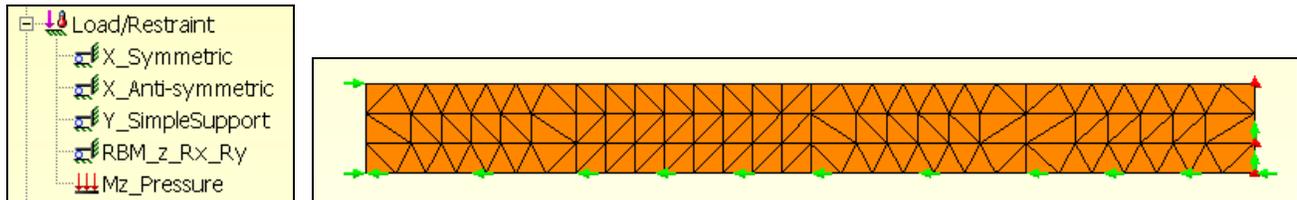


Figure 12

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 $v_{max} = 1.36e-3 m$. The resultant displacement vectors are seen in Figure 13. They are seen to become vertical at the centerline. A probe

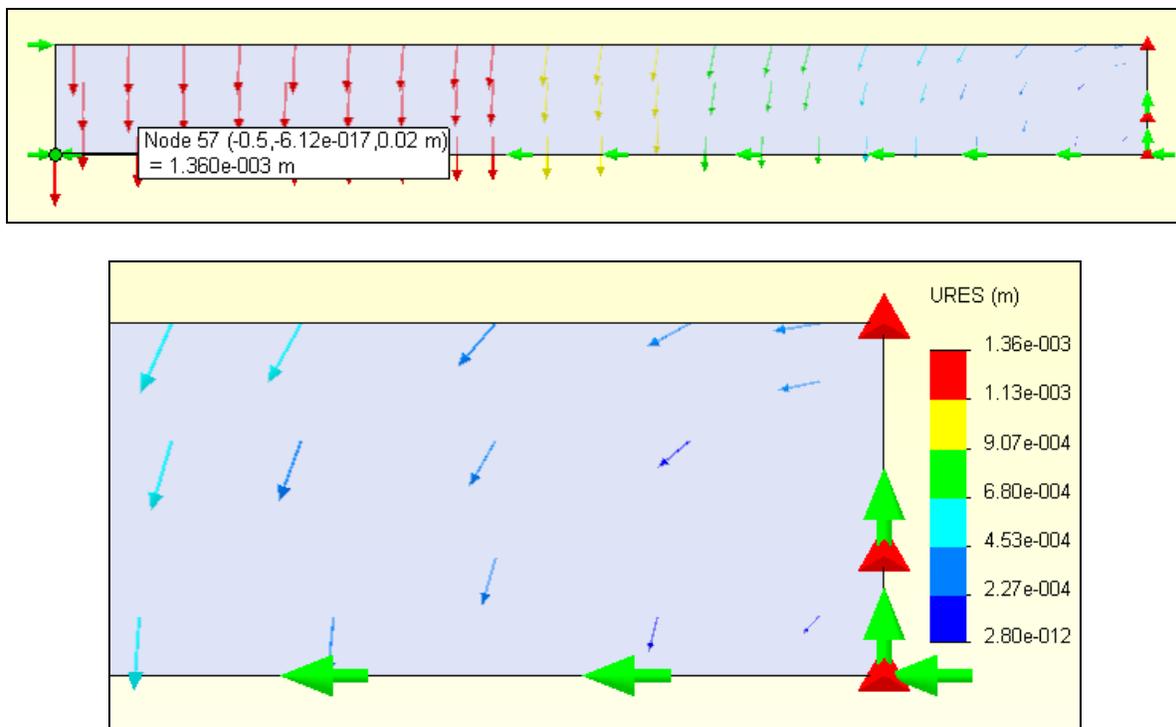


Figure 13

displacement result at the bottom point of the vertical centerline line gives $v_{FEA} = 1.36e-3 \text{ m}$, which agrees to three significant figures with the elementary theory. A detail view of the support region (bottom of Figure 13) shows that the displacement vectors close to the restraint are basically rotating about that support.

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 $\sigma_x = p_{max} = 3.75e7 \text{ N/m}^2$. That seems to agree with the contour range in Figure 14 and indeed, a stress probe there gives a value of $\sigma_x = -3.77e7 \text{ N/m}^2$. 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 15. 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.

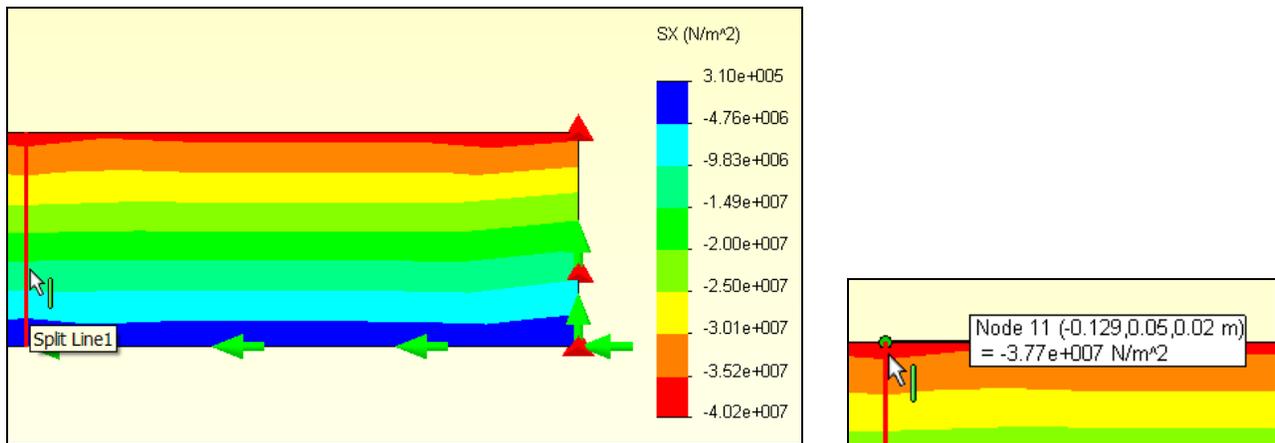


Figure 14

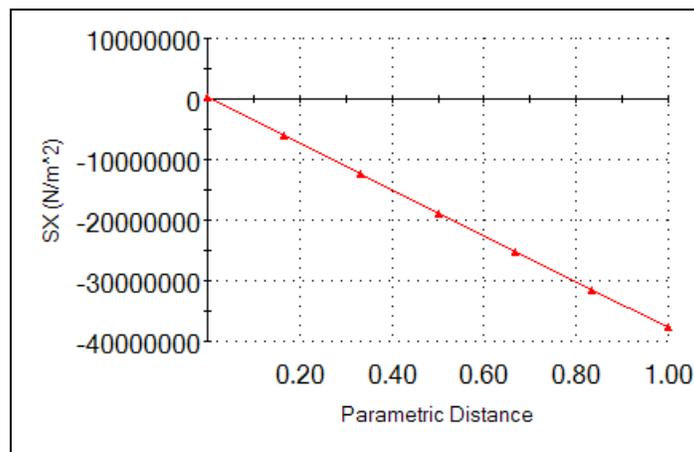


Figure 15

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 $F=18,750\text{ N}$. 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.
2. Examine the horizontal (x) reaction force and verify that its sum is $-18,750\text{ N}$. (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)

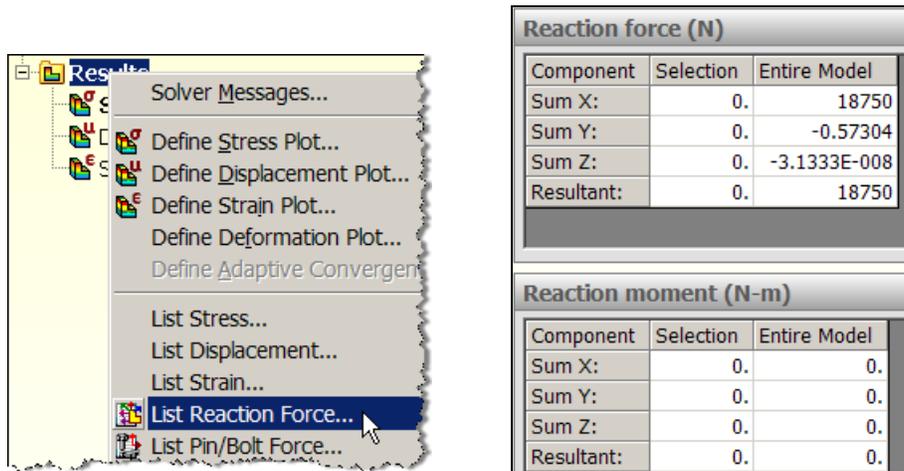


Figure 16

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.

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 $10,000\text{ N}$. Since the part has been reduced to one-fourth through the use of symmetry and anti-symmetry you only need to distribute $2,500\text{ N}$ 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 pressure (which is the better way). Figure 17 (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)..

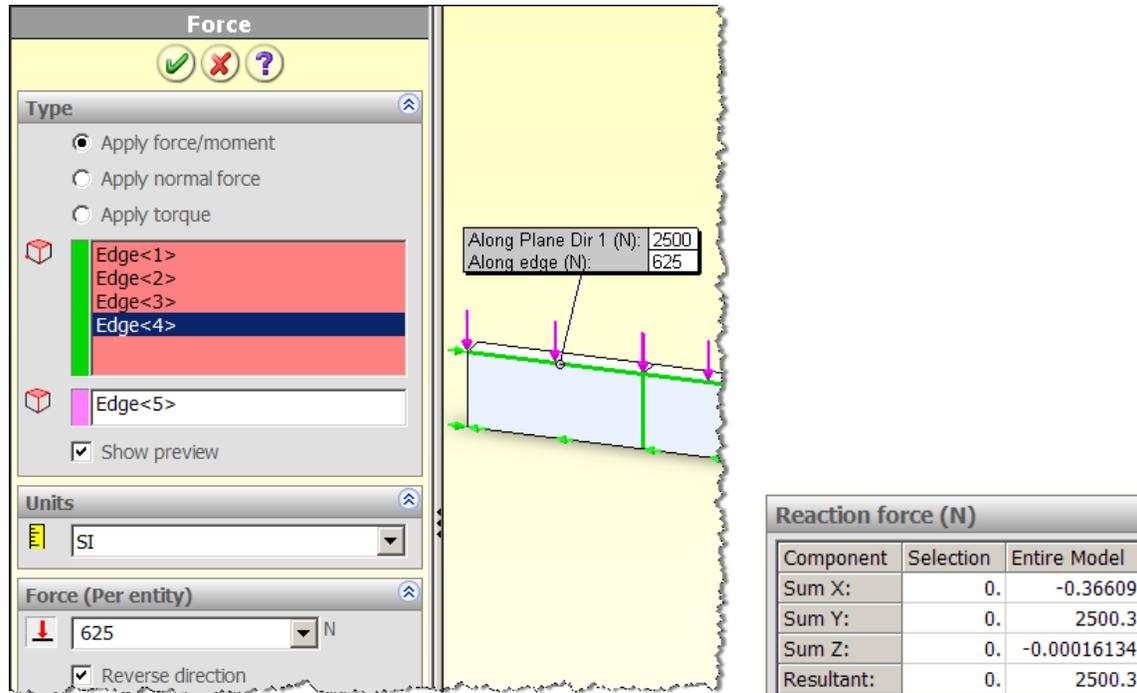


Figure 17

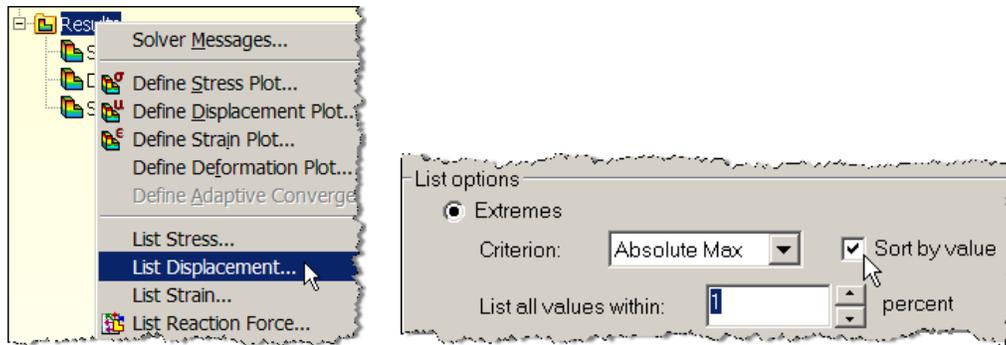
With this second load case in place the study is simply run again with the same restraints and mesh. Since this loading procedure was potentially confusing the first information recovered from the results file was the force reaction data, on the right in Figure 17. Those data verify that the total vertical 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 $v_{max} = 1.13e-3 m$. The plane stress maximum deflection was extracted:

1. Right click in the manager menu **Results**→**List Displacements** (see top of Figure 18) to open the **Displacements List** window.
2. Under **List Options** select **Absolute Max**, **OK**.
3. When the list appears note (bottom of Figure 18) that the maximum deflection is $1.16e-3 m$ at the vertical centerline position. That is very close to the initial estimate.

The contoured magnitude is also given in that figure. It again shows a rotational motion about the simple support end, and vertical translation at the beam centerline, as expected.

The numerical value of the maximum horizontal fiber stress was listed in a similar manner. The maximum compression value, in Figure 19, of $\sigma_x = -4.04e7 N/m^2$ compares well with the



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

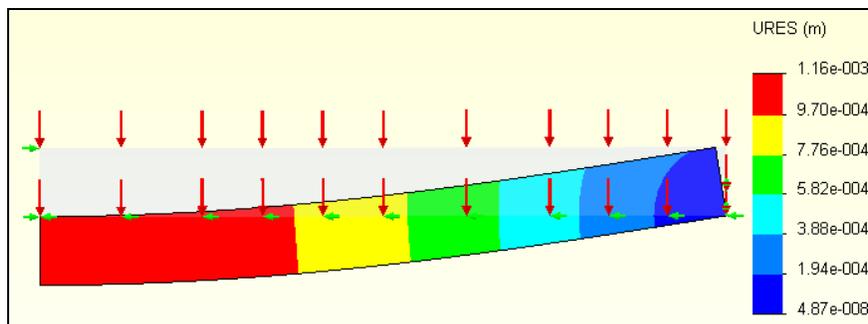


Figure 18

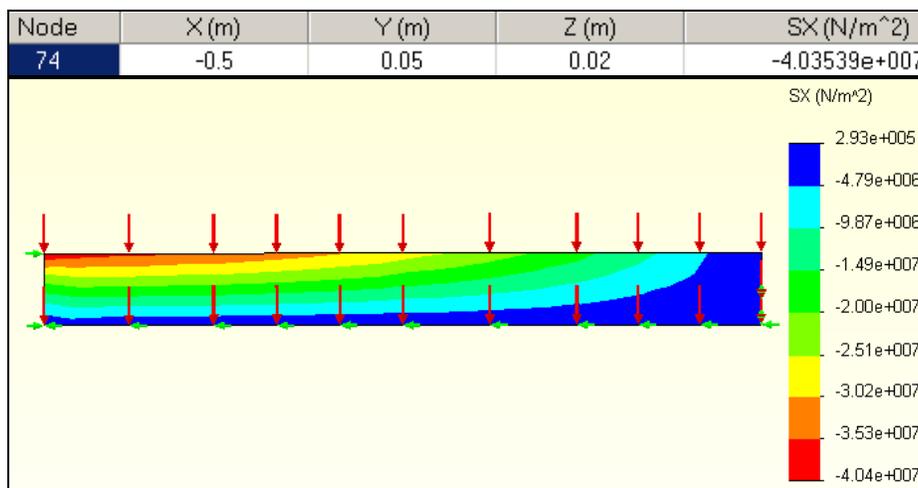


Figure 19

simple beam theory value of $-3.75e7 \text{ N/m}^2$, 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 20 presents the normal stress and shear stress, through the depth, at the L/4 and L/8 positions. 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 $1.88e6 \text{ N/m}^2$. The detail graph values in Figure 21 shows a plane stress maximum shear of $1.84e6 \text{ N/m}^2$ and a minimum of $0.3 e6 \text{ N/m}^2$. 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.

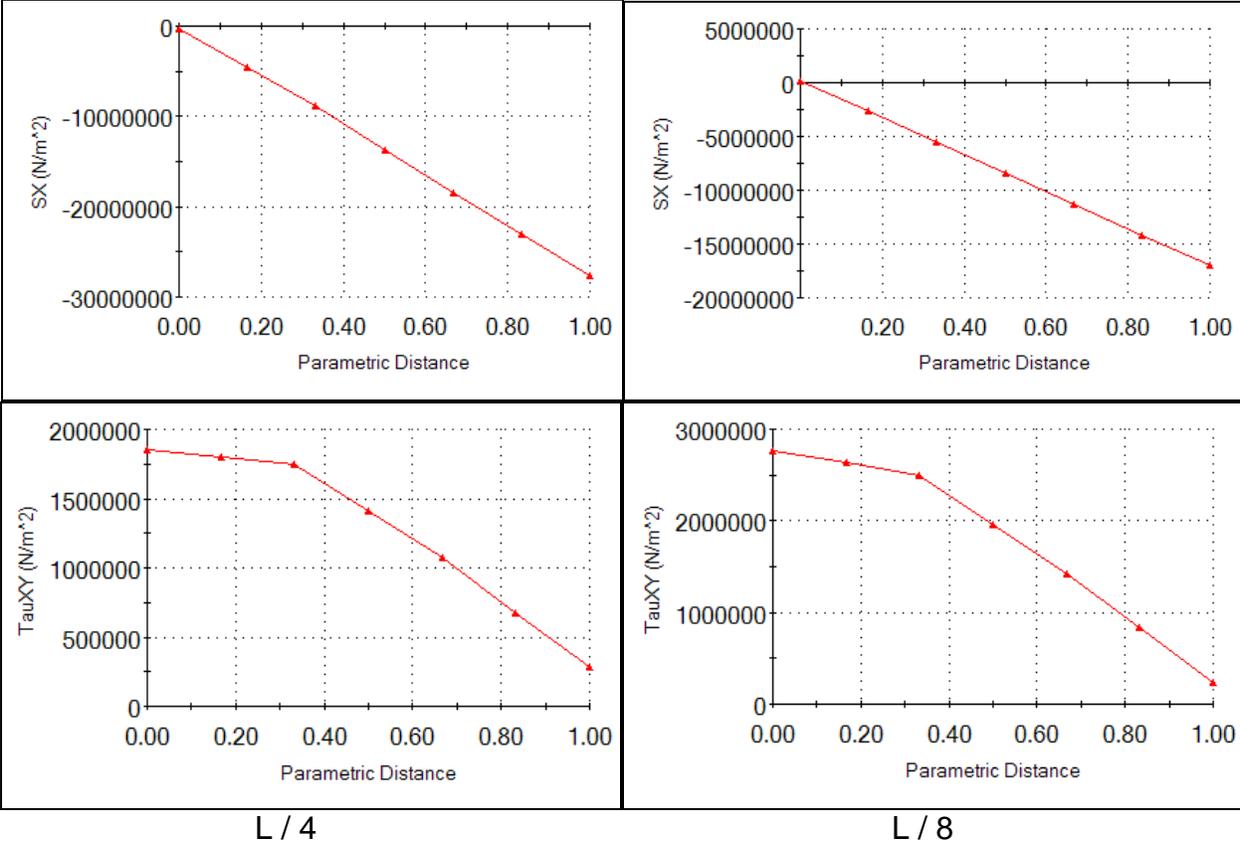


Figure 20

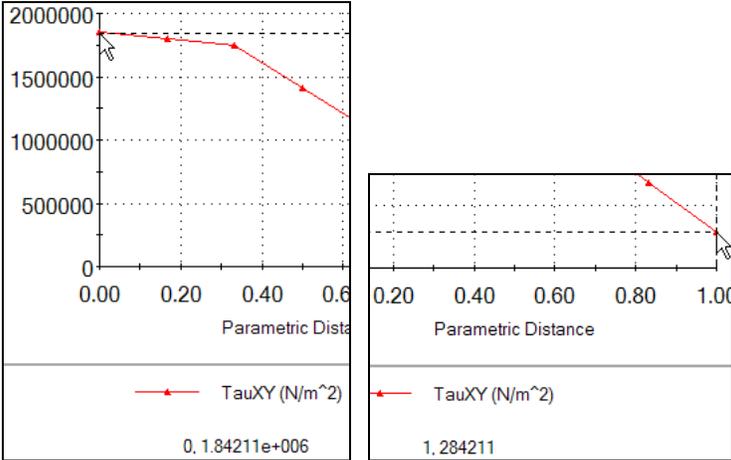


Figure 21

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 stress traction in the vertical direction). To compute the necessary shear stress the quarter model load ($2,500\text{ N}$) was divided by the part face area ($h L/2 = 0.025\text{ m}^2$) and entered as the pressure value tangent to the reference geometry, as shown in Figure 22. This approach gave the exact same results as those in the previous section. This load approach was suppressed for the final combined loading.

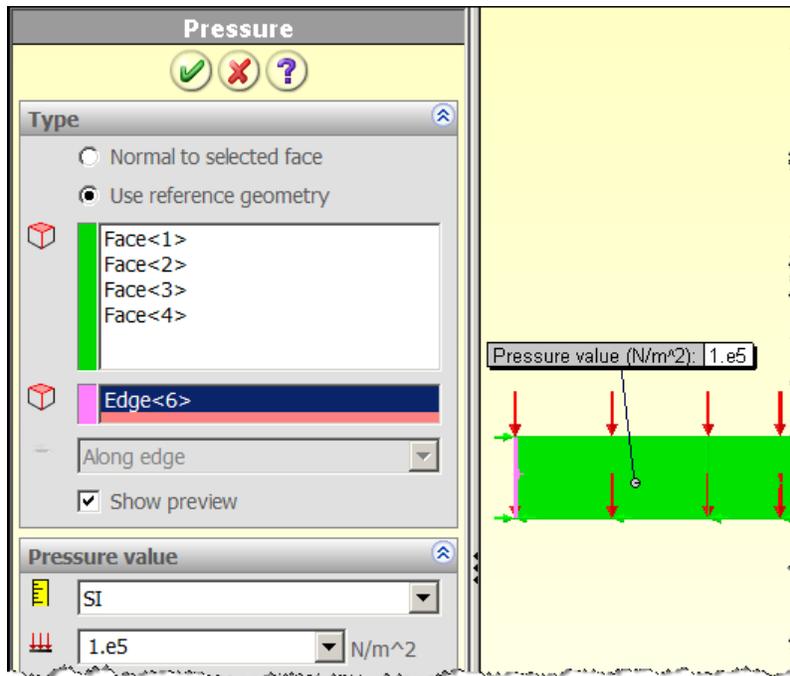


Figure 22

Combined load cases

Having validated each of the two load cases they are combined by un-suppressing the end moment condition (Figure 23 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

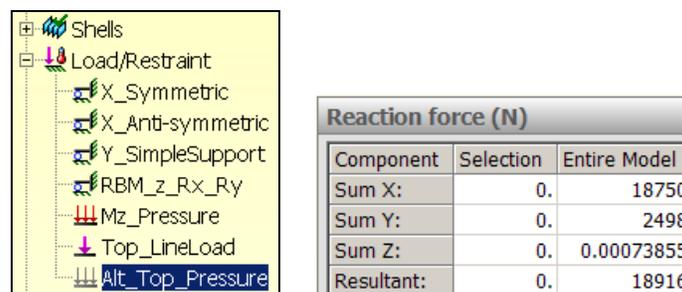


Figure 23

quick spot check verifies the expected results. The reaction force components were verified (Figure 23 right) before listing the maximum deflection and fiber stress (Figure 24).

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 24

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 at the top of Figure 25, twice the maximum shear stress (the stress intensity) is given in the middle of that figure while the bottom 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 need to be compared to the yield point stress of $2.8e7 \text{ N/m}^2$. The arrow in the top part of 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 [1]. 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 eigen-analysis. Since buckling is usually sudden and catastrophic it would require a much higher FOS. Very high precision machines are usually governed by deflection limits. That case is not considered here.

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.

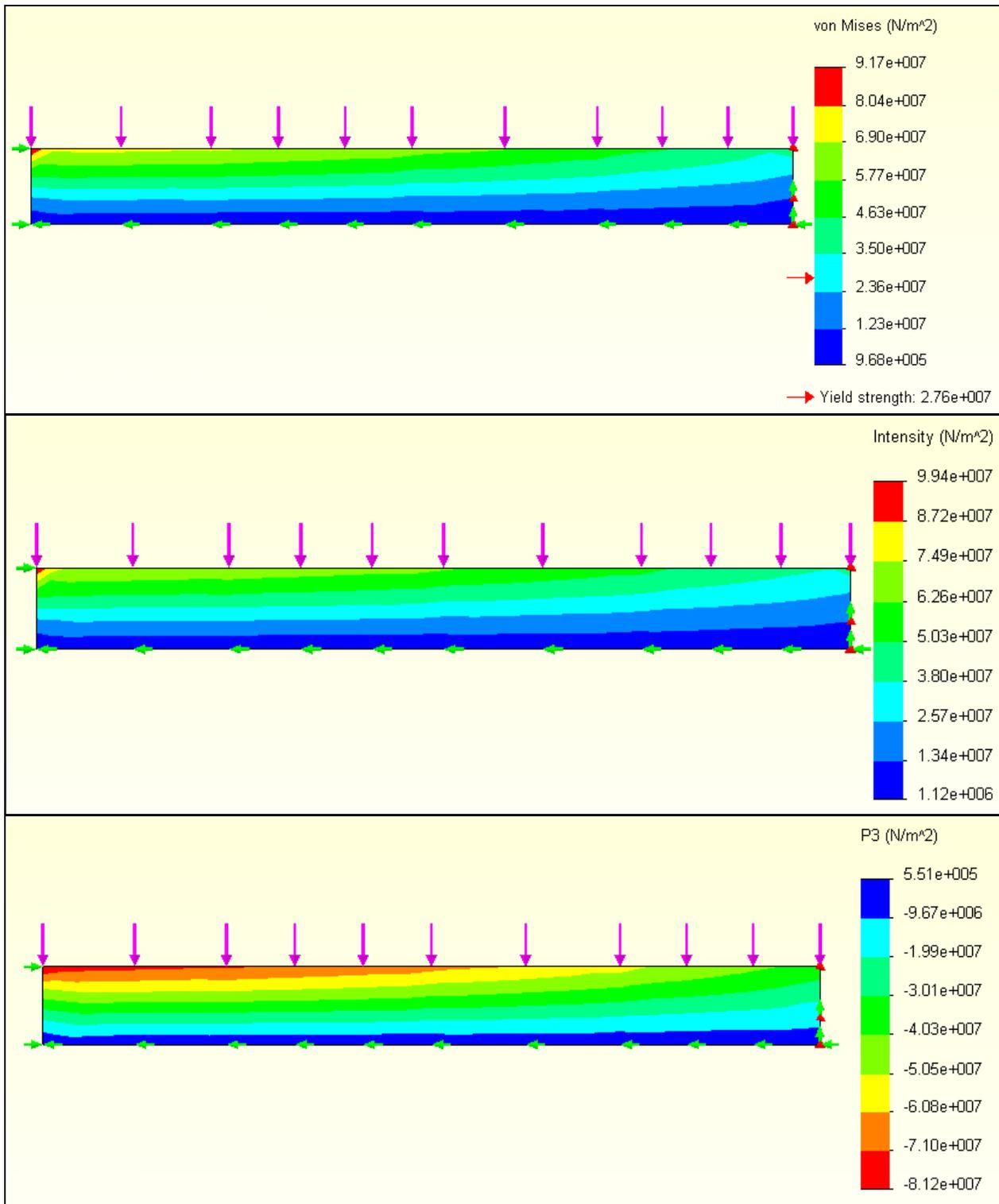


Figure 25

Then the **Report Options** display was used to **Save** those data (node number, deflection value, and x -, y -, z -coordinates) to a named file (Figure 26). Next, the data were opened in Excel, sorted by x -coordinate, and graphed as deflection versus position (Figure 27). You

could add experimental deflection values to the same file and add a second curve for comparison purposes.

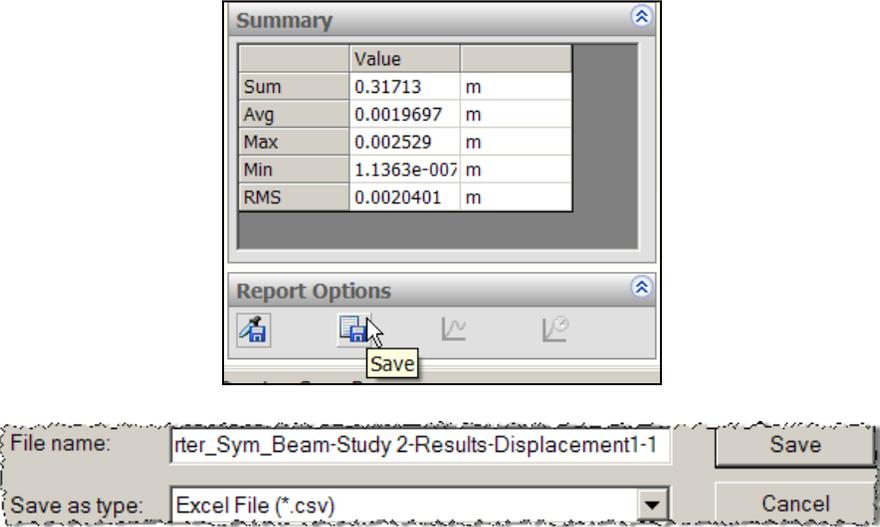


Figure 26

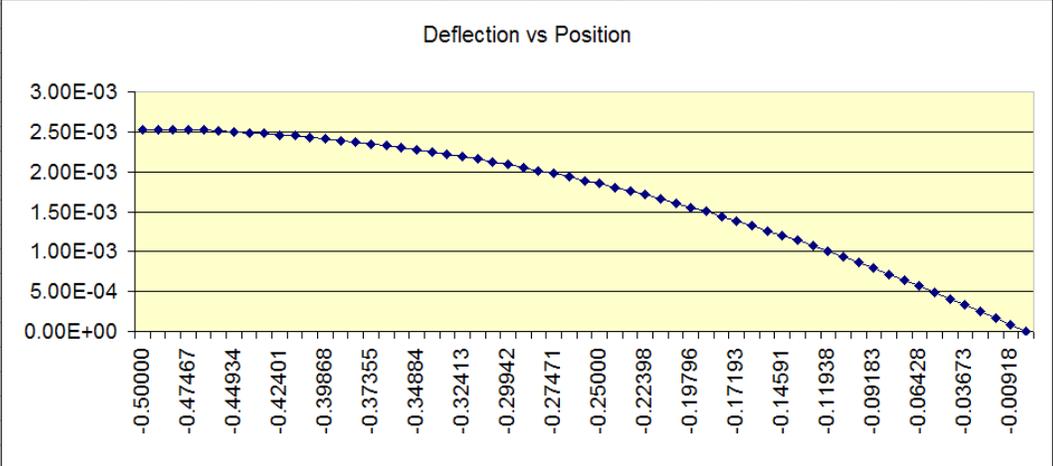


Figure 27

References

[1] R.L. Norton, Machine Design; An Intergrated Approach, Prentice Hall, 1996.
[2] E.P. Popov, Engineering Mechanics of Solids, Prentice Hall, 1990.