What is a Fixed Support? (Draft 3, 10/02/06)

History

A study of the history of mechanics of materials shows that the concept of a ***fixed***, or ***cantilevered***, or ***encastre,*** *or* ***immovable*** support came from elementary beam theory. Originally it meant that a point at the neutral axis of a beam had both a zero displacement and rotation. That also implied that the support was capable of providing a reaction force vector and moment vector.

Elementary beam theory neglects several things including the Poisson's ratio material property. When a material is loaded in one direction, it will always undergo strains perpendicular to that direction as well as in that direction. The ratio of a transverse strain to the axial strain is a material property called ***Poisson's ratio***, $\mu = \varepsilon_{lateral} / \varepsilon_{axial}$. A real material will usually contract in a direction perpendicular to the main load. Introducing an ideal restraint condition that prevents such a contraction can be misleading for 2D or 3D problems.

Engineering practice has developed standard symbols to represent a fixed support. But if you think about it, there are several ways a support can provide a resisting force and moment to prevent a region of material from translating and rotating. Some methods will prevent the secondary contraction required by Poisson's ratio. Such supports therefore introduce very high transverse stresses to hold the material in place. In theory, those "secondary" effects produce stresses that go to infinity at a point. In many applications you ignore those singularities because you know that the material will yield a little (if ductile) and/or the actual support material will move some to allow the point stress to remain finite.

Finite Element Stress Analysis

The use of finite element analysis in 2D or 3D requires you to always consider the effect of Poisson's ratio and how to avoid or model the point singularity in stresses that it causes due to restraint assumptions that you make. As a typical example, consider a horizontal tapered cantilever beam with a single transverse (vertical) force distributed over its unsupported end. Elementary beam theory gives the horizontal (*x*) fiber stress as a function of the distance from the free end, without specific regard the supported end. But in a 2D (or 3D) model the stresses will be different. Beam theory assumes that the vertical, or *y*, normal stress is zero everywhere. That is basically true in the 2D (or 3D) model because the top and bottom surfaces (and front and back faces) of the beam are free to contract, i.e. move vertically (horizontally), as required by Poisson's ratio. However, it suddenly stops being true when you consider the support region, of if there are distributed loads on the top or bottom region.

At the wall in a 2D model the top fiber is being stretched horizontally. Due to Poisson's ratio, that same point wants to contract downward. If that motion is prevented, as it is in the common assumption for a fixed support, then the vertical stress must suddenly jump from basically zero to an extremely high value over a small area to develop the support force necessary to prevent that contraction. Then theoretical stress singularities develop in the 2D (and 3D) theory of elasticity solutions, but probably not in the physical entities.

Exercise modeling choices

With finite element models you are no longer restricted to using a handful of standard symbols to represent a fixed support (or any other of the "stand" support types). You can examine the actual application and extend the solid model to include your knowledge of the actual support conditions.

For the cantilever example, assume it is steel and you know it will be welded to a much larger steel vertical member. ***Saint-Venant's principle***, from the theory of elasticity, basically says that the statically equivalent forces or supports located a few "typical dimensions" away on an object will not affect results in the main regions of the object. Therefore, consider a finite element model where the classic "fixed" support is replaced by a semi-circular region of the vertical steel member. Pick its radius as the depth of the cantilever, and fix (in the classical sense) the outer half circle. That will be statically equivalent to the standard symbol but will produce more realistic stresses near the support region.

Example cantilever member

Consider the study of a tapered cantilever member that is loaded by a vertical force at its free end. It has a length of L = 50 inches, is made of alloy steel, is b = 2 inches thick, and the end force is P = 400 lb. Its depth, h, increases from 3 to 9 inches. Rather than simply construct a trapezoid and fix the deep wall end, the model will be given a little more realistic flexibility by including a semi-circle region of the actual support that is centered on the support end of the member, as seen in Figure 1. Any re-entrant corner will always cause a local weak stress concentration so geometry could be improved by including a pair of fillets.

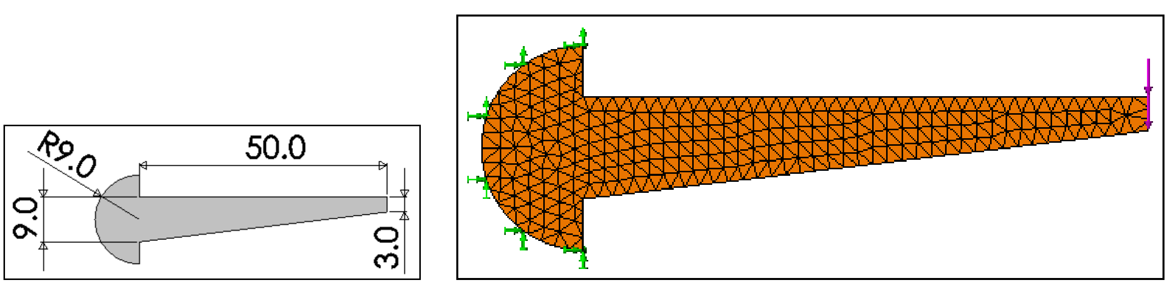

Figure 1 Cantilever member geometry and mesh

The semi-circular arc is restrained to be fixed within CosmosWorks. For a 2D or 3D continuum element (with no rotational unknowns) that means all the displacements there are zero. For a shell model it means that both the displacements and rotations are zero along that arc. In this case, the member cross-section above was extruded to the given thickness. The study will intuitively be governed by a bending behavior, so experience shows that you need a minimum of three layers of quadratic elements through the depth, or five layers of linear elements.

Plane stress assumption

The model lies in a single plane, is relatively thin, and the loading is in the same plane. That fits the definition of the state of *plane stress.* That is the state consider in most undergraduate courses on the mechanics of materials. It is a 2D approximation that assumes the deflections and stresses, but not the strains, normal to the plane are zero. Therefore, for efficiency, that 2D continuum approach is used here. It is available in CosmosWorks by defining a study using mid-surface shells. The loadings will not activate any out of plane bending so such a model actually is using only the membrane (in plane) stiffness of the shell. Only the *x-y* components of the deflection vector and stress tensor will be computed. This is more economical because each node has only two dof versus three for the solid and six for the full shell behavior. A shell mesh was created for this 2D study using a minimal resolution.

Study results

The displacement vectors are seen in Figure 2. As expected, the maximum deflections occur at the free end. The equation for the end deflection of a constant depth cantilever is well known, $u_y = P L^3 / 3 E I$, where the moment of inertia here is $I = b h^3 / 12$. This was used to successfully bound the estimated deflection by using the minimum and maximum h values. The computed value was close to the value obtained with the minimum h.

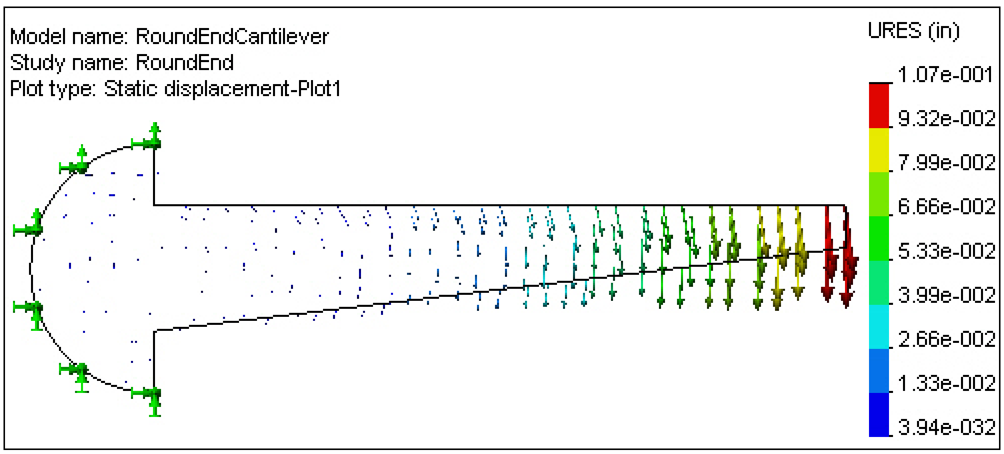

Figure 2 Member displacement vectors

The 2D model has two normal stresses and one shear stress component. Those three stress components are shown in Figure 3. The computed shear stress, τ_{xy}, is largest at the support. Beam theory states that the shear stress is zero at the top and bottom fibers

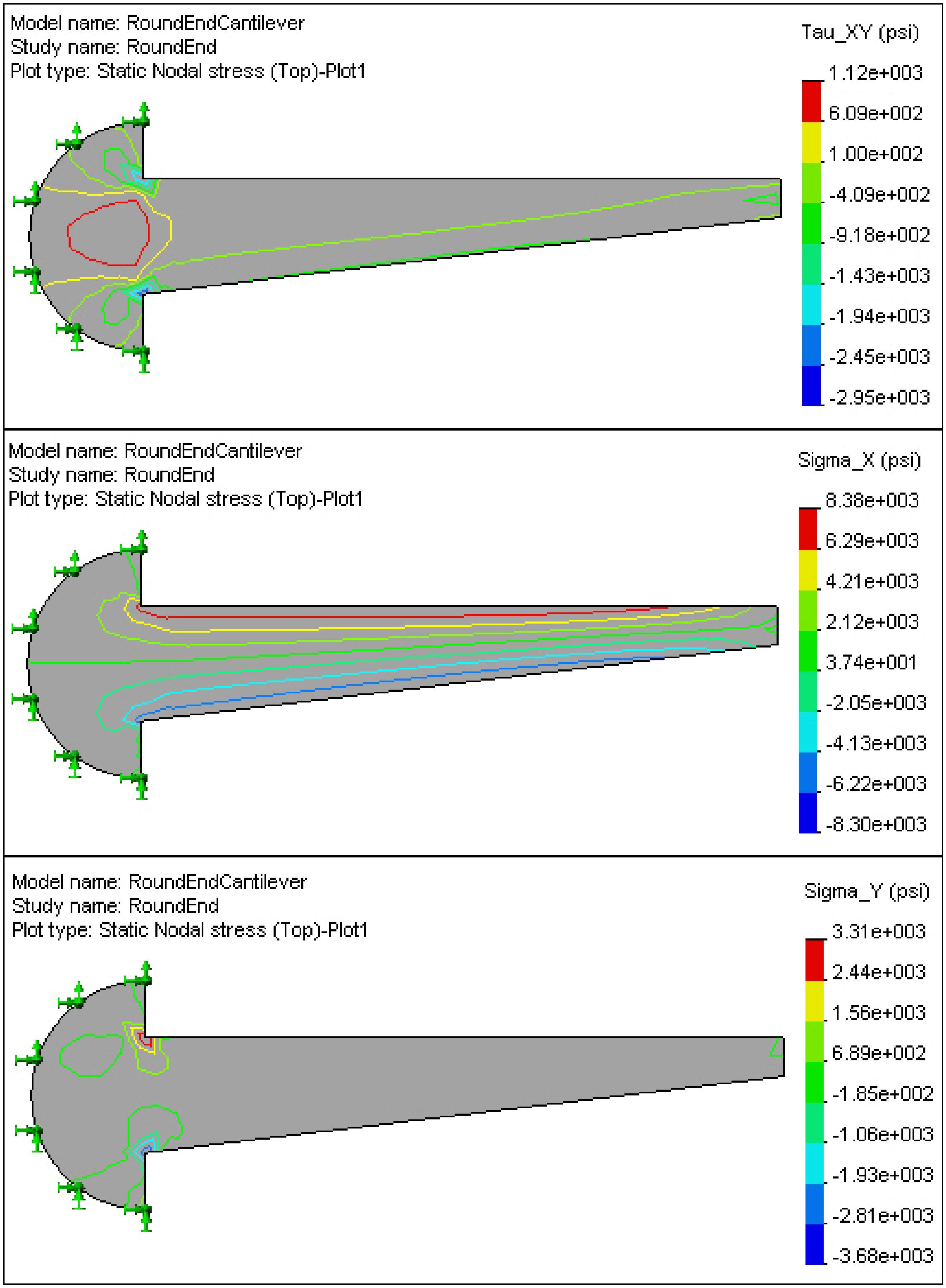

Figure 3 Cantilever x-y stress components

and varies parabolicly to a maximum at the half depth. The computed horizontal stress, σ_x, has its extreme values on the top and bottom edges, and they are largest near L/2.

Those familiar with a constant depth cantilever would know its maximum stress occurs at the support wall. However, this member does not have a constant depth and its maximum stress occurs near the mid-length of the beam. Again the simple beam theory fiber stress was used to bound the extreme values by using the maximum and minimum depth values again. Beam theory states that the horizontal normal stress varies linearly through the depth, is zero at mid-depth, and has extreme values at the top and bottom fibers. That type of distribution agrees well with the computed contours. Beam theory would assume $\sigma_y = 0$, but small values appear in its contoured value at the stress concentration points at the support corners.

Since steel is a ductile material its stress failure can be predicted by the Von Mises distortional energy measure. Thus its contour values are displayed in Figure 4. When that value exceeds the yield stress value determined from a uniaxial tension test the material is said to fail.

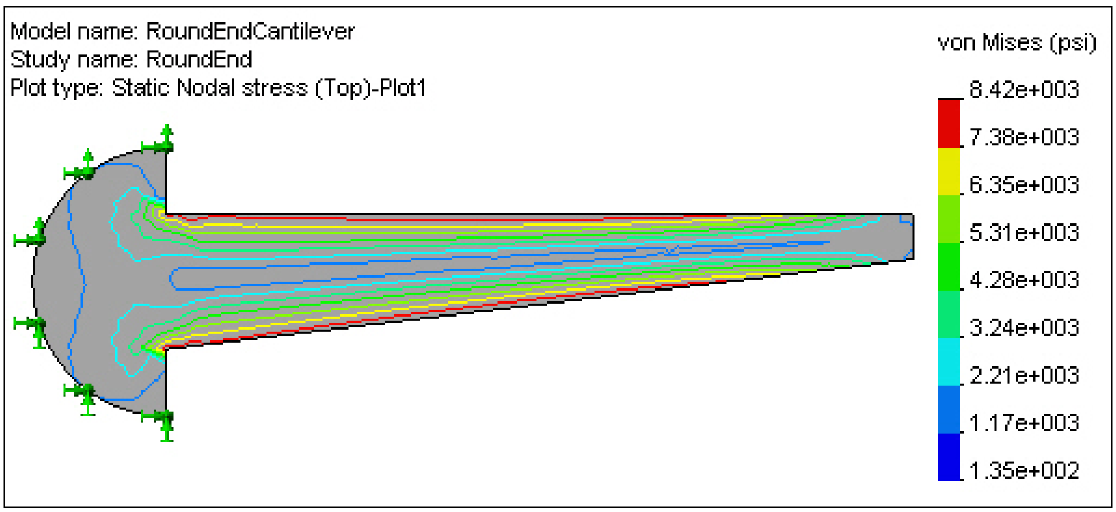

Figure 4 Von MIses failure measure

Of course, at any point the above three stress tensor components can be viewed alternately in terms of its three principle values. Since the cantilever member has both tension and compression stresses the peak values of the principle normal stresses, σ_1 and σ_2 , will be the maximum tension and compression stresses, respectively. At any point, the two normal stresses are perpendicular to each other. The magnitude and direction of the σ_1 normal stress are given in Figure 5 , and the (orthogonal) σ_2 values are seen in Figure 6.

The third principle stress is the maximum shear stress at a point. It occurs on the two orthogonal planes that at 45 degrees to the two orthogonal principle normal stress planes. The maximum shear stress value is half the difference of the maximum and minimum principle normal stresses. Here $\tau_{max} = (\sigma_1 - \sigma_2)/2$. CosmosWorks denotes the maximum difference in the principle normal stresses by INT and refers to it as stress intensity. From

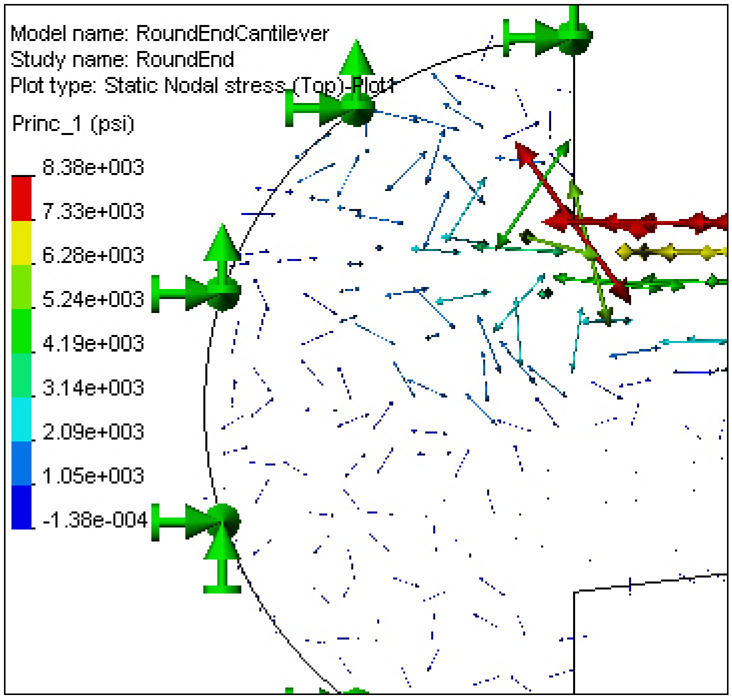

Figure 5 Maximum principle stress directions and values

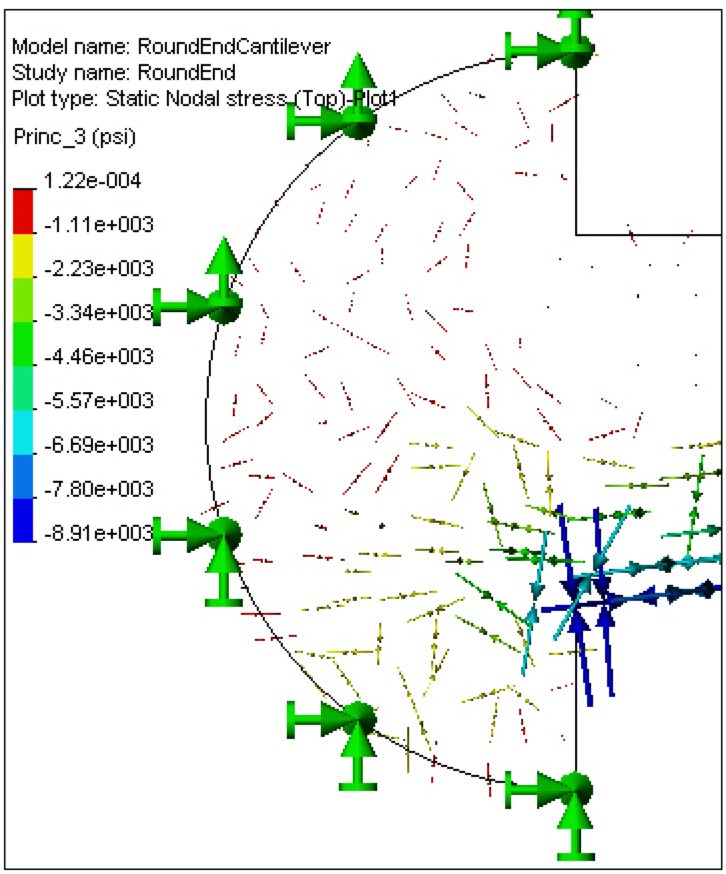

Figure 6 Minimum principle normal stress directions and values

the last equation you can see that it is also twice the maximum shear stress. For this member the stress intensity is given in Figure 7. The maximum shear stress is often also used as the failure criterion for ductile materials. It states that failure occurs when the maximum shear stress exceeds the shear stress in a tensile specimen at yield (i.e., half of the yield stress). Rather than compare the maximum shear stress to half the yield, it is easier to compare twice its value, INT of Figure 7, to the yield stress.

The Von Mises failure criterion for ductile material states that failure occurs when the combined stresses cause a distortional energy value that is equal to that in a tensile specimen at yield. It reduces to comparing a scalar “effective stress” to the yield stress. Therefore in CosmosWorks you want to remember the material yield stress and compare it to the Von Mises value (Figure 4) and the Intensity value (Figure 7). As seen in Figure 8, the maximum shear stress criterion is more conservative since it predicts failure at a lower combined stress value.

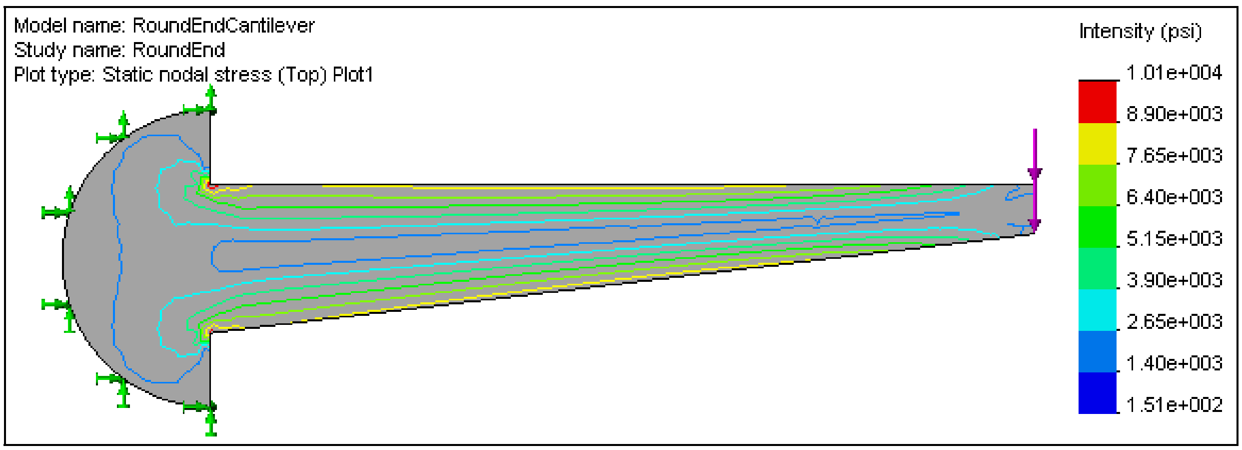

Figure 7 Twice the maximum shear stress

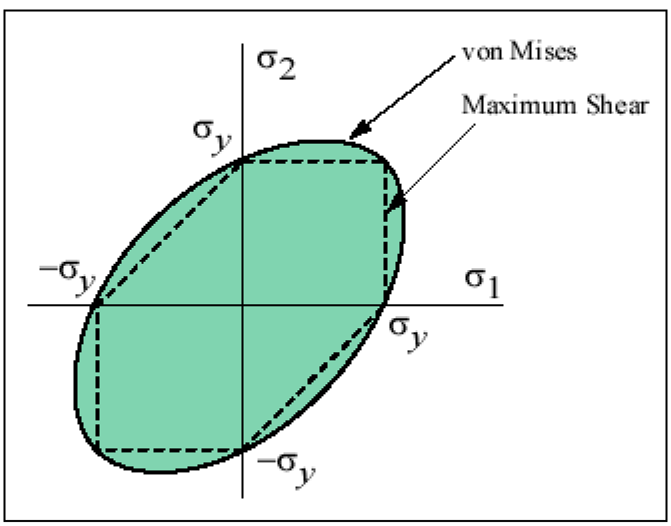

Figure 8 Ductile material failure criteria