# Concepts of Stress Analysis

Part 1, draft 2, 09/30/06

## *Introduction*

In the previous review of heat transfer it was pointed out that the time independent equations for one-dimensional heat transfer and one-dimensional stress analysis are the same. A useful analogy between the two phenomena was also pointed out at that time. 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 thin 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, $\mathbf{u} \leftrightarrow (u, v, w)$. 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 1 for 1D and 2D geometry. 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

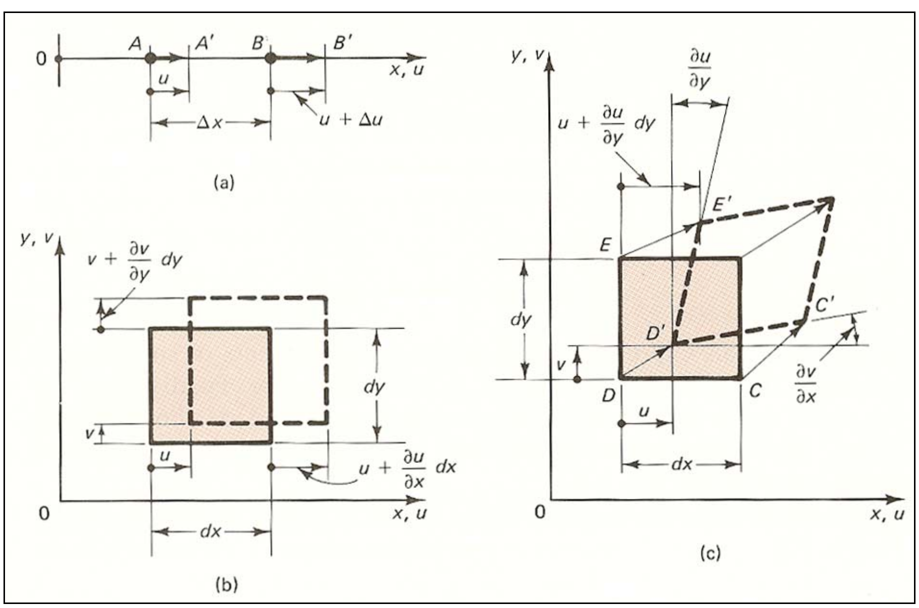



**Figure 1 Geometry of normal strain (a) 1D, (b) 2D, and (c) 2D shear strain**

2D 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 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 it is $\gamma = (\partial u / \partial y + \partial v / \partial x)$, as seen in Figure 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, $\boldsymbol{\varepsilon} = (\varepsilon_x \quad \varepsilon_y \quad \gamma)^T$.

Like the heat transfer case, 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**: $\boldsymbol{\sigma} = \mathbf{E}\, \boldsymbol{\varepsilon}$, (see Figure 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, $\boldsymbol{\sigma} = (\sigma_x \quad \sigma_y \quad \tau)^T$. The **potential energy** stored in the differential element is half the scalar product of the stresses and the strains. The 2D and 3D stress components are shown in Figure 3. The normal and shear stresses represent the normal force per unit area and the tangential forces per unit area, respectively. Therefore they have the units of [N/m^2], or [Pa].

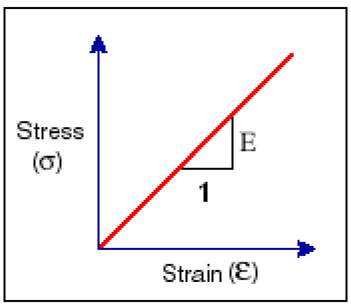



**Figure 2 Hooke's Law for stress-strain**

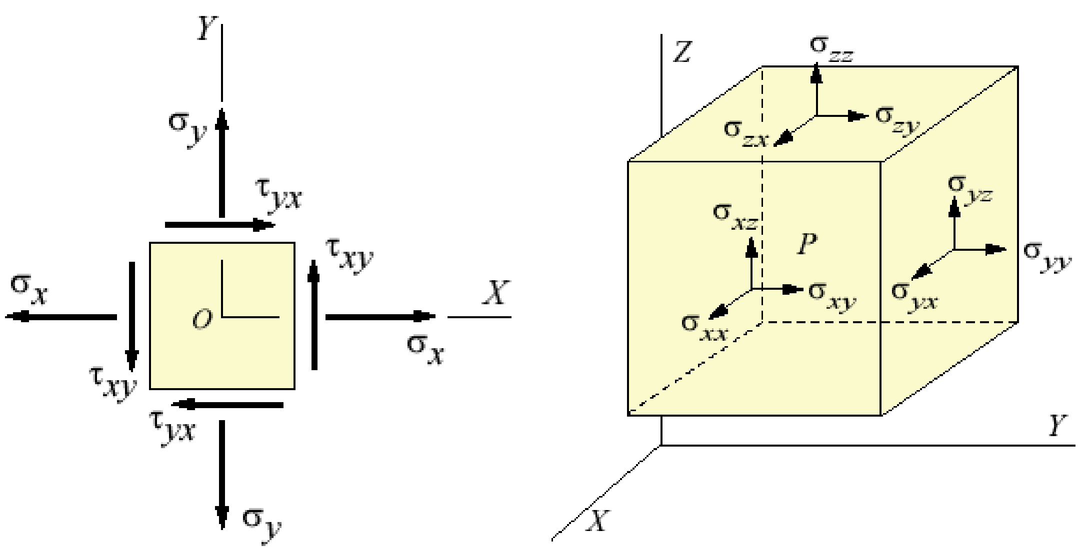
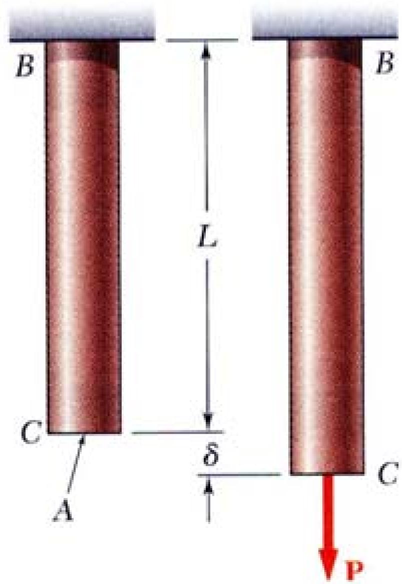



**Figure 3 Stress components in 2D (left) and 3D**

## *Example*

The simplest available example is an axial bar, shown in Figure 4, 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, $u(x) = x\,\delta / L$, so the axial strain is $\varepsilon_x = \partial u / \partial x = \delta / L$, which is a constant. Likewise, the axial stress is everywhere constant, $\sigma = E\,\varepsilon = E\,\delta / L$ which in the case simply reduces to $\sigma = P / A$.




$$\sigma = P / A, \quad \delta = P L / E A$$

**Figure 4 A linearly elastic bar with an axial load**

Like many other more complicated problems, the stress here does not depend on the material properties, but the displacement always does. Therefore you should always carefully check both the deflections and stresses when validating a finite element solution.

Since the 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.

## *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 5**. 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 Mise's  stress, or effective stress, even though it is actually a scalar.  A brittle material requires the use of a higher factor of safety.

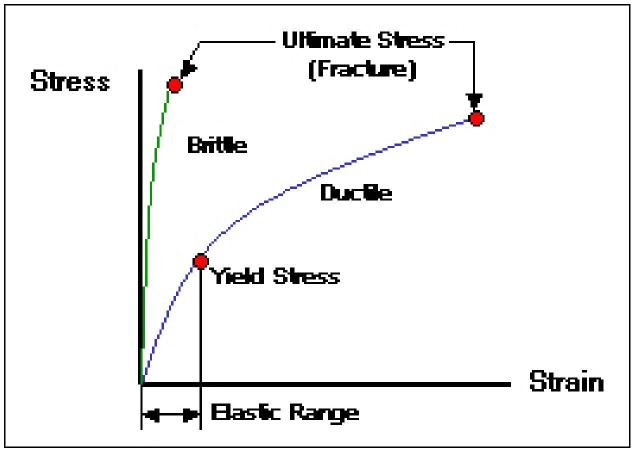



Figure 5  Axial stress-strain experimental results