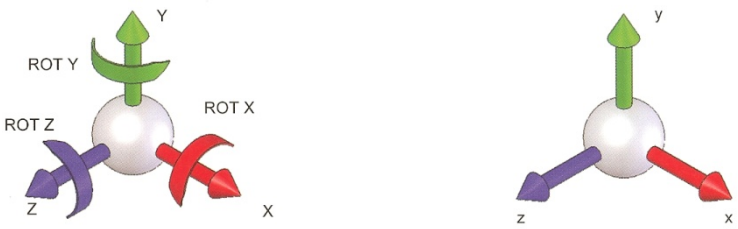
3 Concepts of Stress Analysis

3.1 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 SW Simulation. They differ in that the continuum form utilizes only displacement vectors, while the shell form utilizes displacement vectors *and* infinitesimal rotation vectors at the element nodes.

As illustrated in Figure 3-1, 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.



Shell node Solid node

Figure 3-1 Nodal degrees of freedom for frames and shells; solids and trusses

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 (covered later), 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 is 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 3-2 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 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 3-2 (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 components of the gradient that

 Copyright 2009. All rights reserved.

are 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 3-2(c). Strain has one component in 1D, three components in 2D, and six components in 3D. The 2D strains are commonly written as a column vector in finite element analysis, $\boldsymbol{\varepsilon} = (\varepsilon_x \quad \varepsilon_y \quad \gamma)^T$.

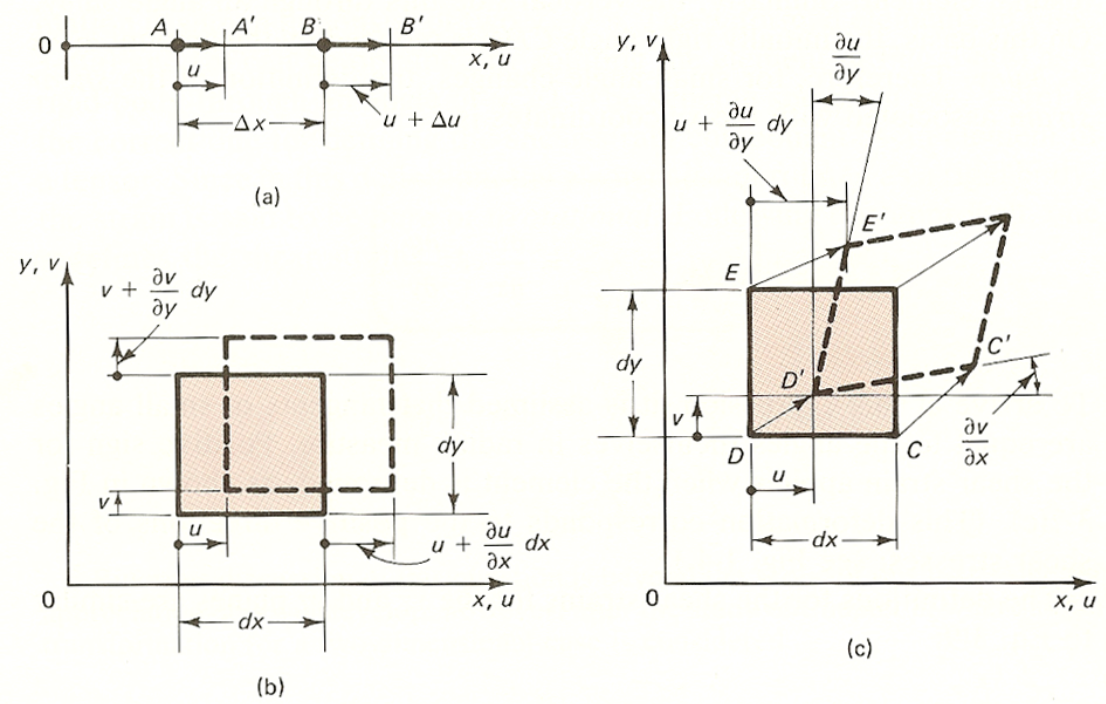


Figure 3-2 Geometry of normal strain (a) 1D, (b) 2D, and (c) 2D shear strain

Stress is a measure of the force per unit area acting on a plane passing through the point of interest in a body. 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**: $\boldsymbol{\sigma = E\,\varepsilon}$, (see Figure 3-3) where the square **material matrix**, **E**, contains the elastic modulus, and Poisson's ratio of the material. The 2D stresses are written as a corresponding column vector, $\boldsymbol{\sigma} = (\sigma_x \quad \sigma_y \quad \tau)^T$. Unless stated otherwise, the applications illustrated here are assume to be in the linear range of a material property.

The 2D and 3D stress components are shown in Figure 3-4. 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 3-5. 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 primarily on the strain energy (or strain energy density).

Figure 3-3 Hooke's Law for linear stress-strain, $\boldsymbol{\sigma = E\,\varepsilon}$

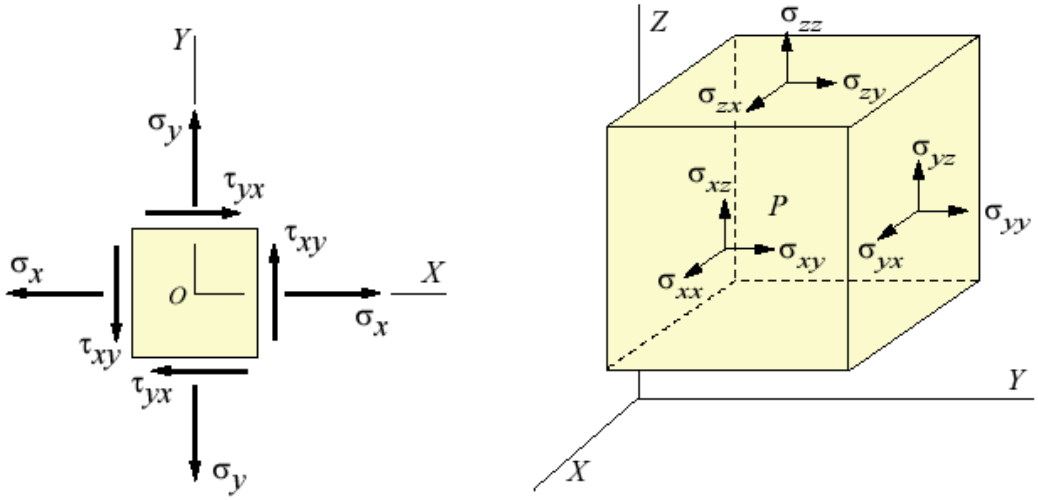


Figure 3-4 Stress components in 2D (left) and 3D

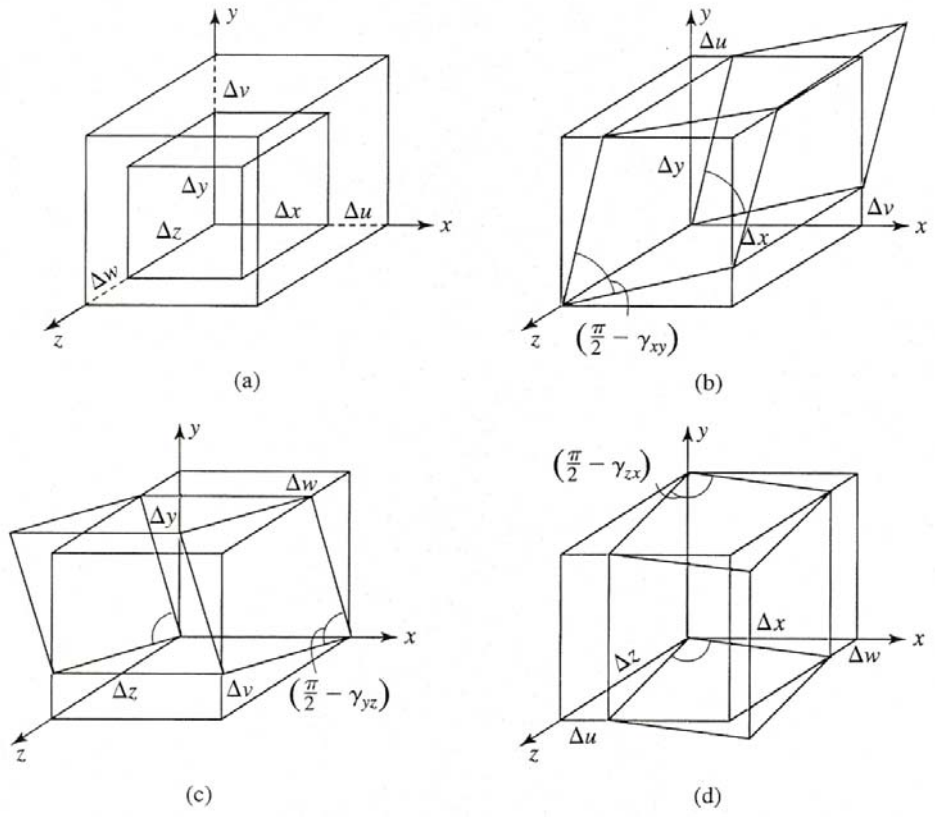


Figure 3-5 Graphical representations of 3D normal strains (a) and shear strains

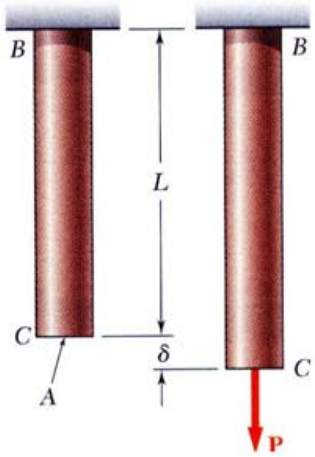
3.2 Axial bar example

The simplest available stress example is an axial bar, shown in Figure 3-6, restrained at one end and subjected to an axial load, *P*, at the other end and the weight is neglected. 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 $\sigma = P / A$. Like many other more complicated problems, the stress here does not depend on the material properties, but the displacement always does, $\delta = PL/EA$. 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 the constant stress results. However, if the load had been the distributed bar weight the exact displacement would be 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. Then the above bar displacement can be written as $\delta = P/k$, like a linear spring.



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

Figure 3-6 A linearly elastic bar with an axial load

3.3 Structural mechanics

Modern structural analysis relies extensively on the finite element method. The 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, and the strains. The total potential energy, Π, is the strain energy, U, of the structure minus the mechanical work, W, done by the external 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 an energy formulation. 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 the given displacement restraint.

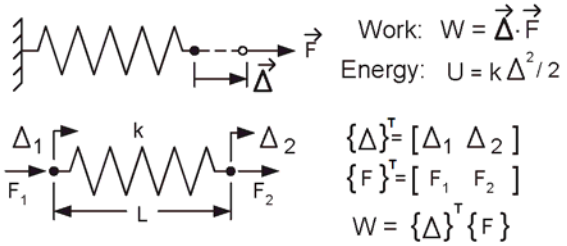


Figure 3-7 The classic and general linear spring element

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

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

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

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

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:

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

That 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 (the axial RBM has not been eliminated). Mathematically, this is clear because the square matrix has a zero determinate and cannot 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 the first row of the matrix equations gives $F_1 = -2\,k\,C$. The second row gives $F_2 = 2\,k\,C$, which is equal and opposite to F_1, as expected. Usually, you know some of the displacements and some of the forces. Then you have to manipulate the matrix equilibrium system to put it in the form of a standard linear algebraic system where a known square matrix multiplied by a vector of unknowns is equal to a known vector: $[A]\{x\} = \{b\}$.

3.4 Equilibrium of restrained systems

Like the original spring problem, now assume the right force, $\boldsymbol{F_2}$, 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 equals, "=". You could (but usually do not) correct that by re-arranging the equation system (not done in practice). First, multiply the first column of the stiffness matrix by the known Δ_{given} value and move it to the right side:

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

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

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

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 energy 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 exact solution:

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

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

so that $\boldsymbol{F_1 = -F_2}$ always, as expected. If $\Delta_{\text{given}} = 0$, as originally stated, then the end displacement is $\Delta_2 = F_2/k$. This sort of re-arrangement of the matrix terms is not done in practice because it destroys the symmetry of the original equations. Algorithms for numerically solving such systems rely on symmetry to reduce both the required storage size and the operations count. They are very important when solving thousands of equations.

Draft 13.0. Copyright 2009. All rights reserved.

3.5 General equilibrium matrix partitions

The above small example gives insight to the most general form of the algebraic system resulting 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 to the problem 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, symmetric matrix equations obtained by minimizing the energy can always be rearranged into the following partitioned matrix form:

$$\begin{bmatrix} \boldsymbol{K_{uu}} & \boldsymbol{K_{ug}} \\ \boldsymbol{K_{gu}} & \boldsymbol{K_{gg}} \end{bmatrix} \begin{Bmatrix} \Delta_{\boldsymbol{u}} \\ \Delta_{\boldsymbol{g}} \end{Bmatrix} = \begin{Bmatrix} \boldsymbol{F_g} \\ \boldsymbol{F_u} \end{Bmatrix}$$

where $\boldsymbol{\Delta_u}$ represents the unknown nodal displacements, and $\boldsymbol{\Delta_g}$ represents the given essential boundary values (restraints, or fixtures) of the other displacements. The stiffness sub-matrices $\boldsymbol{K_{uu}}$ and $\boldsymbol{K_{gg}}$ are square, whereas $\boldsymbol{K_{ug}}$ and $\boldsymbol{K_{gu}}$ are rectangular. In a finite element formulation all of the coefficients in the $\boldsymbol{K}$ matrices are known. The resultant applied nodal loads are in sub-vector $\boldsymbol{F_g}$ and the $\boldsymbol{F_u}$ 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 $\boldsymbol{\Delta_u}$ and $\boldsymbol{F_u}$. 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:

$$\boldsymbol{K_{uu}}\,\Delta_{\boldsymbol{u}} + \boldsymbol{K_{ug}}\Delta_{\boldsymbol{g}} = \boldsymbol{F_g}$$

$$\boldsymbol{K_{gu}}\,\Delta_{\boldsymbol{u}} + \boldsymbol{K_{gg}}\Delta_{\boldsymbol{g}} = \boldsymbol{F_u}.$$

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

$$\Delta_{\boldsymbol{u}} = \boldsymbol{K_{uu}}^{-1}\left(\boldsymbol{F_g} - \boldsymbol{K_{ug}}\Delta_{\boldsymbol{g}}\right).$$

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

$$\boldsymbol{F_u} = \boldsymbol{K_{gu}}\,\Delta_{\boldsymbol{u}} + \boldsymbol{K_{gg}}\Delta_{\boldsymbol{g}}.$$

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.

3.6 Structural 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 brittle and ductile material behaviors is determined by their response to a uniaxial stress-strain test, as in Figure 3-8. You need to know what class of material is being used. SW Simulation, 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.

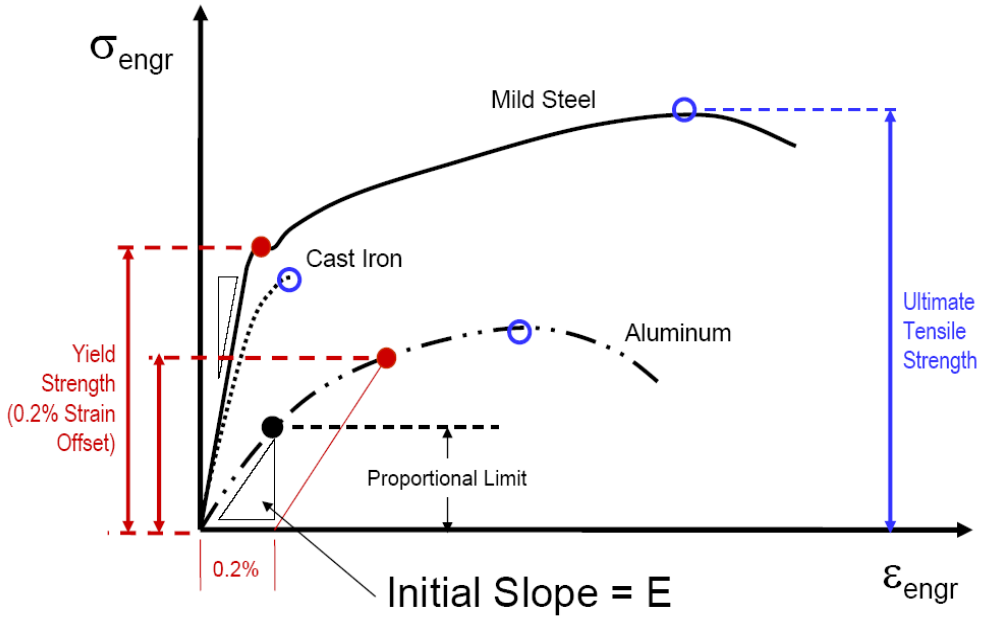


Figure 3-8 Axial stress-strain experimental results

3.7 Factor of Safety

All aspects of a design have some degree of uncertainty, as does how the design will actually be utilized. For all the reasons cited above, you must always employ a Factor of Safety (FOS). Some designers refer to it as the factor of ignorance. 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]. They should be supplemented with the additional uncertainties that come from a FEA. Many authors suggest that the factor of safety should be computed as the product of terms that are all ≥ 1. There is a factor for the certainty of the restraint location and type; the certainty of the load region, type, and value; a material factor; a dynamic loading factor; a cyclic (fatigue) load factor; and an additional factor if failure is likely to result in human injury. Various professional organizations and standards organizations set minimum values for the factor of safety. For example, the standard for lifting hoists and elevators require a minimum FOS of 4, because their failure would involve the clear risk of injuring or killing people. As a guide, consider the FOS as a product of factors: $FOS = \prod_{k=1}^{n} F_k = F_1 F_2 F_3 \ldots F_n$. A set of typical factors is given inTable 3-1.

Table 3-1 Factors to consider when evaluating a design (each ≥ 1)

k	Type	Comments
1	Consequences	Will loss be okay, critical or fatal
2	Environment	Room-ambient or harsh chemicals present
3	Failure theory	Is a part clearly brittle, ductile, or unknown
4	Fatigue	Does the design experience more that ten cycles of use
5	Geometry of Part	Not uncertain, if from a CAD system

6	Geometry of Mesh	Defeaturing can introduce errors. Element sizes and location are important. Looking like the part is not enough.
7	Loading	Are loads precise or do they come from wave action, etc.
8	Material data	Is the material well known, or validated by tests
9	Reliability	Must the reliability of the design be high
10	Restraints	Designs are greatly influenced by assumed supports
11	Stresses	Was stress concentration considered, or shock loads

3.8 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. SW Simulation offers a small element library that includes bars, trusses, beams, frames, thin plates and shells, thick plates and shells, and solid elements. There are also special connector elements called rigid links or multipoint constraints.

The shells and solid elements are considered to be continuum elements. The plate elements are a special case of flat shells with no initial curvature. Solid element formulations include the stresses in all directions. Shells are a mathematical simplification of solids of special shape. Thin shells (like thin beams) do not consider the stress in the direction perpendicular to the shell surface. Thick shells (like deep beams) do consider the stresses through the thickness on the shell, in the direction normal to the middle surface, and account for transverse shear deformations.

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 shells are extensions of thin shell elements that contain additional strain energy terms.

The overlapping h/L ranges for the three continuum element types are suggested in Figure 3-9. 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.

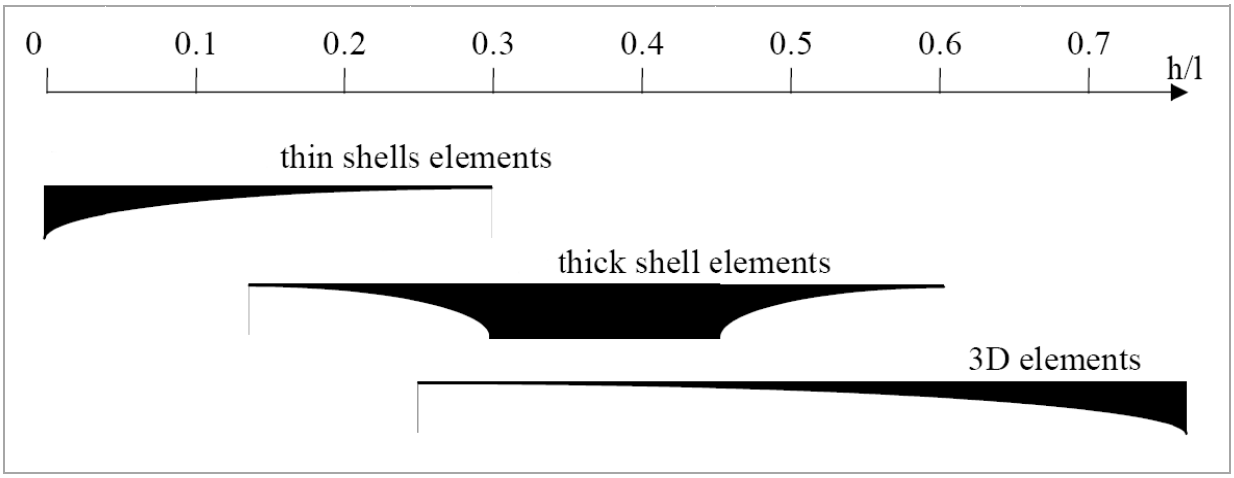


Figure 3-9 Overlapping valid ranges of element types

3.9 SW Simulation Fixture and Load Symbols

The symbols used in SW Simulation to represent a single translational and rotational DOF at a node are shown green in Figure 3-10. 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 SW Simulation nodal symbols for the unknown generalized displacement DOF's for the solid nodes (top) and shell nodes are seen **in** Figure 3-11. You almost always must supply enough restraints to prevent any model from undergoing a rigid body translation or rigid body rotation.

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 3-10 Fixed restraint symbols for solids (top) and shell nodes

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 3-11 Single component symbols for restraints (fixtures) and loads

3.10 Symmetry DOF on a Plane

A plane of symmetry is flat and has mirror image geometry, material properties, loading, and restraints. Symmetry restraints\i are very common for solids and for shells. Figure 3-12 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 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 3-12 Symmetry requires zero normal displacement, and zero in-plane rotation

3.11 Available Loading (Source) Options

Most finite element systems have a wide range of mechanical loads (or sources) that can be applied to points, curves, surfaces, and volumes. The mechanical loading terminology used in SW Simulation is in Table 3-2. Most of those loading options are utilized in later example applications.

Table 3-2 Mechanical loads (sources) that apply to the active structural study

Load Type	Description
Bearing Load	Non-uniform bearing load on a cylindrical face
Centrifugal Force	Radial centrifugal body forces for the angular velocity and/or tangential body forces from the angular acceleration about an axis
Force	Resultant force, or moment, at a vertex, curve, or surface
Gravity	Gravity, or linear acceleration vector, body force loading
Pressure	A pressure having normal and/or tangential components acting on a selected surface
Remote Load / Mass	Allows loads or masses remote from the part to be applied to the part by treating the omitted material as rigid
Temperature	Temperature change at selected curves, surfaces, or bodies (see thermal studies for more realistic temperature transfers)

3.12 Available Material Inputs for Stress Studies

Most applications involve the use of isotropic (direction independent) materials. The available mechanical properties for them in SW Simulation are listed in Table 3-3. 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 material directions reference plane or coordinate axes before entering orthotropic data. Mechanical orthotropic properties are subject to some theoretical

 Copyright 2009. All rights reserved.

relationships that physically possible materials must satisfy (such as positive strain energy). Thus, experimental material properties data may require adjustment before being accepted by SW Simulation.

Table 3-3 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 3-4 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

Note: NUXY, NUYZ, and NUXZ are not independent

Parts can also be made from orthotropic materials (as shown later). However, their utilization is most common in laminated materials (laminates) where they each ply layer has a controllable principal material direction. The concept for constructing laminates from orthotropic material ply's is shown in Figure. Understanding the failure modes of laminates usually requires special study.

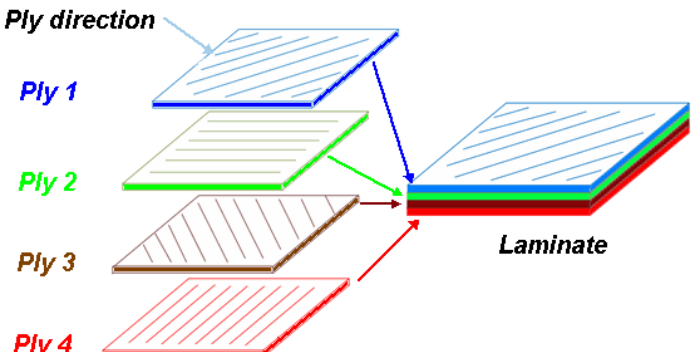


Figure 3-13 Example of a four-ply laminate material

3.13 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 SW Simulation stress study. The available displacement vector components are cited in Table 3-5 and Table 3-6, 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. They can also be transformed to cylindrical or spherical components.

Table 3-5 Output results for solids, shells, and trusses

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

Table 3-6 Additional primary results for beams, plates, and shells

Symbol	Label	Item	Symbol	Label	Item
$\boldsymbol{\theta_x}$	RX	Rotation (X direction)	M_x	RMX:	Reaction moment (X direction)
$\boldsymbol{\theta_y}$	RY	Rotation (Y direction)	M_y	RMY	Reaction moment (Y direction)
$\boldsymbol{\theta_z}$	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 3-7. 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 3-14. 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 3-14. 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.

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. SW Simulation 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 3-7 are often represented graphically in mechanics as a 3D Mohr's circle (seen in Figure 3-15).

Table 3-7: 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)

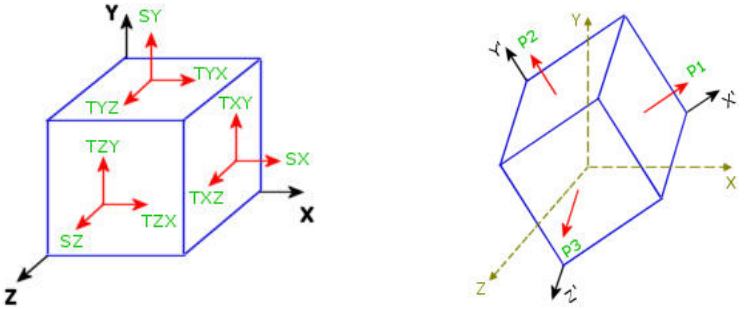


Figure 3-14 The stress tensor (left) and its principal normal values

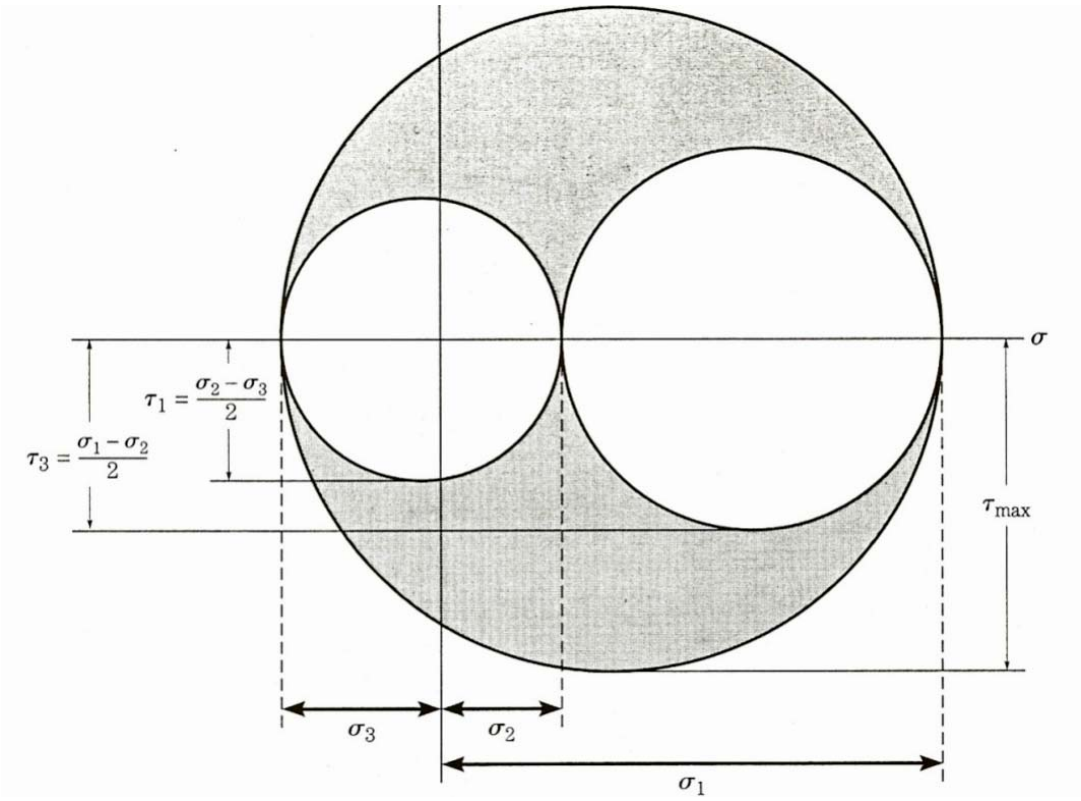


Figure 3-15 The three-dimensional Mohr's circle of stress yield the principal stresses

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.

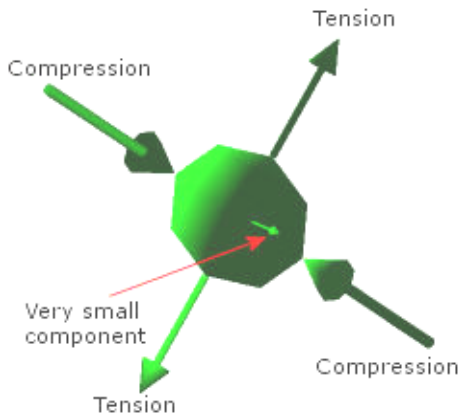


Figure 3-16 A principal stress ellipsoid colored by von Mises value

The available nodal output results in Table 3-7 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 3-8. Table 3-9 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. Additional outputs are available if you conduct an automated adaptive analysis to reduce the (mathematical) error to a specific value, or to recover results from the developed pressure between contacting surfaces. They are listed in Table 3-9.

Table 3-8 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 3-9 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