

Chapter 11

VECTOR FIELDS.

11.1 Introduction

Here we will extend the concepts from the previous chapter on scalar fields to those that involve vector unknowns. The most common (and historically the first) application is that is displacement based stress analysis. Other electrical field applications will only be noted through recent publications on those techniques which often require different 'edge based' interpolation methods not covered here. The main new considerations here are to select ordering for the vector unknowns and related vector or tensor quantities so that they can be cast in an expanded matrix notation.

11.2 Displacement Based Stress Analysis Summary

Let \mathbf{u} denote a displacement vector at a point \mathbf{x} in a solid. The finite element displacement formulation for stress analysis is based on the energy concept of finding $\mathbf{u}(\mathbf{x})$ that both satisfies the essential boundary conditions on \mathbf{u} and minimizes the Total Potential Energy:

$$\Pi(\mathbf{u}) = U - W$$

where W is the external mechanical work due to supplied loading data involving forces per unit volume, \mathbf{X} , surface tractions per unit area, \mathbf{T} , and point loads, \mathbf{P}_j , at point \mathbf{x}_j so

$$W(\mathbf{u}) = \int_{\Omega} \mathbf{u}^T \mathbf{X} d\Omega + \int_{\Gamma} \mathbf{u}^T \mathbf{T} d\Gamma + \sum_j \mathbf{u}_j^T \mathbf{P}_j$$

and $U(\mathbf{u})$ consists of the strain energy due to the deformation, \mathbf{u} , of the material. It is defined as

$$U = \frac{1}{2} \int_{\Omega} \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} d\Omega$$

where $\boldsymbol{\varepsilon}$ are the strain components resulting from the displacements, \mathbf{u} , and $\boldsymbol{\sigma}$ are the stress components that correspond to the strain components.

The strains, $\boldsymbol{\varepsilon}$, are defined by a "Strain-Displacement Relation" for each class of stress analysis. This relation can be represented as a differential operator matrix, say \mathbf{L} ,

acting on the displacements $\boldsymbol{\varepsilon} = \mathbf{L}\mathbf{u}$. There are four commonly used classes that we will consider here. They are

1. Axial stress: $\mathbf{u} = u$, $\mathbf{L} = \partial/\partial x$, $\boldsymbol{\varepsilon} = \varepsilon_x = \partial u/\partial x$

2. Plane Stress and Plane Strain: $\mathbf{u}^T = [u \ v]$

$$\mathbf{L}^T = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & \frac{\partial}{\partial y} \\ 0 & \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix}$$

$$\boldsymbol{\varepsilon}^T = [\varepsilon_x \ \varepsilon_y \ \gamma] = \begin{bmatrix} \frac{\partial u}{\partial x} & \frac{\partial v}{\partial y} & \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \end{bmatrix}$$

3. Axisymmetric solid with radius R and axial position z : $\mathbf{u}^T = [u \ v]$

$$\mathbf{L}^T = \begin{bmatrix} \frac{\partial}{\partial R} & 0 & \frac{\partial}{\partial z} & 1/R \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial R} & 0 \end{bmatrix}$$

$$\boldsymbol{\varepsilon}^T = [\varepsilon_r \ \varepsilon_z \ \gamma \ \varepsilon_\theta] = \begin{bmatrix} \frac{\partial u}{\partial R} & \frac{\partial u}{\partial z} & \left(\frac{\partial u}{\partial z} + \frac{\partial v}{\partial R}\right) & u/R \end{bmatrix}$$

4. The full three-dimensional solid: $\mathbf{u}^T = [u \ v \ w]$

$$\mathbf{L}^T = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 & \frac{\partial}{\partial y} & \frac{\partial}{\partial z} & 0 \\ 0 & \frac{\partial}{\partial y} & 0 & \frac{\partial}{\partial x} & 0 & \frac{\partial}{\partial z} \\ 0 & 0 & \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x} & \frac{\partial}{\partial y} \end{bmatrix}$$

$$\boldsymbol{\varepsilon}^T = \left[\varepsilon_x \ \varepsilon_y \ \varepsilon_z \ \gamma_{xy} \ \gamma_{xz} \ \gamma_{yz} \right] = \left[\frac{\partial u}{\partial x} \ \frac{\partial v}{\partial y} \ \frac{\partial w}{\partial z} \ \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \ \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right) \ \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right) \right].$$

The above notations for strains is called the engineering definition. There is a more general definition, that we will not use, called the strain tensor which is $\boldsymbol{\varepsilon}_{jk} = (u_{j,k} + u_{k,j})/2$. The classic elasticity definitions are expressed in a matrix form and do not yet include any finite element assumptions. Within any element assume that we interpolate the displacement vector in terms of nodal displacement vector components, $\boldsymbol{\delta}^e$:

$$\mathbf{u}(\mathbf{x}) \equiv \mathbf{N}(\mathbf{x})\boldsymbol{\delta}^e \quad \mathbf{x} \in \Omega^e$$

where \mathbf{N} denotes that spatial interpolation for vector components. Then, in all cases, we can define a common notation for the element strains in terms of the element displacement:

$$\boldsymbol{\varepsilon} = \mathbf{L}(\mathbf{x})\mathbf{N}^e(x)\boldsymbol{\delta}^e = \mathbf{B}^e(x)\boldsymbol{\delta}^e$$

where the differential operator action on the displacement vector interpolations defines the "B-matrix":

$$\mathbf{B}^e(\mathbf{x}) = \mathbf{L}(\mathbf{x})\mathbf{N}^e(\mathbf{x})$$

Usually we use the same interpolation for each of the scalar components of the displacement vector, \mathbf{u} . For example

$$u(\mathbf{x}) = \mathbf{H}^e(\mathbf{x})\mathbf{u}^e, \quad v(\mathbf{x}) = \mathbf{H}^e(\mathbf{x})\mathbf{v}^e, \quad w(\mathbf{x}) = \mathbf{H}^e(\mathbf{x})\mathbf{w}^e$$

and we order the unknown element displacements as $\boldsymbol{\delta}^{eT} = [u_1 \ v_1 \ w_1 \ u_2 \ v_2 \ \dots \ v_m \ w_m]$ for an element with "m" nodes. Thus the vector interpolation matrix, \mathbf{N} , simply contains the \mathbf{H} scalar interpolation functions and usually some zeros;

$$\mathbf{N}(\mathbf{x}) = \mathbf{N}(\mathbf{H}(\mathbf{x})) = [\mathbf{N}_1 \ \mathbf{N}_2 \ \dots \ \mathbf{N}_m]$$

With the above choices for the order of the unknowns in $\boldsymbol{\delta}^e$ we note that for our four analysis classes the vector interpolation sub-matrices at node j are:

1. Axial stress: $\mathbf{N}_j(\mathbf{x}) = H_j(x)$
2. - 3. Two-dimensional and axisymmetric cases

$$\mathbf{N}_j(\mathbf{x}) = \begin{bmatrix} H_j(x) & 0 \\ 0 & H_j(x) \end{bmatrix}$$

4. Three-Dimensional Case

$$\mathbf{N}_j(\mathbf{x}) = \begin{bmatrix} H_j(x) & 0 & 0 \\ 0 & H_j(x) & 0 \\ 0 & 0 & H_j(x) \end{bmatrix}$$

or in general if we use an identity matrix, \mathbf{I} , the size of the displacement vector:

$$\mathbf{N}_j(\mathbf{x}) = \mathbf{I}H_j(\mathbf{x}).$$

Returning to the elasticity notations, the corresponding stress components for our four cases are

1. Axial Stress: $\boldsymbol{\sigma} = \sigma_x$
2. Plane Stress, Plane Strain: $\boldsymbol{\sigma}^T = \begin{bmatrix} \sigma_x & \sigma_y & \tau \end{bmatrix}$
3. Axisymmetric Solid: $\boldsymbol{\sigma}^T = \begin{bmatrix} \sigma_R & \sigma_Z & \tau & \sigma_\theta \end{bmatrix}$
4. Three-dimensional Solid: $\boldsymbol{\sigma}^T = \begin{bmatrix} \sigma_x & \sigma_y & \sigma_z & \tau_{xy} & \tau_{xz} & \tau_{yz} \end{bmatrix}$

where the τ terms denote shear stresses and the other terms are normal stresses. For the state of plane strain σ_z is not zero, but is recovered in a post-processing operation. The mechanical stresses are related to the mechanical strains by a material constitutive law usually known as Hooke's Law:

$$\boldsymbol{\sigma} = \mathbf{E} \boldsymbol{\varepsilon}$$

where $\mathbf{E} = \mathbf{E}^T$ is a symmetric material properties matrix that relates the stress and strain components for each case. Actually, it is defined for the three-dimensional case and the

others are special forms of it. For an isotropic material they are:

1. Axial stress: $\mathbf{E} = E$, *Young's Modulus*
2. Plane Stress ($\sigma_z \equiv 0$)

$$\mathbf{E} = \frac{E}{1 - \nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1 - \nu)/2 \end{bmatrix}$$

ν , *Poisson's ratio*, $0 \leq \nu < 0.5$

3. Plane Strain ($\epsilon_z \equiv 0$)

$$\mathbf{E} = \frac{E}{(1 + \nu)(1 - 2\nu)} \begin{bmatrix} (1 - \nu) & 0 & 0 \\ \nu & (1 - \nu) & 0 \\ 0 & 0 & (1 - 2\nu)/2 \end{bmatrix}$$

4. Axisymmetric Solid

$$\mathbf{E} = \frac{E}{(1 + \nu)(1 - 2\nu)} \begin{bmatrix} (1 - \nu) & \nu & 0 & \nu \\ \nu & (1 - \nu) & 0 & \nu \\ 0 & 0 & (1 - 2\nu)/2 & 0 \\ \nu & \nu & 0 & (1 - \nu) \end{bmatrix}$$

5. General Solid

$$E_{11} = E_{22} = E_{33} = (1 - \nu)c$$

$$E_{12} = E_{13} = E_{23} = \nu c = E_{21} = E_{31} = E_{32}$$

$$E_{44} = E_{55} = E_{66} = G$$

$$\text{where } c = \frac{E}{(1 + \nu)(1 - 2\nu)} \quad \text{and} \quad G = \frac{E}{2(1 + \nu)}$$

Note that these data are independent of the element type and so their implementation for isotropic materials just depends on the input order of the material properties. Typical implementations for the plane stress, and axi-symmetric assumptions are shown in Figs. 11.2.1 and 2, respectively. Note that if one has an *incompressible material*, such as rubber where $\nu = 1/2$, then division by zero would cause difficulties for the plane strain and general solid problems, as currently formulated.

The strain energy matrix definition becomes

$$U(\mathbf{u}) = \frac{1}{2} \int_{\Omega} \boldsymbol{\epsilon}^T \mathbf{E} \boldsymbol{\epsilon} d\Omega$$

which when evaluated in an element domain becomes

$$U^e = \frac{1}{2} \int_{\Omega^e} (\mathbf{B}^e \boldsymbol{\delta}^e)^T \mathbf{E}^e (\mathbf{B}^e \boldsymbol{\delta}^e) d\Omega = \frac{1}{2} \boldsymbol{\delta}^{eT} \int_{\Omega^e} \mathbf{B}^{eT}(\mathbf{x}) \mathbf{E}^e \mathbf{B}^e(\mathbf{x}) d\Omega \boldsymbol{\delta}^e = \frac{1}{2} \boldsymbol{\delta}^{eT} \mathbf{S}^e \boldsymbol{\delta}^e$$

where the element stiffness matrix \mathbf{S}^e has the same symbolic matrix form as seen in heat

```

SUBROUTINE E_PLANE_STRESS (E) ! 1
! * * * * * ! 2
! PLANE_STRESS CONSTITUTIVE MATRIX DEFINITION ! 3
! STRESS & STRAIN COMPONENT ORDER: XX, YY, XY, SO N_R_B = 3 ! 4
! PROPERTY ORDER: 1-YOUNG'S MODULUS, 2-POISSON'S RATIO ! 5
! * * * * * ! 6
Use System_Constants ! for DP, N_R_B ! 7
Use Sys_Properties_Data ! for function GET_REAL_LP ! 8
IMPLICIT NONE ! 9
REAL(DP), INTENT(OUT) :: E (N_R_B, N_R_B) !10
REAL(DP) :: C_1, P_R, S_M, Y_M !11
!12
! E = CONSTITUTIVE MATRIX !13
! N_R_B = NUMBER OF ROWS IN B AND E MATRICES !14
! P_R = POISSON'S RATIO !15
! S_M = SHEAR MODULUS !16
! Y_M = YOUNG'S MODULUS OF ELASTICITY !17
!18
! RECOVER ELEMENT PROPERTIES !19
IF ( EL_REAL < 2 ) STOP 'el_real < 2 IN E_PLANE_STRESS' !20
Y_M = GET_REAL_LP (1) ; P_R = GET_REAL_LP (2) !21
S_M = 0.5d0 * Y_M / (1 + P_R) ; C_1 = Y_M / (1 - P_R * P_R) !22
!23
E (1, 1) = C_1 ; E (2, 1) = C_1 * P_R ; E (3, 1) = 0.d0 !24
E (1, 2) = C_1 * P_R ; E (2, 2) = C_1 ; E (3, 2) = 0.d0 !25
E (1, 3) = 0.d0 ; E (2, 3) = 0.d0 ; E (3, 3) = S_M !26
END SUBROUTINE E_PLANE_STRESS !27

```

Figure 10.2.1 Plane-stress isotropic constitutive law

```

SUBROUTINE E_AXISYMMETRIC_STRESS (E) ! 1
! * * * * * ! 2
! AXISYMMETRIC CONSTITUTIVE MATRIX DEFINITION, N_R_B = 4 ! 3
! STRESS & STRAIN COMPONENT ORDER: RR, ZZ, RZ, AND TT ! 4
! PROPERTY ORDER: 1-YOUNG'S MODULUS, 2-POISSON'S RATIO ! 5
! * * * * * ! 6
Use System_Constants ! for DP, N_R_B ! 7
Use Sys_Properties_Data ! for function GET_REAL_LP ! 8
IMPLICIT NONE ! 9
REAL(DP), INTENT(OUT) :: E (N_R_B, N_R_B) ! CONSTITUTIVE !10
REAL(DP) :: C_1, C_2, C_3, P_R, S_M, Y_M !11
!12
! N_R_B = NUMBER OF ROWS IN B AND E MATRICES !13
! P_R = POISSON'S RATIO !14
! S_M = SHEAR MODULUS !15
! Y_M = YOUNG'S MODULUS OF ELASTICITY !16
!17
IF (EL_REAL < 2) STOP 'el_real < 2 IN E_AXISYMMETRIC_STRESS' !18
Y_M = GET_REAL_LP (1) ; P_R = GET_REAL_LP (2) !19
S_M = 0.5d0 * Y_M / (1 + P_R) !20
C_1 = Y_M / (1 + P_R) / (1 - P_R - P_R) !21
C_2 = C_1 * P_R ; C_3 = C_1 * (1 - P_R) !22
!23
E (:, 3) = 0.d0 ; E (3, :) = 0.d0 !24
E (1, 1) = C_3 ; E (2, 1) = C_2 ; E (4, 1) = C_2 !25
E (1, 2) = C_2 ; E (2, 2) = C_3 ; E (4, 2) = C_2 !26
E (1, 4) = C_2 ; E (2, 4) = C_2 ; E (4, 4) = C_3 !27
E (3, 3) = S_M !28
END SUBROUTINE E_AXISYMMETRIC_STRESS !29

```

Figure 10.2.2 Axisymmetric stress isotropic constitutive law

transfer:

$$S^e = \int_{\Omega^e} \mathbf{B}^{eT}(x) \mathbf{E}^e \mathbf{B}^e(x) d\Omega.$$

If additional data on "initial strains", $\boldsymbol{\varepsilon}_0$, and/or "initial stresses", $\boldsymbol{\sigma}_0$, are given then the stress-strain law must be generalized to:

$$\boldsymbol{\sigma} = \mathbf{E}(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}_0) + \boldsymbol{\sigma}_0$$

where $\boldsymbol{\varepsilon}_0$ and $\boldsymbol{\sigma}_0$ are initial strains and stresses, respectively. If not zero they cause additional loading terms (source vectors) and require additional post-processing. The additional work terms due to the initial strains and stresses are

$$W_{\varepsilon_0} = \int_{\Omega} \boldsymbol{\sigma}^T \boldsymbol{\varepsilon}_0 d\Omega, \quad W_{\sigma_0} = \int_{\Omega} \boldsymbol{\varepsilon}^T \boldsymbol{\sigma}_0 d\Omega.$$

There are several common causes of initial strains, such as thermal and swelling effects. It is unusual to know an initial stress state and it is usually assumed to be zero. $\Pi(\boldsymbol{\Delta}) = \boldsymbol{\Delta}^T \mathbf{S} \boldsymbol{\Delta} / 2 - \boldsymbol{\Delta}^T \mathbf{C}$ so minimizing with respect to all the displacements, $\boldsymbol{\Delta}$, gives $\mathbf{S} \boldsymbol{\Delta} - \mathbf{C} = \mathbf{O}$ which are known as the algebraic Equations of Equilibrium.

11.3 Planar Models

The states of *plane stress* and *plane strain* are interesting and useful examples of stress analysis of a two-dimensional elastic solid (in the x - y plane). The assumption of plane stress implies that the component of all stresses normal to the plane are zero ($\sigma_z = \tau_{zx} = \tau_{zy} = 0$) whereas the plane strain assumption implies that the normal components of the strains are zero ($\varepsilon_z = \gamma_{zx} = \gamma_{zy} = 0$). The state of plane stress is commonly introduced in the first course of mechanics of materials. It was also the subject of some of the earliest finite element studies.

The assumption of plane stress means that the solid is very thin and that it is loaded only in the direction of its plane. At the other extreme, in the state of plane strain the dimension of the solid is very large in the z -direction. It is loaded perpendicular to the longitudinal (z) axis, and the loads do not vary in that direction. All cross-sections are assumed to be identical so any arbitrary x - y section can be used in the analysis. These two states are illustrated in Fig. 11.3.1. There are three common approaches to the variational formulation of the plane stress (or plane strain) problem: 1) Displacement formulation, 2) Stress formulation, and 3) Mixed formulation. We will select the common displacement method and utilize the total potential energy of the system. This can be proved to be equal to assuming a Galerkin weighted residual approach. In any event, note that it will be necessary to define all unknown quantities in terms of the displacements of the solid. Specifically, it will be necessary to relate the strains and stresses to the displacements as was illustrated in 1-D in Sec. 3.3.

The finite element form is based on the use of strain energy density, as discussed in Chap. 3. Since it is half the product of the stress and strain tensor components, we do not need, at this point, to consider either stresses or strains that are zero. This means that only three of six products will be used for a planar formulation.

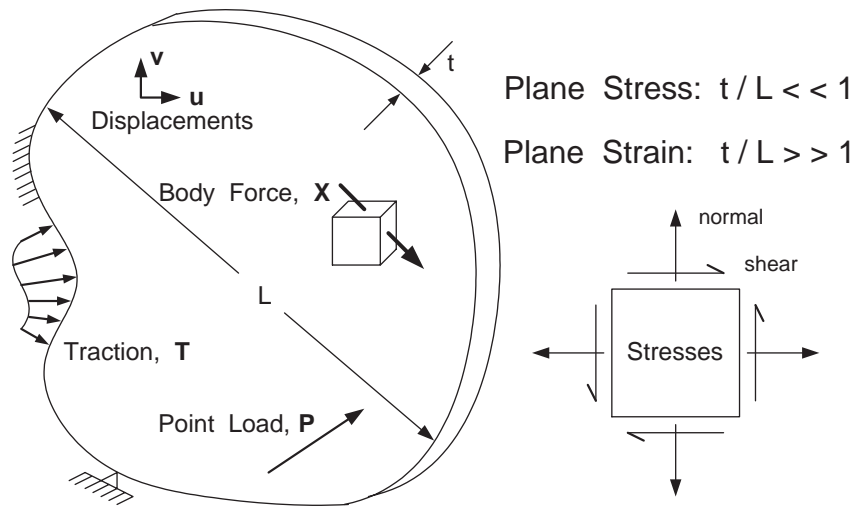


Figure 11.3.1 The states of plane stress and plane strain

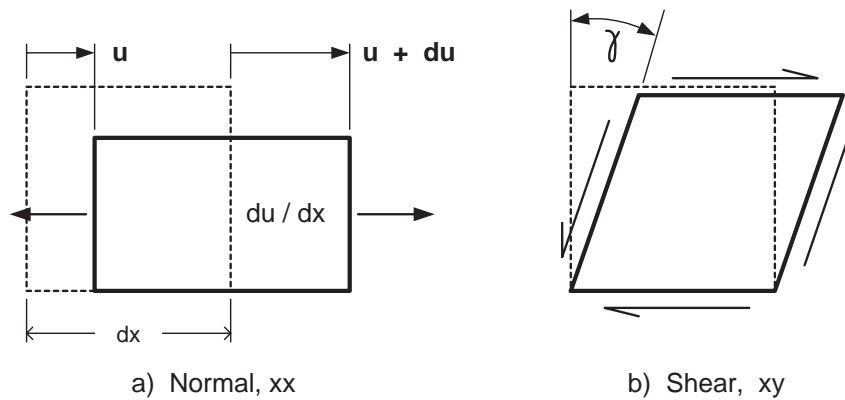


Figure 11.3.2 Engineering notation for planar strains

Our notation will follow that commonly used in mechanics of materials. The displacements components parallel to the x - and y -axes will be denoted by $u(x, y)$ and $v(x, y)$, respectively. The shear stress acting parallel to the x - and y -axes are σ_x and σ_y , respectively. The shear stress acting parallel to the y -axis on a plane normal to the x -axis is τ_{xy} , or simply τ . The corresponding components of strain are ϵ_x , ϵ_y , and γ_{xy} , or simply γ . Figure 11.3.2 summarizes the engineering (versus tensor) strain notations that we will employ in our matrix definitions.

11.3.1 Minimum Total Potential Energy

Plane stress analysis, like other elastic stress analysis problems, is governed by the principle of minimizing the total potential energy in the system. It is possible to write the generalized forms of the element matrices and boundary segment matrices defined in above. The symbolic forms are:

- (1) Stiffness matrix

$$\mathbf{S}^e = \int_{V^e} \mathbf{B}^{eT} \mathbf{D}^e \mathbf{B}^e(x, y) dv \quad (11.1)$$

\mathbf{B}^e = element strain-displacement matrix, \mathbf{D}^e = material constitutive matrix;

- (2) Body Force Matrix

$$\mathbf{C}_x^e = \int_{V^e} \mathbf{N}^e(x, y)^T \mathbf{X}^e(x, y) dv \quad (11.2)$$

\mathbf{N}^e = generalized interpolation matrix, \mathbf{X}^e = body force vector per unit volume;

- (3) Initial Strain Load Matrix

$$\mathbf{C}_o^e = \int_{V^e} \mathbf{B}^{eT}(x, y) \mathbf{D}^e \boldsymbol{\varepsilon}_o^e(x, y) dv \quad (11.3)$$

$\boldsymbol{\varepsilon}_o^e$ = initial strain matrix;

- (4) Surface Traction Load Matrix

$$\mathbf{C}_T^b = \int_{A^b} \mathbf{N}^b(x, y)^T \mathbf{T}^b(x, y) da \quad (11.4)$$

\mathbf{N}^b = boundary interpolation matrix, \mathbf{T}^b = traction force vector per unit area;

and where V^e is the element volume, A^b is a boundary segment surface area, dv is a differential volume, and da is a differential surface area. Now we will specialize these relations for plane stress (or strain). At any point the two displacement components will be denoted by $\mathbf{u}^T = [u \ v]$. Therefore, at each node there are two displacement components to be determined ($N_G_DOF = 2$). The total list of element degrees of freedom will be denoted by $\boldsymbol{\delta}^e$.

11.3.2 Displacement Interpolations

As before, it is necessary to define the spatial approximation for the displacement field. Consider the x -displacement, u , at some point in an element. The simplest approximation of how it varies in space is to assume a complete linear polynomial. In two-dimensions a complete linear polynomial contains three constants. Thus, we select a triangular element with three nodes (see Figs. 4.2.3, 8.2.1, and 8.2.4) and assume u is to be computed at each node. Then

$$u(x, y) = \mathbf{H}^e \mathbf{u}^e = H_1^e u_1^e + H_2^e u_2^e + H_3^e u_3^e. \quad (11.5)$$

The interpolation can either be done in global (x, y) coordinates or in a local system. If global coordinates are utilized then, from Eq. (8.13), the form of the typical interpolation

function is

$$H_i^e(x, y) = (a_i^e + b_i^e x + c_i^e y)/2A^e, \quad 1 \leq i \leq 3 \quad (11.6)$$

where A^e is the area of the element and a_i^e , b_i^e , and c_i^e denote constants for node i that depend on the element geometry. Clearly, we could utilize the same interpolations for the y -displacement:

$$v(x, y) = \mathbf{H}^e \mathbf{v}^e. \quad (11.7)$$

To define the element dof vector $\boldsymbol{\delta}^e$ we chose to order these six constants such that

$$\boldsymbol{\delta}^{eT} = [u_1 \quad v_1 \quad u_2 \quad v_2 \quad u_3 \quad v_3]^e. \quad (11.8)$$

To refer to both displacement components at a point we employ a generalized element interpolation. Then

$$\mathbf{u}(x, y) = \begin{Bmatrix} u(x, y) \\ v(x, y) \end{Bmatrix} = \begin{bmatrix} H_1 & 0 & H_2 & 0 & H_3 & 0 \\ 0 & H_1 & 0 & H_2 & 0 & H_3 \end{bmatrix}^e \boldsymbol{\delta}^e = \mathbf{N}^e \boldsymbol{\delta}^e. \quad (11.9)$$

Of course, more advanced polynomials could be selected to define the \mathbf{H} or \mathbf{N} matrices. Note that the combined interpolation array, \mathbf{N} , could be partitioned into typical partitions from a node. For example, from node j :

$$\mathbf{N}_j = \begin{bmatrix} H_j & 0 \\ 0 & H_j \end{bmatrix}.$$

11.3.3 Strain-Displacement Relations

From mechanics of materials we can define the strains in terms of the displacement. Order the three strain components so as to define $\boldsymbol{\varepsilon}^T = [\varepsilon_x \quad \varepsilon_y \quad \gamma]$. These terms are defined as:

$$\varepsilon_x = \frac{\partial u}{\partial x}, \quad \varepsilon_y = \frac{\partial v}{\partial y}, \quad \gamma = \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)$$

if the common engineering form is selected for the shear strain, γ . Two of these terms are illustrated in Fig. 11.3.2. From Eqs. (11.5) and (11.7) we note

$$\varepsilon_x = \frac{\partial \mathbf{H}^e}{\partial x} \mathbf{u}^e, \quad \varepsilon_y = \frac{\partial \mathbf{H}^e}{\partial y} \mathbf{v}^e, \quad \gamma = \frac{\partial \mathbf{H}^e}{\partial y} \mathbf{u}^e + \frac{\partial \mathbf{H}^e}{\partial x} \mathbf{v}^e. \quad (11.10)$$

These can be combined into a single matrix identity to define

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma \end{Bmatrix}^e = \begin{bmatrix} H_{1,x} & 0 & H_{2,x} & 0 & \cdots & H_{n,x} & 0 \\ 0 & H_{1,y} & 0 & H_{2,y} & \cdots & 0 & H_{n,y} \\ H_{1,y} & H_{1,x} & H_{2,y} & H_{2,x} & \cdots & H_{n,y} & H_{n,x} \end{bmatrix}^e \boldsymbol{\delta}^e \quad (11.11)$$

or symbolically, $\boldsymbol{\varepsilon}^e = \mathbf{B}^e(x, y)\boldsymbol{\delta}^e$, where the shorthand notation $H_{j,x} = \partial H_j / \partial x$, etc. has been employed, and where we have assumed the element has n nodes. Thus, the \mathbf{B}^e matrix size depends on the type of element being utilized. This defines the *element strain-displacement operator* \mathbf{B}^e that would be used in Eqs. (11.1) and (11.3). Note that \mathbf{B} could also be partitioned into 3×2 sub-partitions from each node on the element, as

$$\boldsymbol{\sigma} = \mathbf{E}(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}_o) . \quad (11.14)$$

Here we have allowed for the presence of initial strains, $\boldsymbol{\varepsilon}_o$, that are not usually included in mechanics of materials. For the plane stress assumption note that \mathbf{E} (given above) is a symmetric matrix. This is almost always true. This observation shows that in general the element stiffness matrix, Eq. (11.1), will also be symmetric.

The most common type of initial strain, $\boldsymbol{\varepsilon}_o$, is that due to temperature changes. For an isotropic material these *thermal strains* are

$$\boldsymbol{\varepsilon}_o^T = \alpha \Delta\theta [1 \quad 1 \quad 0] \quad (11.17)$$

where α is the coefficient of thermal expansion and $\Delta\theta = (\theta - \theta_o)$ is the temperature rise from a stress free temperature of θ_o . Usually the $\Delta\theta$ is supplied as data, or θ_o is given as data along with the nodal temperatures computed from the procedures in the previous chapter. Notice that thermal strains in isotropic materials do not include thermal shear strains. If the above temperature changes were present then the additional loading effects could be included via Eq. (11.3). An example of thermal stress loadings of T6 elements is given at the end of Chap. 20.

At this point, we do not know the nodal displacements, $\boldsymbol{\delta}^e$, of the element. Once we do know them, we will wish to use the above arrays to get post-processing results for the stresses and, perhaps, for *failure criteria*. Therefore, for each element we usually store the arrays \mathbf{B}^e , \mathbf{E}^e , and $\boldsymbol{\varepsilon}_o^e$ so that we can execute the products in Eqs. (11.11) and (11.14) after the displacements are known.

11.4 Matrices for the Constant Strain Triangle

Beginning with the simple linear displacement assumption of Eqs. (11.5) to (11.7) we note that for a typical CST interpolation function $\partial H_i^e / \partial x = b_i^e / 2A^e$, and $\partial H_i^e / \partial y = c_i^e / 2A^e$. Therefore, from Eqs. (11.11) and (11.12), the strain components in the triangular element are constant. Specifically,

$$\mathbf{B}^e = \frac{1}{2A^e} \begin{bmatrix} b_1 & 0 & b_2 & 0 & b_3 & 0 \\ 0 & c_1 & 0 & c_2 & 0 & c_3 \\ c_1 & b_1 & c_2 & b_2 & c_3 & b_3 \end{bmatrix}^e. \quad (11.18)$$

For this reason this element is commonly known as the *constant strain triangle*, CST. If we also let the material properties, E and ν , be constant in a typical element then the stiffness matrix in Eq. (11.1) simplifies to

$$\mathbf{S}^e = \mathbf{B}^{eT} \mathbf{E}^e \mathbf{B}^e V^e \quad (11.19)$$

where the element volume is

$$V^e = \int_{V^e} dv = \int_{A^e} t^e(x, y) dx dy \quad (11.20)$$

where t^e is the element thickness. Usually the thickness of a typical element is constant so that $V^e = t^e A^e$. Of course, it would be possible to define the thickness at each node and to utilize the interpolation functions to approximate $t^e(x, y)$. Similarly if the temperature change in the element is also constant within the element then Eq. (11.3) defines the thermal load matrix

$$\mathbf{C}_o^e = \mathbf{B}^{eT} \mathbf{E}^e \boldsymbol{\varepsilon}_o^e \mathbf{t}^e A^e. \quad (11.21)$$

It would be possible to be more detailed and input the temperature at each node and integrate its change over the element.

It is common for plane stress problems to include body force loads due to gravity, centrifugal acceleration, etc. For simplicity, assume that the body force vector \mathbf{X}^e , and the thickness, t^e , are constant. Then the body force vector in Eq. (11.2) simplifies to

$$\mathbf{C}_X^e = \mathbf{t}^e \int_{A^e} \mathbf{N}^{eT}(x, y) dx dy \mathbf{X}^e. \quad (11.22)$$

From Eq. (11.9) it is noted that the non-zero terms in the integral typically involve scalar terms such as

$$I_i^e = \int_{A^e} H_i^e(x, y) da = \frac{1}{2A^e} \int_{A^e} (a_i^e + b_i^e x + c_i^e y) da. \quad (11.23)$$

These three terms can almost be integrated by inspection. The element geometric constants can be taken outside parts of the integrals. Then from the concepts of the first moment (centroid) of an area

$$a_i^e \int da = a_i^e A^e, \quad \int b_i^e x da = b_i^e \bar{x}^e A^e, \quad \int c_i^e y da = c_i^e \bar{y}^e A^e \quad (11.24)$$

where \bar{x} and \bar{y} denote the *centroid* coordinates of the triangle, $\bar{x}^e = (x_1 + x_2 + x_3)^e / 3$, and $\bar{y}^e = (y_1 + y_2 + y_3)^e / 3$. In view of Eq. (11.24), the integral in Eq. (11.23) becomes $I_i^e = A^e (a_i^e + b_i^e \bar{x}^e + c_i^e \bar{y}^e) / 2A^e$. Reducing the algebra to its simplest form, using Table 9.1, yields

$$I_i^e = A^e / 3, \quad 1 \leq i \leq 3. \quad (11.25)$$

Therefore, for the CST the expanded form of Eq. (11.22) is

$$\mathbf{C}_X^e = \frac{t^e A^e}{3} \begin{bmatrix} 1 & 0 \\ 0 & 1 \\ 1 & 0 \\ 0 & 1 \\ 1 & 0 \\ 0 & 1 \end{bmatrix} \left\{ \begin{matrix} X_x \\ X_y \end{matrix} \right\}^e = \frac{t^e A^e}{3} \left\{ \begin{matrix} X_x \\ X_y \\ X_x \\ X_y \\ X_x \\ X_y \end{matrix} \right\}^e \quad (11.26)$$

where X_x and X_y denote the components of the body force vector. To assign a physical meaning to this result note that $t^e A^e X_x^e$ is the resultant force in the x -direction. Therefore, the above calculation has replaced the distributed load with a statically equivalent set of three nodal loads. Each of these loads is a third of the resultant load. These *consistent loads* are illustrated in Figs. 11.4.1.

A body force vector, \mathbf{X} , can arise from several important sources. An example is one due to acceleration (and gravity) loads. We have been treating only the case of equilibrium. When the acceleration is unknown, we have a dynamic system. Then, instead of using Newton's second law,

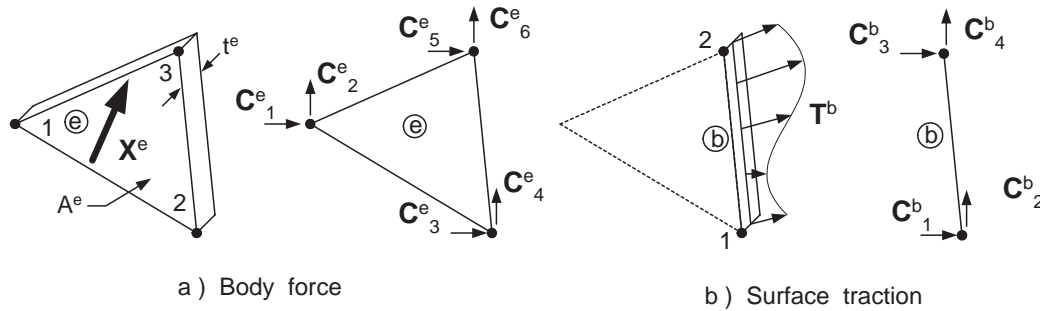


Figure 11.4.1 Element loads and consistent resultants

$$\sum \mathbf{F} = m \mathbf{a}$$

where $\mathbf{a}(t)$ is the acceleration vector, we invoke the D'Alembert's principle and rewrite this as a pseudo-equilibrium problem

$$\sum \mathbf{F} - m \mathbf{a} = \mathbf{0}, \quad \text{or} \quad \sum \mathbf{F} + \mathbf{F}_I = \mathbf{0}$$

where we have introduced an inertial body force vector due to the acceleration, that is, we use $\mathbf{X} = -\rho \mathbf{a}$ for the equilibrium integral form. Since the acceleration is the second time derivative of the displacement vector, we can write

$$\mathbf{a}(t) = \mathbf{N}^e(x, y) \ddot{\boldsymbol{\delta}}^e$$

in a typical element. The typical element inertial contribution is, therefore,

$$-\mathbf{m}^e \mathbf{a}^e = - \int_{\Omega^e} \mathbf{N}^{eT} \zeta \mathbf{N}^e d\Omega \ddot{\boldsymbol{\delta}}^e$$

where \mathbf{m}^e is the element mass matrix. Since the acceleration vector is unknown, we move it to the LHS of the system equations:

$$\mathbf{M} \ddot{\boldsymbol{\delta}} + \mathbf{K} \boldsymbol{\delta} = \mathbf{F}.$$

Here, \mathbf{M} is the assembled system mass matrix, and the above are the *structural dynamic equations*. This class of problem will be considered in later. If we had free ($\mathbf{F} = 0$) *simple harmonic motion*, so that $\boldsymbol{\delta}(t) = \boldsymbol{\delta}_j \text{Sin}(\omega_j t)$, then we get the alternate class known as the *eigen-problem*, $[\mathbf{K} - \omega_j^2 \mathbf{M}] \boldsymbol{\delta}_j = \mathbf{0}$ where ω_j is the eigenvalue, or natural frequency, and $\boldsymbol{\delta}_j$ is the mode shape, or eigenvector. The computational approaches to eigen-problems are covered in detail in the texts by Bathe [1], and by others.

Another type of body force that is usually difficult to visualize is that due to electromagnetic effects. In the past, they were usually small enough to be ignored. However, with the advances in superconducting materials, very high electrical current densities are possible, and they can lead to significant mechanical loads. Similar loads develop in medical scan devices, and in fusion energy reactors which are currently experimental. Recall from basic physics that the mechanical force, \mathbf{F} , due to a current density vector, \mathbf{J} , in a field with a magnetic flux density vector of \mathbf{B} is the vector cross

product $\mathbf{F} = \mathbf{J} \times \mathbf{B}$. For a thin wire conductor, \mathbf{J} is easy to visualize since it is tangent to the conductor. However, the \mathbf{B} field, like the earth's gravity field, is difficult to visualize. This could lead to important forces on the system which might be overlooked.

The final load to be considered is that acting on a typical boundary segment. As indicated in Fig. 11.4.1, such a segment is one side of an element being loaded with a traction. In plane stress problems these pressures or distributed shears act on the edge of the solid. In other words, they are distributed over a length L^b that has a known thickness, t . Those two quantities define the surface area, A^b , on which the tractions in Eq. (11.4) are applied. Similarly, the differential surface area is $da = t dL$. We observe that such a segment would have two nodes. We can refer to them as local boundary nodes 1 and 2. Of course, they are a subset of the three element nodes and also a subset of the system nodes. Before Eq. (11.4) can be integrated to define the consistent loads on the two boundary nodes it is necessary to form the boundary interpolation, \mathbf{N}^b . That function defines the displacements, u and v , at all points on the boundary segment curve. By analogy with Eq. (11.8) we can denote the dof of the boundary segment as

$$\mathbf{a}^{bT} = [u_1 \quad v_1 \quad u_2 \quad v_2]^b.$$

Then the requirement that $\mathbf{u} = \mathbf{N}^b \boldsymbol{\delta}^b$, for all points on L^b , defines the required \mathbf{N}^b . There are actually two ways that its algebraic form can be derived:

- 1) Develop a consistent (linear) interpolation on the line between the nodal dof.
- 2) Degenerate the element function \mathbf{N}^e in Eq. (11.9) by restricting the x and y coordinates to points on the boundary segment.

If the second option is selected then all the H_i^e vanish except for the two associated with the two boundary segment nodes. Those two H^e are simplified by the restriction and thus define the two H_i^b functions. While the result of this type of procedure may be obvious, the algebra is tedious in global coordinates. (For example, let $y^b = mx^b + n$ in Eq. (11.6).) It is much easier to get the desired results if local coordinates are used. (For example, set $s^b = 0$ in Eq. (9.6).) The net result is that one obtains a one-dimensional linear interpolation set for \mathbf{H}^b that is analogous to Eq. (4.11). If we assume constant thickness, t^b , and constant tractions, \mathbf{T}^b , then Eq. (11.4) becomes

$$\mathbf{C}_T^b = t^b \int_{L^b} \mathbf{N}^{bT} dL \mathbf{T}^b.$$

Repeating the procedure used for Eq. (5.5) a typical non-zero contribution to the integral is

$$I_i^b = \int_{L^b} H_i^b dL = L^b/2, \quad 1 \leq i \leq 2$$

and the final result for the four force components is

$$\mathbf{C}_T^{bT} = \frac{t^b L^b}{2} [T_x \quad T_y \quad T_x \quad T_y]^b \quad (11.27)$$

where T_x and T_y are the two components of T . Physically, this states that half of the resultant x -force, $t^b L^b T_x^b$ is lumped at each of the two nodes. The resultant y -force is lumped in the same way as illustrated in Fig. 11.4.1.

In the case of stress analysis there are times when it is desirable to rearrange the constitutive matrix, \mathbf{E} , into two parts. One part, \mathbf{E}_n , is due to normal strain effects, and the other, \mathbf{E}_s , is related to the shear strains. Therefore, in general it is possible to write Eq. (11.5) as

$$\mathbf{E} = \mathbf{E}_n + \mathbf{E}_s . \quad (11.28)$$

In this case such a procedure simply makes it easier to write the CST stiffness matrix in closed form. Noting that substituting Eq. (11.28) into Eq. (11.1) allows the stiffness to be separated into parts $\mathbf{S}^e = \mathbf{S}_n^e + \mathbf{S}_s^e$, where

$$\mathbf{S}_n^e = \frac{EV}{4A^2(1-\nu^2)} \begin{bmatrix} b_1^2 & & & & & \text{sym} \\ \nu b_1 c_1 & c_1^2 & & & & \\ b_1 b_2 & \nu c_1 b_2 & b_2^2 & & & \\ \nu b_1 c_2 & c_1 c_2 & \nu b_2 c_2 & c_2^2 & & \\ b_1 b_3 & \nu c_1 b_3 & b_2 b_3 & \nu c_2 b_3 & b_3^2 & \\ \nu b_1 c_3 & c_1 c_3 & \nu b_2 c_3 & c_2 c_3 & \nu b_3 c_3 & c_3^2 \end{bmatrix}$$

$$\mathbf{S}_s^e = \frac{EV}{8A^2(1-\nu)} \begin{bmatrix} c_1^2 & & & & & \text{sym} \\ c_1 b_1 & b_1^2 & & & & \\ c_1 c_2 & b_1 c_2 & c_2^2 & & & \\ c_1 b_2 & b_1 b_2 & c_2 b_2 & b_2^2 & & \\ c_1 c_3 & b_1 c_3 & c_2 c_3 & b_2 c_3 & c_3^2 & \\ c_1 b_3 & b_1 b_3 & c_2 b_3 & b_2 b_3 & c_3 b_3 & b_3^2 \end{bmatrix}$$

and where V is the volume of the element. For constant thickness $V = At$.

The strain-displacement matrix \mathbf{B}^e can always be partitioned into sub-matrices associated with each node. Thus, the square stiffness matrix \mathbf{S} can also be partitioned into square sub-matrices, since it is the product of $\mathbf{B}^T \mathbf{E} \mathbf{B}$. For local nodes j and k , they interact to give a contribution defined by:

$$\mathbf{S}_{jk} = \int_{\Omega} \mathbf{B}_j^T \mathbf{E} \mathbf{B}_k d\Omega .$$

If we choose to split \mathbf{E} into two distinct parts, say $\mathbf{E} = \mathbf{E}_n + \mathbf{E}_s$, then we likewise have two contributions to the partitions of \mathbf{S} , namely

$$\mathbf{S}_{jk}^n = \int_{\Omega} \mathbf{B}_j^T \mathbf{E}_n \mathbf{B}_k d\Omega , \quad \mathbf{S}_{jk}^s = \int_{\Omega} \mathbf{B}_j^T \mathbf{E}_s \mathbf{B}_k d\Omega .$$

Sometimes we may use different numerical integration rules on these two parts. For the constant strain triangle, CST, we have the nodal partitions of \mathbf{B}^{e_j} :

$$\mathbf{B}_j^e = \frac{1}{2A^e} \begin{bmatrix} b_j & 0 \\ 0 & c_j \\ c_j & b_j \end{bmatrix}^e = \begin{bmatrix} d_{xj} & 0 \\ 0 & d_{yj} \\ d_{yj} & d_{xj} \end{bmatrix}$$

which involve the geometric constants defined earlier. Once we know the gradients of the scalar interpolation functions, \mathbf{H} we can compute \mathbf{B}^e from Eq. 11.11 as shown earlier in Fig. 11.3.3, for any planar element in our element library

For a constant isotropic \mathbf{E} , the integral gives the partitions

$$\mathbf{S}_{jk}^s = \frac{E_{33}}{4A} \begin{bmatrix} c_j c_k & b_j c_k \\ c_j b_k & b_j b_k \end{bmatrix}, \quad \mathbf{S}_{jk}^n = \frac{1}{4A} \begin{bmatrix} E_{11} b_j b_k & E_{12} c_j b_k \\ E_{12} b_j c_k & E_{11} c_j d_k \end{bmatrix}$$

where E_{33} brings in the shear effects, and E_{11} , E_{12} couple the normal stress effects. If we allow j and k to range over the values 1, 2, 3, we would get the full 6×6 stiffness matrix

$$\mathbf{S} = \begin{bmatrix} \mathbf{S}_{11} & \mathbf{S}_{12} & \mathbf{S}_{13} \\ \mathbf{S}_{21} & \mathbf{S}_{22} & \mathbf{S}_{23} \\ \mathbf{S}_{31} & \mathbf{S}_{32} & \mathbf{S}_{33} \end{bmatrix}.$$

Since \mathbf{E} is symmetric, it should be clear that $\mathbf{S}_{jk} = \mathbf{S}_{kj}^T$. A similar split can be made utilizing the constitutive law in terms of the Lamé constants,

$$\lambda = K - \frac{2G}{3} = \frac{E\nu}{(1+\nu)(1-2\nu)}, \quad \mu = G = \frac{E}{2(1+\nu)},$$

where K and G are the bulk modulus and the shear modulus, respectively. Then the plane strain \mathbf{E} matrix can be split as

$$\mathbf{E} = \lambda \begin{bmatrix} 1 & 1 & 0 \\ 1 & 1 & 0 \\ 0 & 0 & 0 \end{bmatrix} + \mu \begin{bmatrix} 2 & 0 & 0 \\ 0 & 2 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

$$\mathbf{E} = K \begin{bmatrix} 1 & 1 & 0 \\ 1 & 1 & 0 \\ 0 & 0 & 0 \end{bmatrix} + \frac{G}{3} \begin{bmatrix} 4 & -2 & 0 \\ -2 & 4 & 0 \\ 0 & 0 & 3 \end{bmatrix}.$$

Likewise, for the full three-dimensional case with six strains and six stresses, we have a similar form

$$\mathbf{E} = \lambda \begin{bmatrix} 1 & 1 & 1 & 0 & 0 & 0 \\ 1 & 1 & 1 & 0 & 0 & 0 \\ 1 & 1 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} + \mu \begin{bmatrix} 2 & 0 & 0 & 0 & 0 & 0 \\ 0 & 2 & 0 & 0 & 0 & 0 \\ 0 & 0 & 2 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}.$$

This type of split means that we have also split the strain energy into two corresponding parts: the distortional strain energy and the volumetric strain energy. We define an *incompressible material* as one that has no change in volume as it is deformed. For such a material $\nu = \frac{1}{2}$. For a nearly incompressible material, we note that as $\nu \rightarrow \frac{1}{2}$, we see that $\lambda \rightarrow \infty$ and $K \rightarrow \infty$. Since many rubber materials are nearly incompressible, we can expect to encounter this difficulty in several practical problems. Since incompressibility means no volume change, it also means there is no volumetric strain. In two-dimensions, this means that we have the *incompressibility constraint* :

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \equiv 0.$$

In such a case, we must either use an alternate variational form that involves the displacements and the mean stress (pressure), or we must undertake numerical corrections to prevent the solution from *locking*.

As an example of the use of the CST, consider the thin structure shown in Fig. 11.4.2. Its elastic modulus is $15 \times 10^9 \text{ N/m}^2$, Poisson's ratio is 0.25, and the yield stress is $10.5 \times 10^6 \text{ N/m}^2$. The uniform thickness of the material is $5 \times 10^{-3} \text{ m}$. The corresponding input data file is given in Fig. 11.4.3. From Eq. (9.14) the element geometric constants are

$e = 1$			$e = 2$		
i	b_i	c_i	i	b_i	c_i
1	-2	-2	1	+2	+2
2	+2	0	2	-2	0
3	0	+2	3	0	-2

For the given data the constants multiplying the \mathbf{S}_n and \mathbf{S}_s matrices are 1×10^7 and $6 \times 10^7 \text{ N/m}$, respectively. For the first element the two contributions to the element stiffness matrix are

$$\mathbf{S}_n^e = \frac{1 \times 10^7}{2} \begin{bmatrix} +8 & & & & & & \text{Sym.} \\ +2 & +8 & & & & & \\ -8 & -2 & +8 & & & & \\ 0 & 0 & 0 & 0 & & & \\ 0 & 0 & 0 & 0 & 0 & & \\ -2 & -8 & +2 & 0 & 0 & +8 & \end{bmatrix}$$

$$\mathbf{S}_s^e = \frac{3 \times 10^7}{2} \begin{bmatrix} +1 & & & & & & \text{Sym.} \\ +1 & +1 & & & & & \\ 0 & 0 & 0 & & & & \\ -1 & -1 & 0 & +1 & & & \\ -1 & -1 & 0 & +1 & +1 & & \\ 0 & 0 & 0 & 0 & 0 & 0 & \end{bmatrix}$$

Thus, the first element stiffness is:

$$\mathbf{S}^e = 5 \times 10^6 \begin{bmatrix} +11 & & & & & & \text{Sym.} \\ +5 & +11 & & & & & \\ -8 & -2 & +8 & & & & \\ -3 & -3 & 0 & +3 & & & \\ -3 & -3 & 0 & +3 & +3 & & \\ -2 & -8 & +2 & 0 & 0 & +8 & \end{bmatrix}$$

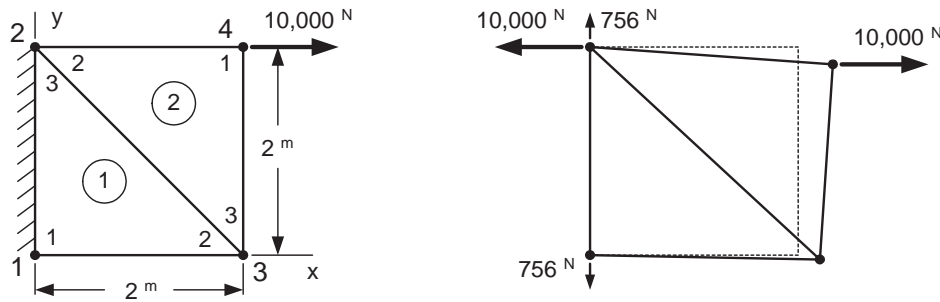


Figure 11.4.2 An example plane stress structure

```

title "Two element plane stress example"           ! 1
area_thick 5e-3 ! Thickness of all planar elements ! 2
nodes      4 ! Number of nodes in the mesh        ! 3
elems      2 ! Number of elements in the system   ! 4
dof        2 ! Number of unknowns per node       ! 5
el_nodes   3 ! Maximum number of nodes per element ! 6
space      2 ! Solution space dimension           ! 7
b_rows     3 ! Number of rows in the B matrix     ! 8
shape      2 ! Element shape, 1=line, 2=tri, 3=quad ! 9
gauss      1 ! number of quadrature points        !10
el_real    3 ! Number of real properties per element !11
el_homo    ! Element properties are homogeneous  !12
pt_list    ! List the answers at each node point !13
loads      ! An initial source vector is input   !14
post_1     ! Require post-processing              !15
example    201 ! Source library example number    !16
remarks    5 ! Number of user remarks            !17
quit ! keyword input, remarks follow             !18
      2-----4 ----> P = 1e4 N                    !19
      Fixed : \ (2): width = height = 2 m,         !20
      edge  : \ : E = 15e9 N/m^2, nu = 0.25,       !21
      : (1)\ : thickness = 5e-3 m                  !22
      1-----3                                     !23
1      11  0.0  0.0 ! node, U-V BC flag, x, y      !24
2      11  0.0  2.0 ! node, U-V BC flag, x, y      !25
3      00  2.0  0.0 ! node, U-V BC flag, x, y      !26
4      00  2.0  2.0 ! node, U-V BC flag, x, y      !27
1  1  3  2 ! element, 3 nodes                       !28
2  4  2  3 ! element, 3 nodes                       !29
      1  1  0.0 ! node, direction, BC value (U)    !30
      1  2  0.0 ! node, direction, BC value (V)    !31
      2  1  0.0 ! node, direction, BC value (U)    !32
      2  2  0.0 ! node, direction, BC value (V)    !33
1 15e9 0.25 10.5e6 ! elem, E, nu, yield stress     !34
4 1 10000. ! node, direction, force                !35
4 2 0. ! terminate with last force in the system !36

```

Figure 11.4.3 Plane stress data file

and its global and local degree of freedom numbers are the same. The second stiffness matrix happens to be the same due to its 180° rotation in space. Of course, its global dof numbers are different. That list is: 7, 8, 3, 4, 5, and 6. Since there are no body forces or surface tractions these matrices can be assembled to relate the system stiffness

to the applied point load, P , and the support reactions. Applying the direct assembly procedure gives

$$5 \times 10^6 \begin{bmatrix} +11 & & & & & & & & & & \\ & +5 & +11 & & & & & & & & \\ & -3 & -3 & +11 & & & & & & & \\ & -2 & -8 & 0 & +11 & & & & & & \\ & -8 & -2 & 0 & +5 & +11 & & & & & \\ & -3 & -3 & +5 & 0 & 0 & +11 & & & & \\ & 0 & 0 & -8 & -3 & -3 & -2 & +11 & & & \\ & 0 & 0 & -2 & -3 & -3 & -8 & +5 & +11 & & \end{bmatrix} \Delta = \begin{bmatrix} R_1 \\ R_2 \\ R_3 \\ R_4 \\ 0 \\ 0 \\ 10^4 \\ 0 \end{bmatrix}.$$

Applying the conditions of zero displacement at nodes 1 and 2 reduces this set to

$$5 \times 10^6 \begin{bmatrix} 11 & & & & & & & & & & \\ & 0 & 11 & & & & & & & & \\ & -3 & -2 & 11 & & & & & & & \\ & -3 & -8 & 5 & 11 & & & & & & \end{bmatrix} \begin{bmatrix} u_3 \\ v_3 \\ u_4 \\ v_4 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 10^4 \\ 0 \end{bmatrix}.$$

Inverting the matrix and solving gives the required displacement vector, transposed: $10^5 \times u_r = [2.52 \quad -6.72 \quad 24.65 \quad -15.41] m$. Substituting to find the reactions yields $\mathbf{R}_g^T = [-0.002 \quad -756.3 \quad -10,000 \quad -756.3] N$. The deformed shape and resulting reactions are also shown in Fig. 11.4.2. One should always check the equilibrium of the reactions and applied loads. Checking $\sum \mathbf{F}_x = 0$, $\sum \mathbf{M} = 0$ does show minor errors in about the sixth significant figure. Thus, the results are reasonable.

At this point we can recover the displacements for each element, and then compute the strains and stress. The element dof vectors (in meters) are, respectively

$$\begin{aligned} \boldsymbol{\delta}^{e^T} &= [0 \quad 0 \quad 2.521 \quad -6.723 \quad 0 \quad 0] \times 10^{-5} \\ \boldsymbol{\delta}^{e^T} &= [24.650 \quad -15.406 \quad 0 \quad 0 \quad 2.521 \quad -6.723] \times 10^{-5} \end{aligned}$$

and the strain-displacement matrices, from Eq. (11.18) are

$$\mathbf{B}^e = \frac{1}{4} \begin{bmatrix} -2 & 0 & 2 & 0 & 0 & 0 \\ 0 & -2 & 0 & 0 & 0 & 2 \\ -2 & -2 & 0 & 2 & 2 & 2 \end{bmatrix}$$

for $e = 1$, while for $e = 2$

$$\mathbf{B}^e = \frac{1}{4} \begin{bmatrix} 2 & 0 & -2 & 0 & 0 & 0 \\ 0 & 2 & 0 & 0 & 0 & -2 \\ 2 & 2 & 0 & -2 & -2 & 0 \end{bmatrix}.$$

Recovering the element strains, $\boldsymbol{\varepsilon}^e = \mathbf{B}^e \boldsymbol{\delta}^e$ in meters/meter gives

```

TITLE: "Two element plane stress example"           ! 1
                                                    ! 2
*** ELEMENT PROPERTIES ***                       ! 3
ELEMENT, 3 PROPERTY & REAL_VALUE PAIRS          ! 4
  1 1 1.50000E+10 2 2.50000E-01 3 1.05000E+07   ! 5
                                                    ! 6
NOTE: 2-D DOMAIN THICKNESS SET TO 5.000000000000E-03 ! 7
                                                    ! 8
*** INITIAL FORCING VECTOR DATA ***             ! 9
NODE   PARAMETER   VALUE   EQUATION             !10
  4     1           1.00000E+04   7           !11
  4     2           0.00000E+00   8           !12
                                                    !13
*** REACTION RECOVERY ***                       !14
NODE, PARAMETER, REACTION, EQUATION             !15
  1, DOF_1, 6.8212E-13 1                       !16
  1, DOF_2, 7.5630E+02 2                       !17
  2, DOF_1, -1.0000E+04 3                      !18
  2, DOF_2, -7.5630E+02 4                      !19
                                                    !20
*** OUTPUT OF RESULTS IN NODAL ORDER ***        !21
NODE, X-Coord, Y-Coord, DOF_1, DOF_2,         !22
  1 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 !23
  2 0.0000E+00 2.0000E+00 0.0000E+00 0.0000E+00 !24
  3 2.0000E+00 0.0000E+00 2.5210E-05 -6.7227E-05 !25
  4 2.0000E+00 2.0000E+00 2.4650E-04 -1.5406E-04 !26
                                                    !27
*** STRESSES AT INTEGRATION POINTS ***         !28
COORDINATES STRESSES                          !29
POINT X Y XX YY                               !30
POINT XY EFFECTIVE                             !31
      ELEMENT NUMBER 1                         !32
  1 6.667E-01 6.667E-01 2.01681E+05 5.04202E+04 !33
  1 -2.01681E+05 3.93794E+05                 !34
      ELEMENT NUMBER 2                         !35
  1 1.333E+00 1.333E+00 1.79832E+06 -2.01681E+05 !36
  1 2.01681E+05 1.93890E+06                 !37

```

Figure 10.4.4 Selected CST results for the two element model

$$e = 1, \quad \boldsymbol{\epsilon}^{e^T} = 10^{-5} [1.261 \quad 0.000 \quad -3.361]$$

$$e = 1, \quad \boldsymbol{\epsilon}^{e^T} = 10^{-5} [12.325 \quad -4.342 \quad 3.361] .$$

Utilizing the constitute law in Eq. (11.15), with no initial strains, $\boldsymbol{\epsilon}_o = \mathbf{0}$, gives

$$\mathbf{E}^e = \frac{15 \times 10^9}{(15/16)} \begin{bmatrix} 1 & 1/4 & 0 \\ 1/4 & 1 & 0 \\ 0 & 0 & 3/8 \end{bmatrix} = 2 \times 10^9 \begin{bmatrix} 8 & 2 & 0 \\ 2 & 8 & 0 \\ 0 & 0 & 3 \end{bmatrix},$$

and the element stresses, in Newtons/meter², are

$$e = 1, \quad \boldsymbol{\sigma}^{e^T} = 10^4 [20.17 \quad 5.04 \quad -20.17]$$

$$e = 2, \quad \boldsymbol{\sigma}^{e^T} = 10^4 [179.83 \quad -20.17 \quad 20.17] .$$

```

!      ***  ELEM_SQ_MATRIX PROBLEM DEPENDENT STATEMENTS ***      ! 1
!      PLANE_STRESS ANALYSIS, NON-ISOPARAMETRIC                  ! 2
! STRESS AND STRAIN COMPONENT ORDER: XX, YY, XY, SO N_R_B = 3    ! 3
!                                                                    ! 4
INTEGER   :: IP          ! loops                                ! 5
REAL(DP)  :: DET, DET_WT, THICK ! volume                      ! 6
! PROPERTIES: 1-YOUNG'S MODULUS, 2-POISSON'S RATIO, AND          ! 7
!              3-YIELD STRESS, IF PRESENT                        ! 8
!                                                                    ! 9
CALL STORE_FLUX_POINT_COUNT ! Save LT_QP                        !10
!                                                                    !11
THICK = 1                ! DEFINE CONSTANT PROPERTIES          !12
IF ( AREA_THICK /= 1.d0 ) THICK = AREA_THICK                    !13
!                                                                    !14
! FORM THE CONSTITUTIVE MATRIX (OR GET_APPLICATION_E_MATRIX )    !15
CALL E_PLANE_STRESS (E)                                         !16
!                                                                    !17
DO IP = 1, LT_QP        !      NUMERICAL INTEGRATION LOOP      !18
  G      = GET_G_AT_QP  (IP)          ! GEOMETRY INTERPOLATIONS !19
  GEOMETRY = COORD (1:LT_GEOM,:)      ! GEOMETRY NODES         !20
  XYZ     = MATMUL ( G, GEOMETRY )    ! COORDINATES OF POINT   !21
!                                                                    !22
  DLG     = GET_DLG_AT_QP (IP)        ! GEOMETRIC DERIVATIVES !23
  AJ      = GEOMETRIC_JACOBIAN ( )    ! JACOBIAN               !24
  CALL INVERT_2BY2 (AJ, AJ_INV, DET)  ! INVERSE, DET           !25
  DET_WT  = DET * WT (IP) * THICK     !                        !26
!                                                                    !27
  H       = GET_H_AT_QP  (IP)        ! SCALAR INTERPOLATIONS !28
  DLH    = GET_DLH_AT_QP (IP)        ! SCALAR DERIVATIVES    !29
  DGH    = MATMUL ( AJ_INV, DLH )    ! PHYSICAL DERIVATIVES  !30
!                                                                    !31
!----> FORM STRAIN-DISPLACEMENT, B (OR GET_APPLICATION_B_MATRIX) !32
CALL ELASTIC_B_PLANAR (DGH, B)      !33
!                                                                    !34
! EVALUATE ELEMENT MATRICES          !35
S = S + DET_WT * MATMUL (TRANSPOSE(B), MATMUL (E, B))          !36
!                                                                    !37
! SAVE PT, CONSTITUTIVE & STRAIN_DISP FOR POST_PROCESS & SCP    !38
CALL STORE_FLUX_POINT_DATA (XYZ, E, B)                             !39
END DO ! Over quadrature points                                       !40
!      ***  END ELEM_SQ_MATRIX PROBLEM DEPENDENT STATEMENTS ***    !41

```

Figure 11.4.5 A general plane stress isotropic stiffness matrix

A good engineer should have an estimate of the desired solution before approaching the computer. For example, if the load had been at the center of the edge, then

$$\sigma_x = P/A = 10^4/(2) (5 \times 10^{-3}) = 10^6 \text{ N/m}^2,$$

and $\sigma_y = 0 = \tau$. The values are significantly different from the computed values. A better estimate would consider both the axial and bending effects so $\sigma_x = P/A \pm Mc/I$. At the centroid of these two elements ($y = 0.667$ and $y = 1.333$) the revised stress estimates are $\sigma_x = 0$ and $\sigma_x = 2 \times 10^6 \text{ N/m}^2$, respectively. The revised difference between the maximum centroidal stress and our estimate is only 10 %. Of course, with the insight gained from the mechanics of materials our mesh was not a good selection. We know that while an axial stress would be constant across the depth of the member, the bending effects would vary linearly with y . Thus, it was poor judgement to select a single element

through the thickness. These hand calculations are validated in the selected output file shown in Fig. 11.4.4.

To select a better mesh we should imagine how the stress would vary through the member. Then we would decide how many constant steps are required to get a good fit to the curve. Similarly, if we employed linear stress triangles (LST) we would estimate the required number of piece-wise linear segments needed to fit the curve. For example, consider a cantilever beam subjected to a bending load at its end. We know the exact normal stress is linear through the thickness and the shear stress varies quadratically through the thickness. Thus, through the depth we would need several CST, or a few LST, or a single cubic triangle (QST).

Converting to such higher order elements is relatively simple if we employ numerical integration. Extending the numerically integrated scalar element square matrix of Fig. 10.8.1 a plane stress formulation, independent of element type, is obtained as shown in Fig. 11.4.5. The corresponding element stress recovery at each quadrature point is given in Fig. 11.4.6. The stiffness matrix given in Fig. 11.4.5 is actually basically the form that would be needed for any 1-, 2-, 3-dimensional, or axisymmetric solid. For example, one would mainly need to change two calls and make them more general. Line 16 recovers the constitutive matrix. It could be replaced with a call, say to `E_ISOTROPIC_STRESS (E)`, that had the necessary logic to treat the 5 cases cited above. The choices for \mathbf{E}^e mainly depend on the dimension of the space (`N_SPACE` set by keyword `space`) and whether the keyword `axisymmetric` is present. Usually the state of plane stress is taken as the default in a 2-dimensional model (if not axisymmetric) so one would have to provide another logical control variable (say keyword `plane_strain`) to allow for activating that condition. Likewise, the strain-displacement call for \mathbf{B}^e (line 33) could be changed to say, `ELASTIC_B_MATRIX (DGH, H, XYZ, B)`, as a general form that works for any of the above 4 cases by allowing for an axisymmetric model to use the current point, `XYZ`, and the interpolation, `H`, needed to obtain the hoop strain. The choices for \mathbf{B}^e depend only on the dimension of the space and whether the keyword `axisymmetric` is present.

It is always wise to test such implementations by means of a numerical patch test. Data for one such test is given in Fig. 11.4.7 where both the u and v displacement components are prescribed, by the same equation, at all the boundary nodes as essential conditions. Then the one interior node displacement is computed. All displacements are then employed to obtain the generalize flux components (here mechanical stresses) and compared to the corresponding constant values assumed in the analytic expression picked to define the patch test. Here we see that the results for displacements and stresses, in Fig. 11.4.8, are everywhere exact and we pass the patch test and thus assume the programming is reasonably correct. In the latter figure the one interior node displacement vector actually computed is node 5 (line 13) while the other 8 node displacement vectors were prescribed as essential boundary conditions. The element post-processing results are given in lines 43-55. The flux (stress) components at the integration points (lines 19-29), and there smoothed values at the nodes (lines 31-41) are default output when numerical integration is used (keyword `gauss` > 0) unless they are turned off by keyword `no_scp`. We will consider the super-convergent patch (scp) methods in later chapters.

```

!   *** POST_PROCESS_ELEM PROBLEM DEPENDENT STATEMENTS FOLLOW ***   ! 1
!   PLANE_STRESS ANALYSIS                                           ! 2
! STRESS AND STRAIN COMPONENT ORDER: XX, YY, XY, SO N_R_B = 3      ! 3
!                                                                     ! 4
! PROPERTIES: 1-YOUNG'S MODULUS, 2-POISSON'S RATIO, AND           ! 5
!   3-YIELD STRESS, IF PRESENT                                     ! 6
INTEGER   :: J, N_IP                                             ! LOOPS   ! 7
REAL(DP), SAVE :: YIELD                                         ! FAILURE DATA ! 8
!                                                                     ! 9
IF ( IE == 1 ) THEN ! PRINT TITLES & INITIALIZE                   !10
  STRAIN = 0.d0 ; STRAIN_0 = 0.d0 ! INITIALIZE ALL OF "STRAIN" !11
  IF ( EL_REAL > 2 ) THEN ! INITIALIZE YIELD STRESS             !12
    YIELD = GET_REAL_LP (3) !13
  ELSE ; YIELD = HUGE (1.d0) ; END IF ! YIELD DATA             !14
!                                                                     !15
  WRITE (6, 50) ; 50 FORMAT ( /,                                     & !16
    '*** STRESSES AT INTEGRATION POINTS ***', /, & !17
    ' COORDINATES STRESSES', /, & !18
    'POINT X Y XX YY', /, & !19
    'POINT XY EFFECTIVE') !20
  END IF ! NEW HEADINGS !21
!                                                                     !22
  WRITE (6, * ) ' ELEMENT NUMBER ', IE !23
  CALL READ_FLUX_POINT_COUNT (N_IP) ! NUMBER OF QUADRATURE POINTS !24
  DO J = 1, N_IP ! AT QUADRATURE POINTS !25
!                                                                     !26
    CALL READ_FLUX_POINT_DATA (XYZ, E, B) ! PT, PROP, STRAIN_DISP !27
!                                                                     !28
! MECHANICAL STRAINS & STRESSES !29
    STRAIN (1:N_R_B) = MATMUL (B, D) ! STRAINS AT THE POINT !30
    STRESS = MATMUL (E, STRAIN) ! CALCULATE STRESSES !31
!                                                                     !32
! VON_MISES FAILURE CRITERION (EFFECTIVE STRESS, ADD TO END) !33
    STRESS (4) = SQRT ( (STRESS (1) - STRESS (2) ) **2 & !34
      + (STRESS (2) ) **2 + (STRESS (1) ) **2 & !35
      + 6.d0 * STRESS (3) **2 ) * 0.7071068d0 !36
    IF ( STRESS (4) >= YIELD ) PRINT *, & !37
      'WARNING: FAILURE CRITERION EXCEEDED IN ELEMENT =', IE !38
!                                                                     !39
! LIST STRESSES AND FAILURE CRITERION AT POINT !40
    WRITE (6, 52) J, XYZ (1:2), STRESS (1:2) !41
    WRITE (6, 51) J, STRESS (3:4) !42
    52 FORMAT ( I3, 2(1PE11.3), 5(1PE14.5) ) !43
    51 FORMAT ( I3, 22X, 5(1PE14.5) ) !44
!                                                                     !45
  END DO ! AT QUADRATURE POINTS !46
!   *** END POST_PROCESS_ELEM PROBLEM DEPENDENT STATEMENTS ***   !47

```

Figure 11.4.6 Plane stress mechanical stress recovery

```

title "2D STRESS PATCH TEST, T6 ESSENTIAL BC" ! 1
nodes 9 ! Number of nodes in the mesh ! 2
elems 2 ! Number of elements in the system ! 3
dof 2 ! Number of unknowns per node ! 4
el_nodes 6 ! Maximum number of nodes per element ! 5
space 2 ! Solution space dimension ! 6
b_rows 3 ! Number of rows in the B (operator) matrix ! 7
shape 2 ! Element shape, 1=line, 2=tri, 3=quad, 4=hex ! 8
gauss 4 ! number of quadrature points ! 9
el_real 2 ! Number of real properties per element !10
el_homo ! Element properties are homogeneous !11
post_el ! Require element post-processing !12
example 201 ! Source library example number !13
data_set 01 ! Data set for example (this file) !14
exact_case 12 ! Exact analytic solution !15
list_exact ! List given exact answers at nodes, etc !16
list_exact_flux ! List given exact fluxes at nodes, etc !17
remarks 8 ! Number of user remarks !18
quit ! keyword input !19
Note: Patch test yields constant gradient and strains !20
3--6---9 Mesh to left. Exact solution u = 1 + 3x - 4y !21
:(2) / : du/dx = 3, du/dy = -4 !22
2 5 8 Exact solution v = 1 + 3x - 4y !23
: / (1): dv/dx = 3, dv/dy = -4 !24
1/--4--7 Thus answer at node 5 is u = v = -1 !25
Strains: 3, -4, -1. Stresses: 3, -4, -0.5, for E=1, nu=0 !26
Stresses: 2.13333, -3.46667, -0.4, for E=1, nu=0.25 !27
1 11 0.0 0.0 ! begin nodes !28
2 11 0.0 2.0 !29
3 11 0.0 4.0 !30
4 11 2.0 0.0 !31
5 0 2.0 2.0 ! only unknown !32
6 11 2.0 4.0 !33
7 11 4.0 0.0 !34
8 11 4.0 2.0 !35
9 11 4.0 4.0 !36
1 1 7 9 4 8 5 ! begin elements !37
2 1 9 3 5 6 2 !38
1 1 1.0 ! essential bc !39
2 1 -7.0 !40
3 1 -15.0 !41
4 1 7.0 !42
6 1 -9.0 !43
7 1 13.0 !44
8 1 5.0 !45
9 1 -3.0 !46
1 2 1.0 ! essential bc !47
2 2 -7.0 !48
3 2 -15.0 !49
4 2 7.0 !50
6 2 -9.0 !51
7 2 13.0 !52
8 2 5.0 !53
9 2 -3.0 !54
1 1.0 0.25 0.0 ! el, E, Nu, yield !55

```

Figure 11.4.7 Plane stress patch test data


```

TITLE: "2D STRESS PATCH TEST, T6 ESSENTIAL BC" ! 1
! 2
*** SYSTEM GEOMETRIC PROPERTIES *** ! 3
VOLUME = 1.60000E+01 ! 4
CENTROID = 2.00000E+00 2.00000E+00 ! 5
! 6
*** OUTPUT OF RESULTS AND EXACT VALUES IN NODAL ORDER *** ! 7
NODE X-Coord Y-Coord DOF_1 DOF_2 EXACT1 EXACT2 ! 8
1 0.00E+0 0.00E+0 1.0000E+0 1.0000E+0 1.0000E+0 1.0000E+0 ! 9
2 0.00E+0 2.00E+0 -7.0000E+0 -7.0000E+0 -7.0000E+0 -7.0000E+0 !10
3 0.00E+0 4.00E+0 -1.5000E+1 -1.5000E+1 -1.5000E+1 -1.5000E+1 !11
4 2.00E+0 0.00E+0 7.0000E+0 7.0000E+0 7.0000E+0 7.0000E+0 !12
5 2.00E+0 2.00E+0 -1.0000E+0 -1.0000E+0 -1.0000E+0 -1.0000E+0 !13
6 2.00E+0 4.00E+0 -9.0000E+0 -9.0000E+0 -9.0000E+0 -9.0000E+0 !14
7 4.00E+0 0.00E+0 1.3000E+1 1.3000E+1 1.3000E+1 1.3000E+1 !15
8 4.00E+0 2.00E+0 5.0000E+0 5.0000E+0 5.0000E+0 5.0000E+0 !16
9 4.00E+0 4.00E+0 -3.0000E+0 -3.0000E+0 -3.0000E+0 -3.0000E+0 !17
!18
*** FE AND EXACT FLUX COMPONENTS AT INTEGRATION POINTS *** !19
EL X-Coord Y-Coord FLUX_1 FLUX_2 FLUX_3 EXACT1 EXACT2 EXACT3 !20
1 2.67E+0 1.33E+0 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !21
1 3.20E+0 8.00E-1 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !22
1 3.20E+0 2.40E+0 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !23
1 1.60E+0 8.00E-1 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !24
EL X-Coord Y-Coord FLUX_1 FLUX_2 FLUX_3 EXACT1 EXACT2 EXACT3 !25
2 1.33E+0 2.67E+0 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !26
2 2.40E+0 3.20E+0 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !27
2 8.00E-1 3.20E+0 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !28
2 8.00E-1 1.60E+0 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !29
!30
** SUPER_CONVERGENT AVERAGED NODAL FLUXES & EXACT FLUXES ** !31
PT X-Coord Y-Coord FLUX_1 FLUX_2 FLUX_3 EXACT1 EXACT2 EXACT3 !32
1 0.00E+0 0.00E+0 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !33
2 0.00E+0 2.00E+0 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !34
3 0.00E+0 4.00E+0 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !35
4 2.00E+0 0.00E+0 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !36
5 2.00E+0 2.00E+0 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !37
6 2.00E+0 4.00E+0 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !38
7 4.00E+0 0.00E+0 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !39
8 4.00E+0 2.00E+0 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !40
9 4.00E+0 4.00E+0 2.13E0 -3.47E0 -4.00E-1 2.13E0 -3.47E0 -4.00E-1 !41
!42
*** STRESSES AT INTEGRATION POINTS *** !43
COORDINATES STRESSES !44
PT X Y XX YY XY EFFECTIVE !45
ELEMENT NUMBER 1 !46
1 2.667E+0 1.333E+0 2.1333E+0 -3.4667E+0 -4.0000E-1 4.9441E+0 !47
2 3.200E+0 8.000E-1 2.1333E+0 -3.4667E+0 -4.0000E-1 4.9441E+0 !48
3 3.200E+0 2.400E+0 2.1333E+0 -3.4667E+0 -4.0000E-1 4.9441E+0 !49
4 1.600E+0 8.000E-1 2.1333E+0 -3.4667E+0 -4.0000E-1 4.9441E+0 !50
ELEMENT NUMBER 2 !51
1 1.333E+0 2.667E+0 2.1333E+0 -3.4667E+0 -4.0000E-1 4.9441E+0 !52
2 2.400E+0 3.200E+0 2.1333E+0 -3.4667E+0 -4.0000E-1 4.9441E+0 !53
3 8.000E-1 3.200E+0 2.1333E+0 -3.4667E+0 -4.0000E-1 4.9441E+0 !54
4 8.000E-1 1.600E+0 2.1333E+0 -3.4667E+0 -4.0000E-1 4.9441E+0 !55

```

Figure 11.4.8 Correct plane stress patch test results

11.5 Stress and Strain Transformations *

Having computed the global stress components at a point in an element, we may wish to find the stresses in another direction. This can be done by employing the transformations associated with *Mohr's circle*. Mohr's circles of stress and strain are usually used to produce graphical solutions. However, here we wish to rely on automated numerical solutions. Thus, we will review the *stress transformation* laws. Refer to Fig. 11.5.1 where the quantities used in Mohr's transformation are defined. The alternate coordinate set (n, s) is used to describe the surfaces on which the normal stresses, σ_n and σ_s , and the shear stress, τ_{ns} , act. The n -axis is rotated from the x -axis by a positive (counter-clockwise) angle of β . By considering the equilibrium of the differential element, it is shown in mechanics of materials that

$$\sigma_n = \sigma_x \cos^2 \beta + \sigma_y \sin^2 \beta + 2\tau_{xy} \sin \beta \cos \beta . \tag{11.29}$$

Likewise the shear stress component is found to be

$$\tau_{ns} = -\sigma_x \sin \beta \cos \beta + \sigma_y \sin \beta \cos \beta + \tau_{xy}(\cos^2 \beta - \sin^2 \beta) . \tag{11.30}$$

For Mohr's circle only these two stresses are usually plotted in the $\sigma_n - \tau_{ns}$ space. However, for a useful analytical statement we also need to define σ_s . Again from equilibrium considerations it is easy to show that

$$\sigma_s = \sigma_x \sin^2 \beta + \sigma_y \cos^2 \beta - 2\tau_{xy} \sin \beta \cos \beta . \tag{11.31}$$

Prior to this point we have employed matrix notation to represent the stress components. Then we were considering only the global coordinates. But now when we refer to the stress components it will be necessary to indicate which coordinate system is being utilized. We will employ the subscripts xy and ns to distinguish between the two systems. Thus, our previous stress component array will be denoted by

$$\boldsymbol{\sigma}^T = \boldsymbol{\sigma}_{xy}^T = [\sigma_x \quad \sigma_y \quad \tau_{xy}]$$

while the new stress components will be ordered in a similar manner and denoted by

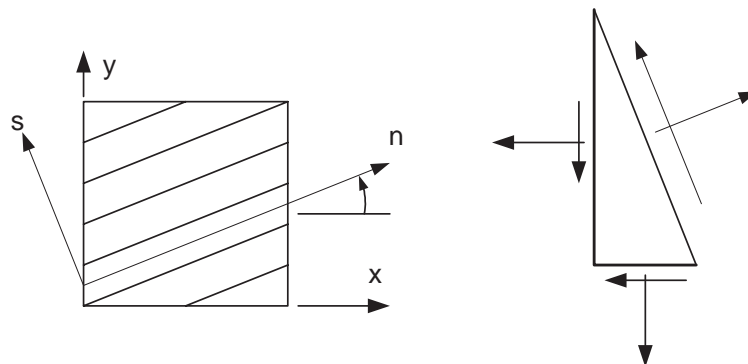


Figure 11.5.1 Local material or stress axes

$$\boldsymbol{\sigma}_{ns}^T = [\sigma_n \quad \sigma_s \quad \tau_{ns}].$$

In this notation the stress transformation laws can be written as

$$\begin{Bmatrix} \sigma_n \\ \sigma_s \\ \tau_{ns} \end{Bmatrix} = \begin{bmatrix} +C^2 & +S^2 & +2SC \\ +S^2 & +C^2 & -2SC \\ -SC & +SC & (C^2 - S^2) \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} \quad (11.32)$$

where $C \equiv \text{Cos } \beta$ and $S \equiv \text{Sin } \beta$ for simplicity. In symbolic matrix form this is

$$\boldsymbol{\sigma}_{ns} = \mathbf{T}(\beta) \boldsymbol{\sigma}_{xy} \quad (11.33)$$

where \mathbf{T} will be defined as the stress transformation matrix. Clearly, if one wants to know the stresses on a given plane one specifies the angle β , forms \mathbf{T} , and computes the results from Eq. (11.32).

A similar procedure can be employed to express Mohr's circle of strain as a strain matrix transformation law. Denoting the new strains as $\boldsymbol{\epsilon}_{ns}^T = [\epsilon_n \quad \epsilon_s \quad \gamma_{ns}]$ then the strain transformation law is

$$\begin{Bmatrix} \epsilon_n \\ \epsilon_s \\ \gamma_{ns} \end{Bmatrix} = \begin{bmatrix} +C^2 & +S^2 & +SC \\ +S^2 & +C^2 & -SC \\ -2SC & +2SC & (C^2 - S^2) \end{bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix} \quad (11.34)$$

or simply

$$\boldsymbol{\epsilon}_{ns} = \mathbf{t}(\beta) \boldsymbol{\epsilon}_{xy}. \quad (11.35)$$

Note that the two transformation matrices, \mathbf{T} and \mathbf{t} , are not identical. This is true because we have selected the engineering definition of the shear strain (instead of using the tensor definition). Also note that both of the transformation matrices are square. Therefore, the reverse relations can be found by inverting the transformations, that is,

$$\boldsymbol{\sigma}_{xy} = \mathbf{T}(\beta)^{-1} \boldsymbol{\sigma}_{ns}, \quad \boldsymbol{\epsilon}_{xy} = \mathbf{t}(\beta)^{-1} \boldsymbol{\epsilon}_{ns}. \quad (11.36)$$

These two transformation matrices have the special property that the inverse of one is the transpose of the other, that is, it can be shown that

$$\mathbf{T}^{-1} = \mathbf{t}^T, \quad \mathbf{t}^{-1} = \mathbf{T}^T. \quad (11.37)$$

This property is also true when generalized to three-dimensional properties. Another generalization is to note that if we partition the matrices into normal and shear components, then

$$\mathbf{T} = \begin{bmatrix} \mathbf{T}_{11} & \mathbf{T}_{12} \\ \mathbf{T}_{21} & \mathbf{T}_{22} \end{bmatrix}, \quad \mathbf{t} = \begin{bmatrix} \mathbf{T}_{11} & \mathbf{T}_{12}/2 \\ 2\mathbf{T}_{21} & \mathbf{T}_{22} \end{bmatrix}.$$

In mechanics of materials it is shown that the principle normal stresses occur when the angle is given by $\text{Tan}(2\beta_p) = 2\tau_{xy}/(\sigma_x - \sigma_y)$. Thus, if β_p were substituted into Eq. (11.31) one would compute the two principle normal stresses. In this case it may be easier to use the classical form that

$$\sigma_p = \frac{\sigma_x + \sigma_y}{2} \pm \left[\left(\frac{\sigma_x - \sigma_y}{2} \right)^2 + \tau_{xy}^2 \right]^{\frac{1}{2}}.$$

However, to illustrate the use of Eq. (11.31) we will use the results of the previous two element plane stress example to find the maximum normal stress at the second element centroid. Then

$$\tan(2\beta_p) = 2(20.17)/(179.83 - 20.17) = 0.2017$$

so $\beta_p = 5.70^\circ$, $\cos \beta_p = 0.995$, $\sin \beta_p = 0.099$, and the transformation is

$$\begin{Bmatrix} \sigma_n \\ \sigma_s \\ \tau_{ns} \end{Bmatrix} = \begin{bmatrix} 0.9901 & 0.0099 & 0.1977 \\ 0.0099 & 0.9901 & -0.1977 \\ -0.0989 & 0.0989 & 0.9803 \end{bmatrix} \begin{Bmatrix} 179.83 \\ -20.17 \\ 20.17 \end{Bmatrix}$$

or

$$\boldsymbol{\sigma}_{ns}^T = [181.84 \quad -22.18 \quad -0.00] \text{ N/m}^2.$$

The maximum shear stress is $\tau_{\max} = (\sigma_n - \sigma_s)/2 = 102.01 \text{ N/m}^2$. These shear stresses occur on planes located at $(\beta_p \pm 45^\circ)$. The classical form for τ_{\max} for two-dimensional problems is $\tau_{\max}^2 = [(\sigma_x - \sigma_y)/2]^2 + \tau_{xy}^2$.

The above example is not finished at this point. In practice, we probably would want to check the failure criterion for this material, and obtain an error estimate to begin an adaptive solution. There are many failure criteria. The three most common ones are the Maximum Principal Stress, the Maximum Shear Stress, and the Von Mises Strain Energy of Distortion. The latter is most common for ductile materials. It can be expressed in terms of a scalar measure known as the *Effective Stress*, σ_E :

$$\sigma_E = \frac{1}{\sqrt{2}} \sqrt{(\sigma_x - \sigma_y)^2 + (\sigma_x - \sigma_z)^2 + (\sigma_y - \sigma_z)^2 + 6(\tau_{xy}^2 + \tau_{xz}^2 + \tau_{yz}^2)}$$

$$\sigma_E = \frac{1}{\sqrt{2}} \sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_1 - \sigma_3)^2 + (\sigma_2 - \sigma_3)^2}$$

in terms of the stress tensor components and principal stresses, respectively. For yielding in a simple tension test, $\sigma_x = \sigma_{yield}$, and all the other stresses are zero. Then, the effective stress becomes $\sigma_E = \sigma_{yield}$ which implies failure. This is the general test for ductile materials. For brittle materials, one may use the maximum stress criteria where failure occurs at $\sigma_1 = \sigma_{yield}$. The Tresca maximum shear stress criteria is also commonly used. With it failure occurs at $\tau_{\max} = \sigma_{yield}/2$. For the plane stress state, all the z -components of the stress tensor are zero. However, in the state of plane strain, σ_z is not zero and must be recovered using the Poisson ratio effect. For an isotropic material (without an initial thermal strain) the result is

$$\sigma_z = \frac{E}{(1 + \nu)(1 - 2\nu)} \left[\nu \sigma_x + \nu \sigma_y \right].$$

11.6 Axisymmetric Solids*

There is another common elasticity problem class that can also be formulated as a two-dimensional problem involving two unknown displacement components. It is an axisymmetric solid subject to axisymmetric loads and axisymmetric supports. That is, the geometry, properties, loads, and supports do not have any variation around the circumference of the solid. The problem is usually discussed in terms of axial and radial position, and axial and radial displacements. The solid is defined by the shape in the radial-axial plane as it is completely revolved about the axis. Let (R, Z) denote the coordinates in the plane of revolution, and u, v denote the corresponding radial and axial displacements at any point. This is an extension of the methods in Chap. 6 in that we now allow changes in the axial, Z , direction. The axisymmetric solid has four stress and strain components, three of them the same as those in the state of plane stress. We simply replace the x, y subscripts with R, Z .

The fourth strain is the so-called *hoop strain*. It arises because of the material around the circumference changes length as it moves radially. The circumferential strain at a radial position, R , is defined in Fig. 6.2.1 as

$$\varepsilon_{\Theta} = \frac{\Delta L}{L} = \frac{2\pi(R + u) - 2\pi R}{2\pi R} = \frac{u}{R}. \quad (11.45)$$

This is a normal strain, and it is usually placed after the other two normal strains. Note that on the axis of revolution, $R = 0$. It can be shown that both $u = 0$ and $\varepsilon_{\Theta} = 0$ on the axis of revolution. However, one can encounter numerical problems if numerical integration is employed with a rule that has quadrature points on the edge of an element.

We typically order the strains as $\boldsymbol{\varepsilon}^T = [\varepsilon_R \ \varepsilon_Z \ \varepsilon_{\Theta} \ \gamma_{RZ}]$ and the corresponding stresses as $\boldsymbol{\sigma} = [\sigma_R \ \sigma_Z \ \sigma_{\Theta} \ \tau_{RZ}]$ where σ_{Θ} is the corresponding *hoop stress*. From the above definition we now see that the contribution of a typical node j to the strain displacement $\mathbf{B}^e = \mathbf{L}\mathbf{N}^e$ matrix is

$$\mathbf{B}_j^e = \begin{bmatrix} \frac{\partial H_j}{\partial R} & 0 \\ 0 & \frac{\partial H_j}{\partial Z} \\ H_j/R & 0 \\ \frac{\partial H_j}{\partial Z} & \frac{\partial H_j}{\partial R} \end{bmatrix}. \quad (11.46)$$

Therefore, we now see that in addition to the physical derivatives of the interpolation functions, we now must also include the actual interpolation functions and the radial coordinate. The implementation of the axisymmetric version is shown in Fig. 11.6.1.

With the above changes and the observation that $d\Omega = 2\pi R da$, we note that the analytic integrals involve terms with $1/R$. These introduce logarithmic terms where we used to have only polynomial terms. Some of these become indeterminate at $R = 0$. For this and other practical considerations, one almost always employs numerical integration to form the element matrices. Clearly, one must interpolate from the given data to find the radial coordinate, R_q , at a quadrature point. That requires that we need to select quadrature rules that do not give points on the element boundary which does occur for

```

SUBROUTINE ELASTIC_B_AXISYMMETRIC (DGH, R, B) ! 1
! * * * * * ! 2
! AXISYMMETRIC ELASTICITY STRAIN-DISPLACEMENT RELATIONS (B) ! 3
! STRESS & STRAIN COMPONENT ORDER: RR, ZZ, RZ, AND TT ! 4
! * * * * * ! 5
Use System_Constants ! for DP, N_R_B, N_G_DOF, N_SPACE ! 6
Use Elem_Type_Data ! for LT_FREE, LT_N, H (LT_N) ! 7
  IMPLICIT NONE ! 8
  REAL(DP), INTENT(IN) :: DGH (N_SPACE, LT_N) ! Gradients ! 9
  REAL(DP), INTENT(IN) :: R ! Radius !10
  REAL(DP), INTENT(OUT) :: B (N_R_B, LT_N * N_G_DOF) ! Strains !11
  INTEGER :: J, K, L !12
!13
! B = STRAIN-DISPLACEMENT MATRIX (RETURNED) !14
! DGH = GLOBAL DERIVATIVES OF H !15
! H = ELEMENT INTERPOLATION FUNCTIONS !16
! LT_N = NUMBER OF NODES PER ELEMENT TYPE !17
! N_G_DOF = NUMBER OF PARAMETERS PER NODE = 2 here (U & V) !18
! N_R_B = NUMBER OF STRAINS (ROWS IN B) = 4: XX, YY, XY, HOOP !19
! N_SPACE = DIMENSION OF SPACE = 2 here !20
! R,Z,T DENOTE RADIAL, AXIAL, CIRCUMFERENCE !21
!22
  B = 0.d0 !23
  DO J = 1, LT_N ! ROW NUMBER !24
    K = N_G_DOF * (J - 1) + 1 ! FIRST COLUMN, U !25
    L = K + 1 ! SECOND COLUMN, V !26
!27
    B (1, K) = DGH (1, J) ! DU/DX FOR XX NORMAL !28
    B (3, K) = DGH (2, J) ! DU/DY FOR XY SHEAR !29
    IF ( R <= 0.d0 ) STOP 'R=0, IN ELASTIC_B_AXISYMMETRIC' !30
    B (4, K) = H (J) / R ! U/R HOOP, ZZ NORMAL !31
    B (2, L) = DGH (2, J) ! DV/DY FOR YY NORMAL !32
    B (3, L) = DGH (1, J) ! DV/DX FOR XY SHEAR !33
  END DO !34
END SUBROUTINE ELASTIC_B_AXISYMMETRIC !35

```

Figure 11.6.1 Axisymmetric strain-displacement matrix

some triangular rules and the Lobatto rules.

11.7 General Solids

For the completely general three-dimensional solid, there are three displacement components, $\mathbf{u}^T = [u \ v \ w]$, and the corresponding load vectors have three components. There are six stresses, $\boldsymbol{\sigma}^T = \begin{bmatrix} \sigma_x & \sigma_y & \sigma_z & \tau_{xy} & \tau_{xz} & \tau_{yz} \end{bmatrix}$, and six corresponding strain components, $\boldsymbol{\epsilon}^T = \begin{bmatrix} \epsilon_x & \epsilon_y & \epsilon_z & \gamma_{xy} & \gamma_{xz} & \gamma_{yz} \end{bmatrix}$, and thus the constitutive array \mathbf{E} is 6 by 6 in size. The engineering strains are defined by a partition at node j of $\mathbf{B}^e = \mathbf{L}\mathbf{N}^e$:

$$\mathbf{B}_j^e = \begin{bmatrix} \frac{\partial H_j}{\partial x} & 0 & 0 \\ 0 & \frac{\partial H_j}{\partial y} & 0 \\ 0 & 0 & \frac{\partial H_j}{\partial z} \\ \frac{\partial H_j}{\partial y} & \frac{\partial H_j}{\partial x} & 0 \\ \frac{\partial H_j}{\partial z} & 0 & \frac{\partial H_j}{\partial x} \\ 0 & \frac{\partial H_j}{\partial z} & \frac{\partial H_j}{\partial y} \end{bmatrix}. \quad (11.47)$$

11.8 Anisotropic Materials*

A material is defined to be *isotropic* if its material properties do not depend on direction. Otherwise it is called *anisotropic*. Most engineering materials are considered to be isotropic. However, there are many materials that are anisotropic. Examples of anisotropic materials include reinforced concrete, plywood, and filament wound fiberglass. Probably the most common case is that of an *orthotropic material*. An orthotropic material has structural (or thermal) properties that can be defined in terms of two principal material axis directions. Let (n, s) be the principal material axis directions. For anisotropic materials it is usually easier to define the generalized constitutive law in the form:

$$\boldsymbol{\varepsilon}_{ns} = \mathbf{E}_{ns}^{-1} \boldsymbol{\sigma}_{ns} + \boldsymbol{\varepsilon}_{0ns}. \quad (11.38)$$

Note by way of comparison that Eq. (11.38) is written relative to the global coordinate axes. In Eq. (11.14) the square matrix contains the mechanical properties as experimentally measure relative to the principal material directions. For a two-dimensional orthotropic material the constitutive law is

$$\begin{Bmatrix} \varepsilon_n \\ \varepsilon_s \\ \gamma_{ns} \end{Bmatrix} = \begin{bmatrix} 1/E_n & -\nu_{sn}/E_s & 0 \\ -\nu_{ns}/E_n & 1/E_s & 0 \\ 0 & 0 & 1/G_{ns} \end{bmatrix} \begin{Bmatrix} \sigma_n \\ \sigma_s \\ \tau_{ns} \end{Bmatrix} + \boldsymbol{\varepsilon}_{0ns}. \quad (11.39)$$

Here the moduli of elasticity in the two principal directions are denoted by E_n and E_s . The shear modulus, G_{ns} , is independent of the elastic moduli. The two Poisson's ratios are defined by the following notation:

$$\nu_{ij} = \varepsilon_i / \varepsilon_j \quad (11.40)$$

where i denotes the direction of the load, ε_i is the normal strain in the load directions, and ε_j is the normal strain in the transverse (orthogonal) direction. Symmetry considerations result in the additional requirement that

$$E_n \nu_{sn} = E_s \nu_{ns}. \quad (11.41)$$

Thus, four independent constants must be measured to define the orthotropic material

mechanical properties. If the material is isotropic then $\nu = \nu_{ns} = \nu_{sn}$, $E = E_s = E_n$, and $G = G_{ns} = E/[2(1 + \nu)]$. In that case only two constants (E and ν) are required and they can be measured in any direction. When the material is isotropic then Eq. (11.39) reduces to Eq. (11.15).

The axisymmetric stress-strain law for an isotropic material was given earlier. For an orthotropic axisymmetric material, we utilize the material properties in the principal material axis (n, s, Θ) direction. In that case, the compliance matrix, \mathbf{E}_{ns}^{-1} , and initial strain matrix, $\boldsymbol{\varepsilon}_{\Theta ns}$ are

$$\begin{Bmatrix} \varepsilon_n \\ \varepsilon_s \\ \varepsilon_\Theta \\ \gamma_{ns} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_n} & -\frac{\nu_{sn}}{E_s} & -\frac{\nu_{\Theta n}}{E_\Theta} & 0 \\ -\frac{\nu_{ns}}{E_n} & \frac{1}{E_s} & -\frac{\nu_{\Theta s}}{E_\Theta} & 0 \\ -\frac{\nu_{n\Theta}}{E_n} & -\frac{\nu_{s\Theta}}{E_s} & \frac{1}{E_\Theta} & 0 \\ 0 & 0 & 0 & \frac{1}{G_{ns}} \end{bmatrix} + \Delta\Theta \begin{Bmatrix} \alpha_n \\ \alpha_s \\ \alpha_\Theta \\ 0 \end{Bmatrix}.$$

For the general anisotropic three-dimensional solid, there are nine independent material constants. However, due to the axisymmetry, there are only seven independent constants. When the material is also *transversely isotropic*, with the $n-t$ plane being the plane of isotropic properties, then there are only five material constants and they are related by $E_n = E_\Theta$, $\nu_{n\Theta} = \nu_{\Theta n}$, and $E_\Theta \nu_{s\Theta} = E_s \nu_{ns}$. In practical design problems with anisotropic materials, it is difficult to get accurate material constant measurements. To be a physically possible material, both \mathbf{E} and \mathbf{E}^{-1} should have a positive determinant. When that is not the case, the program should issue a warning and terminate the analysis. The possible values of an orthotropic Poisson's ratio can be bounded by

$$|\nu_{sn}| < \sqrt{E_s/E_n}, \quad -1 < \nu_{n\Theta} < 1 - 2E_n \nu_{sn}^2/E_s.$$

Orthotropic materials also have thermal properties that vary with direction. If $\Delta\theta$ denotes the temperature change from the stress free state then the local initial thermal strain is $\boldsymbol{\varepsilon}_{0ns}^T = \Delta\theta[\alpha_n \quad \alpha_s \quad 0]$ where α_n and α_s are the principal coefficients of thermal expansion. If one is given the orthotropic properties it is common to numerically invert \mathbf{E}_{ns}^{-1} to give the usual form

$$\boldsymbol{\sigma}_{ns} = \mathbf{E}_{ns}(\boldsymbol{\varepsilon}_{ns} - \boldsymbol{\varepsilon}_{0ns}). \quad (11.42)$$

This is done since the form of \mathbf{E}_{ns} is algebraically much more complicated than its inverse. Due to experimental error in measuring the anisotropic constants there is a potential difficulty with this concept. For a physically possible material it can be shown that both \mathbf{E} and \mathbf{E}^{-1} must be positive definite. This means that the determinant must be greater than zero. Due to experimental error it is not unusual for this condition to be violated. When this occurs the program should be designed to stop and require acceptable data. Then the user must adjust the experimental data to satisfy the condition that $(E_n - E_s \nu_{ns}^2) > 0$.

From the previous section on stress and strain transformations we know how to obtain $\boldsymbol{\varepsilon}_{xy}$ and $\boldsymbol{\sigma}_{xy}$ from given $\boldsymbol{\varepsilon}_{ns}$ and $\boldsymbol{\sigma}_{ns}$. But how do we obtain \mathbf{E}_{xy} from \mathbf{E}_{ns} ? We

must have \mathbf{E}_{xy} to form the stiffness matrix since it must be integrated relative to the x - y axes, that is,

$$\mathbf{K}_{xy} = \int \mathbf{B}_{xy}^T \mathbf{E}_{xy} \mathbf{B}_{xy} dv .$$

Thus, the use of the $n-s$ axes, in Fig. 11.4.1, to define (input) the material properties requires that we define one more transformation law. It is the transformation from \mathbf{E}_{ns} to \mathbf{E}_{xy} . There are various ways to derive the required transformation. One simple procedure is to recall that the strain energy density is a scalar. Therefore, it must be the same in all coordinate systems. The strain energy density at a point is $dU = \boldsymbol{\sigma}^T \boldsymbol{\varepsilon} / 2 = \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} / 2$. In the global axes it is

$$dU = \frac{1}{2} \boldsymbol{\sigma}_{xy}^T \boldsymbol{\varepsilon}_{xy} = \frac{1}{2} (\mathbf{E}_{xy} \boldsymbol{\varepsilon}_{xy})^T \boldsymbol{\varepsilon}_{xy} = \frac{1}{2} \boldsymbol{\varepsilon}_{xy}^T \mathbf{E}_{xy} \boldsymbol{\varepsilon}_{xy} .$$

In the principal material directions it is

$$dU = \frac{1}{2} \boldsymbol{\sigma}_{ns}^T \boldsymbol{\varepsilon}_{ns} = \frac{1}{2} (\mathbf{E}_{ns} \boldsymbol{\varepsilon}_{ns})^T \boldsymbol{\varepsilon}_{ns} = \frac{1}{2} \boldsymbol{\varepsilon}_{ns}^T \mathbf{E}_{ns} \boldsymbol{\varepsilon}_{ns} .$$

But from our Mohr's circle transformation for strain $\boldsymbol{\varepsilon}_{ns} = \mathbf{t}_{ns} \boldsymbol{\varepsilon}_{xy}$ so in the $n-s$ axes

$$dU = \frac{1}{2} (\mathbf{t}_{ns} \boldsymbol{\varepsilon}_{xy})^T \mathbf{E}_{ns} (\mathbf{t}_{ns} \boldsymbol{\varepsilon}_{xy}) = \frac{1}{2} \boldsymbol{\varepsilon}_{xy}^T (\mathbf{t}_{ns}^T \mathbf{E}_{ns} \mathbf{t}_{ns}) \boldsymbol{\varepsilon}_{xy} . \quad (11.43)$$

Comparing the two forms of dU gives the *constitutive transformation* law that

$$\mathbf{E}_{xy} = \mathbf{t}_{ns}^T \mathbf{E}_{ns} \mathbf{t}_{ns} . \quad (11.44)$$

The same concept holds for general three-dimensional problems.

Before leaving the concept of anisotropic materials we should review the initial thermal strains. Recall that for an isotropic material or for an anisotropic material in principal axes a change in temperature does not induce an initial shear strain. However, an anisotropic material does have initial thermal shear strain in other coordinate directions. From Eqs. (11.36) and (11.37) we have

$$\boldsymbol{\varepsilon}_{0,xy} = \mathbf{t}_{ns}^{-1} \boldsymbol{\varepsilon}_{0,ns} , \quad \left\{ \begin{array}{c} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \end{array} \right\} = \left[\begin{array}{ccc} +C^2 & +S^2 & -SC \\ +S^2 & +C^2 & +SC \\ +2SC & -2SC & (C^2 - S^2) \end{array} \right] \left\{ \begin{array}{c} \varepsilon_n^0 \\ \varepsilon_s^0 \\ 0 \end{array} \right\} .$$

Thus, the thermal shear strain is $\gamma_{xy}^0 = 2 \sin \beta \cos \beta (\varepsilon_n^0 - \varepsilon_s^0)$. This is not zero unless the two axes systems are the same ($\beta = 0$ or $\beta = \pi/2$). Therefore, one must replace the previous null terms in Eq. (11.17).

11.9 Circular Hole in an Infinite Plate

A classical problem in elasticity is that of a two-dimensional solid having a traction free hole of radius a at its center and symmetrically loaded by a uniform stress of $\sigma_x = \sigma_\infty$ along the lines of $x = \pm \infty$. The analytic solution for the stresses and the displacements are known. Let θ be the angle from the x -axis and $r \geq a$ the radius from the center point. Then the displacement components are

$$u_x = \frac{\sigma_\infty a}{8G} \left[\frac{r}{a} (\kappa + 1) \cos \theta + 2 \frac{a}{r} ((1 + \kappa) \cos \theta + \cos 3\theta) - 2 \frac{a^3}{r^3} \cos 3\theta \right]$$

$$u_y = \frac{\sigma_\infty a}{8G} \left[\frac{r}{a} (\kappa - 3) \sin \theta + 2 \frac{a}{r} ((1 - \kappa) \sin \theta + \sin 3\theta) - 2 \frac{a^3}{r^3} \sin 3\theta \right]$$

where $G = E/(1 - 2\nu)$ is the shear modulus, ν is Poisson's ratio, E is the elastic modulus and $\kappa = (3 - 4\nu)$ for plain strain or $\kappa = (3 - \nu)/(1 + \nu)$ for plane stress. Note that the displacements depend on the material properties (E, ν), but the stresses given below do not depend on the material but depend on the geometry only. That is a common situation in solid mechanics and this serves as a reminder that one must always validate both the displacements and stresses before accepting the results of a stress analysis problem. We can use the above displacement components as essential boundary conditions on the boundary of a finite domain to compare a finite element solution to the exact results as one way to validate the program for stress analysis or error estimation.

As an aside remark recall that the MODEL program allows the keyword `use_exact_bc` to employ a supplied analytic solution to override any user supplied essential boundary conditions. First one must identify the `exact_case` number assigned to the corresponding part of the exact source code library. Here for the circular hole in an infinite plate the value is 23. In this specific example the exact boundary conditions depend on material properties and the geometry. Therefore one must also supply data to the analytic solution by using the keyword `exact_reals 3` and then supply those data (E, ν, a) immediately after the user remarks and before the usual finite element nodal data. (Most examples in this book that compare to analytic solutions do not have that additional data requirement since they usually depend only on location.) In other words, if so desired one can generate a mesh that will use an analytic solution but it is only necessary to give data on where the essential boundary conditions occur and supply dummy null values which will be overwritten by the selected analytic solution, if the control keyword `use_exact_bc` is present in the data. If you want to use an analytic solution not built-in to the existing library (`exact_case 0`) then you must supply your own exact source code via the include file `my_exact_inc` (and re-compile).

The corresponding stress components in the infinite plate are

$$\sigma_x = \sigma_\infty \left[1 - \frac{a^2}{r^2} \left(\frac{3}{2} \cos 2\theta + \cos 4\theta \right) + \frac{3}{2} \frac{a^4}{r^4} \cos 4\theta \right]$$

$$\sigma_y = \sigma_\infty \left[-\frac{a^2}{r^2} \left(\frac{1}{2} \cos 2\theta - \cos 4\theta \right) - \frac{3}{2} \frac{a^4}{r^4} \cos 4\theta \right]$$

$$\tau_{xy} = \sigma_\infty \left[-\frac{a^2}{r^2} \left(\frac{1}{2} \sin 2\theta + \sin 4\theta \right) + \frac{3}{2} \frac{a^4}{r^4} \sin 4\theta \right].$$

In polar components the radial and shear stress on the hole surface $r = a$ are both zero, $\sigma_r = 0 = \tau_{r\theta}$ while the circumferential stress varies as $\sigma_\theta = \sigma_\infty(1 - 2\cos 2\theta)$. Thus it (and σ_x) has a maximum value of $3\sigma_\infty$ at $\theta = 90$ degrees. Likewise, it (and σ_y) has a

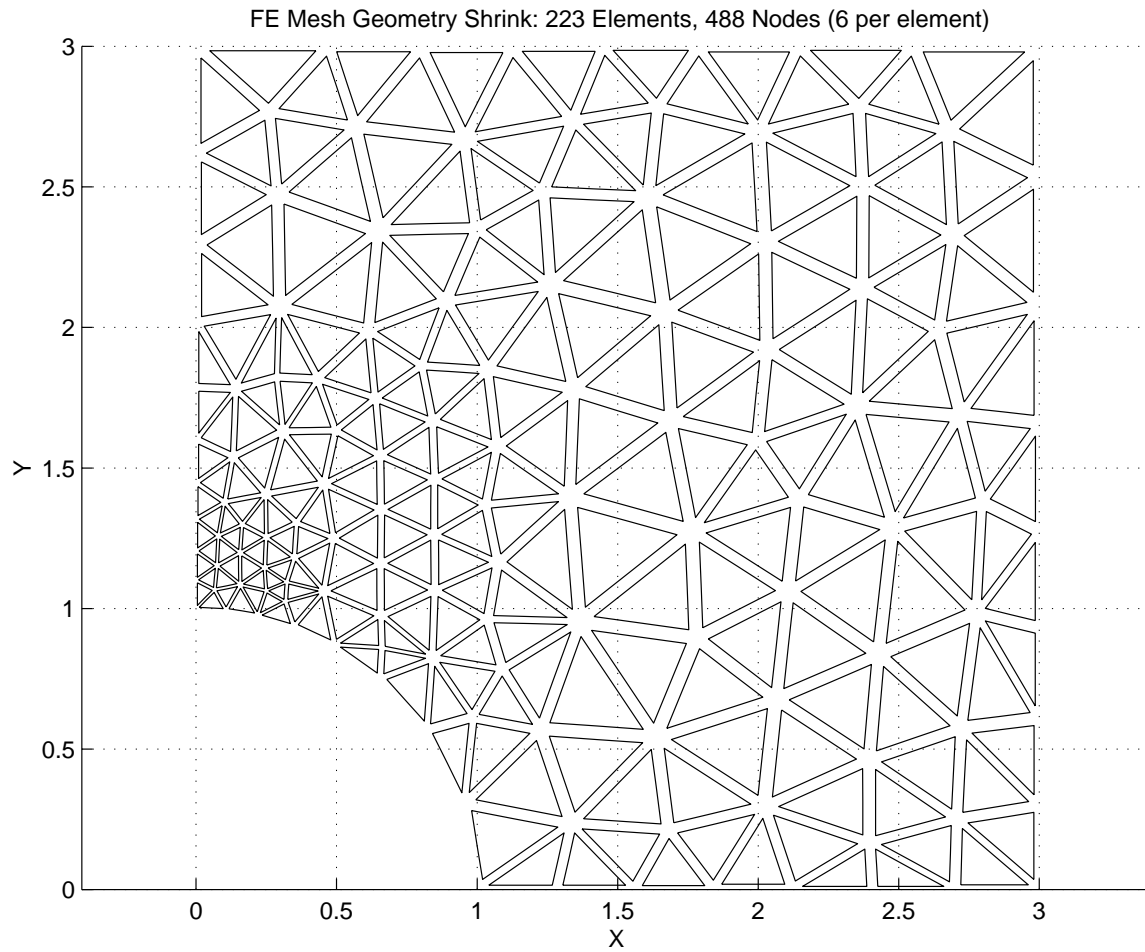


Figure 11.9.1 Exploded quadratic triangular mesh

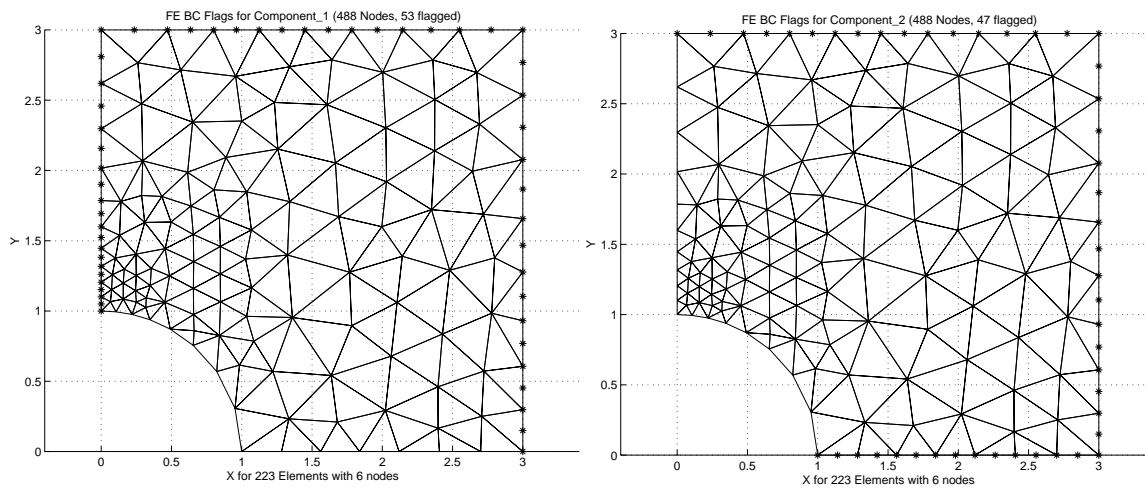


Figure 11.9.2 Nodes with essential boundary conditions (in x-, y-directions)

minimum value is $-\sigma_\infty$ at $\theta = 0$.

If we solve a finite rectangular plate with $\sigma_x = \sigma_\infty$ along $x_{\max} \gg a$ and $\sigma_y = 0$ along $y_{\max} > a$ then the peak stress values should be similar to those in the above special case. Of course, the maximum stresses will be somewhat higher since the hole takes up a bigger percentage of the center ($x = 0$) section.

Here we will present the common example of the hole in an infinite plate to illustrate finite element stress analysis and the usefulness of an error estimator. We will zoom-in on the region around the hole and use a radius of $a = 1\text{ m}$ and bound the upper right quarter of the domain with a width and height of $h = 3\text{ m}$ each. The material is taken to be aluminum ($E = 70\text{ GPa}$, $\nu = 0.33$), and we assume a traction of $T_x = 100\text{ N/m}$ at $x = \infty$. The analytic solution will specify both x - and y -displacements on the right and top edges of the mesh. The edges along the axes will invoke symmetry conditions ($v = 0$, and $u = 0$, respectively) which are consistent with the analytic solution. The edge of the circular arc is traction free and has no prescribed displacements. From undergraduate training we expect the maximum stress concentration to occur at the top point of the arc so the initial mesh is slightly refined in that region. Here we employ the 6 noded quadratic triangular (T6) elements. The initial exploded view of the mesh is given in Fig. 11.9.1, while Fig. 11.9.2 shows an asterisk at nodes where one or more displacements are prescribed.

The scaled deformed geometry and displacements are shown in Figs. 11.9.3 and 9.4. The first error estimate determines that the largest error occurs in the elements shown in Fig. 11.9.5. They are the relatively large elements around the hole and in the region near the x -axis where we will see secondary stress components are varying very rapidly. The error range is cited in the top caption of that figure. As the mesh is refined we expect that the error will become more uniformly distributed over the mesh. The total system error was 0.118 % in the energy norm.

The computed mesh refinement is shown in Fig. 11.9.6 and will be passed to the mesh generation code for creating the next analysis. In that figure the estimated new local element size is plotted relative to the centroid of the current element that was used to compute the new size. In most cases one will note a refinement giving a smaller suggested element size. In a few cases a de-refinement gives a larger new element. In most elements the projected change in size is not noticeable.

Before going on to the next mesh stage we can compare the finite element stress estimates to the exact ones. Since the plate is made of a ductile material (aluminum) we begin with the Von Mises failure criterion which gives a positive value that is compared to the yield stress of the material (about 140 MPa). The shaded planar view and carpet plot view of that criterion are shown in Figs. 11.9.7 and 8, respectively. The maximum value should be about 300.0 MPa , and it is.

The error estimates outlined in Fig. 11.9.5 are based on the nodal stress averages from the super-convergent patch processing (defined in Chapters 12 and 13). The finite element nodal averages for the horizontal normal stress σ_x are shown in Fig. 11.9.9 as a "carpet plot" and should be compared to the exact nodal distribution in while Fig. 11.9.10. The vertical normal stresses, σ_y , finite element and exact values are shown in Figs. 11.9.11 and 12, respectively. Likewise, the comparisons for the shear stress component, τ_{xy} , are in Figs. 11.9.13 and 14. In general we see that the spatial

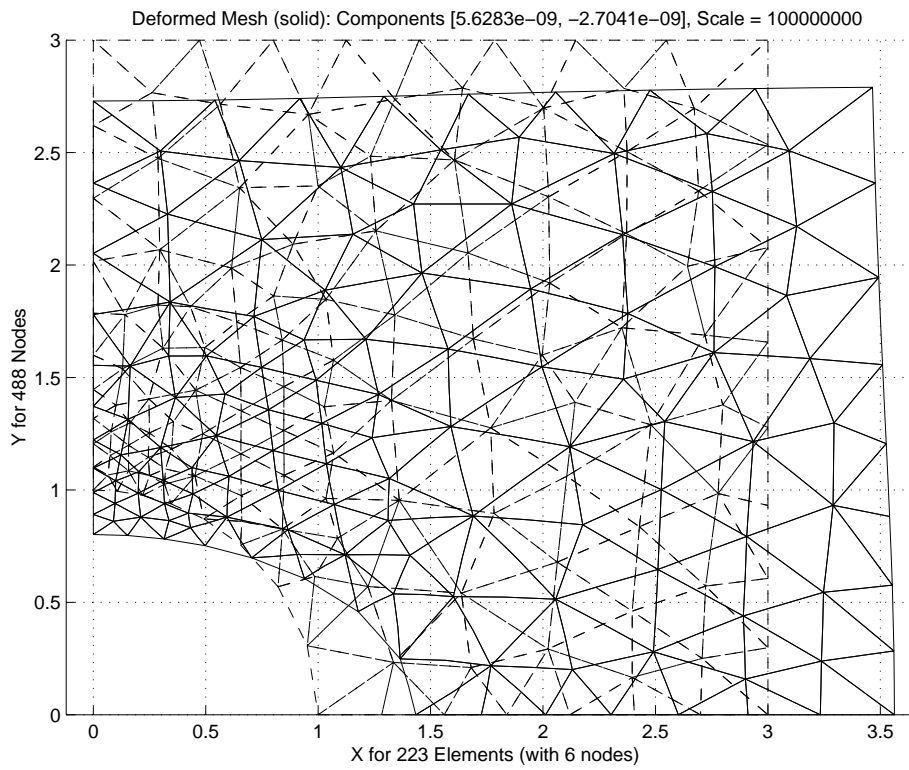


Figure 11.9.3 Original and scaled deformed mesh

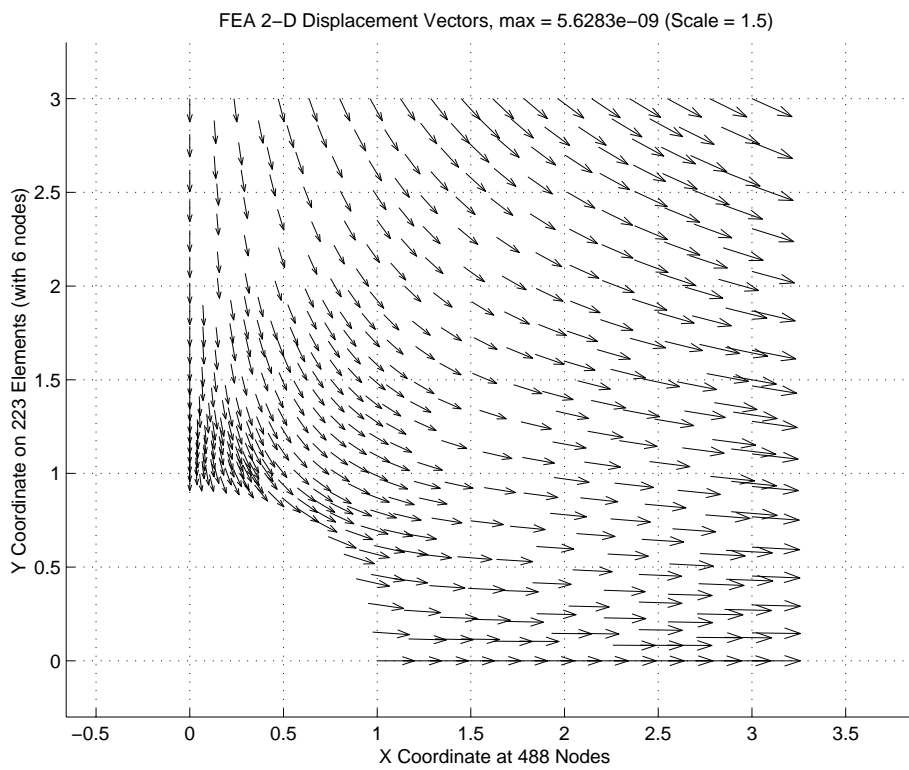


Figure 11.9.4 Nodal displacement vectors

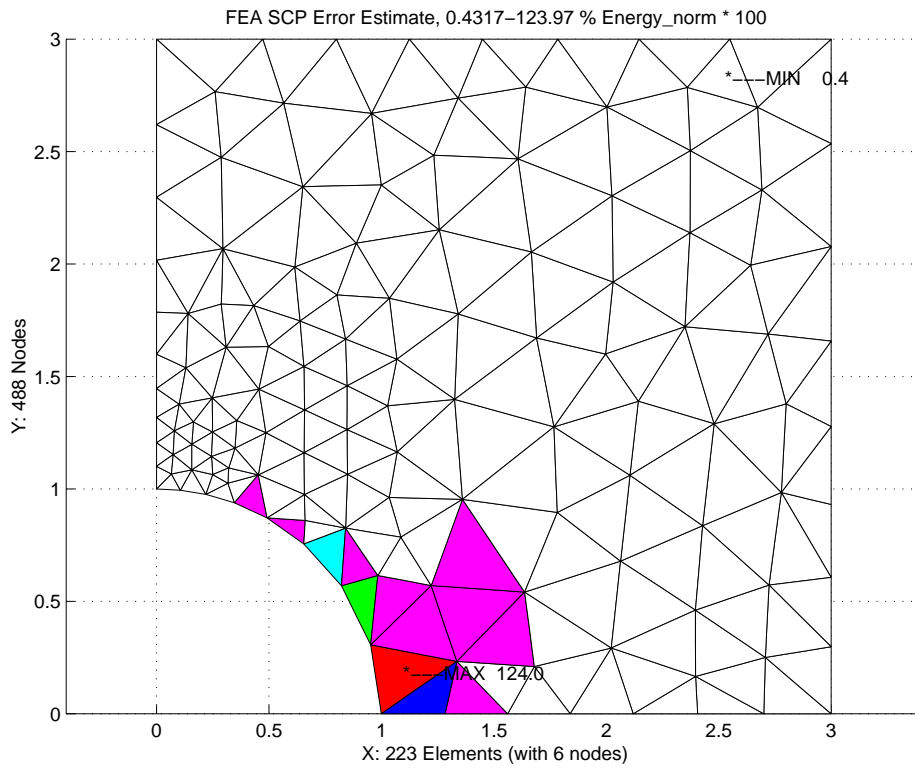


Figure 11.9.5 Largest element error in initial mesh

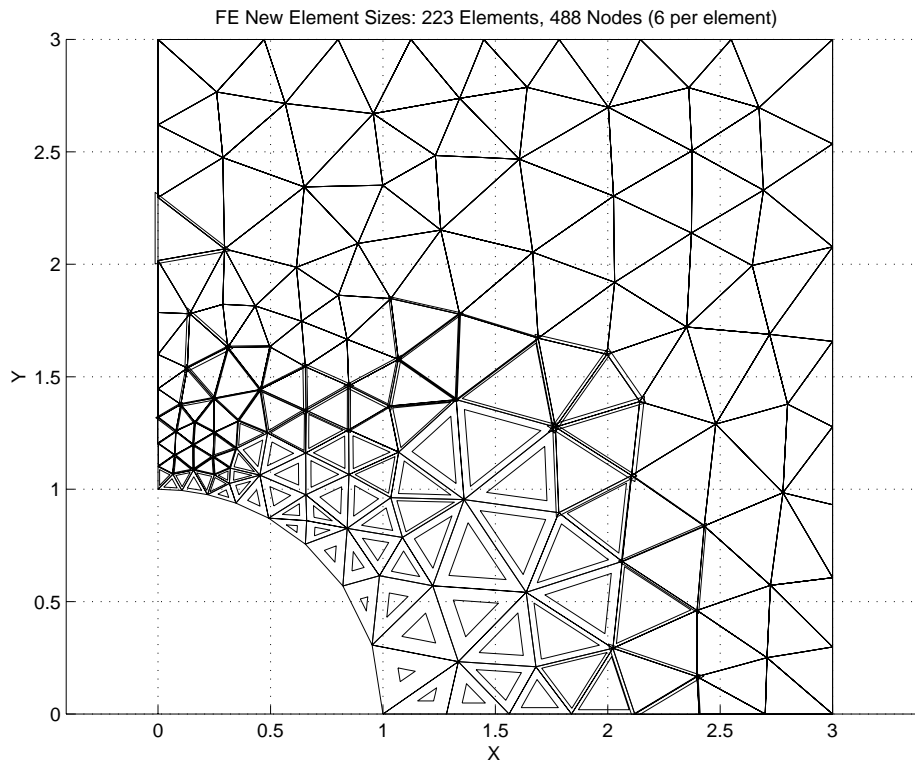


Figure 11.9.6 First recommended mesh refinement

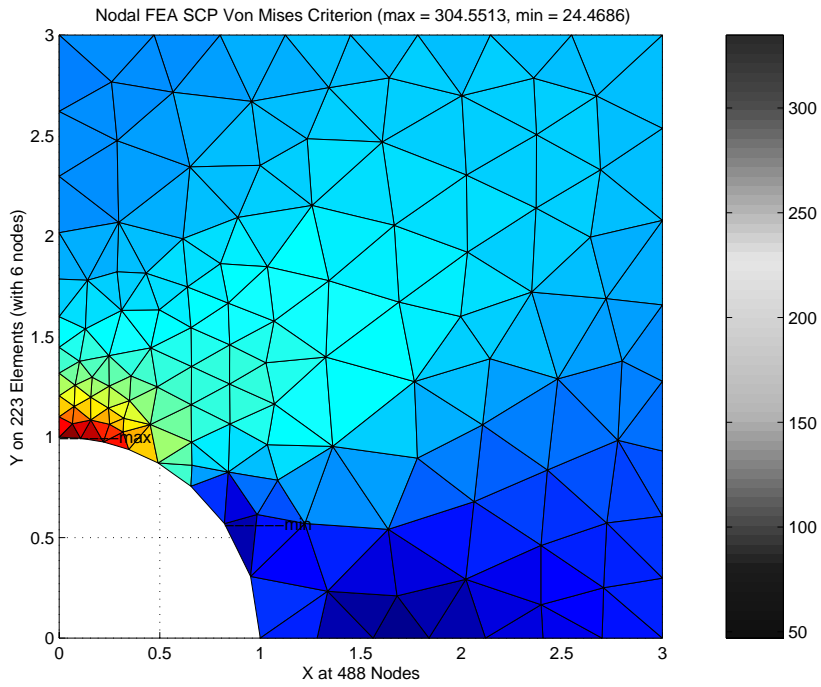


Figure 11.9.7 Shaded Von Mises failure criterion

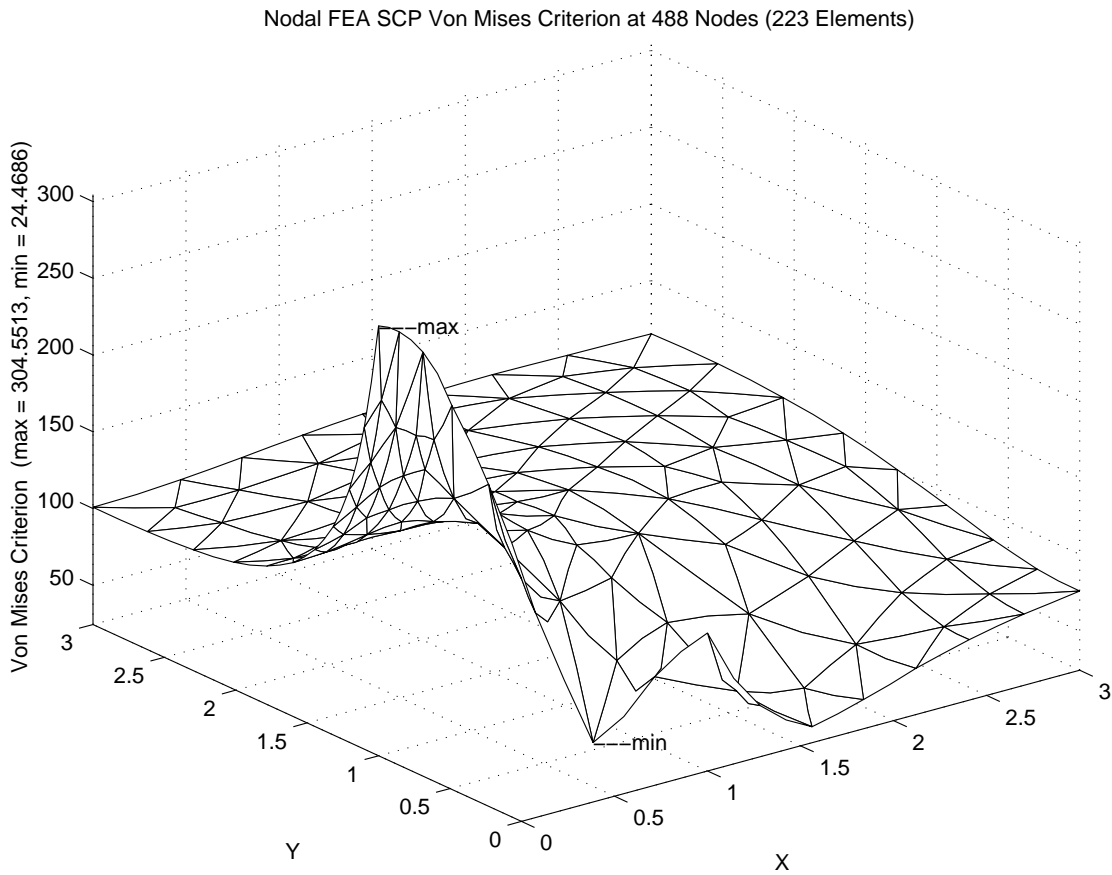


Figure 11.9.8 Von Mises failure criterion surface

distributions are similar but the amplitudes of the finite element stresses are a little high. Hopefully the mesh refinements directed by the error estimates will lead to an efficient and accurate solution for all the stresses.

The first mesh revision shown in Fig. 11.9.6, leads to the second mesh, in Fig. 11.9.15, and that in turn lead to a new distribution of error estimates as seen in Fig. 11.9.16. In the latter figure we see that the error is becoming more dispersed over the mesh. The system energy norm error has been cut almost in half to 0.0669 %. Likewise the maximum error in a single element has been cut by about 40 % and its location moved from near the hole edge to the largest interior element. Repeating this process gives the third mesh and its recommended mesh refinement given in Fig. 11.9.17. The refinement in element sizes is based on the error distribution in Fig. 11.9.16. There the lowest errors (in the energy norm) occur near the edge of the hole, where we expect the maximum stresses and failure criterion. In the second mesh the maximum single element error has been further reduced and is localized near the plate center where the stresses are low and slowly changing. The system wide error has been reduced to 0.051 %. The fourth, and final, mesh is shown in Fig. 11.9.18 along with the final error estimates. The corresponding finite element and exact Von Mises values are given in Figs. 11.9.19 and 20. Clearly, one could continue this process but the resulting figures are too dense to show here. The third set of maximum single element error estimates is about a factor of 150 times smaller than the maximum element error in the first mesh). The rate of system error reduction is given in Fig. 11.9.21, and that for the maximum error in any element is in Fig. 11.9.22.

11.10

Exercises

1. Implement the isotropic \mathbf{E}^e matrix for the plane strain assumption. Assume the stress components are in the XX, YY, XY order.
2. Implement the isotropic \mathbf{E}^e matrix for the general solid. Assume the stress components are in the XX, YY, XY, ZZ, XZ, YZ order.
3. Give an implementation of the \mathbf{B}^e matrix for a general solid. Assume the strain components are in the XX, YY, XY, ZZ, XZ, YZ order.
4. Review the scalar implementation for the square matrix for the T3 element, given in Fig. 10.4.1, and extend it to the case of a two-dimensional vector displacement model (that will double the matrix size).
5. Implement a general evaluation of the elasticity \mathbf{E}^e matrix that will handle all the 1-, 2-, and 3-dimensional cases. Solution:
6. Implement a general evaluation of the elasticity \mathbf{B}^e matrix that will handle all the 1-, 2-, and 3-dimensional cases.

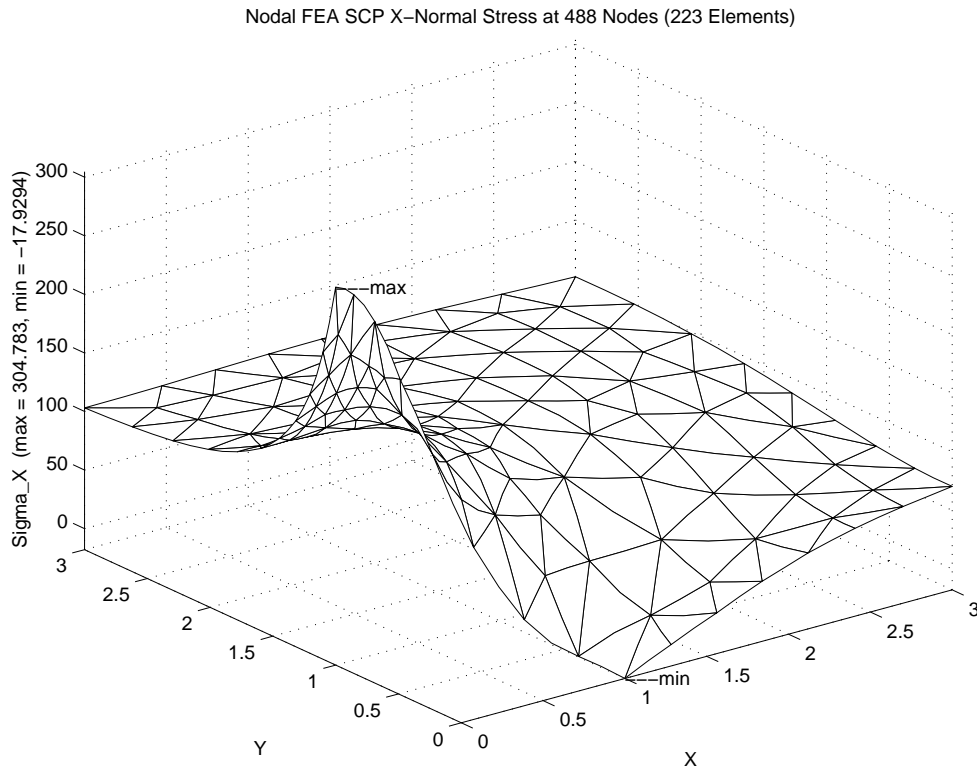


Figure 11.9.9 Initial estimate of x-normal stress

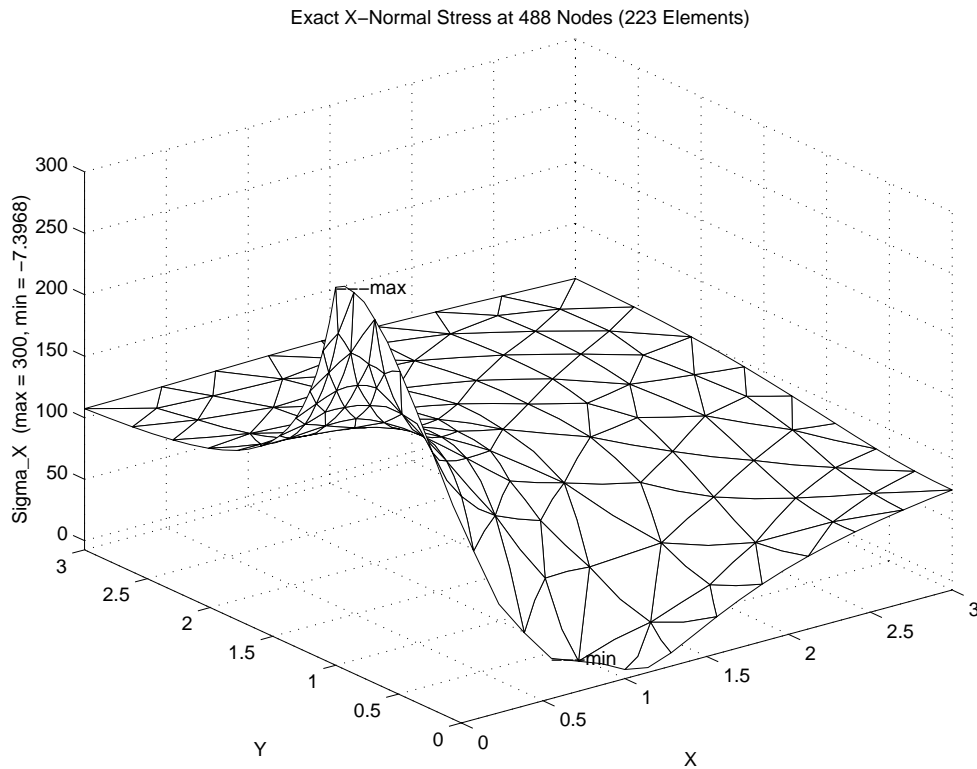
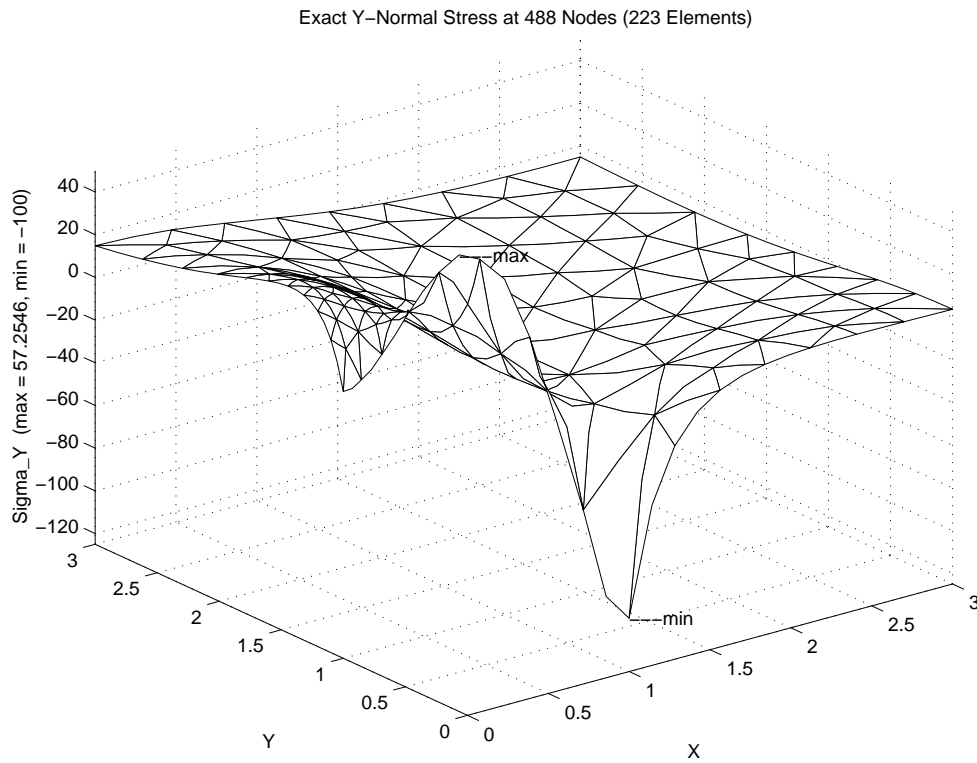
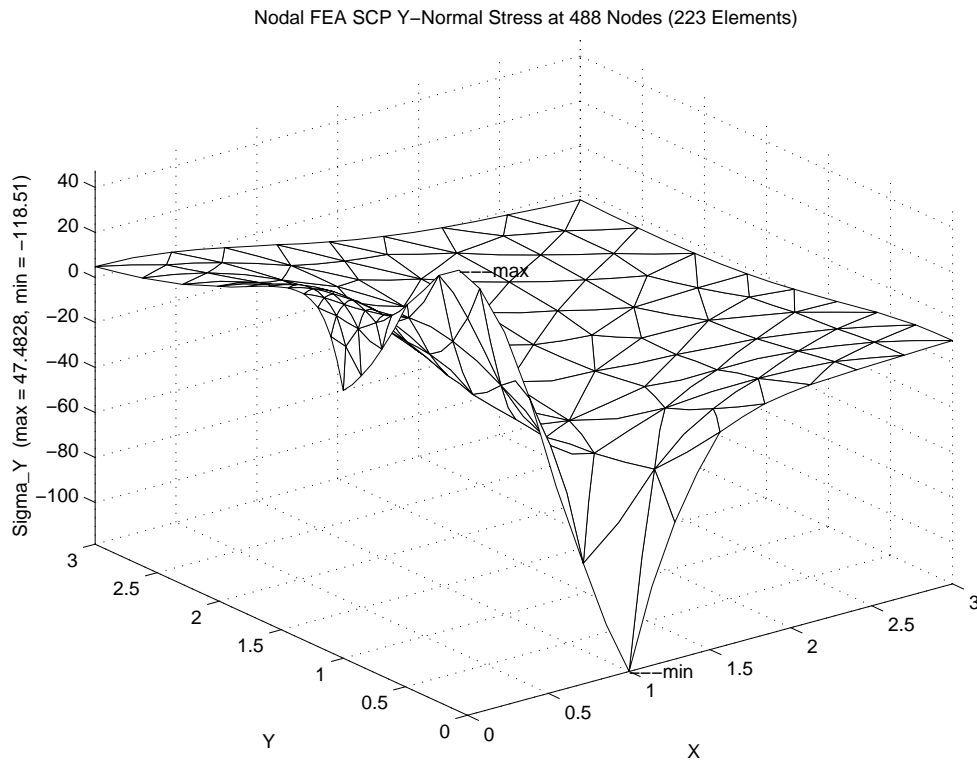


Figure 11.9.10 Exact distribution of x-normal stress



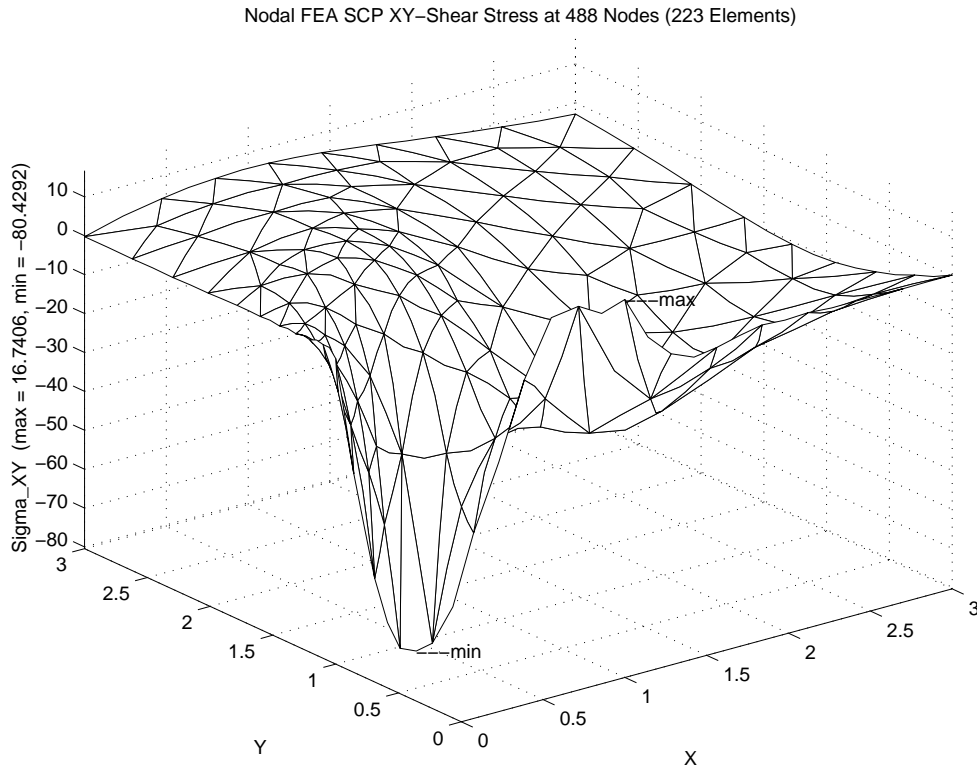


Figure 11.9.13 Initial estimate of xy-shear stress

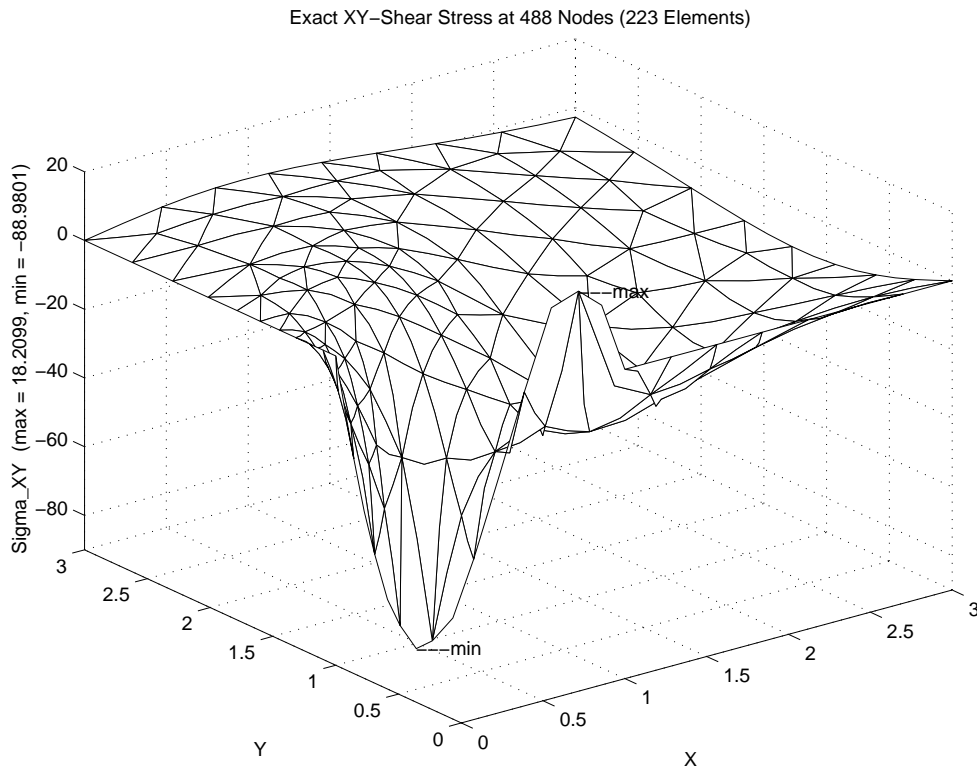


Figure 11.9.14 Exact distribution of xy-shear stress

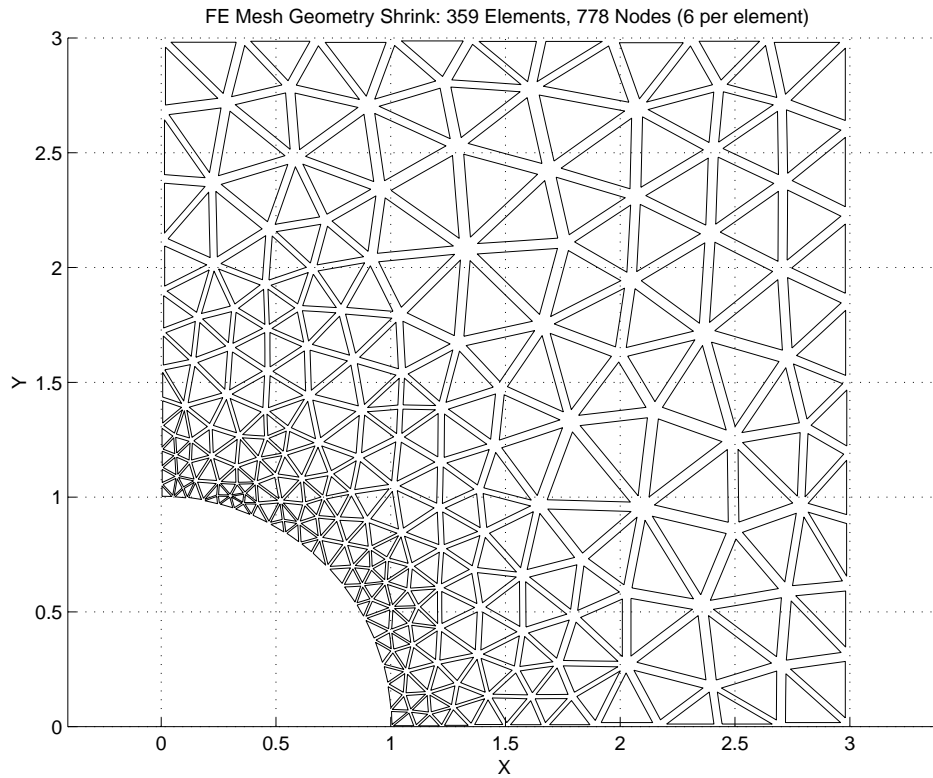


Figure 11.9.15 Second mesh generation (exploded)

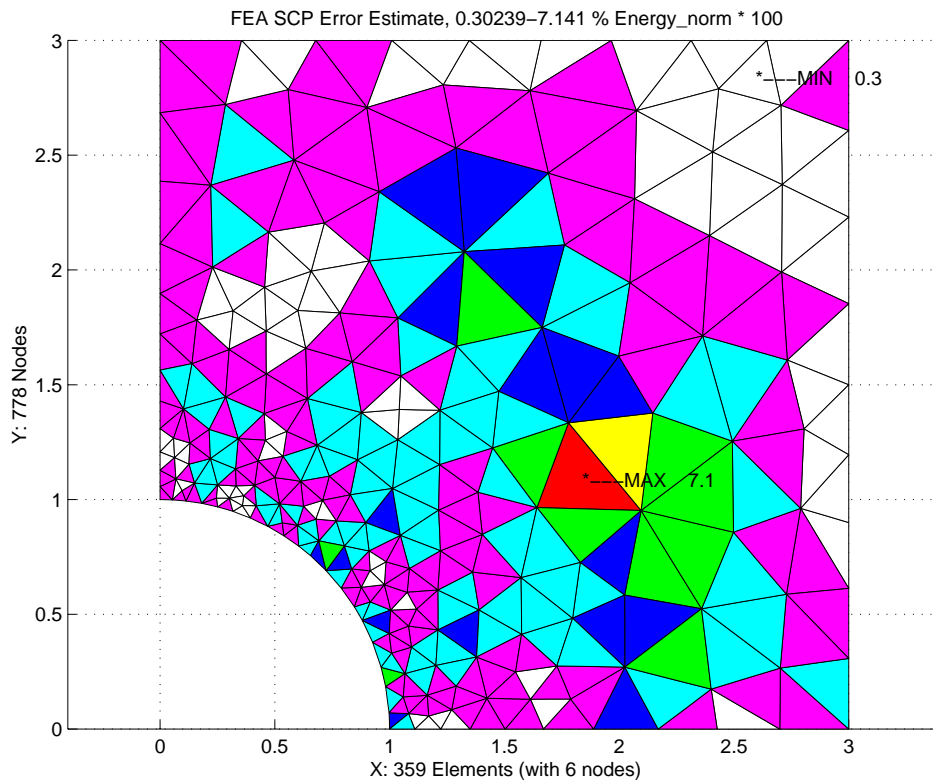


Figure 11.9.16 Second error estimate distribution

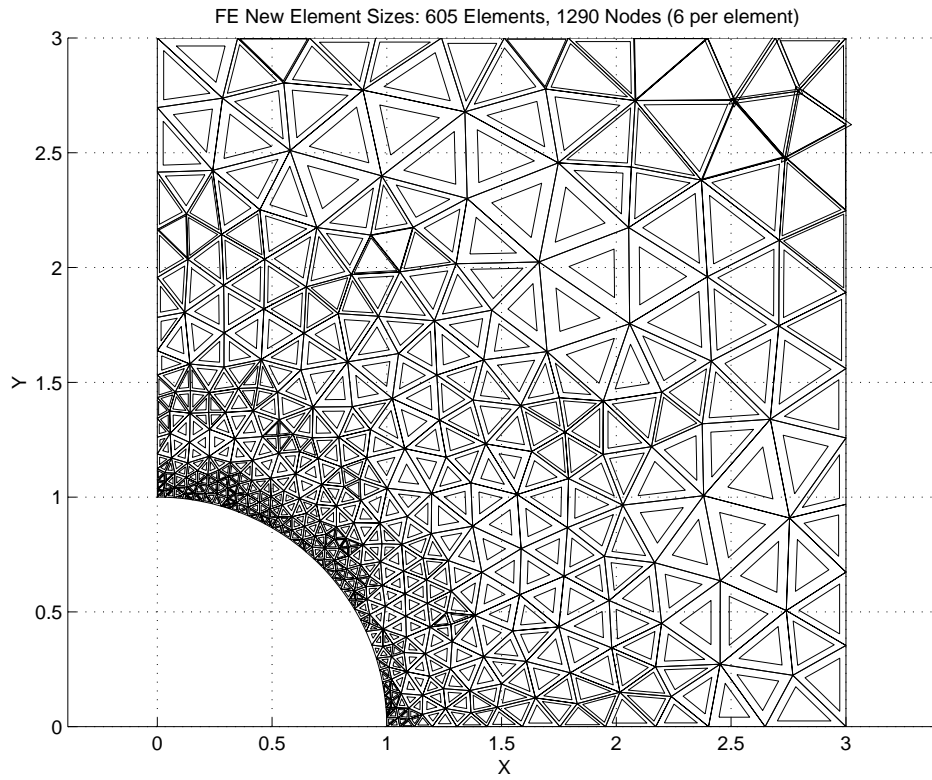


Figure 11.9.17 Third mesh generation and its suggested revision

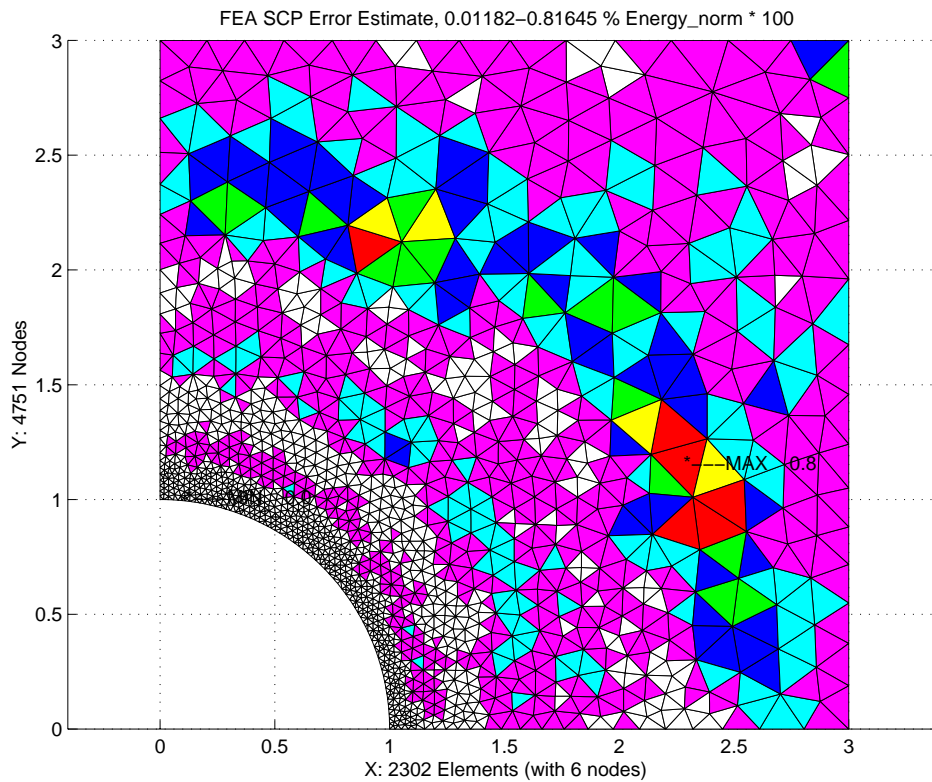


Figure 11.9.18 Fourth mesh and resulting final error estimates

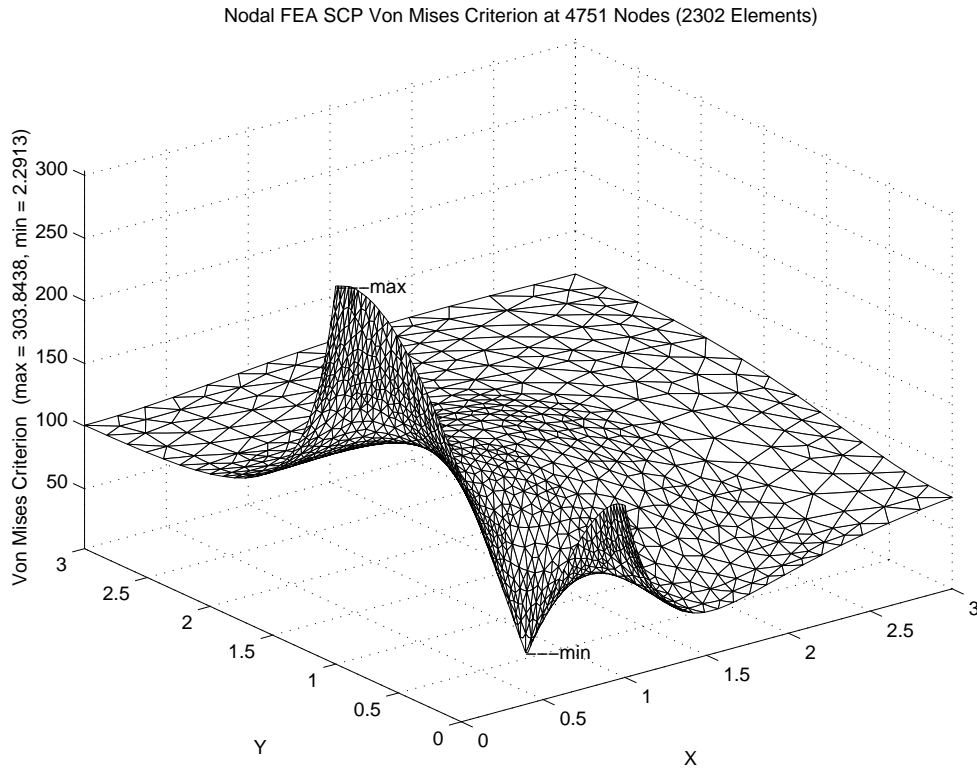


Figure 11.9.19 Final Von Mises failure criterion

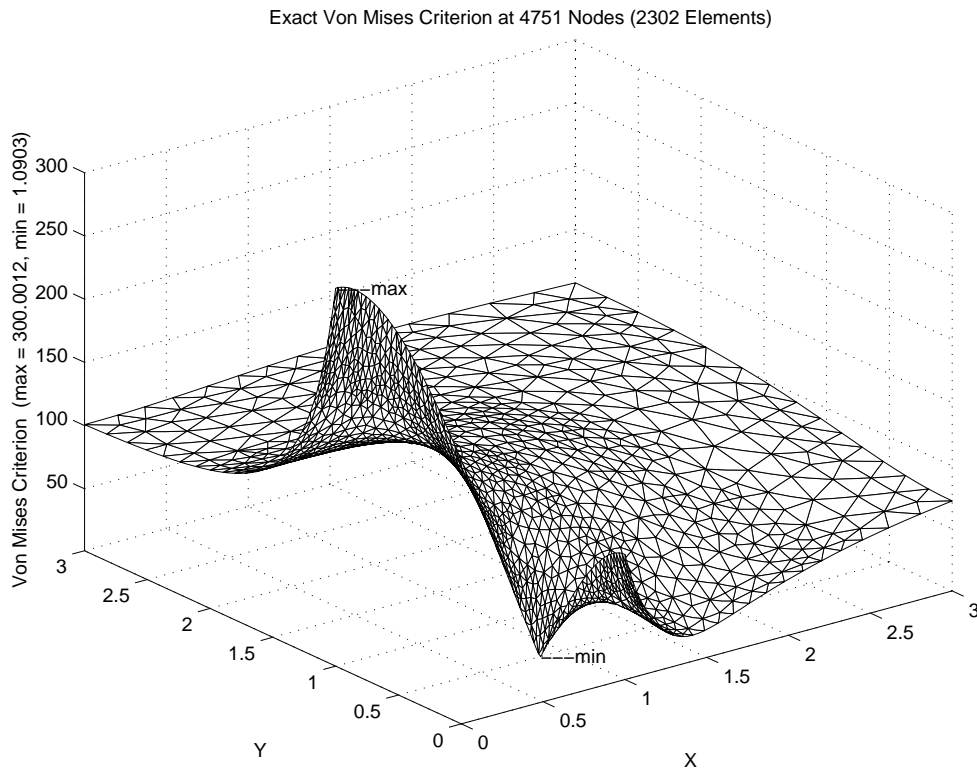


Figure 11.9.20 Exact final Von Mises failure criterion

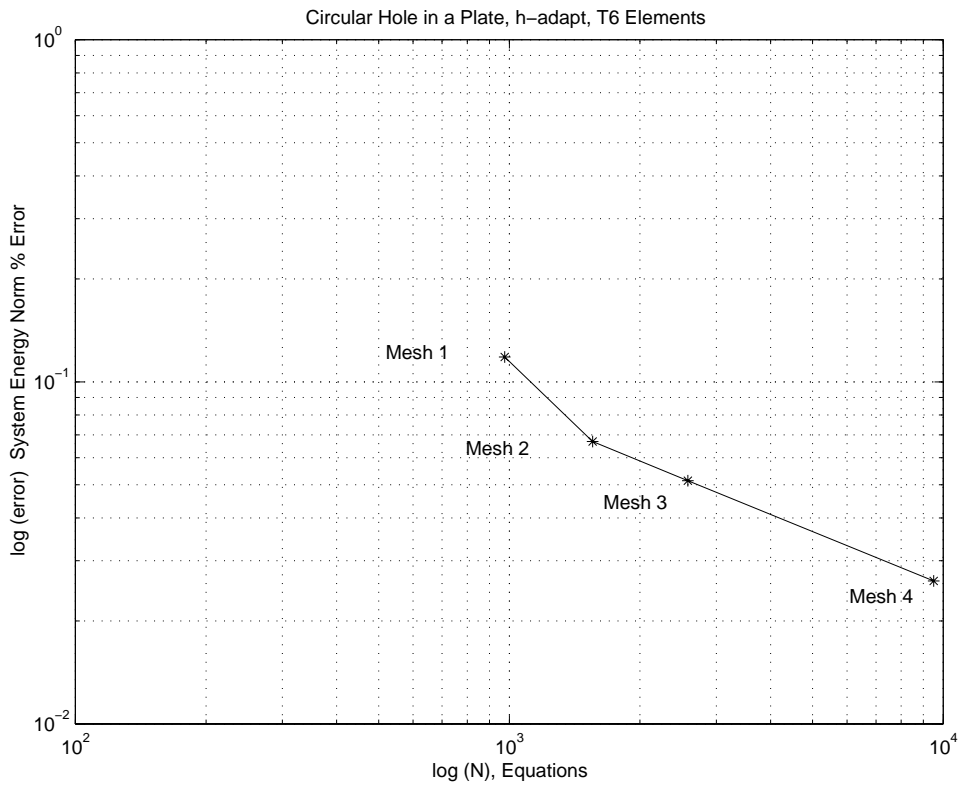


Figure 11.9.21 System energy norm error versus number of equations solved

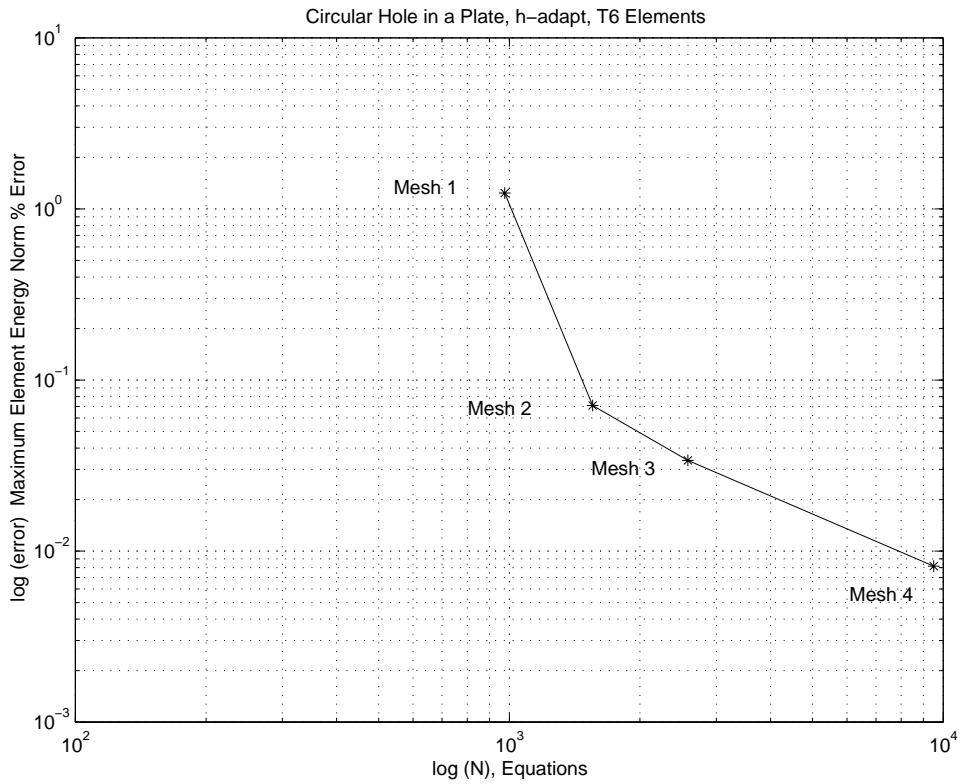


Figure 11.9.22 Maximum element error versus number of equations

11.11 References

- [1] Bathe, K.J., *Finite Element Procedures*, Englewood Cliffs: Prentice-Hall (1996).
- [2] Becker, E.B., Carey, G.F., and Oden, J.T., *Finite Elements – An Introduction*, Englewood Cliffs: Prentice-Hall (1981).
- [3] Belytschko, T., Liu, W.K., and Moran, B., *Nonlinear Finite Elements for Continua and Structures*, New York: John Wiley (2000).
- [4] Cook, R.D., Malkus, D.S., Plesha, N.E., and Witt, R.J., *Concepts and Applications of Finite Element Analysis*, New York: John Wiley (2002).
- [5] Desai, C.S. and Abel, J.F., *Introduction to the Finite Element Method*, New York: Van Nostrand-Reinhold (1972).
- [6] Gallagher, R.H., *Finite Element Analysis Fundamentals*, Englewood Cliffs: Prentice-Hall (1975).
- [7] Gupta, K.K and Meek, J.L., *Finite Element Multidisciplinary Analysis*, Reston: AIAA (2000).
- [8] Hinton, E. and Owen, D.R.J., *Finite Element Programming*, London: Academic Press (1977).
- [9] Hughes, T.J.R., *The Finite Element Method*, Englewood Cliffs: Prentice-Hall (1987).
- [10] Meek, J.L., *Computer methods in structural analysis*, London: E & FN Spon (1991).
- [11] Segerlind, L.J., *Applied Finite Element Analysis*, New York: John Wiley (1987).
- [12] Smith, I.M. and Griffiths, D.V., *Programming the Finite Element Method*, 3rd Edition, Chichester: John Wiley (1998).
- [13] Szabo, B. and Babuska, I., *Finite Element Analysis*, New York: John Wiley (1991).
- [14] Weaver, W.F., Jr. and Johnston, P.R., *Finite Elements for Structural Analysis*, Englewood Cliffs: Prentice-Hall (1984).
- [15] Zienkiewicz, O.C. and Taylor, R.L., *The Finite Element Method*, 5th Edition, London: Arnold (2000).