Steel Plate Analysis by Finite Elements

NORMAN L. FIRKINS

THE FINITE ELEMENT method is becoming a popular as well as practical way of analyzing plates, shells, and related structures. The technique of modeling* an elastic structure into an assemblage of discrete bars or sections and interconnecting these elements by a finite number of joints or nodal points is not new.[1] The growth and development of the finite element method has been greatly enhanced by the high speed digital computer and matrix methods in structural mechanics.

Extended finite element formulation is being carried on to increase the potential of this method in the areas of plasticity, vibration, and three-dimensional stress applications.

Although continuous investigation is being undertaken, it might be in order to review the basic finite element formulation and provide a simplified method of evaluation for the practicing engineer.

ELEMENT CHARACTERISTICS

In order to analyze and express the element properties in matrix form, certain characteristics[2] of an elastic element must be assumed, i.e., the material making up the element must be isotropic and homogeneous; Hooke's Law and Saint Venant compatibility equations apply under small deflection theory.

GENERAL EQUATION FORMULATIONS

With these characteristics accounted for, a set of simultaneous equations may be used to describe the linear relationship that exists between the nodes in Fig. 1 under rigid body displacements:

$$\begin{aligned} u_i &= a_1 + a_2 x_i + a_3 y_i \\ u_j &= a_1 + a_2 x_j + a_3 y_j \\ u_n &= a_1 + a_2 x_n + a_3 y_n \end{aligned} \quad (1)$$

* *Providing the ability to reduce, synthesize, and properly represent the structural configuration.*

Norman L. Firkins is a Civil Engineer with Henningson, Durham & Richardson, Consulting Engineers and Architects, Omaha, Nebraska.

Using the coordinates of the assumed element configuration in Fig. 1, the area of the element may be expressed in determinate form as

$$A = \frac{1}{2}\begin{vmatrix} 1 & x_i\, y_i \\ 1 & x_j\, y_j \\ 1 & x_n\, y_n \end{vmatrix} \quad (2)$$

It can be shown[3] that through certain relationships, substitutions, and elimination of a_1, a_2, and a_3, an area-coordinate matrix $[B]$ will relate element strain and nodal displacement, and may be expressed thus:

$$\{\epsilon\} = [B]\{\Delta\} \quad (3)$$

In matrix form,

$$[B] = \frac{1}{2A}\begin{bmatrix} c_i & o & c_j & o & c_n & o \\ o & d_i & o & d_j & o & d_n \\ d_i & c_i & d_j & c_j & d_n & c_n \end{bmatrix} \quad (4)$$

and $c_i = y_j - y_n$, $c_j = y_n - y_i$, $c_n = y_i - y_j$

and $d_i = x_n - x_j$, $d_j = x_i - x_n$, $d_n = x_j - x_i$

In this discussion, it is assumed that the elastic element will obey Hooke's Law and that the following stress-strain relationship[4] exists for plane stress as well as plane strain. Using the following formulations:

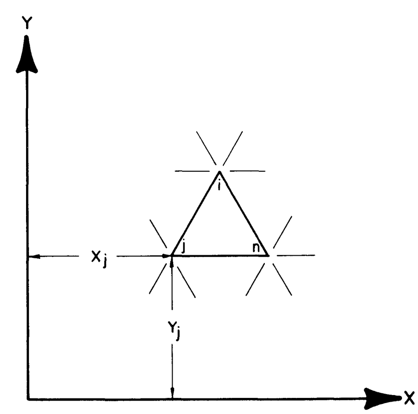

Figure 1

$$\epsilon_x = \frac{1}{E}(\sigma_x - \nu\,\sigma_y)$$

$$\epsilon_y = \frac{1}{E}(\sigma_y - \nu\,\sigma_x) \qquad (5)$$

$$\gamma_{xy} = \frac{2\,(1+\nu)}{E}\,\tau_{xy}$$

Expressed in matrix form:

$$\{\sigma\} = [D]\{\epsilon\} \qquad (6)$$

Plane stress and plane strain may be determined by using the appropriate matrix $[D]$.

For plane stress,

$$[D] = \frac{E}{(1-\nu^2)}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \dfrac{(1-\nu)}{2} \end{bmatrix} \qquad (7)$$

and for plane strain,

$$[D] = \frac{E}{(1-\nu^2)^2 - [\nu(1+\nu)]^2} \times$$

$$\begin{bmatrix} (1-\nu^2) & \nu(1+\nu) & 0 \\ \nu(1+\nu) & (1-\nu^2) & 0 \\ 0 & 0 & \dfrac{(1-\nu^2)^2 - [\nu(1+\nu)]^2}{2\,(1+\nu)} \end{bmatrix} \qquad (8)$$

Using the principal of virtual work and equating internal and external work equations, a generalized nodal point stiffness matrix $[k]$ can be defined for an element in the matrix form:

$$[k] = \int_{Vol} [B]^T[D][B]dVol \qquad (9)$$

or simply for an element:

$$[k] = [B]^T[D][B]tA \qquad (10)$$

or in final matrix expansion, adaptable for computer processing, thus:

$$[k]_e = \frac{1}{2A}\begin{bmatrix} c_i & o & d_i \\ o & d_i & c_i \\ c_j & o & d_j \\ o & d_j & c_j \\ c_n & o & d_n \\ o & d_n & c_n \end{bmatrix} \times \frac{E}{(1-\nu^2)}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \dfrac{(1-\nu)}{2} \end{bmatrix} \times$$

$$\frac{1}{2A}\begin{bmatrix} c_i & o & c_j & o & c_n & o \\ o & d_i & o & d_j & o & d_n \\ d_i & c_i & d_j & c_j & d_n & c_n \end{bmatrix} tA$$

$$= \begin{bmatrix} K_{ii} & K_{ij} & K_{in} \\ K_{ji} & K_{jj} & K_{jn} \\ K_{ni} & K_{nj} & K_{nn} \end{bmatrix} \qquad (11)$$

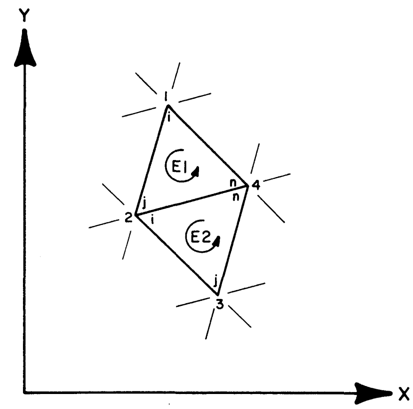

Figure 2

The normal counter-clockwise assignment of i, j, and n to the nodes of assembled elements is shown in Fig. 2.

To develop the total stiffness of a particular structural system, the elemental stiffnesses must then be assembled into a total stiffness matrix to express the compatibility of adjacent triangles and their respective influence on nodal point displacements. This total stiffness assemblage takes the form shown in matrix (12), See Fig. 3.

$$[K]_{TOTAL} = \begin{bmatrix} K^{E1}_{ii} & K^{E1}_{ij} & 0 & K^{E1}_{in} \\ K^{E1}_{ji} & K^{E1}_{jj} + K^{E2}_{ii} & K^{E2}_{ij} & K^{E1}_{jn} + K^{E2}_{in} \\ 0 & K^{E2}_{ji} & K^{E2}_{jj} & K^{E2}_{jn} \\ K^{E1}_{ni} & K^{E1}_{nj} + K^{E2}_{ni} & K^{E2}_{nj} & K^{E1}_{nn} + K^{E2}_{nn} \end{bmatrix} \qquad (12)$$

Figure 3

It will be noted in Eq. (11) that each K_{rc} is a 2 by 2 matrix and that each row (r) and column (c) thereof must contribute properly to the overall makeup of the final stiffness matrix. This relationship must exist[5] with respect to off diagonal as well as diagonal terms.

With the final stiffness matrix generated, the desired stiffness equation can be written in matrix form:

$$\{F\} = [K]\{\Delta\} \qquad (13)$$

Inverting the appropriate $[K]$ matrix, the respective nodal point displacements may be found:

$$\{\Delta\} = [K]^{-1}\{F\} \qquad (14)$$

Using these nodal displacements, the following may be determined:

1. Element stresses:

$$\{\sigma\} = [D][B]\{\Delta\} \quad (15)$$

2. Element strains:

$$\{\epsilon\} = [B]\{\Delta\} \quad (16)$$

3. Angle of maximum and minimum normal stress:

$$\tan 2\theta = \frac{2\,\tau_{xy}}{\sigma_x - \sigma_y} \quad (17)$$

4. Maximum and minimum principal stresses:

$$\sigma_{\max} = \sigma_1 = \frac{\sigma_x + \sigma_y}{2} + \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \tau_{xy}^2} \quad (18)$$

$$\sigma_{\min} = \sigma_2 = \frac{\sigma_x + \sigma_y}{2} - \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \tau_{xy}^2} \quad (19)$$

ELEMENT REPRESENTATION

In triangular finite element representation, it has been established that for proper accuracy the element configuration should resemble or closely approximate an equilateral triangle. Although quadrilateral elements are currently being used in analysis with excellent results, triangular division seems to be desirable.

STEEL PLATE SAMPLE PROBLEMS

Example 1—Consider a typical steel plate shown in Fig. 4.

It is required to compute the displacements and internal stresses and strains on the loaded plate according to the finite element method. The structure modeled into finite elements is shown in Fig. 5.

Using matrices (2), (4), (7), compute the areas, $[B]^T$, $[D]$, $[B]$, and $[k]$ for elements $\mathbf{E}_1$ and $\mathbf{E}_2$, as in matrices (20) and (21), Fig. 6.

With these elemental stiffness matrices, generate the total stiffness matrix (22) (see Fig. 7).

The total stiffness matrix (22) may then be blocked or deleted, according to boundary conditions, to provide the appropriate stiffness matrix $[K]$; a normal inversion computer program will then provide the inverse of $[K]$:

$$[K] = \begin{bmatrix} 0.2799\text{ E }07 & -0.9999\text{ E }06 & -0.3999\text{ E }06 & 0.3999\text{ E }06 \\ -0.9999\text{ E }06 & 0.1966\text{ E }07 & 0.5999\text{ E }06 & -0.1066\text{ E }07 \\ -0.3999\text{ E }06 & 0.5999\text{ E }06 & 0.2799\text{ E }07 & 0.0 \\ 0.3999\text{ E }06 & -0.1066\text{ E }07 & 0.0 & 0.1966\text{ E }07 \end{bmatrix} \quad (23)$$

$$[K]^{-1} = \begin{bmatrix} 0.4394\text{ E}-06 & 0.2433\text{ E}-06 & 0.1064\text{ E}-07 & 0.4257\text{ E}-07 \\ 0.2433\text{ E}-06 & 0.9286\text{ E}-06 & -0.1642\text{ E}-06 & 0.4541\text{ E}-06 \\ 0.1064\text{ E}-07 & -0.1642\text{ E}-06 & 0.3938\text{ E}-06 & -0.9124\text{ E}-07 \\ 0.4257\text{ E}-07 & 0.4541\text{ E}-06 & -0.9124\text{ E}-07 & 0.7461\text{ E}-06 \end{bmatrix} \quad (24)$$

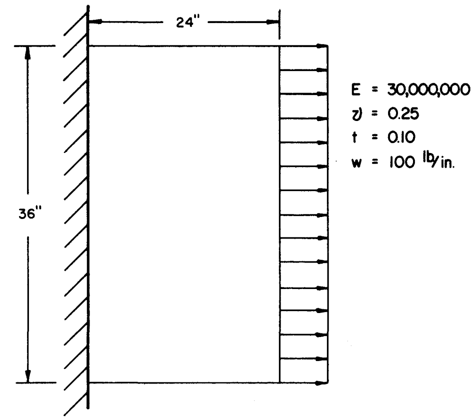

Figure 4

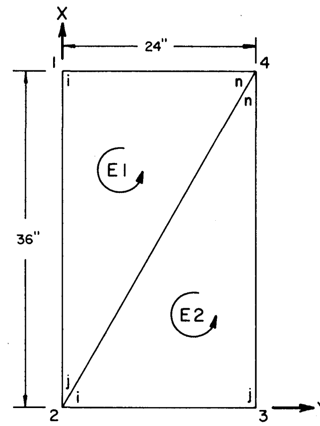

Figure 5

With the distributed load, w, a load vector $\{F\}$ i assembled to express the required nodal forces. Usin this load vector and (24), the displacements of nodɛ points **3** and **4** are found to be

$$\begin{aligned} \{\Delta\} &= [K]^{-1}\{F\} \\ x_3 &= 0.0008102 \text{ in.} \\ y_3 &= 0.0001423 \text{ in.} \\ x_4 &= 0.0007281 \text{ in.} \\ y_4 &= -0.0000875 \text{ in.} \end{aligned} \quad (2\!\!\!\;$$

where

$$F_{x3} = F_{x4} = 1800.0 \text{ lbs}$$
$$F_{y3} = F_{y4} = 0.0 \text{ lbs}$$

Normally, for triangular elements, it is assumed tha the computed stresses are to be applied at the centroi of the element under consideration.

Using the computed nodal displacements (25), a element, such as $\mathbf{E}_2$, can be analyzed for point stresse and strains. This is demonstrated as follows:

FOR ELEMENT E1

$$A_1 = \text{.432000E 03}$$

$$[B]_1^T = \begin{bmatrix} \text{-.416666E-01} & \text{.000000E-99} & \text{.277777E-01} \\ \text{.000000E-99} & \text{.277777E-01} & \text{-.416666E-01} \\ \text{.000000E-99} & \text{.000000E-99} & \text{-.277777E-01} \\ \text{.000000E-99} & \text{-.277777E-01} & \text{.000000E-99} \\ \text{.416666E-01} & \text{.000000E-99} & \text{.000000E-99} \\ \text{.000000E-99} & \text{.000000E-99} & \text{.416666E-01} \end{bmatrix}$$

$$[D]_1 = \begin{bmatrix} \text{.320000E 08} & \text{.800000E 07} & \text{.000000E-99} \\ \text{.800000E 07} & \text{.320000E 08} & \text{.000000E-99} \\ \text{.000000E-99} & \text{.000000E-99} & \text{.120000E 08} \end{bmatrix}$$

$$[B]_1 = \begin{bmatrix} \text{-.416666E-01} & \text{.000000E-99} & \text{.000000E-99} & \text{.000000E-99} & \text{.416666E-01} & \text{.000000E-99} \\ \text{.000000E-99} & \text{.277777E-01} & \text{.000000E-99} & \text{-.277777E-01} & \text{.000000E-99} & \text{.000000E-99} \\ \text{.277777E-01} & \text{-.416666E-01} & \text{-.277777E-01} & \text{.000000E-99} & \text{.000000E-99} & \text{.416666E-01} \end{bmatrix}$$

$$[k]_1 = \begin{bmatrix} \text{.279999E 07} & \text{-.999999E 06} & \text{-.399999E 06} & \text{.399999E 06} & \text{-.239999E 07} & \text{.599999E 06} \\ \text{-.999999E 06} & \text{.196666E 07} & \text{.599999E 06} & \text{-.106666E 07} & \text{.399999E 06} & \text{-.899999E 06} \\ \text{-.399999E 06} & \text{.599999E 06} & \text{.399999E 06} & \text{-.000000E-99} & \text{-.000000E-99} & \text{-.599999E 06} \\ \text{.399999E 06} & \text{-.106666E 07} & \text{-.000000E-99} & \text{.106666E 07} & \text{-.399999E 06} & \text{.000000E-99} \\ \text{-.239999E 07} & \text{.399999E 06} & \text{-.000000E-99} & \text{-.399999E 06} & \text{.239999E 07} & \text{.000000E-99} \\ \text{.599999E 06} & \text{-.899999E 06} & \text{-.599999E 06} & \text{.000000E-99} & \text{.000000E-99} & \text{.899999E 06} \end{bmatrix} \quad (20)$$

FOR ELEMENT E2

$$A_2 = \text{.432000E 03}$$

$$[B]_2^T = \begin{bmatrix} \text{-.416666E-01} & \text{.000000E-99} & \text{.000000E-99} \\ \text{.000000E-99} & \text{.000000E-99} & \text{-.416666E-01} \\ \text{.416666E-01} & \text{.000000E-99} & \text{-.277777E-01} \\ \text{.000000E-99} & \text{-.277777E-01} & \text{.416666E-01} \\ \text{.000000E-99} & \text{.000000E-99} & \text{.277777E-01} \\ \text{.000000E-99} & \text{.277777E-01} & \text{.000000E-99} \end{bmatrix}$$

$$[D]_2 = \begin{bmatrix} \text{.320000E 08} & \text{.800000E 07} & \text{.000000E-99} \\ \text{.800000E 07} & \text{.320000E 08} & \text{.000000E-99} \\ \text{.000000E-99} & \text{.000000E-99} & \text{.120000E 08} \end{bmatrix}$$

$$[B]_2 = \begin{bmatrix} \text{-.416666E-01} & \text{.000000E-99} & \text{.416666E-01} & \text{.000000E-99} & \text{.000000E-99} & \text{.000000E-99} \\ \text{.000000E-99} & \text{.000000E-99} & \text{.000000E-99} & \text{-.277777E-01} & \text{.000000E-99} & \text{.277777E-01} \\ \text{.000000E-99} & \text{-.416666E-01} & \text{-.277777E-01} & \text{.416666E-01} & \text{.277777E-01} & \text{.000000E-99} \end{bmatrix}$$

$$[k]_2 = \begin{bmatrix} \text{.239999E 07} & \text{-.000000E-99} & \text{-.239999E 07} & \text{.399999E 06} & \text{.000000E-99} & \text{-.399999E 06} \\ \text{-.000000E-99} & \text{.899999E 06} & \text{.599999E 06} & \text{-.899999E 06} & \text{-.599999E 06} & \text{-.000000E-99} \\ \text{-.239999E 07} & \text{.599999E 06} & \text{.279999E 07} & \text{-.999999E 06} & \text{-.399999E 06} & \text{.399999E 06} \\ \text{.399999E 06} & \text{-.899999E 06} & \text{-.999999E 06} & \text{.196666E 07} & \text{.599999E 06} & \text{-.106666E 07} \\ \text{.000000E-99} & \text{-.599999E 06} & \text{-.399999E 06} & \text{.599999E 06} & \text{.399999E 06} & \text{.000000E-99} \\ \text{-.399999E 06} & \text{-.000000E-99} & \text{.399999E 06} & \text{-.106666E 07} & \text{.000000E-99} & \text{.106666E 07} \end{bmatrix} \quad (21)$$

Figure 6

$[K]_{TOTAL} =$

.279E 07	-.999E 06	-.399E 06	.399E 06	.000E-99	.000E-99	-.239E 07	.599E 06
-.999E 06	.196E 07	.599E 06	-.106E 07	.000E-99	.000E-99	.399E 06	-.899E 06
-.399E 06	.599E 06	.280E 07	-.000E-99	-.239E 07	.399E 06	.000E-99	-.100E 07
.399E 06	-.106E 07	.000E-99	.196E 07	.599E 06	-.899E 06	-.100E 07	.000E-99
.000E-99	.000E-99	-.239E 07	.599E 06	.279E 07	-.999E 06	-.399E 06	.399E 06
.000E-99	.000E-99	.399E 06	-.899E 06	-.999E 06	.196E 07	.599E 06	-.106E 07
-.239E 07	.399E 06	.000E-99	-.100E 07	-.399E 06	.599E 06	.280E 07	.000E-99
.599E 06	-.899E 06	-.100E 07	-.000E-99	.399E 06	-.106E 07	.000E-99	.196E 07

(2

Figure 7

For element $\mathbf{E}_2$:

Compute stresses:

$$\begin{aligned} \{\sigma\}_{E_2} &= [D][B]\{\Delta\} \\ \sigma_x &= 1029.198 \text{ psi} \\ \sigma_y &= 65.692 \text{ psi} \\ \tau_{xy} &= 43.795 \text{ psi} \end{aligned} \quad (26)$$

Compute strains:

$$\begin{aligned} \{\epsilon\}_{E_2} &= [B]\{\Delta\} \\ \epsilon_x &= 0.3375\,E - 04 \text{ in./in.} \\ \epsilon_y &= 0.6387\,E - 05 \text{ in./in.} \\ \gamma_{xy} &= 0.3650\,E - 05 \text{ in./in.} \end{aligned} \quad (27)$$

From (26), compute the maximum and minimum principal stresses:

$$\sigma_{\max} = \frac{\sigma_x + \sigma_y}{2} + \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \tau_{xy}^2}$$
$$= 1031.184 \text{ psi}$$

$$\sigma_{\min} = \frac{\sigma_x + \sigma_y}{2} - \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \tau_{xy}^2}$$
$$= 63.706 \text{ psi}$$

Model Accuracy—It will be noted in Eq. (25) that the symmetrically loaded plate did not produce symmetrical displacements. This situation can be eliminated when finer mesh size (more elements) are used and relatively constant triangle shape is maintained. To represent this, consider modeling Fig. 4 into a mesh size of 3, 4, and 108 elements as shown in Figs. 8a, 8b, and 8c, respectively. As indicated in Fig. 10, the nodal displacements of the fine mesh system are in close agreement with those shown in Fig. 9.

The previous two, three and four element configurations were analyzed on an IBM 1620 computer and the 108 element fine mesh plate on an IBM 360/65 with a 2314 storage device.

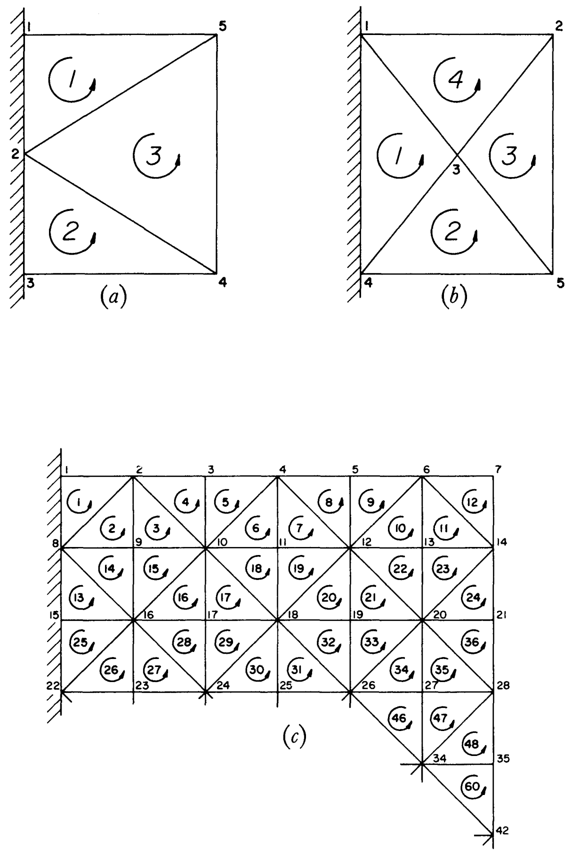

Figure 8

	MESH CONFIGURATION	
	3-ELEMENT	4-ELEMENT
DISPLACEMENTS	$x_5 = 0.0007698$	$x_2 = 0.0007777$

Figure 9

```
SAMPLE PROBLEM PLATE 108 -ELEMENTS
         NO OF PARTITIONS              10
         NO OF NODAL POINTS            70
         NO OF ELEMENTS               108
         NO OF BOUNDARY POINTS         10
         NO OF LOADING                  1
         NO OF MATERIAL TYPES           1
         TYPE OF ANALYSIS --PLANE STRESS
         NO OF DEGREES OF FREEDOM       2
         FORCE PER UNIT VOLUME  X           0.0
         FORCE PER UNIT VOLUME  Y           0.0
JOINT               COORDINATES
               X                   Y
  1           0.0               36.00000
  2           4.00000           36.00000
  3           8.00000           36.00000
  4          12.00000           36.00000
  5          16.00000           36.00000
  6          20.00000           36.00000
  7          24.00000           36.00000
  8           0.0               32.00000
  9           4.00000           32.00000
         ELEMENT    DESCRIPTION
           JOINT            MATERIAL    ANGLE OF
NO      CONNECTIVITY          TYPE      ORTHOTROPY      THICKNESS

  1      1    8    2            1       0.0                 0.10000
  2      8    9    2            1       0.0                 0.10000
  3      2    9   10            1       0.0                 0.10000
  4      2   10    3            1       0.0                 0.10000


NODE X-DISPLACEMENTS Y-DISPLACEMENTS
  1      0.00000000     -0.00000000
  2      0.00014710     -0.00006847
  3      0.00028421     -0.00010621
  4      0.00041395     -0.00012849
  5      0.00054372     -0.00014075
  6      0.00067482     -0.00015080
  7      0.00080776     -0.00015997
  8      0.00000000     -0.00000000
  9      0.00013270     -0.00004501


ELEMENT NUMBER    FIRST  NODE     SECOND NODE     THIRD  NODE      X  AND  Y   COORDINATES OF CENTROID
   X-STRESS         Y-STRESS        XYSTRESS        STRESS-1        STRESS-2      PRINCIPAL ANGLE

  1    1    8    2         1.33333302       34.66665649

  1176.82104492     294.20507812    -205.40046692   1222.27978516    248.74609375    -77.52044678

  2    8    9    2         2.66666603       33.33332825

  1014.56333008      77.72729492     -91.81152344   1023.57519531     68.81542969    -84.45571899

  3    2    9   10         5.33333302       33.33332825

  1003.74780273      74.99780273     -49.02273560   1006.32812500     72.41748047    -86.98681641

  4    2   10    3         6.66666603       34.66665649

  1035.98339844      30.56640625     -52.67968750   1038.73583984     27.81396484    -87.00878906

  5    3   10    4         9.33333302       34.66665649

  977
```

Figure 10

Example 2—Given the cantilever plate shown in Fig. 11a, determine the stress distribution on a stress block taken from element 32 under the given loading condition.

The plate configuration is first modeled into a working finite element mesh (Fig. 11b). Next, the proper boundary conditions and specific load vectors are applied. Using the appropriate matrix formulation, the state of stress at the centroid of element 32 is found to be the following:

$$\begin{aligned}\sigma_x &= 3175.4 \text{ psi}\\ \sigma_y &= -733.7 \text{ psi}\\ \tau_{xy} &= 161.2 \text{ psi}\end{aligned}$$

A unit square stress block is shown in Fig. 12a. The state of stress at that point (centroid of element) may be shown more clearly by expanding the point into a free body and by arbitrarily verifying that the vertical stress components are in equilibrium (Fig. 12b). Computations are as follows:

Compute maximum principal stress (positive):

$$\sigma_{\max} = \frac{\sigma_x + \sigma_y}{2} + \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \tau_{xy}{}^2}$$

$$= \frac{3175.4 - 733.7}{2} + \sqrt{\left(\frac{3175.4 + 733.7}{2}\right)^2 + (161.2)^2}$$

$$= 3182.8 \text{ psi}$$

Compute principal angle:

$$\begin{aligned}\tan 2\alpha &= \frac{2\tau_{xy}}{\sigma_x - \sigma_y}\\ &= \frac{2\ (161.2)}{3175.4 - (-733.7)}\\ \alpha &= 2.35°\\ \measuredangle &= 87.65° \text{ (from vertical)}\end{aligned}$$

Check vertical components for equilibrium ($\Sigma F_y = 0$):

Down:
161.1 (1) (1) = 161.0

Up:

733.76 (1) (0.042)	= 30.0
3182.13 (1) (1.00176) (0.041)	= 131.0
	161.0

Check

161.0 = 161.0 O.K.

It must be stated here that this mesh is usable but it should be realized that high stresses will occur near and at points of applied loads and boundary supports. These areas of high stress concentration may be easily examined by creating a finer mesh of elements near the points in question and reanalyzing the structure.

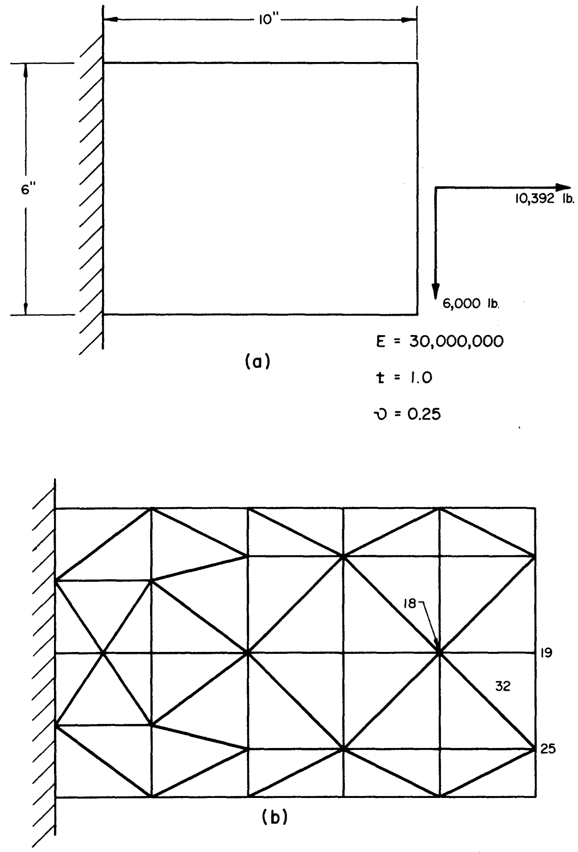

Figure 11

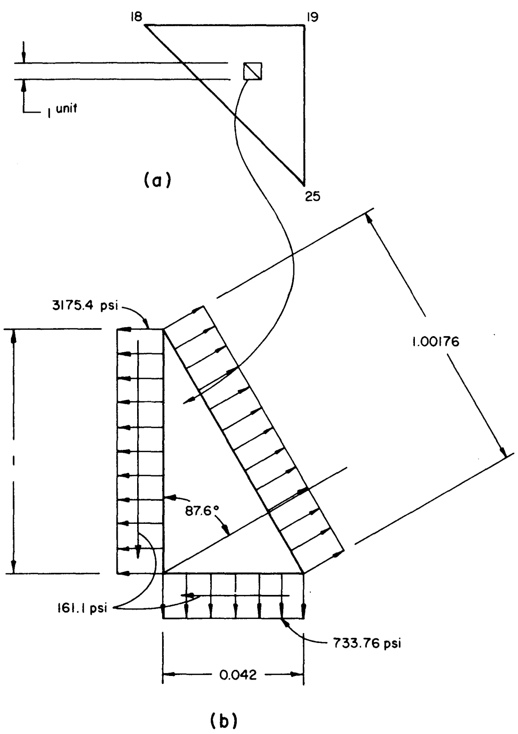

Figure 12

CONCLUSIONS

It has been found that many types of irregular plate configurations may be analyzed for areas of high stress concentration using the finite element approach. Basic element stiffness matrices and solutions to "small element" structures may be programmed on small to medium size digital computers. However, when trying to evaluate irregular structures using fine mesh subdivision, it becomes necessary to go to large capacity computers.

Basic investigation has shown that with regular mesh modeling, the element stiffness equations are identical to those of the finite difference approach and, in addition, provide a more systematic method for computer programming.

SYMBOL NOMENCLATURE

i, j, n = Element nodal points or joints
A = Area of element
$[B]$ = Area-coordinate matrix
$\{\epsilon\}$ = Strain vector
$\{\sigma\}$ = Stress vector
$\{\Delta\}$ = Displacement vector
$[D]$ = Elastic stress-strain matrix
$[k]_e$ = Element stiffness matrix
$[K]_{total}$ = Total stiffness matrix
$[K]$ = Final deleted stiffness matrix
$\{F\}$ = Nodal force vector
α = Principal angle
ν = Poisson ratio
τ = Shearing stress

REFERENCES

1. *Hrennikoff, A.* Solution of Elasticity by the Framework Method *ASCE Journal of Applied Mechanics, Vol. 63, pp. 169–175, 1941.*
2. *Chou, Pei Chi and Pagano, Nicholas J.* Elasticity Tensor, Dyadic and Engineering Approaches *D. Von Nostrand Company, Inc., 1967, Chapters 1 and 2.*
3. *Dodson, Ronald G.* Discussion of the Finite Element Method of Structural Analysis—Report—CE 516 *University of Nebraska at Omaha, May 16, 1968.*
4. *Zienkiewicz, O. C. and Cheung, Y. K.* The Finite Element Method in Structural and Continuum Mechanics *McGraw-Hill Publishing Company, 1967, Chapters 2, 3 and 14.*
5. *Rubinstein, Moshe F.* Matrix Computer Analysis of Structures *Prentice Hall, Inc., 1966, Chapter 12.*