MIT

ANALYSIS OF PLATE STRUCTURES BY A DUAL FINITE ELEMENT METHOD

by

Peter K. Ho

Supervised by

Jerome J. Connor, Jr.

DEPARTMENT OF CIVIL ENGINEERING

SCHOOL OF ENGINEERING
MASSACHUSETTS INSTITUTE OF TECHNOLOGY
Cambridge, Massachusetts 02139

Sponsored by
The National Aeronautics and Space Administration
Research Grant NGR-22-009-059

R69-58

September 1969

Research Report R69-58

Department of Civil Engineering

ANALYSIS OF PLATE STRUCTURES BY A DUAL FINITE ELEMENT METHOD

BY

PETER KEI-KIN HO

Supervised by
Jerome J. Connor, Jr.

September 1969

Sponsored by
The National Aeronautics and Space Administration
Research Grant NGR-22-009-059

School of Engineering
Massachusetts Institute of Technology
Cambridge, Massachusetts

ABSTRACT

ANALYSIS OF PLATE STRUCTURES BY A DUAL FINITE ELEMENT METHOD

by

PETER KEI-KIN HO

A dual finite element method is developed for the analysis of the stretching and bending of linearly elastic, orthotropic plates. This finite element method is based on the duality that exists between the problems of plate stretching and bending. Nodal displacements are the unknowns in the stretching problem while nodal stress functions are those in the bending problem. A variational principle is used in formulating the governing system of equations. The boundary conditions considered are those of stress, displacement, mixed, elastic, edge beam, and strain in stretching; and those of displacement, stress, mixed, and stress function in bending.

The finite element method is implemented into a computer system named the PLANAL System, representing the Plate Analysis Language. The PLANAL System is developed as a subsystem of the Integrated Civil Engineering System (ICES). A user's manual and examples of application of PLANAL are included. Results from the examples are in close agreement with theoretical values.

ACKNOWLEDGMENT

The author wishes to express his sincere gratitude to Professor Jerome J. Connor, Jr. under whose patient guidance this work was completed. He is indebted also to Professor Ziad M. Elias under whose supervision the initial phase of this work was conducted.

The author wishes to thank the National Aeronautics and Space Administration, the sponsor of this work, and the Information Processing Center at the Massachusetts Institute of Technology where computational work was performed. He is also grateful to Messrs. George Will and Robert A. Wells, Jr. who offered useful suggestions during the preparation of computer programs, and to his brother Michael, Miss Mary Chiu, and Mr. Ven H. Shui who typed this work.

To the author's parents, Mr. and Mrs. Paul T. O. Ho, who made his education possible, this work is humbly dedicated.

CONTENTS

FIGURES AND TABLES

NOTATION

Important symbols used in this work are listed below. (The symbols (S) and (B) refer to the stretching and bending problems, respectively.)

a_i = component along x-axis of oriented side i of triangular element;

B = boundary term in potential energy (S);

B', B'' = boundary terms in complementary potential energy (B);

b_i = component along y-axis of oriented side i of triangular element;

D_x, D_y = flexural rigidities (B);

E_x, E_y = moduli of elasticity;

G = shear modulus;

h = plate thickness;

$\mathbf{i}$, $\mathbf{j}$, $\mathbf{k}$ = unit vectors along the coordinate axes;

$\mathbf{K}$ = coefficient matrix of system equations;

K_x, K_y = noncompatible curvatures in y- and x-directions, respectively, associated with particular solution of equilibrium equations, Eqs. (1.33), or particular solution functions (B);

l_i = length of side i of boundary;

M = Eq. (1.50) (B);

M_{nx}, M_{ny} = components of stress couple vector at boundary (B);

M_x, M_{xy}, M_{yx}, M_y = stress couples (B);

M_x°, M_y° = initial stress couples due to thermal causes (B);

$\mathbf{N}_n$ = edge load vector;

N_{nx}, N_{ny} = components of stress resultant vector at boundary (S);

N_x°, N_y° = initial stress resultants due to thermal causes (S);

n = subscript associated with direction normal to the boundary equations (B);

$\mathbf{P}$ = generalized force matrix (S), or generalized rotation matrix (B);

$\mathbf{p}$ = surface load vector;

P = potential energy density of surface load (S);

P'' = term in complementary potential energy (B);

p = superscript associated with particular solution of equilibrium equations (B);

p_x, p_y, p_z = components of surface load;

Q_n = transverse shear at boundary (B);

Q_{ne} = effective transverse shear (B);

Q_x, Q_y = transverse shears (B);

q = surface load intensity (B);

R_{xi}, R_{yi} = generalized force components at node i due to edge loads of one triangular element (S);

R'_{xi}, R'_{yi} = generalized rotation components at node i due to edge curvatures of one triangular element (B);

s = arclength of boundary or subscript referring thereto;

$\mathbf{U}$ = displacement matrix (S), or stress function matrix (B);

$\mathbf{u}$ = displacement vector;

U, V = stress functions (B);

U_i, V_i = U and V at node i (B);

u, v = displacement components (S);

u_i, v_i = u and v at node i (S);

W = strain energy density of plate (S);

W', W'' = complementary strain energy density of plate (B);

w = deflection of plate (B);

x, y = Cartesian coordinates in middle plane of plate; axes of elastic symmetry of orthotropic triangular element;

$\varepsilon_x, \varepsilon_{xy}, \varepsilon_{yx}, \varepsilon_y$ = linear components of strain (S);

θ_{xi}, θ_{yi} = generalized force components at node i due to thermal effects in one triangular element (S);

θ'_{xi}, θ'_{yi} = generalized rotation components at node i due to thermal effects in one triangular element (B);

ν_x, ν_y = Poisson ratios;

ξ_1, ξ_2, ξ_3 = triangular coordinates;

Π = potential energy of plate (S);

Π', Π'' = complementary potential energy expressions of plate (B);

χ_x°, χ_y° = thermal curvatures (B);

χ_x, χ_{xy}, χ_{yx}, χ_y = curvatures and twist of plate (B);

Ω_i = defined in Eq. (3.67) (B); and

* = superscript associated with the solution of the homogeneous equilibrium equations, i.e., with the portions of the force quantities obaained through the stress functions (B).

INTRODUCTION

The finite element method has been the subject of considerable research effort in structural mechanics in recent years. In this method, a continuum is represented by a number of elements joined together at a number of nodes and along interelement boundaries. Variational principles or other methods may be applied in formulating a system of equations describing the problem. In a displacement method, displacement quantities at the nodes are chosen as the unknowns of the equations; whereas in a force method, force quantities are chosen.

Displacement methods are used extensively in the analysis of plate and shell structures. In the problem of plate stretching where two displacements per node are the unknowns, satisfactory results are reported by Clough [2],† using triangular elements and linear displacement functions. However, in the problem of plate bending where three displacements per node are the unknowns, some difficulties seem to exist in obtaining equally satisfactory results [1,2,3,26].

Force methods have, on the other hand, received relatively little attention. A stress method has been presented by De Veubeke [7] and mixed methods by Herrmann [12] and Prato [20]. Recognizing the mathematical duality that exists between the problems of stretching and bending of plates, a finite element method in bending using stress functions as unknowns is presented by Elias [9].

A stiffness method for the stretching problem with unknown in-plane displacements can be interpreted as the dual of a flexibility method for the bending problem with unknown stress functions. Similarly, a stiffness method for the bending problem with an unknown deflection is the dual of a flexibility method for the stretching problem with an unknown Airy's stress function. Making use of this duality, a finite element

† Numerals in brackets refer to items in the References.

method using stress functions for the analysis of plate bending has the same behavior as the method using in-plane displacements for the analysis of plate stretching. The dual stress function method in plate bending has been shown by Elias [9,10] to produce satisfactory results.

It has been shown that the displacement method and the stress function method provide, respectively, lower and upper bounds to the deflection of a plate in bending [10]. This provides a method for evaluating the deviation of finite element solutions from an exact solution.

In the bending problem, the stress function method involves two unknowns per node, whereas the displacement method involves three unknowns per node. This results in a significant difference in computation effort in solving the governing system of equations.

In the present work, the dual finite element method (displacement method in stretching and stress function method in bending) for the analysis of plate structures is presented in Chapters 1, 2, and 3. The method is implemented into a computer system called the PLANAL System, representing Plate Analysis Language. The system is developed as a subsystem of the Integrated Civil Engineering System (ICES). Implementation logic, a user's manual, and examples of application of the PLANAL System are presented in Chapters 4, 5, and 6. Documentation and listing of computer programs in the system are included in the appendices.

CHAPTER 1

DUALITY IN STRETCHING AND BENDING OF ORTHOTROPIC PLATES

1.1. Introduction.

The basic equations and variational formulations of the problems of stretching and bending of a plate are presented in this chapter. It may be noted from the basic equations that duality exists between the two problems.

Throughout this work, the right-handed Cartesian coordinate system (x, y, z) is adopted. The middle surface of the undeformed plate is assumed to lie in the xy-plane. Unit vectors along the x-, y-, and z-axes are denoted by $\mathbf{i}$, $\mathbf{j}$, and $\mathbf{k}$, respectively.†

Differentiation with respect to an independent variable is indicated by a comma followed by that variable, for example,

$$f_{,x} = \frac{\partial f}{\partial x}, \qquad f_{,s} = \frac{\partial f}{\partial s}.$$

1.2. Basic Equations.

Presented in this section are the basic equations which describe the stretching and bending of a thin plate under the *small* deflection theory. These equations are reduced from the general equations in three dimensions by neglecting the extensional stress normal to the plate and

† Depending on the context, boldface types here denote vectors.

adopting Kirchhoff's hypothesis concerning the deformation of normals to the plate. The material of the plate is considered to be linearly elastic and orthotropic (i.e., there are two orthogonal planes of elastic symmetry normal to the plane of the plate).

Equilibrium Equations.

Consider a thin plate in equilibrium (Fig. 1.1) under a surface load of vector intensity

$$\mathbf{p} = p_x \mathbf{i} + p_y \mathbf{j} + p_z \mathbf{k}, \tag{1.1}$$

and an edge load of vector intensity

$$\mathbf{N}_n = N_{nx} \mathbf{i} + N_{ny} \mathbf{j} + Q_n \mathbf{k}. \tag{1.2}$$

The *differential equations of equilibrium* of the plate may be written in the form

$$\begin{aligned} N_{x,x} + N_{yx,y} + p_x &= 0, \\ N_{xy,x} + N_{y,y} + p_y &= 0, \\ N_{xy} - N_{yx} &= 0, \end{aligned} \tag{1.3}$$

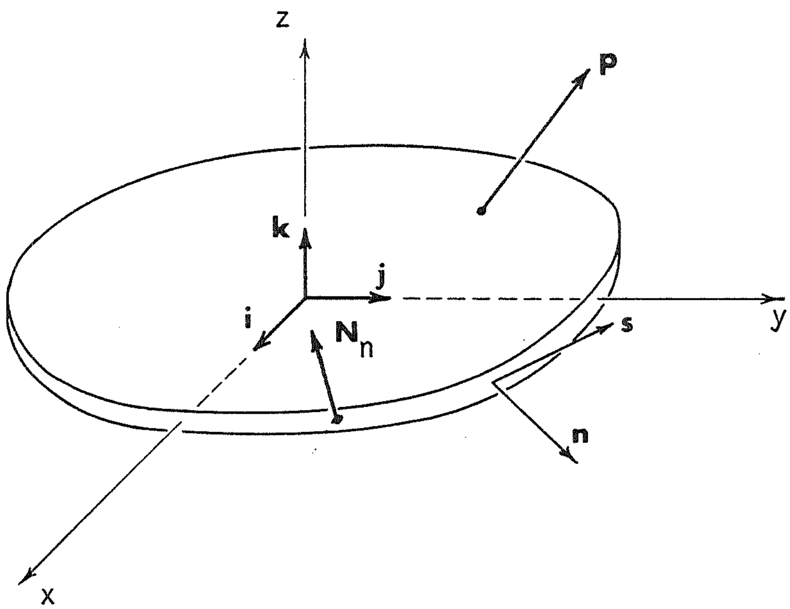


Fig. 1.1. A thin plate in equilibrium under surface and edge loads.

and

$$M_{x,x} + M_{yx,y} - Q_x = 0,$$

$$M_{xy,x} + M_{y,y} - Q_y = 0,$$

$$Q_{x,x} + Q_{y,y} + p_z = 0, \qquad (1.4)$$

$$M_{xy} - M_{yx} = 0,$$

where N_x, N_{xy}, N_{yx}, N_y are the in-plane stress resultants, Q_x, Q_y the transverse shears, and M_x, M_{xy}, M_{yx}, M_y the stress couples (Fig. 1.2).

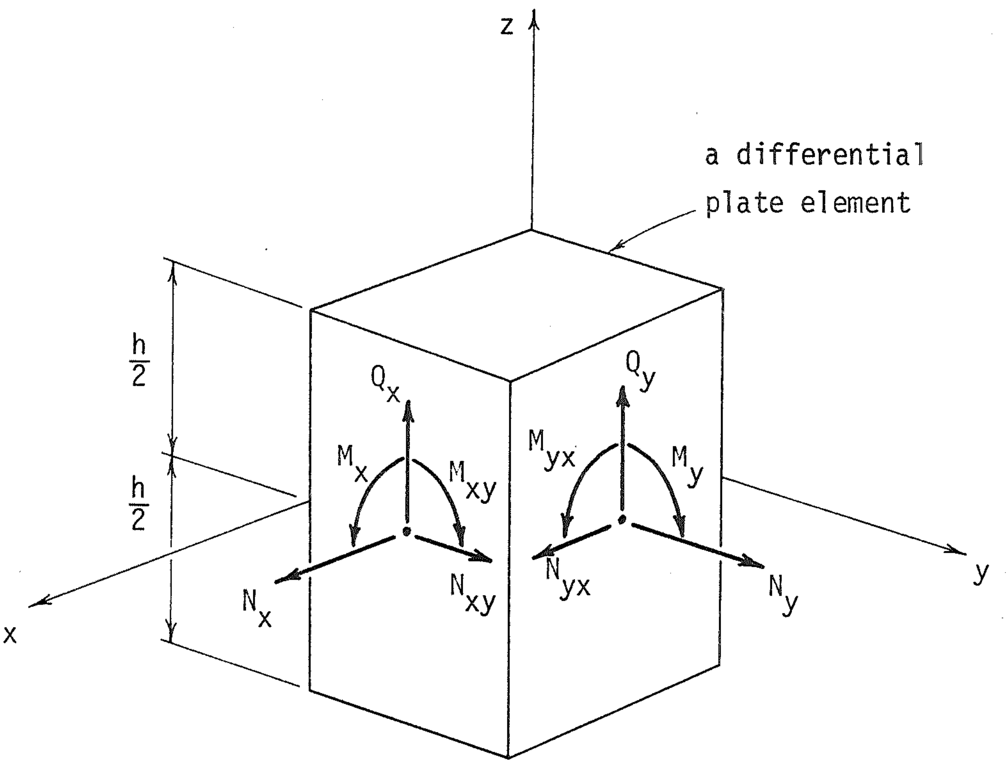


Fig. 1.2. Definition of stress resultants, transverse shears, and stress couples acting on a differential plate element.

It can be seen that the in-plane stress resultants in (1.3) are *uncoupled* from the transverse shears and stress couples in (1.4). Thus, (1.3) are the equilibrium equations of the *stretching* problem, and (1.4) are those of the *bending* problem.

Stress-strain Relations.

The displacement vector of the plate is defined as

$$\mathbf{u} = u\,\mathbf{i} + v\,\mathbf{j} + w\,\mathbf{k}. \tag{1.5}$$

In the *stretching* problem, the generalized strains are ε_x, ε_y, and $\varepsilon_{xy} = \varepsilon_{yx} = \frac{1}{2}\gamma_{xy} = \frac{1}{2}\gamma_{yx}$ which are defined by

$$\varepsilon_x = u_{,x}, \qquad \varepsilon_y = v_{,y}, \qquad \varepsilon_{xy} = \varepsilon_{yx} = \frac{1}{2}(u_{,y} + v_{,x}). \tag{1.6}$$

The generalized strains are related to the generalized stresses (in-plane stress resultants) through the *stress-strain relations*

$$\begin{Bmatrix} \varepsilon_x - \varepsilon_x^\circ \\ \varepsilon_y - \varepsilon_y^\circ \\ \varepsilon_{xy} \end{Bmatrix} = \frac{1}{h} \begin{bmatrix} \frac{1}{E_x} & -\frac{\nu_y}{E_x} & 0 \\ -\frac{\nu_x}{E_y} & \frac{1}{E_y} & 0 \\ 0 & 0 & \frac{1}{2G} \end{bmatrix} \begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix}, \tag{1.7}$$

where ε_x° and ε_y° are initial strains due to temperature change, and h is the thickness of the plate. The coefficient matrix in (1.7) is symmetrical; hence,

$$\frac{\nu_y}{E_x} = \frac{\nu_x}{E_y}. \tag{1.8}$$

The elastic constants are E_x, E_y, ν_x, ν_y and G where E_x, E_y are the Young's moduli in the x-, y-directions, respectively; ν_x, ν_y Poisson's ratios in the x-, y-directions, respectively; and G the shear modulus. As a result of (1.8), there are only four *distinct* elastic constants in an orthotropic plate.† The inverse relations of (1.7) are

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix} = \frac{h}{1 - \nu_x \nu_y} \begin{bmatrix} E_x & \nu_y E_y & 0 \\ \nu_x E_x & E_y & 0 \\ 0 & 0 & 2G(1 - \nu_x \nu_y) \end{bmatrix} \begin{Bmatrix} \varepsilon_x - \varepsilon_x^\circ \\ \varepsilon_y - \varepsilon_y^\circ \\ \varepsilon_{xy} \end{Bmatrix} . \qquad (1.9)$$

In the *bending* problem, the generalized strains are χ_x, χ_y, and $\chi_{xy} = \chi_{yx}$ which are defined by

$$\chi_x = - w_{,xx}, \qquad \chi_y = - w_{,yy}, \qquad \chi_{xy} = \chi_{yx} = - w_{,xy}. \qquad (1.10)$$

The stress couples and transverse shears can be expressed in terms of two stress functions U and V in the form

$$\begin{gathered} M_x = V_{,y}, \quad M_y = U_{,x}, \quad M_{xy} = M_{yx} = -\frac{1}{2}(U_{,y} + V_{,x}), \\ Q_x = \frac{1}{2}(V_{,xy} - U_{,yy}), \quad Q_y = -\frac{1}{2}(V_{,xx} - U_{,yx}). \end{gathered} \qquad (1.11)$$

The generalized stresses (the stress couples) are related to the generalized strains through the *stress-strain relations*

$$\begin{Bmatrix} M_x - M_x^\circ \\ M_y - M_y^\circ \\ M_{xy} \end{Bmatrix} = \frac{h^3}{12(1-\nu_x\nu_y)} \begin{bmatrix} E_x & \nu_y E_y & 0 \\ \nu_x E_x & E_y & 0 \\ 0 & 0 & 2G(1-\nu_x\nu_y) \end{bmatrix} \begin{Bmatrix} \chi_x \\ \chi_y \\ \chi_{xy} \end{Bmatrix} , \qquad (1.12)$$

where M_x° and M_y° are initial stress couples due to temperature change. The inverse relations of (1.12) are

---

† For solids in *three* dimensions, there are, in general, 21 distinct elastic constants in an anisotropic material, and nine distinct elastic constants in an orthotropic material.

$$\begin{Bmatrix} \chi_x \\ \\ \chi_y \\ \\ \chi_{xy} \end{Bmatrix} = \frac{12}{h^3} \begin{bmatrix} \frac{1}{E_x} & -\frac{\nu_y}{E_x} & 0 \\ \\ -\frac{\nu_x}{E_y} & \frac{1}{E_y} & 0 \\ \\ 0 & 0 & \frac{1}{2G} \end{bmatrix} \begin{Bmatrix} M_x - M_x^\circ \\ \\ M_y - M_y^\circ \\ \\ M_{xy} \end{Bmatrix} . \tag{1.13}$$

If we define

$$D_x = \frac{E_x h^3}{12(1-\nu_x\nu_y)},$$
$$D_y = \frac{E_y h^3}{12(1-\nu_x\nu_y)}, \tag{1.14}$$

where D_x and D_y are flexural rigidities of the plate, then (1.12) can be expressed in the form

$$\begin{Bmatrix} M_x - M_x^\circ \\ M_y - M_y^\circ \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} D_x & \nu_x D_x & 0 \\ \nu_y D_y & D_y & 0 \\ 0 & 0 & \frac{Gh^3}{6} \end{bmatrix} \begin{Bmatrix} \chi_x \\ \chi_y \\ \chi_{xy} \end{Bmatrix} . \tag{1.15}$$

Compatibility Equations.

In the *stretching* problem, the compatibility equations are

$$\varepsilon_{y,x} - \varepsilon_{xy,y} - \chi_{yz} = 0,$$
$$\varepsilon_{yx,x} - \varepsilon_{x,y} - \chi_{xz} = 0, \tag{1.16}$$
$$\chi_{yz,x} - \chi_{xz,y} = 0,$$

where

$$\chi_{yz} = \frac{1}{2}(v_{,xy} - u_{,yy}),$$
$$\chi_{xz} = \frac{1}{2}(v_{,xx} - u_{,yx}). \tag{1.17}$$

In the *bending* problem, the compatibility equations are

$$\chi_{y,x} - \chi_{xy,y} = 0,$$
$$\chi_{yx,x} - \chi_{x,y} = 0. \tag{1.18}$$

1.3. Stretching-Bending Duality.

As shown in the last section, the basic equations of the plate separate into two uncoupled systems: the stretching and the bending problems. There is a duality between the two systems of equations which is a particular case of the static-geometry analogy of shell theory where it is established, however, for zero surface load [8]. To include the case of non-zero surface load, there is more than one way that the analogy may be made. For this purpose, the superscript * will denote quantities associated with the homogeneous solution of the equilibrium equation.

For example, (1.3) in stretching with load terms deleted has the same form as (1.18) in bending. On the other hand, (1.4) in bending with load terms deleted has the same form as (1.16) in stretching.

It may be seen that the basic equations of the stretching problem can be transformed into the basic equations of the bending problem, and vice versa, by interchanging dual dependent variables and certain forms of the elastic constants in the two problems. The stretching-bending duality in the basic equations are tabulated in Table 1.1.† The dual

† In Tables 1.1 and 1.2, the solution of the homogeneous equilibrium equations in *bending* is taken. Hence, the dependent variables with superscript * are associated with the portions of the force quantities obtained through the stress functions. See Reference [8] for a full listing of duality in the basic equations.

Table 1.1. Stretching-Bending Duality in the Basic Equations.

Stretching Problem	Bending Problem
Equilibrium Equations (1.3): $N_{x,x} + N_{yx,y} = 0,$ $N_{xy,x} + N_{y,y} = 0.$	Compatibility Equations (1.18): $\chi^*_{y,x} - \chi_{xy,y} = 0,$ $\chi_{yx,x} - \chi^*_{x,y} = 0.$
Compatibility Equations (1.16): $\varepsilon_{y,x} - \varepsilon_{xy,y} - \chi_{yz} = 0,$ $\varepsilon_{yx,x} - \varepsilon_{x,y} - \chi_{xz} = 0,$ $\chi_{yz,x} - \chi_{xz,y} = 0.$	Equilibrium Equations (1.4): $M^*_{x,x} + M_{yx,y} - Q^*_x = 0,$ $M_{xy,x} + M^*_{y,y} - Q^*_y = 0,$ $Q^*_{x,x} + Q^*_{y,y} = 0.$
Stress-strain Relations (1.7): $\varepsilon_x - \varepsilon^\circ_x = \frac{1}{E_x h} N_x - \frac{\nu_y}{E_x h} N_y,$ $\varepsilon_y - \varepsilon^\circ_y = -\frac{\nu_x}{E_y h} N_x + \frac{1}{E_y h} N_y,$ $\varepsilon_{xy} = \frac{1}{2G} N_{xy}.$	Stress-strain Relations (1.15): $M^*_x - M^\circ_x = D_x\chi^*_x + \nu_x D_x \chi^*_y,$ $M^*_y - M^\circ_y = \nu_y D_y \chi^*_x + D_y \chi^*_y,$ $M_{xy} = \frac{Gh^3}{6} \chi_{xy}.$
Strain-displacement Relations (1.6) (1.17): $\varepsilon_x = u_{,x},\quad \varepsilon_y = v_{,y},$ $\varepsilon_{xy} = \varepsilon_{yx} = \frac{1}{2}(u_{,y} + v_{,x}),$ $\chi_{yz} = \frac{1}{2}(v_{,xy} - u_{,yy}),$ $\chi_{xz} = \frac{1}{2}(v_{,xx} - u_{,yx}).$	Stress-stress function Relations (1.11): $M^*_x = V_{,y},\quad M^*_y = U_{,x},$ $M_{xy} = M_{yx} = -\frac{1}{2}(U_{,y} + V_{,x}),$ $Q^*_x = \frac{1}{2}(V_{,xy} - U_{,yy}),$ $Q^*_y = -\frac{1}{2}(V_{,xx} - U_{,yx}).$

Table 1.2. Stretching-Bending Duality in the Dependent Variables and Elastic Constants.

Stretching Problem	Bending Problem
Dependent Variables	
u, v	U, V
p_x, p_y	$K_{x,x}$, $K_{y,y}$
ε_x, ε_{xy}, ε_y	M_y^*, $-M_{xy}^*$, M_x^*
ε_n, ε_{ns}, ε_s	M_s^*, $-M_{ns}^*$, M_n^*
N_x, N_{xy}, N_y	$-\chi_y^*$, χ_{xy}^*, $-\chi_x^*$
N_{nx}, N_{ny}	$-\chi_{sy}^*$, $-\chi_{sx}^*$
χ_{xz}, χ_{yz}	$-Q_y^*$, Q_x^*
Elastic Constants	
E_xh, E_yh, Gh	$-D_y^{-1}$, $-D_x^{-1}$, $-\left(\frac{Gh^3}{3}\right)^{-1}$
ν_x, ν_y	$-\nu_x$, $-\nu_y$

dependent variables and the dual elastic constants are listed in Table 1.2.†

1.4. Variational Formulation of the Stretching Problem in Terms of the Displacements.

Consider a plate in equilibrium under surface load components p_x and

† See the previous footnote.

p_y and edge load components N_{nx} and N_{ny} (Fig. 1.3). The plate is considered to be linearly elastic and orthotropic. The strain energy density function W has the form

$$W = \frac{E_x E_y h}{2(1-\nu_x \nu_y)} \left[\frac{\varepsilon_x^2}{E_y} + \frac{\varepsilon_y^2}{E_x} + \left(\frac{\nu_x}{E_y} + \frac{\nu_y}{E_x} \right) \varepsilon_x \varepsilon_y \right]$$
$$+ 2Gh\varepsilon_{xy}^2 + N_x^\circ \varepsilon_x + N_y^\circ \varepsilon_y. \tag{1.19}$$

N_x° and N_y° are initial stress resultants related to thermal strains ε_x° and ε_y° through the relations

$$N_x^\circ = - \frac{E_x h}{1-\nu_x \nu_y} (\varepsilon_x^\circ + \nu_x \varepsilon_y^\circ)$$
$$N_y^\circ = - \frac{E_y h}{1-\nu_x \nu_y} (\varepsilon_y^\circ + \nu_y \varepsilon_x^\circ) \tag{1.20}$$

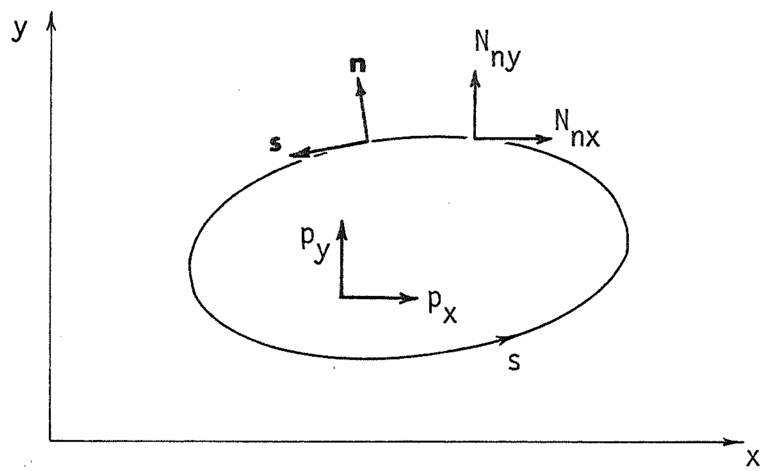


Fig. 1.3. Stretching of a plate under surface and edge loads.

The potential energy due to surface load is $\iint P\, dA$, where

$$P = -p_x u - p_y v. \tag{1.21}$$

The potential energy due to edge load is $\oint B\, ds$, where

$$B = -N_{nx} u - N_{ny} v. \tag{1.22}$$

Therefore, the potential energy of the plate takes the form

$$\Pi = \iint (W + P) dA + \oint B\, ds. \tag{1.23}$$

The above functional can be expressed in terms of the displacements through strain-displacement relations.

The principle of stationary potential energy (sometimes known as the principle of virtual displacements) requires the first variations of the functional Π with respect to the displacements to vanish. That is, the displacements satisfy the variational equation

$$\delta \Pi = 0. \tag{1.24}$$

1.5. Variational Formulation of the Bending Problem in Terms of Stress Functions.

Consider a plate in equilibrium under a surface load component p_z, an edge load component Q_n, and a stress couple at the boundary of vector intensity

$$\mathbf{M}_n = M_{nx}\,\mathbf{i} + M_{ny}\,\mathbf{j}. \tag{1.25}$$

The plate is again considered to be linearly elastic and orthotropic. The complementary strain energy density function W' has the form

$$W' = \frac{6}{h^3}\left[\frac{M_x^2}{E_x} + \frac{M_y^2}{E_y} - \left(\frac{\nu_y}{E_x} + \frac{\nu_x}{E_y}\right) M_x M_y + \frac{M_{xy}^2}{G}\right] + \chi_x^o M_x + \chi_y^o M_y. \tag{1.26}$$

χ_x° and χ_y° are thermal curvatures related to initial stress couples M_x° and M_y° through the relations

$$M_x^\circ = - D_x\ (\chi_x^\circ + \nu_x\chi_y^\circ)$$
$$M_y^\circ = - D_y\ (\chi_y^\circ + \nu_y\chi_x^\circ) \tag{1.27}$$

The complementary potential energy due to edge load is $\oint B'ds$, where

$$B' = - M_{nx}w_{,y} + M_{ny}w_{,x} - Q_n w. \tag{1.28}$$

In a manner similar to that in the preceding section, the variational formulation in the form

$$\delta\Pi' = 0 \tag{1.29}$$

with respect to the stress functions is obtained where

$$\Pi' = \iint W'\ dA + \oint B'\ ds. \tag{1.30}$$

To arrive at a form of the variational formulation which is completely dual of the stretching problem, we proceed as follows.

The stress couples and transverse shear in (1.26) and (1.28) must satisfy the equilibrium equations (1.4). This is accomplished by writing the general solution of (1.4) as the superposition of a particular solution, denoted by the superscript *, of the corresponding homogeneous system.

$$\begin{aligned} M_x &= M_x^p + M_x^*, \\ M_y &= M_y^p + M_y^*, \\ M_{xy} &= M_{xy}^p + M_{xy}^*, \\ Q_x &= Q_x^p + Q_x^*, \\ Q_y &= Q_y^p + Q_y^*. \end{aligned} \tag{1.31}$$

From (1.11), the homogeneous solution is expressed in terms of the stress functions, thus

$$
\begin{aligned}
M_x^* &= V_{,y}, \\
M_y^* &= U_{,x}, \\
M_{xy}^* &= -\frac{1}{2}(U_{,y} + V_{,x}), \\
Q_x^* &= \frac{1}{2}(V_{,xy} - U_{,yy}), \\
Q_y^* &= -\frac{1}{2}(V_{,xx} - U_{,yx}).
\end{aligned} \tag{1.32}
$$

For convenience, the particular solution can be taken in the form

$$
\begin{aligned}
M_x^p &= -D_x(K_y + \nu_x K_x), \\
M_y^p &= -D_y(K_x + \nu_y K_y), \\
M_{xy}^p &= 0, \\
Q_x^p &= M_{x,x}^p = -\left[D_x(K_y + \nu_x K_x)\right]_{,x}, \\
Q_y^p &= M_{y,y}^p = -\left[D_y(K_x + \nu_y K_y)\right]_{,y},
\end{aligned} \tag{1.33}
$$

in which two *particular solution functions* K_x, K_y have been introduced. Comparing (1.33) with (1.15), it can be seen that $-K_y$ and $-K_x$ are curvature quantities, and are indeed the curvatures in the x- and y-directions, respectively.

Eqs. (1.33) satisfy the first two equations of (1.4) identically. To satisfy the third equation of (1.4), K_x and K_y must satisfy the differential equation

$$\left[D_x(K_y + \nu_x K_x)\right]_{,xx} + \left[D_y(K_x + \nu_y K_y)\right]_{,yy} = p_z. \tag{1.34}$$

Eqs. (1.31), (1.32), and (1.33) are then substituted into (1.26), (1.28), and (1.30). After use of Green's theorem in the area integral, integration by parts in the boundary integral, and deletion of non-varying terms, we obtain the functional

$$\Pi'' = \iint (W'' + P'')\, dA + \oint B''\, ds, \tag{1.35}$$

where

$$W'' = \frac{6}{h^3}\left[\frac{V_{,y}^2}{E_x} + \frac{U_{,x}^2}{E_y} - \left(\frac{\nu_y}{E_x} + \frac{\nu_x}{E_y}\right)U_{,x}V_{,x} + \frac{(U_{,y} + V_{,x})^2}{4G}\right]$$

$$+ \chi_x^\circ V_{,y} + \chi_y^\circ U_{,x}, \tag{1.36}$$

$$P'' = K_{x,x}U + K_{y,y}V, \tag{1.37}$$

$$B'' = (w_{,ys} - y_{,s}K_x)U + (-w_{,xs} + x_{,s}K_y)V. \tag{1.38}$$

The principle of complementary potential energy (sometimes known as the principle of virtual forces) requires the first variation of the functional Π'' with respect to the stress functions to vanish. That is, the stress functions satisfy the variational equation

$$\delta\Pi'' = 0. \tag{1.39}$$

To obtain the stress couples and curvatures in the bending problem, we procede as follows.

First, an appropriate choice of K_x and K_y is made (Section 1.6). Then stress functions U and V are obtained from (1.39). The stress couples M_x^*, M_y^*, M_{xy}^* and M_x^p, M_y^p, M_{xy}^p are computed through (1.32) and (1.33), and then summed as in (1.31). Curvatures χ_x^*, χ_y^*, χ_{xy}^* are defined in terms of M_x^*, M_y^*, M_{xy}^* by means of the stress-strain relations (1.13). Curvatures χ_x, χ_y, χ_{xy} are then obtained through

$$\begin{aligned} \chi_x &= \chi_x^* - K_y, \\ \chi_y &= \chi_y^* - K_x, \\ \chi_{xy} &= \chi_{xy}^*. \end{aligned} \tag{1.40}$$

1.6. Determination of a Particular Solution of the Bending Problem.

In solving the bending problem involving a surface load, it is necessary to determine a particular solution of the bending equilibrium equation if the method of stretching-bending duality is to be applied. A particular solution has to satisfy only the equilibrium equation and it does not have to satisfy the boundary conditions of the problem under consideration. It may be expected, however, that the closer a particular solution compares with the actual behavior, the more accurate is the finite element solution to the problem.

A particular solution may be determined in the form of two particular solution functions K_x and K_y, introduced in (1.33), which must satisfy the governing differential equation (1.34). Several schemes by which particular solutions may be determined are discussed below.

1. Determination of K_x and K_y by Fourier Series.

In this scheme, certain limi ations on the geometry and material properties of the plate are adopted. Only rectangular plates are considered, and the plate material is assumed to be isotropic, so that $D_x = D_y = D$ and $\nu_x = \nu_y = \nu$. The surface load p_z is assumed to be expressible in the form

$$p_z = c_1 x + c_2 y + c_3, \tag{1.41}$$

where c_1, c_2, and c_3 are arbitrary constants.

Using the simplification of

$$K_x = K_y = K, \tag{1.42}$$

Eq. (1.34) becomes

$$\Delta K = \frac{p_z}{D(1+\nu)}, \tag{1.43}$$

where Δ is Laplace's operator.

Each term of the right-hand member in (1.41) is expressed as a Fourier series by standard procedure [14]. If the center of the plate, with dimensions 2a by 2b, is located at the origin of the coordinate system (Fig. 1.4), then the terms of the right-hand member in (1.41) can be expressed in the forms be expressed in the forms

$$c_1 x = \frac{8c_1 a}{\pi^2} \sum_{m=1}^{\infty} \sum_{n}^{odd} \left(-1\right)^{\frac{2m+n+1}{2}} \frac{1}{mn} \sin\frac{m\pi x}{a} \cos\frac{n\pi y}{2b},$$

$$c_2 y = \frac{8c_2 b}{\pi^2} \sum_{m}^{odd} \sum_{n=1}^{\infty} \left(-1\right)^{\frac{n+2n+1}{2}} \frac{1}{mn} \cos\frac{m\pi x}{2a} \sin\frac{n\pi y}{b}, \tag{1.44}$$

$$c_3 = \frac{8c_3}{\pi^2} \sum_{m}^{odd} \sum_{n}^{odd} \left(-1\right)^{\frac{2m+2n-1}{2}} \frac{1}{mn} \cos\frac{m\pi x}{2a} \cos\frac{n\pi y}{2b}.$$

Substituting (1.44) into (1.43) finally leads to

$$K = \frac{K_1 + K_2 + K_3}{D(1+\nu)}, \tag{1.45}$$

where

$$K_1 = \frac{8c_1 a}{\pi^4} \sum_{m=1}^{\infty} \sum_{n}^{odd} \frac{\left(-1\right)^{\frac{2m+n-1}{2}}}{mn\left[\left(\frac{m}{a}\right)^2 + \left(\frac{n}{2b}\right)^2\right]} \sin\frac{m\pi x}{a} \cos\frac{n\pi y}{2b},$$

$$K_2 = \frac{8c_2 b}{\pi^4} \sum_{m}^{odd} \sum_{n=1}^{\infty} \frac{(-1)^{m+2n-1}}{mn\left[\left(\frac{m}{2a}\right)^2 + \left(\frac{n}{b}\right)^2\right]} \cos\frac{m\pi x}{2a} \sin\frac{n\pi y}{b}, \tag{1.46}$$

$$K_3 = \frac{64c_3}{\pi^4} \sum_{m}^{odd} \sum_{n}^{odd} \frac{\left(-1\right)^{\frac{m+n}{2}}}{mn\left[\left(\frac{m}{a}\right)^2 + \left(\frac{n}{b}\right)^2\right]} \cos\frac{m\pi x}{2a} \cos\frac{n\pi y}{2b}.$$

2. Determination of K_x and K_y by Strips.

The stress couples and transverse shears of the particular solution defined in (1.33) are the internal forces that would occur if the plate is imagined to be comprised of two families of strips parallel to the coordinate axes [9]. In this scheme, the load p_z may be subdivided arbitrarily between the two families of strips which behave independently of each other. The end conditions of the strips may be arbitrary. To obtain a definite particular solution, the boundary conditions of the strips and the portion of load p_z carried by one family of strips must be specified. If $c(x, y)$ is the portion of load carried by the strips parallel to the x-axis, (1.34) may be replaced by the two equations

$$\left[D_x\ (K_y + \nu_x K_x)\right]_{,xx} = c p_z,$$
$$\left[D_y\ (K_x + \nu_y K_y)\right]_{,yy} = (1 - c) p_z. \qquad (1.47)$$

Once K_x and K_y are solved, the dual stretching problem is well defined.

As an example, consider the plate in Fig. 1.4 to be a homogeneous and isotropic plate with a uniform load p_z. For simplicity, we take the case of $c = 1$, which means that only the family of strips in the x-direction exists. Eq. (1.47) may be satisfied by letting

$$K_x = 0,$$
$$K_{y,xx} = \frac{p_z}{D}. \qquad (1.48)$$

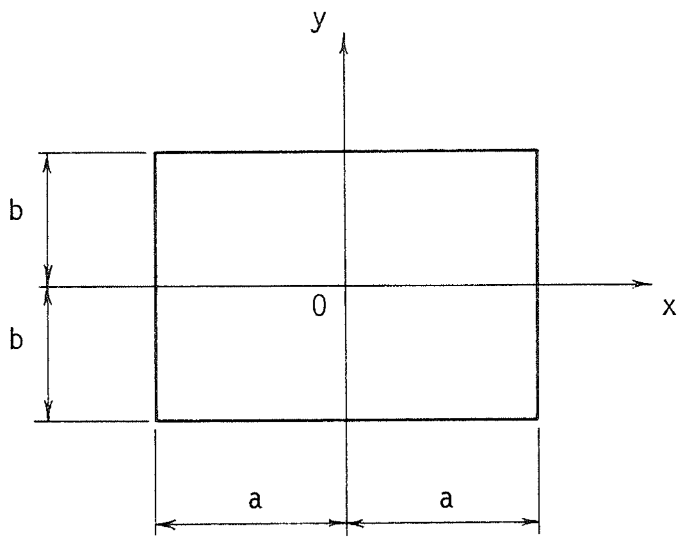


Fig. 1.4. Center of a rectangular plate located at the origin of the coordinate system.

By taking the strips as simply supported, we have

$$K_y = \frac{p_z(x^2 - a^2)}{2D}. \tag{1.49}$$

The particular solution in (1.48) and (1.49) is that of cylindrical bending in the x-direction.

3. Determination of K_x and K_y by a Finite Element Method.

In this scheme, a particular solution is determined through a finite element method using one unknown per node [10]. By letting

$$\begin{aligned} M_x^p &= M_y^p = M, \\ M_{xy}^p &= 0, \end{aligned} \tag{1.50}$$

the equilibiium equations (1.4) becomes

$$\Delta M + p_z = 0. \tag{1.51}$$

A variational formulation of (1.51) has the form

$$\iint \left[(M_{,x})^2 + (M_{,y})^2 - 2p_z M \right] dA = 0. \tag{1.52}$$

Eq. (1.52) may be used with an arbitrary subsidiary condition specifying M at the boundary.

CHAPTER 2

FORMULATION BY THE FINITE ELEMENT METHOD

2.1. Introduction.

In the finite element method, the body under study is discretized into elements and certain points in the body, known as nodes, are selected for analysis. In the present work, the plate structure under study is subdivided into triangular elements and the nodes are taken as the vertices of the elements. For the stretching problem, the unknowns are the two in-plane displacements at each node; for the bending problem, the unknowns are the two stress functions at each node.

The plate is taken to lie on the xy-plane of a right-handed Cartesian coordinate system. The material of the plate is considered to be linearly elastic and orthotropic.

2.2. Triangular Coordinates.

The selection of suitable displacement expansions is simplified considerably if one works with triangular coordinates ξ_1, ξ_2, and ξ_3 rather than with Cartesian coordinates [5,27]. Consider the triangle shown in Fig. 2.1. The nodes of the triangle are numbered 1, 2, and 3 in the direction from x- to y-axis around the boundary,† and the side opposite to node i is defined as side (i).

† That is, the counter-clockwise direction is taken in a right-handed coordinate system, while the clockwise direction is taken in a left-handed coordinate system.

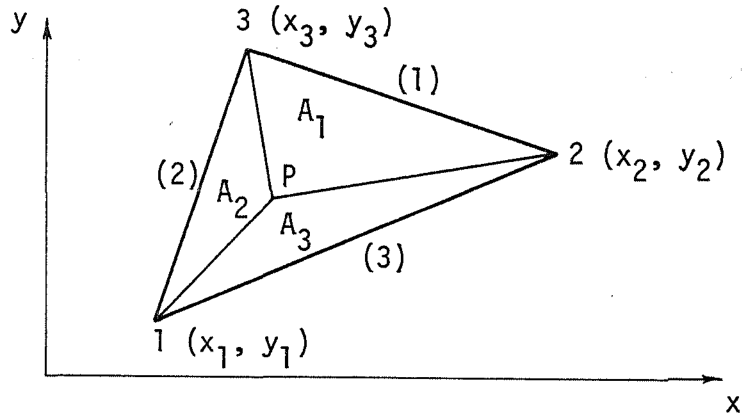


Fig. 2.1. Coordinates of a triangular element.

Consider a point P inside the triangle. Line segments joining the vertices and P divide the triangle into three subtriangles of area A_1, A_2, and A_3 such that

$$A_1 + A_2 + A_3 = A, \tag{2.1}$$

where A is the area of the triangle. The triangular coordinates of P are defined as the *dimensionless* quantities

$$\xi_i = \frac{A_i}{A}, \qquad i=1,2,3. \tag{2.2}$$

It can be seen from (2.1) that

$$\xi_1 + \xi_2 + \xi_3 = 1. \tag{2.3}$$

If we take **12** and **13** as vectors oriented along sides (3) and (2), respectively, and recall the definition of the vector cross product, the area is given by

$$2A = (\mathbf{12} \times \mathbf{13})\cdot \mathbf{k},$$

which leads to

$$2A = \begin{vmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{vmatrix}, \tag{2.4}$$

where x_i and y_i are the Cartesian coordinates of node i.

By applying (2.4) to each of the subtriangles, we obtain the relations between the triangular and Cartesian coordinates:

$$\xi_i = \frac{a_i y - b_i x + c_i}{2A}, \qquad i=1,2,3, \tag{2.5}$$

where

$$\begin{aligned} a_1 &= x_3 - x_2, & b_1 &= y_3 - y_2, & c_1 &= x_2 y_3 - x_3 y_2, \\ a_2 &= x_1 - x_3, & b_2 &= y_1 - y_3, & c_2 &= x_3 y_1 - x_1 y_3, \\ a_3 &= x_2 - x_1, & b_3 &= y_2 - y_1, & c_3 &= x_1 y_2 - x_2 y_1, \end{aligned} \tag{2.6}$$

which are obtained by cyclic permutation of the subscripts according to $1 \to 2 \to 3 \to 1$, etc. The quantities a_i and b_i can be considered as the components of side (i) of the triangle taken as a vector and oriented in the direction from x- to y-axis. It can be noted from (2.3), (2.5), and (2.6) that

$$\begin{aligned} a_1 + a_2 + a_3 &= 0, \\ b_1 + b_2 + b_3 &= 0, \\ c_1 + c_2 + c_3 &= 2A. \end{aligned} \tag{2.7}$$

Solving (2.5) for x and y, we have the inverse relations

$$\begin{aligned} x &= \xi_1 x_1 + \xi_2 x_2 + \xi_3 x_3, \\ y &= \xi_1 y_1 + \xi_2 y_2 + \xi_3 y_3. \end{aligned} \tag{2.8}$$

Expressions for partial derivatives with respect to the Cartesian coordinates can be readily established. For the first derivative of

$f(\xi_1, \xi_2, \xi_3)$, we have†

$$\begin{aligned}\frac{\partial f}{\partial x} &= \sum_i \frac{\partial f}{\partial \xi_i}\frac{\partial \xi_i}{\partial x} = \sum_i -\frac{b_i}{2A}\frac{\partial f}{\partial \xi_i},\\ \frac{\partial f}{\partial y} &= \sum_i \frac{\partial f}{\partial \xi_i}\frac{\partial \xi_i}{\partial y} = \sum_i \frac{a_i}{2A}\frac{\partial f}{\partial \xi_i}.\end{aligned} \tag{2.9}$$

By using two oblique coordinates, it can be shown that the integral of $f(\xi_1, \xi_2, \xi_3)$ over the triangle is given by

$$\iint_A f\,dA = 2A\int_{\xi_2=0}^{\xi_2=1}\left[\int_{\xi_1=0}^{\xi_1=1-\xi_2} f\,d\xi_1\right]d\xi_2. \tag{2.10}$$

The results of the first and second degree terms in ξ_i are listed below:

$$\begin{aligned}\iint_A \xi_i\,dA &= \frac{A}{3}, & i &= 1, 2, 3,\\ \iint_A \xi_i^2\,dA &= \frac{A}{6}, & i &= 1, 2, 3,\\ \iint_A \xi_i\xi_j\,dA &= \frac{A}{12}, & i &\neq j.\end{aligned} \tag{2.11}$$

2.3. Stretching of a Triangular Plate.

An approximate solution of the problem of stretching of an element in the form of a triangular plate is now obtained by applying a direct method to the variational equation (1.24).

Consider a triangular plate element in equilibrium (Fig. 2.2) under a surface load of vector intensity

$$\mathbf{p} = p_x\mathbf{i} + p_y\mathbf{j}, \tag{2.12}$$

edge loads of vector intensity

---

† Unless otherwise stated, index i or j under a summation sign indicates that the summation is to be taken over the subscripts 1, 2, and 3.

$$\mathbf{N}_i = N_{xi}\,\mathbf{i} + N_{yi}\,\mathbf{j}\,, \qquad i = 1,\ 2,\ 3, \tag{2.13}$$

concentrated nodal forces

$$\mathbf{F}_i = F_{xi}\,\mathbf{i} + F_{yi}\,\mathbf{j}\,, \qquad i = 1,\ 2,\ 3, \tag{2.14}$$

and a temperature change causing initial strains ε_x° and ε_y° which result in initial stresses N_x° and N_y° given by (1.20).

The displacement vector

$$\mathbf{u}(x,\ y) = u(x,\ y)\,\mathbf{i} + v(x,\ y)\,\mathbf{j} \tag{2.15}$$

describes the displacement of a point on the middle surface of the element. The displacement components u and v are sought as *linear* functions of the coordinates, and the result is in the form

$$\begin{aligned} u &= \xi_1 u_1 + \xi_2 u_2 + \xi_3 u_3, \\ v &= \xi_1 v_1 + \xi_2 v_2 + \xi_3 u_3. \end{aligned} \tag{2.16}$$

where u_i and v_i are the displacement components at node i.

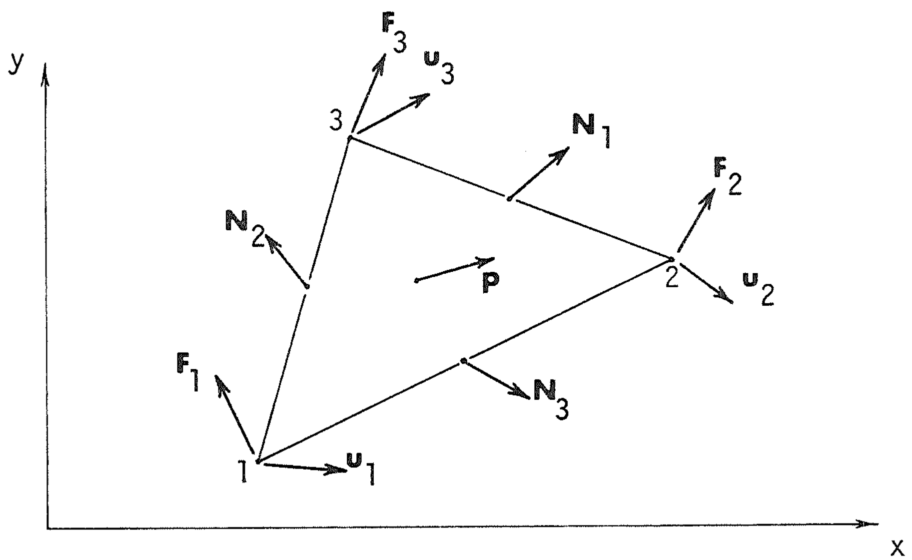


Fig. 2.2. Loads and displacements of a triangular plate element.

The strains are obtained by substituting (1.6) into (2.9), yielding

$$\varepsilon_x = \sum_i - \frac{b_i u_i}{2A},$$

$$\varepsilon_y = \sum_i \frac{a_i v_i}{2A}, \tag{2.17}$$

$$2\varepsilon_{xy} = \sum_i \frac{a_i u_i - b_i v_i}{2A}.$$

It can be noted that the strains given above are constant throughout the element, which is therefore called a *constant strain* element.

The total potential energy Π will now be expressed in terms of the nodal displacements.

The potential energy due to surface load, Eq. (1.21), takes the form

$$\iint_A P\, dA = \sum_i \left(- P_{xi} u_i - P_{yi} v_i\right), \tag{2.18}$$

where

$$\begin{aligned} P_{xi} &= \iint_A p_x \xi_x\, dA, \\ P_{yi} &= \iint_A p_x \xi_y\, dA, \end{aligned} \qquad i = 1, 2, 3. \tag{2.19}$$

The potential energy due to edge load, Eq. (1.22), takes the form

$$\oint_S B\, ds = \sum_i \left(- R_{xi} u_i - R_{yi} v_i\right), \tag{2.20}$$

where

$$R_{xi} = \sum_{j}^{j\neq i} \int_0^{l_j} N_{xj}\xi_i \, ds_j = \sum_{j}^{j\neq i} \frac{1}{j} \int_0^{l_j} N_{xj}s_j \, ds_j,$$

$$i=1,2,3. \quad (2.21)$$

$$R_{yi} = \sum_{j}^{j\neq i} \int_0^{l_j} N_{yj}\xi_i \, ds_j = \sum_{j}^{j\neq i} \frac{1}{j} \int_0^{l_j} N_{yj}s_j \, ds_j,$$

In (2.21), j refers to the two sides of the triangular plate intersecting at node i. On each side j, l_j is the length of the side and s_j is the arc-length oriented positively *toward* node i.

The potential energy due to concentrated nodal forces take the form

$$\oint_s - \mathbf{F}_i \cdot \mathbf{u}_i \, ds = \sum_i \left(- F_{xi}u_i - F_{yi}v_i \right). \quad (2.22)$$

The potential energy involving N_x° and N_y° in (1.19), after using (2.17), takes the form

$$\iint_A \left(N_x^\circ \varepsilon_x + N_y^\circ \varepsilon_y \right) dA = \sum_i \left(- \theta_{xi}u_i - \theta_{yi}v_i \right), \quad (2.23)$$

where

$$\theta_{xi} = \frac{b_i}{2A} \iint_A N_x^\circ \, dA,$$

$$i = 1, 2, 3. \quad (2.24)$$

$$\theta_{yi} = - \frac{a_i}{2A} \iint_A N_y^\circ \, dA,$$

The total potential energy Π may now be written in the form

$$\Pi = \sum_i \sum_j \left\{ \frac{E_x E_y h}{8A(1-\nu_x\nu_y)} \left[\frac{(b_i u_i)^2}{E_y} + \frac{(a_i v_1)^2}{E_x} - \left(\frac{\nu_x}{E_y} + \frac{\nu_y}{E_x} \right) b_i a_j u_i v_j \right] \right.$$

$$\left. + \frac{Gh}{A}(a_i u_i - b_i v_i)^2 - (P_{xi} + R_{xi} + \theta_{xi})u_i - (P_{yi} + R_{yi} + \theta_{yi})v_i \right\}. \quad (2.25)$$

In the case when p_x and p_y are *linear* in x and y, that is,

$$p_x = p_{x1}\xi_1 + p_{x2}\xi_2 + p_{x3}\xi_3,$$
$$p_y = p_{y1}\xi_1 + p_{y2}\xi_2 + p_{y3}\xi_3, \tag{2.26}$$

the integrals in (2.19) may be expressed in terms of the nodal values p_{xi} and p_{yi}, i = 1, 2, 3. The resulting expressions are

$$P_{xi} = \iint_A \left(\sum_j p_{xj}\xi_j\right)\xi_i \, dA = \frac{A}{12}(p_{xi} + p_{x1} + p_{x2} + p_{x3}),$$
$$i=1,2,3. \tag{2.27}$$
$$P_{yi} = \iint_A \left(\sum_j p_{yj}\xi_j\right)\xi_i \, dA = \frac{A}{12}(p_{yi} + p_{y1} + p_{y2} + p_{y3}),$$

Similarly, when N_x° and N_y° are *linear* in x and y, the integrals in (2.24) takes the form

$$\theta_{xi} = \frac{b_i}{2A}\iint_A \sum_j N_{yj}^\circ\xi_j \, dA = \frac{b_i}{6}(N_{x1}^\circ + N_{x2}^\circ + N_{x3}^\circ),$$
$$i=1,2,3, \tag{2.28}$$
$$\theta_{yi} = -\frac{a_i}{2A}\iint_A \sum_j N_{yj}^\circ\xi_j \, dA = -\frac{a_i}{6}(N_{y1}^\circ + N_{y2}^\circ + N_{y3}^\circ),$$

where N_{xi}° and N_{yi}° are the values at node i.

The variational equation (1.24) yields at each node k the two equations

$$\frac{\partial\Pi}{\partial u_k} = 0, \qquad k = 1, 2, 3, \tag{2.29a}$$

$$\frac{\partial\Pi}{\partial v_k} = 0, \qquad k = 1, 2, 3. \tag{2.29b}$$

Finally, using (2.25) in (2.29), we obtain for the plate element the *equilibrium equations*

$$\sum_i \frac{h}{4A(1-\nu_x\nu_y)} \Big\{ \Big[E_x b_k b_i + G(1 - \nu_x\nu_y) a_k a_i \Big] u_i$$

$$- \Big[E_x \nu_x b_k a_i + G(1 - \nu_x\nu_y) a_k b_i \Big] v_i \Big\} = P_{xk} + R_{xk} + \theta_{xk} + F_{xk},$$

$$k = 1, 2, 3, \qquad (2.30a)$$

$$\sum_i \frac{h}{4A(1-\nu_x\nu_y)} \Big\{ - \Big[E_y \nu_y a_k b_i + G(1 - \nu_x\nu_y) b_k a_i \Big] u_i$$

$$+ \Big[E_y a_k a_i + G(1 - \nu_x\nu_y) b_k b_i \Big] v_i \Big\} = P_{yk} + R_{yk} + \theta_{yk} + F_{yk},$$

$$k = 1, 2, 3. \qquad (2.30b)$$

The right-hand members of (2.30) can be considered as generalized nodal forces at node k.

2.4. Assembly of the System of Equations.

The system of equations governing the stretching of a plate may now be assembled. For convenience, matrix notation is used wherever appropriate.

First, the equations for a typical element n of the plate are assembled from (2.30). Letting

$$\mathbf{U}_i = \{ u_i \quad v_i \}, \quad i = 1, 2, 3, \qquad (2.31)$$

the nodal displacements are denoted by $\mathbf{U}_1$, $\mathbf{U}_2$, and $\mathbf{U}_3$.†

Edge loads and concentrated nodal forces will be considered after the equations for the entire plate have been assembled. Thus, the right-hand members of (2.30) can be replaced by $\mathbf{P}_k$, where

$$\mathbf{P}_k = \begin{Bmatrix} P_{xk} + \theta_{xk} \\ P_{yk} + \theta_{yk} \end{Bmatrix}. \qquad (2.32)$$

† Depending on the context, boldface types here denote matrices.

By introducing element stiffness matrices $\mathbf{k}_{ij}$, (2.30) can now be written in the form

$$\begin{bmatrix} \mathbf{k}_{11} & \mathbf{k}_{12} & \mathbf{k}_{13} \\ \mathbf{k}_{21} & \mathbf{k}_{22} & \mathbf{k}_{23} \\ \mathbf{k}_{31} & \mathbf{k}_{32} & \mathbf{k}_{33} \end{bmatrix} \begin{Bmatrix} \mathbf{U}_1 \\ \mathbf{U}_2 \\ \mathbf{U}_3 \end{Bmatrix} = \begin{Bmatrix} \mathbf{P}_1 \\ \mathbf{P}_2 \\ \mathbf{P}_3 \end{Bmatrix}, \tag{2.33}$$

where $\mathbf{k}_{ij}$ are 2 x 2 submatrices and have the form

$$\mathbf{k}_{ij} = \frac{h}{4A} \left[\begin{array}{c|c} \dfrac{E_x b_i b_j}{1-\nu_x\nu_y} + Ga_i a_j & -\dfrac{E_x \nu_x b_i a_j}{1-\nu_x\nu_y} - Ga_i b_j \\ \hline -\dfrac{E_y \nu_y a_i b_j}{1-\nu_x\nu_y} - Gb_i a_j & \dfrac{E_y a_i a_j}{1-\nu_x\nu_y} + Gb_i b_j \end{array} \right] \tag{2.34}$$

where i, j = 1, 2, 3. The numerical values of $\mathbf{k}_{ij}$ are element dependent, i.e., they depend on the geometric and material properties of a particular element.

Next, an illustration for a typical subassembly of elements incident on a node is presented. Consider m elements with n nodes arranged and named as shown in Fig. 2.3. Using superscripts to identify the elements and subscripts to identify the nodes, the equilibrium equations of elements 1, 2, ..., m for node 1 are

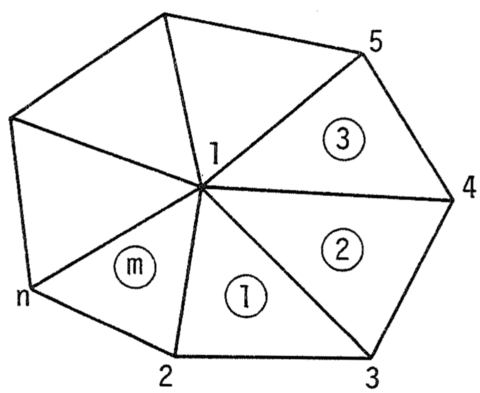


Fig. 2.3. A typical subassembly of elements.

$$\mathbf{k}_{11}^{(1)}\,\mathbf{U}_1 + \mathbf{k}_{12}^{(1)}\,\mathbf{U}_2 + \mathbf{k}_{13}^{(1)}\,\mathbf{U}_3 = \mathbf{P}_1^{(1)},$$

$$\mathbf{k}_{11}^{(2)}\,\mathbf{U}_1 + \mathbf{k}_{13}^{(2)}\,\mathbf{U}_3 + \mathbf{k}_{14}^{(2)}\,\mathbf{U}_4 = \mathbf{P}_1^{(2)}, \qquad (2.35)$$

$$\cdots$$

$$\mathbf{k}_{11}^{(m)}\,\mathbf{U}_1 + \mathbf{k}_{1n}^{(m)}\,\mathbf{U}_n + \mathbf{k}_{12}^{(m)}\,\mathbf{U}_2 = \mathbf{P}_1^{(m)}.$$

Summing Eqs. (2.35) yields the equilibrium equations of the subassembly of elements for node 1:

$$\left(\sum_{j=1}^{m} \mathbf{k}_{11}^{(j)}\right)\mathbf{U}_1 + \left(\mathbf{k}_{12}^{(m)} + \mathbf{k}_{12}^{(1)}\right)\mathbf{U}_2 + \left(\mathbf{k}_{13}^{(1)} + \mathbf{k}_{13}^{(2)}\right)\mathbf{U}_3$$
$$+ \ldots + \left(\mathbf{k}_{1n}^{(m-1)} + \mathbf{k}_{1n}^{(m)}\right)\mathbf{U}_n = \sum_{j=1}^{m} \mathbf{P}_1^{(j)}. \qquad (2.36)$$

Finally, the system of equations governing the plate can be assembled by applying (2.36) to all of the n nodes of the plate. The nodes are numbered, for convenience, consecutively from 1 through n. The displacements of all the nodes is represented by

$$\mathbf{U} = \left\{ \mathbf{U}_1, \; \mathbf{U}_2, \; \ldots \;, \; \mathbf{U}_n \right\}. \qquad (2.37)$$

Edge loads of an element are provided by the internal stresses from an adjacent element. Edge loads along the common edge of two elements are equal and opposite for the two elements. Therefore, for all interior edges, edge loads do not contribute to the total generalized nodal forces and are not considered. Edge loads along exterior edges are considered separately under stress boundary conditions (Section 3.4) and are neglected here.

Concentrated nodal forces are not considered in (2.36) when the system of equations are being assembled. They are added only after the assembly has been completed so that they are considered only once. The concentrated forces at node k are defined by

$$\mathbf{F}_k = \left\{ F_{xk} \quad F_{yk} \right\}. \qquad (2.38)$$

The assembled equations may be called, in the stretching problem, the global stiffness equations. In matrix notation, the system of equations has the form

$$\mathbf{K}\mathbf{U} = \mathbf{P}. \tag{2.39}$$

Eqs. (2.39) are called system equations in later discussion. In submatrix form, it can be written as

$$\begin{bmatrix} \mathbf{K}_{11} & \mathbf{K}_{12} & \cdots & \mathbf{K}_{1n} \\ \mathbf{K}_{21} & \mathbf{K}_{22} & \cdots & \mathbf{K}_{2n} \\ \cdots & \cdots & \cdots & \cdots \\ \mathbf{K}_{n1} & \mathbf{K}_{n2} & \cdots & \mathbf{K}_{nn} \end{bmatrix} \begin{Bmatrix} \mathbf{U}_1 \\ \mathbf{U}_2 \\ \vdots \\ \mathbf{U}_n \end{Bmatrix} = \begin{Bmatrix} \mathbf{P}_1 \\ \mathbf{P}_2 \\ \vdots \\ \mathbf{P}_n \end{Bmatrix}. \tag{2.40}$$

The *procedure* for assembly of $\mathbf{K}$ and $\mathbf{P}$ follows from (2.33) and (2.36):

1. *Before assembly,* $\mathbf{K}$ *and* $\mathbf{P}$ *are null matrices.*
2. *For element* m, *with nodes* n_1, n_2, n_3:
   (1). *Add* $\mathbf{k}_{ij}^{(m)}$ *computed by* (2.34) *to submatrix* $\mathbf{K}_{n_i n_j}$ *in hyper-row* n_i, *hyper-column* n_j *of* $\mathbf{K}$. *Repeat for* i, j = 1, 2, 3.
   (2). *Add* $\mathbf{P}_i^{(m)}$ *computed by* (2.32) *to submatrix* $\mathbf{P}_{n_i}$ *in hyper-row* n_i *of* $\mathbf{P}$. *Repeat for* i = 1, 2, 3.

   Repeat step 2 *for every element in the plate.*
3. *For node* k, *add* $\mathbf{F}_k$ *to submatrix* $\mathbf{P}_k$ *in hyper-row* k *of* $\mathbf{P}$.
   Repeat for all nodes with concentrated forces.

It should be noted that the system of equations (2.39) is singular, i.e., there exists a non-trivial solution $\mathbf{U}^\circ$ to the system when $\mathbf{P} = \mathbf{O}$. $\mathbf{U}^\circ$ represents the nodal displacements of a rigid body motion. To fix the plate against rigid-body motion, three independent displacement components must be specified, e.g., the two displacements at a node and the rotation about that node. These displacement components must be specified in order to solve the system. Once these are specified, the system is modified according to the algorithm described under displacement boundary condition in stretching, and the resulting system becomes non-singular.

2.5. Formulation for the Bending Problem.

The results obtained in Sections 2.3 and 2.4 are directly applicable to the dual bending problem by means of the correspondence in Table 1.2. Applying the stretching-bending duality to (2.30), and neglecting F_{xk} and F_{yk}, yields for the bending problem

$$\sum_i \frac{3}{Ah^3}\left\{\left[\frac{b_k b_i}{E_y} + \frac{a_k b_i}{4G}\right]U_i + \left[\frac{\nu_x b_k a_i}{E_y} - \frac{a_k b_i}{4G}\right]V_i\right\} = -P'_{xk} - R'^{*}_{xk} - \theta'_{xk}$$

$$k = 1,\ 2,\ 3, \qquad (2.41a)$$

$$\sum_i \frac{3}{Ah^3}\left\{\left[\frac{\nu_y a_k b_i}{E_x} - \frac{b_k a_i}{4G}\right]U_i + \left[\frac{a_k a_i}{E_x} + \frac{b_k b_i}{4G}\right]V_i\right\} = -P'_{yk} - R'^{*}_{yk} - \theta'_{yk}$$

$$k = 1,\ 2,\ 3, \qquad (2.41b)$$

where P'_{xk}, P'_{yk}, R'^{*}_{xk}, R'^{*}_{yk}, θ'_{xk}, and θ'_{yk} are dual of P_{xk}, P_{yk}, R_{xk}, R_{yk}, θ_{xk}, and θ_{yk}, respectively, and may be expressed through equations dual of (2.19), (2.21), and (2.24). The right-hand members of (2.41) can be considered as generalized nodal rotations at node k.

Assembly of the system of equations governing the bending of a plate is effected by applying the stretching-bending duality to the equations in Section 2.4. After material properties dual of those in bending have been replaced, the system of equations is assembled by the same procedure as used in the stretching problem. The assembled equations may be called, in the bending problem, the global flexibility equations. They are also called system equations in later discussion.

In computing the contribution of the particular solution functions of one element to the generalized nodal rotations P'_{xk} and P'_{yk} at node k, the equations

$$P'_{xk} = \iint_A K_{x,x}\xi_k \, dA,$$
$$P'_{yk} = \iint_A K_{y,y}\xi_k \, dA \qquad (2.42)$$

which are dual of (2.19) are used. However, in the schemes outlined in Section 1.6, it is the particular solution functions K_x and K_y themselves that are computed. It is possible to use K_x and K_y directly in the computation of P'_{xk} and P'_{yk}. Using Green's theorem and (2.5), (2.24) becomes

$$\begin{aligned} P'_{xk} &= \frac{b_k}{2A} \iint_A K_x \, dA + \oint K_x \xi_k \, dy, \\ P'_{yk} &= -\frac{a_k}{2A} \iint_A K_y \, dA - \oint K_y \xi_k \, dy. \end{aligned} \tag{2.43}$$

The total generalized nodal rotations G'_{xk} and G'_{yk} at node k due to the particular solution functions are obtained by superposition of P'_{xk} and P'_{yk}, respectively, of the elements having node k in common.

At an interior node, the line integrals in (2.43) add up to zero because $\xi_k = 0$ on the sides opposite to node k, and the integrands take opposite values on sides common to the triangular elements. Therefore,

$$\begin{aligned} G'_{xk} &= \sum \frac{b_k}{2A} \iint_A K_x \, dA, \\ G'_{yk} &= \sum - \frac{a_k}{2A} \iint_A K_y \, dA, \end{aligned} \tag{2.44}$$

where the summation extends over the elements having node k in common. It can be shown that (2.44) can also be used at a boundary node.

Use of triangular coordinates shows that, for example,

$$f = \xi_1 f_1 + \xi_2 f_2 + \xi_3 f_3, \tag{2.45}$$

where f_i is the nodal value of a function f at node i. It can be easily proved that the integral of f over a triangular element takes the form

$$\iint_A f \, dA = \frac{1}{3} A (f_1 + f_2 + f_3). \tag{2.46}$$

(2.46) can be conveniently used in evaluating the integrals in (2.44).

CHAPTER 3

MODIFICATION OF EQUATIONS FOR BOUNDARY CONDITIONS

3.1. Introduction.

The system of simultaneous equations governing the plate stretching problem are called the global stiffness equations, while those governing the plate bending problem are called the global flexibility equations. Since the forms of the two systems of equations are the same, both systems are called the *system equations* for generality. The procedure for the assembly of the system equations are the nodal values of the displacements or stress functions in the stretching or bending problems, respectively.

Let n be the total number of nodes in the plate. Then, there are 2n unknowns and the size of the coefficient matrix in (2.40) is 2n x 2n. In submatrix form, as in (2.40), the coefficient matrix is n x n.

The independent variable along a plate boundary is defined as s. The positive s-direction is taken as the one along which the outward normal points to the right. Boundary conditions may be specified at the nodes or along the segments between the nodes. Distributed quantities, such as edge loads, may vary linearly along a boundary segment and may be discontinuous at a node. An example showing the values of N_{yi}^- and N_{yi}^+ at the negative and positive sides, respectively, of node i is illustrated in Fig. 3.1.

In analytic methods, solutions to the differential equilibrium equations are obtained as expressions in term of arbitrary constants.

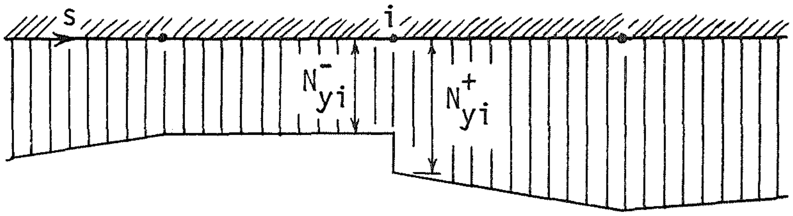


Fig. 3.1. Example showing values at the negative and positive sides of a node.

Boundary conditions are then imposed so that definite values of those constants can be determined. In a numerical method such as the one presented in this work, boundary conditions are incorporated in the form of modifications of the system equations.

The algorithms, or procedures, for modifications applicable to each of the boundary conditions considered are presented in the remainder of the chapter [9,19]. For clarity and conciseness, matrix notation is used wherever appropriate. However, some of the matrix multiplications indicated are not carried out explicitly in the computer implementation for the sake of efficient computations. The matrices $\mathbf{I}_2$ and $\mathbf{O}_2$ stand for the 2 x 2 unit and null matrices, respectively. The symbol " $\leftarrow$ " means that the quantities on the left are to be replaced by the quantities resulting from the operations indicated on the right.

3.2. Duality in Boundary Conditions.

The stretching-bending duality applies to the boundary conditions of the stretching and bending problems as well as to their basic equations. It can be seen that a wider class of boundary conditions appears than is usually considered in each of the two problems.

The dual of stress boundary conditions in the stretching problem are displacement boundary conditions in which curvature quantities have to be computed from the prescribed displacement quantities. The dual of displacement boundary conditions in stretching are stress function boundary conditions. For mixed boundary conditions in stretching, the dual mixed boundary conditions requires the specification of a stress function component and a curvature in the perpendicular direction. Elastic in stretching is dual of edge beam in bending, whereas edge beam in stretching is dual of elastic in bending. The dual of stress boundary conditions in bending are strain boundary conditions in which the extensional strain and the in-plane curvature of a boundary curve are specified. The duality in boundary conditions and their corresponding boundary values are listed in Table 3.1.

The coefficient matrix of the system equations is symmetric when it was assembled originally. However, symmetry may be destroyed when the equations are modified to incorporate certain boundary conditions. For example, in strain boundary conditions, certain rows in the coefficient matrix are replaced without changing the corresponding columns. In Table 3.1, an asterisk * in a boundary condition indicates that the coefficient matrix becomes non-symmetric, in general, after modifications for the boundary condition.

3.3. Geometric Relations.

Certain geometric relations required in subsequent sections are presented here.

Transformation of Vectors. A vector at a node may have its components referenced to a local coordinate system x^* and y^* which is different from the global coordinate system x and y. The local system at node i may be defined by an angle ϕ_i measured from the x-axis to the x^*-axis (Fig. 3.2). For example, the displacement components

$$\mathbf{U}_i^* = \{ u_i^* \quad v_i^* \} \tag{3.1}$$

in the local system may be transformed to those in the global system by the relations

Table 3.1. Stretching-Bending Duality in Boundary Conditions.

Stretching	Bending
Stress N_x, N_y	Displacement $-\chi_y$, $-\chi_x$
Displacement u, v	Stress function U, V
Mixed u_r, N_q	Mixed U_r, χ_q
Elastic u^s, v^s, k_{xx}, k_{xy}, k_{yx}, k_{yy}	Edge beam* U^s, V^s, f_{xx}, f_{xy}, f_{yx}, f_{yy}
Edge beam N_x, N_y, EA, EI	Elastic* $-\chi_y$, $-\chi_y$, $-f_{ss}$, $-f_{zz}$
Strain* ε_s, χ_s	Stress* M_n, Q_{ne}

* Coefficient matrix becomes non-symmetric after modifications for boundary condition.

$$\mathbf{U}_i = \mathbf{R}^i \mathbf{U}_i^*, \tag{3.2}$$

where the rotation matrix at node i is given by

$$\mathbf{R}^i = \begin{bmatrix} \cos\phi_i & -\sin\phi_i \\ \sin\phi_i & \cos\phi_i \end{bmatrix}. \tag{3.3}$$

Strain and Rotation of a Side. Consider line segment (i) of length l_i connecting nodes i and i+1 represented by A and B, respectively, in

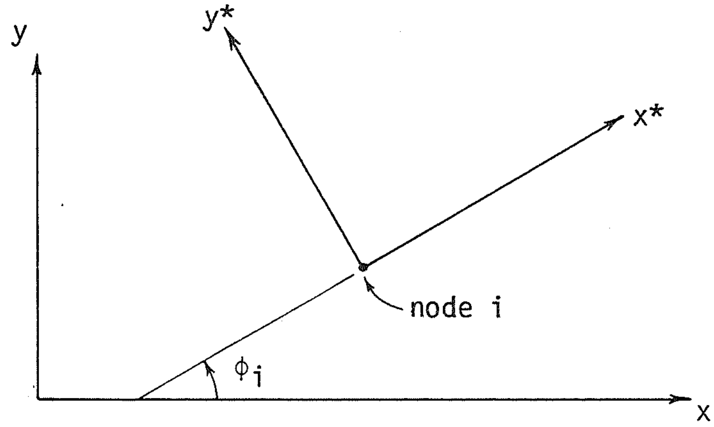


Fig. 3.2. Transformation of vectors.

Fig. 3.3. The deformed segment is translated to position AB'. The strain ε_i and rotation ω_i of the side are required. For small deformations, BB' may be taken as the elongation, $(CD)/l_i$ as the rotation, and

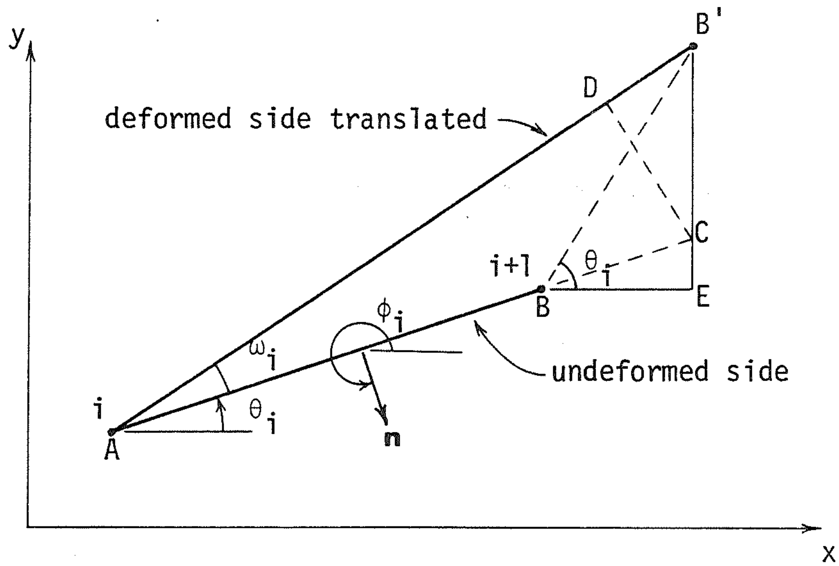


Fig. 3.3. Computation of strain and rotation of a side.

angle B'BE as θ_i. The orientation angle ϕ_i of the side is measured from the x-axis to the outward normal $\mathbf{n}$ of the side. Since

$$BE = u_{i+1} - u_i, \qquad EB' = v_{i+1} - v_i,$$

and

$$\sin \theta_i = \cos \phi_i, \qquad \cos \theta_i = - \sin \phi_i,$$

it can be shown that ε_i and ω_i are given by

$$\varepsilon_i l_i = - (u_{i+1} - u_i) \sin \phi_i + (v_{i+1} - v_i) \cos \phi_i, \tag{3.4}$$

$$\omega_i l_i = - (u_{i+1} - u_i) \cos \phi_i - (v_{i+1} - v_i) \sin \phi_i. \tag{3.5}$$

Curvature at a Node. In the finite element method, a curved boundary is considered to be comprised of a number of line segments. The in-plane curvature in this idealization does not exist and must be interpreted instead as the divided difference between rotations of two adjacent boundary segments. If sides (i-1) and (i) intersect at node i, then the in-plane curvature at node i is given by

$$\chi_i = \frac{2\,(\omega_i - \omega_{i-1})}{l_i + l_{i-1}}. \tag{3.6}$$

3.4. Modification for Boundary Conditions in Stretching.

The boundary conditions in stretching considered in this section are: stress, displacement, mixed, elastic, edge beam, and strain.

1. Stress Boundary Conditions.

It was stated in Section 2.4 that edge loads along exterior edges are considered under stress boundary conditions. Edge load intensities N_x and N_y specified on a side of length l connecting two nodes i and j (Fig. 3.4) contribute to the generalized nodal forces $R_{\alpha i}$ and $R_{\alpha j}$ at the

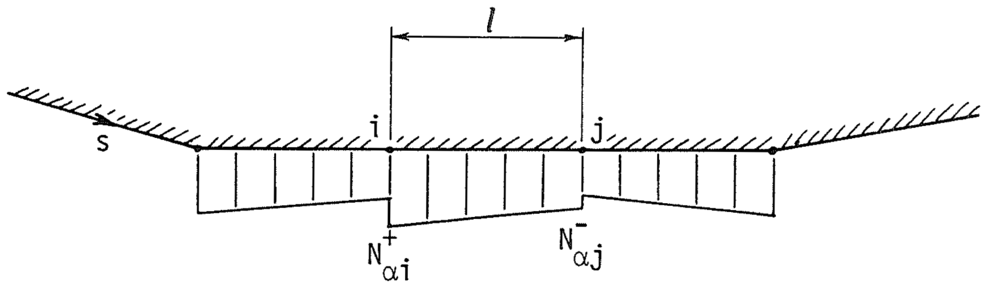


Fig. 3.4. Stresses specified on a side along the boundary.

nodes. From (2.21), we have

$$R_{\alpha i} = \frac{1}{l} \int_0^l N_\alpha \,(l - s)\, ds,$$
$$R_{\alpha j} = \frac{1}{l} \int_0^l N_\alpha s \, ds, \tag{3.7}$$

where $\alpha = x, y$, If N_α is a linear function in s, then (3.7) becomes

$$R_{\alpha i} = \frac{l}{6} (2N^{+}_{\alpha i} + N^{-}_{\alpha j}),$$
$$R_{\alpha j} = \frac{l}{6} (N^{+}_{\alpha i} + 2N^{-}_{\alpha j}), \tag{3.8}$$

where $N^{+}_{\alpha i}$ is the value of N_α at the positive side of node i and $N^{-}_{\alpha j}$ is the value of N_α at the negative side of node j (Section 3.1). N_α may be discontinuous at a node as shown in Fig. 3.4.

In the system equations, submatrices

$$\mathbf{R}_i = \{ R_{xi} \quad R_{yi} \},$$
$$\mathbf{R}_j = \{ R_{xj} \quad R_{yj} \}, \tag{3.9}$$

are added to $\mathbf{P}_i$ and $\mathbf{P}_j$, respectively, for every side along the boundary with specified stress boundary conditions.

In the case when edge loads are specified on the entire boundary, a rigid body displacement in the form of three appropriate displacement components (Section 2.4) must be specified so that the resulting coefficient matrix will be non-singular.

2. Displacement Boundary Conditions.

If the displacements $\mathbf{U}_i$ are prescribed at node i, then there are two less unknown nodal displacements. The two equilibrium equations associated with that node can be deleted from the system equations.† Terms involving $\mathbf{U}_i$ in the other equations of the system are then transposed to the right-hand members.

The algorithm for modifying the system equation for node i is as follows:

1. *In* $\mathbf{K}$:

$$\mathbf{K}_{ij} \leftarrow \mathbf{O}_2, \qquad j \neq i,\ j = 1, 2, \ldots, n. \tag{3.10}$$

$$\mathbf{K}_{ji} \leftarrow \mathbf{O}_2, \qquad j \neq i,\ j = 1, 2, \ldots, n. \tag{3.11}$$

$$\mathbf{K}_{ii} \leftarrow \mathbf{I}_2. \tag{3.12}$$

2. *In* $\mathbf{P}$:

$$\mathbf{P}_j \leftarrow \mathbf{P}_j - \mathbf{K}_{ji}\,\mathbf{U}_i, \qquad j \neq i,\ j = 1, 2, \ldots, n. \tag{3.13}$$

$$\mathbf{P}_i \leftarrow \mathbf{U}_i. \tag{3.14}$$

3. Mixed Boundary Conditions.

In mixed boundary conditions, one displacement component and an edge load component in a normal direction may be prescribed. The displacement component u_{ri} is taken at node i in a direction $\mathbf{r}$. The edge

---

† During assembly of the system equations, if those equations associated with prescribed displacements are assembled from the bottom upwards, the coefficient matrix would remain "compact" after those equations have been deleted.

load component of magnitudes N_q^- and N_q^+ at the negative and positive sides, respectively, of node i may be prescribed in the direction $\mathbf{q}$ normal to $\mathbf{r}$ (Fig. 3.5). The direction of $\mathbf{r}$ at node i is given by an angle ϕ_i measured from the positive x-axis to $\mathbf{r}$, and $\mathbf{q}$ is taken to be $\pi/2$ radians *ahead* of $\mathbf{r}$.

Eq. (3.7) or (3.8), with α replaced by q, are used to compute N_{qi}, the generalized nodal forces contributed by N_q specified on the two sides issuing from node i.

The algorithm for modifying the system equations for node i is presented below:

1. *Four matrices* $\mathbf{E}$, $\mathbf{G}$, $\mathbf{u}_i^*$, *and* $\mathbf{N}_i^*$ *given by*

$$\mathbf{E} = \begin{bmatrix} 0 & 0 \\ 0 & 1 \end{bmatrix}, \qquad \mathbf{G} = \begin{bmatrix} 1 & 0 \\ 0 & 0 \end{bmatrix},$$
$$\mathbf{u}_i^* = \begin{Bmatrix} u_{ri} \\ 0 \end{Bmatrix}, \qquad \mathbf{N}_i^* = \begin{Bmatrix} 0 \\ N_{qi} \end{Bmatrix}, \tag{3.15}$$

are defined.

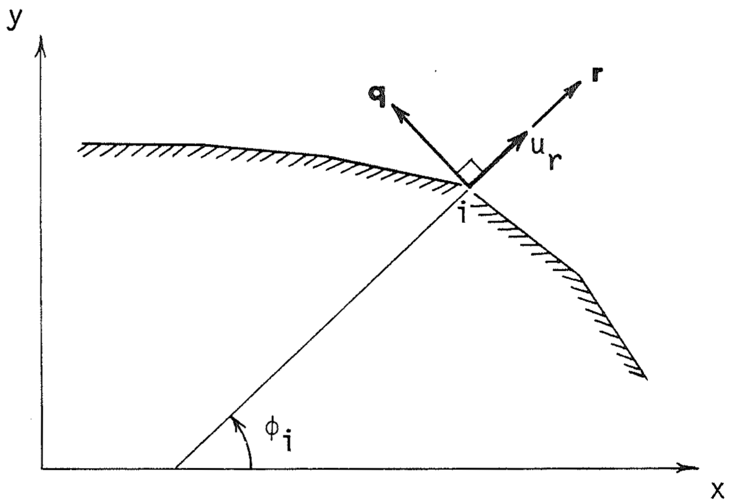


Fig. 3.5. To specify mixed boundary conditions.

2. **P** *and* **K** *are then modified according to:*†

(1) $\mathbf{P}_j \leftarrow \mathbf{P}_j - \mathbf{K}_{ji}\,\mathbf{R}^i\,\mathbf{u}^*_i, \qquad j \neq i,\ j = 1, 2, \ldots, n. \qquad (3.16)$

$\mathbf{K}_{ji} \leftarrow \mathbf{K}_{ji}\,\mathbf{R}^i\,\mathbf{E}, \qquad j \neq i,\ j = 1, 2, \ldots, n. \qquad (3.17)$

(2) $\mathbf{P}_i \leftarrow \mathbf{N}^*_i + \mathbf{E}\,\mathbf{R}^{i,T}\,(\mathbf{P}_i - \mathbf{K}_{ii}\,\mathbf{R}^i\,\mathbf{u}^*_i) + \mathbf{u}^*_i. \qquad (3.18)$

$\mathbf{K}_{ii} \leftarrow (\mathbf{E}\,\mathbf{R}^{i,T})\,\mathbf{K}_{ii}\,(\mathbf{R}^i\mathbf{E}) + \mathbf{G}. \qquad (3.19)$

(3) $\mathbf{K}_{ij} \leftarrow (\mathbf{E}\,\mathbf{R}^{i,T})\,\mathbf{K}_{ij}, \qquad j \neq i,\ j = 1, 2, \ldots, n. \qquad (3.20)$

After the system equations have been modified, the solution of the equations will yield

$$\mathbf{U}^*_i = \{\,u_{ri} \quad u_{qi}\,\} \tag{3.21}$$

which are oriented in the local axes defined by ϕ_i. The displacements $\mathbf{U}_i$ oriented in the global axes can be obtained by using (3.2).

4. Elastic Boundary Supports.

When a boundary is elastically supported, the stress resultants on the boundary are functions of the unknown nodal displacements along the boundary. If the stiffness coefficients of the elastic support are k_{xx}, k_{xy}, k_{yx}, and k_{yy}, then the boundary stresses N_x and N_y are given by

$$\begin{aligned} N_x &= k_{xx}(u^s - u) + k_{xy}(v^s - v), \\ N_y &= k_{yx}(u^s - u) + k_{yy}(v^s - v), \end{aligned} \tag{3.22}$$

where u^s and v^s are the specified displacements of the elastic support.

We now consider the elastic edge stresses along a side of length connecting two nodes i and j. Substituting (3.22) into (3.7), the generalized nodal forces at the two nodes can be expressed in the form

---

† $\mathbf{P}_j$ is modified before $\mathbf{K}_{ji}$ because $\mathbf{K}_{ji}$ on the right-hand sides are those *before* modification.

$$\begin{aligned} \mathbf{R}_i &= \mathbf{S}_{ii}(\mathbf{U}_i^s - \mathbf{U}_i) + \mathbf{S}_{ij}(\mathbf{U}_j^s - \mathbf{U}_j), \\ \mathbf{R}_j &= \mathbf{S}_{ji}(\mathbf{U}_i^s - \mathbf{U}_i) + \mathbf{S}_{jj}(\mathbf{U}_j^s - \mathbf{U}_j), \end{aligned} \qquad (3.23)$$

where

$$\mathbf{U}_k^s = \{ u_k^s \quad v_k^s \}, \qquad k = i,\ j. \qquad (3.24)$$

If k_{xx}, k_{xy}, k_{yx}, and k_{yy} are linear functions of s, then the 2 x 2 elastic stiffness matrices are given by

$$\begin{aligned} \mathbf{S}_{ii} &= \frac{l}{12}\left[\begin{array}{c|c} 3k_{xx}^i + k_{xx}^j & 3k_{xy}^i + k_{xy}^j \\ \hline 3k_{yx}^i + k_{yx}^j & 3k_{yy}^i + k_{yy}^j \end{array}\right], \\ \mathbf{S}_{ij} = \mathbf{S}_{ji} &= \frac{l}{12}\left[\begin{array}{c|c} k_{xx}^i + k_{xx}^j & k_{xy}^i + k_{xy}^j \\ \hline k_{yx}^i + k_{yx}^j & k_{yy}^i + k_{yy}^j \end{array}\right], \\ \mathbf{S}_{jj} &= \frac{l}{12}\left[\begin{array}{c|c} k_{xx}^i + 3k_{xx}^j & k_{xy}^i + 3k_{xy}^j \\ \hline k_{yx}^i + 3k_{yx}^j & k_{yy}^i + 3k_{yy}^j \end{array}\right], \end{aligned} \qquad (3.25)$$

where k_{xx}^i is the value of k_{xx} at node i, and so forth. It may be noted that if $k_{xy} = k_{yx}$, then $\mathbf{S}_{ii}$, $\mathbf{S}_{ij}$, and $\mathbf{S}_{jj}$ will be symmetric matrices.

In (3.23), the terms involving the unknown displacements must be transposed to the left-hand members of the system equations.

For every side (connecting nodes i and j) on elastic boundary support, the following modifications to the system equations are required:

1. *In* **K**:

$$\begin{aligned} \mathbf{K}_{ii} &\leftarrow \mathbf{K}_{ii} + \mathbf{S}_{ii}, \\ \mathbf{K}_{ij} &\leftarrow \mathbf{K}_{ij} + \mathbf{S}_{ij}, \\ \mathbf{K}_{ji} &\leftarrow \mathbf{K}_{ji} + \mathbf{S}_{ji}, \\ \mathbf{K}_{jj} &\leftarrow \mathbf{K}_{jj} + \mathbf{S}_{jj}. \end{aligned} \qquad (3.26)$$

2. *In* **P**:

$$\mathbf{P}_i \leftarrow \mathbf{P}_i + \mathbf{S}_{ii}\,\mathbf{U}_i^s + \mathbf{S}_{ij}\,\mathbf{U}_j^s,$$
$$\mathbf{P}_j \leftarrow \mathbf{P}_j + \mathbf{S}_{ji}\,\mathbf{U}_i^s + \mathbf{S}_{jj}\,\mathbf{U}_j^s. \tag{3.27}$$

5. Plate Bounded by an Edge Beam.

When a plate is bounded by an edge beam, the strain energy of the beam must be included in the total potential energy of the plate given by (1.23). The strain energy W^b of the beam takes the form

$$W^b = \frac{1}{2} \oint_s \left[EA(\varepsilon - \varepsilon^\circ)^2 + EI(\chi - \chi^\circ)^2 \right] ds, \tag{3.28}$$

where A is the cross-sectional area of the beam, I is the moment of area about the centroidal axis normal to the plane of the beam, and E is Young's modulus. Using piecewise linear displacements and a piecewise constant thermal strain, (3.28) can be expressed in the form

$$W^b = \frac{1}{2} \sum_i E_i A_i (\varepsilon_i - \varepsilon_i^\circ)^2 l_i + \frac{1}{8} \sum_k (E_k I_k + E_{k-1} I_{k-1})\,(\chi_k - \chi_k^\circ)^2\,(l_k + l_{k-1}), \tag{3.29}$$

where i refers to a boundary segment of length l_i, k refers to a boundary node, and the summations extend over all the segments and nodes along the boundary.

We now introduce the notation

$$s_i = \frac{\sin \phi_i}{l_i}, \tag{3.30}$$

$$c_i = \frac{\cos \phi_i}{l_i}, \tag{3.31}$$

$$\alpha_i = E_i A_i l_i, \tag{3.32}$$

$$\beta_k = \frac{1}{2} (l_k + l_{k-1}), \tag{3.33}$$

$$d_k = \frac{E_k I_k + E_{k-1} I_{k-1}}{2\beta}. \tag{3.34}$$

Substituting (3.4), (3.5), and (3.6) into (3.29) yields

$$\begin{aligned} W^b = \frac{1}{2} \sum_i \alpha_i \Big[- s_i u_{i+1} + s_i u_i + c_i v_{i+1} - c_i v_i - \varepsilon_i^\circ \Big]^2 \\ + \sum_k d_k \Big[- c_k u_{k+1} + c_k + c_{k-1} u_k - c_{k-1} u_{k-1} - s_k v_{k+1} \\ + s_k + s_{k-1} v_k - s_{k-1} v_{k-1} - \beta_k \chi_k^\circ \Big]^2. \end{aligned} \tag{3.35}$$

To simplify notation here, it is convenient to name the boundary nodes by consecutive integers beginning with 1 (Fig. 3.6). Examination of (3.35) reveals that the coefficient of a typical variable, say, u_3, is a linear combination of the variables u_1, u_2, u_3, u_4, and u_5. Thus, if we include W^b in Π in Eq. (2.29), the two sums

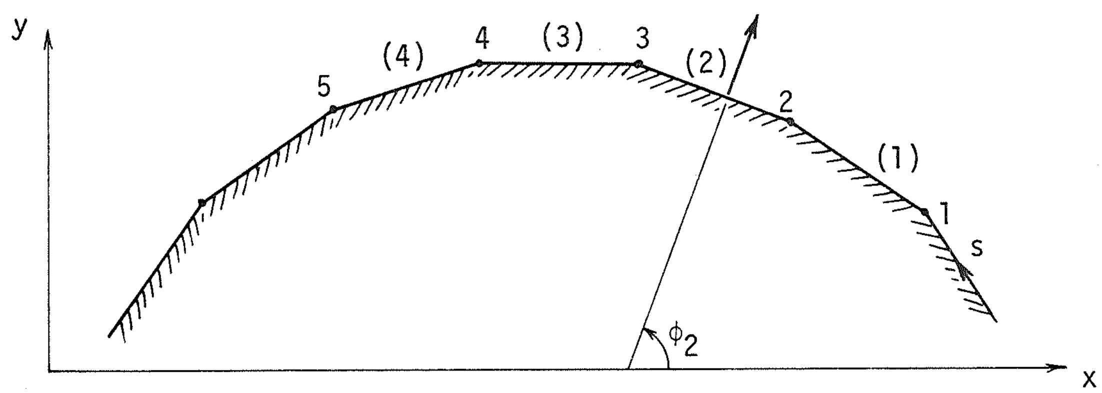


Fig. 3.6. Naming of nodes along an edge beam bounding a plate.

$$\sum_{m=1}^{5}\left(k_{3m}^{xx}\,u_m + k_{3m}^{xy}\,v_m\right),$$

$$\sum_{m=1}^{5}\left(k_{3m}^{yx}\,u_m + k_{3m}^{yy}\,v_m\right) \tag{3.36}$$

must be added to the left-hand members of (2.30a) and (2.30b), respectively, which corresponds to the equilibrium equations at node 3. To the right-hand members of the same equilibrium equations must be *subtracted* the quantities F°_{x3} and F°_{y3}, respectively. The quantities mentioned above are defined by

$$k_{31}^{xx} = d_2c_1c_2, \tag{3.37a}$$

$$k_{32}^{xx} = -\alpha_2s_2^2 - d_3c_2(c_2 + c_3) - d_2c_2(c_1 + c_2), \tag{3.37b}$$

$$k_{33}^{xx} = \alpha_2s_2^2 + \alpha_3s_3^2 + d_3(c_2 + c_3)^2 + d_2c_2^2 + d_4c_3^2, \tag{3.37c}$$

$$k_{34}^{xx} = -\alpha_3s_3^2 - d_3c_3(c_2 + c_3) - d_4c_3(c_3 + c_4), \tag{3.37d}$$

$$k_{35}^{xx} = d_4c_3c_4, \tag{3.37e}$$

$$k_{31}^{xy} = d_2s_1c_2, \tag{3.38a}$$

$$k_{32}^{xy} = \alpha_2s_2c_2 - d_3s_2(c_2 + c_3) - d_2c_2(s_1 + s_2), \tag{3.38b}$$

$$k_{33}^{xy} = -\alpha_2s_2c_2 - \alpha_3s_3c_3 + d_3(s_2 + s_3)(c_2 + c_3) + d_2s_2c_2 + d_4s_3c_3, \tag{3.38c}$$

$$k_{34}^{xy} = \alpha_3s_3c_3 - d_3s_3(c_2 + c_3) - d_4c_3(s_3 + s_4), \tag{3.38d}$$

$$k_{35}^{xy} = d_4s_4c_3, \tag{3.38e}$$

and

$$F^\circ_{x3} = \alpha_2 s_2 \varepsilon^\circ_2 - \alpha_3 s_3 \varepsilon^\circ_3 - d_3 \beta_3 (c_2 + c_3) \chi^\circ_3$$
$$+ d_2 \beta_2 c_2 \chi^\circ_2 + d_4 \beta_4 c_3 \chi^\circ_4, \tag{3.39a}$$

$$F^\circ_{y3} = - \alpha_2 c_2 \varepsilon^\circ_2 + \alpha_3 c_3 \varepsilon^\circ_3 - d_3 \beta_3 (s_2 + s_3) \chi^\circ_3$$
$$+ d_2 \beta_2 s_2 \chi^\circ_2 + d_4 \beta_4 s_3 \chi^\circ_4. \tag{3.39b}$$

The quantities k^{yy}_{3m} and k^{yx}_{3m}, m = 1, ..., 5 can be obtained from (3.37) and (3.38) by interchanging x and y, and s_n and c_n, n = 1, ..., 4.

We define the 2 x 2 edge beam stiffness matrices as

$$\mathbf{S}^{3m} = \begin{bmatrix} k^{xx}_{3m} & k^{xy}_{3m} \\ k^{yx}_{3m} & k^{yy}_{3m} \end{bmatrix}, \qquad m = 1, \ldots, 5. \tag{3.40}$$

We also define

$$\mathbf{F}^\circ_3 = \left\{ F^\circ_{x3} \quad F^\circ_{y3} \right\}. \tag{3.41}$$

Then, *for each typical node* 3,

1. *In row* 3 *of* $\mathbf{K}$:

$$\mathbf{K}_{3m} \leftarrow \mathbf{K}_{3m} + \mathbf{S}^{3m}, \qquad m = 1, \ldots, 5. \tag{3.42}$$

2. *In row* 3 *of* $\mathbf{P}$:

$$\mathbf{P}_3 \leftarrow \mathbf{P}_3 - \mathbf{F}^\circ_3. \tag{3.43}$$

Repeat the above two steps for all other nodes along the edge beam, using similar relations, with the subscript and superscript 3 *replaced by the node in question.*

6. Strain Boundary Conditions.

Extensional strain ε_s and in-plane curvature χ_s are specified along a portion of the plate boundary under strain boundary conditions. The strains are specified for each segment and are given by (3.4); and the curvatures are specified at each node and are given by (3.5) and (3.6).

Let m be the total number of nodes, including the end nodes, along the strain boundary portion; hence there are 2m unknown nodal displacements. One equation like (3.4) can be written for each of the m-1 segments, and one equation like (3.6) can be written for each of the m-2 nodes other than the end nodes. This results in a total of 2m-3 equations. The remaining three equations required to solve the strain boundary portion are supplied in *one* of two conditions:

First, three components of a rigid body motion of the boundary portion may be specified (i.e., two displacements at a node and the rotation of a segment).

Secondly, two force resultants and a moment about some point of the boundary forces acting on the boundary portion may be computed to provide three scalar equations.

It may be noted that the equations to be assembled for the strain boundary portion are compatibility equations or strain-displacement relations which are to replace the original equilibrium equations. This will result in certain rows being replaced without replacing the corresponding columns, and the coefficient matrix will become, in general, non-symmetric.

We now number the nodes along the strain boundary portion consecutively from 1 through m in the positive s-direction, with segment (i) following node i. Eq. (3.4) for segment (i) can be combined with (3.5) substituted in (3.6) for node i, and the result takes the form

$$- \mathbf{J}_{i-1} \mathbf{U}_{i-1} + (\mathbf{J}_{i-1} + \mathbf{H}_i) \mathbf{U}_i - \mathbf{H}_i \mathbf{U}_{i+1} = \mathbf{C}_i \tag{3.44}$$

where

$$\mathbf{J}_n = \frac{1}{l_n} \begin{bmatrix} 0 & 0 \\ \cos \phi_n & \sin \phi_n \end{bmatrix}, \tag{3.45}$$

$$\mathbf{H}_n = \frac{1}{l_n} \begin{bmatrix} \sin \phi_n & - \cos \phi_n \\ \cos \phi_n & \sin \phi_n \end{bmatrix}, \tag{3.46}$$

$$\mathbf{c}_i = \begin{Bmatrix} \varepsilon_i \\ \\ \frac{1}{2}\chi_i(l_i + l_{i-1}) \end{Bmatrix}. \tag{3.47}$$

For each of the m-2 nodes of the boundary portion other than the two end nodes, the original equilibrium equation is replaced by (3.44) written for that node. Consequently, 2m-4 scalar equations are obtained.

It may be noted that (3.4) written for segment (1) is not included in the above equations, and it can be written in the form

$$\mathbf{a}_1 \mathbf{U}_1 - \mathbf{a}_1 \mathbf{U}_2 = \varepsilon_1 l_1, \tag{3.48}$$

where

$$\mathbf{a}_i = \begin{bmatrix} \sin\phi_i & -\cos\phi_i \end{bmatrix}. \tag{3.49}$$

The remaining equations required to solve the strain boundary portion are now considered.

In the first case when three components of a rigid body motion of the boundary portion are specified, the two displacement components given for any node i are treated as in the case of displacement boundary conditions. The specified rotation ω_j for segment (j) is substituted into (3.5), yielding

$$\mathbf{b}_j \mathbf{U}_j - \mathbf{b}_j \mathbf{U}_{j+1} = \omega_j l_j, \tag{3.50}$$

where

$$\mathbf{b}_j = \begin{bmatrix} \cos\phi_j & \sin\phi_j \end{bmatrix}. \tag{3.51}$$

Eqs. (3.48) and (3.50) can be combined to replace the original equations for node 1.

In the second case, two force resultants and a moment about some point are to be computed. To obtain the two force resultants, we sum all the m matrix equations associated with the m nodes on the strain boundary. The two force resultants then appear on the right-hand member of the resulting matrix equation which is to replace the original equation for node 1. The operations can be represented by the relations

$$\mathbf{K}_{1j} \leftarrow \sum_{i=1}^{m} \mathbf{K}_{ij}, \qquad j = 1, 2, \ldots, n \tag{3.52}$$

$$\mathbf{P}_{1} \leftarrow \sum_{i=1}^{m} \mathbf{P}_{i}. \tag{3.53}$$

The moment about a point, say, node 1, of the boundary forces can be obtained by premultiplying each of the m matrix equations considered above by the matrix

$$\mathbf{d}_i = \begin{bmatrix} y_1 - y_i & x_i - x_1 \end{bmatrix}, \tag{3.54}$$

which contains the differences in coordinates between node 1 and node i. After the products are summed, the required moment appears on the right-hand member of the resulting scalar equation. This equation and (3.48) can be combined to replace the original equation of node m. The operations can be represented by the relations

$$\mathbf{K}_{m1} \leftarrow \sum_{i=1}^{m} \mathbf{D}_i \mathbf{K}_{i1} + \mathbf{A}_1, \tag{3.55}$$

$$\mathbf{K}_{m2} \leftarrow \sum_{i=1}^{m} \mathbf{D}_i \mathbf{K}_{i2} - \mathbf{A}_1, \tag{3.56}$$

$$\mathbf{K}_{mj} \leftarrow \sum_{i=1}^{m} \mathbf{D}_i \mathbf{K}_{ij}, \qquad j = 3, 4, \ldots, n, \tag{3.57}$$

$$\mathbf{P}_m \leftarrow \begin{Bmatrix} \sum_{i=1}^{m} \mathbf{d}_i \mathbf{P}_i \\ \varepsilon_1 l_1 \end{Bmatrix}, \tag{3.58}$$

where

$$\mathbf{D}_i = \begin{bmatrix} y_1 - y_i & x_i - x_1 \\ 0 & 0 \end{bmatrix} \tag{3.59}$$

$$\mathbf{A}_i = \begin{bmatrix} 0 & 0 \\ \sin \phi_i & -\cos \phi_i \end{bmatrix} \tag{3.60}$$

The algorithm of modification for strain boundary conditions can be summarized as follows:

Case 1. A rigid body motion is specified.

1. *Replace original equation for node* 1 *by* (3.48) *and* (3.50).
2. *Replace original equations for other nodes by* (3.44).
3. *Treat specified displacements at a node as in displacement boundary conditions.*

Case 2. No rigid body motion is specified.

1. *Replace original equation for node* 1 *by applying* (3.52) *and* (3.53).
2. *Replace original equations for nodes* 2, 3, ..., m-1 *by* (3.44).
3. *Replace original equation for node* m *by applying* (3.55), (3.56), (3.57), *and* (3.58).

3.5. Modification for Boundary Conditions in Bending.

The boundary conditions in bending considered in this section are: displacement, stress, mixed, stress function, and standard boundary conditions. Stretching-bending duality can be applied in the algorithm of modification for boundary conditions in stretching developed in the previous section.

1. Displacement Boundary Conditions.

The quantities to be specified for a boundary portion with displacement boundary conditions are the nodal displacement w and the slope $w_{,n}$ of the plate edge in the direction of the outward normal $\mathbf{n}$. In the finite element method, only the average value of $w_{,n}$ along a side need be specified. The average value of the slope $w_{,s}^{(i)}$ in the s-direction for side (i) of length l is given by

$$w_{,s}^{(i)} = (w_j - w_i)/l \tag{3.61}$$

where nodes i and j are connected by side (i). The components $w_{,x}$ and $w_{,y}$ of the average edge slope in the global coordinate system may be obtained from $w_{,n}$ and $w_{,s}$ by relations similar to (3.2). Explicitly, the relations are

$$\begin{aligned} w_{,x} &= w_{,n} \cos \phi_i - w_{,s} \sin \phi_i, \\ w_{,y} &= w_{,n} \sin \phi_i + w_{,s} \cos \phi_i. \end{aligned} \tag{3.62}$$

The generalized nodal rotations due to edge slope may be computed through equations dual of (3.7), in which the curvatures $w_{,xs}$ and $w_{,ys}$ are required. Through integration by parts, however, they can be computed directly from $w_{,x}$ and $w_{,y}$ and the results are

$$\begin{aligned} R'_{xi} &= - w_{,y}^{i} + w_{,y}^{(i)}, \\ R'_{yi} &= w_{,x}^{i} - w_{,x}^{(i)}, \\ R'_{xj} &= w_{,y}^{j} - w_{,y}^{(i)}, \\ R'_{yj} &= - w_{,x}^{j} + w_{,x}^{(i)}, \end{aligned} \tag{3.63}$$

where the superscripts k and (k) denote the average quantities at node k and side (k), respectively.

In the system equations, submatrices

$$\begin{aligned} \mathbf{R}'_i &= \{ R'_{xi} \quad R'_{yi} \}, \\ \mathbf{R}'_j &= \{ R'_{xj} \quad R'_{yj} \} \end{aligned} \tag{3.64}$$

are added to $\mathbf{P}'_i$ *and* $\mathbf{P}'_j$*, respectively, for every side along the boundary with specified displacement boundary condition.*

2. Stress Boundary Conditions.

In stress boundary conditions, edge stress couple M_n and edge effective shear Q_{ne} are specified for a portion of the boundary. From these values and the particular solution, we obtain

$$\begin{aligned} M_n^* &= M_n - M_n^p, \\ Q_{ne}^* &= Q_{ne} - Q_{ne}^p, \end{aligned} \tag{3.65}$$

where M_n^* and Q_{ne}^* are dual of ε_s and χ_s, respectively, in the stretching problem. The algorithm for modification of the system equations is exactly the same as strain boundary conditions in stretching.

The quantities U_i, V_i, and Ω_j are quantities dual of a rigid body motion in stretching and may be specified for the stress boundary portion.

The equations dual of (3.4), (3.5), and (3.6) take the form

$$M_i^* l_i = -(U_{i+1} - U_i)\sin\phi_i + (V_{i+1} - V_i)\cos\phi_i, \tag{3.66}$$

$$\Omega_i l_i = -(U_{i+1} - U_i)\cos\phi_i - (V_{i+1} - V_i)\sin\phi_i, \tag{3.67}$$

$$Q_i^* = \frac{2(\Omega_i - \Omega_{i-1})}{l_i + l_{i-1}}, \tag{3.68}$$

where M_i^* is M_n^* for side (i) and Q_i^* is Q_{ne}^* at node i.

3. Mixed Boundary Conditions.

In mixed boundary conditions, one stress function component and a curvature component in the same direction are prescribed. The notation for directions of specified quantities (Fig. 3.5) and the algorithm for modification of the system equations are exactly the same as its dual in stretching.

4. Stress Function Boundary Conditions.

If the stress functions $\mathbf{U}_i^!$ are prescribed at node i, then there are two less unknown nodal stress functions. The algorithm for modification of the equations are exactly the same as that under displacement boundary conditions in stretching.

5. Standard Boundary Conditions.

Several standard boundary conditions which are special cases of the previous boundary conditions are presented here.

SIMPLE SUPPORT:

Simple support is a special case of mixed boundary conditions. Displacement component w and stress couple M_n are both zero along the boundary. With w zero along the s-direction, χ_s is also zero. We take the particular solution functions K_x and K_y which are zero along the boundary. Since K_s and K_n are then zero by transformation, M_n^p becomes zero by using (1.33). Boundary conditions for the homogeneous problem which is dual of the stretching problem are obtained as follows:

From (3.65), $M_n^* = 0$. With $U_{s,s} = M_n^*$, U_s is constant and is taken, for convenience, to be zero. Since

$$\chi_s^* = \chi_s + K_n,$$

χ_s^* becomes zero. Thus, the required boundary conditions are that both U_s and χ_s^* are zero along the boundary.

LINE OF SYMMETRY:

The line of symmetry boundary results when there is symmetry in geometry and loading. By using the symmetry boundary, only half or a quarter of a plate need be solved. The symmetry boundary is a special case of mixed boundary conditions.

Along the line of symmetry, the normal slope $w_{,n}$ and effective shear Q_{ne} are both zero, which leads to zero the curvature χ_{ns}. We take the particular solution functions which results in $K_{s,n}$ and $K_{n,n}$ both being zero.

Boundary conditions for the homogeneous problem which is dual of the stretching problem are obtained in the form

$$\chi^*_{ns} = \chi_{ns} = 0$$

and

$$U_{n,ss} = Q_{ne} - D(K_{s,n} + \nu K_{n,n}) = 0.$$

To eliminate the quantity dual of a rigid body motion in stretching, we take U_n as zero, for convenience. Thus, the required boundary conditions are that both U_n and χ^*_{ns} are zero along the boundary.

FREE:

This is a special case of stress boundary conditions in which both M_n and Q_{ne} are zero.

FIXED SUPPORT:

This is a special case of displacement boundary conditions in which both w and $w_{,n}$ are zero.

CHAPTER 4

COMPUTER IMPLEMENTATION OF THE PLANAL SYSTEM

4.1. Introduction.

The dual finite element method described in the previous chapters is implemented into a system employing a large scale digital computer. This computer system which is described in the remainder of this work is called the PLANAL System, representing the Plate Analysis Language. The scope of the system is limited to solutions of plate problems in stretching and bending.

The PLANAL System is developed as a subsystem of the Integrated Civil Engineering System (ICES) at the Department of Civil Engineering, Massachusetts Institute of Technology. Externally, an ICES subsystem consists of a series of commands, which serve as communication links between a user and the subsystem. Internally, each command is processed by a command interpreter which calls translation programs (Command Definition Blocks, or CDBs) written in the Command Definition Language (CDL). A CDB in turn calls computer programs (subroutines) written in ICETRAN (ICES FORTRAN) which is a FORTRAN-based, procedure-oriented language. The subroutines finally perform the intended tasks in the system. A complete description of ICES, CDL, and ICETRAN may be found in [15,16,21].

A number of advantages result in developing the PLANAL System in ICES. The input commands are formed in a free, problem-oriented style, using vocabulary already familiar to the user (Chapter 5). The features of dynamic memory allocation (DMA) does not limit the size of a problem (e.g., the maximum number of nodes) that can be handled by the system. Finally, related programs are formed into units called load modules; thus, efficient use of the core of a computer may be realized by bringing into core only those modules which are necessary for the current computation.

The organization and sequence of operations of PLANAL are similar to those of STRUDL [17,18] which is another ICES subsystem. The system is also partially based on works by Nagy [19] and Ferrante [11].

4.2. Organization and Sequence of Operations.

After PLANAL has been initialized as a subsystem of ICES, addresses for COMMON variables are assigned for transmitting data between CDBs and ICETRAN programs. A COMMON map is included in Appendix C. Most of the arrays and scalars used in PLANAL are COMMON variables, and are described briefly in the COMMON map. A more detailed description of many of these variables is presented in Appendix D. The subroutines in PLANAL are organized into 19 load modules. Documentations of the load modules and subroutines are given in Appendices E and F, respectively. A complete listing of the CDBs and ICETRAN programs is included in Appendix G.

The sequence of operations in PLANAL is illustrated in Fig. 4.1. Each operation calls for one or more load modules, and each load module may be called more than once under different aliases (which are also entry points to a module).

1. Data Input.

   Topology of the plate to be analyzed is processed by Load Modules STINCI and STEJPR. Informations on element properties, boundary conditions, and loadings are then processed (STHGEN, STHINI).

2. Finite Element Analysis.

   When all input data has been provided, the FINITE ELEMENT ANALYSIS command is issued by the user. Control is then transferred

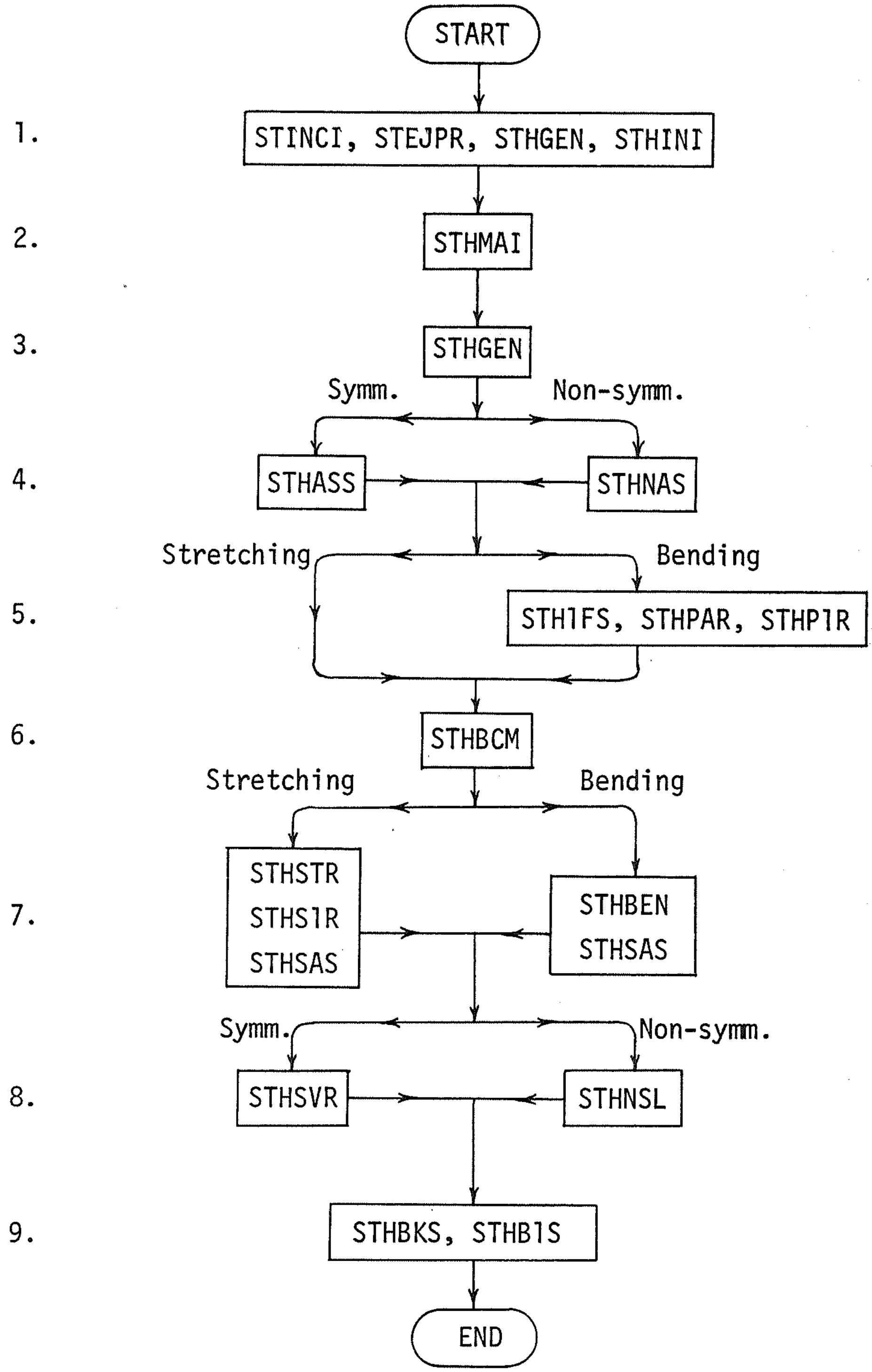


Fig. 4.1. Sequence of operations in PLANAL.

to STHMAI which sets up subsequent operations.

3. Generation of Local Coefficient Matrices.

   Local coefficient matrices for all the elements are generated by Load Module STHGEN.

4. Assembly of the Global Coefficient Matrix.

   The global coefficient matrix is assembled from the local coefficient matrices according to the connectivity of the nodes. Depending on the symmetry of the global coefficient matrix, either STHASS or STHNAS is used.

5. Bending Particular Solution.

   In the bending problem, when particular solution functions are not provided, standard PLANAL procedure will be used for their construction (STH1FS, STHPAR, STHP1R).

6. Management of Modification of System Equations.

   After the global coefficient matrix has been assembled, the right-hand members of the system equations are modified for the loading (STHBCM). STHBCM also controls the calling sequence of the processing of different boundary conditions along the plate boundary.

7. Boundary Conditions Modifications.

   The system of equations is modified according to existing boundary conditions. Different load modules are called depending on whether the problem is one of stretching or one of bending (STHSTR, STHS1R, STHBEN, STHSAS).

8. Solution of the System Equations.

   The unknowns (displacements or stress functions) are solved from the modified system equations by calling the proper load module (STHSVR, STHNSL).

9. Back-substitution.

Quantities related to the unknowns can be computed by back-substitution after the unknowns have been solved (STHBKS, STHB1S).

4.3. Information for Installation of PLANAL.

ICES contains a number of subsystems and a *basic* system that controls all the subsystems. The operation of a subsystem is independent of any other subsystems. Since the PLANAL System is a part of ICES, any execution or modification of PLANAL will require the use of the basic system of ICES itself.

For development, modification and execution of PLANAL or any ICES subsystem, the "ICES/360 Basic System and Language Processors," a package of basic system programs, is required. For execution only, the "ICES/360 Basic System," a subset of the above, is needed. (The sole distributer of ICES programs is the IBM Corporation, and the Program Order Numbers of the above two packages are 360D 16.2.005 and 360D 16.2.004, respectively.)

Because of interface requirements during development, ICES at present operates only in an IBM Operating System/360 environment. PLANAL requires as a minimum machine an S.360 Model 40, with a 128K-byte core, two 2311 disk drives (or their equivalent), and input/output devices. The above packages with their proper documentation may be obtained from IBM Corporation by writing: IBM Corporation, Program Information Department, 40 Saw Mill River Road, Hawthorne, New York 10532, U.S.A.

The PLANAL System as described here has not been released to the public. Further information on PLANAL and ICES may be obtained from: Headquarters, Department of Civil Engineering, Room 1-290, 77 Massachusetts Avenue, Cambridge, Massachusetts 02139, U.S.A.

CHAPTER 5

USER'S MANUAL OF THE PLANAL SYSTEM

5.1. Introduction.

The user's manual in this chapter provides a complete description of all the commands in PLANAL, the Plate Analysis Language. The commands in the PLANAL System are written in a problem-oriented style that is easily recognizable and does not require a fixed format.

Input information describing a problem to be solved is supplied to the PLANAL System through a set of commands. Each command is interpreted by a language processor, called the command interpreter. Control is ultimately transferred to the appropriate subroutines in the PLANAL System to perform the intended task. By suitably assembling a set of commands, a user can solve a problem using the PLANAL System.

5.2. Capabilities of the System.

At present, the analysis capabilities of the PLANAL System fall into two catagories: plate stretching problems, and plate bending problems.

Plate Stretching. In the plate stretching problem, the system can analyze a plate of arbitrary shape, variable thickness and material properties, and under arbitrary in-plane loading. The boundary conditions available are those of displacement, stress, mixed, elastic support, edge beam, and strain.

Plate Bending. In the plate bending problem, when there is no lateral loading, or when there is a lateral loading and corresponding particular solution functions are supplied, the system can analyze a plate of arbitrary shape, and variable thickness and material properties.

When there is a lateral loading but no particular solution functions are supplied, the present system will construct appropriate particular solution functions only if certain requirements in geometry and loading are satisfied. The plate must be of rectangular shape, and uniform thickness and material properties. The loading is restricted to one which varies linearly in two orthogonal directions x and y. This load function q is expressible in the form

$$q = c_1x + c_2y + c_3,$$

where c_1, c_2, and c_3 are arbitrary constants. (Uniformly distributed loads and hydrostatic loads are examples of this form of loading.) The system can also analyze the case of a concentrated lateral force applied at the intersecting point of the lines of symmetry of the plate.

The boundary conditions available are those of displacement, stress, and mixed. The same boundary conditions listed under simple support, fixed support, free, and symmetry are also available.

5.3. Format of Commands.

All commands in PLANAL have a free format in the sense that there are no requirements for certain information to appear in certain prescribed columns in an input card. However, the following rules must be observed in preparing input for PLANAL:

1. All 80 columns of a card may be used.
2. Embedded blanks in words are not allowed.
3. Where one blank is required, several may be used.
4. The first character on a card can be placed in any column.
5. If more than one card is needed to complete a command, continuation

cards are allowed. To continue a command, a minus sign preceded by at least one blank is placed on the card to be continued. (The minus sign is to be the last character typed on that card.)

Example:

1 3 4 5 9 14 THICKNESS 1.0 EX 30000000.0 -

EY 30000000.0 PX 0.25 PY 0.25 G 12000000.0

6. Comments may be interspersed among the commands at the user's discretion. The card columns after a $ sign preceded by at least one blank are available for user's comments. Cards with a $ sign in card column 1 are likewise available for comments.

   Example:

   $ THIS IS A UNIFORMLY LOADED PLATE.

7. All alphameric *data* must be placed between single quotes, ' '. Words such as NODE COORDINATES, ELEMENT, or THICKNESS are in the standard vocabulary of PLANAL and are not data; therefore, they must not be placed between single quotes.

   Example:

   ELEMENT PROPERTIES TYPE 'CST'

8. If data items in a command are supplied in the order specified, no labels need be used. If a label is used with any data item in a command, all *succeeding* data items for that command must be labeled. For example, in the NODE COORDINATES command,

   1 X 10. Y 20.

   1   10.   20.

   1   10. Y 20.

   1 Y 20. X 10.

   are all acceptable forms (here, X and Y are labels). But

   1 X 10.   20.

   is not acceptable to the system.

5.4. Convention.

Throughout the remainder of this chapter, certain notational conventions will be followed in describing the commands.

Underlined Characters. In the command description, characters which are underlined are necessary symbols for identification by the command interpreter and must appear in the commands. Other characters or words listed in the command but not underlined may be included for clarity or otherwise omitted. For example, in the TYPE specification command (Section 5.6),

TYPE PLATE STRETCHING
TYPE STRETCH
TYP STR

provide the same information for the system.

Mode of Data. Data are either real, integer, or alphameric as designated. A real data item requires a decimal point while an integer data item does not. An alphameric data item consists of one or more characters each of which can be either a letter or a numeral. In the command description, real and integer quantities are designated by v and n, respectively, with identifying subscripts. Words placed between single quotes shown in the form of a command are the only data that must be alphameric.

Names and Lists. The names of nodes, elements or boundaries may be integer or alphameric. Some of the commands require a node name list or an element name list. A node name list may consist of the name of a single node, or the names of a number of nodes. If the names of the nodes are consecutive integers n_1, n_1+1, ..., n_2, then the list may be supplied in the form n_1 TO n_2. When a name is alphameric, it must be enclosed by single quotes. The conventions for an element name list are the same as for a node name list.

Example:

4 THICKNESS 1.0
3 TO 11 THICKNESS 1.0
2 7 'AZ' THICKNESS 1.0

However, when node names (not node name list) are indicated, the names of one or more (up to ten) nodes can be specified, but the option of

n_1 TO n_2 is no longer available.

Brackets and Braces. In the commands, square brackets [ ] and the information they contain are to be replaced by the appropriate input form representing the information required. Braces { } are used to indicate where *choices* are available in the input.

5.5. Preparation of Input.

PLANAL commands can be classified into ten groups. Each group provides a certain type of information and is made up of one or more input cards. The ten groups are:

1. Problem initiation,
2. Type specification,
3. Unit declaration,
4. Geometry and topology,
5. Element properties specification,
6. Boundary condition specification,
7. Loading specification,
8. Particular solution functions for the bending problem,
9. Output and analysis commands,
10. Termination statement.

It is recommended that the order of groups of commands as given above should be followed in describing a problem, although certain minor variations are acceptable. (For a comparison with the details of input to a parallel system STRUDL, the STRUDL User's Manual [18] may be consulted.)

All the above groups of commands except Groups 3, 7, and 8 must be supplied before a problem can be solved in the PLANAL System. If standard units (Section 5.6) are assumed, unit declaration in Group 3 can be omitted. When there are no loadings, Group 7 can be neglected. Group 8 is excluded from the input commands in the stretching problem or in the bending problem when particular solution functions are unknown.

5.6. Description of Commands.

The ten groups of PLANAL commands are now described in detail in this section. Examples are included where appropriate.

1. Problem Initiation.

PLANAL ['name'] ['title']

The word PLANAL signifies the beginning of a new problem to be solved by the system. The 'name' is an alphameric name chosen by the user to identify his problem. It must be enclosed in single quotes and may have a maximum length of eight characters. The 'title' is *optional* (may be omitted); it contains the title of the problem or any other comments, and may have a maximum length of 64 characters.

Example:

PLANAL 'U44LSSL1' 'S.S.SQ. PLATE, 25 NODES, 32 ELEMENTS.'

The following two commands are *optional* and are placed, if used, after the problem initiation card. They are usually not included in a normal PLANAL execution job.

DEBUG { ALL / COMMON }

PLDEBUG

When certain system errors are detected during execution, processing of the problem in the computer will be interrupted. A DEBUG ALL command will cause the listing (dump) of the entire core of the computer at the time of interruption. A DEBUG COMMON command will cause the listing of the COMMON area of the core. The DEBUG command is useful only for system debugging.

The PLDEBUG command causes the printing of the names of all the important subroutines whenever they are called by the system. It is useful if the calling sequence of subroutines is desired.

Example:

PLANAL 'EXAMPLE'

DEBUG COMMON

PLDEBUG

2. Type Specification.

$$\underline{\text{TYP}}\text{E} \quad \left\{ \begin{array}{l} \text{PLATE } \underline{\text{STR}}\text{ETCHING} \\ \text{PLATE } \underline{\text{BEND}}\text{ING} \end{array} \right\} \quad \left\{ \begin{array}{l} \text{leave blank} \\ \underline{\text{SYM}}\text{METRICAL} \\ \underline{\text{NON}}\text{SYMMETRICAL} \end{array} \right\}$$

This command is used to specify the type of the problem in question. The types available at the present are PLATE STRETCHING and PLATE BENDING.

The form of the stiffness coefficient matrix in the system equation may be symmetrical or non-symmetrical depending on the types of boundary conditions involved. If the matrix for a particular problem is symmetrical, the user may specify the SYMMETRICAL form, or (perhaps for comparison) the NONSYMMETRICAL form, which is also acceptable to the system in this case. However, by leaving the last item blank, the correct form of symmetry will be automatically selected by the system.

Example:

TYPE PLATE BENDING

3. Unit Declaration.

$$\underline{\text{UNI}}\text{TS} \quad \left\{ \begin{array}{l} \text{[length unit]} \\ \text{[force unit]} \\ \text{[angular unit]} \\ \text{[temperature unit]} \\ \text{[time unit]} \end{array} \right\}$$

The UNITS command specifies the units of input data following the statement and designates the units for output. The command is optional (may be omitted) and may be used any number of times in the same problem. If they are not specified, the units assumed as *standard* in the five unit types are inches, pounds, radians, Fahrenheit, and seconds, respectively. The following are the available units for the system:

Length unit:	$\underline{\text{IN}}\text{CHES}$, $\underline{\text{FE}}\text{ET}$, $\underline{\text{FT}}$, $\underline{\text{CENTIM}}\text{ETERS}$, $\underline{\text{CM}}$, or $\underline{\text{M}}\text{ETERS}$,
Force unit:	$\underline{\text{POU}}\text{NDS}$, $\underline{\text{LB}}$, $\underline{\text{KIP}}\text{S}$, $\underline{\text{TON}}\text{S}$, $\underline{\text{KIL}}\text{OGRAMS}$, $\underline{\text{KG}}$, or $\underline{\text{MTON}}$,
Angular unit:	$\underline{\text{RAD}}\text{IANS}$, or $\underline{\text{DEG}}\text{REES}$,
Temperature unit:	$\underline{\text{FAH}}\text{RENHEIT}$, or $\underline{\text{CENTIG}}\text{RADE}$,
Time unit:	$\underline{\text{SEC}}\text{ONDS}$, $\underline{\text{MIN}}\text{UTES}$, or $\underline{\text{HOU}}\text{RS}$.

Any unit types not given in a UNITS command are assumed to remain unchanged from those previously specified (or the standard values).

Example:

UNITS FEET KIPS FAH

UNITS SEC RADIANS INCHES LB

4. Geometry and Topology.

NODE COORDINATES

[node name] X [v_x] Y [v_y] $\left\{ \begin{array}{l} \text{BOUNDARY} \\ \text{leave blank} \end{array} \right\}$

.
.
.

ELEMENT INCIDENCES

[element name] [node name 1] [node name 2] [node name 3]

.
.
.

BOUNDARY INCIDENCES

.
.
.

The NODE COORDINATES command specifies the coordinates of each node with respect to an arbitrarily chosen right-handed global frame. The xy-plane is to be taken as the plane of the plate. The x- and y-coordinates are designated by v_x and v_y, respectively, and they can be supplied in any order. When no labels (i.e., X,Y) are given, the values are assumed to be given in the order of v_x and v_y. For nodes which are on the boundary, the letter B is required to be placed after the coordinate values.

The ELEMENT INCIDENCES command specifies the connectivity of the elements. The nodes of an element must be given in a direction sweeping from the positive x-axis to the positive y-axis, i.e., in a counter-clockwise order with respect to a right-handed reference axes (Fig. 5.1). The first node given can be any node of the element. For example, the element incidence for element 5 in Fig. 5.1 may be specified in one of

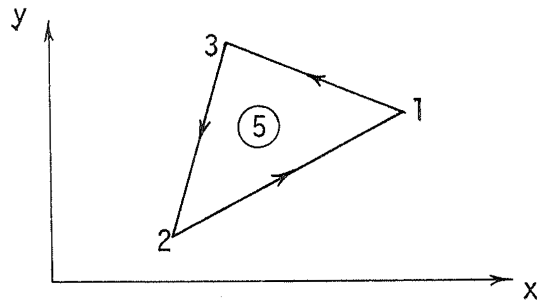


Fig. 5.1. Order for specifying element incidence.

the following forms:

5 1 3 2

5 3 2 1

5 2 1 3

The BOUNDARY INCIDENCES command is used to assign names to the boundaries of the plate for subsequent identification. Node name may be the name of any node located on the boundary being named. A boundary is defined here as a completely closed path bounding the plate. There are more than one boundaries bounding a plate with interior openings. This command causes the chain of boundary nodes for each boundary to assembled. Thus it must be used after the NODE COORDINATES and ELEMENT INCIDENCES commands, but before any boundary conditions are specified.

Node names, element names, and boundary names may be either integer or alphameric. In the case of alphameric identification, the name must be enclosed in single quotes.

Examples:

NODE COORDINATES

3 X 3.75 Y 0.00 BOUNDARY

'N7' Y 0.5 X -1.50 B

4 3.75 1.00 B

ELEMENT INCIDENCES

10 8 7 14

'E2' '3N' 14 'A2'

BOUNDARY INCIDENCES

1 1

'BOUNDARY' 32

5. Element Properties Specification.

ELEMENT PROPERTIES TYPE ['type']

[element name list] THICKNESS [v_t] EX [v_{ex}] EY [v_{ey}] -

PX [v_{px}] PY [v_{py}] CTX [v_{cx}] CTY [v_{cy}] G [v_g] DENSITY [v_d]

.
.
.

The thickness and material properties of the elements are specified in this command and can be given in any order. The type of elements to be used in the problem is specified in 'type', and the only type that can be used at present is 'CST', representing Constant Strain Triangle element. The element name list may consist of the name of a single element, or a group of elements. The list may be replaced by the word ALL (no quotes) if all elements of the plate have the same properties. The variables, which are to be real, have the following meaning:

v_t = average thickness of the element,

v_{ex} = Young's modulus in the x-direction,

v_{ey} = Young's modulus in the y-direction,

v_{px} = Poisson's ratio in the x-direction,

v_{py} = Poisson's ratio in the y-direction,

v_{cx} = thermal expansion coefficient in the x-direction,

v_{cy} = thermal expansion coefficient in the y-direction,

v_g = shear modulus,

v_d = material density of the element.

When v_{ey} is not given, it is assumed to be v_{ex}; when v_{py} is not listed, it is equated to $v_{px}v_{ex}/v_{ey}$. When the other variables are not given, they will be taken as zero.

Example:

ELEMENT PROPERTIES TYPE 'CST'

1 TO 16 TH 1.0 EX 30000000.0 PX 0.25 DEN 0.3 G 12000000.0

ALL TH 1.0 EX 10000000.0 PX 0.25 DEN 0.1 G 4000000.0

6. Boundary Condition Specification.

BOUNDARY CONDITION ['boundary name'] type

[boundary portion] [quantity 1] [v_1] [quantity 2] [v_2] ...

.
.
.

This command specifies the boundary conditions on all boundaries of the plate. The 'boundary name' is the name of the particular boundary for which boundary values are tabulated. Again, a boundary is defined as a completely closed path bounding the plate.

The type of boundary condition can be one of the following: In stretching: DISPLACEMENT, STRESS, MIXED STRETCHING, ELASTIC, EDGE BEAM, and STRAIN; in bending: DISPLACEMENT, STRESS, FUNCTION, MIXED BENDING, SIMPLE SUPPORT, FIXED SUPPORT, FREE, and SYMMETRY. Different appropriate boundary quantities are to be specified for different types of boundary conditions, and they are listed together on the following pages. Only one type of boundary condition can appear in one BOUNDARY CONDITION command. For a boundary with more than one types of boundary conditions, several BOUNDARY CONDITION commands will be required, and the order in which they are supplied is immaterial.

The positive s-direction along a boundary is taken to be the positive sense along the boundary. When one traverses in the positive s-direction along a boundary, the normal vector outward from the plate points to the right of the boundary. In the right-handed Cartesian coordinate system adopted here, this direction is counter-clockwise for an exterior boundary, and clockwise for an interior boundary.

Boundary portion defines the boundary nodes and/or the element edges between boundary nodes that have the prescribed boundary values.

There are four forms of boundary portions to accommodate various situations of specifying boundary values:

1. [node name] <u>POS</u>ITIVE [quantity 1] [v_1] . . .
2. [node name] <u>NEG</u>ATIVE [quantity 1] [v_1] . . .
3. [node name] [quantity 1] [v_1] . . .
4. [node name 1] <u>TO</u> [node name 2] [quantity 1] [v_1] . . .

In the *first* form, the values v_1, ... specified are the limiting values approached from the positive side of [node name]. In the *second* form, the values specified are the limiting values approached from the negative side of [node name]. The first and second forms of boundary portion allow discontinuous boundary values to be specified. When the values at the positive and negative sides of a node are the same, the *third* form can be used. In the *fourth* form, the *same* nodal values are assigned to [node name 1] POSITIVE, to [node name 2] NEGATIVE, and to all intermediate nodes along the boundary between [node name 1] and [node name 2], traversed in the positive sense. When the values of a quantity at the two end nodes of an element edge are given, linear variation of that quantity along the edge, wherever applicable, is assumed. The following example illustrates the use of the four forms of boundary portions.

Example 5.1. Consider a rectangular plate subjected to distributed boundary stresses as shown in Fig. 5.2. The plate is divided into elements and the ten nodes are named as shown. The positive sense of the boundary goes from node 10 to node 9, and so forth. The boundary is named 'B1' and the boundary condition is that of stress. Boundary condition for the complete boundary can be specified by the following statements:

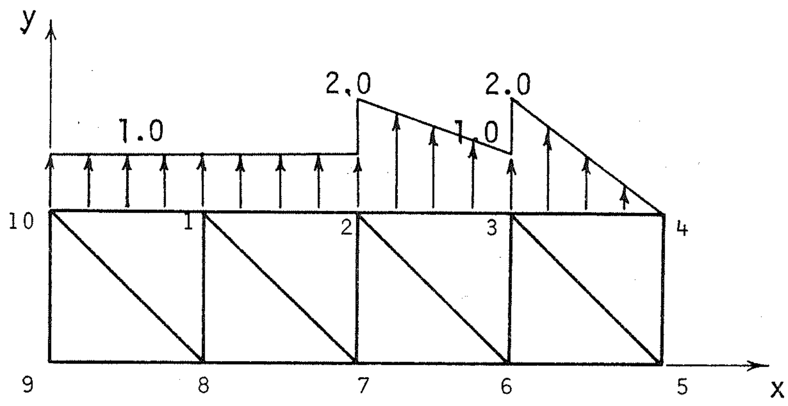


Fig. 5.2. Example to illustrate forms of boundary portion.

BOUNDARY CONDITION 'B1' STRESS
4 POS NY 0.0
3 NEG NY 2.0
3 POS NY 1.0
2 NEG NY 2.0
2 TO 10 NY 1.0
10 TO 4 NY 0.0

To illustrate an alternate form, the second last card above can be replaced by three cards:

2 POS NY 1.0
1 NY 1.0
10 NEG NY 1.0

It should be noted that 1 TO 2 involves the complete boundary *except* the side between nodes 1 and 2. 1 TO 1 implies that the same boundary values are specified for the complete boundary.

Each type of boundary condition requires certain boundary quantities for its complete description. In the command format, boundary quantities are designated by quantity 1, quantity 2, etc. The boundary quantities can be specified in any order. Quantities not specified will be taken as zero unless stated otherwise. Whenever components of vectors are indicated, they are taken with respect to the *global frame* unless stated otherwise. The types of boundary conditions with their associated boundary quantities are described below:

(1) Type: DISPLACEMENT

Quantities: U [v_u] V [v_v] W [v_w] R [v_r]

v_u and v_v are the x- and y-components of the displacement for all nodes along the specified boundary portion, and are to be entered for the plate stretching problem.

v_w is the z-component of nodal displacement, and v_r is the edge rotation $\beta = - w_{,n}$. They are to be entered in the plate bending problem.

(2) Type: STRESS

Quantities: NX [v_{nx}] NY [v_{ny}] Q [v_q] M [v_m] ROTATION [v_r]

In the stretching problem, v_{nx} and v_{ny} are the x- and y-components of the edge stress resultant (force/unit length) along the boundary portion. The values specified are nodal values, and linear variations of these values are assumed between nodes.

In the bending problem, v_q is the z-component of the edge effective shear Q^*_{ne} (force/unit length), and v_m is the edge stress couple M^*_{nn} (bending moment/unit length) whose vector is oriented in the positive s-direction. If the quantity dual to rotation in stretching is known, it can be specified in v_r. If v_r is not specified, it will not be automatically taken as zero. (See Type (6) for use of v_r.)

(3) Type: MIXED STRETCHING

Quantities: UR [v_{ur}] NR [v_{nr}] ANGLE [v_a]

This boundary condition is applicable to the stretching problem only. v_{ur} is the nodal displacement in the r-direction in the plane of the plate (Fig. 5.3). v_{nr} is the edge stress resultant (force/unit length) in the direction perpendicular to, and $\pi/2$ radians ahead of, the r-direction. And v_a is the positive (counterclockwise) angle from the positive x-axis to the r-direction.

If v_{ur} is not specified it would not be taken as zero. The ends of a boundary portion of MIXED STRETCHING may be adjacent to a boundary portion of DISPLACEMENT or STRESS in stretching. By not specifying v_{ur} at such end nodes, v_{ur} will either take on the value specified under DISPLACEMENT or be determined by the governing system of simultaneous equations.

(4) Type: <u>ELAST</u>IC

Quantities: <u>US</u> [v_{us}] <u>VS</u> [v_{vs}] <u>KXX</u> [v_{kxx}] <u>KXY</u> [v_{kxy}] -
<u>KYX</u> [v_{kyx}] <u>KYY</u> [v_{kyy}]

In this command, which is available for the stretching problem only, v_{kxx}, v_{kxy}, v_{kyx}, v_{kyy} are the elastic constants of the elastic support, v_{us}, v_{vs} are the x- and y-components of support displacement.

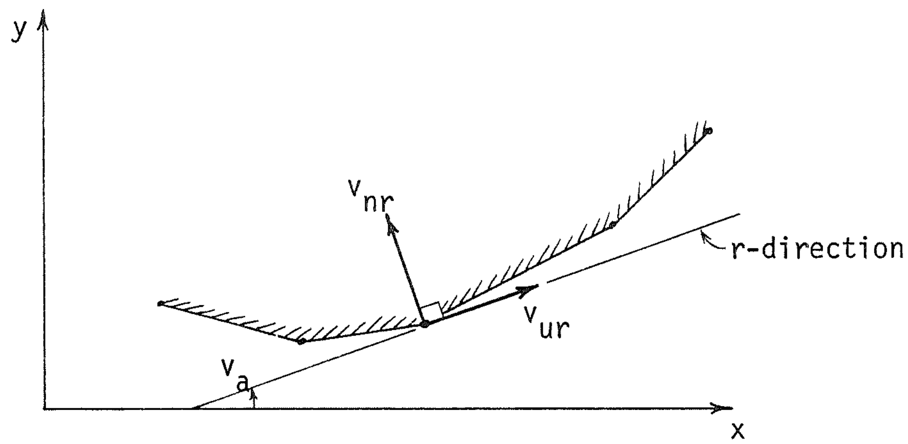


Fig. 5.3. Notation for mixed boundary condition.

(5) Type: EDGE BEAM

Quantities: NX [v_{nx}] NY [v_{ny}] EB [v_e] AB [v_a] IZ [v_i]

This command is available for the stretching problem only. The quantities v_{nx} and v_{ny} are the x- and y-components of the edge stress resultant (force/unit length) applied along the edge beam within the specified boundary portion. The values specified are nodal values, and linear variations of these values are assumed between nodes. v_e is the Young's modulus of the beam material in the direction of the beam, and v_i is the moment of area about a centroidal axis in the z-direction.

When edge beam is specified anywhere along a boundary, the *entire* boundary must be specified as an edge beam. Dummy portions of the edge beam can be effected by taking v_e, v_a, and v_i as zero.

(6) Type: STRAIN

Quantities: EPSILON [v_e] CHI [v_c] ROTATION [v_r]

This boundary condition is applicable to the stretching problem only. It is used when the extensional strain v_e of a boundary segment and the curvature v_c at the junction of two adjacent boundary segments are known along a boundary portion. If the rotation v_r is not specified, it will not be automatically taken as zero. Internally, v_c specified at a node is taken as the specified curvature of the boundary at that *node*. The quantities v_e and v_r specified at a node are taken as the extensional strain and rotation, respectively, of the *segment* following that node in a positive s-direction.

It should be noted that the purpose of specifying rotation of a segment together with the specifying of displacements of a node is to fix a rigid body displacement of the plate considered. Such a rigid body displacement can be specified uniquely only once. Therefore, when the rotation of one segment along a boundary portion is specified, the displacements at one of the nodes along that boundary portion must also be specified through a DISPLACEMENT command.

In the case when an entire boundary is of STRAIN boundary condition, a special condition exists. Let there be n boundary nodes (therefore n segments) along the boundary. The strains along only n-1 segments and the curvatures at only n-2 nodes need be specified, in addition to the necessary specification of a rigid body displacement (3 quantities). That the above specification is sufficient can be verfied by the fact that (n-1) + (n-2) + 3 = 2n, which is equal to the 2n unknown displacements along the boundary (two displacements at each of the n nodes).

(7) Type: FUNCTION
Quantities: U [v_u] V [v_v]

In this boundary condition, which is applicable to plate bending problems only, stress functions U and V are specified. It is dual of the DISPLACEMENT boundary condition in stretching.

(8) Type: MIXED BENDING
Quantities: UR [v_{ur}] CHI [v_c] ANGLE [v_a]

In this boundary condition, which is applicable to plate bending problems only, the quantities dual of those in MIXED STRETCHING boundary condition are specified. The quantities are stress function v_{ur}, curvature v_c and angle v_a. (See Type (3).)

(9) Type: SIMPLE SUPPORT
Quantities: None.

This command is available for the bending problem only. Internally, the system changes this boundary condition to that of MIXED BENDING, assigning a constant value to the s-component of the stress function vector along the specified boundary portion.

(10) Type: FIXED SUPPORT
Quantities: None.

This command is available for the bending problem only. Internally, the system changes this boundary condition to that of DISPLACEMENT, equating to zero the displacements and rotations along the specified boundary portion.

(11) Type: FREE

Quantities: None.

This command is available for the bending problem only. Internally, the system changes this boundary condition to that of STRESS, equating to zero the edge effective shear and the edge stress couple along the specified boundary portion.

(12) Type: SYMMETRY

Quantities: None.

This command is available for the bending problem only. Internally, the system changes this boundary condition to that of MIXED BENDING, assigning a linear function U_n to the n-component of the stress function vector along the specified boundary portion. In the case of a distributed load (limited to linear functions of x and y), U_n will be a constant. This command can be applied only to a line of symmetry in both *geometry* and *loading*.

Example 5.2. As an example to illustrate the combination of some boundary conditions commands, consider a rectangular plate in stretching subjected to boundary stresses as shown in Fig. 5.4. The boundary is named 'EXTERIOR'. The prescribed displacements are u = v = 0 at nodes 3 and 4; u = 0 at nodes 7 and 8. The boundary conditions indicated can be specified thus:

```
BOUNDARY CONDITION 'EXTERIOR' STRESS
1 TO 2 NX -1. NY  1.
2 TO 3 NX  1. NY -1.
4 TO 5 NX  1. NY -1.
5 TO 6 NX  1. NY  1.
BOUNDARY CONDITION 'EXTERIOR' DISPLACEMENT
3 TO 4 U 0. V 0.
```

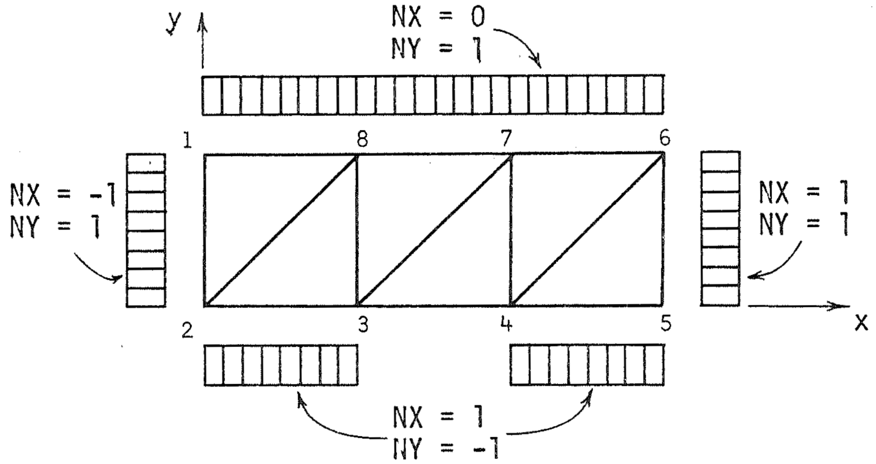


Fig. 5.4. Example to illustrate the use of BOUNDARY CONDITION command.

BOUNDARY CONDITION 'EXTERIOR' MIXED STRETCHING

6 POS NR 1. ANG 0.

7 UR 0. NR 1. ANG 0.

8 UR 0. NR 1. ANG 0.

1 NEG NR 1. ANG 0.

It may be noted that in the last and fourth last cards above, UR is not specified.

7. Loading Specification.

LOADING

$\left\{ \begin{array}{l} \underline{\text{NOD}}\text{ES [node names]} \\ \underline{\text{UNI}}\text{FORM} \end{array} \right\} \left\{ \begin{array}{l} \underline{\text{INT}}\text{ENSITY} \\ \underline{\text{FOR}}\text{CE} \end{array} \right\} \underline{X}\ [v_x]\ \underline{Y}\ [v_y]\ \underline{Z}\ [v_z]$

.

.

.

The LOADING command specifies the loading applied to the plate. If there are no loadings, this command must be ignored completely. The node names may be the name of a single node having the specified values, or may be a list of nodes (up to ten nodes) having the same specified values. If the loading at all the nodes are identical, the word UNIFORM can be used. The load vector can be either an intensity or a concentrated force. The three components of the load vector are specified by v_x, v_y, and v_z. Components not specified will be taken as zero.

Example:

```
LOADING
NODES 1 2 3 4 5 INTENSITY X 1.0 Y 2.0
NODES 6         INTENSITY   1.5   2.0
NODES 7 8 9 10  INT       Y 2.0 X 1.5
```

In the bending problem, the present version of the system can process a lateral load intensity only if it is linear in x and y. Such a loading is defined uniquely if the load intensity is specified at three non-collinear points. This form of specifying such a loading is the only form acceptable to the system.

Example:

```
LOADING
NODE  1 INTENSITY Z -1.0
NODE  5 INTENSITY Z -3.0
NODE 14 INTENSITY Z -2.5
```

In the above example, nodes 1, 5, and 14 must be non-collinear. If the three points defining the loading are collinear, an error message will be issued by the system.

If the loading is a uniform load, the following is an acceptable form:

```
LOADING
UNIFORM INTENSITY Z 1.0
```

If the loading is a concentrated force applied at the intersecting point of two lines of symmetry, the acceptable form is:

```
LOADING
NODE 3 FORCE Z 50.0
```

8. Particular Solution Functions for the Bending Problem.

BENDING PARTICULAR SOLUTION

NODES [node names] KX [K_x] KY [K_y] KXX [$K_{x,x}$] KYY [$K_{y,y}$]

.

.

.

This command is applicable only to the bending problem in which the particular solution functions K_x and K_y or their derivatives $K_{x,x}$ and $K_{y,y}$ are known. The node names may be the name of a single node having the specified values, or may be the names of several nodes (up to ten nodes) having the same specified values.

Example:

BENDING PARTICULAR SOLUTION
NODES 1 4 9 KX 0.0 KY -0.08736
NODES 2 3 8 KX 0.0 KY -0.08190

If particular solution functions are unknown in the bending problem, standard functions will be constructed by summing a Fourier series, provided certain limitations in geometry and loading are met (see Section 5.2). In such a case, and when particular solution functions are not applicable, this command must be ignored completely.

9. Output and Analysis Commands.

$$\text{OUTPUT} \begin{Bmatrix} \text{NODES} \\ \text{ELEMENTS} \end{Bmatrix} \text{ quantities}$$

FINITE ELEMENT ANALYSIS

Once all data required to perform an analysis have been supplied, the output and analysis command can be issued.

Output can be computed at the nodes, at the elements, or both. If output at both the nodes and elements is requested, two separate OUTPUT commands designating NODES and ELEMENTS will be required. The quantities to be printed are different in the stretching and bending problems:

quantities in the stretching problem	quantities in the bending problem
DISPLACEMENTS	FUNCTIONS
STRAINS	MOMENTS
STRESSES	CURVATURES
PRINCIPAL STRAINS	PRINCIPAL MOMENTS
PRINCIPAL STRESSES	PRINCIPAL CURVATURES
ALL	ALL

ALL denotes that all quantities will be printed. When principal values (such as strains or moments) are required, the principal direction is also computed. The direction is computed as the angle swept from the positive x-axis to the direction of the *major* principal value in the positive (counter-clockwise) sense. The ranges of that angle are from 0 to $\pi/2$ radians and from $3\pi/2$ to 2π radians.

Example:

OUTPUT NODES DISPLACEMENTS PRINCIPAL STRESSES
OUTPUT ELEMENTS ALL

Note. If quantities at a node are required, grid lines parallel to the axes are passed through all the nodes to effect differentiations with respect to x and y. (For example, strains are derivatives of displacements.) When a line in the grid pattern is formed by only one node, the approximation to a derivative at that node cannot be made, and that derivative is taken to be zero. For *such* nodes, quantities listed in the output are thus invalid.

The analysis command must be the last card describing any one problem to be analyzed.

Example:

FINITE ANALYSIS

For the purpose of understanding the internal working of the PLANAL System, a user may wish to print out certain arrays used in the process of analysis. These intermediate print-outs can be effected through the use of a number of control parameters in the analysis command described above. (These parameters were frequently used during development of the system.) The *modified* command format when intermediate print-outs are also required is:

FINITE ELEMENT ANALYSIS <u>K1</u> [k_1] <u>K2</u> [k_2] <u>K3</u> [k_3] <u>K4</u> [k_4] <u>K5</u> [k_5] <u>K6</u> [k_6] - <u>K7</u> [k_7] <u>K8</u> [k_8] <u>K9</u> [k_9] <u>K10</u> [k_{10}]

Any control parameters can be supplied, and in any order. They have the following meaning:

$k_1 = 1$ means to print global stiffness matrices before boundary condition modification (symmetric: KDIAG, KOFDG, KPPRI; non-symmetric: FCMAT, IREL1, ICUREL, KPPRI).

$k_2 = 1$ means to print global stiffness matrices after boundary condition modification.

$k_3 = 1$ means to print BDCOND before boundary condition modification.

$k_4 = 1$ means to print BDCOND after boundary condition modification.

$k_5 = 1$ means to print ELSTMT.

$k_6 \geqslant 1$ means to print KPPRI at each step of solver.

$k_6 \geqslant 2$ means to print KPPRI, FCMAT, ICUINT at each step of solver (applicable only to non-symmetric coefficient matrices).

$k_7 = 0$ means that K_x, K_y are to be used in forming KPPRI.

$k_7 = 1$ means that K_x, K_y are not to be used in forming KPPRI.

$k_7 \geqslant 2$ means that $K_{x,x}$, $K_{y,y}$ are to be used in forming KPPRI.

$k_7 = 3$ means that K_x, K_y are to be used in boundary correction of particular solution.

$k_8 = 1$ means that particular solution functions are computed by double integration with c as a function of x and y.

$k_8 = 2$ means that particular solution functions are computed by double integration with c = 0.5.

$k_9 \geqslant 1$ means to print PBSOLN as assembled by system.

k_9 = 2 means to print PBNTEM, KPBSLN, GRIDPR whenever applicable.

k_{10} = 1 means to compute load function by Fourier series and print result.

For a description of the arrays listed, see Appendix D.

Example:

FINITE ELEMENT ANALYSIS K2 1 K10 1

10. Termination Statement.

FINISH

This command requests control to exit from the ICES System of which PLANAL is a subsystem. Therefore, the FINISH command must be placed after all the cards describing a problem to be analyzed. If there are more than one problem to be analyzed (requiring more than one FINITE ELEMENT ANALYSIS commands), the cards describing each problem must be stacked together, and then *one* FINISH card is placed after the combined deck (Fig. 5.5).

A summary of all the PLANAL commands can be found in Appendix B.

5.7. Formation of Input Deck.

An input deck of cards submitted to a computer for execution must contain a number of control cards in addition to the PLANAL commands that describe the problem to be solved. These control cards are usually written in a job control language (JCL). Initial control cards are placed before the PLANAL commands and final control cards are placed at the end (Fig. 5.5). These control cards provide information for job identification, accounting, and setting up the proper program libraries for execution. Information to be supplied on these cards depend on the computer configuration at a particular organization and must be determined by that organization.

Listed here are the control cards for using the PLANAL System at the Information Processing Center, the Massachusetts Institute of Technology at the time when this work was prepared.

Initial Control Cards:

```
// SMITH
/*MITID PROB=M1234,PROG=5678
/*SRI   DEFER
/*MAIN  TIME=3,LINES=2
/*SETUP DDNAME=PACK16,UNIT=2314,ID=(234016,,SAVE),A=QFM
//JOBLIB DD DSNAME=ICES.LINKLIB,DISP=OLD,VOLUME=(PRIVATE,RETAIN)
// DD DSNAME=ICES.HO,DISP=OLD,VOLUME=(PRIVATE,RETAIN)
// DD DSNAME=ICES.MODULES.STRUDL2,DISP=OLD,VOLUME=(PRIVATE,RETAIN)
// EXEC ICES
//GO.SYSIN DD *
```

Final Control Card:

```
/*
```

```
Initial Control Cards

PLANAL 'PROB1'
.
.
.
FINITE ELEMENT ANALYSIS
.
.                              one or more
PLANAL 'PROBN'                 PLANAL problems
.
.
.
FINITE ELEMENT ANALYSIS

FINISH

Final Control Cards
```

Fig. 5.5. Formation of input deck.

CHAPTER 6

APPLICATIONS OF THE PLANAL SYSTEM

6.1. Introduction.

Examples of applications of the PLANAL System to both the stretching and bending problems are presented in this chapter. Sample input cards for problems in the examples are listed to illustrate the use of various PLANAL commands, especially the boundary condition and loading specifications.

Nodes should be numbered consecutively in such a pattern that the difference between the node numbers of any two *adjacent* nodes should be as small as possible. In this way, the band widths of non-zero entries in the coefficient matrix of the system equations (2.40) may be minimized. When the nodes of a plate form a rectangular grid pattern, they should be numbered consecutively in the direction parallel to the *short* side (see Example 6.1). Proper numbering of nodes may save computation time in solving the system equations by as much as three times or more.

Samples of output from the PLANAL System are also presented. The boundaries in all the examples are named 'BOUND' in the input.

6.2. Examples in Stretching.

Four examples are included here to illustrate combinations of different boundary conditions in stretching problems.

Example 6.1. *Tension Specimen*. Consider a homogeneous, isotropic, long plate of constant thickness t with dimensions as shown in Fig. 6.1.

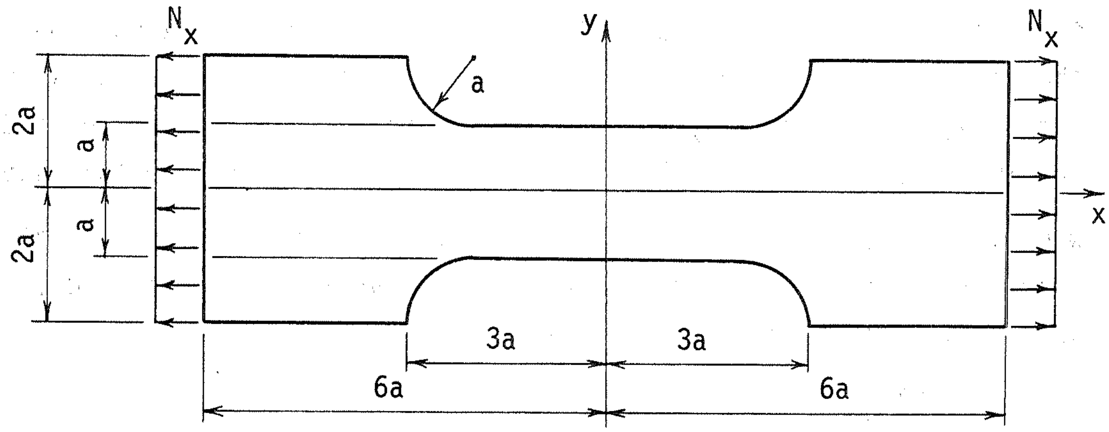


Fig. 6.1. Dimensions and loading of a tension specimen.

It is subjected to a tensile stress N_x applied at the ends. Taking advantage of symmetry, we need to analyze only the portion of the plate in the first quadrant which is discretized into triangular elements in Fig. 6.2. It can be noted that at the region where the sample narrows, a denser grid is used. The nodes are numbered consecutively along the shorter grid lines. We now illustrate the PLANAL input for the case when a = 2 in., t = 1 in., E = 10^5 psi, ν = 0.3, and N_x = 1 lb/in.

A sample of input cards to the PLANAL System for the tension specimen problem is shown in Fig. 6.3 (some cards for NODE COORDINATES and ELEMENT INCIDENCES which are similar to the ones shown have been omitted). The x- and y-axes are lines of symmetry along which displacements v and u, respectively, are suppressed. These lines are specified under a boundary condition of MIXED STRETCHING. Moreover, since u = v = 0 at node 1, this node is specified under a boundary condition of DISPLACEMENT. Hence, at node 1 POSITIVE and node 1 NEGATIVE, UR is not specified under MIXED STRETCHING. ∎

Example 6.2. *Circular Disk Subjected to Compressive Forces.* Consider a circular disk of radius a subjected to a pair of diametrically opposite compressive forces P (Fig. 6.4a). Theoretical expressions for the stresses may be found in Timoshenko and Goodier [23]. Because of symmetry, we analyze only the disk in the first quadrant, which is discretized into triangular elements in Fig. 6.4b. We analyze the case when a = 1 in., $E = 10^5$ psi, $\nu = 0.3$, and P = 1 lb. The portion of the input cards for the problem pertaining to boundary condition and loading specifications are shown below:

```
BOUNDARY CONDITION 'BOUND' DISPLACEMENT
1  U 0.0 V 0.0
BOUNDARY CONDITION 'BOUND' MIXED STRETCHING
 1 POS          NR 0.0 ANGLE 1.5707963
 7 NEG   UR 0.0 NR 0.0 ANGLE 1.5707963
 7 TO 28 UR 0.0 NR 0.0 ANGLE 1.5707963
 6 TO 2  UR 0.0 NR 0.0 ANGLE 0.0
 2 POS   UR 0.0 NR 0.0 ANGLE 0.0
 1 NEG          NR 0.0 ANGLE 0.0
```

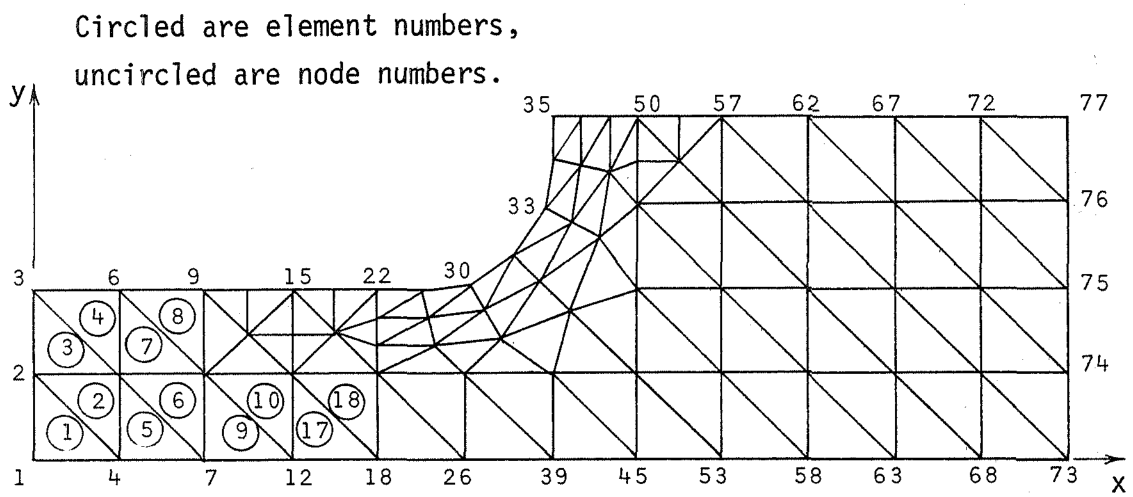


Fig. 6.2. Discretization of a quarter of the tension specimen.

```
PLANAL 'TENSION' 'TENSION SPECIMEN.'
DEBUG COMMON
PLDEBUG
TYPE PLATE STRETCHING
NODE COORDINATES
 1  0.   0. B
 2  0.   1. B
 3  0.   2. B
 4  1.   0. B
 5  1.   1.
 6  1.   2. B
 7  2.   0. B
 8  2.   1.
 9  2.   2. B
10  2.5 1.5
11  2.5 2.0 B

ELEMENT INCIDENCES
  1  1  4  2
  2  4  5  2
  3  3  2  5
  4  5  6  3
  5  5  4  7
  6  7  8  5
  7  5  8  6
  8  8  9  6
  9  7 12  8
 10 12 13  8
 11  8 13 10
 12  8 10  9

BOUNDARY INCIDENCE
'BOUND' 1
ELEMENT PROPERTIES TYPE 'CST'
ALL THICK 1. EX 100000. PX 0.3 G 38461.538
BOUNDARY CONDITION 'BOUND' DISPLACEMENT
1  U 0.0 V 0.0
BOUNDARY CONDITION 'BOUND' MIXED STRETCHING
 1 POS          NR 0.0 ANGLE 1.5707963
 4 NEG   UR 0.0 NR 0.0 ANGLE 1.5707963
 4 TO 73 UR 0.0 NR 0.0 ANGLE 1.5707963
 3 POS   UR 0.0 NR 0.0 ANGLE 0.0
 2       UR 0.0 NR 0.0 ANGLE 0.0
 1 NEG          NR 0.0 ANGLE 0.0
BOUNDARY CONDITION 'BOUND' STRESS
73 TO 77 NX 1.0 NY 0.0
77 TO  3 NX 0.0 NY 0.0
OUTPUT NODES ALL
OUTPUT ELEMENTS ALL
FINITE ELEMENT ANALYSIS
```

Fig. 6.3. PLANAL input cards for the tension specimen problem.

```
BOUNDARY CONDITION 'BOUND' STRESS
28 TO  6 NX 0.0 NY 0.0
LOADING
NODE 6 FORCE Y -0.5
```

A sample of the output from the PLANAL System is shown in Fig. 6.5. Theoretical and PLANAL results are compared in Fig. 6.6. ■

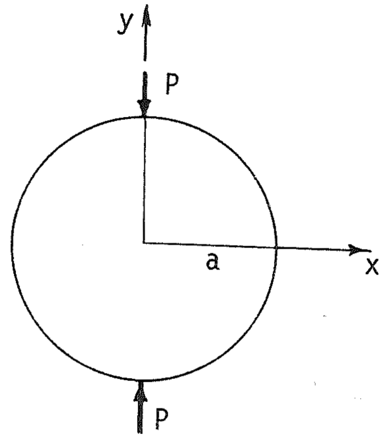
Example 6.3. *Beam on Elastic Foundation.* A beam of length 2a and depth b rests on air elastic foundation and is subjected to a distributed load N_y over a length of 2c as shown in Fig. 6.7a. We analyze only half the beam (Fig. 6.7b) because of symmetry. We consider the case when a = 16 in., b = 1 in., c = 4 in., E = 10^5 psi, ν = 0.3, and $N_y = 10^4$ lb/in. If we take the stiffness coefficients of the elastic foundation as K_{xx} = $K_{xy} = K_{yx} = K_{yy}$ 3 x 10^4 lb/in./in., then the input cards for boundary condition are:

```
BOUNDARY CONDITION 'BOUND' ELASTIC
1 TO 33 KXX 30000.0 KXY 30000.0 KYX 30000.0 KYY 30000.0
BOUNDARY CONDITION 'BOUND' STRESS
33 TO 10 NY 0.0
10 TO  2 NY -10000.0
BOUNDARY CONDITION 'BOUND' MIXED STRETCHING
2 TO 1 UR 0.0 NR 0.0 ANGLE 0.0
```

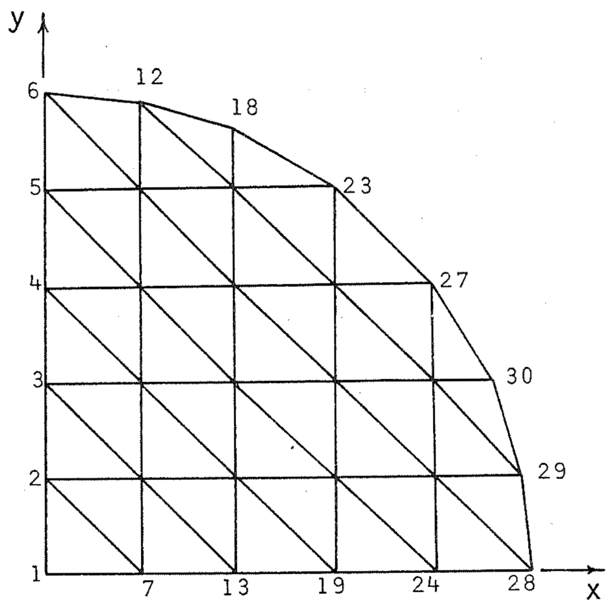
The shape of the deformed beam is shown in Fig. 6.7c. ■

Example 6.4. *Rectangular Plate with Edge Beam.* A homogeneous, isotropic plate considered as a deep beam is simply supported as shown in Fig. 6.8a. Its lower edge is attached to an edge beam of cross-sectional area A_b, and its upper edge is subjected to a distributed load p. Because of symmetry, we analyze only half the plate (Fig. 6.8b). The behavior of the plate is dependent on the ratio of Young's Moduli for the plate and the edge beam, denoted by E_p and E_b, respectively. We now consider the case when a =12 in., b = 8 in., A_b = 0.955 in.^{2}, E_p = 10^5 psi, E_b = 3 x 10^6 psi, ν = 0, and p = 1 lb/in.

In preparing input cards for the problem, displacements at the sup-



a. Dimensions and loading.



b. Discretization of a quarter disk.

Fig. 6.4. Circular disk subjected to compressive forces.

```
PLANAL 'DISK' 'DISK WITH CONCENTRATED FORCES.'

          ***********************************************
          *                                             *
          *                 ICES PLANAL                 *
          *         THE PLATE ANALYSIS LANGUAGE         *
          *                                             *
          *               MODIFICATION  0               *
          *              SEPTEMBER, 1969                *
          *                                             *
          *         2:52:16          8/21/69            *
          *                                             *
          ***********************************************
TYPE PLATE STRETCHING

NODE COORDINATES

 1  0.0  0.0 R

 [illegible]
        ****  ****  RESULTS  ****  ****

 NODAL DISPLACEMENTS

    NODE              U              V

     1             0.0            0.0
     2             0.0           -0.2246E-05
     3             0.0           -0.4755E-05
     4             0.0           -0.7885E-05
     5             0.0           -0.1232E-04
     6             0.0           -0.1944E-04
     7             0.1195E-05    -0.3917E-12
                   [illegible]   -0.1917E-05
** GRID PATTERN FOR DIFFERENTIATION.

    LINES PARALLEL TO X-AXIS.

         6 NODES.     1   7  13  19  24    28
         6 NODES.     2   8  14  20  25    29
         [illegible]

    LINES PARALLEL TO Y-AXIS.

         6 NODES.     1   2   3   4   5     6
         6 NODES.     7   8   9  10  11    12
         [illegible]

* NOTE *

     WHEN A LINE IN THE GRID PATTERN FOR DIFFERENTIATION (SEE ABOVE) IS FORMED
BY ONE NODE, THE APPROXIMATION TO A DERIVATIVE AT THAT NODE CANNOT BE MADE.
THAT DERIVATIVE IS TAKEN TO BE ZERO.  FOR SUCH NODES, QUANTITIES LISTED BELOW
ARE THUS INVALID.


NODAL STRAINS AND PRINCIPAL STRAINS

   NODE          EX            EY         GAMMA-XY*          E1           E2               THETA-1 (X TO E1)

    1        0.6595E-05   -0.1058E-04   -0.2161E-11     0.6595E-05   -0.1058E-04    6.283 RAD  =  359 D 59 M 59.12 S
    2        0.7101E-05   -0.1189E-04    0.1040E-05     0.7115E-05   -0.1190E-04    0.027 RAD  =    1 D 34 M  4.83 S
    3        0.7886E-05   -0.1410E-04    0.3012E-05     0.7989E-05   -0.1420E-04    0.068 RAD  =    3 D 54 M  3.23 S
    4        0.9065E-05   -0.1892E-04    0.7835E-05     0.9603E-05   -0.1945E-04    0.137 RAD  =    7 D 49 M 16.57 S
             0.8118E-05   -0.2888E-04    0.2177E-04     0.1108E-04   -0.3184E-04    0.266 RAD  =   15 D 14 M  7.44 S
                                                        0.0          -0.4228E-04    0.0   RAD  =    0 D  0 M  0.0  S
                                    *GAMMA-XY = 2 EXY                               0.007 RAD  =    0 D 25 M 10.89 S
                                                                                                             27.17 S
NODAL STRESSES AND PRINCIPAL STRESSES

   NODE          SX            SY           SXY            S1           S2               THETA-1 (X TO S1)

    1        0.3760E 00   -0.9450E 00   -0.8312E-07     0.3760E 00   -0.9450E 00    6.283 RAD  =  359 D 59 M 59.12 S
    2        0.3884E 00   -0.1072E 01    0.4001E-01     0.3895E 00   -0.1073E 01    0.027 RAD  =    1 D 34 M  4.83 S
    3        0.4019E 00   -0.1289E 01    0.1158E 00     0.4098E 00   -0.1297E 01    0.068 RAD  =    3 D 54 M  3.23 S
    4        0.3725E 00   -0.1780E 01    0.3013E 00     0.4139E 00   -0.1821E 01    0.137 RAD  =    7 D 49 M 16.57 S
    5       -0.6004E-01   -0.2906E 01    0.8372E 00     0.1680E 00   -0.3134E 01    0.266 RAD  =   15 D 14 M  7.44 S
    6       -0.1394E 01   -0.4646E 01    0.0           -0.1394E 01   -0.4646E 01    0.0   RAD  =    0 D  0 M  0.0  S
                                                        0.2797E 00   -0.8535E 00    0.007 RAD  =    0 D 25 M 10.89 S
ELEMENT STRAINS AND PRINCIPAL STRAINS

  ELEMENT        EX            EY         GAMMA-XY*          E1           E2               THETA-1 (X TO E1)

    1        0.5976E-05   -0.1123E-04   -0.1958E-11     0.5976E-05   -0.1123E-04    6.283 RAD  =  359 D 59 M 59.12 S
    2        0.6216E-05   -0.9585E-05    0.1887E-05     0.6272E-05   -0.9641E-05    0.059 RAD  =    3 D 24 M 17.30 S
    3        0.6216E-05   -0.1254E-04    0.1647E-05     0.6252E-05   -0.1258E-04    0.044 RAD  =    2 D 30 M 32.37 S
    4        0.6504E-05   -0.1001E-04    0.4467E-05     0.6801E-05   -0.1031E-04    0.132 RAD  =    7 D 34 M  7.02 S
    5        0.6504E-05   -0.1565E-04    0.4179E-05     0.6699E-05   -0.1585E-04    0.093 RAD  =    5 D 20 M 25.20 S
    6        0.6647E-05   -0.1064E-04    0.9333E-05     0.7826E-05   -0.1182E-04    0.247 RAD  =   14 D 10 M 49.58 S
    7        0.6647E-05   -0.2218E-04    0.9190E-05     0.7362E-05   -0.2289E-04    0.154 RAD  =    8 D 50 M 29.25 S
                                         [illegible]    0.9193E-05   -0.1492E-04    0.454 RAD  =   25 D 59 M 43.74 S
ELEMENT STRESSES AND PRINCIPAL STRESSES

  ELEMENT        SX            SY           SXY            S1           S2               THETA-1 (X TO S1)

    1        0.2864E 00   -0.1037E 01   -0.7532E-07     0.2864E 00    0.0           6.283 RAD  =  359 D 59 M 59.12 S
    2        0.3671E 00   -0.8484E 00    0.7257E-01     0.3909E 00   -0.1383E-01    0.059 RAD  =    3 D 24 M 17.30 S
    3        0.2696E 00   -0.1173E 01    0.6334E-01     0.2838E 00   -0.1414E-01    0.044 RAD  =    2 D 30 M 32.37 S
    4        0.3848E 00   -0.8855E 00    0.1718E 00     0.4503E 00   -0.6555E-01    0.132 RAD  =    7 D 34 M  7.02 S
    5        0.1987E 00   -0.1506E 01    0.1607E 00     0.2883E 00   -0.8961E-01    0.093 RAD  =    5 D 20 M 25.20 S
    6        0.3796E 00   -0.9504E 00    0.3590E 00     0.5958E 00   -0.2163E 00    0.247 RAD  =   14 D 10 M 49.58 S
```

Fig. 6.5. A sample of PLANAL output for the circular disk problem.

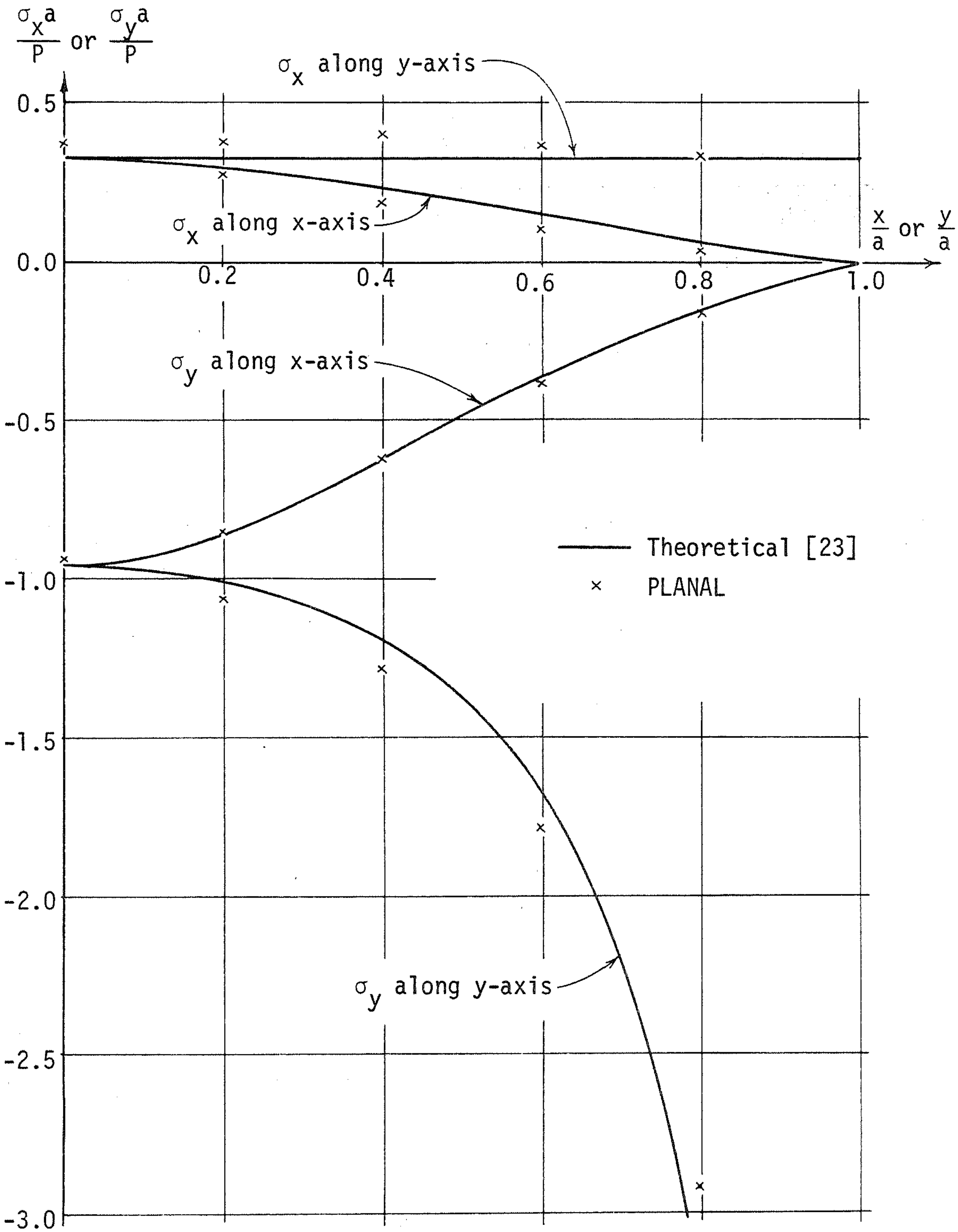
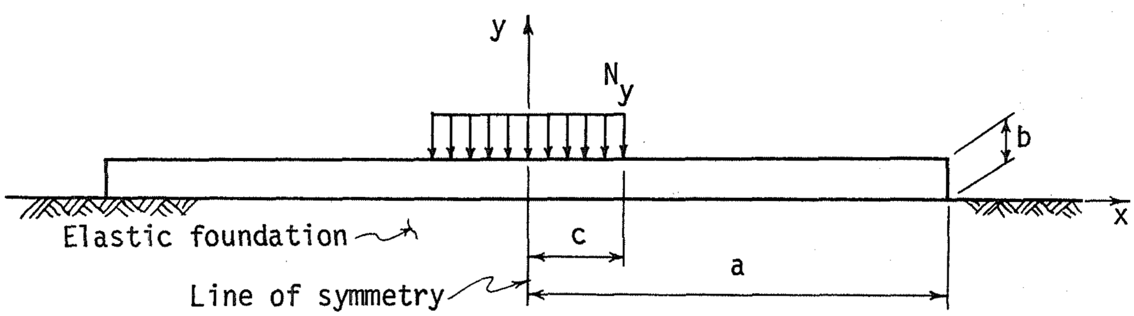
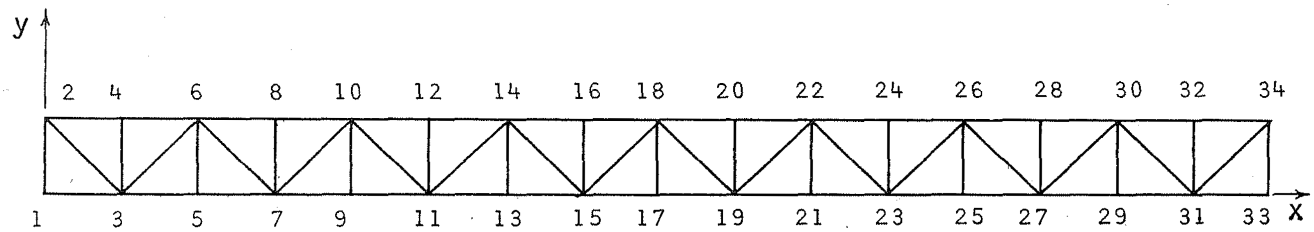


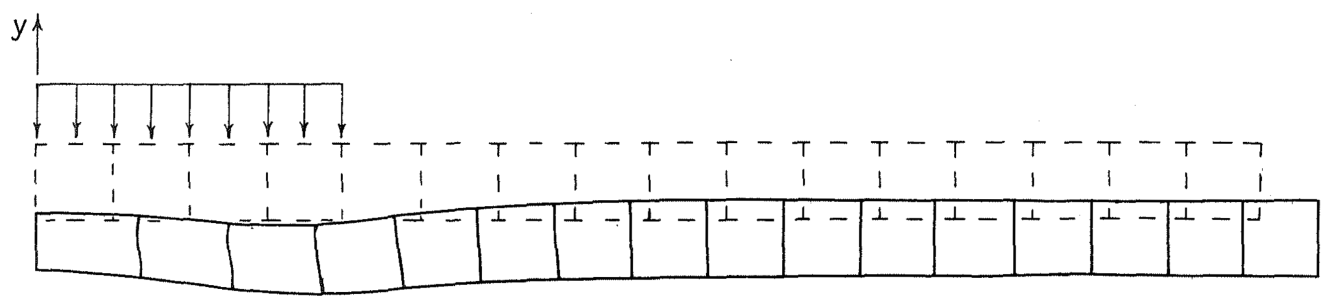
Fig. 6.6. Stresses along the axes of circular disk subjected to compressive forces.



a. Dimensions and loading.

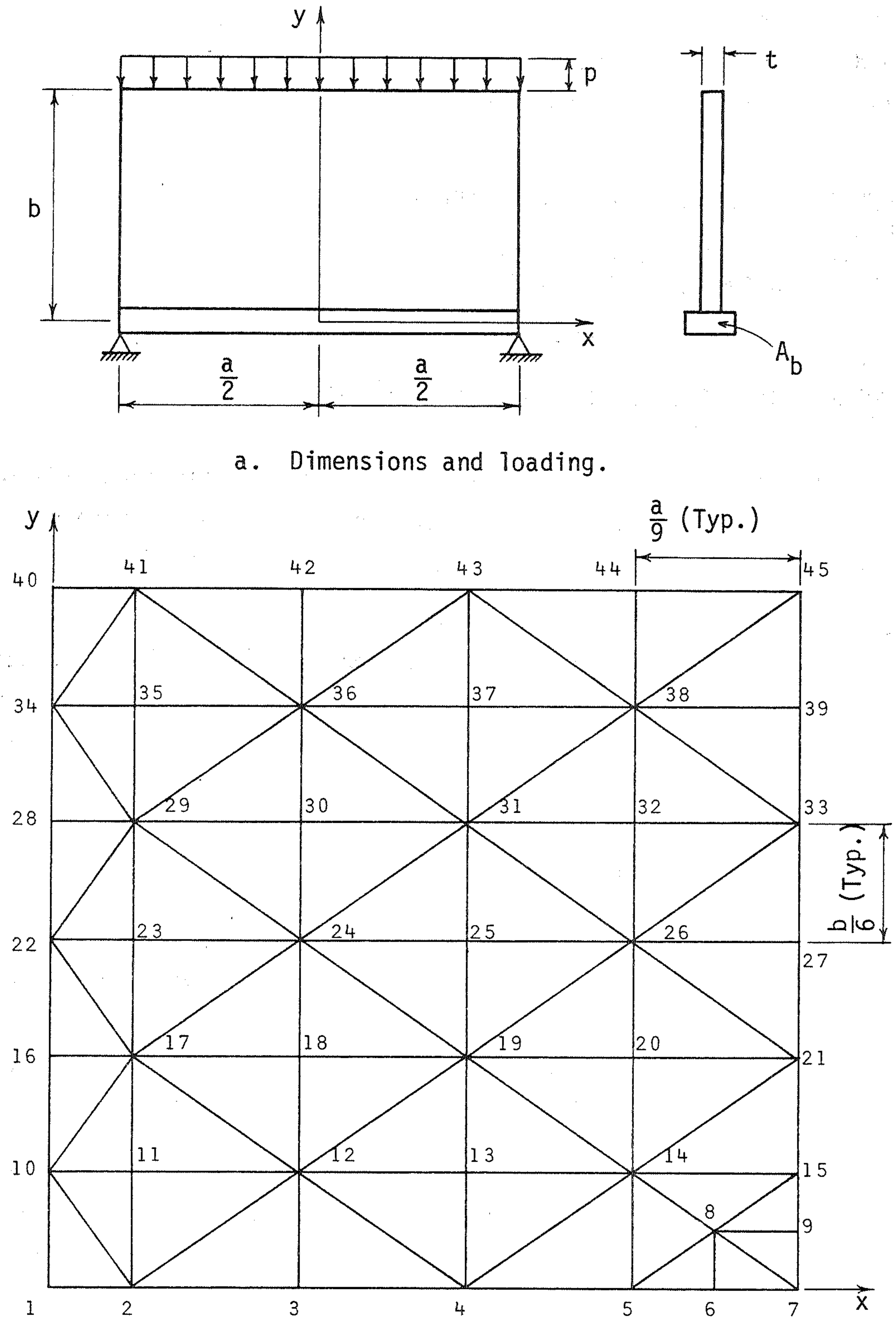


b. Discretization of half the beam.



c. The deformed beam.

Fig. 6.7. Beam on elastic foundation.



a. Dimensions and loading.

b. Discretization of half the plate.

Fig. 6.8. Plate with edge beam.

port are suppressed, and a "dummy" edge beam is specified along the boundary portion that has no edge beam. A MIXED STRETCHING boundary condition is *superimposed* along the line of symmetry. The resulting input cards are:

```
BOUNDARY CONDITION 'BOUND' DISPLACEMENT
7 U 0.0 V 0.0
BOUNDARY CONDITION 'BOUND' EDGE BEAM
 1 TO  7 EB 3000000.0 AB 0.955
 7 TO 45 EB 0.0 AB 0.0
45 TO 40 EB 0.0 AB 0.0 NY -1.0
40 TO  1 EB 0.0 AB 0.0
BOUNDARY CONDITION 'BOUND' MIXED STRETCHING
40 TO 1 UR 0.0 NR 0.0 ANGLE 0.0
```

A finite difference solution is obtained by Rosenhaupt [22] for the above problem which he considers a masonry wall with a reinforced concrete foundation beam acting as a tension tie. Stresses σ_x, σ_y, σ_{xy} and major principal directions θ_1† obtained from PLANAL and [22] are listed in Table 6.1 in which the nodes are as named in Fig. 6.8b. The same stresses are plotted in Fig. 6.9 for comparison. The two sets of results are very close except along the free edges since constant strain triangular elements are used in PLANAL. ■

6.3. Examples in Bending.

First, two short examples are given to illustrate plates in pure bending and pure twist. Then, in six examples following, rectangular plates with various aspect ratios are analyzed for different edge conditions and loadings. Results from PLANAL are compared with the theoretical values tabulated in Timoshenko and Woinowsky-Krieger [24].

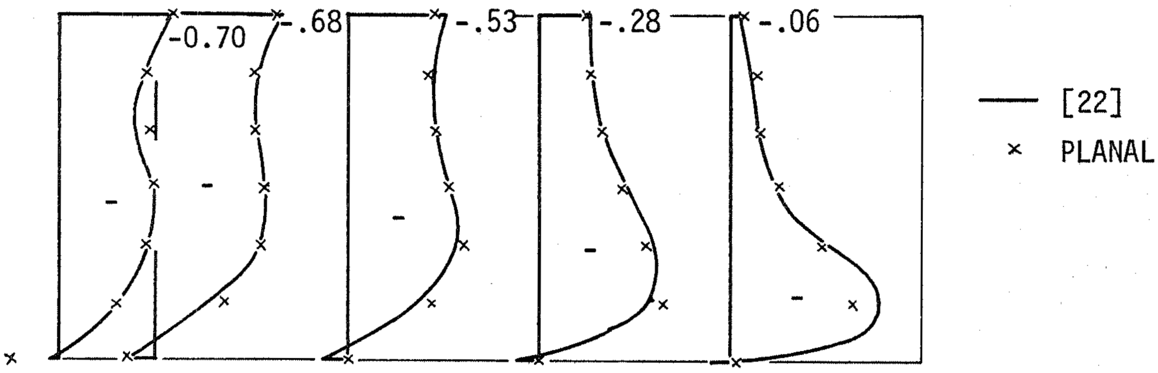
Example 6.5. *Rectangular Plate in Pure Bending*. A rectangular plate of thickness h (Fig. 6.10a) is placed in a state of pure bending by prescribing, along the boundary, displacement w and edge rotation β

---

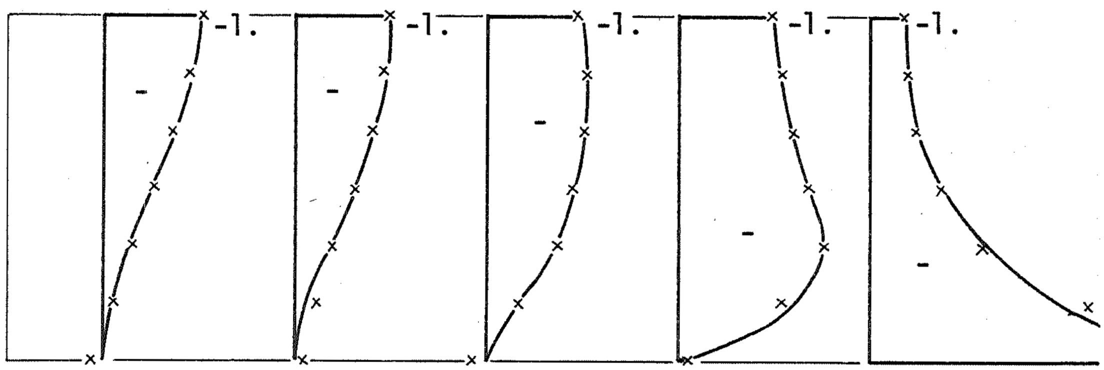
† θ_1 is measured from the x-axis to the direction of the major principal stress.

Table 6.1. Stresses and Principal Directions in Plate with Edge Beam.

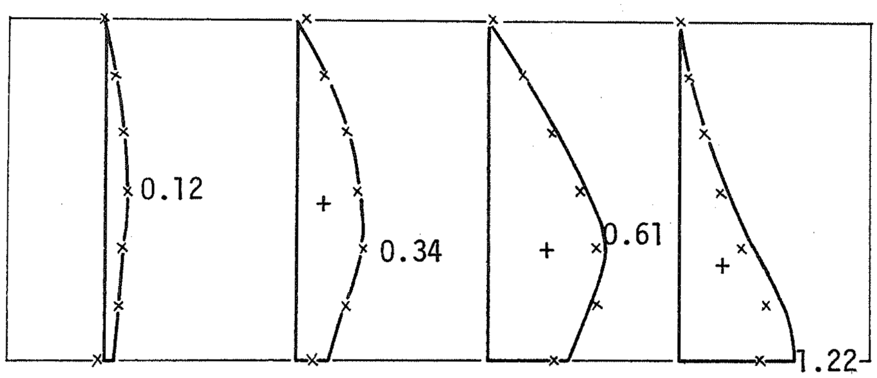
Node	$\sigma_x t/p$		$\sigma_y t/p$		$\sigma_{xy} t/p$		θ_1 (degrees)	
	PLANAL	[22]	PLANAL	[22]	PLANAL	[22]	PLANAL	[22]
1	0.271	0.08	-0.032	0.0	-0.072	0.0	-12.4	0.0
2	0.136	0.130	0.119	0.0	-0.023	0.051	-34.6	0.0
3	-0.003	0.125	-0.097	0.0	0.096	0.177	31.9	0.0
4	-0.017	0.113	0.176	0.0	0.346	0.427	52.8	0.0
5	-0.039	0.085	-0.077	0.0	0.829	1.216	44.3	0.0
6	-0.069		-0.784	0.0	2.004		39.9	0.0
7	-0.115	0.0	3.975	-9.000	3.542		60.0	0.0
8	0.153		-0.784		3.093		40.7	
9	0.153	0.0	-4.325		3.368	0.0	28.2	0.0
10	-0.321	-0.34	-0.129		0.062	0.0	73.7	0.0
11	-0.360	-0.320	-0.118	-0.105	0.083	0.075	72.7	72.6
12	-0.442	-0.411	-0.213	-0.125	0.273	0.251	56.3	58.6
13	-0.655	-0.599	-0.347	-0.363	0.579	0.531	52.5	51.3
14	-0.640	-0.788	-1.101	-1.250	0.908	1.095	37.9	39.0
15	-0.270	0.0	-6.841	-5.255	1.048	0.0	8.8	0.0
16	-0.576	-0.56	-0.298		-0.013	0.0	-87.4	0.0
17	-0.567	-0.543	-0.332	-0.300	0.104	0.109	69.2	69.1
18	-0.617	-0.587	-0.407	-0.402	0.339	0.341	53.6	37.4
19	-0.550	-0.627	-0.749	-0.721	0.572	0.612	40.1	42.8
20	-0.475	-0.532	-1.543	-1.533	0.688	0.788	26.1	28.8
21	-0.574	0.0	-2.956	-3.086	0.644	0.0	14.2	0.0
22	-0.578	-0.60	-0.533		-0.003	0.0	-86.5	0.0
23	-0.584	-0.575	-0.539	-0.539	0.113	0.115	50.6	49.4
24	-0.531	-0.556	-0.658	-0.649	0.320	0.338	39.4	41.7
25	-0.464	-0.479	-0.926	-0.945	0.490	0.519	32.4	32.9
26	-0.261	-0.240	-1.408	-1.376	0.434	0.512	18.5	21.0
27	0.130	0.0	-1.941	-1.981	0.256	0.0	6.9	0.0
28	-0.560	-0.52	-0.745		-0.011	0.0	3.3	0.0
29	-0.528	-0.536	-0.749	-0.760	0.094	0.096	20.2	20.3
30	-0.472	-0.479	-0.833	-0.836	0.255	0.266	27.4	18.2
31	-0.336	-0.350	-0.994	-1.008	0.332	0.360	22.6	23.8
32	-0.162	-0.164	-1.247	-1.218	0.284	0.304	13.8	15.0
33	-0.036	0.0	-1.356	-1.354	0.092	0.0	4.0	0.0
34	-0.536	-0.53	-0.897		0.012	0.0	2.0	0.0
35	-0.514	-0.530	-0.910	-0.923	0.059	0.060	8.4	8.6
36	-0.425	-0.450	-0.932	-0.952	0.144	0.161	14.8	16.3
37	-0.285	-0.295	-1.015	-1.014	0.188	0.199	13.6	14.5
38	-0.136	-0.120	-1.100	-1.081	0.127	0.143	7.4	8.4
39	0.019	0.0	-1.092	-1.057	0.007	0.0	0.4	0.0
40	-0.711	-0.70	-1.008	-1.000	-0.015	0.0	-3.0	0.0
41	-0.641	-0.679	-1.037	-1.000	0.004	0.0	0.5	0.0
42	-0.477	-0.526	-1.008	-1.000	0.052	0.0	5.5	0.0
43	-0.266	-0.277	-0.994	-1.000	0.026	0.0	2.0	0.0
44	-0.084	-0.057	-0.999	-1.000	0.007	0.0	0.5	0.0
45	0.047	0.0	-0.939	-1.000	-0.073	0.0	-4.0	0.0



a. $\sigma_x t/p$.



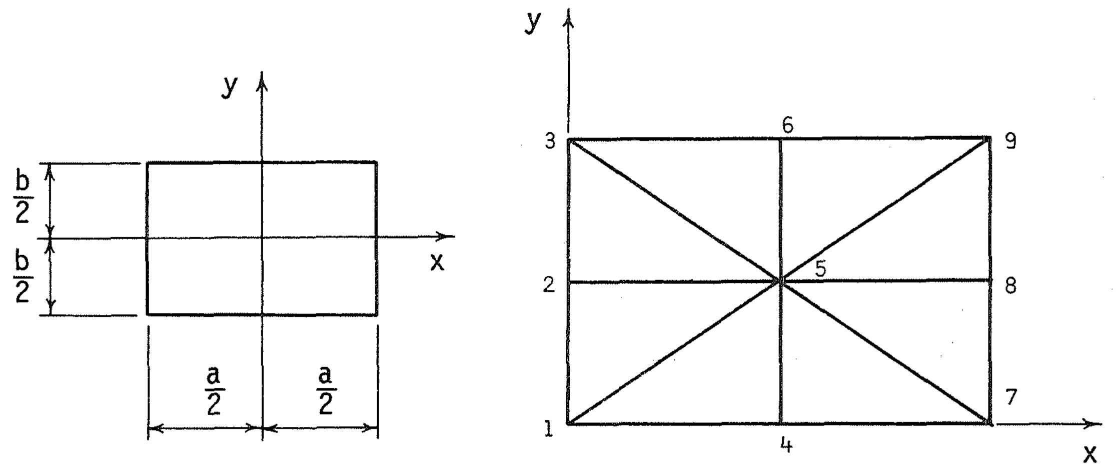
b. $\sigma_y t/p$.



Note. In this figure, geometry of plate is distorted.

c. $\sigma_{xy} t/p$.

Fig. 6.9. Comparison of stresses in plate with edge beam.



a. Dimensions.

b. Discretization.

Fig. 6.10. Rectangular plate in pure bending and pure twist.

of the form

$$w = c\ (x^2 + y^2),$$
$$\beta = -\ w_{,n}, \tag{6.1}$$

where c is an arbitrary constant. The resulting bending stress couple M is constant throughout the plate and is given by

$$M = -\ \frac{Eh^3c}{6(1-\nu)}. \tag{6.2}$$

The twisting couple M_{xy} is identically zero.

Analyzing only a quarter of the plate (Fig. 6.10b), we take a = 6 in., b = 4 in., h = 1 in., c = 0.1, E = 10^5 psi and ν = 0.2. The values of w and β along the boundary can be computed from (6.1). In this case, if we specify only w and β, the stress functions will be indeterminate. Hence, we also specify a quantity dual of a rigid body displacement. The input cards for boundary conditions are:

```
BOUNDARY CONDITION 'BOUND' DISPLACEMENT
1      W 0.0   R 0.0
4      W 0.225 R 0.0
7 NEG  W 0.900 R 0.0
7 POS  W 0.900 R 0.6
8      W 1.000 R 0.6
9 NEG  W 1.300 R 0.6
9 POS  W 1.300 R 0.4
6      W 0.625 R 0.4
3 NEG  W 0.400 R 0.4
3 POS  W 0.400 R 0.0
2      W 0.100 R 0.0
BOUNDARY CONDITION 'BOUND' FUNCTION
4 U 0.0 V 0.0
BOUNDARY CONDITION 'BOUND' MIXED BENDING
9 POS         CHI 0.0 ANGLE 0.0
6      UR 0.0 CHI 0.0 ANGLE 0.0
3 NEG         CHI 0.0 ANGLE 0.0
```

Stress couples from PLANAL are $M_x = M_y = -2083$ lb-in./in. and $M_{xy} = 0$ at all nodes, which are in agreement with theoretical values. ■

Example 6.6. *Rectangular Plate in Pure Twist.* The plate in Example 6.5 may be placed in a state of pure twist by prescribing along the boundary w and β of the form

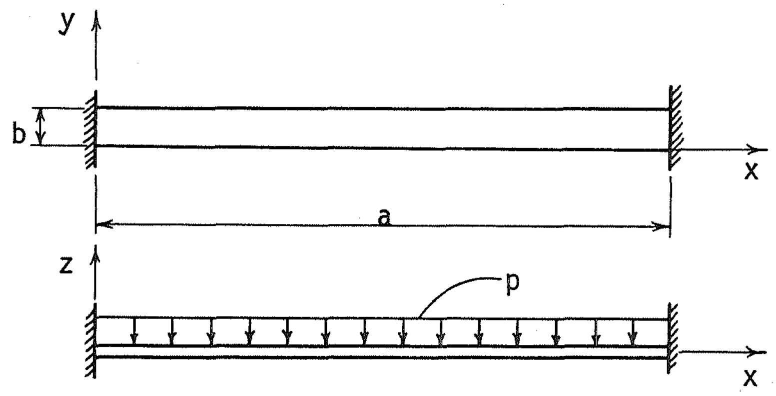
$$w = cxy,$$
$$\beta = - w_{,n}. \tag{6.3}$$

The resulting bending stress couple is identically zero and the twisting couple is given by

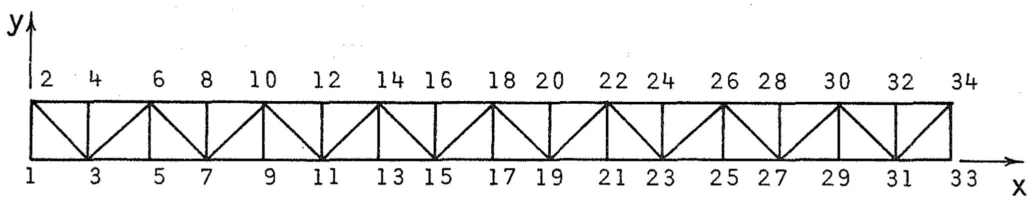
$$M_{xy} = - \frac{Eh^3c}{12(1+\nu)}. \tag{6.4}$$

Proceeding as in Example 6.5, we obtain from PLANAL $M_x = M_y = 0$ and $M_{xy} = -694.4$ lb-in./in. at all nodes, which are in agreement with theoretical values. ■

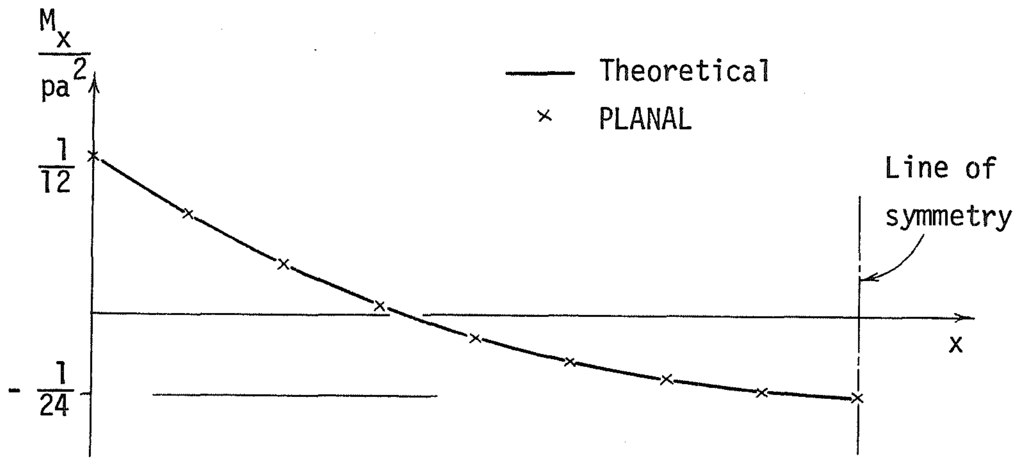
Example 6.7. *A Uniformly Loaded Long Plate.* A long plated fixed at its ends is subjected to a uniformly distributed load p (Fig. 6.11a). When b is small compared to a, the plate behaves like a fixed-ended beam. A statically equivalent load may be supplied in the form of an effective



a. Plan and elevation.



b. Discretization.



c. Moments along the x-axis.

Fig. 6.11. A uniformly loaded long plate.

edge shear $Q = \frac{1}{2}$ pb applied along the two free edges. The plate is discretized as shown in Fig. 6.11b and the values of a = 16 in., b = 1 in., p = 1 psi are taken in the PLANAL analysis. Input cards for boundary conditions are:

```
BOUNDARY CONDITION 'BOUND' FIXED SUPPORT
33 TO 34
 2 TO  1
BOUNDARY CONDITION 'BOUND' STRESS
 1 TO 33 Q -0.5
34 TO  2 Q -0.5
```

The moments from PLANAL are plotted in Fig. 6.11c and they compare closely with the theoretical values. ■

Example 6.8. *Rectangular Plates with Simply Supported Edges*. A homogeneous, isotropic rectangular plate is simply supported along its edges (Fig. 6.12). Cases with different aspect ratios (a : b) and under different loadings are analyzed in the PLANAL System for $E = 10^5$ psi and $\nu = 0.3$.

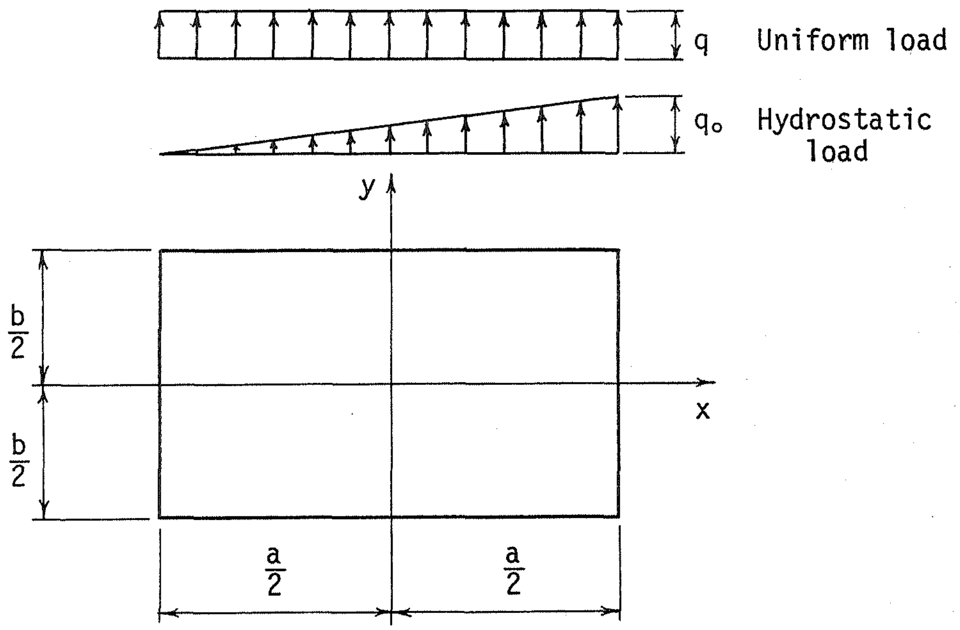


Fig. 6.12. Dimensions and loadings of a rectangular plate.

Uniform Load. When the plate is under uniform load, the coordinate axes are lines of symmetry; therefore, only the first quadrant of the plate is analyzed. We take b = 1 in. and q = 1 psi, and the aspect ratios a/b of 1 and 2, using a 4 x 4 grid shown in Fig. 6.13. The input cards for boundary conditions and loading are:

```
BOUNDARY CONDITION 'BOUND' SYMMETRY
 1 TO 21
 5 TO  1
BOUNDARY CONDITION 'BOUND' SIMPLE SUPPORT
21 TO  5
LOADING
UNIFORM INTENSITY Z 1.0
```

The bending particular solution functions K_x and K_y, and q (for checking) at each node are constructed internally by the system, and the results for a/b = 1 are shown in Fig. 6.14. A sample of the output from the system for a/b = 1 is shown in Fig. 6.15. Results for both aspect ratios are shown in Fig. 6.16.

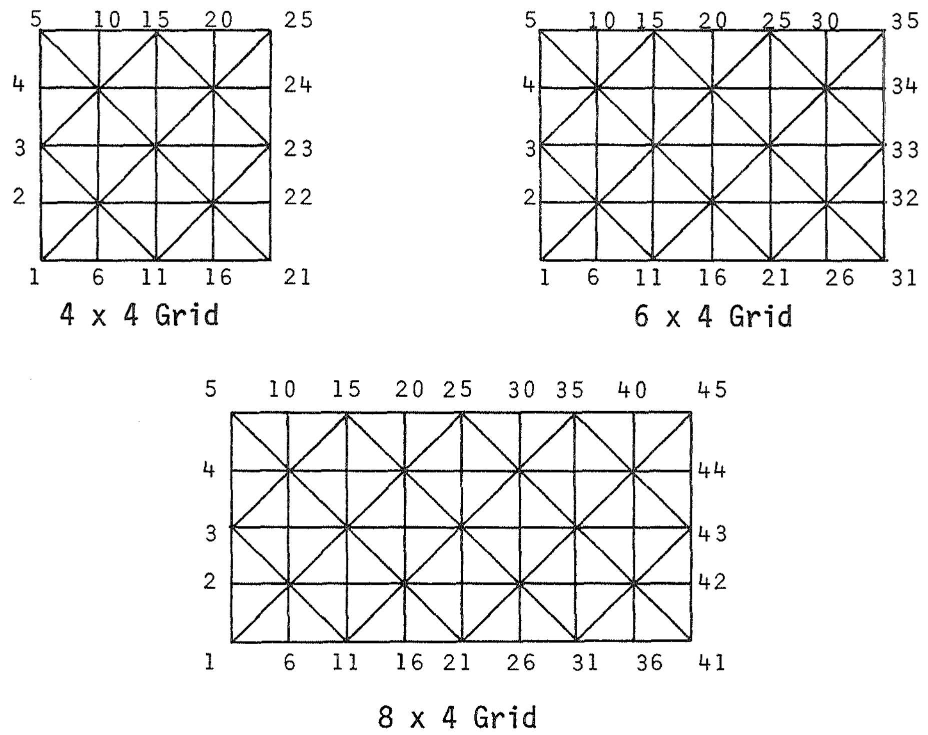


Fig. 6.13. Grid patterns in rectangular plates.

BENDING PARTICULAR SOLUTION CONSTRUCTED FROM FOURIER SERIES.

NODE	KX = KY	MAXIMUM SUMMATION INDEX	LOAD FUNCTION (FOR CHECKING)	MAXIMUM SUMMATION INDEX
1	-0.6190E-05	17	0.9757E 00	51
2	-0.5857E-05	7	0.9879E 00	51
3	-0.4817E-05	11	0.1005E 01	51
4	-0.2935E-05	13	0.9863E 00	51
5	0.0	1	0.5312E-06	1
6	-0.5857E-05	7	0.9879E 00	51
7	-0.5550E-05	13	0.1000E 01	51
8	-0.4569E-05	9	0.1017E 01	51
9	-0.2800E-05	13	0.9986E 00	51
10	0.0	1	0.4908E-06	1
11	-0.4817E-05	11	0.1005E 01	51
12	-0.4569E-05	9	0.1017E 01	51
13	-0.3804E-05	17	0.1035E 01	51
14	-0.2375E-05	11	0.1016E 01	51
15	0.0	1	0.3756E-06	1
16	-0.2935E-05	13	0.9863E 00	51
17	-0.2800E-05	13	0.9986E 00	51
18	-0.2375E-05	11	0.1016E 01	51
19	-0.1531E-05	23	0.9971E 00	51
20	0.0	1	0.2033E-06	1
21	0.0	1	0.5312E-06	1
22	0.0	1	0.4908E-06	1
23	0.0	1	0.3756E-06	1
24	0.0	1	0.2033E-06	1
25	0.0	1	0.1741E-12	1

Fig. 6.14. Particular solution functions for a quadrant of a uniformly loaded rectangular plate.

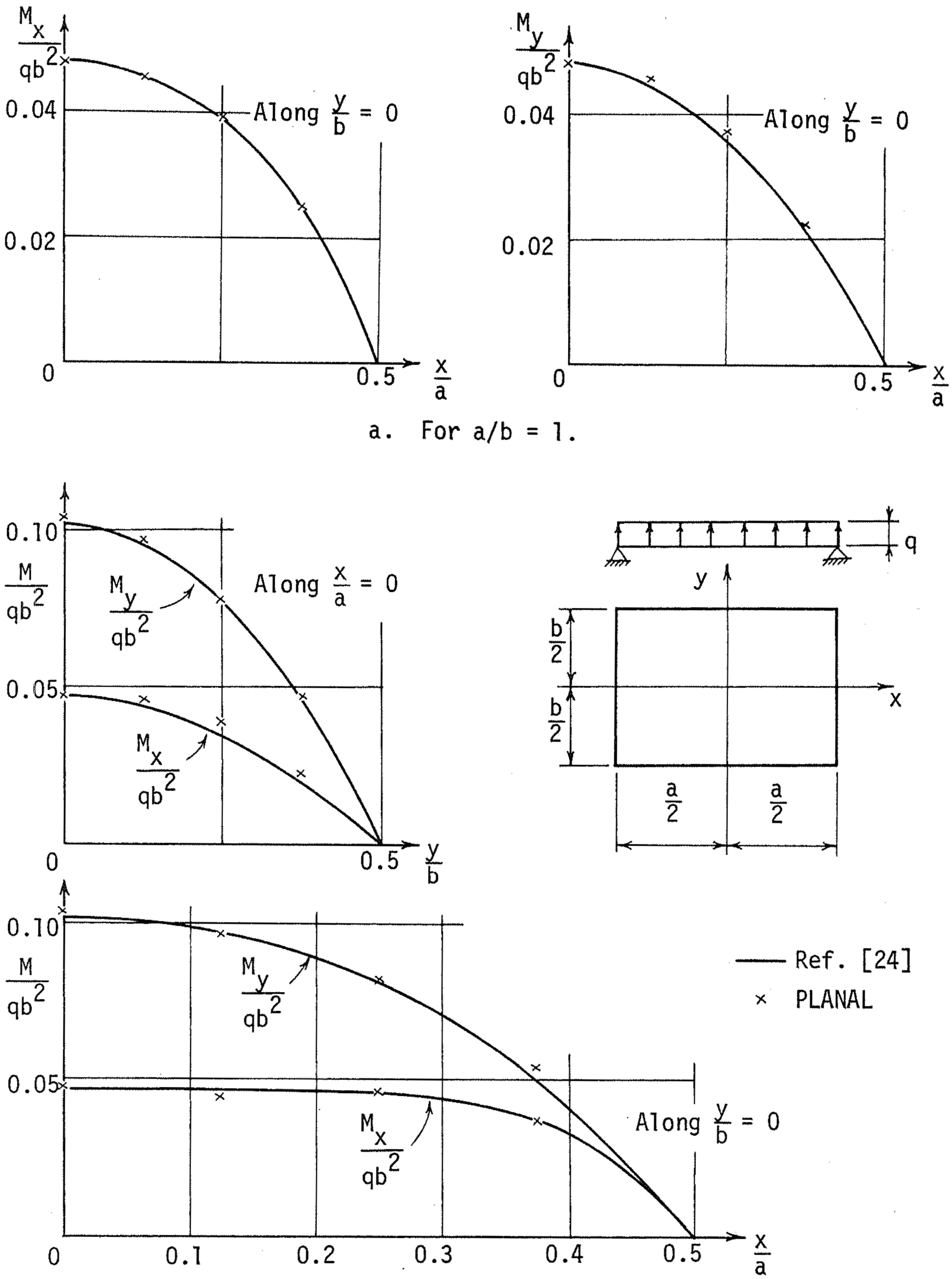
Hydrostatic Load. When the plate is under a hydrostatic load which varies linearly in x, the x-axis is the line of symmetry. Half of the plate is analyzed, with b = 1 in. and q_o = 1 psi. The grids for aspect ratios a/b of 0.5, 1, and 2 are the 4 x 4, 8 x 4, and 8 x 4 grids, respectively, in Fig. 6.13. The input cards for loading when a/b = 1 are:

```
LOADING
NODES  1 5 INTENSITY Z 0.0
NODE  25   INTENSITY Z 1.0
```

The results are presented in Fig. 6.17. ■

```
PLANAL 'U44LSSL1' 'HALF BY HALF UNIT LENGTH.'

** BENDING PARTICULAR SOLUTION CONSTRUCTED FROM FOURIER SERIES.

                          MAXIMUM                             MAXIMUM
                          SUMMATION      LOAD FUNCTION        SUMMATION
NODE        KX = KY       INDEX          (FOR CHECKING)       INDEX

  1     -0.6190E-05        17            0.9757E 00           51
  2     -0.5857E-05         7            0.9879E 00           51
                                                              51

        ****  ****  RESULTS  ****  ****

NODAL STRESS FUNCTIONS

  NODE         U             V

   1        0.0           0.0
   2       -0.2031E-08   -0.3099E-02
   3       -0.3926E-08   -0.5991E-02
   4       -0.5319E-08   -0.8116E-02
   5       -0.6001E-08   -0.9157E-02

** GRID PATTERN FOR DIFFERENTIATION.

  LINES PARALLEL TO X-AXIS.

       5 NODES.     1   6  11  16  21
       5 NODES.     2   7  12  17  22
                           13  18  23
  LINES PARALLEL TO Y-AXIS.

       5 NODES.     1   2   3   4   5
       5 NODES.     6   7   8   9  10
                   11  12  13  14  15

NODAL MOMENTS AND PRINCIPAL MOMENTS

  NODE       MX            MY            MXY            M1            M2                 THETA-1 (X TO M1)

   1      0.4808E-01    0.4808E-01    0.1259E-07     0.4808E-01    0.4808E-01    0.594 RAD  =   34 D  3 M 22.26 S
   2      0.4576E-01    0.4554E-01    0.9313E-04     0.4579E-01    0.4550E-01    0.347 RAD  =   19 D 53 M 30.37 S
   3      0.3727E-01    0.3904E-01   -0.4265E-03     0.3914E-01    0.3718E-01    4.938 RAD  =  282 D 54 M  3.16 S
   4      0.2227E-01    0.2484E-01    0.8573E-03     0.2510E-01    0.2201E-01    1.276 RAD  =   73 D  7 M 13.80 S
   5     -0.3980E-02    0.1914E-08   -0.1460E-02     0.4784E-03   -0.4459E-02    5.029 RAD  =  288 D  8 M  9.55 S
                                                                                 1.224 RAD  =   70 D  6 M 12.27 S
NODAL CURVATURES AND PRINCIPAL CURVATURES

  NODE      CHI-X         CHI-Y         CHI-XY         CHI-1         CHI-2               THETA-1 (X TO CHI-1)

   1      0.4039E-05    0.4039E-05    0.1964E-11     0.4039E-05    0.4039E-05    0.594 RAD  =   34 D  3 M 22.26 S
   2      0.3852E-05    0.3817E-05    0.1453E-07     0.3857E-05    0.3812E-05    0.347 RAD  =   19 D 53 M 30.37 S
   3      0.3068E-05    0.3343E-05   -0.6653E-07     0.3358E-05    0.3052E-05    4.938 RAD  =  282 D 54 M  3.16 S
   4      0.1779E-05    0.2179E-05    0.1337E-06     0.2219E-05    0.1738E-05    1.276 RAD  =   73 D  7 M 13.80 S
   5     -0.4776E-06    0.1433E-06   -0.2278E-06     0.2179E-06   -0.5523E-06    5.029 RAD  =  288 D  8 M  9.55 S
                                                     0.3857E-05    0.3812E-05    1.224 RAD  =   70 D  6 M 12.27 S

NODAL MOMENTS - HOMOGENEOUS, PARTICULAR, AND TOTAL.

  NODE       MXH           MXP           MX             MYH           MYP           MY

   1     -0.2561E-01    0.7369E-01    0.4808E-01    -0.2561E-01    0.7369E-01    0.4808E-01
   2     -0.2396E-01    0.6972E-01    0.4576E-01    -0.2419E-01    0.6972E-01    0.4554E-01
   3     -0.2007E-01    0.5735E-01    0.3727E-01    -0.1831E-01    0.5735E-01    0.3904E-01
   4     -0.1266E-01    0.3494E-01    0.2227E-01    -0.1010E-01    0.3494E-01    0.2484E-01
   5     -0.3980E-02    0.0          -0.3980E-02     0.1914E-08    0.0           0.1914E-08
                                      0.4554E-01    -0.2396E-01    0.6972E-01    0.4576E-01
ELEMENT MOMENTS AND PRINCIPAL MOMENTS

 ELEMENT     MX            MY            MXY            M1            M2                 THETA-1 (X TO M1)

   1      0.4671E-01    0.4504E-01   -0.8320E-03     0.4705E-01    0.4470E-01    5.890 RAD  =  337 D 30 M  0.88 S
   2      0.4504E-01    0.4671E-01   -0.8320E-03     0.4705E-01    0.4470E-01    5.105 RAD  =  292 D 30 M  0.0  S
   3      0.4124E-01    0.4126E-01   -0.8320E-03     0.4208E-01    0.4042E-01    5.494 RAD  =  314 D 46 M 30.53 S
   4      0.3827E-01    0.4151E-01   -0.4583E-02     0.4475E-01    0.3503E-01    5.328 RAD  =  305 D 15 M 57.13 S
   5      0.3116E-01    0.3060E-01   -0.5907E-02     0.3679E-01    0.2496E-01    5.522 RAD  =  316 D 22 M 22.09 S
                                                     0.3255E-01    0.2444E-01    4.943 RAD  =  283 D 14 M  0.23 S

ELEMENT CURVATURES AND PRINCIPAL CURVATURES

 ELEMENT    CHI-X         CHI-Y         CHI-XY         CHI-1         CHI-2               THETA-1 (X TO CHI-1)

   1      0.3983E-05    0.3724E-05   -0.1298E-06     0.3987E-05   -0.4225E-08    5.890 RAD  =  337 D 30 M  0.88 S
   2      0.3724E-05    0.3983E-05   -0.1298E-06     0.3728E-05   -0.4518E-08    5.105 RAD  =  292 D 30 M  0.0  S
   3      0.3464E-05    0.3466E-05   -0.1298E-06     0.3469E-05   -0.4856E-08    5.494 RAD  =  314 D 46 M 30.53 S
   4      0.3098E-05    0.3603E-05   -0.7150E-06     0.3255E-05   -0.1570E-06    5.328 RAD  =  305 D 15 M 57.13 S
   5      0.2638E-05    0.2550E-05   -0.9215E-06     0.2928E-05   -0.2900E-06    5.522 RAD  =  316 D 22 M 22.09 S
                                                                                 4.943 RAD  =  283 D 14 M  0.23 S

ELEMENT MOMENTS - HOMOGENEOUS, PARTICULAR, AND TOTAL.

 ELEMENT     MXH           MXP           MX             MYH           MYP           MY

   1     -0.2312E-01    0.6983E-01    0.4671E-01    -0.2479E-01    0.6983E-01    0.4504E-01
   2     -0.2479E-01    0.6983E-01    0.4504E-01    -0.2312E-01    0.6983E-01    0.4671E-01
   3     -0.2314E-01    0.6438E-01    0.4124E-01    -0.2312E-01    0.6438E-01    0.4126E-01
   4     -0.2100E-01    0.5927E-01    0.3827E-01    -0.1776E-01    0.5927E-01    0.4151E-01
   5     -0.1719E-01    0.4836E-01    0.3116E-01    -0.1776E-01    0.4836E-01    0.3060E-01
                                      0.2487E-01    -0.9747E-02    0.4187E-01    0.3212E-01
FINISH

GOOD-BYE
```

Fig. 6.15. A sample of PLANAL output for a bending problem.



a. For a/b = 1.

b. For a/b = 2.

Fig. 6.16. Bending moments of simply supported rectangular plates under uniform load, $\nu = 0.3$.

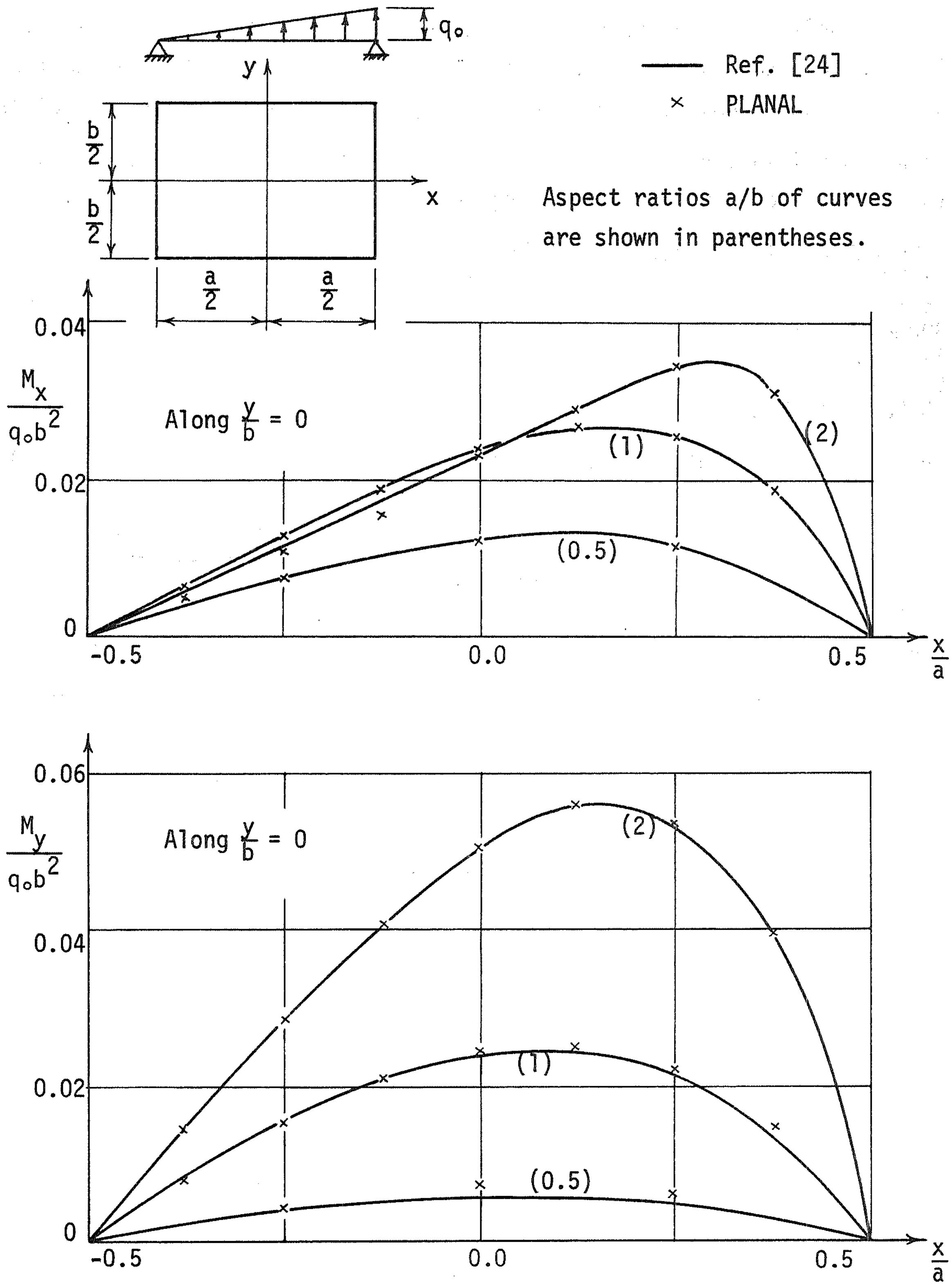


Fig. 6.17. Bending moments of simply supported rectangular plates under hydrostatic load, $\nu = 0.3$.

Example 6.9. Square Plate with Fixed Edges. A square plate (Fig. 6.12) with fixed edges is subjected to a uniform load q. Only a quarter of the plate is analyzed, using a 4 x 4 grid (Fig. 6.13). For the values of a = b = 1 in., $E = 10^5$ psi, $\nu = 0.3$, and q = 1 psi, the input cards for boundary conditions and loading are:

```
BOUNDARY CONDITION 'BOUND' SYMMETRY
 1 TO 21
 5 TO  1
BOUNDARY CONDITION 'BOUND' FIXED SUPPORT
21 TO 5
LOADING
UNIFORM INTENSITY Z 1.0
```

The results are plotted in Fig. 6.18. ■

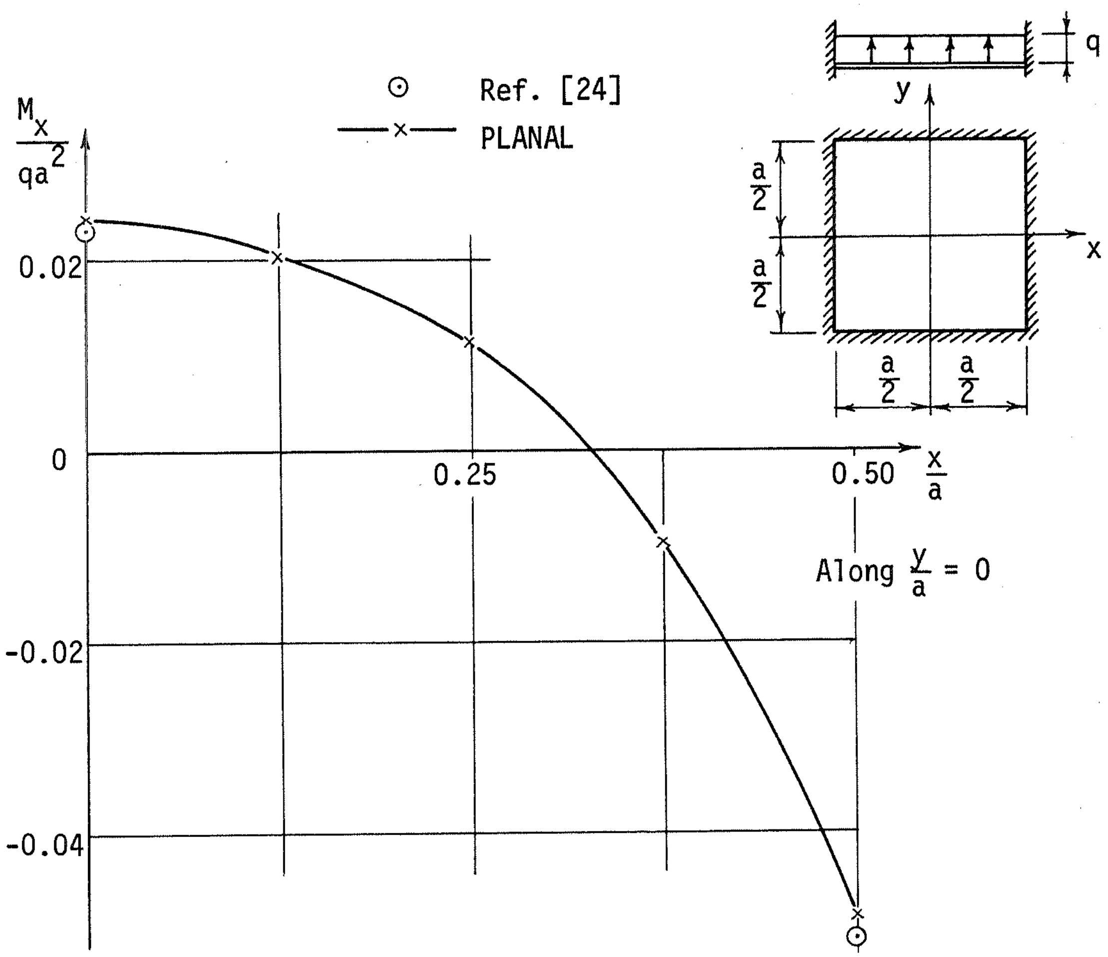


Fig. 6.18. Bending moments of a square plate with fixed edges under uniform load, $\nu = 0.3$.

Example 6.10. *Square Plate with Two Edges Simply Supported and Two Edges Fixed.* A homogeneous, isotropic square plate is simply supported along the edges parallel to the y-axis and fixed along the others (Fig. 6.12). It is subjected to a uniform load and we analyze the first quadrant of the plate for a = b = 1 in., $E = 10^5$ psi, $\nu = 0.3$, and q = 1 psi. Using the 4 x 4 grid in Fig. 6.13, the input cards for boundary conditions are:

```
BOUNDARY CONDITION 'BOUND' SYMMETRY
1 TO 21
5 TO  1
BOUNDARY CONDITION 'BOUND' SIMPLE SUPPORT
21 TO 25
BOUNDARY CONDITION 'BOUND' FIXED SUPPORT
25 TO 5
```

The results are plotted in Fig. 6.19. ∎

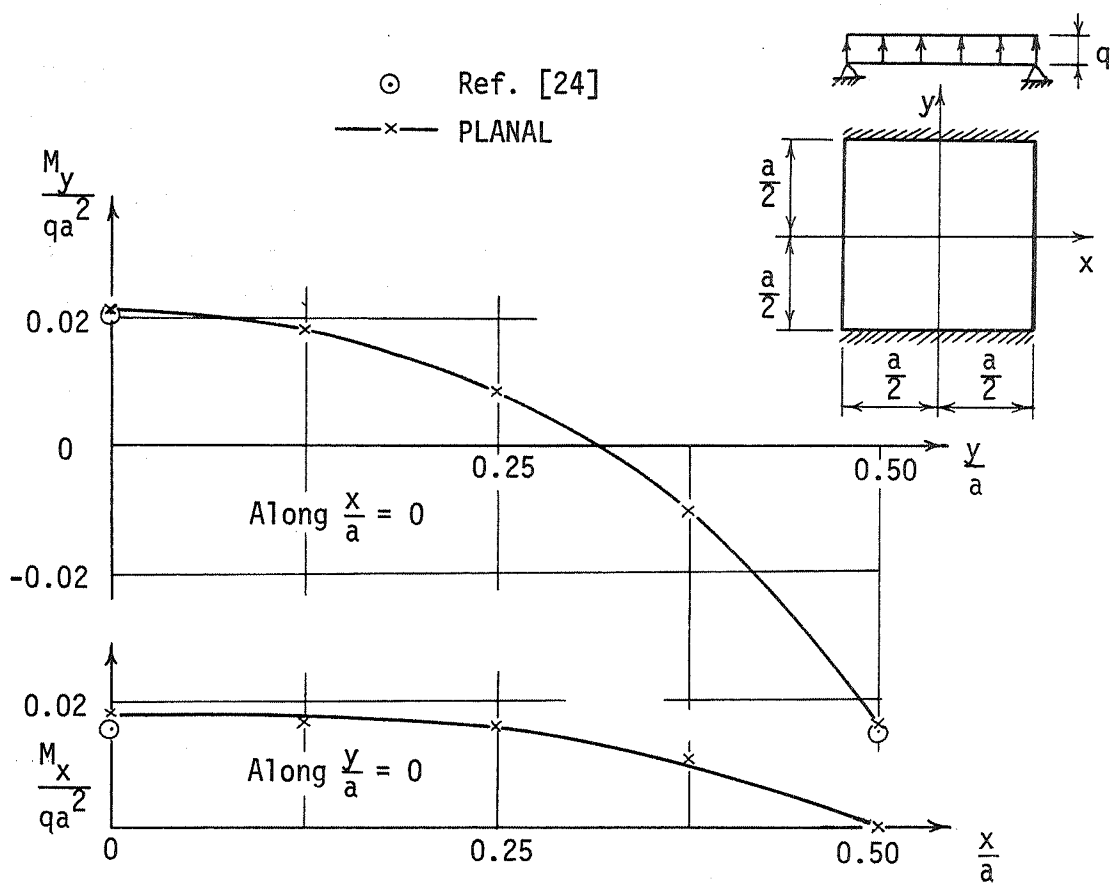


Fig. 6.19. Bending moments of a square plate with two edges simply supported and two edges fixed, $\nu = 0.3$.

Example 6.11. *Rectangular Plate with Three Edges Simply Supported and One Edge Fixed.* A homogeneous, isotropic rectangular plate shown in Fig. 6.12 is fixed along the edge x = a/2 and simply supported along the others. It is subjected to a uniform load and a hydrostatic load that varies linearly in x. The x-axis is the line of symmetry, and we analyze half the plate for b = 1 in., E = 10^5 psi, and ν = 0.3.

Uniform Load. We consider the aspect ratios a/b of 0.75 and 1 (using the 6 x 4 and 8 x 4 grids, respectively, in Fig. 6.13) for q = 1 psi. The input cards for boundary conditions when a/b = 1 are:

```
BOUNDARY CONDITION 'BOUND' SYMMETRY
1 TO 41
BOUNDARY CONDITION 'BOUND' FIXED SUPPORT
41 TO 45
BOUNDARY CONDITION 'BOUND' SIMPLE SUPPORT
45 TO 1
```

The results are shown in Fig. 6.20.

Hydrostatic Load. We analyze the case when a = b = 1 in. and q_o = 1 psi (8 x 4 grid used), and the results are plotted in Fig. 6.21. ■

Example 6.12. *Rectangular Plates Under Central Loads.* A homogeneous, isotropic plate is subjected to a concentrated load P applied at the center (Fig. 6.22a). Because of symmetry, only the first quadrant of the plate is analyzed. The quadrant is discretized in two patterns (Fig. 6.22b): Grid A has a uniform mesh and grid B has a finer mesh at the load point. The plate is analyzed for b = 1 in., E = 10^5 psi, ν = 0.3, and P = 1 lb.

Simply Supported Edges. The case when all edges are simply supported are analyzed for aspect ratios a/b of 1 and 2, and using both grids A and B. The input cards for boundary conditions and loading for grid B are:

```
BOUNDARY CONDITION 'BOUND' SYMMETRY
 1 TO 30
14 TO  1
BOUNDARY CONDITION 'BOUND' SIMPLE SUPPORT
30 TO 14
LOADING
NODE 1 FORCE Z 1.0
```

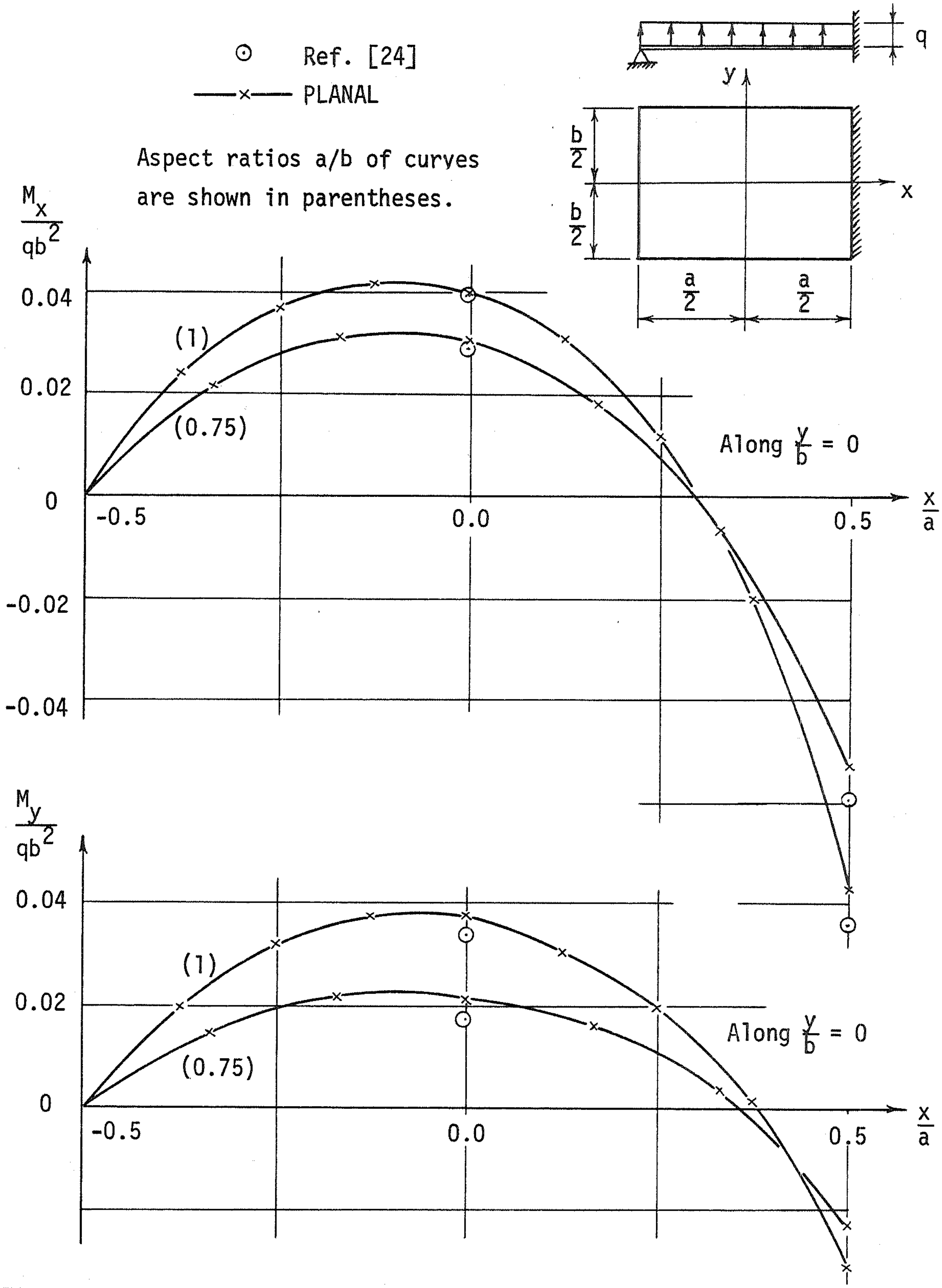


Fig. 6.20. Bending moments of rectangular plates with three edges simply supported and one edge fixed under uniform load, $\nu = 0.3$.

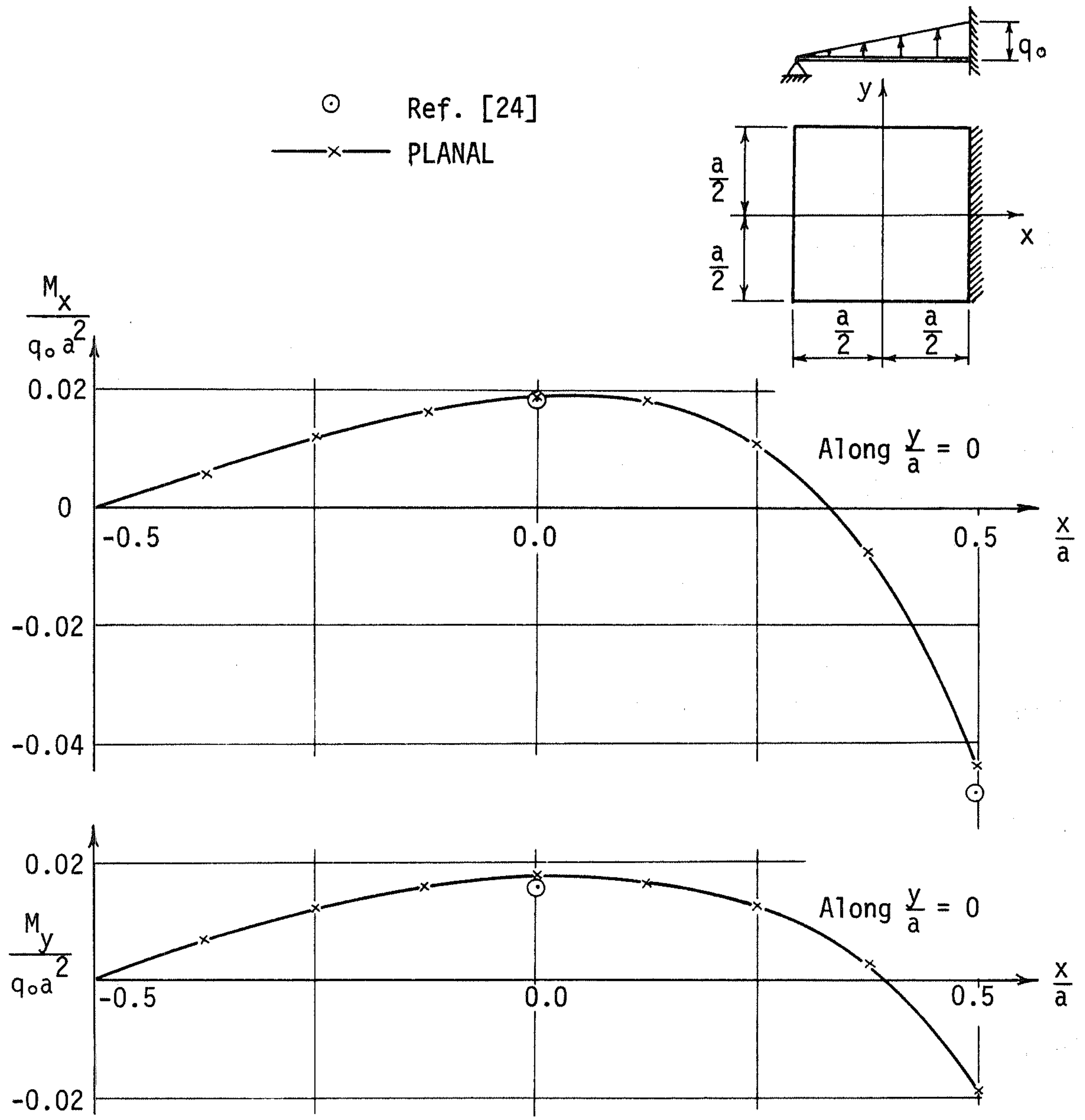
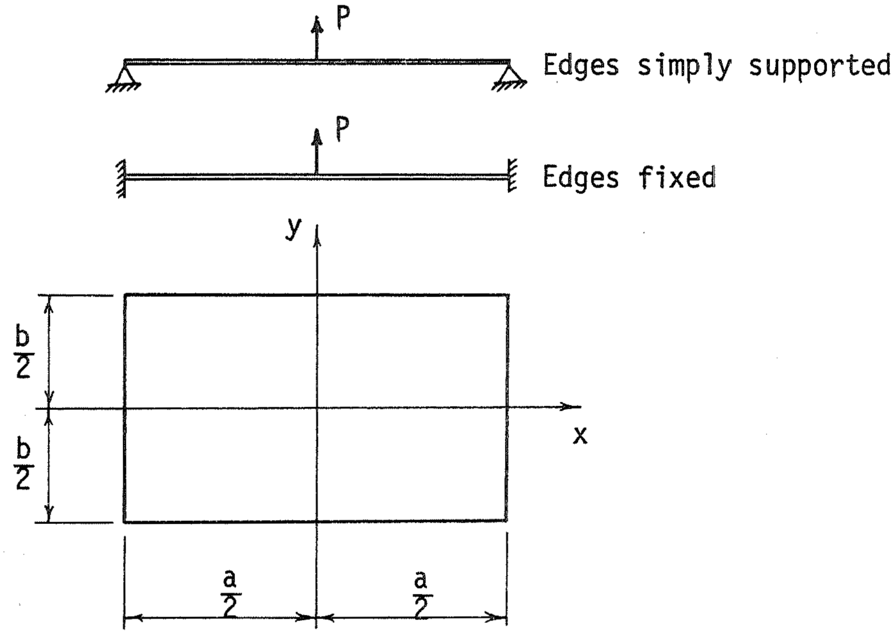


Fig. 6.21. Bending moments of a square plate with three edges simply supported and one edge fixed under hydrostatic load, $\nu = 0.3$.

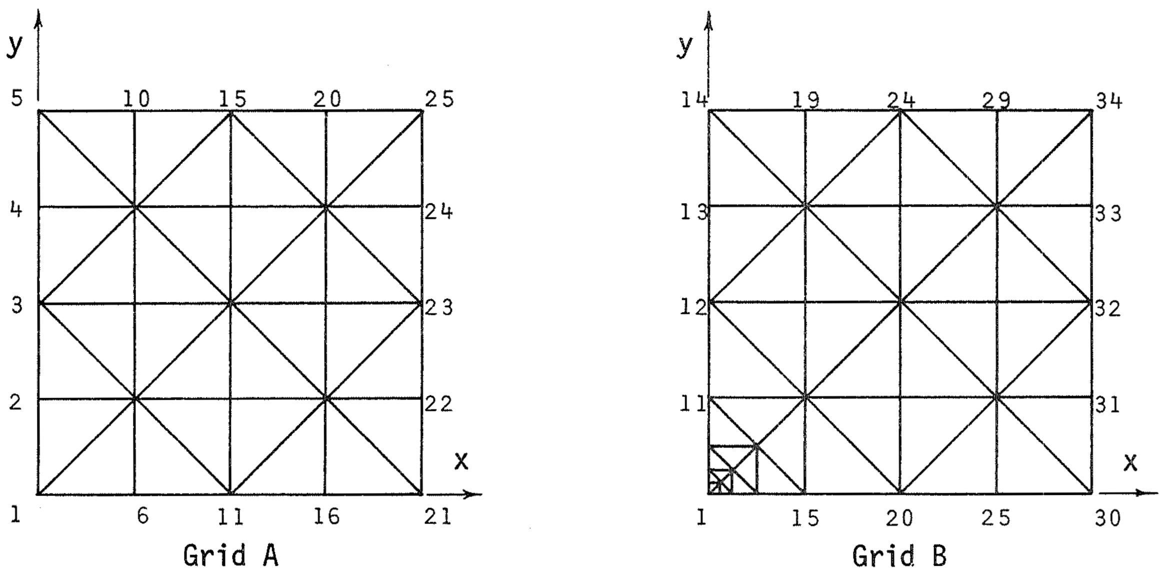
The results are shown in Fig. 6.23.

Fixed Edges. The case with all edges fixed is also analyzed for aspect ratios a/b of 1 and 2, and using both grids. The results are shown in Fig. 6.24.

It can be seen that results from grid B with the finer mesh approach the approximations at the vincinity of load point. ■

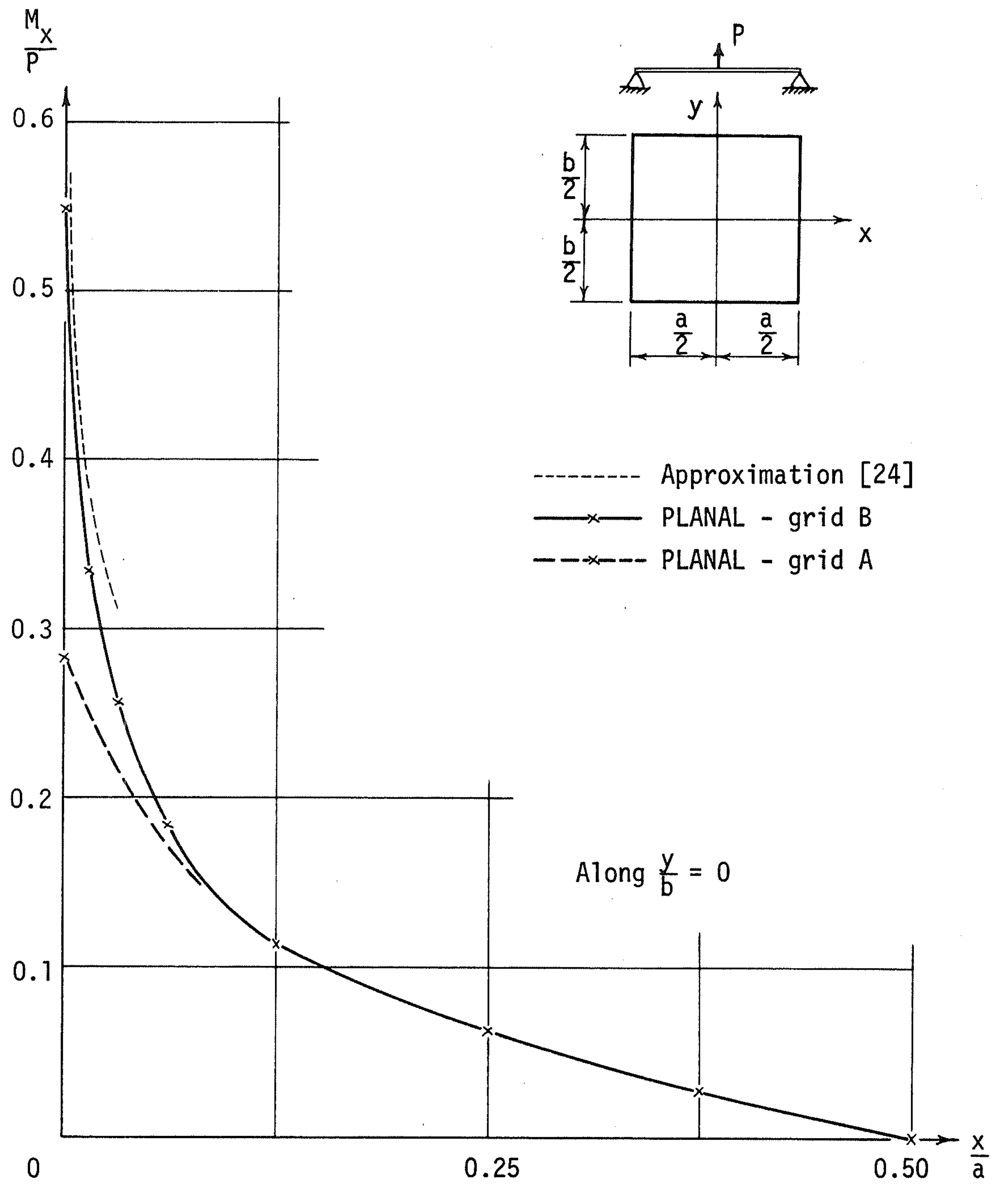


a. Dimensions and loading.



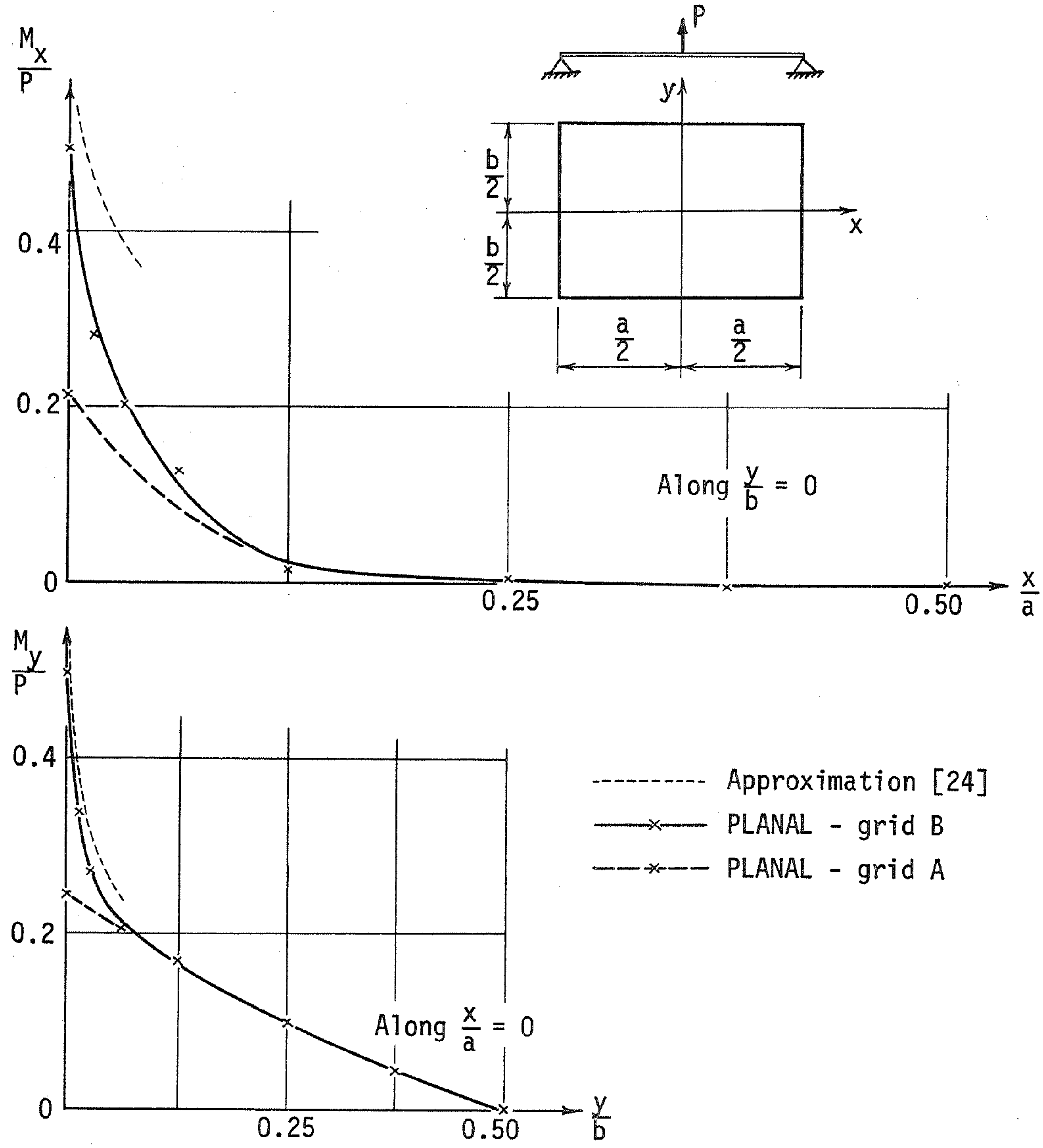
b. Discretization of a quarter of the plate.

Fig. 6.22. Rectangular plates under central loads.



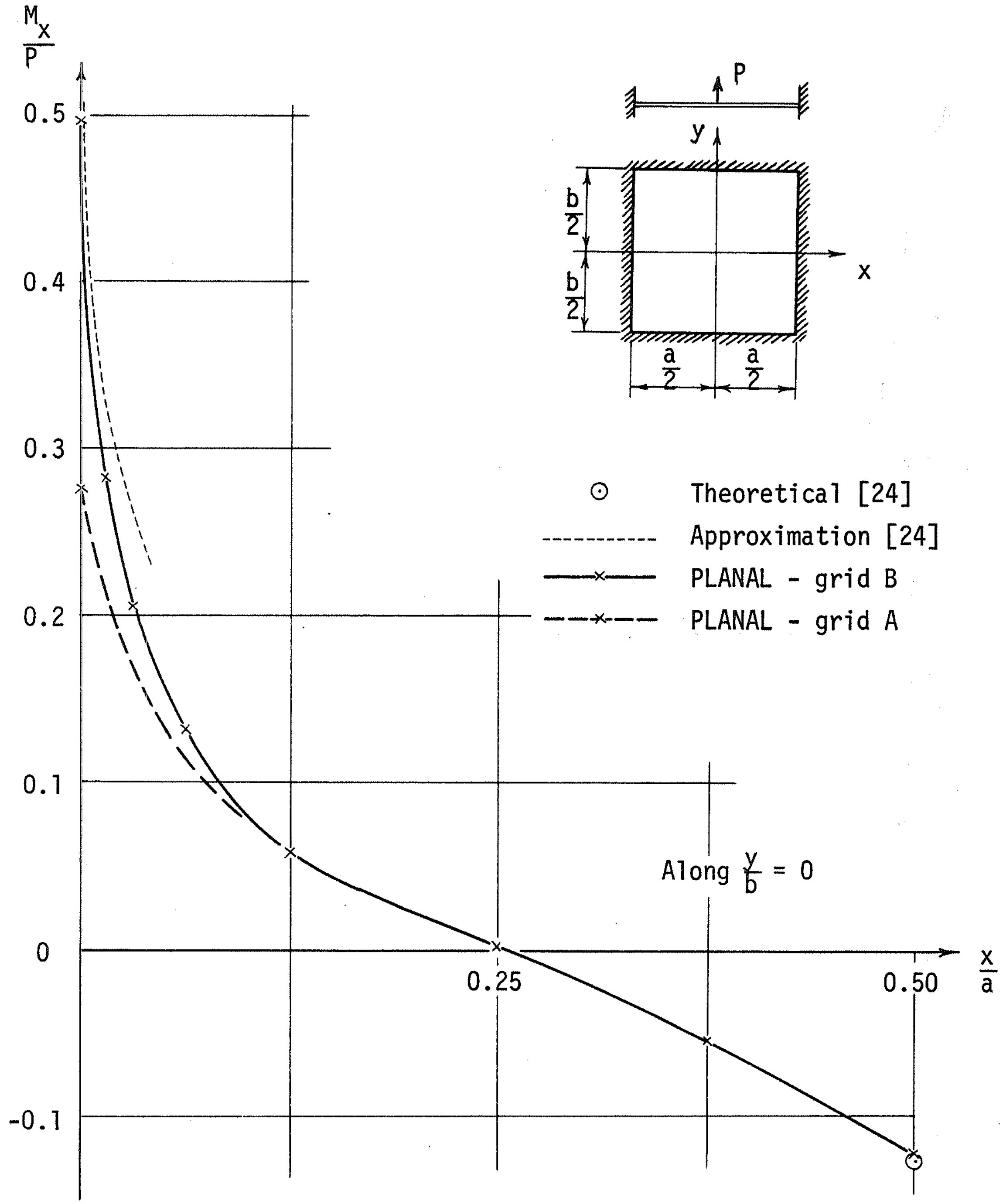
a. For a/b = 1.

Fig. 6.23. Bending moments of rectangular plates with simply supported edges under central load, $\nu = 0.3$.



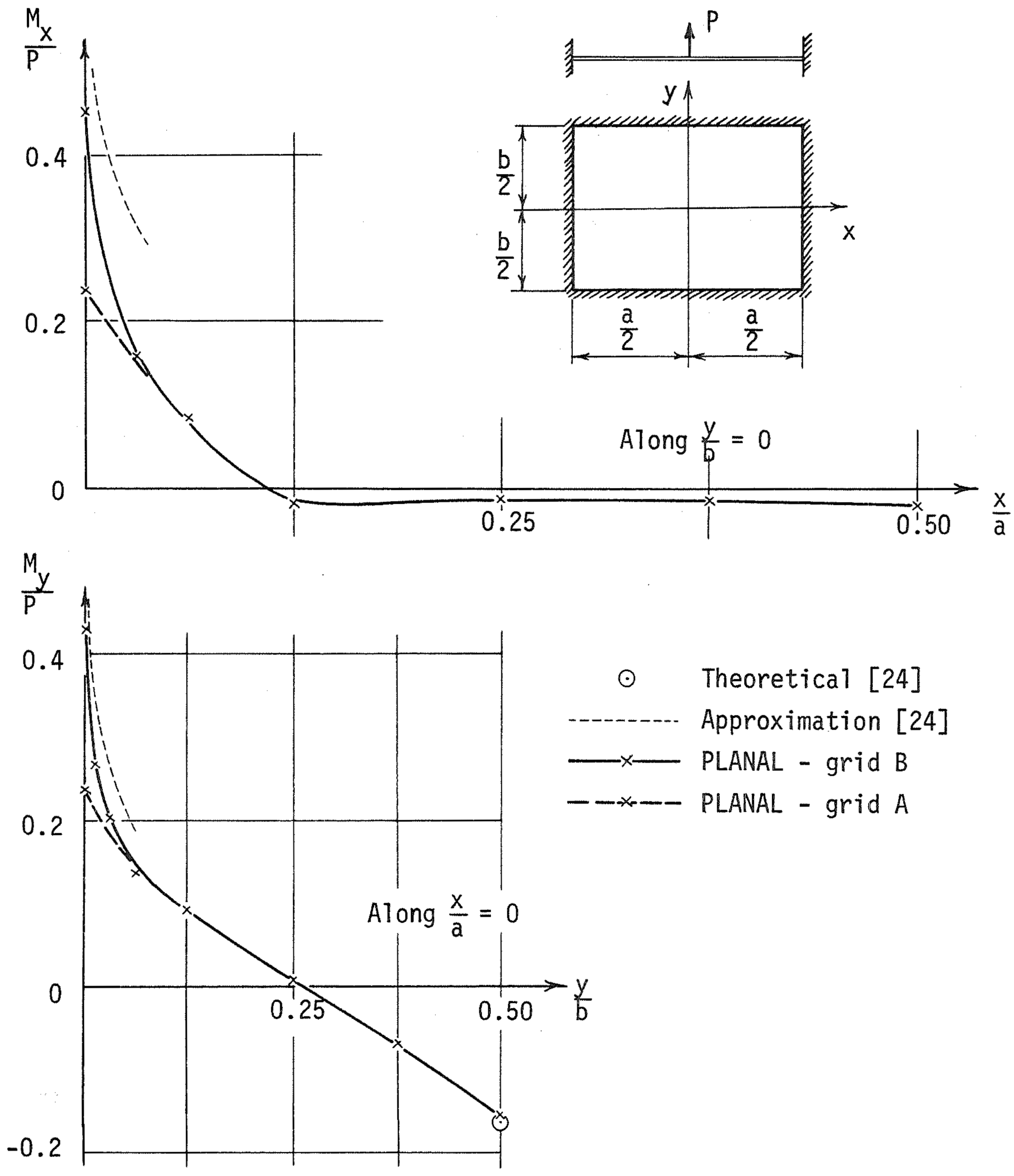
b. For a/b = 2.

Fig. 6.23. Continued.



a. For a/b = 1.

Fig. 6.24. Bending moments of rectangular plates with fixed edges under central load, $\nu = 0.3$.



b. For a/b = 2.

Fig. 6.24. Continued.

6.4. Computation Time.

Computation time required for execution in the PLANAL System is studied in a sample of 21 problems. These problems were executed on the IBM 360/65 computer at the Information Processing Center, Massachusetts Institute of Technology in August, 1969. Most of the examples in the previous sections are included in this sample.

Total execution time (time elapse between entry into and exit from PLANAL) depends on the number of nodes, number of elements, boundary conditions, and other factors. A reasonably simple time study is to plot total execution time versus the total number of nodes. Such a plot for the sample taken is shown in Fig. 6.25.

Execution time for assembling the global coefficient matrix is dependent on both the number of elements and number of nodes. Modification for boundary conditions takes between two and six seconds in the sample. Execution time for solution of the system equations is approximately proportional to the square of NSOL, the number of nodes without completely prescribed displacements or stress functions. The solution operation takes between 0.4 and 13.7 seconds. When the construction of particular solution functions is required, it takes about $(1 + 0.055n)$ seconds, where n is the total number of nodes.

On the same computer, a sample of nine plate bending problems are executed in the STRUDL System [18] using flat plate triangular elements termed 'CPT'. Total execution times for this sample are also plotted in Fig. 6.25. A STRUDL bending problem is solved by a displacement method with three unknowns per node, whereas a PLANAL bending problem is solved by a force method with two unknowns per node. While the time difference may not be entirely due to the difference in the numbers of unknowns, the former apparently takes longer to solve than the latter having the same number of nodes.

In addition to the total execution time, there is an overhead of about 20 seconds per job submitted to the computer. It takes 10 seconds for control to reach ICES and another 10 seconds to reach PLANAL (or STRUDL).

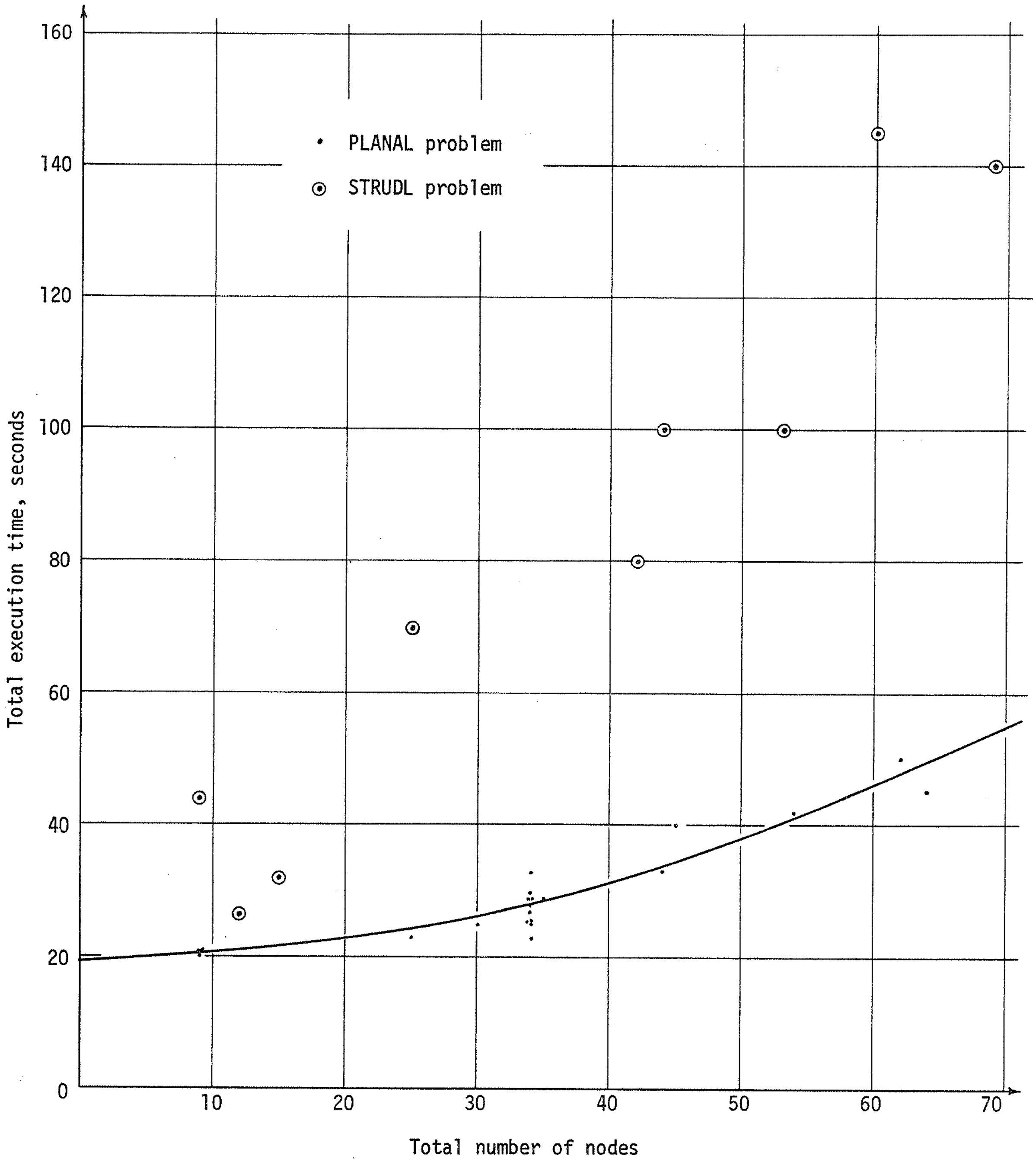


Fig. 6.25. Computation times for samples of PLANAL and STRUDL problems.

CHAPTER 7

CONCLUSIONS AND RECOMMENDATIONS

7.1. Conclusions.

The dual finite element method for analysis of plate structures is implemented into the PLANAL System. The present form of the system is capable of solving problems of plate stretching and bending (Section 5.2).

In the stretching problem, the system can analyze an arbitrary plate under arbitrary loading. Results are obtained in the form of nodal displacements, from which strains and then stresses are computed. These results are in good agreement with theoretical values when they are available.

In the bending problem when particular solutions are known, or not required at all, the system can analyze an arbitrary plate under arbitrary loading. When particular solutions are not known, standard procedure is implemented into the system to generate such solutions for linear loadings and rectangular plates. Results are obtained in the form of stress functions at the nodes, from which moments and curvatures are computed. In the examples studied, results from the system agree closely with theoretical values. Since two unknowns per node are taken in this method, shorter computation time in solving the system equations is realized when compared to a displacement method in which three unknowns per node are taken.

Programming capabilites of the Integrated Civil Engineering System are utilized in the PLANAL System. Features of a problem-oriented language, unrestricted problem size, and efficient programming management are the results of using ICES. The advantages of ICES in the development of structural analysis systems are demonstrated.

Parallel algorithms are implemented to perform a number of operations when the global coefficient matrix is symmetric and when it is non-symmetric. These operations are the assemblage of the global coefficient matrix, modification for boundary conditions, and solution of the system equations.

7.2. Recommendations.

Constant strain triangular elements are used in the development of the dual method in the system. Higher order elements, such as linear strain triangles, may be added to improve the analysis capabilities. In addition to the boundary conditions that can be processed by the present system, a few more may be included, such as: dislocations in multiply-connected plates in stretching; edge beam and elastic boundary in bending. Algorithm for obtaining deflections in the bending problem may be implemented into the system through integration from the computed curvatures.

Standard procedures for obtaining particular solutions in the bending problem for plates with arbitrary geometry and loading may be investigated further. Elias suggests that a finite element method with one unknown moment per node may be used [10].

The dual finite element method is formulated for the plate stretching and bending problems in this work. The method may be readily extended to shallow shells as well as shells approximated by flat triangular elements.

REFERENCES

1. Bazely, G. P., Y. K. Cheung, B. M. Irons, and O. C. Zienkiewicz, "Triangular Elements in Plate Bending--Conforming and Non-conforming Solutions," *Proceedings*, Conference on Matrix Methods in Structural Mechanics, Wright-Patterson Air Force Base, Ohio, Oct., 1965.

2. Clough, R. W., "The Finite Element Method in Structural Mechanics," *Stress Analysis*, O. C. Zienkiewicz and G. S. Holister, editors, John Wiley and Sons, Inc., New York, 1965, p.100.

3. Clough, R. W., and J. L. Tocher, "Finite Element Stiffness Matrices for Analysis of Plate Bending," *Proceedings*, Conference on Matrix Methods in Structural Mechanics, Wright-Patterson Air Force Base, Ohio, Oct., 1965.

4. Connor, J. J., "Lecture Notes on the Finite Element Displacement Method," Dept. of Civil Engrg., M.I.T., Cambridge, Mass., 1966 (unpublished).

5. Connor, J. J., "Application to Plane Stress and Plane Strain--Linear Elastic Material," notes for Special Summer Program on Finite Element Methods in Solid Mechanics, M.I.T., Cambridge, Mass., June, 1968.

6. Connor, J. J., and G. Will, "Computer-aided teaching of the Finite Element Displacement Method," *Research Report 69-23*, Dept. of Civil Engrg., M.I.T., Cambridge, Mass., Feb., 1969.

7. De Veubeke, B. F., "Displacement and Equilibrium Models in the Finite Element Method," *Stress Analysis*, O. C. Zienkiewicz and G. S. Holister, editors, John Wiley and Sons, Inc., New York, 1965, p.173.

8. Elias, Z. M., "On the Duality Between the Problems of Stretching and Bending of Plates," *CR-705*, National Aeronautics and Space Administration, Washington, D.C., Jan., 1967.

9. Elias, Z. M., "Dual Finite Element Method for Stretching and Bending of Plates," *Tech. Report R67-16*, Dept. of Civil Engrg., M.I.T., Cambridge, Mass., May, 1967.

10. Elias, Z. M., "Duality in Finite Element Methods," *Journal of the Engineering Mechanics Division*, Am. Soc. of Civil Engr., Vol. 94, No. EM4, Proc. Paper 6074, Aug., 1968, pp. 931-946.

11. Ferrante, A. J., "A System for Finite Analysis," S.M. Thesis, Dept. of Civil Engrg., M.I.T., Cambridge, Mass., Jan., 1967.

12. Herrmann, L. R., "A Bending Analysis for Plates," *Proceedings*, Conference on Matrix Methods in Structural Mechanics, Wright-Patterson Air Force Base, Ohio, Oct., 1965.

13. Hildebrand, F. B., *Introduction to Numerical Analysis*, McGraw-Hill Book Co., Inc., New York, N.Y., 1956.

14. Hildebrand, F. B., *Advanced Calculus for Applications*, Prentice-Hall, Inc., Englewood Cliffs, N.J., 1962.

15. "ICES: Concepts and Facilities," Preliminary Edition, Civil Engineering Systems Laboratory, Dept. of Civil Engrg., M.I.T., Cambridge, Mass., 1965.

16. "ICES: Programmer's Guide," Preliminary Edition, Civil Engineering Systems Laboratory, Dept. of Civil Engrg., M.I.T., Cambridge, Mass., 1965.

17. "ICES STRUDL-I: The Structural Design Language - Internal Logic Manual," Structures Division and Civil Engineering Systems Laboratory, Dept. of Civil Engrg., M.I.T., Cambridge, Mass., Apr., 1968. (Released by IBM Corporation, Hawthorne, N.Y. as documentation to Distribution 360D 16.2.015.)

18. "ICES STRUDL-I: The Structural Design Language - Engineering User's Manual," Structures Division and Civil Engineering Systems Laboratory, Dept. of Civil Engrg., M.I.T., Cambridge, Mass., Sep., 1967.

19. Nagy, D. A., "A Dual Finite Element Computer System for Stretching and Bending of Orthotropic Plates," S.M. Thesis, Dept. of Civil Engrg., M.I.T., Cambridge, Mass., Aug., 1967.

20. Prato, C. A., "A Mixed Finite Element Method for Thin Shell Analysis," Ph.D. Thesis, Dept. of Civil Engrg., M.I.T., Cambridge, Mass., May, 1968.

21. Roos, D., *ICES System Design*, M.I.T. Press, Cambridge, Mass., 1966.

22. Rosenhaupt, S., "Stresses in Point Supported Composite Walls," *Journal of the American Concrete Institute*, Vol. 61, No. 7, July, 1964.

23. Timoshenko, S., and J. N. Goodier, *Theory of Elasticity*, Second Edition, McGraw-Hill Book Co., Inc., New York, N.Y., 1951.

24. Timoshenko, S., and S. Woinowsky-Krieger, *Theory of Plates and Shells*, Second Edition, McGraw-Hill Book Co., Inc., New York, N.Y., 1959.

25. Whang, B., "Elasto-plastic Analysis of Orthotropic Plates and Shells," Ph.D. Thesis, Dept. of Civil Engrg., M.I.T., Cambridge, Mass., Oct., 1968.

26. Zienkiewicz, O. C., "Finite Element Procedures in the Solution of Plate and Shell Problems," *Stress Analysis*, O. C. Zienkiewicz and G. S. Holister, editors, John Wiley and Sons, Inc., New York, N.Y., 1965, p.121.

27. Zienkiewicz, O. C., *The Finite Element Method in Structural and Continuum Mechanics*, McGraw-Hill Publishing Company Limited, London, England, 1967.

BIOGRAPHY

The author was born in Hong Kong in 1942 and he received his primary and secondary education there. He attended Purdue University in 1961 where he received his bachelor's degree in 1964 and master's degree in 1965. He was later awarded a graduate assistantship to study at the Massachusetts Institute of Technology.

During several summers, he was first associated with the Indiana State Highway Commission in Indianapolis, and later the U.S. Army Engineers Waterways Experiment Station at Vicksburg, Mississippi.

The author is a member of Omicron Delta Kappa, Tau Beta Pi, Chi Epsilon, and Phi Eta Sigma. He is also an associate member of the American Society of Civil Engineers.

APPENDIX A

NUMERICAL APPROXIMATIONS

When computations of certain quantities are made in the PLANAL System, numerical differentiation, integration, and interpolation are approximated by Lagrangian methods [13]. For example, nodal strains may be obtained from nodal displacements through differentiation. Using Langrange's interpolation formula of second degree, we obtain the following approximations for a function f having ordinates f_1, f_2, f_3 at x_1, x_2, x_3, respectively (Fig. A.1).

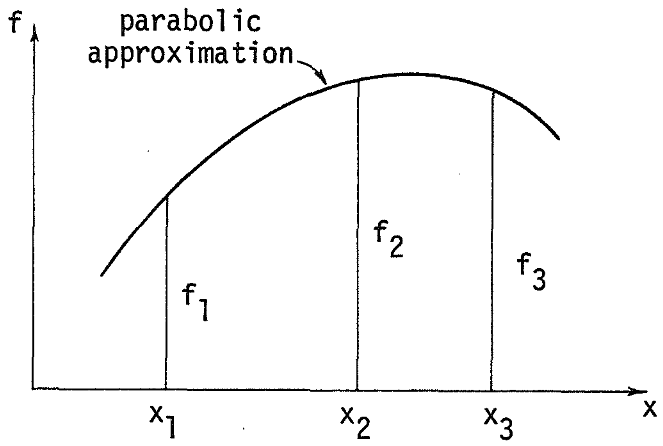


Fig. A.1. Function approximated by a parabola.

Numerical Differentiation.

$$f'(x_1) \simeq -\frac{2d_1 + d_2}{d_1(d_1 + d_2)} f_1 + \frac{d_1 + d_2}{d_1 d_2} f \quad - \frac{d_1}{d_2(d_1 + d_2)} f_3,$$

$$f'(x_2) \simeq -\frac{d_2}{d_1(d_1 + d_2)} f_1 - \frac{d_1 - d_2}{d_1 d_2} f_2 + \frac{d_1}{d_2(d_1 + d_2)} f_3,$$

$$f'(x_3) \simeq \frac{d_2}{d_1(d_1 + d_2)} f_1 - \frac{d_1 + d_2}{d_1 d_2} f_2 + \frac{d_1 + 2d_2}{d_2(d_1 + d_2)} f_3.$$

Numerical Integration.

$$\int_{x_1}^{x_2} f\,dx \simeq \frac{d_1(2d_1 + 3d_2)}{6(d_1 + d_2)} f_1 + \frac{d_1(d_1 + 3d_2)}{6d_2} f_2 - \frac{d_1^3}{6d_2(d_1 + d_2)} f_3,$$

$$\int_{x_2}^{x_3} f\,dx \simeq -\frac{d_2^3}{6d_1(d_1 + d_2)} f_1 + \frac{d_2(3d_1 + d_2)}{6d_1} f_2 + \frac{d_2(3d_1 + 2d_2)}{6(d_1 + d_2)} f_3.$$

Numerical Interpolation.

$$f(x) \simeq \frac{e_2 e_3}{d_1(d_1 + d_2)} f_1 - \frac{e_3 e_1}{d_1 d_2} f_2 + \frac{e_1 e_2}{d_2(d_1 + d_2)} f_3,$$

where $e_i = x - x_i$.

APPENDIX B

SUMMARY OF PLANAL COMMANDS

A summary of all the commands in PLANAL are listed here for user's reference. These commands are explained in detail in Chapter 5.

1. *Problem Initiation.*

$$\underline{\text{PLANAL}}\ \text{['name'] ['title']}$$

$$\underline{\text{DEBUG}} \begin{Bmatrix} \underline{\text{ALL}} \\ \underline{\text{COMMON}} \end{Bmatrix}$$

$$\underline{\text{PLD}}\text{EBUG}$$

2. *Type Specification.*

$$\underline{\text{TYPE}} \begin{Bmatrix} \text{PLATE } \underline{\text{STR}}\text{ETCHING} \\ \text{PLATE } \underline{\text{BEN}}\text{DING} \end{Bmatrix} \begin{Bmatrix} \text{leave blank} \\ \underline{\text{SYM}}\text{METRICAL} \\ \underline{\text{NON}}\text{SYMMETRICAL} \end{Bmatrix}$$

3. *Unit Declaration.*

$$\underline{\text{UNI}}\text{TS} \begin{Bmatrix} \underline{\text{IN}}\text{CHES, } \underline{\text{FE}}\text{ET, } \underline{\text{FT}}\text{, } \underline{\text{CENTIM}}\text{ETERS, } \underline{\text{CM}}\text{, or } \underline{\text{M}}\text{ETERS} \\ \underline{\text{POU}}\text{NDS, } \underline{\text{LB}}\text{, } \underline{\text{KIPS}}\text{, } \underline{\text{TONS}}\text{, } \underline{\text{KIL}}\text{OGRAMS, } \underline{\text{KG}}\text{, or } \underline{\text{MTON}} \\ \underline{\text{RAD}}\text{IANS, or } \underline{\text{DEG}}\text{REES} \\ \underline{\text{FAH}}\text{RENHEIT, or } \underline{\text{CENTIG}}\text{RADE} \\ \underline{\text{SEC}}\text{ONDS, } \underline{\text{MIN}}\text{UTES, or } \underline{\text{HOURS}} \end{Bmatrix}$$

4. *Geometry and Topology.*

NODE COORDINATES

[node name] X [v_x] Y [v_y] $\left\{\begin{array}{l}\text{BOUNDARY}\\ \text{leave blank}\end{array}\right\}$

ELEMENT INCIDENCES

[element name] [node 1] [node 2] [node 3]

BOUNDARY INCIDENCES

[boundary name] [node name]

5. *Element Properties Specification.*

ELEMENT PROPERTIES TYPE ['type']

[element name list] THICKNESS [v_t] EX [v_{ex}] EY [v_{ey}] -

PX [v_{px}] PY [v_{py}] CTX [v_{cx}] CTY [v_{cy}] G [v_g] DENSITY [v_d]

6. *Boundary Condition Specification.*

(1) BOUNDARY CONDITION ['boundary name'] DISPLACEMENT
[boundary portion] U [v_u] V [v_v] W [v_w] R [v_r]

(2) BOUNDARY CONDITION ['boundary name'] STRESS
[boundary portion] NX [v_{nx}] NY [v_{ny}] Q [v_q] M [v_m] -
ROTATION [v_r]

(3) BOUNDARY CONDITION ['boundary name'] MIXED STRETCHING
[boundary portion] UR [v_{ur}] NR [v_{nr}] ANGLE [v_a]

(4) BOUNDARY CONDITION ['boundary name'] ELASTIC
[boundary portion] US [v_{us}] VS [v_{vs}] KXX [v_{kxx}] KXY [v_{kxy}] -
KYX [v_{kyx}] KYY [v_{kyy}]

(5) BOUNDARY CONDITION ['boundary name'] EDGE BEAM
[boundary portion] NX [v_{nx}] NY [v_{ny}] EB [v_e] AB [v_a] IZ [v_i]

(6) BOUNDARY CONDITION ['boundary name'] STRAIN
[boundary portion] EPSILON [v_e] CHI [v_c] ROTATION [v_r]

(7) BOUNDARY CONDITION ['boundary name'] FUNCTION
[boundary portion] U [v_u] V [v_v]

(8) BOUNDARY CONDITION ['boundary name'] MIXED BENDING
[boundary portion] UR [v_{ur}] CHI [v_c] ANGLE [v_a]

(9) BOUNDARY CONDITION ['boundary name'] SIMPLE SUPPORT
[boundary portion]

(10) BOUNDARY CONDITION ['boundary name'] FIXED SUPPORT
[boundary portion]

(11) BOUNDARY CONDITION ['boundary name'] FREE
[boundary portion]

(12) BOUNDARY CONDITION ['boundary name'] SYMMETRY
[boundary portion]

7. *Loading Specification.*

LOADING

$\left\{\begin{array}{l}\text{NODES [node names]}\\ \text{UNIFORM}\end{array}\right\}$ $\left\{\begin{array}{l}\text{INTENSITY}\\ \text{FORCE}\end{array}\right\}$ X [v_x] Y [v_y] Z [v_z]

8. *Particular Solution Functions for the Bending Problem.*

BENDING PARTICULAR SOLUTION

NODES [node names] KX [K_x] KY [K_y] KXX [$K_{x,x}$] KYY [$K_{y,y}$]

9. *Output and Analysis Commands.*

$$\underline{\text{OUT}}\text{PUT} \begin{Bmatrix} \underline{\text{NOD}}\text{ES} \\ \underline{\text{ELE}}\text{MENTS} \end{Bmatrix} \begin{Bmatrix} \underline{\text{DISPL}}\text{ACEMENTS} \\ \underline{\text{STRA}}\text{INS} \\ \underline{\text{STRESS}}\text{ES} \\ \underline{\text{PRINC}}\text{IPAL } \underline{\text{STRA}}\text{IN} \\ \underline{\text{PRINC}}\text{IPAL } \underline{\text{STRESS}}\text{ES} \\ \underline{\text{ALL}} \end{Bmatrix} \text{ or } \begin{Bmatrix} \underline{\text{FUNCT}}\text{IONS} \\ \underline{\text{MOM}}\text{ENTS} \\ \underline{\text{CURV}}\text{ATURES} \\ \underline{\text{PRINC}}\text{IPAL } \underline{\text{MOM}}\text{ENTS} \\ \underline{\text{PRINC}}\text{IPAL } \underline{\text{CURV}}\text{A-} \\ \text{TURES} \\ \underline{\text{ALL}} \end{Bmatrix}$$

(in stretching) (in bending)

<u>FINITE</u> ELEMENT ANALYSIS

10. *Termination Statement.*

<u>FINISH</u>

APPENDIX C

COMMON MAP

The COMMON map of the PLANAL System is presented in this appendix. ICES requires that all variables used in the Command Definition Blocks (programs written in CDL) and all dynamic arrays must appear in COMMON. The relative addresses and the displacements (both in hexadecimals and decimals) from the beginning of COMMON of all such variables and arrays are listed. When the mode of a variable [16] does not conform to the FORTRAN convention of naming a variable, it will be so indicated: D = double word, H = half word integer, R = real variable. A dynamic array base pointer is indicated by P. Remarks or brief definitions of the variables are also given. Dummy areas which are not used by the system are also shown.

Name	Rel. Add.	Displacement Hex.	Displacement Dec.	Remarks
		ICES COMMON POOL		
QQDUB(1)	1	000	0000	
QQDUB(2)	2	004	0004	
ICOM	3	008	0008	
IERROR	4	00C	0012	
ICOML	5	010	0016	
QQCOM(1)	6	014	0020	
•	•	•	•	
•	•	•	•	
QQCOM(75)	80	13C	0316	

Name		Rel. Add.	Displacement Hex.	Dec.	Remarks
		SCRATCH COMMOM POOL			
I1		81	140	0320	
•		•	•	•	
•		•	•	•	
I36		116	1CC	0460	
T1		117	1D0	0464	
•		•	•	•	
•		•	•	•	
T36		152	25C	0604	
D1	D	153	260	0608	
•		•	•	•	
•		•	•	•	
D10	D	171	2A8	0680	
		173	2B0	0688	DUMMY.
		•	•	•	•
		•	•	•	•
		180	2CC	0716	DUMMY.
NSOL		181	2D0	0720	NO. OF NODES AT WHICH DISPLACEMENTS ARE NOT FULLY PRESCRIBED.
NDIS		182	2D4	0724	NO. OF PRESCRIBED DISP. COMPONENTS.
NLDSI		183	2D8	0728	NO. OF INDEPENDENT LOADING. CONDITIONS.
		184	2DC	0732	DUMMY.
		•	•	•	•
		•	•	•	•
		236	3AC	0940	DUMMY.
IBAND	P	237	3B0	0944	SEMIBANDWIDTH OF HYPERCOLUMNS OF COEFFICIENT MATRIX.
IFDT	P	239	3B8	0952	BIT PICTURE OF COEFFICIENT MATRIX.
KDIAG	PD	241	3C0	0960	DIAGONAL SUBMATRICES OF COEF. MAT.
KOFDG	PD	243	3C8	0968	OFF-DIAG. SUBMATRICES OF COEF. MAT.
IOFDG	PH	245	3D0	0976	NON-ZERO SUBMATRICES OF EACH ROW OF COEFFICIENT MATRIX.
KPPRI	PD	247	3D8	0984	RIGHT-HAND MEMBERS OF MAT. EQ.
FCMAT	PD	249	3E0	0992	NON-SYMMETRIC COEF. MATRIX ELEMENTS.
ICUREL	P	251	3E8	1000	NON-ZERO ROWS IN EACH COLUMN OF NON-SYMMETRIC COEF. MATRIX.
IREL1	P	253	3F0	1008	NON-ZERO COLUMNS IN EACH ROW OF NON-SYMMETRIC COEF. MATRIX.
ICUINT	P	255	3F8	1016	INVERSE USE OF ICUREL.
		NON-DICTIONARY COMMON POOL			
		257	400	1024	DUMMY.
		•	•	•	•
		•	•	•	•
		262	414	1044	DUMMY.
ISCAN		263	418	1048	SCANNING MODE INDICATOR.

Name		Rel. Add.	Displacement Hex.	Dec.	Remarks
		264	41C	1052	DUMMY.
		265	420	1056	DUMMY.
		266	424	1060	DUMMY.
IDUMP		267	428	1064	INDICATOR FOR INTERMEDIATE OUTPUT.
IPOOL		268	42C	1068	SIZE OF DATA POOL.
		269	430	1072	DUMMY.
		.	.	.	.
		.	.	.	.
		315	4E8	1256	DUMMY.
CFLEN		316	4EC	1260	CONVERSION FACTOR FOR LENGTH.
CFWT		317	4F0	1264	CONVERSION FACTOR FOR WEIGHT.
CFANG		318	4F4	1268	CONVERSION FACTOR FOR ANGLE.
CFTEMP		319	4F8	1272	CONVERSION FACTOR FOR TEMPEATURE.
CFTIME		320	4FC	1276	CONVERSION FACTOR FOR TIME.
		321	500	1280	DUMMY.
		.	.	.	.
		.	.	.	.
		330	524	1316	DUMMY.

DICTIONARY COMMON POOL

LDID	PD	331	528	1320	LOADING NAMES.
		333	530	1328	DUMMY.
LEXTN		334	534	1332	TOTAL NO. OF LOADINGS.
LTYP	P	335	538	1336	LOADING TYPE.
		337	540	1344	DUMMY.
LDLIST	P	338	548	1352	LOADING LIST.
LDTLE	PR	340	54C	1356	LOADING TITLES.
JTID	PD	342	554	1364	NODE NAMES.
		344	55C	1372	DUMMY.
JEXTN		345	560	1376	TOTAL NO. OF NODES.
JTYP	PH	346	564	1380	NODE TYPE.
		348	56C	1388	DUMMY.
		349	570	1392	DUMMY.
		350	574	1396	DUMMY.
JTXYZ	PR	351	578	1400	NODE COORDINATES.
JTLOD	PR	353	580	1408	NODAL LOADS.
IDLDND	PH	355	588	1416	EXT. NO. OF NODES WITH SPEC. LOAD.
LODTYP		357	590	1424	GENERAL LOADING TYPE.
		358	594	1428	DUMMY.
		.	.	.	.
		.	.	.	.
		399	638	1592	DUMMY.
NJ		400	63C	1596	
		401	640	1600	DUMMY.
NLDS		402	644	1604	NO. OF ACTIVE LOADING CONDITIONS.
JF		403	648	1608	NO. OF DEGREES OF FREEDOM.
ID		404	64C	1612	PROBLEM TYPE.
		405	650	1616	DUMMY.
		.	.	.	.

Name		Rel. Add.	Displacement Hex.	Dec.	Remarks
		.	.	.	.
		409	660	1632	DUMMY.
JINT	PH	410	664	1636	ARRAY IN CORRESPONDENCE WITH JEXT.
JEXT	PH	412	66C	1644	LOCATION OF NODES IN MATRIX EQ.
		414	674	1652	DUMMY.
		.	.	.	.
		.	.	.	.
		439	6D8	1752	DUMMY.
ELID	PD	440	6DC	1756	ELEMENT NAMES.
ELTYP	P	442	6E4	1764	ELEMENT TYPE.
ELPROP	P	444	6EC	1772	ELEMENT PROPERTIES.
ELTOP	PH	446	6F4	1780	NODE INCIDENCE ON ELEMENTS.
ELOADS	P	448	6FC	1788	ELEMENT LOADS.
ELTOP1	PH	450	704	1796	ELEMENT INCIDENCE ON NODES.
IUNIPR		452	70C	1804	INDICATOR FOR UNIF. EL. PROPERTIES.
NDPROP	PR	453	710	1808	NODE PROPERTIES.
		455	718	1816	DUMMY.
		.	.	.	.
		.	.	.	.
		463	738	1848	DUMMY.
NBXTEL		464	73C	1852	TOTAL NO. OF ELEMENTS.
NBEL		465	740	1856	NO. OF ACTIVE ELEMENTS.
NSYM		466	744	1860	SYMMETRY INDICATOR.
NGEN		467	748	1864	GENERAL INDICATOR.
ELSTDE		468	74C	1868	STANDARD YOUNG'S MODULUS.
ELSTDG		469	750	1872	STANDARD SHEAR MODULUS.
ELSTCT		470	754	1876	STANDARD COEF. OF THERMAL EXPANSION.
ELSTDS		471	758	1880	STANDARD DENSITY.
ELSTPO		472	75C	1884	STANDARD POISSON'S RATIO.
		473	760	1888	DUMMY.
		474	764	1892	DUMMY.
		475	768	1896	DUMMY.
		476	76C	1900	DUMMY.
ELSTMT	P	477	770	1904	ELEMENT STIFFNESS.
NODISP	PR	479	778	1912	NODAL DISPLACEMENTS.
VALUEN	P	481	780	1920	OUTPUT VALUES AT NODES.
VALUEE	P	483	788	1928	OUTPUT VALUES OF ELEMENTS.
GRIDPR	P	485	790	1936	PROPERTIES OF GRID LINES.
KODOUT	P	487	798	1944	CODE FOR OUTPUT.
GRID	PH	489	7A0	1952	GRID PATTERN OF NODES.
KPBSLN	PH	491	7A8	1960	TEMPORARY ARRAY FOR PARTICULAR SOL.
IDEBUG		493	7B0	1968	INDICATOR FOR DEBUGGING.
IPROB		494	7B4	1972	PROBLEM PHASE INDICATOR.
IBCON		495	7B8	1976	NO. OF CLOSED BOUNDARY CURVES.
BDID	PD	496	7BC	1980	BOUNDARY NAMES.
BDCOND	P	498	7C4	1988	BOUNDARY CONDITIONS.
PBNTEM	P	500	7CC	1996	TEMPORARY ARRAY FOR PARTICULAR SOL.
PBSOLN	P	502	7D4	2004	NODAL VALUES OF PARTICULAR SOLUTION.
PBSOLE	P	504	7DC	2012	ELEMENT VALUES OF PARTICULAR SOL.
RINTND	P	506	7E4	2020	NODAL LOAD INTENSITY.

Name		Rel. Add.	Displacement Hex.	Dec.	Remarks
RFORND	P	508	7EC	2028	NODAL LOAD FORCE.
IPRTIC		510	7F4	2036	INDICATOR FOR PARTICULAR SOLUTION.
ILOADN		511	7F8	2040	TYPE OF NODE LOAD (INT. OR FORCE).
BDPOS	PH	512	7FC	2044	BOUNDARY POSITION OF NODES.
		514	804	2052	DUMMY.
		515	808	2056	DUMMY.
TEMOUT	PH	516	80C	2060	CODE FOR TEMPORARY OUTPUT.

APPENDIX D

DATA STRUCTURE

The definition and structure of the data used in the PLANAL System is presented in this appendix. All arrays defined here are dynamic arrays which must appear in COMMON (as compared with dimensioned arrays) unless specified otherwise. Some scalar quantities in COMMON are also defined here (others are defined in Appendix C). The arrays and scalars are listed alphabetically for easy reference.

Each node, element, or boundary of a structure has both a name and an external number. A name is used for identification by the user and can be alphameric. An external number is an integer assigned by the system to a node, element, or boundary according to the order of their appearance in the input. Separate sets of consecutive integers starting from 1 are assigned to the nodes, elements, and boundaries. In addition, each node is assigned an internal number according to the position of the unknowns related to that node in the system equations.

BDCOND Three level full word array to store the boundary conditions (B.C.).

```
DEFINE BDCOND,1,POINTER,STEP = 1
   DEFINE BDCOND(I),10,POINTER,STEP = 10
      DEFINE BDCOND(I,J),5,STEP = 5
```

where I = external number of a boundary;

J = order of nodes in counter-clockwise direction around the boundary starting with the node specified in the 'BOUNDARY INCIDENCE' command.

The boundary values to be entered to the third level (indicated by K) are assembled according to the type of boundary condition encountered (Table A.1).

BDCOND(I,J,1) = external number (EN) of the boundary node N considered,

BDCOND(I,J,2) = code for stretching B.C. at negative side of N,

BDCOND(I,J,3) = code for stretching B.C. at positive side of N,

BDCOND(I,J,4) = code for bending B.C. at negative side of N,

BDCOND(I,J,5) = code for bending B.C. at positive side of N,

Boundary values (n = 3,4,...,19):

BDCOND(I,J,2n) = boundary value at negative side of N,

BDCOND(I,J,2n+1) = boundary value at positive side of N.

The code for B.C. is cumulative so that more than one B.C. that exist at a node can be indicated. The types of B.C. are:

(1) displacement,
(2) stress,
(3) elastic,
(4) edge beam,
(5) mixed stretching,
(6) strain,
(7) function,
(8) mixed bending,
(9) simple support,
(10) fixed support,
(11) free,
(12) symmetry.

BDID Two level double word array to store the alphameric name of a boundary, number of nodes on the boundary, and processing information.

Table A.1. Data Structure of BDCOND.

Value of K	Type of Boundary Condition											
	Displacement (1)	Stress (2)	Elastic (3)	Edge beam (4)	Mixed stret. (5)	Strain (6)	Function (7)	Mixed bend. (8)	Simple supp. (9)	Fixed supp. (10)	Free (11)	Symmetry (12)
1	EN	EN	EN	EN	EN	EN	EN	EN	EN	EN	EN	EN
2 3	1	2	4	8	16	32						
4 5	1	2					64	128	256	512	1024	2048
6 7		NX	US	NX		EPS						
8 9		NY	VS	NY		CHI						
10 11			KXX	EB		ROT						
12 13			KXY	AB								
14 15			KYX	IZ								
16 17			KYY									
18 19												
20 21	W	Q										
22 23	R	M										
24 25		ROT										
26 27												
28 29												
30 31	U						U					
32 33	V						V					
34 35					UR			UR				
36 37					NR			CHI				
38 39					ANG			ANG				

DEFINE BDID,1,POINTER,STEP = 1

DEFINE BDID(I),3,DOUBLE

where I is the external number of the boundary.

BDID(I,1) = name of boundary,

BDID(I,2) = number of nodes on boundary,

BDID(I,3) = indicator for necessity to re-process boundary as a stretching problem in an actually bending problem. Necessary if > 1. Set in STHBOU.

BDNORM Two level full word array to store boundary normals.

DEFINE BDNORM,IBCON,POINTER

DEFINE BDNORM(I),J

where J = number of nodes on current boundary.

Defined in HPSSLS. Constructed if ISSLSB ⩾ 1.

BDPOS One level half word array to store boundary position of nodes on boundary.

DEFINE BDPOS,JEXTN,HALF

BDPOS(I) = boundary position, where I is the external number of a node. Defined and constructed in STHTCE.

ELEXT One level half word array to store external numbers of elements.

DEFINE ELEXT,NBEL,HALF

ELID One level double word array to store the alphameric identification of an element.

DEFINE ELID,10,DOUBLE,STEP = 10

ELPROP Two level full word array to store element properties.

DEFINE ELPROP,10,POINTER,STEP = 10

DEFINE ELPROP(I),13

where I = external number of an element.

ELPROP(I,1) = element type name,
ELPROP(I,2) = thickness,
ELPROP(I,3) = area,
ELPROP(I,6) = Young's modulus in x-direction,
ELPROP(I,7) = Young's modulus in y-direction,
ELPROP(I,8) = Poisson's ratio in x-direction,
ELPROP(I,9) = Poisson's ratio in y-direction,
ELPROP(I,10) = coefft. of therm. exp. in x-direction,
ELPROP(I,11) = coefft. of therm. exp. in y-direction,
ELPROP(I,12) = shear modulus,
ELPROP(I,13) = density.

ELSTMT Three level full word array to store the lower elements of the local stiffness matrix of each element.

DEFINE ELSTMT,NBEL,6,JF*JF

A typical element is ELSTMT(I,J,K). The external number of an element is indicated by I. The 6x6 element stiffness matrix of each element I is partitioned by nodes. Because of symmetry, only the lower submatrices numbered (indicated by J) are stored:

$$\begin{bmatrix} 1 & & \\ 2 & 3 & \\ 4 & 5 & 6 \end{bmatrix}$$

The matrix elements of each submatrix J are stored in the following order (indicated by K):

$$\begin{bmatrix} 1 & 2 \\ 3 & 4 \end{bmatrix}$$

The structure of ELSTMT for each element I can be summarized by indicating the last two subscripts of the matrix elements in their positions in the element stiffness matrix:

$$\begin{bmatrix} 1,1 & 1,2 & & & & \\ 1,3 & 1,4 & & & & \\ 2,1 & 2,2 & 3,1 & 3,2 & & \\ 2,3 & 2,4 & 3,3 & 3,4 & & \\ 4,1 & 4,2 & 5,1 & 5,2 & 6,1 & 6,2 \\ 4,3 & 4,4 & 5,3 & 5,4 & 6,3 & 6,4 \end{bmatrix}$$

ELTOP Two level half word array to store node incidence on the elements.

DEFINE ELTOP,10,POINTER,STEP = 10

DEFINE ELTOP(I),6,HALF

where I = external number of an element.

ELTOP(I,1) = total number of nodes in the element,

ELTOP(I,n) = node incidence in counter-clockwise direction (n = 2, ..., 6), with two nodes repeated for convenience.

ELTOP1 Two level half word array to store element incidence on the nodes.

DEFINE ELTOP1,JEXTN,5,HALF,STEP = 5

ELTOP1(I,1) = total number of elements incident on a node,

ELTOP1(I,n) = elements incident on the node (external numbers used), n = 2, 3, ...,

where I = external node number.

Defined and constructed in STHTCE.

FCMAT Three level double word array to store non-symmetric global coefficient matrix.

DEFINE FCMAT,NJ,5,POINTER,STEP = 5

DEFINE FCMAT(I,J),4,DOUBLE

In FCMAT(I,J,K), the matrix elements (indicated by K) are stored in the following order:

$$\begin{bmatrix} 1 & 2 \\ 3 & 4 \end{bmatrix}$$

Defined in STHNAS,HNSASS,HNSLAS,STHSAS.

GRID Three level half word array - to store the rectangular grid pattern of the nodes. External numbers of the nodes are used for identification.

DEFINE GRID(I),J,K,HALF

where I designates the axis to which the grid lines are parallel (1 = x, 2 = y),

J is the number of lines in a direction,

K: first element contains the number of nodes of line J, subsequent elements contain the external number of the nodes ordered in positive x- or y-direction.

Defined in STHGRI.

GRIDPR Three level full word array to store the properties of a grid line used in STHP1R.

DEFINE GRIDPR(I,J),4

GRIDPR(I,J,1) = type of grid line,

GRIDPR(I,J,2) = type of end condition combination,

GRIDPR(I,J,3) = I-coordinate of first node,

GRIDPR(I,J,4) = I-coordinate of last node.

Defined and constructed in STHP1R.

IBCON Scalar - number of closed boundary curves bounding the plate.

ID Scalar - indicator for problem type.

10: plate stretching,

11: plate bending,

12: general.

IDUMP Scalar - indicator for intermediate output.

ICUINT Two level half word array - an inverse use of ICUREL.

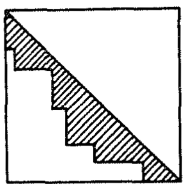
DEFINE ICUINT,NSOL,5,HALF,STEP = 5

ICUINT(I,1) = the hyper-column position K at which hyper-row I in the banded region* of global stiffness matrix starts,

ICUINT(I,n) = position of submatrix(I,J) of the global stiffness matrix in FCMAT(I), n = 2, 3, ...,

where (k-1) + (n-1) = J.

Defined and initialized in STHNSL.



* The banded region is defined such that the first entry (column position) of any row cannot be greater than that of any subsequent row. This situation is shown in the diagram.

ICUREL Two level half word array to contain information on the row structure of the non-symmetric global coefficient matrix.

DEFINE ICUREL,NJ,5,HALF,STEP = 5

ICUREL(I,1) = number of non-zero submatrices in hyper-row I,

ICUREL(I,n) = position (hyper-column number) of FCMAT(I,n-1) in the global stiffness matrix, where n = 2, 3, ...

Defined and initialized in STHNAS.

IDLDND One level half word array to contain external number of nodes at which loads are specified when special loading applies. For example, "planar" distributed load is defined by load intensities at three nodes.

DEFINE IDLDND,5,HALF,STEP = 5

IDLDND(1) = number of such "special loading" nodes,

IDLDND(n) = external numbers of these nodes, n = 2, 3, ...

Defined in STHLOD.

ILOADN Scalar - indicator for type of load applied at node (code is cumulative). Set in STHLOD.

1: load intensity,

2: load force.

IOFDG Two level half word array to contain information on the row structure of the symmetric global coefficient matrix.

DEFINE IOFDG,NJ,6,HALF,STEP = 5

IOFDG(I,1) = number of non-zero submatrices to the left of the diagonal in row I,

IOFDG(I,n) = position (hypercolumn number) of the non-zero submatrices in the array KOFDG (n = 2, 3, ...).

IPROB Scalar - indicator of problem phase.

0: stretching problem,

-1: bending problem,

2: general problem (both stretching and bending).

IPRTIC Scalar - indicator of the type of particular bending solution specified by the user.

0: particular solution not given,

1: nodal values of particular solution specified or computed,

2: area integral of particular solution specified in PBSOLE.

IPRTIC is set to 1 in STH2FS so that KPPRI can be computed in STHBLV.

IREL1 Two level full word array to contain the bit picture of the non-symmetric global coefficient matrix.

DEFINE IREL1,NJ,I,HALF

where I = (NJ+31)/32

First level denotes the position of a hyper-column of global coefficient matrix.

Defined and initialized in STHNAS.

ISCAN Scalar - scanning mode indicator.
1: normal execution of programs,
2: execution inhibited.

ISSLSB Scalar - indicator for the presence of simple support or line of symmetry boundary conditions in bending.
0: not present,
>0: present.

IUNIPR Scalar - indicator for uniformity of thickness and material properties in all elements.
0: not uniform,
1: uniform.

JEXT One level half word array to store information for location of nodes in the system equation. External numbers of the nodes are used for storage. Nodes are assigned contiguous locations in the order of their appearance in the input. Nodes without complete restraints fill the array downwards starting from the top; nodes with complete restraints fill the array upwards from the bottom.
DEFINE JEXT,NJ,HALF

JEXTN Scalar - total number of nodes in plate.

JF Scalar - number of degrees of freedom. For the PLANAL System, JF = 2.

JINT One level half word array assembled in correspondence with JEXT. If the number i is stored in location j of JEXT, then the number j is stored in location i of JINT.
DEFINE JINT,NJ,HALF

JTID One level double word array to store the alphameric identification of a node.

DEFINE JTID,10,DOUBLE,STEP = 10

JTXYZ Two level full word array to store coordinates of nodes.

DEFINE JTXYZ,10,POINTER,STEP = 10

DEFINE JTXYZ(I),2

where I = external number of node.

JTXYZ(I,1) = x-coordinate,

JTXYZ(I,2) = y-coordinate.

JTYP One level half word array to indicate the type and status of a node. The code is cumulative.

DEFINE JTYP,10,HALF,STEP = 10

Before calling STHTCE, boundary nodes have code of 2; after calling STHTCE, such nodes have code of 4. After calling HSTORE, nodes with prescribed displacements or stress functions have code of 2 (also updated in HPSSLS for nodes with implied FUNCTION boundary condition).

KDIAG Two level double word array to contain the diagonal submatrices of the symmetrical global coefficient matrix.

DEFINE KDIAG, NSOL, JF*JF, DOUBLE

KODOUT One level half word array to contain code for selective output.

KODOUT(1) = cumulative code for element in stretching,

KODOUT(2) = cumulative code for element in bending,

KODOUT(3) = cumulative code for node in stretching,

KODOUT(4) = cumulative code for node in bending,

KODOUT(5) = 1234, if output at elements is required,

KODOUT(6) = 1234, if output at nodes is required.

Defined in STHOUT.

KOFDG Three level double word array - to contain the non-zero lower half off-diagonal submatrices of the global stiffness matrix.

DEFINE KOFDG,NJ,5,POINTER,STEP = 5

DEFINE KOFDG(I,J),JF*JF,DOUBLE

where I = internal number of a node,

J = order of non-zero submatrix, whose position in the matrix is indicated by array IOFDG.

KPBSLN Two level half word array to contain information during construction of particular solution functions in bending.

DEFINE KPBSLN,NJ,2,HALF

KPBSLN(I,1) = condition in x-direction of K_y,

DPBSLN(I,2) = condition in y-direction of K_x.

0: K_x or K_y not computed yet,

1: K_x or K_y computed.

Defined in STHBPS.

Also defined in STH1FS for another temporary use.

KPPRI Two level double word array to contain the right-hand members of the system equations.

DEFINE KPPRI,NJ,JF,DOUBLE

LEXTN Scalar - total number of loadings.

LODTYP Scalar - indicator for type of loading specified.

1: load intensity specified at each node,

2: load force specified at each node,

3: uniform load intensity specified,

4: uniform load force specified.

Set in STHLOD.

NBEL Scalar - number of active elements.

NBXTEL Scalar - total number of elements.

NDPROP Two level full word array to contain properties of plate at nodes.

DEFINE NDPROP,NJ,6

NDPROP(I,1) = thickness,

NDPROP(I,2) = Young's modulus in x-direction,

NDPROP(I,3) = Young's modulus in y-direction,

NDPROP(I,4) = Poisson's ratio in x-direction,

NDPROP(I,5) = Poisson's ratio in y-direction,

NDPROP(I,6) = shear modulus.

Defined and constructed in STHGEN only when IUNIPR = 0.

NJ Scalar - number of active nodes.

NLDS Scalar - number of active loading conditions.

NLDSI Scalar - number of independent loading conditions.

NODISP Two level full word array to store the computed nodal values of displacements or stress functions.

DEFINE NODISP,JEXTN,2

NODISP(I,1) = U,

NODISP(I,2) = V.

where I = external number of a node.

NSOL Scalar - number of nodes at which displacements or stress functions are not fully prescribed.

NSYM Scalar - indicator for symmetry of global coefficient matrix.

1: symmetric,

>1: non-symmetric.

Set in STHBOU,STHINI.

PBNTEM Two level full word array for temporary storage in constructing particular solution functions at the nodes.

DEFINE PBNTEM,NJ,8

PBNTEM(I,1) = effective distance in x-direction,

PBNTEM(I,2) = effective distance in y-direction,

PBNTEM(I,3) = $K_{y,xx}$,

PBNTEM(I,4) = $K_{x,yy}$,

PBNTEM(I,5) = $K_{y,x}$,

PBNTEM(I,6) = $K_{x,y}$,

PBNTEM(I,7) = K_y,

PBNTEM(I,8) = K_x.

Defined in STHBPS.

PBSOLE Two level full word array to store the element centered values of the particular solution functions K_x and K_y.

DEFINE PBSOLE,NBXTEL,POINTER

DEFINE PBSOLE(I),2

where I = external number of the element.

PBSOLE(I,1) = K_x,

PBSOLE(I,2) = K_y,

PBSOLE(I,3) = $K_{x,x}$,

PBSOLE(I,4) = $K_{y,y}$.

PBSOLN Two level full word array to store nodal values of the particular solution functions K_x and K_y. Nodes ordered according to external numbers.

DEFINE PBSOLN,JEXTN,POINTER

DEFINE PBSOLN(I),2

where I = external number of the node.

PBSOLN(I,1) = K_x,

PBSOLN(I,2) = K_y,

PBSOLN(I,3) = $K_{x,x}$,

PBSOLN(I,4) = $K_{y,y}$,

Defined in STHPAR,STH1FS.

RFORND Two level full word array to store load forces at nodes.
External numbers used for nodes.
DEFINE RFORND,JEXTN,3
RFORND(I,1) = force in x-direction,
RFORND(I,2) = force in y-direction,
RFORND(I,3) = force in z-direction.
Defined in STHLOD.

RINTND Two level full word array to store load intensities at nodes.
External numbers used for nodes.
DEFINE RINTND,JEXTN,3
RFORND(I,1) = intensity in x-direction,
RFORND(I,2) = intensity in y-direction,
RFORND(I,3) = intensity in z-direction.
Defined in STHLOD.

TEMOUT One level half word array to store the parameters controlling intermediate output of arrays.
DEFINE TEMOUT,10,HALF,STEP = 5
TEMOUT(n) = K_n,for n = 1,...,10.

$K_1,\ldots,K_{10}$ are defined under output and analysis commands in Section 5.6.

VALUEE Two level full word array to store the output quantities at the elements according to Table A.2.
DEFINE VALUEE,NBXTEL,15

VALUEN Two level full word array to store the output quantities at the nodes according to Table A.2.

DEFINE VALUEN,NJ,17

Table A.2. Data Structure of VALUEE and VALUEN.

VALUEE				VALUEN			
Data level	(1)	(2)	(3)	Data level	(1)	(2)	(3)
1	ε_x	M_y^*	M_y^*	1	ε_x	M_y^*	M_y^*
2	ε_y	M_x^*	M_x^*	2	$v_{,x}$	$V_{,x}$	$V_{,x}$
3	ε_{xy}	$-M_{xy}$	$-M_{xy}$	3	$u_{,y}$	$U_{,y}$	$U_{,y}$
4	σ_x	χ_x	χ_x	4	ε_y	M_x^*	M_x^*
5	σ_y	χ_y	χ_y	5	ε_{xy}	$-M_{xy}$	$-M_{xy}$
6	σ_{xy}	χ_{xy}	χ_{xy}	6	σ_x	χ_x	χ_x
7	ε_1	M_1	M_1	7	σ_y	χ_y	χ_y
8	ε_2	M_2	M_2	8	σ_{xy}	χ_{xy}	χ_{xy}
9	θ_1	θ_1	θ_1	9	ε_1	M_1	M_1
10	σ_1	χ_1	χ_1	10	ε_2	M_2	M_2
11	σ_2	χ_2	χ_2	11	θ_1	θ_1	θ_1
12			M_x^p	12	σ_1	χ_1	χ_1
13			M_y^p	13	σ_2	χ_2	χ_2
14		$M_x = M_x^*$	M_x	14			M_x^p
15		$M_y = M_y^*$	M_y	15			M_y^p
				16		$M_x = M_x^*$	M_x
				17		$M_y = M_y^*$	M_y

(1) Stretching problem.
(2) Bending problem when particular solution is not involved.
(3) Bending problem when particular solution is involved.

APPENDIX E

LOAD MODULES DOCUMENTATION

The load modules of PLANAL are documented by listing the input to SETGEN. SETGEN is one of the steps in computer operation when load modules are formed from source and object decks of subprograms (subroutines). Input to SETGEN provides all the required information for module formation. The input is to be punched in the first six card columns, left-justified, in the order as shown under the heading "input to SETGEN." Remarks (not to be punched) are added here to describe the function of each input card. The remarks indicate the name and structure of the load module, the entry and non-entry points, and the subprograms with COMMON and those without. Wherever numbers are used in the input, a format of I2 is required. The functions of the load modules are also stated.

```
STHBCM

FUNCTION
     THIS MODULE INITIATES THE MANAGEMENT OF BOUNDARY COND-
ITIONS.  IT SETS UP THE RIGHT-HAND SIDE OF THE GOVERNING
SYSTEM OF SIMULTANEOUS EQUATIONS CONTRIBUTED BY EXTERNAL
LOADS.

INPUT TO
SETGEN          REMARKS

NODECK
PLANAL          NAME OF SUBSYSTEM.
SIMPLE          STRUCTURE OF LOAD MODULE.
STHBCM          NAME OF LOAD MODULE.
```

1	NO. OF SUBPROGRAMS WITH COMMON, TO BE ENTRIES.
STHBCM	NAME OF SUCH SUBPROGRAM.
6	NO. OF SUBPROG. WITH COMMON, TO BE NON-ENTRIES.
HATAN	NAME OF SUCH SUBPROGRAM.
HPHI	NAME OF SUCH SUBPROGRAM.
NEXNOD	NAME OF SUCH SUBPROGRAM.
STHAVG	NAME OF SUCH SUBPROGRAM.
STHBLV	NAME OF SUCH SUBPROGRAM.
STHSLV	NAME OF SUCH SUBPROGRAM.
0	NO. OF SUBPROGRAMS W/O COMMON, TO BE ENTRIES.
2	NO. OF SUBPROG. W/O COMMON, TO BE NON-ENTRIES.
AND	NAME OF SUCH SUBPROGRAM.
LCDBLE	NAME OF SUCH SUBPROGRAM.
**EOF	

* * * * *

STHBEN

FUNCTION

THIS MODULE CONTAINS THE DICTIONARY SUBPROGRAM IN PLATE BENDING (STHBEN), WHICH LEADS TO THE PROPER SUBPROGRAM IN AN OVERLAY STRUCTURE THAT PROCESSES ONE OF THE BOUNDARY CONDITIONS.

INPUT TO SETGEN	REMARKS
NODECK	
PLANAL	NAME OF SUBSYSTEM.
SIMPLE	STRUCTURE OF LOAD MODULE.
STHBEN	NAME OF LOAD MODULE.
1	NO. OF SUBPROGRAMS WITH COMMON, TO BE ENTRIES.
STHBEN	NAME OF SUCH SUBPROGRAM.
6	NO. OF SUBPROG. WITH COMMON, TO BE NON-ENTRIES.
HATAN	NAME OF SUCH SUBPROGRAM.
HPHI	NAME OF SUCH SUBPROGRAM.
NEXNOD	NAME OF SUCH SUBPROGRAM.
STHBDI	NAME OF SUCH SUBPROGRAM.
STHFFB	NAME OF SUCH SUBPROGRAM.
STHSLB	NAME OF SUCH SUBPROGRAM.
0	NO. OF SUBPROGRAMS W/O COMMON, TO BE ENTRIES.
1	NO. OF SUBPROG. W/O COMMON, TO BE NON-ENTRIES.
AND	NAME OF SUCH SUBPROGRAM.
**EOF	

* * * * *

STHBKS

FUNCTION

THIS MODULE IS CALLED AFTER THE UNKNOWNS OF THE GOVERNING SYSTEM OF SIMULTANEOUS EQUATIONS HAVE BEEN SOLVED. STRAINS AND STRESSES, OR STRESS COUPLES AND CURVATURES, ARE THEN COMPUTED BY BACK-SUBSTITUTION. OUTPUT SUBPROGRAMS ARE ALSO INCLUDED.

INPUT TO SETGEN	REMARKS
NODECK	
PLANAL	NAME OF SUBSYSTEM.
SIMPLE	STRUCTURE OF LOAD MODULE.
STHBKS	NAME OF LOAD MODULE.
1	NO. OF SUBPROGRAMS WITH COMMON, TO BE ENTRIES.
STHBKS	NAME OF SUCH SUBPROGRAM.
6	NO. OF SUBPROG. WITH COMMON, TO BE NON-ENTRIES.
DE1BUG	NAME OF SUCH SUBPROGRAM.
HCODE	NAME OF SUCH SUBPROGRAM.
HTRANS	NAME OF SUCH SUBPROGRAM.
H1OUT	NAME OF SUCH SUBPROGRAM.
STHDER	NAME OF SUCH SUBPROGRAM.
TPFORM	NAME OF SUCH SUBPROGRAM.
0	NO. OF SUBPROGRAMS W/O COMMON, TO BE ENTRIES.
1	NO. OF SUBPROG. W/O COMMON, TO BE NON-ENTRIES.
AND	NAME OF SUCH SUBPROGRAM.
**EOF	

*   *   *   *   *

STHB1S

FUNCTION

THIS MODULE IS A CONTINUATION OF STHBKS.

INPUT TO SETGEN	REMARKS
NODECK	
PLANAL	NAME OF SUBSYSTEM.
SIMPLE	STRUCTURE OF LOAD MODULE.
STHB1S	NAME OF LOAD MODULE.
1	NO. OF SUBPROGRAMS WITH COMMON, TO BE ENTRIES.
STHB1S	NAME OF SUCH SUBPROGRAM.
9	NO. OF SUBPROG. WITH COMMON, TO BE NON-ENTRIES.
DE2BUG	NAME OF SUCH SUBPROGRAM.
DE3BUG	NAME OF SUCH SUBPROGRAM.
HANGLE	NAME OF SUCH SUBPROGRAM.
HATAN	NAME OF SUCH SUBPROGRAM.

```
HELEMT         NAME OF SUCH SUBPROGRAM.
HNSTAN         NAME OF SUCH SUBPROGRAM.
HNSTES         NAME OF SUCH SUBPROGRAM.
H2OUT          NAME OF SUCH SUBPROGRAM.
H3OUT          NAME OF SUCH SUBPROGRAM.
 0             NO. OF SUBPROGRAMS W/O COMMON, TO BE ENTRIES.
 0             NO. OF SUBPROG. W/O COMMON, TO BE NON-ENTRIES.
**EOF
```

* * * * *

STHGEN

FUNCTION

THIS MODULE GENERATES THE LOCAL COEFFICIENT MATRICES (STIFFNESS/FLEXIBILITY MATRICES) BEFORE THE MATRIX FOR THE ENTIRE SYSTEM IS ASSEMBLED.

```
INPUT TO
SETGEN         REMARKS

NODECK
PLANAL         NAME OF SUBSYSTEM.
SIMPLE         STRUCTURE OF LOAD MODULE.
STHGEN         NAME OF LOAD MODULE.
 2             NO. OF SUBPROGRAMS WITH COMMON, TO BE ENTRIES.
STHGEN         NAME OF SUCH SUBPROGRAM.
STHSEP         NAME OF SUCH SUBPROGRAM.
 2             NO. OF SUBPROG. WITH COMMON, TO BE NON-ENTRIES.
HDUAL          NAME OF SUCH SUBPROGRAM.
STHESM         NAME OF SUCH SUBPROGRAM.
 0             NO. OF SUBPROGRAMS W/O COMMON, TO BE ENTRIES.
 0             NO. OF SUBPROG. W/O COMMON, TO BE NON-ENTRIES.
**EOF
```

* * * * *

STHINI

FUNCTION

THIS MODULE CONTAINS THE SUBPROGRAMS THAT INITIALIZE THE PLANAL SYSTEM.

```
INPUT TO
SETGEN         REMARKS

NODECK
PLANAL         NAME OF SUBSYSTEM.
SIMPLE         STRUCTURE OF LOAD MODULE.
```

```
STHINI          NAME OF LOAD MODULE.
 6              NO. OF SUBPROGRAMS WITH COMMON, TO BE ENTRIES.
STHINI          NAME OF SUCH SUBPROGRAM.
STHBOU          NAME OF SUCH SUBPROGRAM.
STHLOD          NAME OF SUCH SUBPROGRAM.
STHOUT          NAME OF SUCH SUBPROGRAM.
STHTCE          NAME OF SUCH SUBPROGRAM.
STHTRA          NAME OF SUCH SUBPROGRAM.
 2              NO. OF SUBPROG. WITH COMMON, TO BE NON-ENTRIES.
GETNOS          NAME OF SUCH SUBPROGRAM.
HSTORE          NAME OF SUCH SUBPROGRAM.
 0              NO. OF SUBPROGRAMS W/O COMMON, TO BE ENTRIES.
 3              NO. OF SUBPROG. W/O COMMON, TO BE NON-ENTRIES.
AND             NAME OF SUCH SUBPROGRAM.
LCDBLE          NAME OF SUCH SUBPROGRAM.
SETCLK          NAME OF SUCH SUBPROGRAM.
**EOF
```

*   *   *   *   *

STHMAI

FUNCTION

THIS MODULE CONTAINS THE SUBPROGRAM THAT IS THE 'MAIN' PROGRAM OF THE SYSTEM. EXECUTION OF OTHER LOAD MODULES IS CONTROLLED BY THE 'MAIN' PROGRAM.

```
INPUT TO
SETGEN          REMARKS

NODECK
PLANAL          NAME OF SUBSYSTEM.
SIMPLE          STRUCTURE OF LOAD MODULE.
STHMAI          NAME OF LOAD MODULE.
 1              NO. OF SUBPROGRAMS WITH COMMON, TO BE ENTRIES.
STHMAI          NAME OF SUCH SUBPROGRAM.
 8              NO. OF SUBPROG. WITH COMMON, TO BE NON-ENTRIES.
HATAN           NAME OF SUCH SUBPROGRAM.
HCLOCK          NAME OF SUCH SUBPROGRAM.
HPSSLS          NAME OF SUCH SUBPROGRAM.
NEXNOD          NAME OF SUCH SUBPROGRAM.
STHCHK          NAME OF SUCH SUBPROGRAM.
STHGRI          NAME OF SUCH SUBPROGRAM.
STHRBD          NAME OF SUCH SUBPROGRAM.
STHTMO          NAME OF SUCH SUBPROGRAM.
 0              NO. OF SUBPROGRAMS W/O COMMON, TO BE ENTRIES.
 2              NO. OF SUBPROG. W/O COMMON, TO BE NON-ENTRIES.
AND             NAME OF SUCH SUBPROGRAM.
SETCLK          NAME OF SUCH SUBPROGRAM.
**EOF
```

STHNAS

FUNCTION

THIS MODULE ASSEMBLES THE COEFFICIENT MATRIX (STIFFNESS/FLEXIBILITY MATRIX) OF THE GOVERNING SYSTEM OF SIMULTANEOUS EQUATIONS WHEN THE MATRIX IS NON-SYMMETRIC.

INPUT TO SETGEN	REMARKS
NODECK	
PLANAL	NAME OF SUBSYSTEM.
SIMPLE	STRUCTURE OF LOAD MODULE.
STHNAS	NAME OF LOAD MODULE.
1	NO. OF SUBPROGRAMS WITH COMMON, TO BE ENTRIES.
STHNAS	NAME OF SUCH SUBPROGRAM.
2	NO. OF SUBPROG. WITH COMMON, TO BE NON-ENTRIES.
HNSASS	NAME OF SUCH SUBPROGRAM.
HPOSIT	NAME OF SUCH SUBPROGRAM.
0	NO. OF SUBPROGRAMS W/O COMMON, TO BE ENTRIES.
2	NO. OF SUBPROG. W/O COMMON, TO BE NON-ENTRIES.
BITON	NAME OF SUCH SUBPROGRAM.
SETCLK	NAME OF SUCH SUBPROGRAM.
**EOF	

* * * * *

STHNSL

FUNCTION

THIS MODULE SOLVES THE SYSTEM OF SIMULTANEOUS EQUATIONS IN THE NON-SYMMETRIC CASE.

INPUT TO SETGEN	REMARKS
NODECK	
PLANAL	NAME OF SUBSYSTEM.
SIMPLE	STRUCTURE OF LOAD MODULE.
STHNSL	NAME OF LOAD MODULE.
1	NO. OF SUBPROGRAMS WITH COMMON, TO BE ENTRIES.
STHNSL	NAME OF SUCH SUBPROGRAM.
9	NO. OF SUBPROG. WITH COMMON, TO BE NON-ENTRIES.
HDEBUG	NAME OF SUCH SUBPROGRAM.
HGAUBK	NAME OF SUCH SUBPROGRAM.
HGAUSS	NAME OF SUCH SUBPROGRAM.
HINTER	NAME OF SUCH SUBPROGRAM.
HNSLAS	NAME OF SUCH SUBPROGRAM.
HNSLSA	NAME OF SUCH SUBPROGRAM.
MATMUL	NAME OF SUCH SUBPROGRAM.
MATSUB	NAME OF SUCH SUBPROGRAM.

```
 0            NO. OF SUBPROGRAMS W/O COMMON, TO BE ENTRIES.
 1            NO. OF SUBPROG. W/O COMMON, TO BE NON-ENTRIES.
SETCLK        NAME OF SUCH SUBPROGRAM.
**EOF
```

* * * * *

STHPAR

FUNCTION

THIS MODULE PROCESSES THE INPUT OR CONSTRUCTION OF A PARTICULAR SOLUTION IN THE BENDING PROBLEM.

```
INPUT TO
SETGEN        REMARKS

NODECK
PLANAL        NAME OF SUBSYSTEM.
SIMPLE        STRUCTURE OF LOAD MODULE.
STHPAR        NAME OF LOAD MODULE.
 2            NO. OF SUBPROGRAMS WITH COMMON, TO BE ENTRIES.
STHPAR        NAME OF SUCH SUBPROGRAM.
STHBPS        NAME OF SUCH SUBPROGRAM.
 4            NO. OF SUBPROG. WITH COMMON, TO BE NON-ENTRIES.
HINTEG        NAME OF SUCH SUBPROGRAM.
INTGRT        NAME OF SUCH SUBPROGRAM.
INTPOL        NAME OF SUCH SUBPROGRAM.
NEXNOD        NAME OF SUCH SUBPROGRAM.
 0            NO. OF SUBPROGRAMS W/O COMMON, TO BE ENTRIES.
 2            NO. OF SUBPROG. W/O COMMON, TO BE NON-ENIRIES.
AND           NAME OF SUCH SUBPROGRAM.
LCDBLE        NAME OF SUCH SUBPROGRAM.
**EOF
```

* * * * *

STHP1R

FUNCTION

THIS MODULE IS A CONTINUATION OF STHPAR.

```
INPUT TO
SETGEN        REMARKS

NODECK
PLANAL        NAME OF SUBSYSTEM.
SIMPLE        STRUCTURE OF LOAD MODULE.
STHP1R        NAME OF LOAD MODULE.
 1            NO. OF SUBPROGRAMS WITH COMMON, TO BE ENTRIES.
```

STHP1R	NAME OF SUCH SUBPROGRAM.
5	NO. OF SUBPROG. WITH COMMON, TO BE NON-ENTRIES.
HATAN	NAME OF SUCH SUBPROGRAM.
HDISLD	NAME OF SUCH SUBPROGRAM.
HDIST	NAME OF SUCH SUBPROGRAM.
HTHETA	NAME OF SUCH SUBPROGRAM.
NEXNOD	NAME OF SUCH SUBPROGRAM.
0	NO. OF SUBPROGRAMS W/O COMMON, TO BE ENTRIES.
1	NO. OF SUBPROG. W/O COMMON, TO BE NON-ENTRIES.
AND	NAME OF SUCH SUBPROGRAM.
**EOF	

*   *   *   *   *

STHSAS

FUNCTION

THIS MODULE TRANSFERS SUBMATRICES OF THE COEFFICIENT MATRIX OF THE GOVERNING EQUATIONS TO LOCAL ARRAYS, AND VICE VERSA, FOR EASE OF MODIFICATION.

INPUT TO SETGEN	REMARKS
NODECK	
PLANAL	NAME OF SUBSYSTEM.
SIMPLE	STRUCTURE OF LOAD MODULE.
STHSAS	NAME OF LOAD MODULE.
2	NO. OF SUBPROGRAMS WITH COMMON, TO BE ENTRIES.
STHSAS	NAME OF SUCH SUBPROGRAM.
STHSSA	NAME OF SUCH SUBPROGRAM.
0	NO. OF SUBPROG. WITH COMMON, TO BE NON-ENTRIES.
0	NO. OF SUBPROGRAMS W/O COMMON, TO BE ENTRIES.
1	NO. OF SUBPROG. W/O COMMON, TO BE NON-ENTRIES.
BITON	NAME OF SUCH SUBPROGRAM.
**EOF	

*   *   *   *   *

STHSTR

FUNCTION

THIS MODULE CONTAINS THE DICTIONARY SUBPROGRAM IN PLATE STRETCHING (STHSTR), WHICH LEADS TO THE PROPER SUBPROGRAM IN AN OVERLAY STRUCTURE THAT PROCESSES ONE OF THE BOUNDARY CONDITIONS.

INPUT TO SETGEN	REMARKS

```
NODECK
PLANAL          NAME OF SUBSYSTEM.
OVERLAY         STRUCTURE OF LOAD MODULE.
STHSTR          NAME OF LOAD MODULE.
 1              NO. OF SUBPROGRAMS WITH COMMON, TO BE ENTRIES.
STHSTR          NAME OF SUCH SUBPROGRAM.
 0              NO. OF SUBPROG. WITH COMMON, TO BE NON-ENTRIES.
 0              NO. OF SUBPROGRAMS W/O COMMON, TO BE ENTRIES.
 2              NO. OF SUBPROG. W/O COMMON, TO BE NON-ENTRIES.
AND             NAME OF SUCH SUBPROGRAM.
BITON           NAME OF SUCH SUBPROGRAM.
 1              NO. OF REGIONS IN OVERLAY STRUCTURE.
REGION 1        INDICATES START OF REGION.
OVERLAY BETA    INDICATES START OF SEGMENT.
SDISPL          NAME OF THE ENTRY TO SEGMENT.
 1              NO. OF SUBPROGRAMS WITH COMMON.
SDISPL          NAME OF SUCH SUBPROGRAM.
 0              NO. OF SUBPROGRAMS W/O COMMON.
OVERLAY BETA    INDICATES START OF SEGMENT.
SEDGEB          NAME OF THE ENTRY TO SEGMENT.
 2              NO. OF SUBPROGRAMS WITH COMMON.
SEDGEB          NAME OF SUCH SUBPROGRAM.
STIFED          NAME OF SUCH SUBPROGRAM.
 0              NO. OF SUBPROGRAMS W/O COMMON.
OVERLAY BETA    INDICATES START OF SEGMENT.
SELAST          NAME OF THE ENTRY TO SEGMENT.
 1              NO. OF SUBPROGRAMS WITH COMMON.
SELAST          NAME OF SUCH SUBPROGRAM.
 0              NO. OF SUBPROGRAMS W/O COMMON.
OVERLAY BETA    INDICATES START OF SEGMENT.
SMIXED          NAME OF THE ENTRY TO SEGMENT.
 2              NO. OF SUBPROGRAMS WITH COMMON.
SMIXED          NAME OF SUCH SUBPROGRAM.
ENDMIX          NAME OF SUCH SUBPROGRAM.
 0              NO. OF SUBPROGRAMS W/O COMMON.
OVERLAY BETA    INDICATES START OF SEGMENT.
SSTRES          NAME OF THE ENTRY TO SEGMENT.
 1              NO. OF SUBPROGRAMS WITH COMMON.
SSTRES          NAME OF SUCH SUBPROGRAM.
 0              NO. OF SUBPROGRAMS W/O COMMON.
END OF OVERLAY INDICATES END OF OVERLAY STRUCTURE.
**EOF
```

*   *   *   *   *

STHSVR

FUNCTION

THIS MODULE SOLVES THE SYSTEM OF SIMULTANEOUS EQUATIONS IN THE SYMMETRIC CASE.

INPUT TO SETGFN	REMARKS
NODECK	
PLANAL	NAME OF SUBSYSTEM.
SIMPLE	STRUCTURE OF LOAD MODULE.
STHSVR	NAME OF LOAD MODULE.
1	NO. OF SUBPROGRAMS WITH COMMON, TO BE ENTRIES.
STHSVR	NAME OF SUCH SUBPROGRAM.
8	NO. OF SUBPROG. WITH COMMON, TO BE NON-ENTRIES.
STADRS	NAME OF SUCH SUBPROGRAM.
STAD1S	NAME OF SUCH SUBPROGRAM.
STDCPY	NAME OF SUCH SUBPROGRAM.
STDMAD	NAME OF SUCH SUBPROGRAM.
STDMMP	NAME OF SUCH SUBPROGRAM.
STDMTR	NAME OF SUCH SUBPROGRAM.
STIVDP	NAME OF SUCH SUBPROGRAM.
SVRBUG	NAME OF SUCH SUBPROGRAM.
0	NO. OF SUBPROGRAMS W/O COMMON, TO BE ENTRIES.
2	NO. OF SUBPROG. W/O COMMON, TO BE NON-ENTRIES.
BITON	NAME OF SUCH SUBPROGRAM.
SETCLK	NAME OF SUCH SUBPROGRAM.
**EOF	

*   *   *   *   *

STHS1R

FUNCTION

THIS MODULE IS A CONTINUATION OF LOAD MODULE STHSTR.

INPUT TO SETGEN	REMARKS
NODECK	
PLANAL	NAME OF SUBSYSTEM.
SIMPLE	STRUCTURE OF LOAD MODULE.
STHS1R	NAME OF LOAD MODULE.
1	NO. OF SUBPROGRAMS WITH COMMON, TO BE ENTRIES.
STHS1R	NAME OF SUCH SUBPROGRAM.
9	NO. OF SUBPROG. WITH COMMON, TO BE NON-ENTRIES.
ENDSTN	NAME OF SUCH SUBPROGRAM.
HATAN	NAME OF SUCH SUBPROGRAM.
HCHECK	NAME OF SUCH SUBPROGRAM.
HINITL	NAME OF SUCH SUBPROGRAM.
HMODIF	NAME OF SUCH SUBPROGRAM.
HPHI	NAME OF SUCH SUBPROGRAM.
HROTAT	NAME OF SUCH SUBPROGRAM.
NEXNOD	NAME OF SUCH SUBPROGRAM.
STRAIN	NAME OF SUCH SUBPROGRAM.

```
 0              NO. OF SUBPROGRAMS W/O COMMON, TO BE ENTRIES.
 2              NO. OF SUBPROG. W/O COMMON, TO BE NON-ENTRIEO.
AND             NAME OF SUCH SUBPROGRAM.
BITON           NAME OF SUCH SUBPROGRAM.
**EOF
```

* * * * *

STH1FS

FUNCTION

THIS MODULE PROCESSES THE CONSTRUCTION OF A PARTICULAR SOLUTION IN THE BENDING PROBLEM BY SUMMATION OF A FOURIER SERIES.

```
INPUT TO
SETGEN          REMARKS

NODECK
PLANAL          NAME OF SUBSYSTEM.
SIMPLE          STRUCTURE OF LOAD MODULE.
STH1FS          NAME OF LOAD MODULE.
 1              NO. OF SUBPROGRAMS WITH COMMON, TO BE ENTRIES.
STH1FS          NAME OF SUCH SUBPROGRAM.
 7              NO. OF SUBPROG. WITH COMMON, TO BE NON-ENTRIES.
HSIGN           NAME OF SUCH SUBPROGRAM.
NEXNOD          NAME OF SUCH SUBPROGRAM.
STHCBC          NAME OF SUCH SUBPROGRAM.
STHCON          NAME OF SUCH SUBPROGRAM.
STH2FS          NAME OF SUCH SUBPROGRAM.
STH3FS          NAME OF SUCH SUBPROGRAM.
STH4FS          NAME OF SUCH SUBPROGRAM.
 0              NO. OF SUBPROGRAMS W/O COMMON, TO BE ENTRIES.
 1              NO. OF SUBPROG. W/O COMMON, TO BE NON-ENTRIEO.
AND             NAME OF SUCH SUBPROGRAM.
**EOF
```

APPENDIX F

PROGRAM DOCUMENTATION

In this appendix is listed brief documentation for the subroutines used in the PLANAL System. The names of a subroutine and the load module in which it resides are listed with description. Internal logic, linkage and calling sequence are indicated wherever appropriate. A missing item means that the item needs no description or is missing.

```
Name:          AND.
Load Module:   Assembly language program used in STHBCM, STHBEN, STHBKS,
               STHINI, STHMAI, STHPAR, STHP1R, STHSTR, STHS1R, STH1FS.
Description:   Program returns the 32 bits logical product of its two
               arguments in general register 0.
Length:        20 bytes.

Name:          BITON.
Load Module:   Assembly language program used in load modules STHNAS,
               STHSAS, STHSTR, STHSVR, STHS1R.
Description    BITON turns on bit N of WORD where N is counted from 1 to
               32 left to right.  BITOFF turns of bit N of WORD.  IFBIT
               returns 0 if bit N is off and 1 if it is on.
Logic:         Forms of calling sequence:
               (1) CALL BITON(WORD,N)
               (2) CALL BITOFF(WORD,N)
               (3) J = IFBIT(WORD,N)
               where N    = bit to be tested or changed in WORD.
                     WORD = full word in which a bit will be turned on or
                            off.
                     J    = value of bit N in WORD.
Length:        136 bytes.
```

Name:	DE1BUG.
Load Module:	STHBKS.
Description:	This subroutine prints out grid lines for differentiation when requested.
Length:	1084 bytes.
Called by:	STHBKS.

Name:	DE2BUG.
Load Module:	STHB1S.
Description:	Program to print the moments of the homogeneous, particular and total problems at the nodes.
Lenth:	864 bytes.
Called by:	STHB1S.

Name:	DE3BUG.
Load Module:	STHB1S.
Description:	Program to print the moments of the homogeneous, particular and total problems of the elements.
Length:	904 bytes.
Called by:	STHB1S.

Name:	ENDMIX.
Load Module:	STHSTR.
Description:	Program to treat the special condition at the ends of a mixed boundary portion in stretching.
Length	1912 bytes.
Calls:	AND.
Called by:	SMIXED.

Name:	ENDSTN.
Load Module:	STHS1R.
Description:	Program to process ends of a boundary portion with strain boundary condition.
Length:	1644 bytes.
Calls:	AND.
Called by:	STRAN.

Name:	GETNOS.
Load Module:	STHINI.
Description:	Program to trace the external numbers of nodes along a boundary portion.
Length:	1804 bytes.
Calls:	LCDBLE.
Called by:	STHBOU.
Message:	Error messages issued when boundaries or nodes are not previously defined.

Name: HANGLE.
Load Module: STHB1S.
Description: Program converts an angle given in radians to one in degrees, minutes, and seconds.
Length: 592 bytes.
Called by: H2OUT.

Name: HATAN.
Load Module: STHBCM, STHBEN, STHB1S, STHP1R.
Description: Program computes the arctangent of an angle.
Logic: For a point with given abscissa and ordinate, the arctangent of the angle swept from the positive x-axis to the point is computed. The range of the angle is from zero to 2π.
Length: 640 bytes.
Called by: HNSTES, HELEMT, HNSTAN, STHRBD, HDHI, HTHETA.

Name: HCHECK.
Load Module: STHS1R.
Description: Program to check whether a rigid body displacement has been specified in strain boundary condition.
Length: 1560 bytes.
Calls: AND, NEXNOD
Called by STRAIN.

Name: HCLOCK.
Load Module: STHMAI.
Description: Program for timing the various operations.
Length: 720 bytes.
Calls: SETCLK.
Called by: STHMAI.

Name: HCODE.
Load Module: STHBKS.
Description: Program to compute control parameters for selective output.
Length: 972 bytes.
Calls: AND.
Called by: STHBKS.

Name: HDEBUG.
Load Module: STHNSL.
Description: Program to print out ICUINT when requested.
Length: 1088 bytes.
Called by: HDEB1G, HGAUSS.

Name: HDEB1G.
Load Module: STHNSL.
Description: Program to print FCMAT.
Length: 924 bytes.
Calls: HDEBUG, HDEB2G.
Called by: HGAUSS.

Name: HDEB2G.
Load Module: STHNSL.
Description: Program to print KPPRI.
Length: 652 bytes.
Called by: HDEB1G, HGAUBK.

Name: HDISLD.
Load Module: STHP1R.
Description: Program to distribute load between strips parallel to the axes in computing particular solution functions.
Length: 3144 bytes.
Calls: HDIST.
Called by: STHP1R.

Name: HDIST.
Load Module: STHP1R.
Description: Program to compute distances to boundary.
Length: 608 bytes.
Called by: HDISLD.

Name: HDUAL.
Load Modules: STHGEN.
Description: Program to perform the duality coversion of material properties for the bending problem.
Length: 620 bytes.
Called by: STHESM.

Name: HELEMT.
Load Module: STHB1S.
Description: Program to compute strains (moments), stresses (curvatures), and their principal values at the elements.
Logic: The strain of an element is computed from displacements of the nodes. Stresses are then computed from stress-strain relations.
Length: 4108 bytes.
Calls: HATAN.
Called by: HELEMT.

Name: HGAUBK.
Load Module: STHNSL.
Description: Program to perform back-substitution for unknowns after Gauss reduction of the non-symmetric system equations.
Length: 1608 bytes.
Calls: HNSLSA, MATMUL, MATSUB, HDEB2G.
Called by: STHNSL.

Name: HGAUSS.
Load Module: STHNSL.
Description: Program to perform Gauss reduction of the non-symmetric system equations.
Length: 2212 bytes.
Calls: HDEBUG, HNSLSA, HINTER, MATMUL, HNSLAS, MATSUB, HDEB1G.
Called by: STHNSL.

Name: HINITL.
Load Module: STHS1R.
Description: Program initializes a row of coefficient matrix for modification.
Length: 984 bytes.
Calls: BITOFF, NEXNOD.
Called by: STRAIN.

Name: HINTEG.
Load Module: STHPAR.
Description: Executive program for integration of function along a grid line.
Length: 2996 bytes.
Calls: INTGRT, INTPOL.

Name: HINTER.
Load Module: STHNSL.
Description: Program to perform interchange of hyper-rows when determinant of diagonal submatrix is zero in Gauss reduction of non-symmetric system equations.
Length: 1376 bytes.
Calls: HNSLSA.
Called by: HGAUSS.

Name: HMODIF.
Load Module: STHS1R.
Description: Program to perform modifications in strain boundary condition.
Length: 1404 bytes.
Links: STHSSA.
Called by: STRAIN.

Name: HNSASS.
Load Module: STHNAS.
Description: Program to manage repeated operation of assemblage and updating of record.
Length: 1480 bytes.
Calls: BITON, HPOSIT.
Called by: STHNAS.

Name:	HNSLAS.
Load Module:	STHNSL.
Description:	Program to transfer temporary submatrix to element of non-symmetric global coefficient matrix.
Length:	816 bytes.
Called by:	HGAUSS.

Name:	HNSLSA.
Load Module:	STHNSL.
Description:	Program to transfer element of non-symmetric global coefficient matrix to temporary submatrix.
Length:	708 bytes.
Called by:	HINTER, HGAUBK, HGAUSS.

Name:	HNSTAN.
Load Module:	STHB1S.
Description:	This program computes the strains (moments) and their principal values at the nodes.
Logic:	Strains are computed by derivatives of the displacements.
Length:	2108 bytes.
Calls:	HATAN.
Called by:	STHB1K.

Name:	HNSTES.
Load Module:	STHB1S.
Description:	This program computes the stresses (curvatures) and their principal values at the nodes.
Logic:	Stresses are computed from the strains through stress-strain relations.
Length:	2380 bytes.
Calls:	HATAN.
Called by:	STHB1S.

Name:	HPHI.
Load Module:	STHBCM, STHBEN, STHS1R.
Description:	Program to compute the direction of the outward normal at a node.
Length:	708 bytes.
Calls:	HATAN.
Called by:	STHBLV, STHBDI, STRAIN.

Name:	HPOSIT.
Load Module:	STHNAS.
Description:	Program to compute position of bit in structure of IREL1.
Length:	420 bytes.
Called by:	HNSASS.

Name: HPSSLS.
Load Module: STHMAI.
Description: Program to process simple support and line of symmetry boundary conditions in bending.
Logic: It constructs the BDNORM array around the boundary. Then it sets the boundary values for the two conditions.
Length: 2472 bytes.
Calls: HATAN, AND, NEXNOD.
Called by: STHMAI.

Name: HROTAT.
Load Module: STHS1R.
Description: Program to match the node with prescribed rotation.
Length: 684 bytes.
Called by: STRAIN.

Name: HSIGN.
Load Module: STH1FS.
Description: Function to compute (-1)**M.
Length: 412 bytes.
Called by: STH4FS.

Name: HSTORE.
Load Module: STHINI.
Description: Program stores the boundary values to the negative and positive sides of a boundary node.
Length: 1940 bytes.
Calls: AND.
Called by: STHBOU.

Name: HTHETA.
Load Module: STHP1R.
Description: Program to compute the acute angle between the x-axis and normal to a line segment.
Length: 660 bytes.
Calls: HATAN.
Called by: STHP1R.

Name: HTRANS.
Load Module: STHBKS.
Description: It transfers results after solving the system equations to array NODISP. It also transforms NODISP, wherever applicable, to global axes.
Logic: For nodes with truly mixed boundary condition, displacements have to be transformed to global axes by premultiplying them by the original rotation matrix transposed.
Length: 1528 bytes.
Called by: STHBKS.

Name: H1OUT.
Load Module: STHBKS.
Description: Program to print nodal displacements (stress functions) when requested
Output: Nodal displacements or stress functions.
Length: 856 bytes.
Calls: AND.
Called by: STHBKS.

Name: H2OUT.
Load Module: STHB1S.
Description: Program to print the strains (moments), stresses (curvatures), and/or their principal values at the nodes.
Length: 3688 bytes.
Calls: HANGLE.
Called by: STHB1S.

Name: H3OUT.
Load Module: STHB1S.
Description: Program to print the strains (moments), stresses (curvatures), and/or their principal values of the elements.
Length: 3548 bytes.
Calls: HANGLE.
Called by: STHB1S.

Name: INTGRT.
Load odule: STHPAR.
Description: Program to perform numerical integration for PBNTEM.
Length: 876 bytes.
Called by: HINTEG.

Name: INTPOL.
Load Module: STHPAR.
Description: Program to perform interpolation of ordinates.
Length: 1072 bytes.
Called by: HINTEG.

Name: LCDBLE.
Load Module: Utility program used in load modules STHBCM, STHINI, STHPAR.
Description: This function performs a logical comparison of two double precision arguments and returns as a code:
0 if arguments are logically equivalent.
1 if first argument is logically less than the second.
2 if first argument is logically greater than the second.
Length: 60 bytes.

Name: MATMUL.
Load Module: STHNSL.
Description: Program to perform matrix multiplication.
Length: 508 bytes.
Called by: HGAUBK, HGAUSS.

Name: MATSUB.
Load Module: STHNSL.
Description: Program to perform matrix subtraction.
Length: 440 bytes.
Called by: HGAUBK, HGAUSS.

Name: NEXNOD.
Load Module: STHBCM, STHBEN, STHS1R, STHPAR, STHP1R, STH1FS.
Description: Function to compute the next node along a boundary position in either a positive or negative s-direction.
Length: 480 bytes.
Called by: STHBLV, STHBDI, STHFFB, STHSLB, STRAIN, STHBPS, STHP1R, STHCON.

Name: SDISPL.
Load Module: STHSTR.
Description: Program to process displacement boundary condition in stretching.
Length: 3780 bytes.
Calls: AND, BITON.
Called by: STHSTR.
Message: "Inconsistent displacement values at node."

Name: SEDGEB.
Load Module: STHSTR.
Description: Program to process edge beam boundary in stretching.
Length: 4100 bytes.
Links: STHSAS, STHSSA.
Calls: STIFED, AND.
Called by: STHSTR.

Name: SELAST.
Load Module: STHSTR.
Description: Program to process elastic boundary condition in stretching.
Length: 4396 bytes.
Links: STHSAS, STHSSA.
Calls: AND.
Called by: STHSTR.
Messages: "Inconsistently prescribed support displacements at node."

Name: SETCLK.
Load Module: Assembly language program used in load modules STHINI, STHMAI, STHNAS, STHNSL, STHSVR.
Description: The entry point SETCLK initializes timing calls. The entry point GETCLK returns the elapsed time, in hundredths of a second, since the last call to SETCLK.
Length: 30 bytes.

Name: SMIXED.
Load Module: STHSTR.
Description: Program to process mixed boundary condition in stretching.
Length: 6208 bytes.
Links: STHSAS, STHSSA.
Calls: AND, ENDMIX.
Called by: STHSTR.
Messages: Error messages will be issued when displacement components and angles at nodes are not specified consistently.

Name: SSTRES.
Load Module: STHSTR.
Description: Program to process stress boundary condition in stretching.
Length: 2436 bytes.
Calls: AND.
Called by: STHSTR.

Name: STHASS.
Load Module: STHASS.
Description: Executive program for assembly of the symmetric global coefficient matrix.
Length: 3834 bytes.
Linked by: STHMAI.
Calls: SETCLK, STORSU, STADRS, STDCPY, STDMAD.

Name: STHAVG.
Load Module: STHBCM.
Description: Program to compute average angle in MIXED BENDING boundary condition.
Logic: In bending problems, MIXED BENDING boundary is constructed internally for simple support and line of symmetry boundary conditions. It checks if there is any disagreement in direction for stress function at a node. The average direction is taken.
Length: 1068 bytes.
Calls: AND.
Called by: STHBCM.

Name: STHBCM.
Load Module: STHBCM.
Description: Executive program in modification for boundary conditions and computation of load vector in system equations.
Logic: It determines whether current problem is stretching or bending. Then it calls appropriate program to construct load vector in system equations. It checks for any node with unspecified boundary condition. It then loops on all the boundaries by calling a dictionary program. For bending problem, some boundary conditions are processed under dual routines in stretching.
Length: 3748 bytes.
Links: STHBEN, STHSTR.
Linked by: STHMAI.

Calls: AND, STHBLV, STHSLV, STHAVG.
Messages: Error messages are printed when:
1. boundary conditions for some portion of a boundary are not specified,
2. instability due to boundary conditions is detected (less than three displacement components specified in stretching, or less than three stress function components specified in bending).

Name: STHBDI.
Load Module: STHBEN.
Description: This program processes the displacement boundary condition in bending.
Length: 2172 bytes.
Calls: AND, HPHI, NEXNOD.
Called by: STHBEN.

Name: STHBEN.
Load Module: STHBEN.
Description: Dictionary program for boundary conditions in bending.
Logic: From the code for boundary conditions, the appropriate routine is called.
Length: 1272 bytes.
Calls: STHBDI, STHSLB, STHFFB.

Name: STHBKS.
Load Module: STHBKS.
Description: Executive program for backsubstitution and output of results.
Length: 1552 bytes.
Links: STHB1S.
Linked by: STHMAI.
Calls: H1OUT, HCODE, HTRANS, DE1BUG, STHDER.

Name: STHBLV.
Load Module: STHBCM.
Description: Program to assemble the generalized nodal rotation vector for the bending problem.
Logic: Nodal rotations are computed from the particular solution functions. A number of computation routines are used.
Length: 4596 bytes.
Calls: HPHI, NEXNOD.
Called by: STHBCM.

Name: STHBOU.
Load Module: STHINI.
Description: Boundary values are stored by program into BDCOND according to boundary condition involved.
Length: 3200 bytes.
Calls: GETNOS, HSTORE.

Name: STHBPS.
Load Module: STHPAR.
Description: Executive program to compute particular solution functions by double integration.
Length: 6416 bytes.
Links: STHP1R.
Calls: AND, HINTEG, NEXNOD.

Name: STHB1S.
Load Module: STHB1S.
Description: It is a continuation of STHBKS. It is the executive program for computing the strains and stresses of the nodes and elements.
Length: 1032 bytes.
Linked by: STHBKS.
Calls: H2OUT, H3OUT, HNSTAN, HNSTES, DE2BUG, HELEMT, DE3BUG.

Name: STHCBC.
Load Module: STH1FS.
Description: Program to modify boundary conditions of plate under concentrated load applied at center.
Length: 2108 bytes.
Calls: AND.
Called by: STHCON.

Name: STHCHK.
Load Module: STHMAI.
Description: Program to check loading and status of each node after input.
Logic: It constructs JEXT, JINT when no error is detected.
Length: 2224 bytes.
Calls: AND.
Called by: STHMAI.

Name: STHCON.
Load Module: STH1FS.
Description: Executive program for modifying boundary conditions of plate under concentrated load applied at center.
Length: 2228 bytes.
Calls: AND, STHCBC, NEXNOD.
Called by: STH1FS.

Name: STHDER.
Load Module: STHBKS.
Description: Program to compute at each node derivatives of the solved unknowns of the system equations.
Logic: Computation is carried out by three-point formulas or divided differences, processed along grid lines parallel to the global axes.
Length: 2232 bytes.
Calls: TPFORM.
Called by: STHBKS.

Name: STHESM.
Load Module: STHGEN.
Description: Program to compute the diagonal and lower half of the element local coefficient matrices. For the bending problem, it calls HDUAL to perform the duality conversion of material properties.
Length: 1216 bytes.
Calls: HDUAL.
Called by: STHGEN.

Name: STHFFB.
Load Module: STHBEN.
Description: This program processes the fixed support or free boundary conditions in bending.
Logic: The equivalent displacement or stress boundary conditions are specified.
Length: 1232 bytes.
Calls: AND, NEXNOD.
Called by: STHBEN.

Name: STHGEN.
Load Module: STHGEN.
Description: Executive program for generation of local coefficient matrices of elements. It also constructs NDPROP when necessary.
Length: 3656 bytes.
Linked by: STHMAI.
Calls: STHESM.
Messages: Error messages issued when:
1. element not of type 'CST'.
2. element of zero thickness.

Name: STHGRI.
Load Module: STHMAI.
Description: Program constructs the rectangular grid pattern of the nodes for differentiations of the final variables.
Logic: The grid pattern is formed as lines parallel to the axes by comparing nodal coordinates.
Length: 1848 bytes.
Called by: STHMAI.

Name: STHINI.
Load Module: STHINI.
Description: Program to initialize BDID, BDCOND, and IPROB.
Length: 780 bytes.
Messages: "Command valid only for plate stretching and bending."

Name: STHLOD
Load Module: STHINI.
Description: Program inputs external loading to the system.
Logic: Routines are written for load intensity and forces for uniform and non-uniform cases.
Length: 2984 bytes.
Calls: LCDBLE.

Name: STHMAI.
Load Module: STHMAI.
Description: The main program of the PLANAL System.
Logic: Depending on the symmetry of the coefficient matrix, the proper assembler and solver are called. Programs for constructing particular solution functions in bending and routines for temporary output are also controlled.
Length: 3484 bytes.
Links: STHNAS, STHNSL, STHASS, STHSVR, STHGEN, STHBKS, STHBCM, STH1FS.
Calls: HPSSLS, STHRBD, HCLOCK, STHCHK, STHGRI, STHTMO.

Name: STHNAS.
Load Module: STHNAS.
Description: Program to assemble the non-symmetric global coefficient matrix.
Length: 1740 bytes.
Linked by: STHMAI.
Calls: SETCLK, HNSASS.

Name: STHNSL.
Load Module: STHNSL.
Description: Program to prepare for so ution of non-symmetric system equations.
Length: 1428 bytes.
Linked by: STHMAI.
Calls: SETCLK, HGAUSS, HGAUBK.

Name: STHOUT.
Load Module: STHINI.
Description: Program transfers information from Command 'OUTPUT' to KODOUT.
Length: 656 bytes.

Name: STHPAR.
Load Module: STHPAR.
Description: Program to store input of particular solution functions in bending.
Length: 1912 bytes.
Calls: LCDBLE.

Name: STHP1R.
Load Module: STHP1R.
Description: Program to prepare for STHBPS by determining types of end conditions of all grid lines.
Length: 4360 bytes.
Linked by: STHBPS.
Calls: AND, NEXNOD, HTHETA, HDISLD.

Name: STHRBD.
Load Module: STHMAI.
Description: This program checks if a quantity in bending dual of a rigid body displacement has been supplied. If not, one would be specified.
Logic: When all boundary nodes are free or fixed, two stress functions and one "rotation" will be specified. When there is simple support or line of symmetry but there is no node with constructed function boundary condition, two stress functions are specified.
Length: 3244 bytes.
Calls: AND, HATAN, NEXNOD.
Called by: STHMAI.

Name: STHSAS.
Load Module: STHSAS.
Description: Program to transfer temporary submatrix to element of global matrix.
Length: 1520 bytes.
Linked by: SMIXED, SEDGEB, SELAST, STRAIN.
Calls: BITON.

Name: STHSEP.
Load Module: STHGEN.
Description: Program to store element properties.
Length: 1344 bytes.

Name: STHSLB.
Load Module: STHBEN.
Description: This program processes the simple or line of symmetry boundary conditions in bending.
Logic: The equivalent mixed bending boundary condition is specified.
Length: 1536 bytes.
Calls: AND, NEXNOD.
Called by: STHBEN.

Name: STHSLV.
Load Module: STHBCM.
Description: Program to assemble the generalized load vector for the stretching problem.
Logic: Load vector is computed from the externally applied loads.
Length: 1872 bytes.
Calls: AND
Called by: STHBCM.

Name: STHSSA.
Load Module: STHSAS.
Description: Program to transfer element of global matrix to temporary submatrix.
Length: 1352 bytes.
Linked by: SMIXED, SEDGEB, SELAST, HMODIF, STRAIN.
Calls: BITON.

Name: STHSTR.
Load Module: STHSTR.
Description: Dictionary program to branch to the appropriate program for processing the stretching boundary conditions.
Length: 956 bytes.
Links: STHS1R.
Linked by: STHBCM.
Calls: SDISPL, SSTRESS, SELAST, SEDGEB, SMIXED.

Name: STHSVR.
Load Module: STHSVR.
Description: Executive program to solve the symmetric system equations.
Length: 7588 bytes.
Linked by: STHMAI.
Calls: SETCLK, STDCPY, STIVDP, STADRS, STDMMP, STDMTR, STDMAD, SVRBUG.

Name: STHS1R.
Load Module: STHS1R.
Description: Continuation program of STHSTR.
Length: 528 bytes.
Linked by: STHSTR.
Calls: STRAIN.

Name: STHTCE.
Load Module: STHINI.
Description: Program traces the chain of boundary nodes in the positive s-direction.
Logic: The chain is formed by examining the nodes following a current boundary node around all elements incident on that node.
Length: 3892.
Calls: AND, SETCLK, LCDBLE.
Messages: Error messages are issued when boundary chain cannot be formed because of input errors.

Name: STHTMO.
Load Module: STHMAI.
Description: Program to make intermediate output of arrays when requested.
Length: 5120 bytes.
Called by: STHMAI.

Name: STHTRA.
Load Module: STHINI.
Description: Program transforms an integer from integer format to alphameric format.
Logic: Integer is converted digit by digit.
Length: 1048 bytes.

Name: STH1FS.
Load Module: STH1FS.
Description: Program to check input of geometry and loading before construction of particular solution functions by Fourier series.
Length: 5956 bytes.
Linked by: STHMAI.
Calls: AND, STHCON, STH2FS.

Name: STH2FS.
Load Module: STH1FS.
Description: Executive program for construction of particular solution functions by Fourier series.
Length: 4984 bytes.
Calls: AND, STHBFS
Called by: STH1FS.

Name: STH3FS.
Load Module: STH1Fs.
Description: Program to perform actual summation of Fourier series.
Length: 4316 bytes.
Calls: STH4FS.
Called by: STH2Fs.

Name: STH4FS.
Load Module: STH1FS.
Description: Program to compute the coefficients for summation by Fourier series.
Length: 2076 bytes.
Calls: HSIGN.
Called by: STHBFS.

Name: STIFED.
Load Module: STHSTR.
Description: Program to compute the local stiffness coefficients for edge beam in stretching.
Length: 1524 bytes.
Called by: SEDGEB.

Name: STRAIN.
Load Module: STHS1R.
Description: Program to process the strain boundary condition in bending.
Length: 6252 bytes.
Links: STHSAS, STHSSA.
Calls: AND, HPHI, NEXNOD, HCHECK, HROTAT, HMODIF, HINITL, ENDSTN.
Called by: STHS1R.

Name:	SVRBUG.
Load Module:	STHSVR.
Description:	Program to point out KPPRI during iterations in Gauss reduction of symmetric system equations.
Length:	652 bytes.
Called by:	STHSVR.

Name:	TPFORM.
Load Module:	STHBKS.
Description:	Program to compute derivatives by a three-point formula.
Logic:	The formula used is selected by code in argument list.
Length:	848 bytes.
Called by:	STHDER.

APPENDIX G

PROGRAM LISTINGS

Complete Listings of the Command Definition Blocks and Icetran programs used in the PLANAL System may be obtained upon request.