

FILE COPY

ESD ACCESSION LIST

DRI Call No. 87872

Copy No. 1 of 2 cys.

Technical Note

1977-17

S. S. Britten

Finite Element Capabilities
in M.I.T. Lincoln Laboratory
Version of STRUDL

30 September 1977

Prepared for the Department of the Air Force
under Electronic Systems Division Contract F19628-76-C-0002 by

Lincoln Laboratory

MASSACHUSETTS INSTITUTE OF TECHNOLOGY

LEXINGTON, MASSACHUSETTS



Approved for public release; distribution unlimited.

ADA048605

BEST AVAILABLE COPY

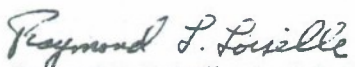
The work reported in this document was performed at Lincoln Laboratory, a center for research operated by Massachusetts Institute of Technology, with the support of the Department of the Air Force under Contract F19628-76-C-0002.

This report may be reproduced to satisfy needs of U.S. Government agencies.

The views and conclusions contained in this document are those of the contractor and should not be interpreted as necessarily representing the official policies, either expressed or implied, of the United States Government.

This technical report has been reviewed and is approved for publication.

FOR THE COMMANDER

A handwritten signature in cursive script, reading "Raymond L. Loiselle".

Raymond L. Loiselle, Lt. Col., USAF
Chief, ESD Lincoln Laboratory Project Office

MASSACHUSETTS INSTITUTE OF TECHNOLOGY
LINCOLN LABORATORY

FINITE ELEMENT CAPABILITIES
IN M.I.T. LINCOLN LABORATORY VERSION OF STRUDL

S. S. BRITTEN

Group 73

TECHNICAL NOTE 1977-17

30 SEPTEMBER 1977

Approved for public release; distribution unlimited.

LEXINGTON

MASSACHUSETTS

ABSTRACT

The well-known, general-purpose structural analysis program ICES-STRUDL has been used extensively at M.I.T. Lincoln Laboratory since its public release in 1967. Since that original release, several updates, issued by the original developers and the ICES Users Group, have produced changes in programs and documentation. Likewise, additional development and enhancement of ICES-STRUDL have occurred at M.I.T. Lincoln Laboratory on those areas (such as finite elements, dynamics, etc.) specifically of interest to the Laboratory.

This report describes the finite element capabilities in the M.I.T. Lincoln Laboratory version of ICES-STRUDL and can therefore serve as a user's manual. Sections are devoted to specification of finite element geometry, element properties, and element loading. Finite element modelling considerations are discussed and three examples are presented to illustrate capabilities often needed at the Laboratory, but not described in other ICES-STRUDL documentation.

TABLE OF CONTENTS

	<u>Page</u>
ABSTRACT	iii
I. FINITE ELEMENT GEOMETRY SPECIFICATION	1
1. Planar Element Coordinate System	1
2. LOCAL and GLOBAL Joints	3
3. Element Nodal Listing	6
4. Convention for Stress Output	9
II. FINITE ELEMENT PROPERTY SPECIFICATION	11
1. Element Type Selection	11
2. Thermal Expansion Coefficients	16
3. Rigidity Matrix Description	17
4. Stiffness/Mass Matrix Description	21
5. Special DUMMY Element Specification	22
6. Element Similarities Specification	23
7. Examples of ELEMENT PROPERTIES Specification	25
III. FINITE ELEMENT LOADING SPECIFICATION	27
1. Edge Forces	27
2. Surface Forces	31
3. Body Forces	34
4. Thermal Forces	35
5. Initial Strains	36
IV. FINITE ELEMENT MODELING CONSIDERATIONS	38
1. Convergence and Accuracy of Model	38
2. Interpretation of Results	39

V. EXAMPLES	42
1. Plane Stress Example	42
2. Fracture Mechanics Example	45
3. Super Finite Element Generation Example	50
VI. APPENDIX: Description of Finite Elements	57
REFERENCES	73

LIST OF ILLUSTRATIONS

	<u>Page</u>
Fig. 1.1. Planar element coordinate system (general).	2
Fig. 1.2. Planar element coordinate system ($ X_G Y_G$).	3
Fig. 1.3. Example of LOCAL/GLOBAL joint designation.	5
Fig. 1.4. Nodal sequence for planar elements.	6
Fig. 1.5. Nodal sequence for tridimensional elements.	7
Fig. 1.6. Nodal sequence for IPQS element.	8
Fig. 1.7. Sign convention for planar element stresses.	9
Fig. 1.8. Sign convention for tridimensional element stresses.	10
Fig. 2.1. Example of rigidity matrix transformation.	20
Fig. 2.2. Rotation angles for element similarities.	24
Fig. 3.1. Edge numbering for planar elements.	28
Fig. 3.2. Edge numbering for IPLS element.	29
Fig. 3.3. Edge numbering for IPQS element.	29
Fig. 3.4. Edge numbering for TRIP element.	30
Fig. 3.5. Local coordinate system for edge loads.	30
Fig. 3.6. Local coordinate systems for surface loads.	32
Fig. 3.7. Face numbering for TRIP element.	33
Fig. 3.8. Face numbering for IPLS element.	33
Fig. 3.9. Face numbering for IPQS element.	34
Fig. 5.1. Solid cantilever beam.	42
Fig. 5.2. Finite element model of beam.	42
Fig. 5.3. Input listing of STRUDL model of beam.	43
Fig. 5.4. Results of solid cantilever example.	44

Fig. 5.5.	Elastic fracture mechanics modes.	45
Fig. 5.6.	Finite element model of fracture specimen.	47
Fig. 5.7.	Input listing of STRUDL model of fracture specimen.	48
Fig. 5.8.	Convergence of K_I for fracture specimen.	49
Fig. 5.9.	Finite element model of plate assembly.	50
Fig. 5.10.	Input listing of STRUDL model used in super finite element generation.	53
Fig. 5.11.	Comparative STRUDL results for differing triangular meshes (skewed vs. symmetric).	54
Fig. 5.12.	Partial listing of STRUDL super finite element model.	55

LIST OF TABLES

	<u>Page</u>
Table 2.1. Plane Stress/Strain Elements.	13
Table 2.2. Plate Bending Elements.	14
Table 2.3. Plate Elements.	14
Table 2.4. Tridimensional Elements.	15
Table 4.1. Output Summary for Finite Elements.	41

I. FINITE ELEMENT GEOMETRY SPECIFICATION

1. Planar Element Coordinate System

Each planar finite element in a STRUDL problem has associated with it a coordinate system in which the results for that element are expressed. This coordinate system is independent of the type of element and depends only upon the orientation of the plane of the element. It is called the planar element coordinate system.

The use of such a reference frame enables the element results (stresses and strains) for all of the elements in a particular plane to be output with respect to a common reference frame. This makes the interpretation of output a relatively easy task, since similar results are all expressed in the same directions.

The planar element coordinate system of a triangular element (nodes 1, 2, 3) is illustrated in Figure 1.1 for the general case.

The orientation of the element affects the determination of its planar element coordinate system. The planar system is located as follows (refer to Figure 1.1):

- (1) The planar Z_p axis is normal to the plane of the element. Its positive direction is determined by applying the right-hand rule to the node order given in the ELEMENT INCIDENCES command.
- (2) The planar X_p axis coincides with the intersection of the element plane and the global $X_G Y_G$ plane.
- (3) The projection of the planar Y_p axis onto the global Z_G axis is defined to be positive in the same direction as the global Z_G axis.
- (4) The planar X_p axis is defined to be positive in the direction determined by the right-hand rule.

The triangular element shown in Figure 1.1 has relative node numbers 1, 2, 3 which correspond to the nodes listed in the ELEMENT INCIDENCES for the element.

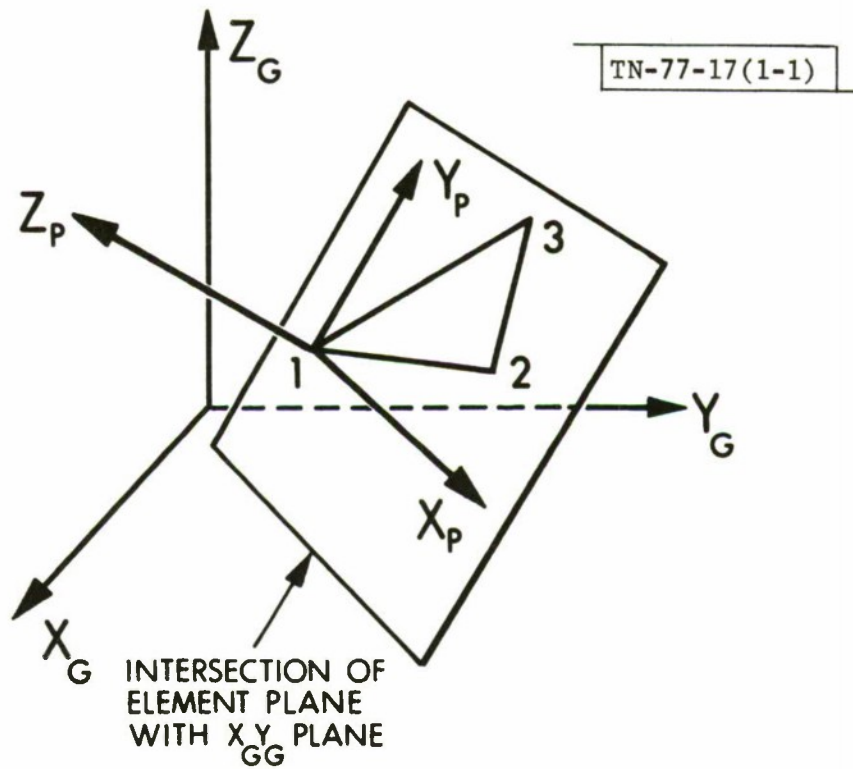


Figure 1.1. Planar element coordinate system (general).

The only exception to the above rules for location of the planar element coordinate system occurs when the element lies in a plane parallel to the global $X_G Y_G$ plane. The planar element coordinate system of such a triangular element is illustrated in Figure 1.2.

The planar system for an element which lies in a plane parallel to the global $X_G Y_G$ plane is located as follows (refer to Figure 1.2):

- (1) The planar Z_P axis is normal to the plane of the element. Its positive direction is determined by applying the right-hand rule to the node order given in the ELEMENT INCIDENCES command.
- (2*) The planar X_P axis is located parallel to the global X_G axis and is positive in the same direction as the global X_G axis.

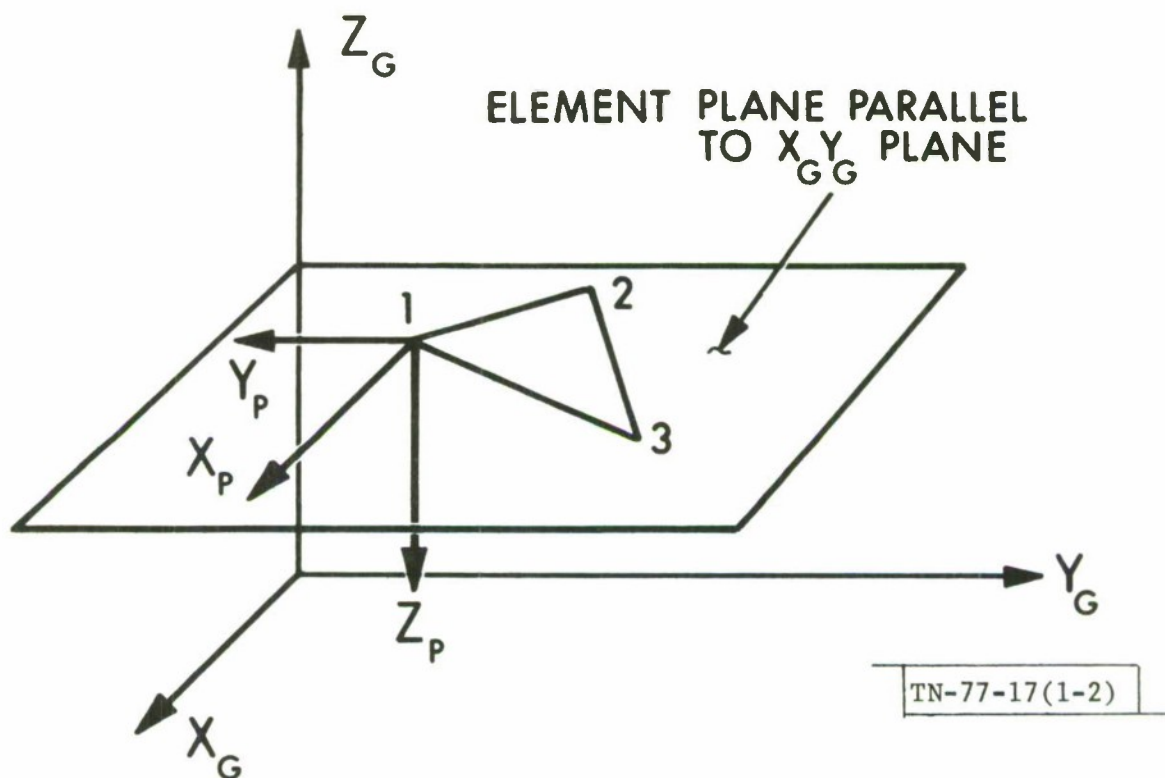


Figure 1.2. Planar element coordinate system ($\parallel X_G Y_G$).

(3*) The planar Y_P axis is defined to be positive in the direction determined by the right-hand rule.

The triangular element shown in Figure 1.2 has relative node numbers 1, 2, 3 which correspond to the nodes listed in the ELEMENT INCIDENCES for the element.

2. LOCAL and GLOBAL Joints

The JOINT COORDINATES command has been modified to provide the capability for analyzing a structure comprised of plates which do not lie in global planes. If all finite elements attached to a joint are co-planar (in other than a global plane), and no members are incident on the joint, the joint could be declared LOCAL. This LOCAL specification would cause the solution at the joint to be carried out in the planar element coordinate system; thus eliminating potential instabilities in the stiffness matrix caused by rotation into the global coordinate system.

All input for a LOCAL joint must be referred to the planar element coordinate system (e.g., JOINT RELEASES, JOINT LOADS, etc.). All output (e.g., displacements) for a LOCAL joint are also referred to the planar element coordinate system.

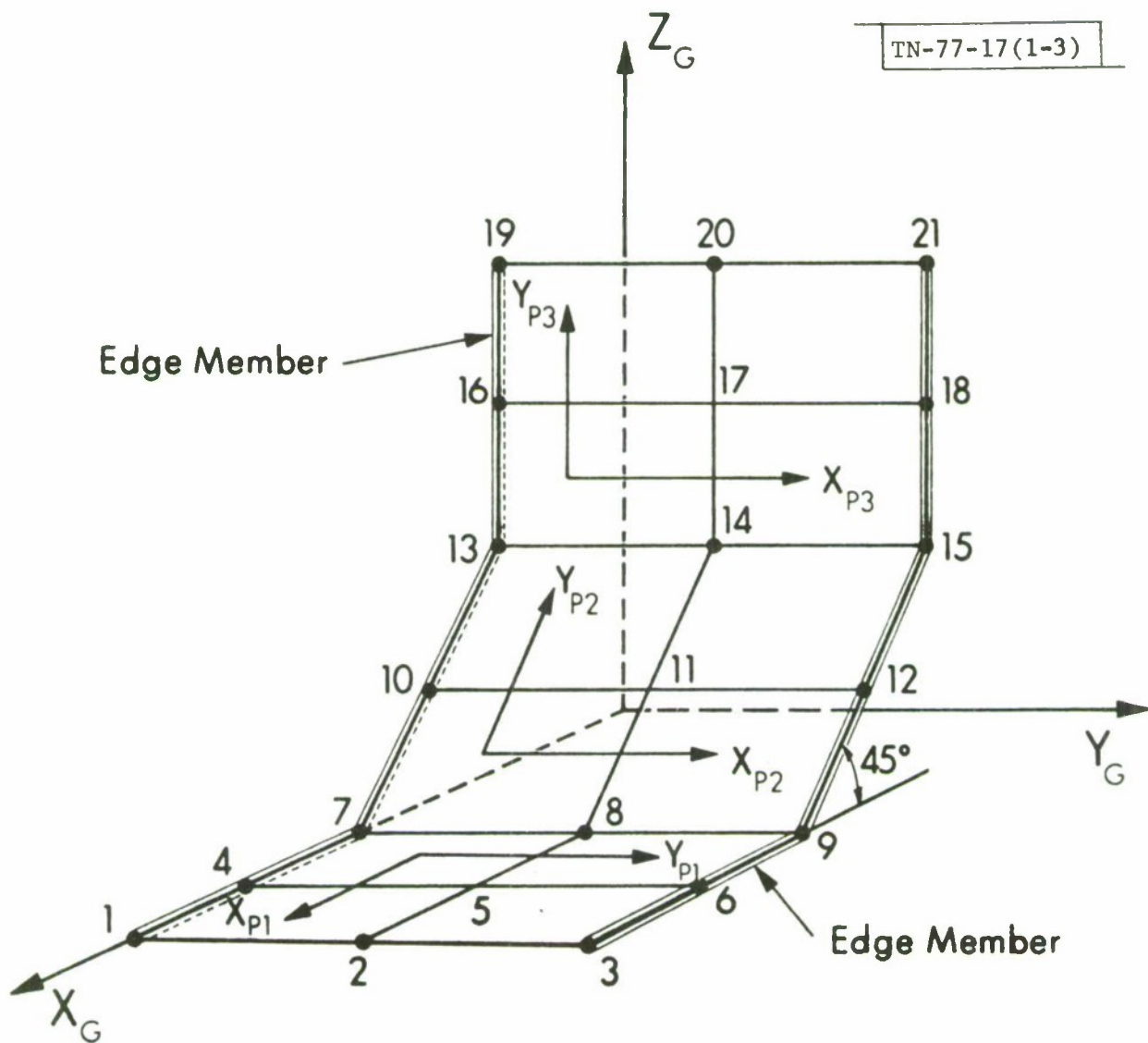
Joints connected only by elements lying in one of the three global planes ($X_G Y_G$, $X_G Z_G$, or $Y_G Z_G$) should be referred to as GLOBAL joints for ease in describing input and interpreting output. Similarly, joints connected by co-planar elements (in other than a global plane) should be referred to as SUPPORTED GLOBAL joints, and JOINT RELEASES (with THETA angles if needed) should be used to constrain the directions of potential instability.

Three cases exist in which joints should not be described as LOCAL joints. These are:

- (1) Nodes at which elements are incident in more than one plane.
- (2) Nodes at which incident elements are all parallel to global planes (instabilities will not be generated and the reduced number of unknowns will automatically be used).
- (3) Nodes at which members are incident.

In case (1) above, the specification of LOCAL will lead to incorrect results. In case (2), the specification of LOCAL will change the orientation of the input loads and output results to the planar element coordinate system. In case (3), the node cannot be specified as LOCAL, as an error message will result, and it will be processed as if a GLOBAL specification had been given.

Joints 7, 8, 9 and 13, 14, 15 in Figure 1.3 illustrate case (1) above and must be described as GLOBAL, since they are nodes at which elements are incident in more than one plane. Nodes 5 and 17 illustrate case (2) and are described as GLOBAL since their incident elements are parallel to global planes. Node 11 may be described as a LOCAL joint but is better described as a SUPPORTED GLOBAL joint with associated JOINT RELEASES and THETA angles. Nodes at which the edge members are incident illustrate case (3) and must be described as GLOBAL.



Preferred Approach:

JOINT COORDINATES \$ DEFAULT IS GLOBAL

1 TO 10

11 SUPPORT

12 TO 21

UNIT DEGREES

JOINT RELEASES

11 FORCE X Y TH1 90. TH3 45.

Figure 1.3. Example of LOCAL/GLOBAL joint designation.

3. Element Nodal Listing

The connectivity of element nodes (or joints) is described by listing the joints associated with each element in the ELEMENT INCIDENCE command.

For planar elements having only corner nodes, the joints are listed consecutively around the element boundary, starting at any arbitrary corner node. For planar elements having side nodes, the corner nodes are listed first and then the side nodes.

The mid-side nodes for the LST and LSR elements must be listed in the JOINT COORDINATES command, but one does not have to specify the joint coordinate values. This also pertains to the isoparametric IPQQ (or IPQS) and IPCQ elements when the boundary is straight and the mid-side node is located at the 1/2- or 1/3-points, respectively. The PRINT Joint COORDINATES command, given

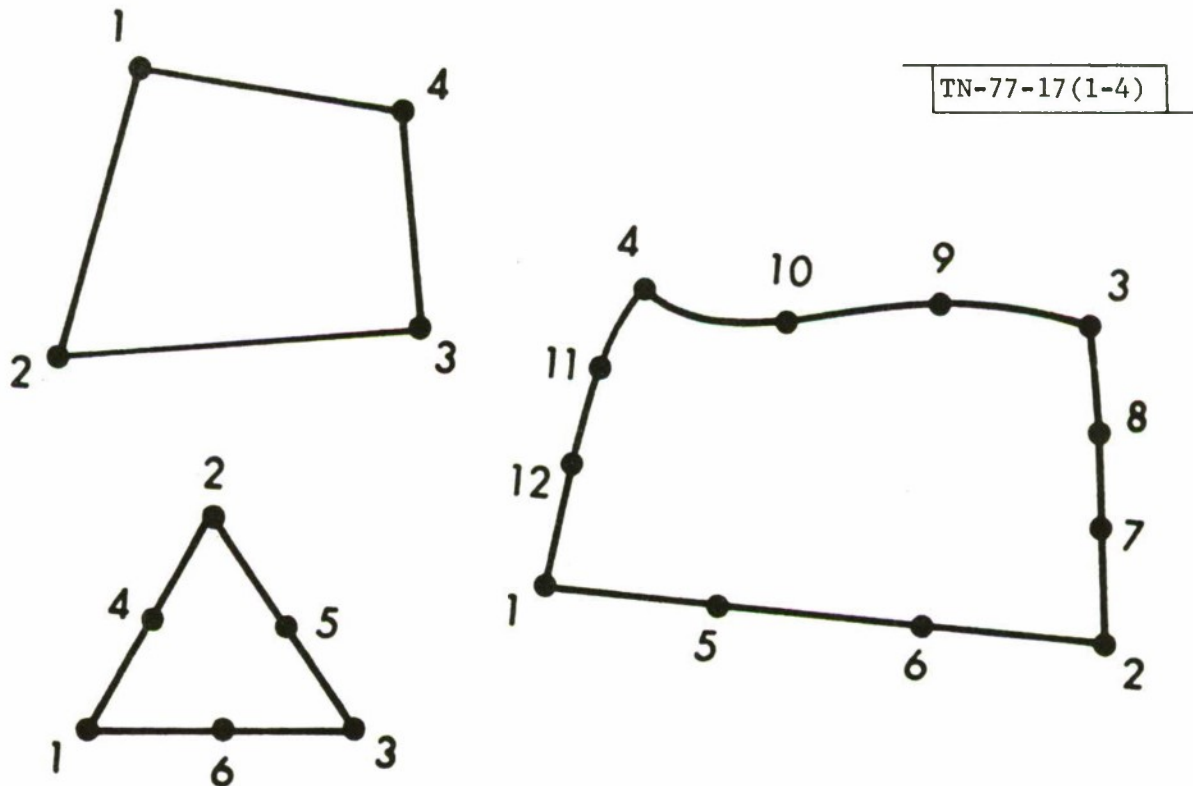


Figure 1.4. Nodal sequence for planar elements.

after an ANALYSIS command, will print the computed joint coordinates of the unspecified mid-side nodes. Before an ANALYSIS is performed, however, the values will be zero.

The nodal sequence for planar elements is illustrated in Figure 1.4. Note that numbering may proceed in either a clockwise or counterclockwise direction. For planar results to make sense on a given plane however, it is preferable to choose one direction for all elements on that given plane.

The relative node numbering for tridimensional elements having only corner joints, such as the IPLS and TRIP elements, is obtained by numbering an arbitrary

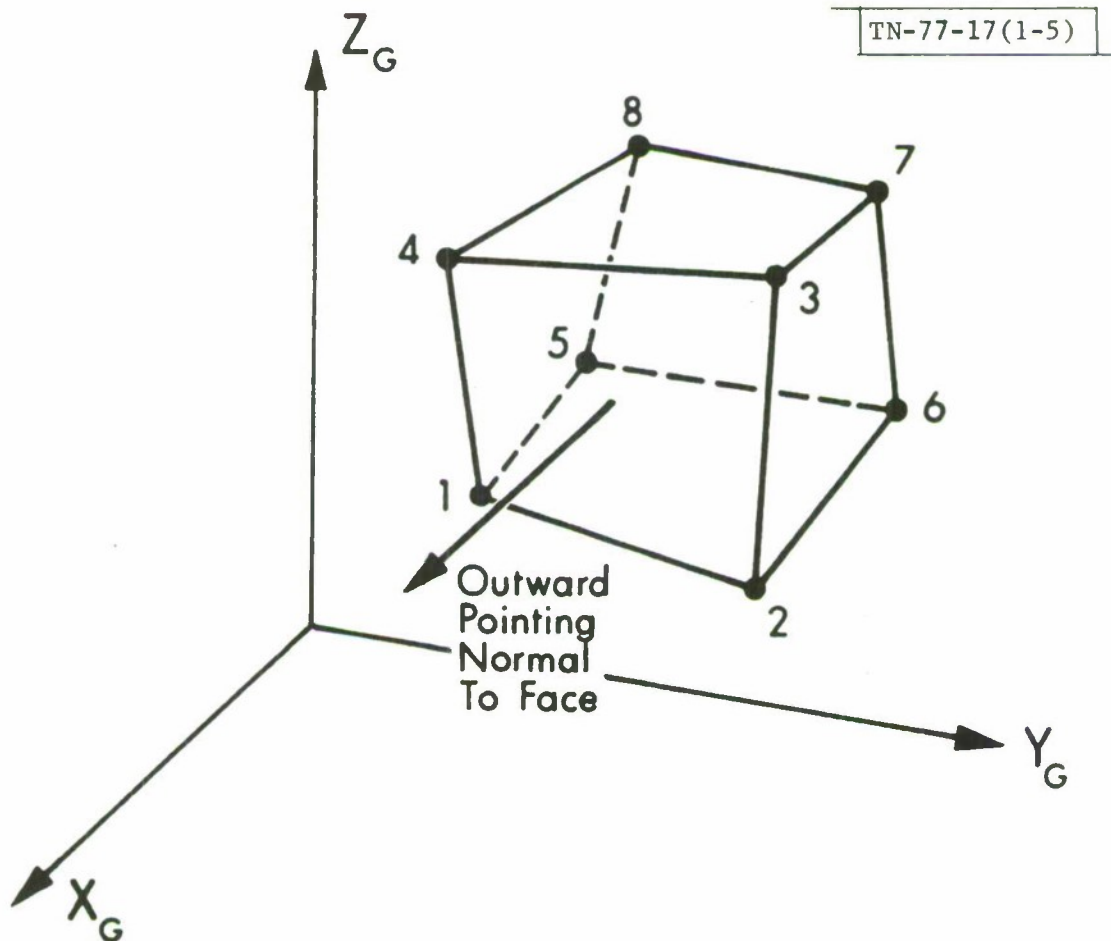


Figure 1.5. Nodal sequence for tridimensional elements.

face. The face is numbered by proceeding around the boundary of the face in a direction such that an outward-pointing normal is produced by applying the right-hand rule to the node ordering. The back face is then numbered by proceeding in the same direction from the joint directly back of the first numbered node. Refer to Figure 1.5.

The scheme for the tridimensional IPQS element is similar. One works from the front face to the mid-face nodes and then to the back face. All side nodes on the front face are numbered similar to planar element side nodes before proceeding to the mid-face nodes. Refer to Figure 1.6.

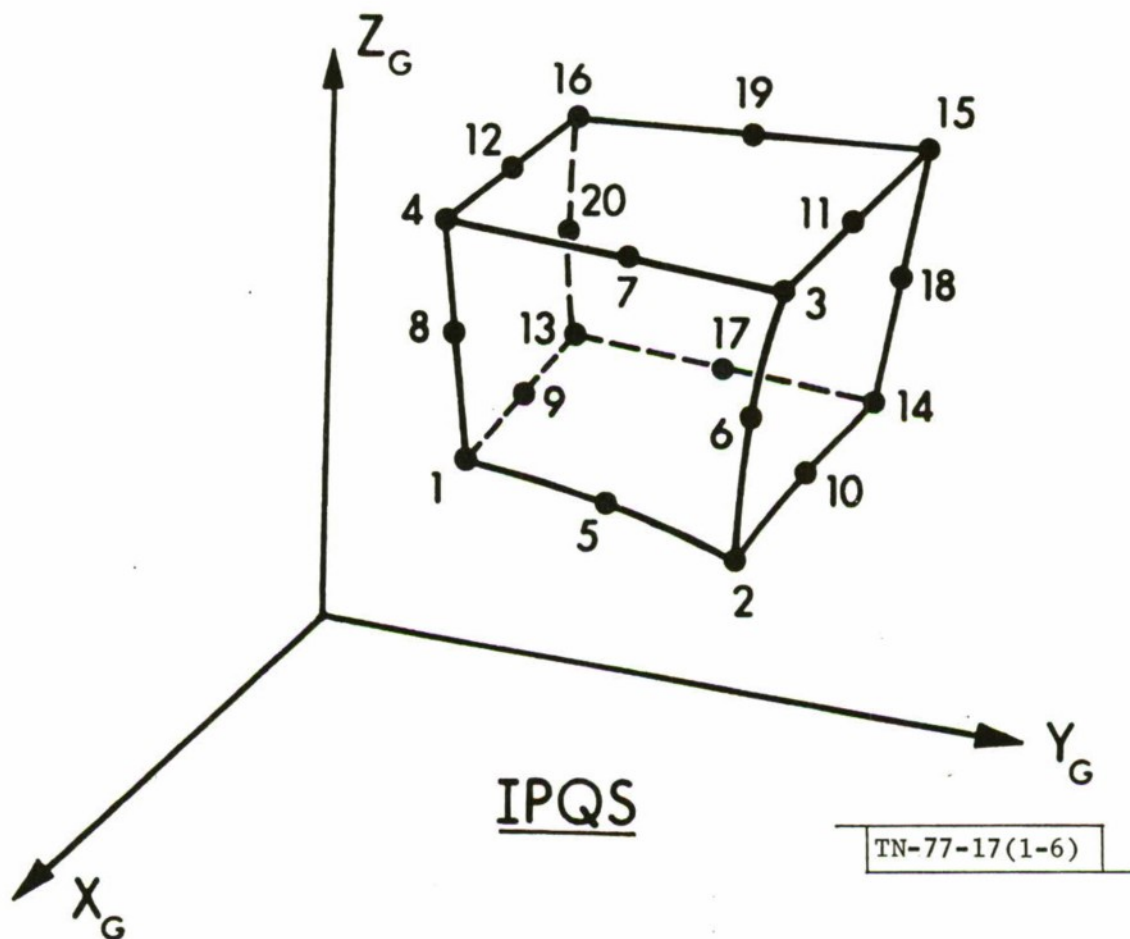


Figure 1.6. Nodal sequence for IPQS element.

4. Convention for Stress Output

Stress and strain results for the planar elements are output with respect to the element planar coordinate system. Actual stresses are output for the plane stress/strain elements, while stress resultants are output for the plate bending elements. (These stress resultants are in units of shear and moment per unit length.) Refer to Figure 1.7 for positive sign convention.

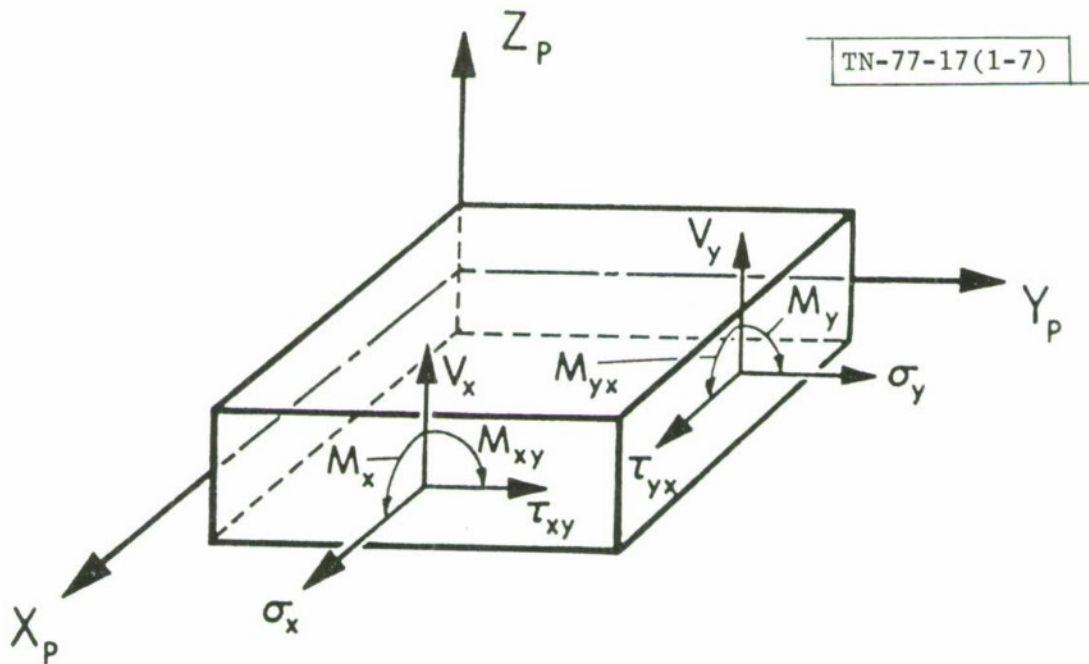


Figure 1.7. Sign convention for planar element stresses.

Principal stress and principal strain values are available for some plane stress/strain elements with the angle between the principal directions and the planar coordinate system also printed.

Stress and strain results for the tridimensional elements are output with respect to the global coordinate system. Principal stresses and strains are also available. Refer to Figure 1.8 for positive sign convention.

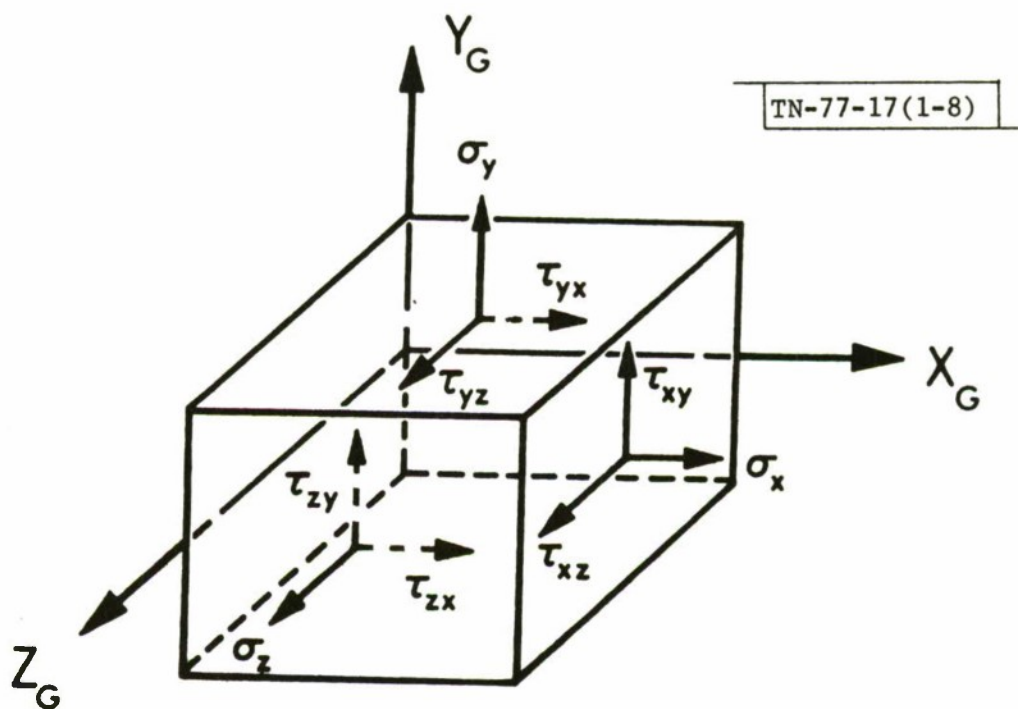


Figure 1.8. Sign convention for tridimensional element stresses.

II. FINITE ELEMENT PROPERTY SPECIFICATION

1. Element Type Selection

The structural TYPE command for finite element analysis consists of options for specifying various finite element types.

PLANE STRESS and PLANE STRAIN specify two-dimensional problems varying only with respect to X_p and Y_p . This includes the surface and body forces, which also must be planar and independent of Z_p . Two degrees-of-freedom are defined at each node, usually the u and v displacements in the element planar coordinate system.

In PLANE STRESS problems the out-of-plane normal stress is considered to be negligible, while the in-plane stresses are assumed to be independent of Z_p . In other words,

$$\begin{aligned} \sigma_z = \tau_{xz} = \tau_{yz} &\approx 0. \\ \text{and } \sigma_x, \sigma_y, \tau_{xy} &\text{ independent of } z. \end{aligned}$$

These statements violate certain compatibility equations, but can be shown to be a reasonable assumption for thin plates.

In PLANE STRAIN problems the out-of-plane strain is considered to be zero, while the in-plane strains are assumed to be independent of Z_p . In other words,

$$\begin{aligned} \epsilon_z = \gamma_{xz} = \gamma_{yz} &\equiv 0 \\ \text{and } \epsilon_x, \epsilon_y, \gamma_{xy} &\text{ independent of } z. \end{aligned}$$

The corresponding physical requirement which justifies this assumption is that the thickness is significant relative to the representative X_p and Y_p dimensions.

PLATE BENDING specifies problems where only bending deformations need to be considered. Plate bending formulations include three degrees-of-freedom at each node; usually the displacement w in the direction normal to the plane

of the element, and the rotations θ_x and θ_y in the element planar coordinate system. The effect of transverse shear flexibility is ignored.

PLATE specifies a combination of PLANE STRESS and PLATE BENDING in which the uncoupled stiffnesses of the two components are superimposed to represent a three-dimensional thin flat plate.

TRIDIMENSIONAL specifies problems where a three-dimensional state of stress exists in the body and requires solid elements rather than planar ones.

At least one TYPE command must precede an ANALYSIS command. A suggested means of specification is to associate the TYPE command with the ELEMENT INCIDENCES command and to input the element incidence data for all elements of the same type at the same time. This also makes it easy to identify related elements at a later time. Since the TYPE command is noted for elements when processing ELEMENT INCIDENCES, it is possible to combine PLANE STRESS and PLANE STRAIN elements in the same ANALYSIS.

The currently available STRUDL finite elements are listed in Tables 2.1 - 2.4 for the element types described in this section. In addition, each finite element is described in detail in the Appendix. The following properties are available for these elements unless noted otherwise:

- (1) Isotropic, orthotropic, and anisotropic material.
- (2) Lumped and consistent mass matrices.
- (3) Element loadings (described in Section III) consisting of:
 - a. Edge Forces
 - b. Surface Forces
 - c. Body Forces
 - d. Thermal Forces
 - e. Initial Strains

TABLE 2.1
PLANE STRESS/STRAIN ELEMENTS

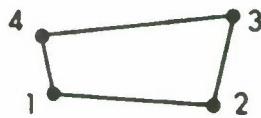
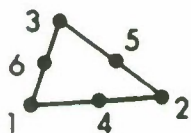
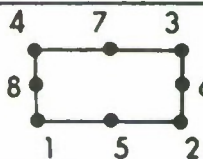
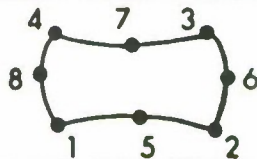
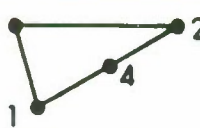
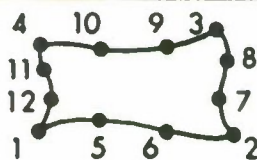
ELEMENT NAME	SHAPE	NO. OF NODES	COMMENTS
CSTG		3	
PSR PSRCSH		4	
IPLQ IPLQCSH		4	
LST		6	
LSR		8	
IPQQ		8	
UTLQ1		4	
IPCQ		12	

TABLE 2.2
PLATE BENDING ELEMENTS

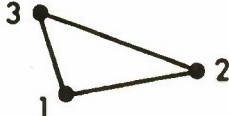
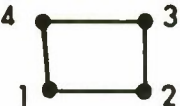
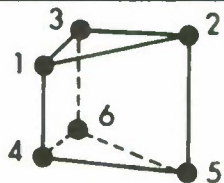
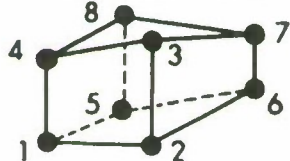
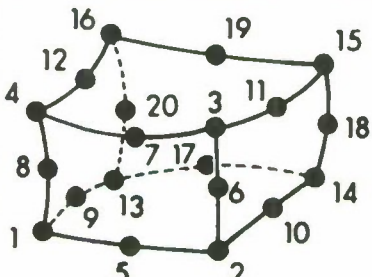
ELEMENT NAME	SHAPE	NO. OF NODES	COMMENTS
CPT		3	No edge or body forces
BPR		4	No edge or body forces
PBQ1		4	No edge or body forces

TABLE 2.3
PLATE ELEMENTS

ELEMENT NAME	SHAPE	NO. OF NODES	COMMENTS
SBCT		3	No initial strains

TABLE 2.4
TRIDIMENSIONAL ELEMENTS

ELEMENT NAME	SHAPE	NO. OF NODES	COMMENTS
TRIP		6	
IPLS IPLSCSH		8	
IPQS		20	

2. Thermal Expansion Coefficients

The initial strains due to temperature are evaluated from:

$$\underline{\epsilon}_0 = \Delta T \underline{\alpha}$$

where

$\underline{\epsilon}_0$ = vector of initial strains

$\underline{\alpha}$ = vector of thermal expansion coefficients

ΔT = temperature differential

The thermal expansion coefficient directions correspond to the planar element coordinate system for planar elements and to the global coordinate system for tridimensional elements.

The ELEMENT PROPERTIES command is used to input the thermal expansion coefficients when the material is orthotropic or anisotropic.

$$\begin{array}{ccccccc} \underline{\text{CTE}} & [\underline{\text{CAX}}] & v_1 & [\underline{\text{CAY}}] & v_2 & [\underline{\text{CAZ}}] & v_3 & - \\ & [\underline{\text{CSXY}}] & v_4 & [\underline{\text{CSXZ}}] & v_5 & [\underline{\text{CSYZ}}] & v_6 & \end{array}$$

where v_1, v_2, v_3 = axial thermal expansion coefficients

v_4, v_5, v_6 = shear thermal expansion coefficients

If the material is isotropic, the shear thermal expansion coefficients are zero and the axial thermal expansion coefficients are a constant value α . We can input α with the CONSTANTS command, since

$$\alpha_x = \alpha_y = \alpha_z = \alpha$$

$$\alpha_{xy} = \alpha_{xz} = \alpha_{yz} = 0$$

3. Rigidity Matrix Description

The ELEMENT PROPERTIES command is used to input element type, thickness, and, optionally, the material rigidity matrix. The planar stretching rigidity matrix, $\underline{D}_S$, is defined as the relationship between in-plane stresses and strains, and is expressed as:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \underline{D}_S \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix}$$

The planar bending rigidity matrix, $\underline{D}_B$, relates the stress couples and the bending deformations as follows:

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \underline{D}_B \begin{Bmatrix} -w_{,xx} \\ -w_{,yy} \\ -2w_{,xy} \end{Bmatrix}$$

The rigidity matrix, $\underline{D}_3$, for the tridimensional case is defined in the same manner as the planar stretching rigidity matrix, i.e.,

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{xz} \\ \tau_{yz} \end{Bmatrix} = \underline{D}_3 \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix}$$

Since the thickness, h , of the plate bending elements is assumed constant,

$$\underline{D}_B = \frac{h^3}{12} \underline{D}_S$$

If the material is orthotropic or anisotropic, we input $\underline{D}_S$ or $\underline{D}_3$ using the ELEMENT PROPERTIES command. The tridimensional rigidity matrix $\underline{D}_3$ is input by rows (6 x 6 array), as is the planar rigidity matrix $\underline{D}_S$ (3 x 3 array). The units are expressed in FL^{-2} terms (usually psi), but depend on the UNITS specifications.

<u>RIGIDITY MATRIX</u>	<u>NC</u>	<u>ic</u>
<u>ROW</u> i_1	V_{11}	----- V_{1n}
.		
.		
.		
<u>ROW</u> i_n	V_{n1}	----- V_{nn}

where:

ic = number of columns in the rigidity matrix;
 $i_1 \dots i_n$ = row numbers of the rows being specified;
 $v_{11} \dots v_{nn}$ = values in the rigidity matrix.

The rigidity matrix for planar elements is specified in the planar element coordinate system, while the rigidity matrix for tridimensional elements is specified in the global coordinate system.

If the material is isotropic, we input E , ν with the CONSTANTS command since $\underline{D}'_S$ for plane strain can be computed internally as:

$$\underline{D}'_S = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} (1-\nu) & \nu & 0 \\ \nu & (1-\nu) & 0 \\ 0 & 0 & \frac{(1-2\nu)}{2} \end{bmatrix}$$

and $\underline{D}_S$ for plane stress as:

$$\underline{D}_S = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix}$$

The isotropic tridimensional rigidity matrix may also be expressed by inputting E, ν with the CONSTANTS command, since

$$\underline{D}_3 = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} (1-\nu) & \nu & \nu & 0 & 0 & 0 \\ \nu & (1-\nu) & \nu & 0 & 0 & 0 \\ \nu & \nu & (1-\nu) & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{(1-2\nu)}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{(1-2\nu)}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{(1-2\nu)}{2} \end{bmatrix}$$

When treating a laminated plate, the equivalent homogeneous stretching and bending rigidity matrices are generated by defining $\underline{D}_S$ and $\underline{D}_B$ as follows:

$$\underline{D}_S = \frac{1}{h} \sum_{\text{layers}} \underline{D}_i h_i$$

$$\underline{D}_B = \frac{12}{h^3} \sum_{\text{layers}} \int_z z^2 \underline{D}_i dz$$

It sometimes happens that orthotropic planar rigidity exists in a local element orientation. It is then necessary to transform the orthotropic local rigidity matrix, $\underline{D}_L$, to an anisotropic planar rigidity matrix, $\underline{D}_P$.

$$\underline{D}_P = \underline{T}_\sigma \underline{D}_L \underline{T}_\sigma^T$$

where the stress transformation $\mathcal{T}_\sigma$ is given by:

$$\mathcal{T}_\sigma = \begin{bmatrix} \cos^2 \theta & \sin^2 \theta & \sin 2\theta \\ \sin^2 \theta & \cos^2 \theta & -\sin 2\theta \\ -\frac{\sin 2\theta}{2} & \frac{\sin 2\theta}{2} & \cos 2\theta \end{bmatrix}$$

An example of such a transformation is illustrated in Figure 2.1 for an orthotropic edge. The positive sign convention for stresses and rotations in the local and planar reference frames is also illustrated.

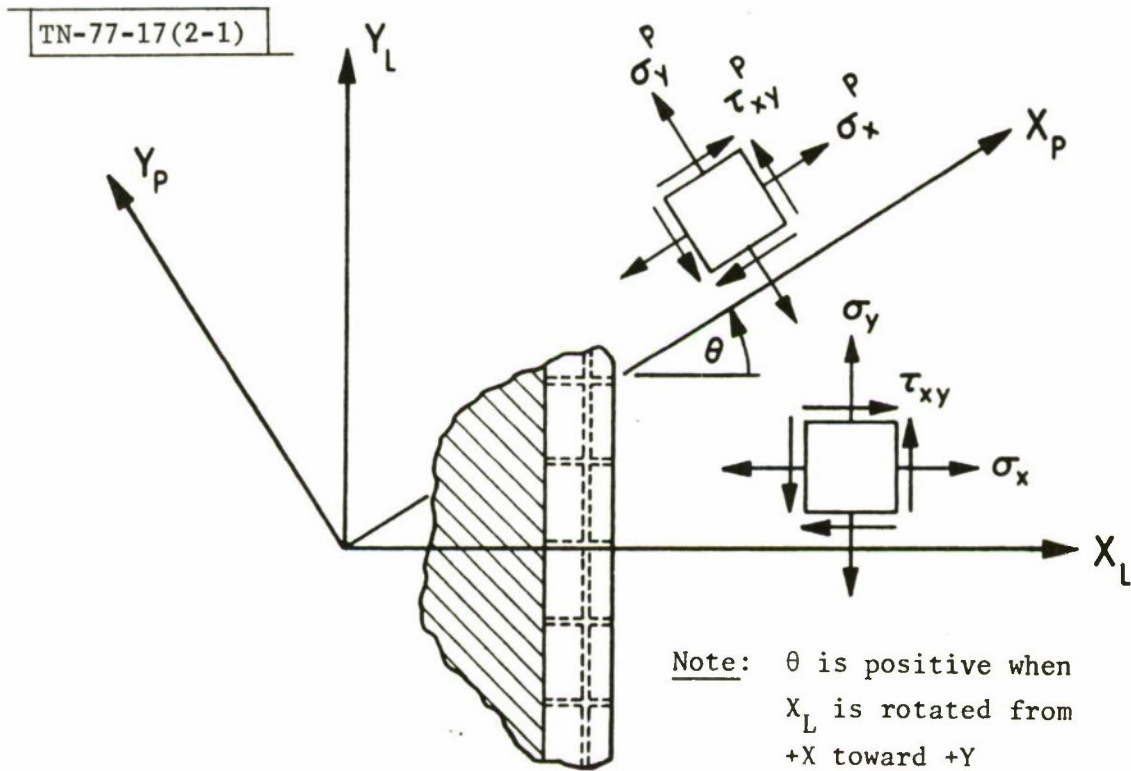


Figure 2.1. Example of rigidity matrix transformation.

4. Stiffness/Mass Matrix Description

The stiffness and mass matrices are expressed in partitioned form, where each submatrix is identified by its hyperrow and hypercolumn subscripts, i and j , respectively. Each submatrix is a square array whose order is the number of degrees of freedom per node. The degrees of freedom at each node must conform to those of the element whose "type name" is given. The partitioned form of the matrix is:

$$\tilde{K} = \begin{bmatrix} K_{11} & & K_{1j} & & K_{1n} \\ \vdots & & \vdots & & \vdots \\ K_{i1} & & K_{ij} & & K_{in} \\ \vdots & & \vdots & & \vdots \\ K_{n1} & & K_{nj} & & K_{nn} \end{bmatrix} \quad \tilde{K}_{ij} = \begin{bmatrix} K_{11} & \dots & K_{1m} \\ \vdots & & \vdots \\ K_{m1} & \dots & K_{mm} \end{bmatrix}$$

Since it is symmetric, only the unique, non-zero partitions need be given (i.e., only the diagonal partitions and either the upper OR lower off-diagonal partitions). If a submatrix has non-zero terms only on the diagonal, the DIAGONAL form (as shown in the command description) may be used.

$$\left\{ \begin{array}{l} \underline{\text{MASS}} \\ \underline{\text{STIFFNESS}} \end{array} \right\} \quad (\underline{\text{MATRIX}}) \quad [\underline{\text{NODES}}] \quad n \quad [\underline{\text{NDF}}] \quad m$$

$$\underline{\text{SUBMATRIX}} \quad i \quad j \quad \left\{ \begin{array}{l} \begin{bmatrix} v_{11} & \dots & v_{1m} \\ \vdots & & \vdots \\ v_{m1} & \dots & v_{mm} \end{bmatrix} \\ \text{DIAGONAL } v_{11} \dots v_{mm} \end{array} \right\}$$

where

n = number of nodes of the element;
m = number of degrees of freedom per node;
i = hyperrow subscript of the submatrix being specified;
j = hypercolumn subscript of the submatrix being specified;
 $v_{11} \dots v_{mm}$ = stiffness values for the submatrix being specified.

The number of hypercolumns (or hyperrows) in the element stiffness/mass matrix will be equal to the number of nodes, n. The number of columns (or rows) within each submatrix is equal to the number of degrees of freedom per joint, m. For example, a triangular plane stress element having three nodes will have three hypercolumns (and hyperrows), while the number of columns (or rows) within each submatrix will equal two (corresponding to the u, v nodal degrees of freedom at each joint).

Again, the stiffness/mass matrix is specified in the planar element coordinate system for planar elements, and in the global coordinate system for tridimensional elements. Units conversion is not performed for stiffness matrix components. They must be given in metric units of meters, and newtons.

5. Special DUMMY Element Specification

When describing ELEMENT PROPERTIES, it is necessary to specify an element "type name" (e.g., CSTG, IPLQ, etc.) even if the stiffness/mass matrix is input directly. This "type name" is used to identify the program name to be called within certain element procedures (e.g., stiffness/mass matrix generation, loads generation, stress backsubstitution, stress/strain output, etc.).

For this reason a special type name, DUMMY, has been added to allow element information such as stiffness/mass matrices to be input without specifying a unique element "type name". Load generation, stress backsubstitution, and output are bypassed for this special element, since it lacks the necessary information to compute and store them.

The specification of degrees of freedom for this DUMMY element is given in the ELEMENT PROPERTIES command as:

$$\underline{\text{DOF}} \quad \{ \underline{\text{DX}} \quad \underline{\text{DY}} \quad \underline{\text{DZ}} \quad \underline{\text{RX}} \quad \underline{\text{RY}} \quad \underline{\text{RZ}} \}$$

These nodal degrees of freedom are global degrees of freedom, and must be the same at all nodes.

6. Element Similarities Specification

When an element is similar to another, it is possible to take advantage of element similarities to reduce the computation time. For this purpose, the two elements need not be identical. Differences may exist in applied loads, boundary node support conditions, and geometric orientation. This similarity is specified with the ELEMENT PROPERTIES command as:

$$\underline{\text{SAME (AS)}} \quad \left\{ \begin{array}{c} i_3 \\ 'a_3' \end{array} \right\} \left(\begin{array}{c} \underline{\text{ROTATED}} \\ \left\{ \begin{array}{cccc} \underline{\text{PLANAR}} & [\underline{\text{TH4}}] & V_4 & \\ & [\underline{\text{TH1}}] & V_1 & [\underline{\text{TH2}}] & V_2 & [\underline{\text{TH3}}] & V_3 \end{array} \right\} \end{array} \right) -$$

(BUT LOADINGS)

where:

$i_3, 'a_3'$ = element name

V_1, V_2, V_3 = values of the global rotation angles $\theta_1, \theta_2, \theta_3$

V_4 = value of the planar rotation angle θ_4

When an element is identical to another, only the identifier for the second element needs to be given in the similarity specification. If, however, their geometric orientation with respect to the global (or planar) coordinate system is different, it is necessary to specify rotation angles. The angles

TH1, TH2, and TH3 indicate how the base element is to be rotated in the global reference frame so that it coincides with the similar element (refer to Figure 2.2).

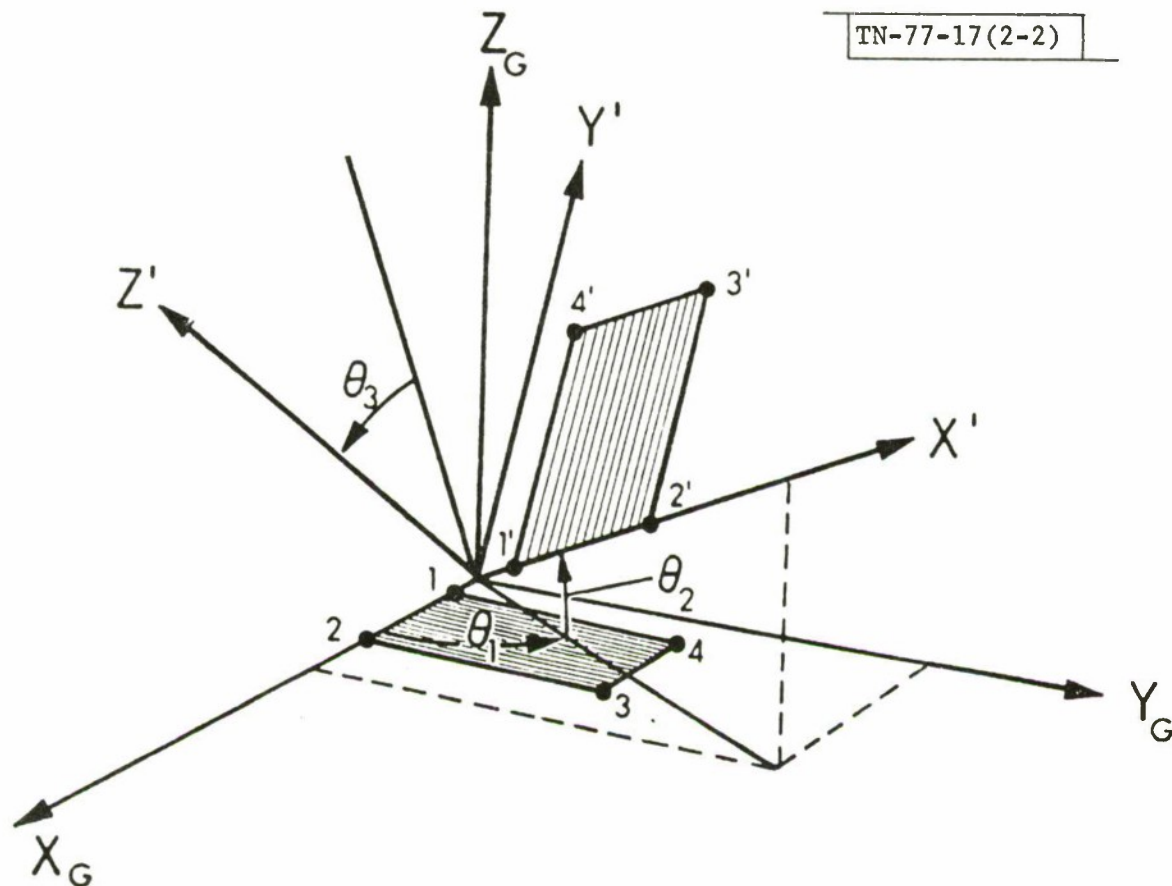


Figure 2.2. Rotation angles for element similarities.

Note that if planar elements are to be rotated, the three-axis rotation procedure is extremely cumbersome. Since the element planar coordinate systems are known for both elements, it is possible to use the angle TH4 to indicate how the planar base element is to be rotated in the planar reference frame so that it coincides with the similar planar element.

When the element loads applied to the similar element differ from those for the base element, it is necessary to specify the option BUT LOADINGS. However, this is not necessary if the only difference in loadings involves applied joint loads or displacements.

7. Examples of ELEMENT PROPERTIES Specification

(1) Element type and thickness

ELEMENT PROPERTIES

1 TO 15 TYPE 'LSR' THICK 0.5

(2) DUMMY Element Type and Stiffness/Mass Matrix

ELEMENT PROPERTIES

1 TO 10 TYPE 'DUMMY' DOF DX DZ -

STIFFNESS MATRIX NODES 2 NDF 2

SUBMATRIX 1 1 DIAG 87575. 98700.

SUB 1 2 51500. 47000.

35000. 63735.

SUB 2 2 DIAG 75000. 841000.

MASS MATRIX NODES 2 NDF 2

SUB 1 1 DIAG .015 .015

SUB 2 2 DIAG .015 .015

(3) Element type, thickness, thermal expansion coefficients, and rigidity matrix.

ELEMENT PROPERTIES

1001 TO 1016 TYPE 'CSTG' THICK 0.1 -

CTE 10.33E-6 10.67E-6 CSXY 0.94E-6

RIGIDITY MATRIX NC 3

ROW 1 0.15156D+07 0.43313D+06 0.46849D+05

ROW 2 0.43313D+06 0.15891D+07 0.57063D+05

ROW 3 0.46849D+05 0.57063D+05 0.57226D+06

(4) Element type and similarities

ELEMENT PROPERTIES

1 TYPE 'PSR' THICK 1.
2 TO 60 SAME AS 1
61 TO 120 SAME AS 1 ROTATED TH4 90.

III. FINITE ELEMENT LOADING SPECIFICATION

1. Edge Forces

Element loads acting on the exterior boundaries of plane stress/strain finite elements or on the edges of tridimensional finite elements must be converted into consistent joint loads. These consistent joint loads are defined as "work equivalent" forces:

$$\sum_{i=1}^{\text{nodes}} P_i \cdot \delta u_i = \int_S \bar{p} \cdot \delta u \, dS$$

where u_i , P_i are the nodal displacements and equivalent joint loads, respectively. As can be seen by the right-hand side of the above equation, these equivalent joint loads depend on the assumed element displacement expansion, u , as well as on the load intensities, $\bar{p}$.

EDGE FORCES may be specified under the ELEMENT LOADS command as:

$$\begin{aligned} \text{list} \quad \underline{\text{EDGE FORCES}} \quad \underline{\text{EDGE}} \quad i_1 & \left\{ \begin{array}{c} \underline{\text{LOCAL}} \\ \underline{\text{PLANAR}} \\ \underline{\text{GLOBAL}} \end{array} \right\} \left\{ \begin{array}{c} * \\ \left(\begin{array}{cc} \underline{\text{LX}} & v_1 \\ \underline{\text{LY}} & v_2 \\ \underline{\text{LZ}} & v_3 \end{array} \right) \\ \text{(variable)} \end{array} \right\} \\ \\ \text{(variable)} = \underline{\text{VARIABLE}} & \quad * \left\{ \begin{array}{c} \underline{\text{VX}} \, x_1 \, x_2 \, \dots \, x_n \\ \underline{\text{VY}} \, y_1 \, y_2 \, \dots \, y_n \\ \underline{\text{VZ}} \, z_1 \, z_2 \, \dots \, z_n \end{array} \right\} \end{aligned}$$

where: list = list of elements for which edge forces are applied;
 i_1 = edge number where load is being applied;
 v_1, v_2, v_3 = value of the load components for uniform loading;
 x_j, y_j, z_j = value of X, Y, Z load components at node j for variable loading.

(Units of applied load are force/unit length)

The edge numbers for planar elements are defined by the nodal order in the ELEMENT INCIDENCES command. The first pair of corner nodes define EDGE 1, the second pair EDGE 2, etc., until all edges have been defined.

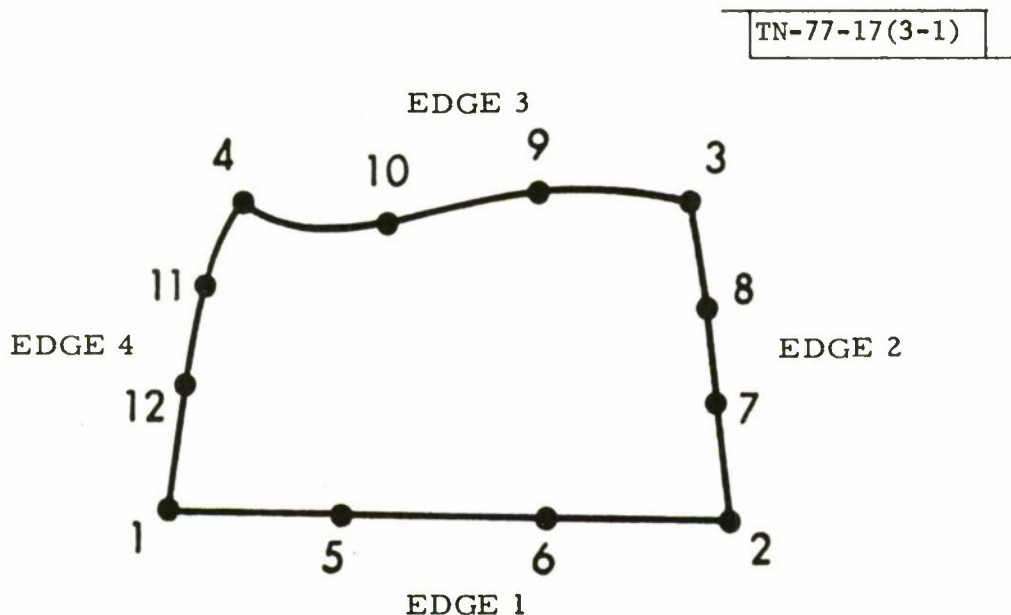
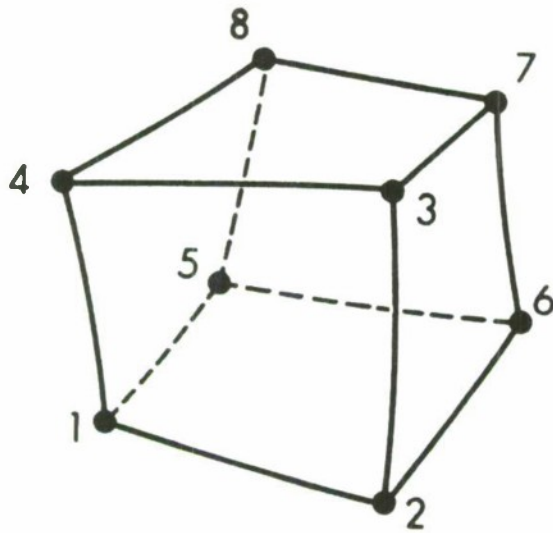


Figure 3.1. Edge numbering for planar elements.

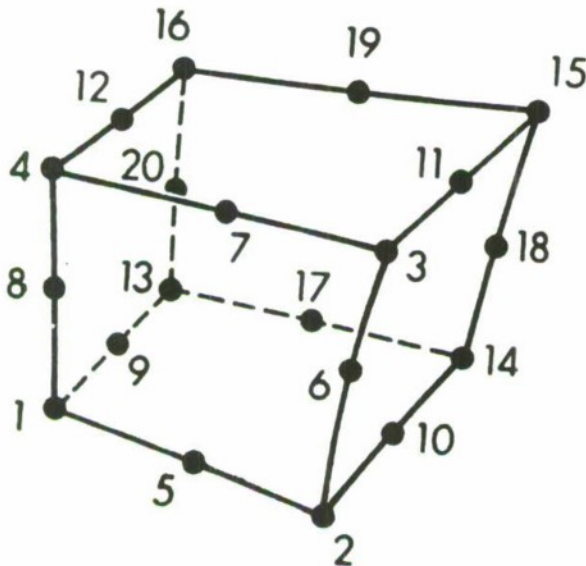
If the VARIABLE loading specification is used, the nodal ordering on the edge proceeds from corner node to corner node in the same direction as the ELEMENT INCIDENCE ordering (e.g., nodes 3, 9, 10, 4 define edge 3 in Figure 3.1).

The edge numbers for tridimensional elements are also defined by the element nodal listing in the ELEMENT INCIDENCES command. The first numbered face (front face) defines the first group of edge numbers, the edges connecting the front face and back face define the second group of edge numbers, and the back face edges are defined last. Edge numbering for the IPLS, IPQS, and TRIP tridimensional elements are illustrated in Figures 3.2-3.4, respectively.

EdgeNode ordering

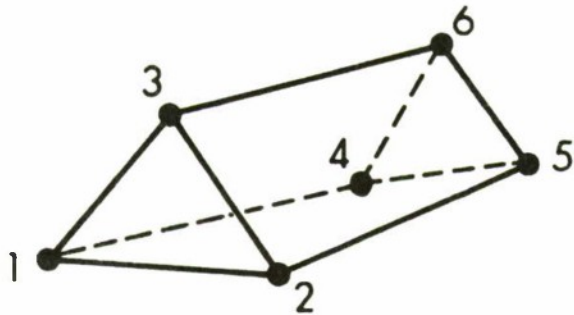
1	1	2
2	2	3
3	3	4
4	4	1
5	1	5
6	2	6
7	3	7
8	4	8
9	5	6
10	6	7
11	7	8
12	8	5

Figure 3.2. Edge numbering for IPLS element.

EdgeNode ordering

1	1	5	2
2	2	6	3
3	3	7	4
4	4	8	1
5	1	9	13
6	2	10	14
7	3	11	15
8	4	12	16
9	13	17	14
10	14	18	15
11	15	19	16
12	16	20	13

Figure 3.3. Edge numbering for IPQS element.



Edge	Node ordering	
1	1	2
2	2	3
3	3	1
4	1	4
5	2	5
6	3	6
7	4	5
8	5	6
9	6	4

Figure 3.4. Edge numbering for TRIP element.

EDGE FORCES acting on tridimensional elements must be specified as GLOBAL loads. If EDGE FORCES acting on planar elements are specified as GLOBAL, the loads will be transformed into the PLANAR element coordinate system for consistent load calculations.

For planar stress/strain elements a LOCAL coordinate system is established at each node in the following manner (refer to Fig. 3.5):

- Z_L is determined by nodal ordering of the ELEMENT INCIDENCES command.
- X_L is the normal to the edge at a joint (positive in the outward-pointing direction).
- Y_L is the tangent to the edge at a joint (positive direction being determined from right-hand rule on Z_L and X_L).

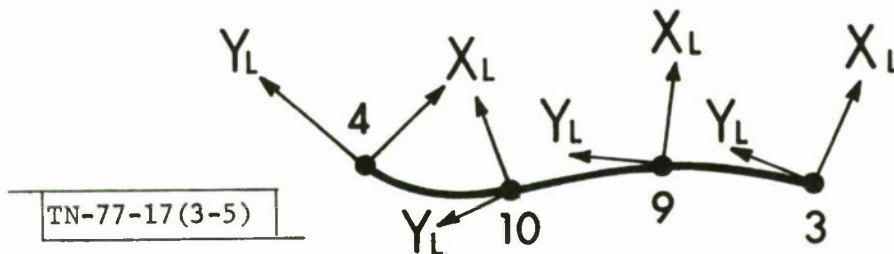


Figure 3.5. Local coordinate system for edge loads.

2. Surface Forces

Element loads acting on the surfaces of planar or tridimensional elements must also be converted to consistent joint loads (refer to Edge Forces for description of this conversion).

SURFACE FORCES are specified under the ELEMENT LOADS command as:

$$\text{list} \quad \text{SURFACE FORCES (FACE } i_1) \quad \left\{ \begin{array}{l} \rightarrow \text{LOCAL} \\ \text{PLANAR} \\ \text{GLOBAL} \end{array} \right\}^* \left\{ \begin{array}{l} \text{PX} \quad v_1 \\ \text{PY} \quad v_2 \\ \text{PZ} \quad v_3 \end{array} \right\} \quad \text{(variable)}$$

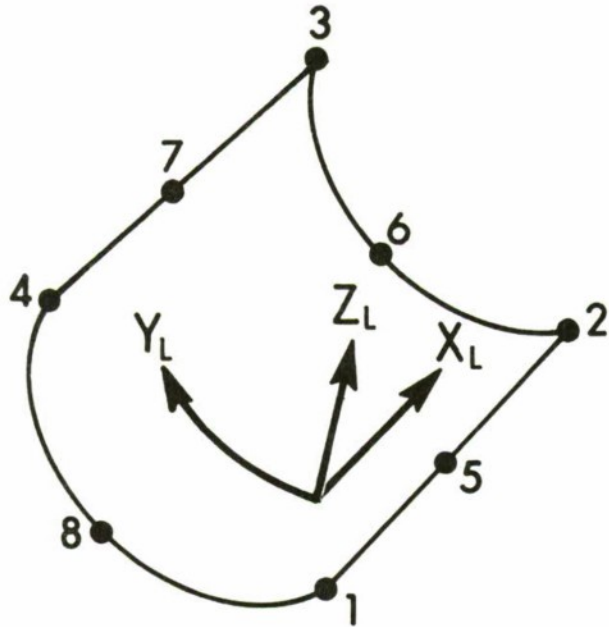
where: list = list of elements for which surface forces are applied;
 i_1 = face number of tridimensional element where load is applied;
 v_1, v_2, v_3 = value of the load components for uniform loading;
(variable) = refer to Edge Forces description.

(Units of applied load are force/unit area).

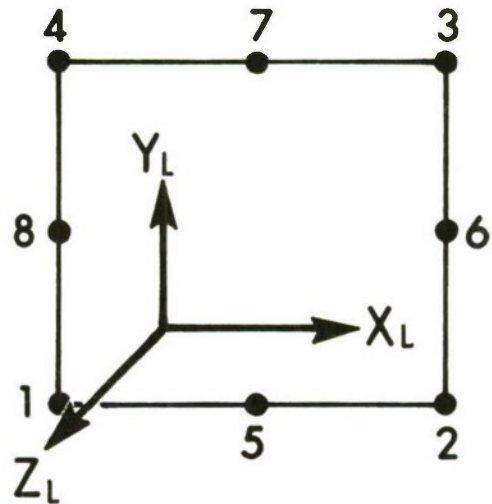
SURFACE FORCES can be specified as either PLANAR or GLOBAL for planar finite elements. The LOCAL option is not allowed. If SURFACE FORCES acting on planar elements are specified as GLOBAL, the loads will be transformed into the planar element coordinate system for consistent load calculations. The FACE i specification must not be given for planar elements, since it is clear where the load is to be applied. The variable option is not available for plate bending elements.

SURFACE FORCES can be specified as either LOCAL or GLOBAL for tridimensional elements. The LOCAL coordinate system refers to the transformed curvilinear coordinates (refer to Figure 3.6).

- (a) Z_L is the outward-pointing normal to the surface.
- (b) X_L is parallel to edges 1 and 3 of the transformed coordinate axes.
- (c) Y_L is parallel to edges 2 and 4 of the transformed coordinate axes.



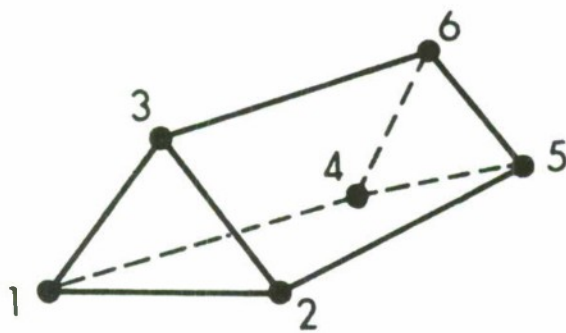
CURVILINEAR COORDINATES



TRANSFORMED LOCAL COORDINATES

Figure 3.6. Local coordinate systems for surface loads.

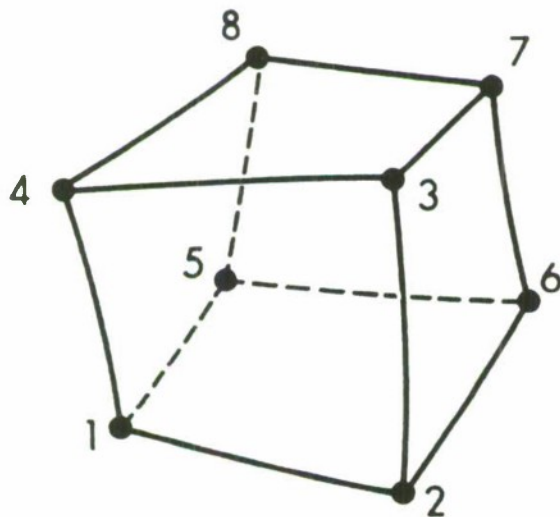
The face numbers for tridimensional elements are defined by the nodal order in the ELEMENT INCIDENCES command. The first numbered face is always FACE 1, and the back face is always FACE 2. The connecting faces are then numbered in the same direction as the node numbering (FACE 3 $\rightarrow$ FACE N). The nodal ordering for each face is started at the first node encountered for that face in the nodal list. The remaining nodes on the face are ordered by proceeding around the face in a direction such that an outward-pointing normal is produced by applying the right-hand rule to the node ordering.



Face	Node Ordering
1	1 2 3
2	4 6 5
3	1 4 5 2
4	2 5 6 3
5	1 3 6 4

TN-77-17(3-7)

Figure 3.7. Face numbering for TRIP element.



Face	Node Ordering
1	1 2 3 4
2	5 8 7 6
3	1 5 6 2
4	2 6 7 3
5	3 7 8 4
6	1 4 8 5

TN-77-17(3-8)

Figure 3.8. Face numbering for IPLS element.

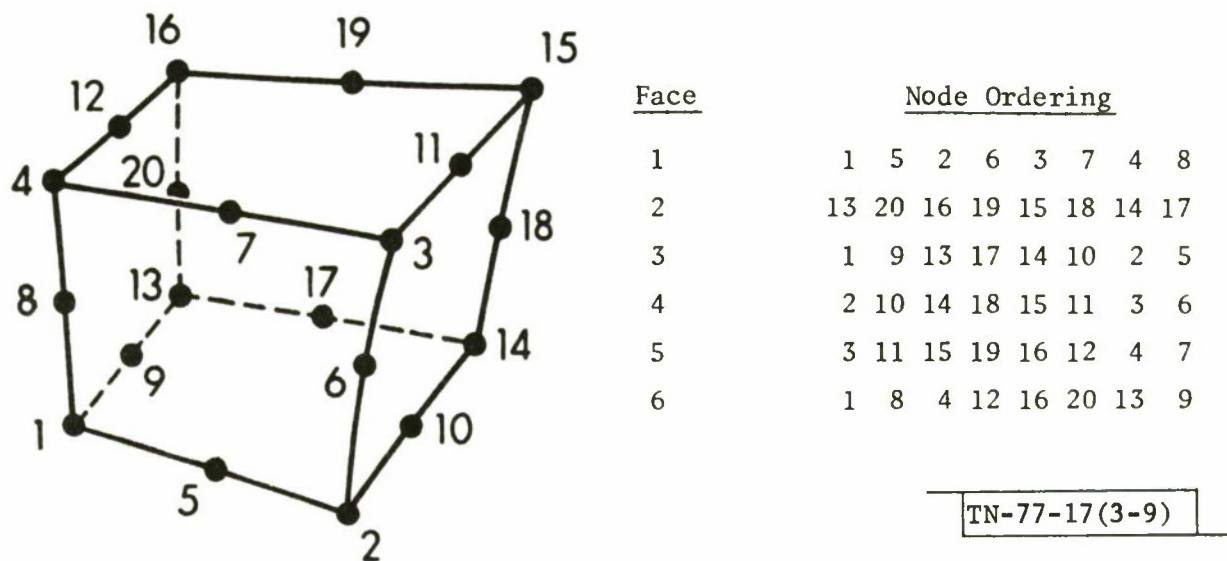


Figure 3.9. Face numbering for IPQS element.

3. Body Forces

Forces, such as dead load, often act on a finite element throughout the entire body of the element. These body forces must be converted into consistent joint loads (as detailed under Edge Forces) for each finite element.

BODY FORCES may be specified under the ELEMENT LOADS command as:

$$\text{list} \quad \underline{\text{BODY}} \quad \underline{\text{FORCES}} \quad \left\{ \begin{array}{l} \rightarrow \underline{\text{PLANAR}} \\ \underline{\text{GLOBAL}} \end{array} \right\} \left(* \begin{array}{l} \left(\begin{array}{l} \underline{\text{BX}} \quad v_1 \\ \underline{\text{BY}} \quad v_2 \\ \underline{\text{BZ}} \quad v_3 \end{array} \right) \\ \text{(variable)} \end{array} \right)$$

where: list = list of elements for which body forces are prescribed;
 v_1, v_2, v_3 = value of the load components for uniform loading;
(variable) = refer to Edge Forces description.

(Units of applied load are force/unit volume).

BODY FORCES must be specified only as GLOBAL for tridimensional elements. However, they may be specified as either PLANAR or GLOBAL for planar finite elements. If BODY FORCES for planar finite elements are specified as GLOBAL, the loads will be transformed into the planar element coordinate system for consistent load calculation.

4. Thermal Forces

Temperature changes and thermal gradients acting on finite elements also require conversion to equivalent consistent joint loads (as detailed under Edge Forces) for each finite element.

Thermal forces may be specified under either the ELEMENT or JOINT TEMPERATURE commands as:

$$\text{list} \quad \left\{ \begin{array}{l} \underline{\text{CHANGE}} \quad v_1 \\ \underline{\text{GRADIENT}} \end{array} \right. * \left\{ \begin{array}{l} \underline{X} \quad v_2 \\ \underline{Y} \quad v_3 \\ \underline{Z} \quad v_4 \end{array} \right\} \left\{ \begin{array}{l} \rightarrow \underline{\text{LOCAL}} \\ \underline{\text{GLOBAL}} \end{array} \right\}$$

where: list = a list of joints/elements at which the thermal forces are applied;
 v_1 = value of temperature change, T;
 v_2, v_3, v_4 = value of X, Y, Z thermal gradients (i.e., $\partial T/\partial X$, $\partial T/\partial Y$, and $\partial T/\partial Z$).

Coefficients of thermal expansion must be given for all elements connected to joints which are subjected to JOINT TEMPERATURES and for all elements subjected to ELEMENT TEMPERATURES.

When only a coarse description of the thermal distribution is known, it is often simplest to use the ELEMENT TEMPERATURE form of the command. On the other hand, if variations in temperature are well defined and rapidly changing across an element, it would be better to use the JOINT TEMPERATURE form of the

command. In this latter case, equivalent nodal forces due to temperature are generated by assuming the temperature expansion coincides with the displacement expansion.

Plane stress/strain elements and tridimensional elements use only the temperature CHANGE to generate thermal forces, while plate bending elements use only temperature GRADIENT information. Only the plate elements make use of both temperature CHANGE and temperature GRADIENT data.

Temperature GRADIENTS can be either LOCAL or GLOBAL. However, if LOCAL is specified, the only applicable load component is Z which defines a gradient normal to the surface. If GLOBAL is given, the component of the gradient vector normal to the plate will be calculated and used. The in-place components will be discarded.

5. Initial Strains

If any initial strains are present in a finite element from such sources as imperfect fit, etc., then they must be accounted for in the model. The elastic strain (i.e., the only one that produces stress) can be expressed as $e = \epsilon - \epsilon_T - \epsilon_I$, where ϵ is the total strain, ϵ_T is the thermal strain, and ϵ_I is the initial strain.

Since the deformed shape of the structure under total loading (external loads, thermal strain, initial strain) is desired, equivalent consistent joint loads are generated for the thermal strain and initial strain effects. The final elastic strains (those computed by STRUDL) are then obtained by subtracting the input values of thermal strain and initial strain from the total strains computed from the displacements.

These initial strains may be specified with the ELEMENT INITIAL STRAINS command as:

$$\text{list} \quad \left\{ \begin{array}{llllll} \underline{\text{AXIAL}} & [\underline{\text{X}}] & v_1 & [\underline{\text{Y}}] & v_2 & [\underline{\text{Z}}] & v_3 \\ \underline{\text{SHEAR}} & [\underline{\text{XY}}] & v_4 & [\underline{\text{XZ}}] & v_5 & [\underline{\text{YZ}}] & v_6 \end{array} \right\}$$

where:

list = a list of elements for which initial strains are prescribed;

v_1, v_2, v_3 = values of initial axial strains;

v_4, v_5, v_6 = values of initial shear strains.

The initial strain components must be given in the GLOBAL reference frame for tridimensional elements and in the PLANAR reference frame for planar elements. The strain curvatures, rather than strains, are specified in the case of plate bending elements. Therefore, initial strains must not be specified for plate elements, as inconsistencies will result. (Plate elements may be remodeled as overlaid plane stress and plate bending elements if initial strains are present.)

IV. FINITE ELEMENT MODELING CONSIDERATIONS

1. Convergence and Accuracy of Model

One objective of the analyst when using finite elements to model a structure is to generate results sufficiently accurate to satisfy his requirements. To have confidence in his results, he must know their level of accuracy (i.e., establish a "percentage of error" from the "true answers"). He also wants his results to converge to the "true solution" as the finite element mesh is made finer.

Two basic criteria must be satisfied to assure convergence.

- (1) Any required state of constant strain (including rigid body movement) can be reproduced on an element;
- (2) Displacements must be continuous between adjacent elements.

If these criteria are met, the solution represents an upper-bound on the total potential energy. The potential energy can be made to converge monotonically if successively finer modeling retains the nodes of the previous models. If the second criterion is unsatisfied, these non-compatible elements will still converge if the first criterion above is satisfied. However, a monotonic convergence cannot be guaranteed, and bounds on the potential energy are not established. Non-compatibility is often associated with plate bending elements (e.g., CPT and SBCT). However, these non-compatible elements have often been shown to exhibit better convergence characteristics than compatible elements which use complex displacement expansions.

In generating the finer mesh, nodes may be added, but none may be removed or eliminated. In addition, the displacement expansions of the elements must not be a function of the element size or orientation.

Since convergence can be assured for all finite element models, one simple method of assessing accuracy is available. The problem can be analyzed with successively finer meshes, whose successive results will approach the

true solution. The results of successive refinements as a function of the number of degrees of freedom allow the true solution to be approximated by extrapolation. This process of successive refinements can be terminated whenever the analyst feels he has achieved sufficient accuracy.

It must be remembered that the degree of refinement required for a particular analysis is a function of both the geometry of the structure being analyzed and its loading environment. Finer meshes, or higher-order elements, are generally required in areas of high stress gradients. High stress gradients are indicated by large stress differences in adjacent elements. If structural loadings do not generate any such high stress gradients, a finer grid will not be required. Similarly, if a curved boundary is being modeled, elements with straight edges can never model the problem exactly, but several higher-order elements may give excellent results with significantly less expense.

Convergence and accuracy studies should be standard procedure for any finite element analysis in order that the analyst can estimate his errors and account for them.

2. Interpretation of Results

To obtain the results of a finite element analysis, it is necessary to use the LIST command. Available for output are the nodal displacements and element stresses and strains. Reactions and loads (statics check results) are not available in problems containing finite elements.

Since the stresses in a finite element problem are a function of derivatives of the displacements, the displacement results will converge more quickly than the stresses. Therefore, acceptable displacement results are often obtained for a mesh which is significantly coarser than that required to obtain acceptable stress results. One technique which enables the analyst to improve his stress results without resorting to re-analysis using a finer discretization is stress averaging. The basic assumption is that if the displacements are accurate, the stresses, on the average, should also be good.

Several variations of the stress averaging technique exist, all of which result in a smoothing of the stress pattern over the structure:

(1) Nodal Stress Averaging

The analyst may be interested in a set of stress values at the nodes. If this is the case, the stresses from all of the elements incident on that node may be averaged (the sum of the stresses in a given direction divided by the number of incident elements). If the elements have stress values at the nodes, those stresses may be used. If the elements have constant stresses, those constant values may be used. In either case, the result is a smoothing of the set of stress values at particular points in the structure.

(2) Element Stress Averaging

Alternatively, the analyst may be interested in a good approximation of the stresses in the elements. If this is the case, the stresses from each of the nodes may be averaged. If the elements are constant stress elements (i.e., there are no nodal stresses), the constant stress value may be used.

It must be noted that this sort of averaging technique may not be valid in stress fields where the smoothing would tend to mask areas of high stress gradients.

In summary, the intent of this section has been to point out the fact that while displacement results may be sufficiently accurate for a particular problem, it may be appropriate to interpret the resulting stresses somewhat more loosely, due to the inaccuracies which may have been introduced by the original finite element modeling assumptions.

The output currently available for the STRUDL finite elements is listed in Table 4.1. In addition, each finite element is also further described in the Appendix.

TABLE 4.1
OUTPUT SUMMARY FOR FINITE ELEMENTS

PROBLEM TYPE	NODAL DISPLACEMENTS	STRAINS	STRESSES	ELEMENT	LOCATION OF STRESS-STRAIN OUTPUT
Plane Stress				CSTG PSR PSRCSH	Centroid
and	$\begin{Bmatrix} u \\ v \end{Bmatrix}$	$\begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix}$	$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix}$	LST LSR IPQQ IPCQ	Nodes
Plane Strain				IPLQ IPLQCSH	Nodes*
Plate				CPT	Centroid
Bending	$\begin{Bmatrix} w \\ \theta_x \\ \theta_y \end{Bmatrix}$	$\begin{Bmatrix} \epsilon_x \\ \epsilon_{y1} \\ \gamma_{xy} \end{Bmatrix}$	$\begin{matrix} V_{xx}, V_{yy} \\ M_{xx}, M_{yy}, M_{xy} \end{matrix}$	BPR PBQ1	Nodes
Plate	$u, v, w, \theta_x, \theta_y$	$\begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix}$	$\begin{matrix} \sigma_x, \sigma_y, \tau_{xy} \\ V_{xx}, V_{yy} \\ M_{xx}, M_{yy}, M_{xy} \end{matrix}$	SBCT	Centroid
					Nodes
Tridimensional	$\begin{Bmatrix} u \\ v \\ w \end{Bmatrix}$	$\begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix}$	$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{xz} \\ \tau_{yz} \end{Bmatrix}$	TRIP IPLS IPLSCSH	Nodes*
				IPQS	Nodes

* Location of output for these elements can be requested at edges or faces by specifying DUMP ON 16384 previous to an ANALYSIS command.

V. EXAMPLES

1. Plane Stress Example

A solid cantilever beam (see Figure 5.1) will be modeled using plane stress finite elements. The cross-section is rectangular, 12 inches in depth, 6 inches in thickness, and 120 inches in length. A concentrated end load ($P = 10$ kips) and a uniform loading ($w = 0.8$ kips/ft) are to be investigated. Tip deflection of the beam is desired.

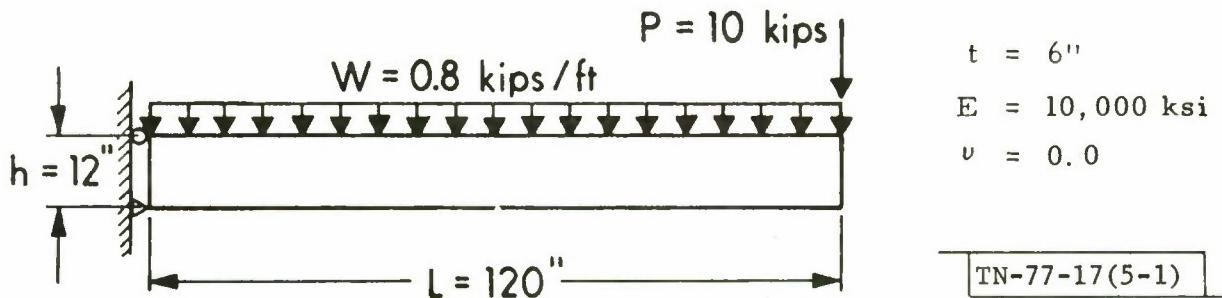


Figure 5.1. Solid cantilever beam.

Since the tip deflection is the desired output (rather than stresses), it is possible to use the CSH elements and reduce the number of linear displacement finite elements (PSR, IPLQ) required for this model. It should also be noted that it is possible to use only one quadratic displacement finite element (e.g., LSR, IPQQ) since the theoretical displacement function for this example happens to be quadratic.

IPLQ and IPLQCSH elements were selected for the model. They also were used to illustrate how edge loads ($w = 0.8$ kips/ft) can be applied to the elements. A typical 2-element model is shown in Figure 5.2.

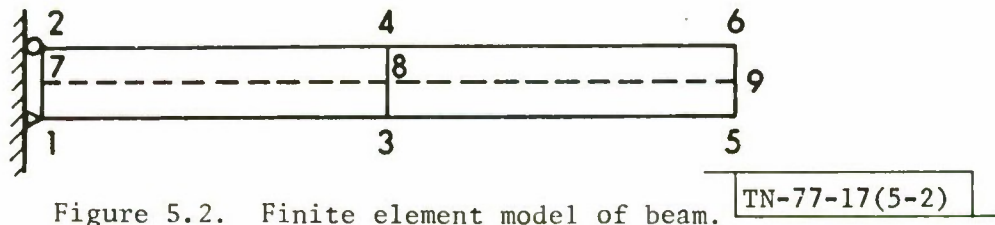


Figure 5.2. Finite element model of beam.

It should be noted that adding joints 7, 8, 9 in Figure 5.2 does not improve the tip displacements at all. Even though the number of finite elements is doubled, the element aspect ratio has also been doubled, and no improvement in tip deflection is obtained. This illustrates the importance of proper discretization for proving convergence of a finite element model.

A typical input deck for this example is listed in Figure 5.3, while the results are illustrated in Figure 5.4. Two cases were considered: (1) a long beam ($h/L = 0.1$) in which bending dominates; and (2) a short beam ($h/L = 0.5$) in which shear deformation is important.

It should be noted that the CSH elements provide almost exact results for the loading cases illustrated in this example.

```

STRU DL 'IPLQCSH' 'PLANE STRESS EXAMPLE'
JOINT COORDINATES
1 0. 0. SUPPORT
2 0. 12. SUPPORT
3 60. 0.
4 60. 12.
5 120. 0.
6 120. 12.
JOINT RELEASES
2 FORCE Y
TYPE PLANE STRESS
ELEMENT INCIDENCES
1 1 3 4 2
2 3 5 6 4
ELEMENT PROPERTIES
1 2 TYPE 'IPLQCSH' THICK 6.
CONSTANTS
E 10.E6 ALL
POISSON 0. ALL
UNITS KIPS FEET
LOADING 1
JOINT LOADS
6 FORCE Y -10.
LOADING 2
ELEMENT LOADS
1 2 EDGE FORCES EDGE 3 LOCAL UNIFORM LY -.8
DUMP TIME
STIFFNESS ANALYSIS
UNITS INCHES
LIST DISPLACEMENTS JOINT 5
FINISH

```

TN-77-17(5-3)

Figure 5.3. Input listing of STRUDL model of beam.

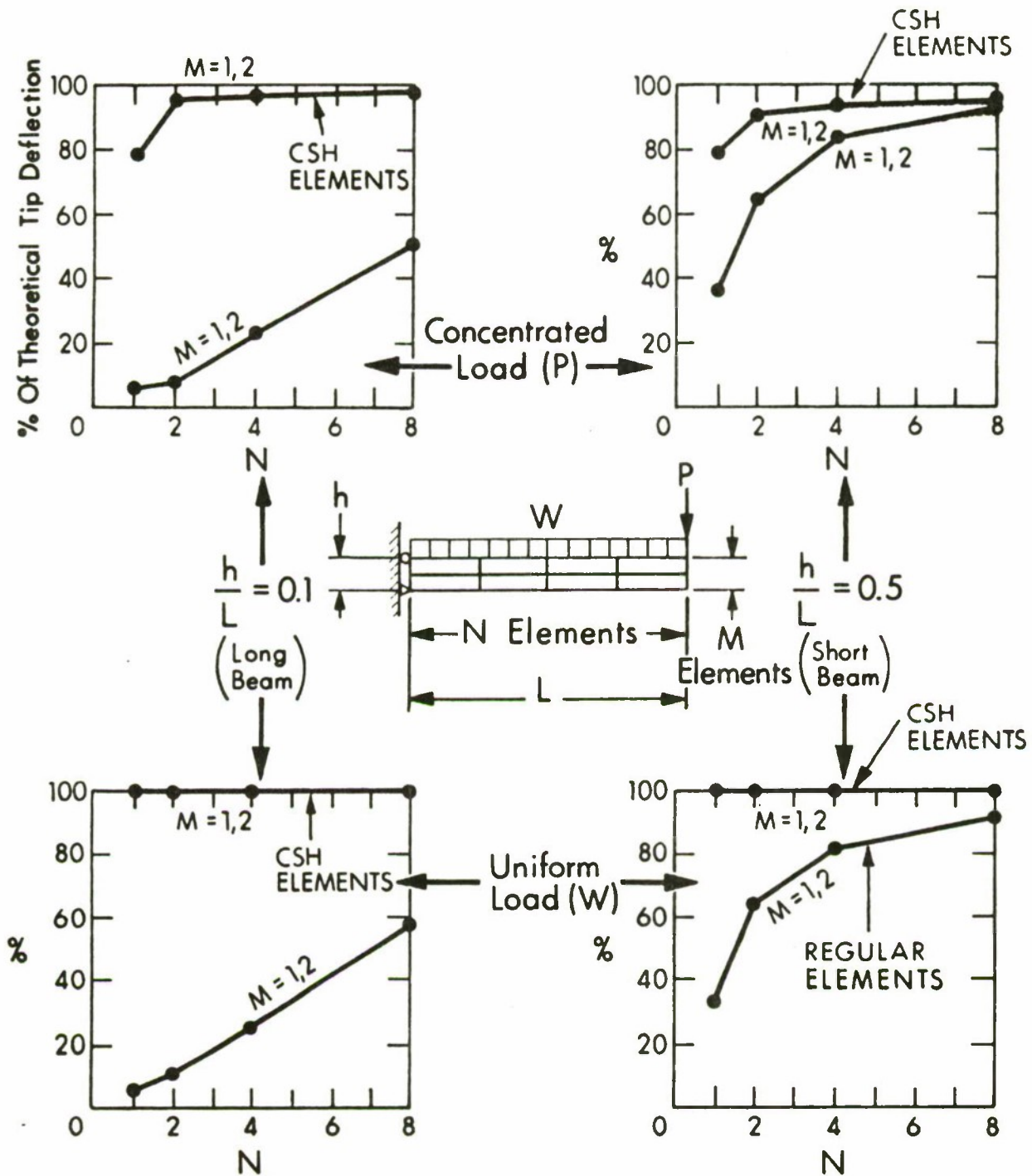


Figure 5.4. Results of solid cantilever example.

2. Fracture Mechanics Example

The quadratic isoparametric elements (IPQQ, IPQS) existing in STRUDL have been shown^[1] to embody $1/\sqrt{r}$ singularity for calculating stress intensity factors of elastic fracture mechanics. The singularity is obtained by placing those mid-side nodes near the crack tip at the quarter points.

The STRUDL input file need not be changed to accommodate fracture analysis, except that the coordinates of the quarter-side nodes must be specified. The distance from the quarter-side nodes to the crack tip should probably be between 2 - 3% of the crack length. Elastic fracture mechanics problems that can be treated are Mode I (crack opening), Mode II (crack sliding), or combined Modes I and II (refer to Figure 5.5).

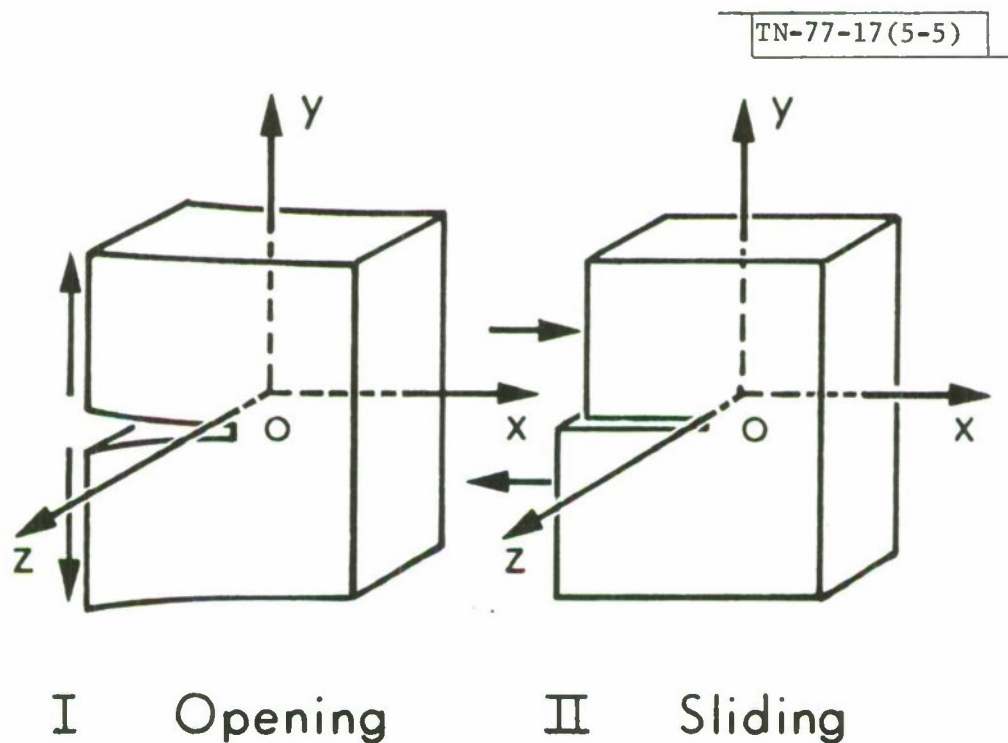


Figure 5.5. Elastic fracture mechanics modes.

Since STRUDEL permits thermal loading, the effects of cracks on structures subjected to various thermal environments can also be studied.

In order to compute the stress intensity factors K_I and K_{II} , the displacements at the quarter-side nodes must be found. The following equations^[2] may then be used to evaluate them.

$$\begin{aligned}
 u = & u_0 + \frac{K_I}{4G} \sqrt{\frac{r}{2\pi}} \left\{ \cos\alpha \left[(2\kappa - 1) \cos \frac{\theta}{2} - \cos \frac{3\theta}{2} \right] \right. \\
 & - \sin\alpha \left[(2\kappa + 1) \sin \frac{\theta}{2} - \sin \frac{3\theta}{2} \right] \left. \right\} + \frac{K_{II}}{4G} \sqrt{\frac{r}{2\pi}} \left\{ \cos\alpha \left[(2\kappa + 3) \sin \frac{\theta}{2} + \sin \frac{3\theta}{2} \right] \right. \\
 & + \sin\alpha \left[(2\kappa - 3) \cos \frac{\theta}{2} + \cos \frac{3\theta}{2} \right] \left. \right\}
 \end{aligned}$$

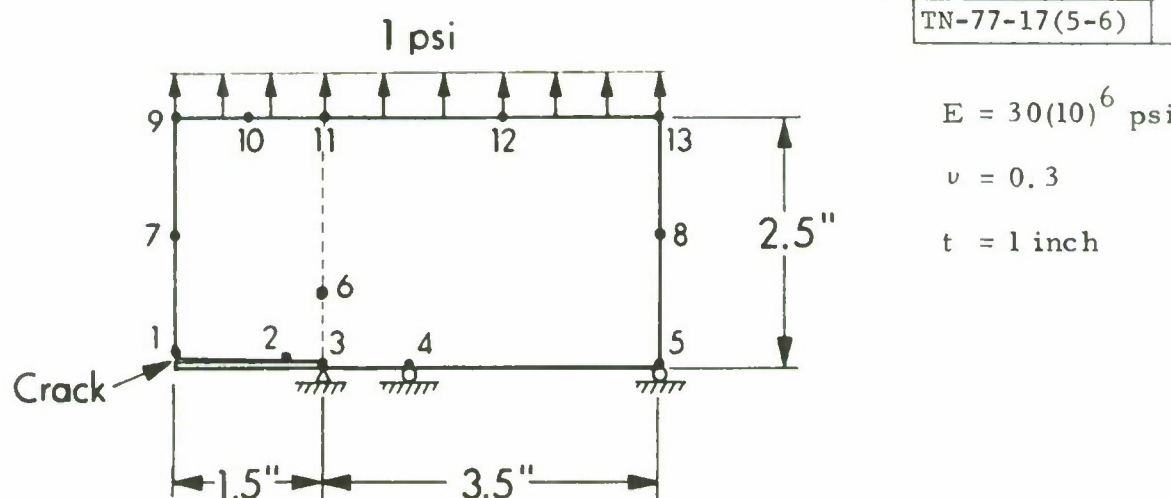
$$\begin{aligned}
 v = & v_0 + \frac{K_I}{4G} \sqrt{\frac{r}{2\pi}} \left\{ \sin\alpha \left[(2\kappa - 1) \cos \frac{\theta}{2} - \cos \frac{3\theta}{2} \right] \right. \\
 & + \cos\alpha \left[(2\kappa + 1) \sin \frac{\theta}{2} - \sin \frac{3\theta}{2} \right] \left. \right\} + \frac{K_{II}}{4G} \sqrt{\frac{r}{2\pi}} \left\{ \sin\alpha \left[(2\kappa + 3) \sin \frac{\theta}{2} + \sin \frac{3\theta}{2} \right] \right. \\
 & - \cos\alpha \left[(2\kappa - 3) \cos \frac{\theta}{2} + \cos \frac{3\theta}{2} \right] \left. \right\}
 \end{aligned}$$

where u_0 and v_0 are the unknown displacements of the crack tip, u and v are the displacements at the quarter-side nodes, G is the shear modulus, α is the angle of the crack as measured positive counterclockwise from the x axis, and r and θ are the crack tip centered polar coordinates of the quarter-side nodes. The quantity κ is dependent on the problem type:

$$\kappa = \begin{cases} 3 - 4\nu & \text{(plane strain or axisymmetric)} \\ (3 - \nu)/(1 + \nu) & \text{(plane stress)} \end{cases}$$

where ν is Poisson's ratio. It should be noted that this displacement assumption, when properly differentiated to yield strains and stresses, yields the correct singularity for stresses at the crack tip, i.e., $\sigma_{ij} = \sigma_{ij}(r^{-1/2}, \theta)$ which tends to infinity as r tends to zero.

Consider a steel compact tensile specimen, 5 inches square with a 1.5-inch crack and a 1 psi tensile loading on the top surface. A 2-element model of the specimen is shown in Figure 5.6.



Note: Joints 2, 4, and 6 are quarter-side nodes.

Figure 5.6. Finite element model of fracture specimen.

The singularity of strains at the crack tip is evidenced by the Jacobian of transformation from curvilinear space to Cartesian space. This singularity is bypassed during strain calculations and a warning is printed indicating the node at which the singularity occurs (i.e., this should be the node at the crack tip). For example, for the above problem

```

**** STRUDL WARNING 4.35 - JACOBIAN VANISHED FOR ISOPARAMETRIC ELEMENT 1
      AT INTEGRATION POINT      2
**** STRUDL WARNING 4.35 - JACOBIAN VANISHED FOR ISOPARAMETRIC ELEMENT 2
      AT INTEGRATION POINT      1

```

where node 3 is the integration point noted in both warning messages.

In order to achieve enough accuracy in the output displacements, it is necessary to apply a larger (e.g., 1000 psi) tensile stress on the top surface and also request 9 digits of output data by the OUTPUT DECIMAL 9 command. The input listing of the data deck for this example is listed in Figure 5.7.

```

STRUDL 'IPQQ' 'FRACTURE MECHANICS EXAMPLE'
JOINT COORDINATES
1 0. 0.
2 1.125 0.
3 1.5 0. SUPPORT
4 2.375 0. SUPPORT
5 5. 0. SUPPORT
6 1.5 .625
7
8
9 0. 2.5
10
11 1.5 2.5
12
13 5. 2.5
JOINT RELEASES
4 5 FORCE X
TYPE PLANE STRAIN
ELEMENT INCIDENCES
1 1 3 11 9 2 6 10 7
2 3 5 13 11 4 8 12 6
ELEMENT PROPERTIES
1 2 TYPE 'IPQQ' THICK 1.
CONSTANTS
E 30.E6 ALL
POISSON .3 ALL
LOADING 1
ELEMENT LOADS
1 2 EDGE FORCES EDGE 3 GLOBAL UNIFORM LY - 1000.
DUMP TIME
STIFFNESS ANALYSIS
OUTPUT DECIMAL 9
LIST DISPLACEMENTS JOINTS 2 4 6
FINISH

```

TN-77-17(5-7)

Figure 5.7. Input listing of STRUDL model of fracture specimen.

The number of quarter-side nodes surrounding a crack tip affects the accuracy of computation of the stress intensity factors. Several grids were used for the above problem and convergence to the exact solution was extremely good. The exact answer for K_I is $4.016 \text{ psi-in}^{1/2}$. Convergence is illustrated in Figure 5.8.

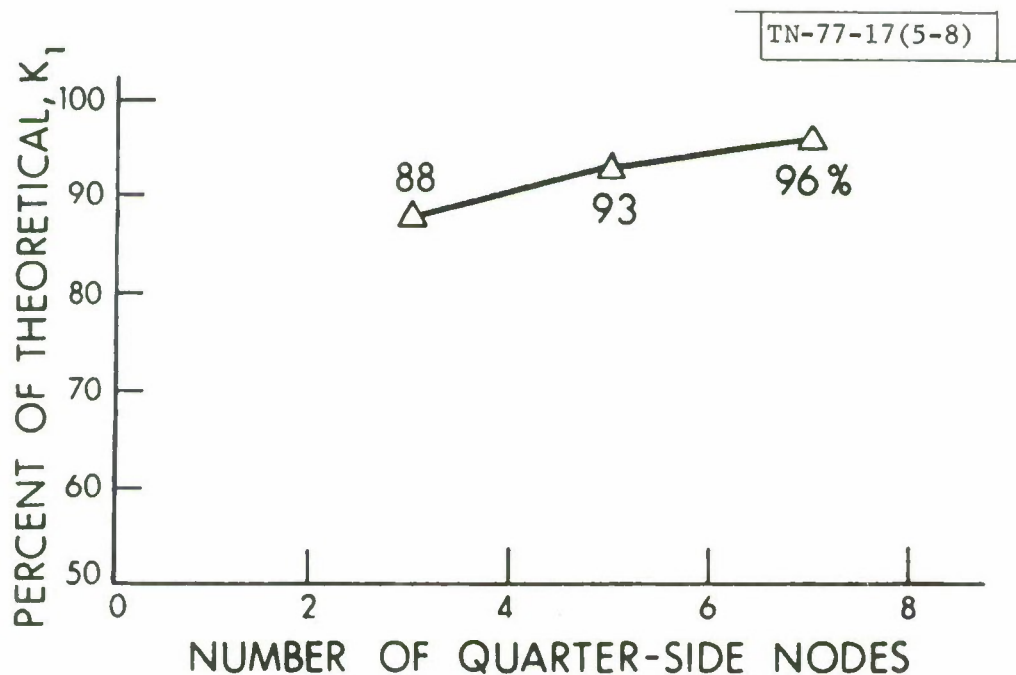


Figure 5.8. Convergence of K_I for fracture specimen.

3. Super Finite Element Generation Example

The construction of super finite elements (or substructures) is often required to analyze large structures. These super finite elements contain only those degrees of freedom needed for structural coupling and those additional degrees of freedom needed to accurately reflect the stiffness and mass properties of the structural component.

Consider a flat plate, support frame, and torque tube assembly (modeled as simply as possible in Figure 5.9 for illustrative purposes). The honeycomb plate section consists of .05" facesheets on a .95" thick core.

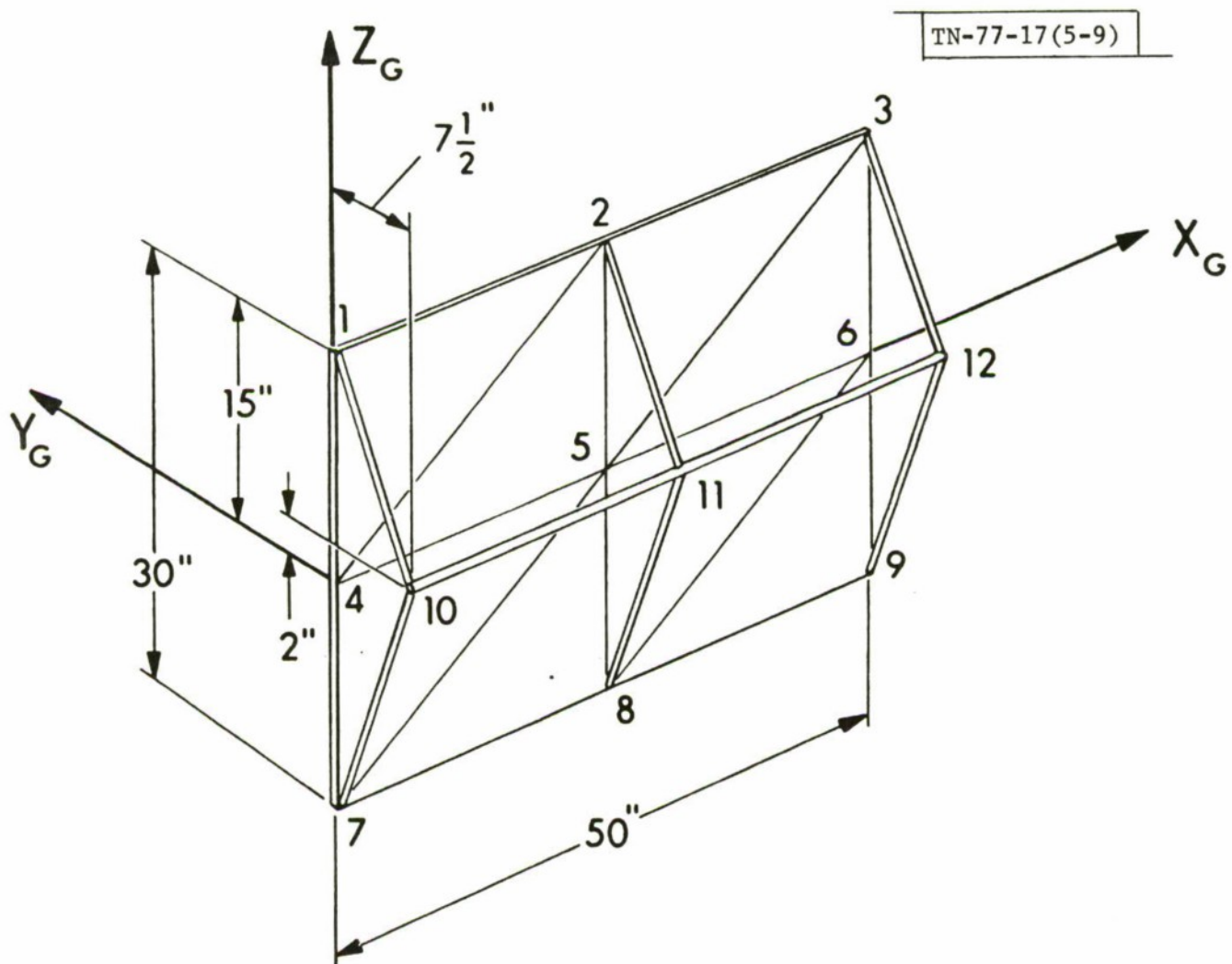


Figure 5.9: Finite element model of plate assembly.

The assembly connects to the total structure at joints 10 and 12. Thus, it is possible to obtain a proper stiffness representation for the above assembly with one super finite element (or member) connecting only those two joints. A proper mass representation would probably require some additional degrees of freedom to be considered for both mass and stiffness matrix development.

Since STRUDL does not compute reactions at supports for problems containing finite elements, it is necessary to obtain them in another way. Two additional joints '10A' and '12A' are constructed differing only from joints 10 and 12 by some very small ΔX_G (in this case, .01 inches). These two additional joints are then made SUPPORTS and rigid-linked (accomplished by specifying very large cross-sectional properties) by two members to their corresponding real joints. The MEMBER FORCES in these two rigid-link members should be identical to the reactions if one is careful to keep the sign convention straight. (It should be simple to transform the results from the member coordinate system into the global coordinate system).

The calculation of the equivalent plate thicknesses to use for the planar ELEMENT PROPERTIES is rather straightforward (refer to Section II-3). The modulus of elasticity of the plane stress and plate bending elements is chosen to be that of the honeycomb plate. Their thicknesses are then computed to be:

$$\text{(stretching)} \quad h_S = 2 h_i = 0.10 \text{ inch}$$

$$\text{(bending)} \quad h_B = \sqrt[3]{24 h_i \left(\frac{d}{2}\right)^2} = 0.67 \text{ inch}$$

One interesting observation should be made at this time. The flat plate is intentionally modeled with triangular finite elements to illustrate that skew symmetric answers will be obtained for this otherwise perfectly symmetric problem. This skew symmetry can be eliminated by overlaying an equal number of finite elements whose diagonals run the other way, and then modifying the plate thicknesses of all the elements (stretching $h'_S = .5 h_S$ and bending

$$h'_B = \sqrt[3]{.5 h_B}).$$

The stiffness matrix relates loads and displacements. One column of this matrix can be constructed by displacing a SUPPORTed degree of freedom one unit, while constraining all others, and then listing the reactions at those other SUPPORTed degrees of freedom. There are 12 degrees of freedom needed for construction of the super finite element stiffness matrix, and thus, 12 loading conditions are required.

Loads acting on the assembly are also transferred to the coupling nodes. The reactions at the fully SUPPORTed coupling nodes represent the super finite element loads to be applied to the coupling nodes in subsequent models.

Note that OUTPUT DECIMAL 2 is specified in Figure 5.10 before listing member forces, so that the large stiffness matrix values (of order 10^7) will not overflow the output field. Also, OUTPUT BY MEMBER has been specified so that the output from the 12 loading cases will be presented under each member separately.

As mentioned earlier, note the lack of agreement in the marked terms in Figure 5.11 between the skewed triangular model and the overlaid symmetric triangular model. Non-zero coupling terms are generated for the skewed triangular model even though coupling does not really exist. This illustrates very vividly that care should be exercised when constructing finite element models with triangular elements.

The member forces (really reactions) are then used as stiffness matrix data for input to another model with no loss of information by this reduction in problem size. While a MEMBER could have been selected to input the stiffness matrix data just generated, a DUMMY finite element was chosen instead, in order to illustrate the input technique for finite element stiffness matrix input. The input portion of the data deck pertaining to the DUMMY element specification is listed in Figure 5.12.

STRUDL 'SUPER' 'SUPER FINITE ELEMENT GENERATION'

JOINT COORDINATES

1 0. 0. 15.
 2 25. 0. 15.
 3 50. 0. 15.
 4 0. 0. 0.
 5 25. 0. 0.
 6 50. 0. 0.
 7 0. 0. -15.
 8 25. 0. -15.
 9 50. 0. -15.
 10 0. -10. 2.
 '10A' -.01 -10. 2. SUPPORT
 11 25. -10. 2.
 12 50. -10. 2.
 '12A' 50.01 -10. 2. SUPPORT

TYPE PLANE STRESS
 ELEMENT INCIDENCES

101 4 2 1
 102 2 4 5
 103 1 5 2
 104 5 1 4
 105 5 3 2
 106 3 5 6
 107 2 6 3
 108 6 2 5
 109 7 5 4
 110 5 7 8
 111 4 8 5
 112 8 4 7
 113 8 6 5
 114 6 8 9
 115 5 9 6
 116 9 5 8

TYPE PLATE BENDING
 ELEMENT INCIDENCES

201 4 2 1
 202 2 4 5
 203 1 5 2
 204 5 1 4
 205 5 3 2
 206 3 5 6
 207 2 6 3
 208 6 2 5
 209 7 5 4
 210 5 7 8
 211 4 8 5
 212 8 4 7
 213 8 6 5
 214 6 8 9
 215 5 9 6
 216 9 5 8

TYPE SPACE FRAME
 MEMBER INCIDENCES

5 1 10
 6 7 10
 7 2 11
 8 8 11
 9 3 12
 10 9 12
 11 10 11
 12 11 12
 13 '10A' 10
 14 12 '12A'

TN-77-17(5-10)

ELEMENT PROPERTIES

101 TO 116 TYPE 'CSTG' THICK .05
 201 TO 216 TYPE 'CPT' THICK .5313
 MEMBER PROPERTIES PRISMATIC
 5 TO 12 AX 3. IX 20. IY 10. IZ 10.
 13 14 AX 30. IX 200. IY 100. IZ 100.

CONSTANTS

E 10.E6 ALL
 POISSON .3 ALL
 LOADING 1
 JOINT DISPLACEMENTS
 '10A' DISP X 1.
 LOADING 2
 JOINT DISPLACEMENTS
 '10A' DISP Y 1.
 LOADING 3
 JOINT DISPLACEMENTS
 '10A' DISP Z 1.
 LOADING 4
 JOINT DISPLACEMENTS
 '10A' ROT X 1.
 LOADING 5
 JOINT DISPLACEMENTS
 '10A' ROT Y 1.
 LOADING 6
 JOINT DISPLACEMENTS
 '10A' ROT Z 1.
 LOADING 7
 JOINT DISPLACEMENTS
 '12A' DISP X 1.
 LOADING 8
 JOINT DISPLACEMENTS
 '12A' DISP Y 1.
 LOADING 9
 JOINT DISPLACEMENTS
 '12A' DISP Z 1.
 LOADING 10
 JOINT DISPLACEMENTS
 '12A' ROT X 1.
 LOADING 11
 JOINT DISPLACEMENTS
 '12A' ROT Y 1.
 LOADING 12
 JOINT DISPLACEMENTS
 '12A' ROT Z 1.

DUMP TIME

STIFFNESS ANALYSIS REDUCE BAND
 OUTPUT BY JOINT
 LIST DISPLACEMENTS ALL
 OUTPUT DECIMAL 2
 OUTPUT BY MEMBER
 LIST FORCES 13 14
 FINISH

Fig.5.10. Input listing of STRUDL model used in super finite element generation.

PROBLEM - SUPER TITLE - SKEWED TRIANGULAR MODEL

TN-77-17(5-11)

MEMBER FORCES								
MEMBER	LOADING	JOINT	FORCES			MOMENTS		
			AXIAL	SHEAR Y	SHEAR Z	TORSION	MOMENT Y	MOMENT Z
13	1	10A	648215.69	3.53	105.98	1578.36	74398.31	-478095.50
		10	-648215.69	-3.53	-105.98	-1578.36	-74399.37	478095.50
	2	10A	3.53	14978.57	802.67	3185.35	-21315.01	374429.50
		10	-3.53	-14978.57	-802.67	-3185.35	21306.98	-374279.69
	3	10A	105.98	802.67	26598.18	120934.06	-665205.31	18758.24
		10	-105.98	-802.67	-26598.18	-120934.06	664939.37	-18750.21
	4	10A	1578.39	3185.56	120934.06	4096715.00	-3042881.00	76995.94
		10	-1578.39	-3185.56	-120934.06	-4096715.00	3041671.00	-76964.06
	5	10A	74398.31	-21314.95	-665205.31	-3042880.00	30660912.00	-1263416.00
		10	-74398.31	21314.95	665205.31	3042880.00	-30654256.00	1263203.00
	6	10A	-478095.50	374429.44	18758.19	76994.81	-1263416.00	16246325.00
		10	478095.50	-374429.44	-18758.19	-76994.81	1263229.00	-16242580.00
	7	10A	-648215.69	-3.53	-105.98	-1578.39	-74398.31	478095.50
		10	648215.69	3.53	105.98	1578.39	74399.37	-478095.56
	8	10A	-3.53	14974.34	802.66	3185.53	21314.95	-374429.12
		10	3.53	-14974.34	-802.66	3185.53	-21306.92	374279.37
	9	10A	-105.98	-802.66	-26581.77	-120934.06	665205.25	-18758.15
		10	105.98	802.66	26581.77	120934.06	-664939.37	18750.12
	10	10A	-1578.39	-3185.21	-120934.06	-4096714.00	3042881.00	-76993.19
		10	1578.39	3185.21	120934.06	4096714.00	-3041672.00	76961.37
	11	10A	-79699.25	-18834.34	-664415.06	-3006241.00	2612640.00	325132.12
		10	79699.25	18834.34	664415.06	3006241.00	-2605996.00	-325320.44
	12	10A	478272.00	374587.37	21391.12	82340.31	197239.44	-2482628.00
		10	-478272.00	-374587.37	-21391.12	-82340.31	-197453.31	2478882.00

PROBLEM - SUPER TITLE - SYMMETRIC TRIANGULAR MODEL

MEMBER FORCES								
MEMBER	LOADING	JOINT	FORCES			MOMENTS		
			AXIAL	SHEAR Y	SHEAR Z	TORSION	MOMENT Y	MOMENT Z
13	1	10A	648346.87	0.00	0.00	0.02	78655.31	-479487.50
		10	-648346.87	-0.00	-0.00	-0.02	-78655.31	479487.50
	2	10A	0.00	14986.55	812.47	3198.29	-20319.92	374886.81
		10	-0.00	-14986.55	-812.47	-3198.29	20311.80	-374736.94
	3	10A	0.00	812.47	26650.64	121072.00	-666327.25	20319.88
		10	-0.00	-812.47	-26650.64	-121072.00	666060.69	-20311.76
	4	10A	0.03	3198.21	121072.06	4110196.00	-3028014.00	79987.19
		10	-0.03	-3198.21	-121072.06	-4110196.00	3026803.00	-79955.25
	5	10A	78655.31	-20319.94	-666327.25	-3028013.00	30806912.00	-1287286.00
		10	-78655.31	20319.94	666327.25	3028013.00	-30800240.00	1287083.00
	6	10A	-479487.50	374886.75	20319.84	79986.69	-1287286.00	16272879.00
		10	479487.50	-374886.75	-20319.84	-79986.69	1287083.00	-16269130.00
	7	10A	-648346.87	-0.00	-0.00	-0.02	-78655.31	479487.50
		10	648346.87	0.00	0.00	0.02	78655.31	-479487.50
	8	10A	0.00	-14989.50	-812.47	-3198.18	20319.95	-374887.00
		10	-0.00	14989.50	812.47	3198.18	-20311.83	374737.12
	9	10A	0.00	-812.47	-26642.73	-121074.81	666327.69	-20319.91
		10	-0.00	812.47	26642.73	121074.81	-666061.25	20311.79
	10	10A	0.02	-3198.32	-121074.81	-4110231.00	3028016.00	-79987.62
		10	-0.02	3198.32	121074.81	4110231.00	-3026805.00	79955.62
	11	10A	-78655.31	-20319.86	-666327.62	-3028016.00	2522800.00	-270884.56
		10	78655.31	20319.86	666327.62	3028016.00	-2516137.00	271087.75
	12	10A	479487.56	374887.00	20319.95	79987.37	270884.69	2478964.00
		10	-479487.56	-374887.00	-20319.95	-79987.37	-271087.87	-2475215.00

Figure 5.11. Comparative STRUDL results for differing triangular meshes (skewed vs. symmetric).

.
.
.
.
.
TYPE SPACE FRAME

ELEMENT INCIDENCES

TN-77-17(5-12)

15 10 12

ELEMENT PROPERTIES

15 TYPE 'DUMMY' THICK 1. DOF DX DY DZ RX RY RZ -
STIFFNESS MATRIX GLOBAL NODES 2 NDF 6

SUBMATRIX 1 1 -

648346.	0.	0.	0.	78655.	-479487.
0.	14986.	812.	3198.	-20319.	374886.
0.	812.	26650.	121072.	-666327.	20319.
0.	3198.	121072.	4110196.	-3028014.	79987.
78655.	-20319.	-666327.	-3028013.	30806912.	-1287286.
-479487.	374886.	20319.	79986.	-1287286.	16272879.

SUBMATRIX 2 1 -

-648346.	-0.	-0.	-0.	-78655.	479487.
0.	-14989.	-812.	-3198.	20319.	-374887.
0.	-812.	-26642.	-121074.	666327.	-20319.
0.	-3198.	-121074.	-4110231.	3028016.	-79987.
-78655.	-20319.	-666327.	-3028016.	2522800.	270884.
479487.	374887.	20319.	79987.	270884.	2478964.

SUBMATRIX 2 2 -

648346.	0.	-0.	-0.	78655.	-479487.
0.	14990.	812.	3197.	20319.	-374886.
-0.	812.	26656.	121072.	666327.	-20319.
-0.	3198.	121072.	4110195.	3028013.	-79985.
78655.	20319.	666327.	3028013.	30806912.	-1287286.
-479487.	-374886.	-20319.	-79985.	-1287286.	16272876.

.
.
.
.
.

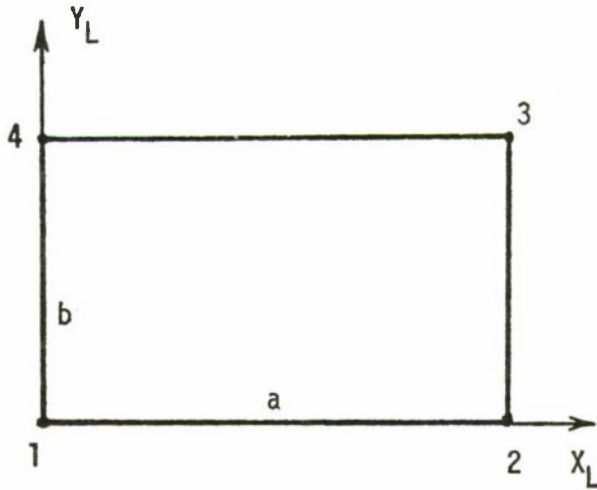
Figure 5.12. Partial listing of STRUDL super finite element model.

APPENDIX

Description of Finite Elements

Element Type: 'BPR' - Bending Plate Rectangle

Shape & Node Order:



Nodal Unknowns: DZ, RX, RY

Results: Planar stress couples and strain curvatures at each node.

Element Loads:

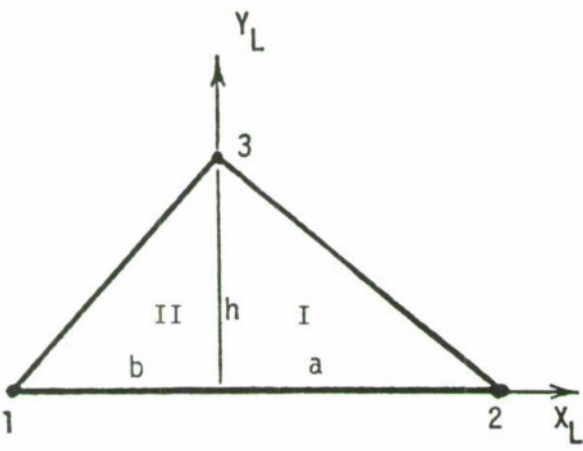
Uniform out-of-plane surface forces

References: 3

Element Characteristics: BPR is a flat rectangular element with four nodes, the corner points. Its transverse displacement expansion is

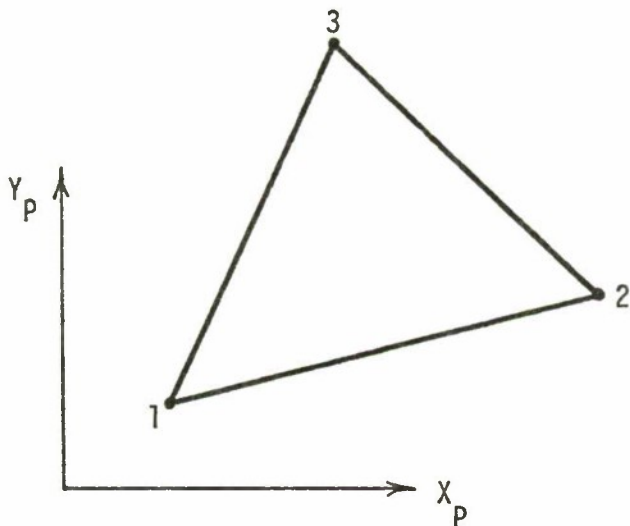
$$w = \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 xy + \alpha_5 x^2 + \alpha_6 y^2 + \alpha_7 x^3 + \alpha_8 y^3 + \alpha_9 x^2 y + \alpha_{10} xy^2 + \alpha_{11} x^3 y + \alpha_{12} xy^3$$

The expansions for the rotations are the derivatives of that for w . The element satisfies inter-element compatibility with respect to translation, but, since only two nodal values for normal slope are available along each side, the quadratic variation is not unique and the normal slope is not continuous. The solution will converge as the mesh size is reduced, but the convergence will not be monotonic, due to the incompatibility of normal slope.

Element Type: 'CPT' - Compatible Plate Triangle	
Shape & Node Order: 	Nodal Unknowns: DZ, RX, RY
	Results: Planar stress couples and strain curvatures at each node.
	Element Loads: Uniform out-of-plane surface forces
	References: 4
Element Characteristics: CPT is a flat triangular element with three nodes, the corner points. The element is divided into two triangles with the following displacement expansion: $w_I = \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 x^2 + \alpha_5 xy + \alpha_7 y^2 + \left(\frac{1}{3} \left(2 - \left(\frac{h}{a} \right)^2 \right) \alpha_8 - \frac{h}{a} \alpha_9 \right) x^3 + \alpha_8 xy^2 + \alpha_9 y^3$ These expansions are not complete cubics, since they contain no $x^2 y$ terms. The expansions for the rotations are the derivatives of that for w. w and $\frac{\partial w}{\partial x}$ are continuous across the interior boundary, $x = 0$. The normal slope varies linearly along each edge. The missing terms cause displacement compatibility not to be completely satisfied along side 1-2, since the displacement expression involves nodal quantities for the third node. The accuracy of the element is quite good, however, in spite of this incompatibility.	

Element Type: 'CSTG' - Constant Strain Triangle

Shape & Node Order:



Nodal Unknowns: DX, DY

Results: Planar stresses and strains at centroid.

Element Loads:

In-plane surface,
body, and edge forces

References: 3, 5, 6

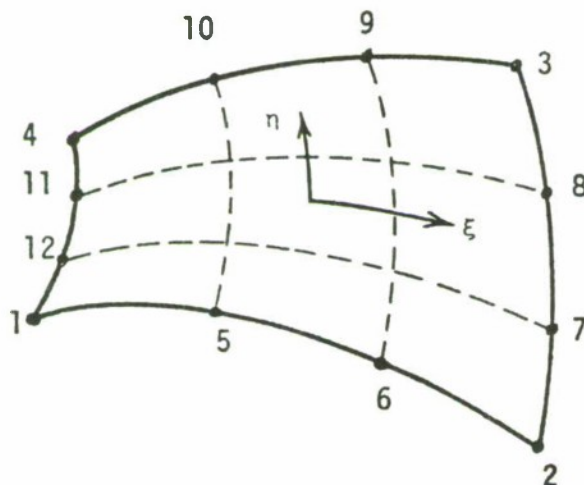
Element Characteristics: CSTG is a triangular element with three nodes, the corner points. The element stiffness matrix is computed based on displacement expansions of the following form

$$\chi = \alpha_1 + \alpha_2 x + \alpha_3 y$$

which yield a linear displacement field over the element and along each side. The stresses and strains are thus constant over the element.

Element Type: 'IPCQ' - Isoparametric Cubic Quadrilateral

Shape & Node Order:



Nodal Unknowns: DX, DY

Results: Planar stresses and strains at each node.

Element Loads:

In-plane surface, body, and edge forces

References: 7, 8

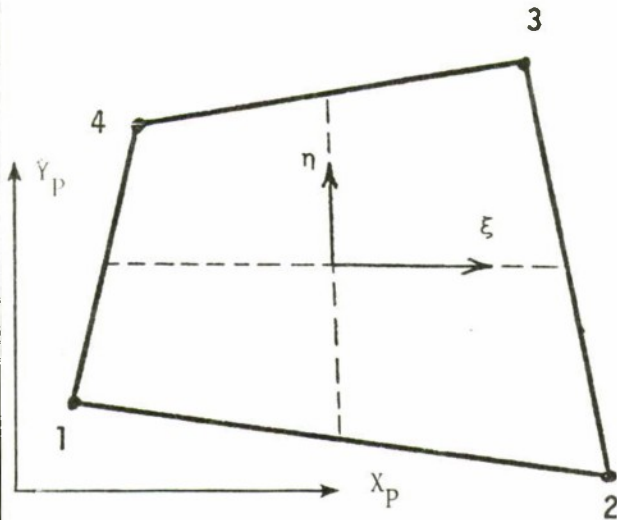
Element Characteristics: IPCQ is a flat quadrilateral element with 12 nodes, the 4 corner points, and the 8 third-points of each side. The displacement expansions are of the form

$$\begin{aligned} \chi = & \alpha_1 + \alpha_2 \xi + \alpha_3 \eta + \alpha_4 \xi \eta + \alpha_5 \eta^2 + \alpha_6 \xi^2 + \alpha_7 \xi^3 + \alpha_8 \xi^2 \eta + \alpha_9 \xi \eta^2 \\ & + \alpha_{10} \eta^3 + \alpha_{11} \eta^3 \xi + \alpha_{12} \xi^3 \eta \end{aligned}$$

which yields a quartic field over the element, but a cubic variation along the edges. ξ and η are generalized curvilinear coordinates which vary from -1 to +1 across the element. The sides of IPCQ may be initially curved if the coordinates of the midside nodes are not colinear with the corner nodes. If coordinates are not specified for the midside nodes, they will be computed assuming that the sides are straight.

Element Type: 'IPLQ' - Isoparametric Linear Quadrilateral
 'IPLQCSH' - Isoparametric Linear Quadrilateral - Constant Shear

Shape & Node Order:



Nodal Unknowns: DX, DY

Results: Planar stresses and strains at each node.

Element Loads:

In-plane surface, body, and edge forces

References: 7, 8, 9

Element Characteristics: IPLQ is a flat quadrilateral element with 4 nodes, the corner points. The displacement expansions are of the form

$$x = \alpha_1 + \alpha_2\xi + \alpha_3\eta + \alpha_4\xi\eta$$

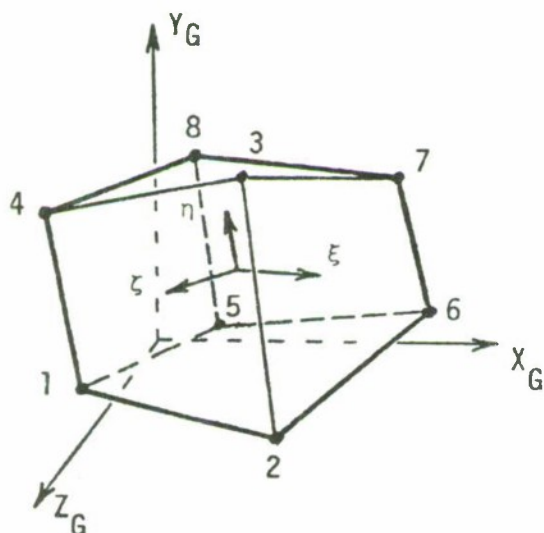
which yields a quadratic field over the element, but a linear variation along the edges. ξ and η are generalized coordinates which vary from -1 to +1 across the element.

IPLQCSH is a flat quadrilateral element with 4 nodes, the corner points. The displacement expansions are the same as the IPLQ, but the assumed shear strain variation in the X-direction is different. The shear strain variation for the IPLQ is assumed to be a constant in the X-direction and equal to the centroidal value of the element; whereas, the shear strain varies linearly in the IPLQ. The resulting formulation tends to give better results than the IPLQ for problems in which the in-plane forces are generated by overall bending.

The stiffness matrix is generated by numerical integration, rather than explicitly, and results in an element with performance characteristics and accuracy similar to the PSRCSH element. The formulations are identical if the element is rectangular in shape.

Element Type: 'IPLS' - Isoparametric Linear Solid
 'IPLSCSH' - Isoparametric Linear Solid - Constant Shear

Shape & Node Order:



Nodal Unknowns: DX, DY, DZ

Results: Global stresses and strains at each node.

Element Loads:

Surface, body,
and edge forces

References: 7, 8, 9

Element Characteristics: IPLS is a six-sided solid element each of whose faces is quadrilateral. The element has 8 nodes, the corner points. The displacement expansions are of the form

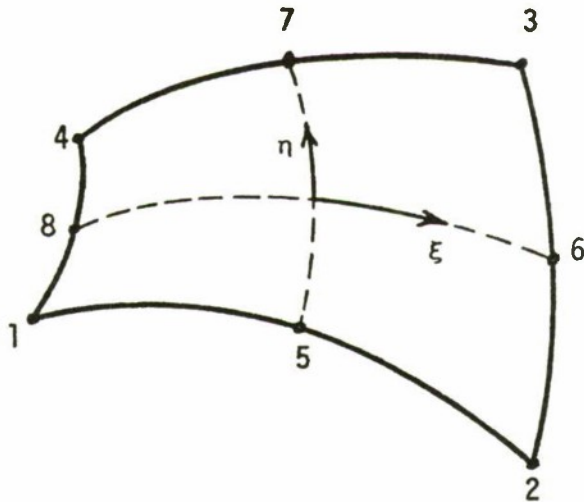
$$x = \alpha_1 + \alpha_2\xi + \alpha_3\eta + \alpha_4\zeta + \alpha_5\xi\eta + \alpha_6\xi\zeta + \alpha_7\eta\zeta + \alpha_8\xi\eta\zeta$$

which yields a cubic field through the element, a quadratic field on the faces, and linear displacements along the edges. ξ , η , ζ are generalized coordinates which vary from -1 to +1 across the element.

IPLSCSH is a six-sided solid element each of whose faces is a quadrilateral. The element has 8 nodes, the corner points. The displacement expansions are the same as for the IPLS, but the assumed shear strain variation in the X-direction is different. The shear strain variation for the IPLSCSH is assumed to be constant in the X-direction and equal to the centroidal value of the element; whereas, the shear strain varies linearly in the IPLS. The resulting formulation tends to give better results than the IPLS for the problems in which the structure is being subjected to overall bending.

Element Type: 'IPQQ' - Isoparametric Quadratic Quadrilateral

Shape & Node Order:



Nodal Unknowns: DX, DY

Results: Planar stresses and strains at each node.

Element Loads:

In-plane surface, body, and edge forces

References: 7, 8

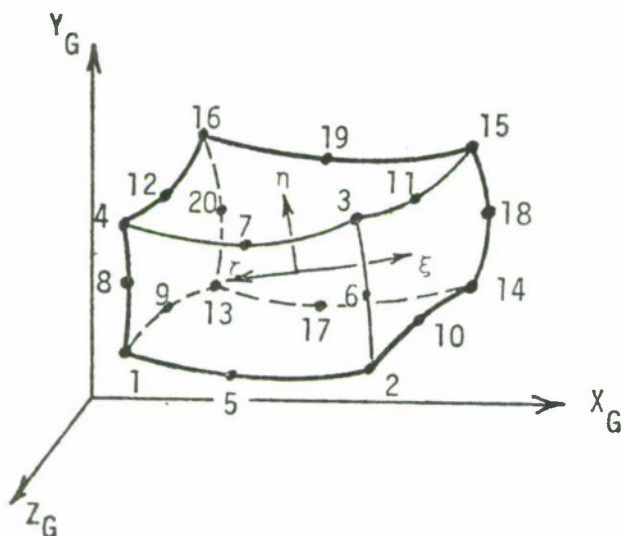
Element Characteristics: IPQQ is a flat quadrilateral element with 8 nodes, the corner points and the midpoints of each side. The displacement expansions are of the form

$$x = \alpha_1 + \alpha_2\xi + \alpha_3\eta + \alpha_4\xi\eta + \alpha_5\xi^2 + \alpha_6\eta^2 + \alpha_7\xi^2\eta + \alpha_8\xi\eta^2$$

which yields a cubic field over the element, but a quadratic variation along the edges. ξ and η are generalized curvilinear coordinates which vary from -1 to +1 across the element. Although the stiffness matrix is generated by numerical integration, rather than explicitly, the result is an element with performance characteristics and accuracy similar to the LSR element. The sides of IPQQ may be initially curved if the coordinates of the midside nodes are not colinear with the corner nodes. If coordinates are not specified for the midside nodes, they will be computed assuming that the sides are straight.

Element Type: 'IPQS' - Isoparametric Quadratic Solid

Shape & Node Order:



Nodal Unknowns: DX, DY, DZ

Results: Global stresses and strains at each node.

Element Loads:

Surface, body, and edge forces

References: 7, 8

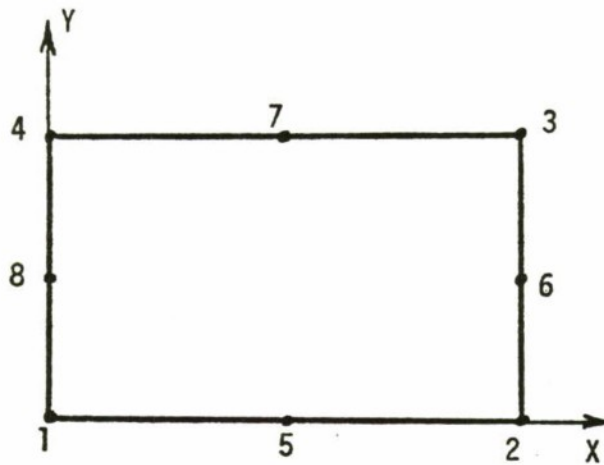
Element Characteristics: IPQS is a six-sided solid element each of whose faces is a quadrilateral. The element has 20 nodes, the corner points and midpoints of each edge. The displacement expansions are of the form

$$\begin{aligned} X = & \alpha_1 + \alpha_2 \xi + \alpha_3 \eta + \alpha_4 \zeta + \alpha_5 \xi \eta + \alpha_6 \xi \zeta + \alpha_7 \eta \zeta + \alpha_8 \xi \eta \zeta + \alpha_9 \xi^2 + \alpha_{10} \eta^2 \\ & + \alpha_{11} \zeta^2 + \alpha_{12} \xi^2 \eta + \alpha_{13} \xi^2 \zeta + \alpha_{14} \xi \eta^2 + \alpha_{15} \eta^2 \zeta + \alpha_{16} \xi \zeta^2 + \alpha_{17} \eta \zeta^2 \\ & + \alpha_{18} \xi^2 \eta \zeta + \alpha_{19} \xi \eta^2 \zeta + \alpha_{20} \xi \eta \zeta^2 \end{aligned}$$

which yields a quartic field through the element, a cubic field on the faces, and quadratic displacements along the edges. ξ , η , and ζ are generalized curvilinear coordinates which vary from -1 to +1 across the element. The edges of IPQS may be initially curved if the coordinates of the midside nodes are not colinear with the corner nodes. If coordinates are not specified for the midside nodes, they will be computed assuming that the edges are straight.

Element Type: 'LSR' - Linear Strain Rectangle

Shape & Node Order:



Nodal Unknowns: DX, DY

Results: Planar stresses and strains at the corner nodes.

Element Loads:

In-plane surface, body, and edge forces

References: 10

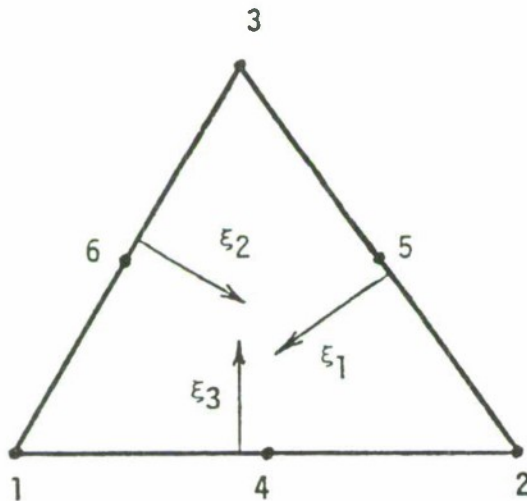
Element Characteristics: LSR is a flat rectangular element with 8 nodes, the corner points and the midpoints of each side. The sides must be straight initially. The displacement expansions are of the form

$$x = \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 xy + \alpha_5 x^2 + \alpha_6 y^2 + \alpha_7 x^2 y + \alpha_8 xy^2$$

which yields a cubic displacement field over the element, but a quadratic variation along the boundaries. The displacement along each boundary depends only upon the nodal quantities on that boundary, satisfying inter-element displacement compatibility.

Element Type: 'LST' - Linear Strain Triangle

Shape & Node Order:



Nodal Unknowns: DX, DY

Results: Planar stresses and strains at the corner nodes.

Element Loads:

In-plane surface, body, and edge forces

References: 11

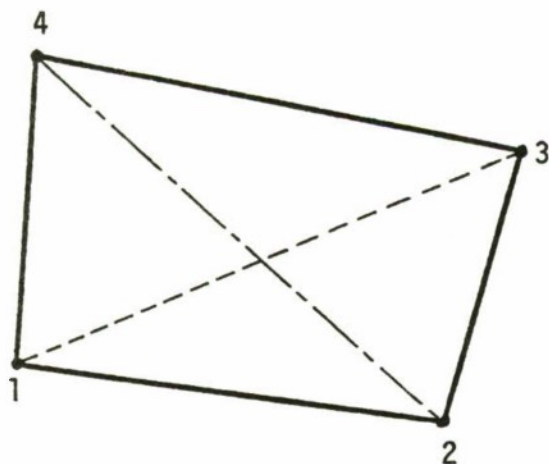
Element Characteristics: LST is a flat triangular element with 6 nodes, the corner points and the midpoints of each side. The sides must be straight initially. The displacement expansions are expressed in terms of the triangular coordinates ξ_1 , ξ_2 , ξ_3 as follows:

$$\chi = \left(\sum_{i=1}^3 \xi_i (2\xi_i - 1) \right) + 4(\xi_1\xi_2 + \xi_1\xi_3 + \xi_2\xi_3)$$

These expansions yield a complete second-degree polynomial, giving a quadratic displacement field over the element and along each boundary. The displacements along each boundary depend only upon the nodal quantities on that boundary, satisfying inter-element displacement compatibility.

Element Type: 'PBQ1' - Plate Bending Quadrilateral

Shape & Node Order:



Nodal Unknowns: DZ, RX, RY

Results: Planar stress couples and strain curvatures at each node.

Element Loads:

Uniform out-of-plane surface forces

References: 10

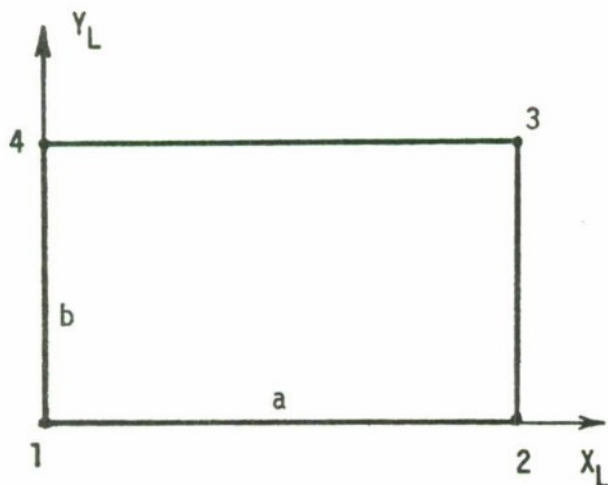
Element Characteristics: PBQ1 is a flat quadrilateral element with 4 nodes, the corner points. The stiffness matrix is formed as the average of the matrices generated by summing the stiffnesses of the two pairs of triangles formed by diagonals across the element.

$$K_{1234} = \frac{1}{2} \left\{ \left[K_{123} + K_{134} \right] + \left[K_{124} + K_{234} \right] \right\}$$

The triangles used in this case are CPT's. The resulting solution will converge as the mesh size is reduced, and the results are more accurate than the worst combination of triangles, but less accurate than the best. Being composed of CPT's, the absolute accuracy is relatively good in any case. The stress results are best when the triangular areas are nearly the same; i.e., the quadrilateral is a parallelogram.

Element Type: 'PSR' - Plane Stress Rectangle
 'PSRCSH' - Plane Stress Rectangle - Constant Shear

Shape & Node Order:



Nodal Unknowns: DX, DY

Results: Planar stresses and strains at the centroid.

Element Loads:

In-plane surface, body, and edge forces

References: 3, 5, 6, 9

Element Characteristics: PSR is a rectangular element with 4 nodes, the corner points. The element stiffness matrix is computed based upon displacement expansions in the local coordinate system of the following form

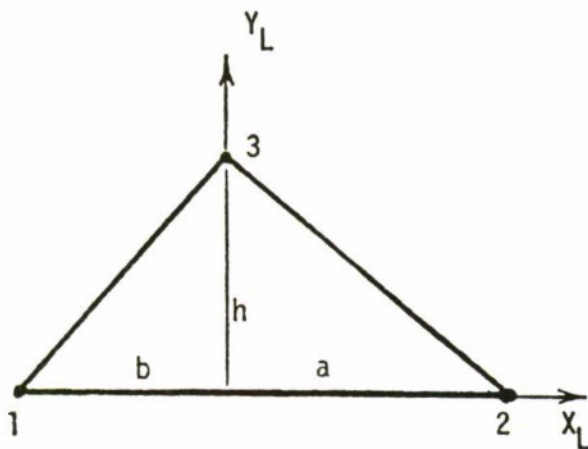
$$\chi = \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 xy$$

which yields a quadratic displacement field over the element, but a linear variation along the edges.

PSRCSH is a rectangular element with 4 nodes, the corner points. The displacement expansions are the same as for the PSR, but the assumption concerning the shear strain variation in the X-direction is different. The shear strain variation for the PSRCSH is assumed to be a constant in the X-direction and equal to the centroidal value of the element; whereas the shear strain varies linearly in the PSR. The resulting formulation tends to give better results than the PSR for problems in which the in-plane forces are generated by overall bending.

Element Type: 'SBCT' - Stretching & Bending Compatible Triangle

Shape & Node Order:



Nodal Unknowns: DX, DY, DZ, RX, RY

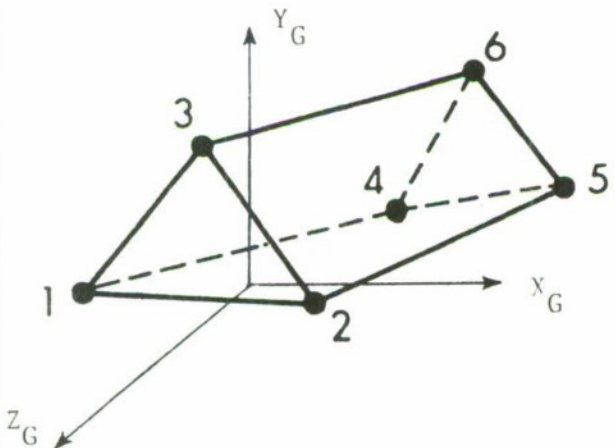
Results: Planar stresses and strains at centroid; planar stress couples and strain curvatures at each node.

Element Loads:

Uniform out-of-plane surface forces. In-plane surface, body, and edge forces.

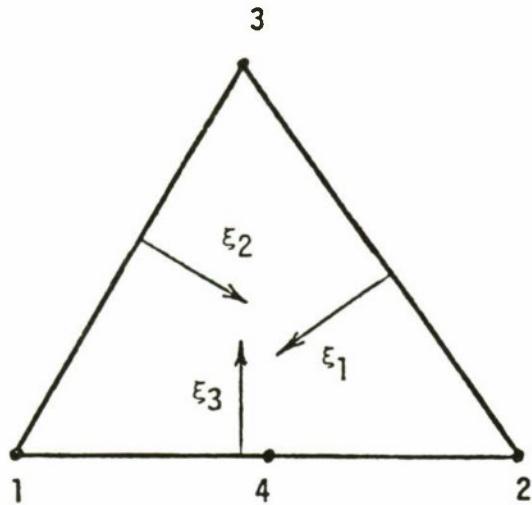
References: 12

Element Characteristics: SBCT is a flat triangular element with 3 nodes, the corner points. The element stiffness matrix is the sum of the CSTG and CPT stiffness matrices and is generated as an overlay of those two elements. The in-plane and out-of-plane stiffnesses and displacements are uncoupled. The in-plane formulation is plane stress.

Element Type: 'TRIP' - Triangular Prism	
Shape & Node Order: 	Nodal Unknowns: DX, DY, DZ
	Results: Global stresses and strains at the nodes.
	Element Loads: Surface, body, and edge forces
References:	
Element Characteristics: TRIP is a five-sided solid element containing two triangular faces and three quadrilateral faces. The element has 6 nodes, the corner points. The displacement expansions are of the same form as the IPLS elements, but the quadrilateral front and back faces are allowed to degenerate to triangles. This expansion yields a quadratic field on the quadrilateral faces, and a linear field on the triangular faces and along the edges.	

Element Type: 'UTLQ1' - Unequal Transition Linear Quadrilateral - 1 Node

Shape & Node Order:



Nodal Unknowns: DX, DY

Results: Planar stresses and strains at the nodes.

Element Loads:

In-plane surface,
body, and edge forces

References: 13

Element Characteristics: UTLQ1 is a flat triangular element with 4 nodes, the corner points and one midpoint on the first side. The displacement expansions are expressed in terms of the triangular coordinates ξ_1 , ξ_2 , ξ_3 as follows:

$$\chi = \sum_{i=1}^3 (\xi_i^2 \xi_i - 1) + 4\xi_1\xi_2$$

These expansions will yield a quadratic displacement field over the element and along the first side, and a linear displacement field over the remaining two sides. The displacements along each boundary depend only upon the nodal quantities on that boundary, satisfying inter-element displacement compatibility.

REFERENCES

1. R. S. Barsoum, "Application of Quadratic Isoparametric Finite Elements in Linear Fracture Mechanics," *Int. J. Fracture* 10, 603-605 (1974).
2. L. Nash Gifford Jr., "APES - Second Generation Two Dimensional Elasticity and Fracture Analyses by Finite Element," NSRDC Report SD-172-188 (July 1974).
3. J. S. Przemieniecki, Theory of Matrix Structural Analysis (McGraw-Hill, New York, 1968).
4. J. J. Connor and G. T. Will, "Triangular Flat Plate Bending Element," Report TR68-3, M.I.T. Department of Civil Engineering (February 1968).
5. J. J. Connor and G. T. Will, "Computer-Aided Teaching of the Finite Element Displacement Method," Report 69-23, M.I.T. Department of Civil Engineering (February 1969).
6. O. C. Zienkiewicz and Y. K. Cheung, The Finite Element Method (McGraw-Hill, New York, 1967).
7. I. Ergatoudis, B. M. Irons and O. C. Zienkiewicz, "Curved, Isoparametric, 'Quadrilateral' Elements for Finite Element Analysis," *Int. J. Solids and Structures*, 4, 31-42 (1968).
8. L. E. Aparicia and J. J. Connor, "Isoparametric Finite Element Displacement Models," Research Report R70-39, M.I.T. Department of Civil Engineering (July 1970).
9. S. S. Britten and J. J. Connor, "A New Family of Finite Elements," Research Report R71-14, M.I.T. Department of Civil Engineering (February 1971).
10. L. E. Aparacio, "Finite Element Implementation for the Structural Design Language," M.S. Thesis, M.I.T. Department of Civil Engineering (September 1969).
11. C. D. Felippa, "Refined Finite Element Analysis of Linear and Non Linear Two Dimensional Structures," SESM Report, University of California at Berkeley (1966).
12. L. Rodriguez, "Finite Element Non-Linear Analysis for Plates and Shallow Shells," Ph.D. Thesis, M.I.T. Department of Civil Engineering (September 1968).
13. C. Caramanlian, K. A. Selby and G. T. Will, "Plane Stress Formulation in Finite Element Method," Publication 76-06, University of Toronto, Department of Civil Engineering (June 1976).

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE		READ INSTRUCTIONS BEFORE COMPLETING FORM
1. REPORT NUMBER ESD-TR-77-266	2. GOVT ACCESSION NO.	3. RECIPIENT'S CATALOG NUMBER
4. TITLE (and Subtitle) Finite Element Capabilities in M.I. T. Lincoln Laboratory Version of STRUDL		5. TYPE OF REPORT & PERIOD COVERED Technical Note
		6. PERFORMING ORG. REPORT NUMBER Technical Note 1977-17
7. AUTHOR(s) Stephen S. Britten		8. CONTRACT OR GRANT NUMBER(s) F19628-76-C-0002
9. PERFORMING ORGANIZATION NAME AND ADDRESS Lincoln Laboratory, M.I. T. P.O. Box 73 Lexington, MA 02173		10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS Program Element No. 63431F Project No. 1227
11. CONTROLLING OFFICE NAME AND ADDRESS Air Force Systems Command, USAF Andrews AFB Washington, DC 20331		12. REPORT DATE 30 September 1977
		13. NUMBER OF PAGES 84
14. MONITORING AGENCY NAME & ADDRESS (if different from Controlling Office) Electronic Systems Division Hanscom AFB Bedford, MA 01731		15. SECURITY CLASS. (of this report) Unclassified
		15a. DECLASSIFICATION DOWNGRADING SCHEDULE
16. DISTRIBUTION STATEMENT (of this Report) Approved for public release; distribution unlimited.		
17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)		
18. SUPPLEMENTARY NOTES None		
19. KEY WORDS (Continue on reverse side if necessary and identify by block number) structural analysis ICES-STRUDL specifications finite elements		
20. ABSTRACT (Continue on reverse side if necessary and identify by block number) The well-known, general-purpose structural analysis program ICES-STRUDL has been used extensively at M.I. T. Lincoln Laboratory since its public release in 1967. Since that original release, several updates, issued by the original developers and the ICES Users Group, have produced changes in programs and documentation. Likewise, additional development and enhancement of ICES-STRUDL have occurred at M.I. T. Lincoln Laboratory on those areas (such as finite elements, dynamics, etc.) specifically of interest to the Laboratory. This report describes the finite element capabilities in the M.I. T. Lincoln Laboratory version of ICES-STRUDL and can therefore serve as a user's manual. Sections are devoted to specification of finite element geometry, element properties, and element loading. Finite element modelling considerations are discussed and three examples are presented to illustrate capabilities often needed at the Laboratory, but not described in other ICES-STRUDL documentation.		

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)