SOUTHWEST RESEARCH INSTITUTE
Post Office Drawer 28510, 6220 Culebra Road
San Antonio, Texas 78284

SUPPORTING INFORMATION FOR THE STRESS ANALYSIS OF THE THIN PIPE REGION IN THE NO. 12 STEAM GENERATOR MAIN STEAM LINE (EB-01-1005-05)

By
Lawrence J. Goland, P.E.

STATE OF TEXAS
L. J. GOLAND
53162
REGISTERED
PROFESSIONAL ENGINEER

Lawrence J. Goland
4/8/88

FINAL REPORT
SwRI Project No. 06-1917-203

Prepared For
Baltimore Gas and Electric
Charles Center
P. O. Box 1475
Baltimore, Maryland 21203

March 1988

Approved:

EmBriggs

Edward M. Briggs, Director
Department of Structural
and Mechanical Systems

ABSTRACT

The Nuclear Regulatory Commission requested additional information concerning the analysis that was performed on the thin pipe region in the Calvert Cliffs Nuclear Power Plant Unit No. 1 main steam line. The items requiring further explanation are:

1.) The theoretical information about the quadrilateral shell element used for the finite element model of the thin pipe and adjacent structures,

2.) The state of stress in the pipe in the plane of the thin pipe region, and

3.) A complete description of the beam model used to determine the deadweight loads that were applied to the detailed analysis of the thin pipe region.

This document presents the requested information.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1. INTRODUCTION

A thin wall region in a Calvert Cliffs Nuclear Power Plant Unit No. 1 main steam line was analyzed for stresses by Southwest Research Institute (SwRI). The results of the analysis are presented in the final report entitled "Stress Analysis Of Thin Pipe Region In No. 12 Steam Generator Main Steam Line (EB-01-1005-05)", (Reference 1).

A telephone conference call was held on Thursday, February 11, 1988 with Mr. Bill Holston of Baltimore Gas and Electric (BG&E), Mr. Larry Goland (SwRI), and persons of the Nuclear Regulatory Commission (NRC) for the purpose of clarifying certain information presented in the original report. As a result of the discussion, the following information was requested by the NRC:

1.) A description of the shell elements used in the finite element analysis of the thin pipe region and elbow,
2.) The state of stress in the pipe 180 degrees away from the thin pipe region, and
3.) Complete information about the pipe beam model used to determine the deadweight forces and moments which were used in the detailed analysis of the thin pipe region.

This report addresses the requested information.

CHAPTER 2. DESCRIPTION OF SHELL ELEMENTS USED IN ANALYSIS

The thin pipe region was analyzed using the general purpose finite element program ANSYS (Reference 2). The elbow, thin pipe region, and a segment of the horizontal straight pipe were modelled using quadrilateral shell elements designated as STIF63 type elements in the program ANSYS. This element has both bending and membrane stiffness, with resulting bending and membrane stresses available for output.

Appendices A and B of this report present the application and theoretical documentation for this element, respectively. This information was copied from the ANSYS User's and Theoretical manuals (References 2 and 3, respectively).

CHAPTER 3. PIPE STRESSES IN PLANE OF THIN PIPE REGION

The location of the thin pipe region is shown diagrammatically in Figure 3.1. This is the same figure as Figure 1 in the original report. Table 3.1 presents the hoop and longitudinal stress components in the plane of the thin pipe at the locations indicated in Figure 3.2. Outside, mid, and inside surface stress components are reported in the table. The loading condition which produced these stress levels was a combination of an internal pressure of 1000 psig and deadweight.

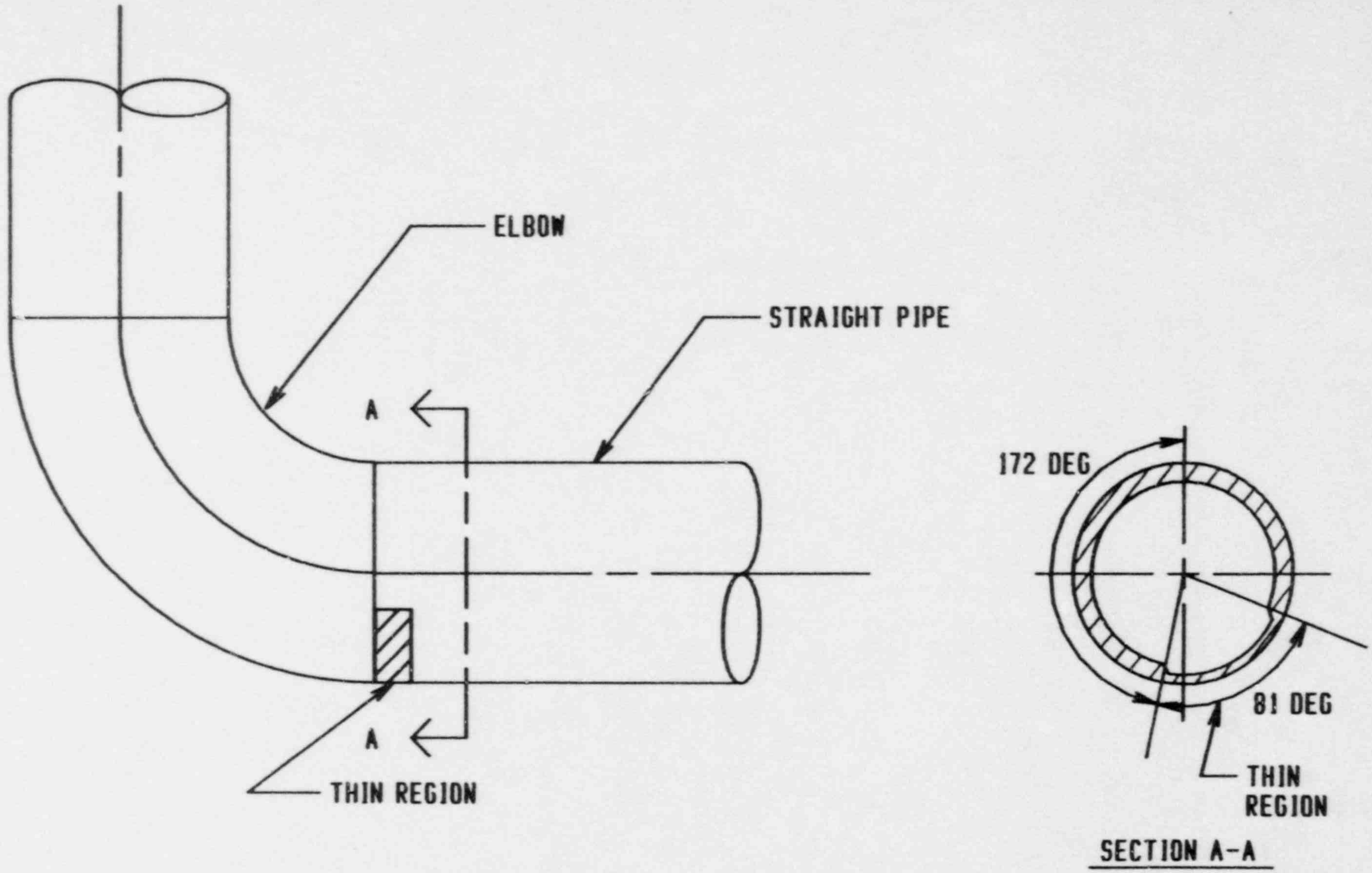


FIGURE 3.1

LOCATION OF THIN PIPE REGION
(SAME AS FIGURE 1 OF REFERENCE 1)

TABLE 3.1

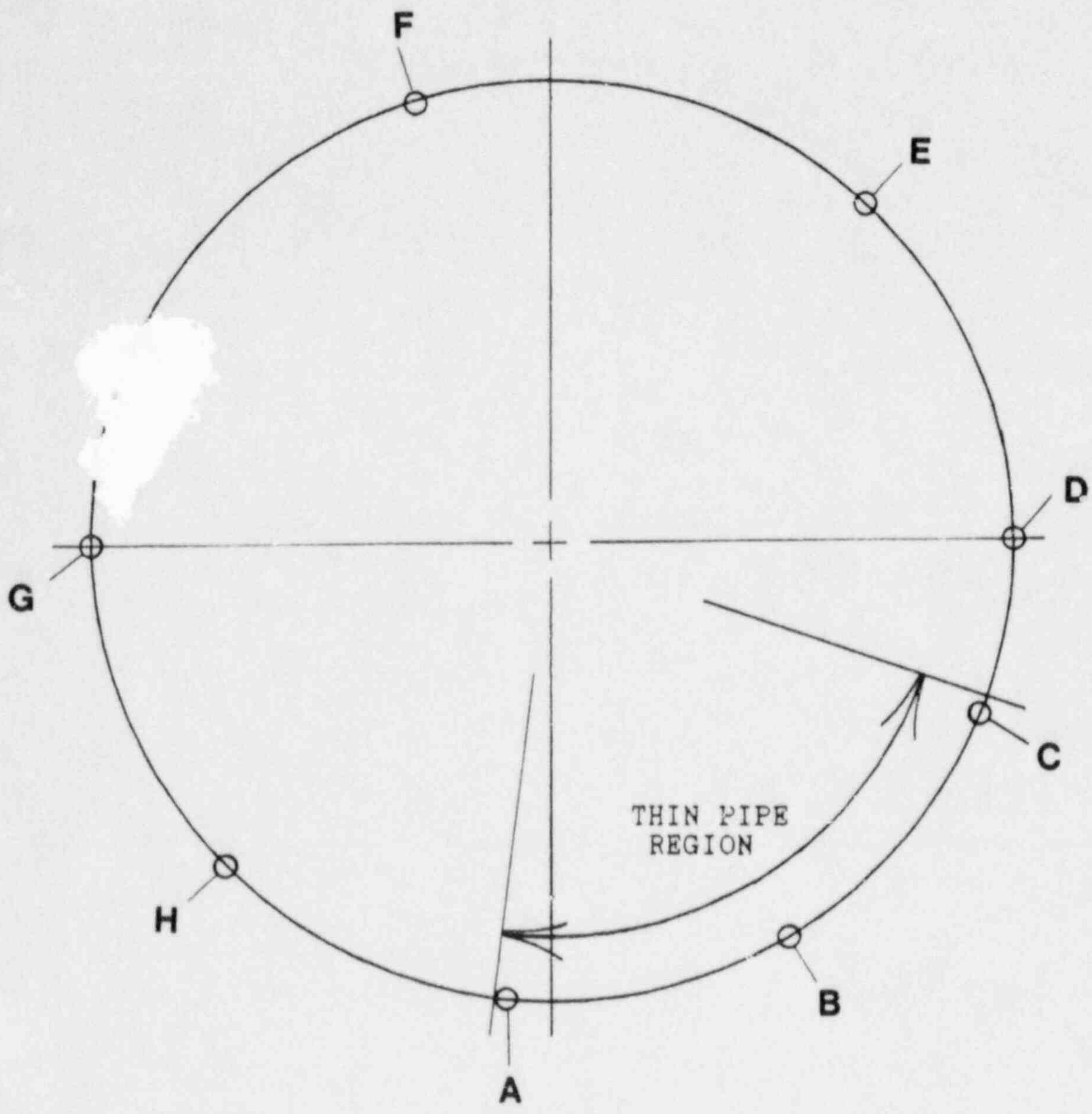
STRESSES IN COMPLETE PIPE SECTION
AT THIN PIPE REGION LOCATION

LOCATION (NOTE 1)	STRESS COMPONENT	STRESSES (PSI)		
		OUTSIDE SURFACE (NOTE 2)	MID SURFACE (NOTE 3)	INSIDE SURFACE (NOTE 2)
A	HOOP	11,600	14,900	18,100
	LONGITUDINAL	7,400	8,700	11,400
B	HOOP	14,200	14,600	15,000
	LONGITUDINAL	6,700	7,900	9,200
C	HOOP	11,400	15,900	19,600
	LONGITUDINAL	6,100	8,400	10,700
D	HOOP	16,600	15,000	13,400
	LONGITUDINAL	7,900	7,600	7,400
E	HOOP	15,500	15,700	15,800
	LONGITUDINAL	7,700	7,600	7,600
F	HOOP	16,900	15,700	14,600
	LONGITUDINAL	7,100	6,600	6,100
G	HOOP	13,500	15,000	16,400
	LONGITUDINAL	6,700	7,400	8,100
H	HOOP	14,300	14,200	14,000
	LONGITUDINAL	6,700	7,500	8,300

NOTE 1: SEE FIGURE 3.2 FOR LOCATIONS WITH RESPECT TO THIN PIPE REGION

NOTE 2: OUTSIDE AND INSIDE STRESS LEVELS ARE COMPRISED OF MEMBRANE PLUS BENDING STRESS COMPONENTS.

NOTE 3: MID SURFACE STRESSES ARE MEMBRANE STRESSES.



(STRESSES ARE PRESENTED IN TABLE 3.1)

FIGURE 3.2

REPORTED PIPE STRESS LOCATIONS
IN PLANE OF THIN PIPE REGION

CHAPTER 4. DEADWEIGHT LOADING ANALYSIS

The thin pipe region was analyzed under a 1000 psig internal pressure and deadweight loading condition. These loads are shown in Figure 4.1, which is Figure 6 in the original report. The force (FZ) on the end of the finite element model represents the axial force due to internal pressure and deadweight loading on the structure. The moments (MX and MY) are the result of structural deadweight. The deadweight loadings were determined from a separate finite element analysis using a 3-dimensional beam model and deadweight loads supplied to SwRI by BG&E personnel.

The finite element program ANSYS was used for the deadweight load analysis. The beam model of the pipe and elbow structure is shown in Figure 4.2. Three-dimensional elastic beam elements were used to model the pipe and elbow structures. The pipe section area and moments of inertia for a 34.0-inch outside diameter, 1.075-inch wall thickness pipe were input as section properties. A modulus of elasticity of 30,000,000 psi and a Poisson's ratio of 0.3 were used as the structure's material properties.

Fixed and symmetry boundary conditions were applied to the model. The end of the elbow, element 19 in Figure 4.2, was restrained against movement in all translational degrees of freedom and against rotation in directions perpendicular to the pipe axis. This end was free to rotate about it's longitudinal axis. The end of the horizontal pipe, element 1 in Figure 4.2, had symmetry boundary conditions applied. This end was restrained against movement in the direction parallel to its axis and against rotation in directions perpendicular to it's axis. These boundary conditions are shown in Figure 4.3.

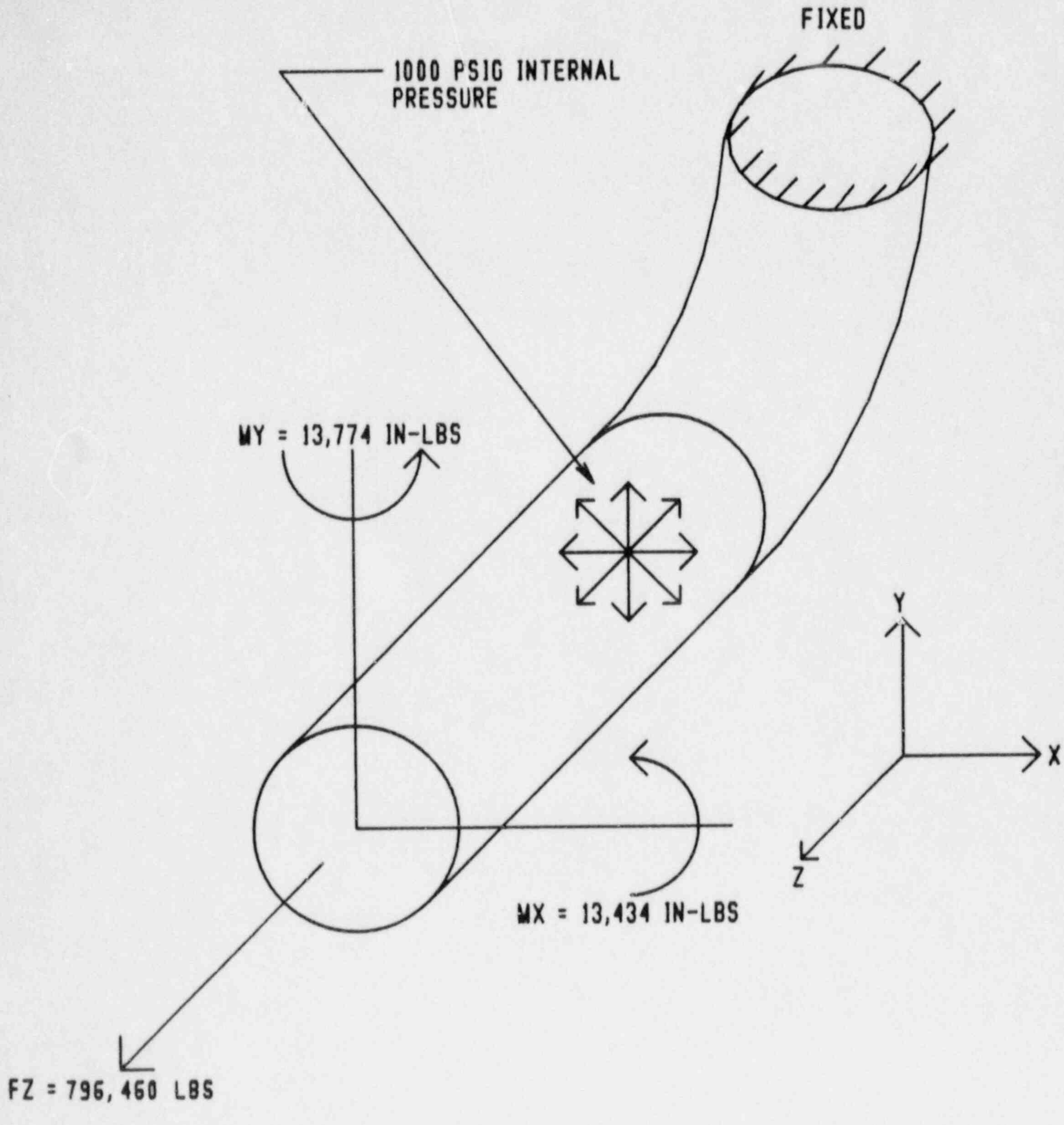


FIGURE 4.1

LOADS AND BOUNDARY CONDITIONS ON SHELL MODEL
(SAME AS FIGURE 6 IN REFERENCE 1)

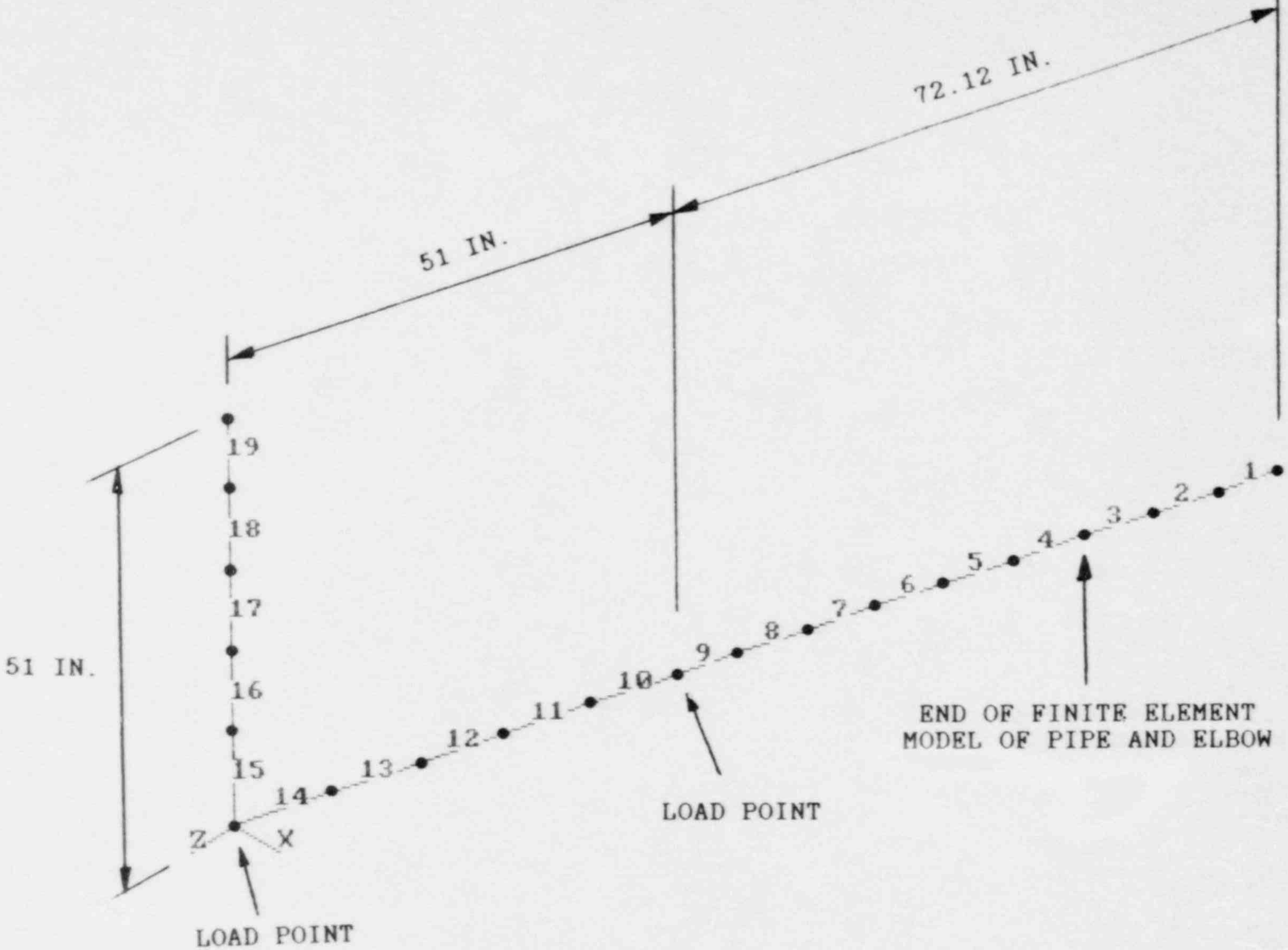


FIGURE 4.2

BEAM MODEL OF PIPE AND ELBOW
USED FOR DEADWEIGHT ANALYSIS

DEAD WEIGHT FORCES AND MOMENTS

	LOCATION	
	A	B
FX	175	175
FY	1568	2833
FZ	91	91
MX	-84,228	-10,320
MY	15,288	8328
MZ	10,284	33,816

FORCES IN POUNDS.
MOMENTS IN INCH-POUNDS.

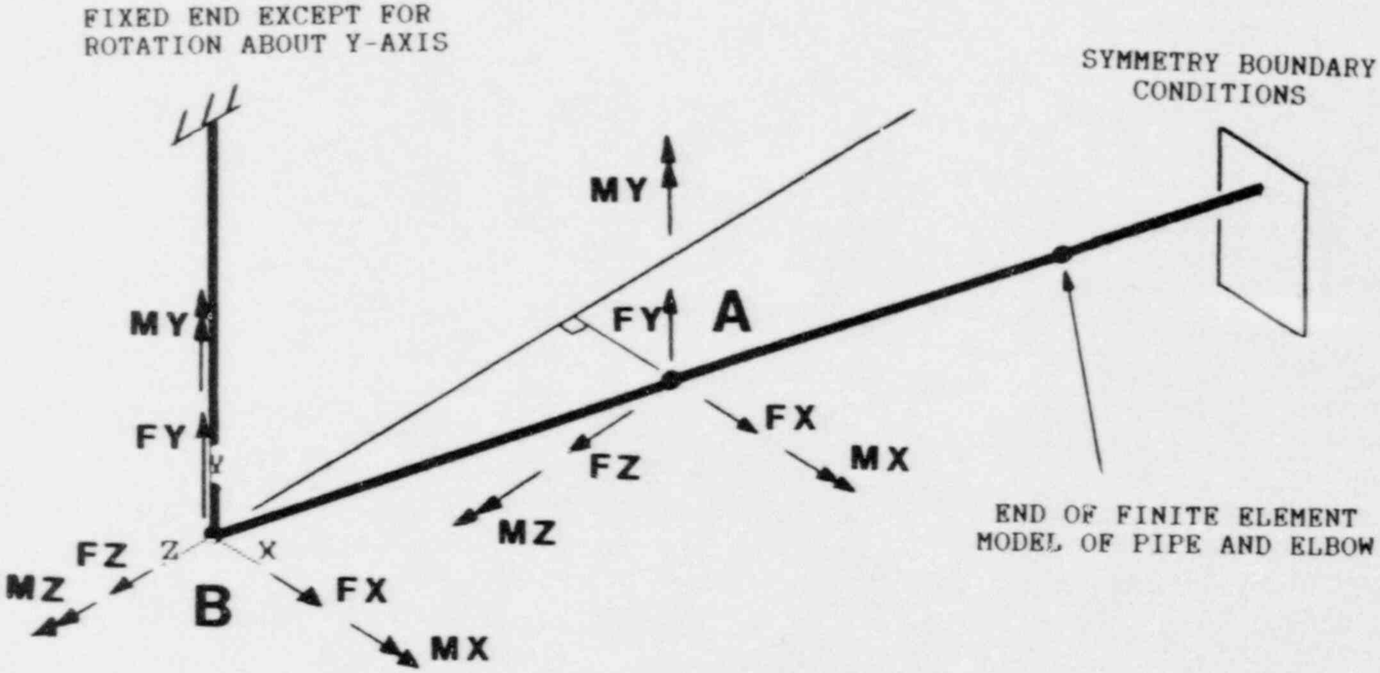


FIGURE 4.3

BOUNDARY CONDITIONS AND LOADS
ON BEAM MODEL

The pipe and elbow model was loaded with deadweight loads obtained from BG&E sources. The deadweight forces and moments applied to the model are shown in Figure 4.3. The deadweight loading applied at point A in the figure, which is the tangent between the pipe and elbow, was obtained from Reference 4. The loading applied at point B in the figure, which is in the elbow, was obtained from Reference 5.

The detailed shell model of the thin pipe structure extended to a location corresponding to the point between elements 3 and 4 of the beam model as indicated in Figure 4.2. The internal pipe forces and moments at this location, as determined from the beam analysis, were applied to the shell model.

The exaggerated displacement of the beam model under the deadweight loading is shown in Figure 4.4. The internal pipe forces and moments at the intersection between elements 3 and 4 are:

FX = 237 lbs (tension)

FY = 0

FZ = 0

MX = 0

MY = 13,774 in.-lbs

MZ = 13,435 in.-lbs

These forces and moments are in the element coordinate system where the x-axis is parallel to the pipe's longitudinal axis. A copy of the computer output indicating the internal forces and moments for each element, along with an explanation of the printed information, is presented in Appendix C.

The axial force (FZ) shown in Figure 4.1 consists of the 237 pound axial force due to deadweight plus the axial force due to the internal pressure of 1000 psig. The moments (MY and MZ) shown in the figure are those due to the deadweight.

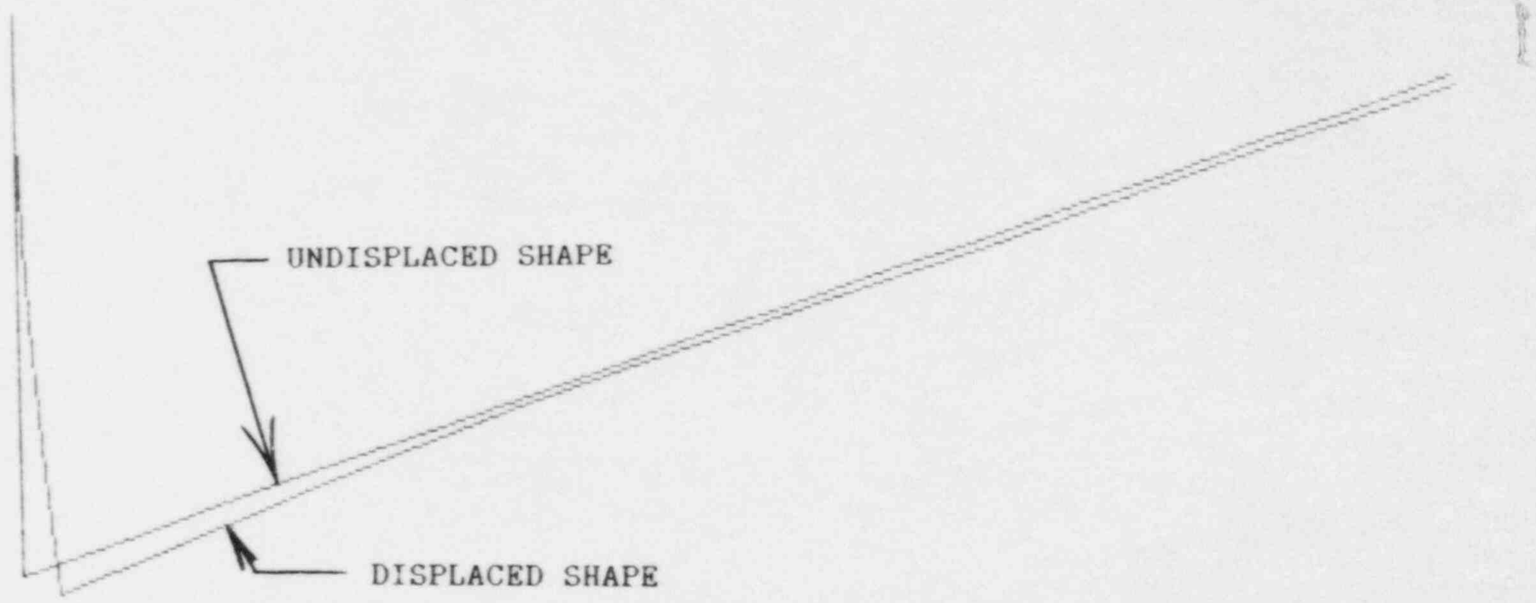


FIGURE 4.4

EXAGGERATED DISPLACEMENT OF BEAM
MODEL UNDER DEADWEIGHT LOADING

CHAPTER 5. CLOSING REMARKS

The information in this report is presented for the purpose of answering questions from the NRC about the original thin pipe region analysis discussed in Reference 1. Within this supporting document, information about the particular shell finite element used in the internal pressure and deadweight analysis of the thin pipe region is presented. Also, complete stresses around the pipe's perimeter in the plane of the thin pipe region are indicated. Finally, the beam analysis used to determine the deadweight loads that were applied to the original analysis is discussed.

It is noted that no torsional moments due to deadweight were applied to the original model of the thin pipe region because the beam analysis did not indicate that one was present. This occurred due to the symmetry boundary conditions applied at the end of the beam model which allowed rotation about the pipe's longitudinal axis. In reality, a torsional moment due to deadweight probably exists and fixing the end of the beam model would have produced one. However, realizing that a 34-inch outside diameter pipe has a large torsional constant, 24,600 in.^{4} for a 0.86 inch thick wall, small shear stresses occur due to any torque derived from the deadweight loads presented in the referenced sources. The reported state of stress in the thin pipe region is primarily caused by the 1000 psig internal pressure, and the stresses due to deadweight can be neglected since they are very low.

REFERENCES

1. Goland, Lawrence J. and Jack R. Maison. Stress Analysis Of Thin Pipe Region In No. 12 Steam Generator Main Steam Line (EB-01-1005-05), Final Report, SwRI Project No. 17-4772-861. San Antonio, Texas: Southwest Research Institute, February 1987.

2. DeSalvo, G. J. and J. A. Swanson. ANSYS Engineering Analysis System User's Manual, Revision 4.2. Houston, Pennsylvania: Swanson Analysis Systems, Inc., 1985 Edition.

3. Kohnke, Peter C. ANSYS Engineering Analysis System Theoretical Manual. Houston, Pennsylvania: Swanson Analysis Systems, Inc., 1986.

4. Telecopied information from Mr. Bernie Rudell, Baltimore Gas and Electric, to Dr. Prasad K. Nair, Southwest Research Institute, dated December 15, 1986. Subject: Pipeline inspection summary and related analyses results.

5. Telecopied information from Mr. W. Holston, Baltimore Gas and Electric, to Mr. L. J. Goland, Southwest Research Institute, dated December 15, 1986. Subject: Forces and moments on data points of elbow and pipe sections.

APPENDIX A

QUADRILATERAL SHELL (STIF63)
APPLICATION DOCUMENTATION

4.63 QUADRILATERAL SHELL

This element has both bending and membrane capabilities. Both in-plane and normal loads are permitted. The element has six degress of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes. Another four-node shell element (STIF43), has rotated material axes and in-plane pressure capabilities available.

The quadrilateral shell has options for variable thicknesses, elastic foundation supports, suppressing extra shapes, and for concentrating pressure loadings. Stress stiffening and large rotation capabilities are included. A similar element with mid-side node capability (STIF93) (but with limited applicability) is described in Section 4.93. A thick shell element (STIF94) is described in Section 4.94.

4.63.1 Input Data

The geometry, nodal point locations, loading, and the coordinate system for this element are shown in Figure 4.63.1. The element is defined by four nodal points, four thicknesses, an elastic foundation stiffness, and the orthotropic material properties. The material X-direction corresponds to the element x-direction. Properties not input default as described in Section 4.0.2.

The thickness is assumed to vary smoothly over the area of the element, with the thickness input at the four nodal points. If the element has a constant thickness, only TK(I) need be input. If the thickness is not constant, all four thicknesses must be input.

The elastic foundation stiffness (EFS) is defined as the pressure required to produce a unit normal deflection of the foundation. The elastic foundation capability is bypassed if EFS is less than, or equal to, zero.

The element loading (see Section 4.0.11) can be either surface temperatures or pressure, or a combination of both. The positive directions of pressure are as shown in Figure 4.63.1. The pressure loading may be uniformly distributed over the face of the element (KEYOPT(6)=0), or a curved shell loading (KEYOPT(6=1) consisting of an equivalent element load applied at the nodal points may be used. The latter loading produces more accurate stress results in curved shells because certain fictitious element bending stresses are eliminated.

The KEYOPT(1) option is available for neglecting the membrane stiffness or the bending stiffness, if desired. A reduced out-of-plane mass matrix is also used when the bending stiffness is neglected. The KEYOPT(2) option allows deleting the nominal in-plane rotational stiffness as described in Section 4.0.7. The KEYOPT(3) option is used to suppress the extra displacement shapes as described in Section 4.0.6.

The KEYOPT(7) option allows a reduced mass matrix formulation (lumping procedure with off-diagonal terms with rotational degrees of freedom terms deleted). This option is useful for improved bending stresses in thin members under mass loading. A summary of the shell element parameters is given in Table 4.63.1. A general description of element input, including the special features, is given in Section 4.0.2.

4.63.2 Output Data

a) Printout - The printout associated with the shell element is summarized in Table 4.63.2. Several items are illustrated in Figure 4.63.2. A general description of element printout is given in Section 4.0.3. Printout includes the moments about the x face (MX), the moments about the y face (MY), and the twisting moment (MXY). The moments are calculated per unit length in the element coordinate system. Edge stress output (KEYOPT(4)) is relative to an edge coordinate reference, i.e., x is along the edge, y is normal to the edge. Edge stresses are based on the average force in the half of the element nearest that edge and are not accurate if a significant force gradient exists across the element and are not available for triangular shapes. Centroid and nodal stress data are accurate.

b) Post Data - The post data associated with the shell element is shown below. The data are written on File12 if requested, as described in Section 4.0.4.

```
    1    TX                       21-24 SX,SY,SXY,SZ(I)<TOP>      129-131 SIG1,SIG2,SIG3<TOP>
    2    TY                       25-36 21-24 @ (J-L)<TOP>        132-133 S.I.,SIGE<TOP>
    3    TXY                      37-68 21-36 @ <MID,BOT>         134-143 129-133 @ <MID,BOT>
  4-5    SPARE,SPARE            2---------------------------      144-146 XC,YC,ZC
  6-8    MX,MY,MXY                69-71 SIG1,SIG2,SIG3(I)<TOP>    147-148 AREA,TTOP
1---------------- -------         72-73 S.I.,SIGE(I)<TOP>         149-151 TBOT,PRESS(1,2)
  9-12   SX,SY,SXY,SZ<TOP>        74-88 69-73 @ (J-L)<TOP>      3----------------------------
 13-20   9-12 @ <MID,BOT>         89-128 69-88 @ <MID,BOT>
```

4.63.3 Theory

The membrane stiffness is the same as for the membrane shell element (STIF41), including the extra shapes. The bending stiffness is formed from the bending stiffness of four triangular shell elements (STIF53). Two triangles have one diagonal of the element as a common side and two triangles have the other diagonal of the element as a common side. The stiffness is obtained from the sum of the four stiffnesses divided by two.

4.63.4 Assumptions and Restrictions

Zero area elements are not allowed. This occurs most often whenever the elements are not numbered properly. Zero thickness elements or elements tapering down to a zero thickness at any corner are not allowed. The applied transverse thermal gradient is assumed to be linear through the thickness and uniform over the shell surface.

An assemblage of flat shell elements can produce a good approximation to a curved shell surface provided that each flat element does not extend over more than a 15° arc. If an elastic foundation stiffness is input, one-fourth of the total is applied at each node. Shear deflection is not included in this thin-shell element.

A triangular element may be formed by defining duplicate K and L node numbers as described in Section 4.0.9. The extra shapes are automatically deleted for triangular elements so that the membrane stiffness reduces to a constant strain formulation.

The four nodal points defining the element should lie in an exact flat plane; however, a small out-of-plane tolerance is permitted so that the element may have a slightly warped shape. A slightly warped element will produce a warning message in the printout. If the warpage is too severe, a fatal message results and a triangular element should be used, see Section 4.0.9. For large deflection analyses, the element is most accurate in the rectangular and triangular configurations. Initially warped quadrilateral elements should not be used in large deflection analyses.

TABLE 4.63.1

QUADRILATERAL SHELL

ELEMENT NAME	STIF63	
NO. OF NODES	4	I,J,K,L
DEGREES OF FREEDOM PER NODE	6	UX,UY,UZ,ROTX,ROTY,ROTZ
REAL CONSTANTS	5	TK(I),TK(J),TK(K),TK(L),EFS (TK(J),TK(K),TK(L) DEFAULT TO TK(I))
MATERIAL PROPERTIES	7	EX,EY,ALPX,ALPY,NUXY,DENS,GXY (DIRECTION I-J IS X)
PRESSURES	2	P1,P2
TEMPERATURES	2	TTOP,TBOTTOM
SPECIAL FEATURES	STRESS STIFFENING, LARGE ROTATION	
KEYOPT(1)	0 - BENDING AND MEMBRANE STIFFNESS 1 - MEMBRANE STIFFNESS ONLY 2 - BENDING STIFFNESS ONLY	
KEYOPT(2)	0 - ZIENKIEWICZ IN-PLANE ROTATIONAL STIFFNESS 1 - NO IN-PLANE ROTATIONAL STIFFNESS	
KEYOPT(3)	0 - INCLUDE EXTRA DISPLACEMENT SHAPES 1 - SUPPRESS EXTRA DISPLACEMENT SHAPES	
KEYOPT(4)	0 - NO EDGE PRINTOUT N - EDGE PRINTOUT AT EDGE N (N = 1,2,3 OR 4) 5 - EDGE PRINTOUT AT ALL FOUR EDGES	
KEYOPT(5)	0 - BASIC ELEMENT PRINTOUT 2 - NODAL STRESS PRINTOUT (ADDS 15 MORE LINES PER ELEMENT)	
KEYOPT(6)	0 - FLAT SHELL PRESSURE LOADING 1 - CURVED SHELL PRESSURE LOADING (MUST BE USED IF KEYOPT(1) = 1)	
KEYOPT(7)	0 - CONSISTENT MASS MATRIX 1 - REDUCED MASS MATRIX	

****

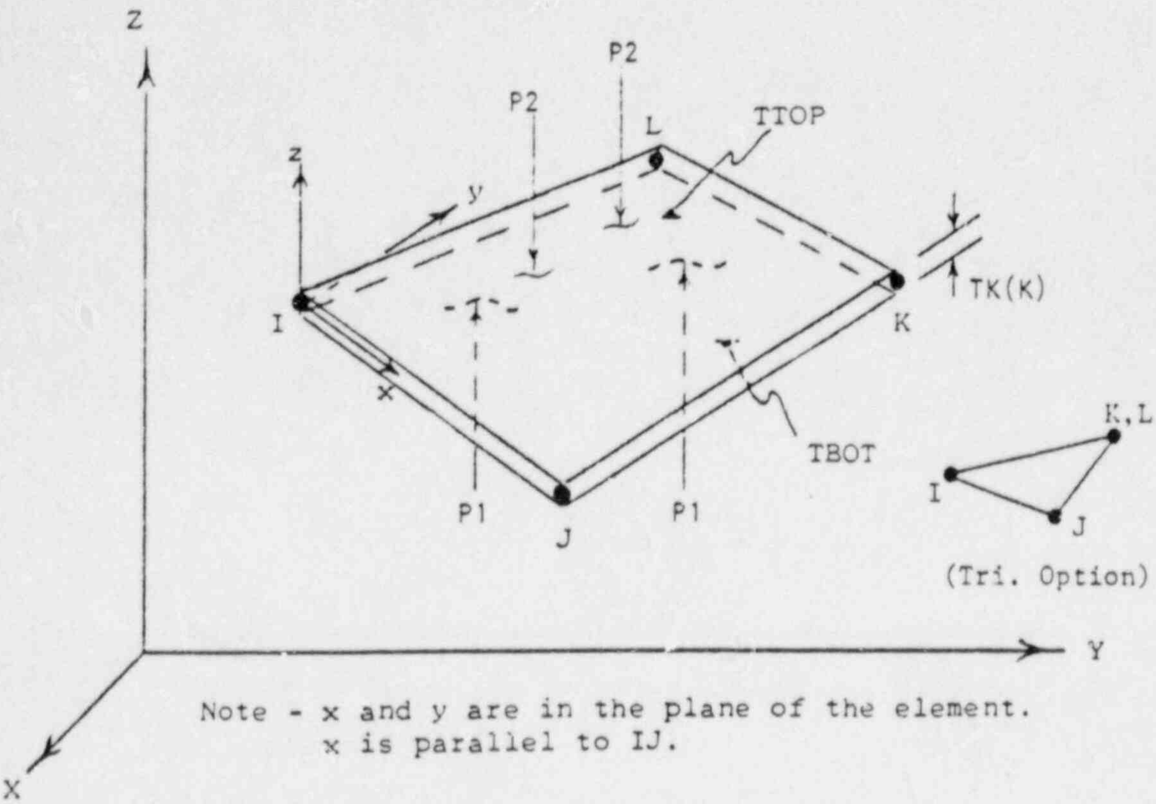


Figure 4.63.1 Quadrilateral Shell

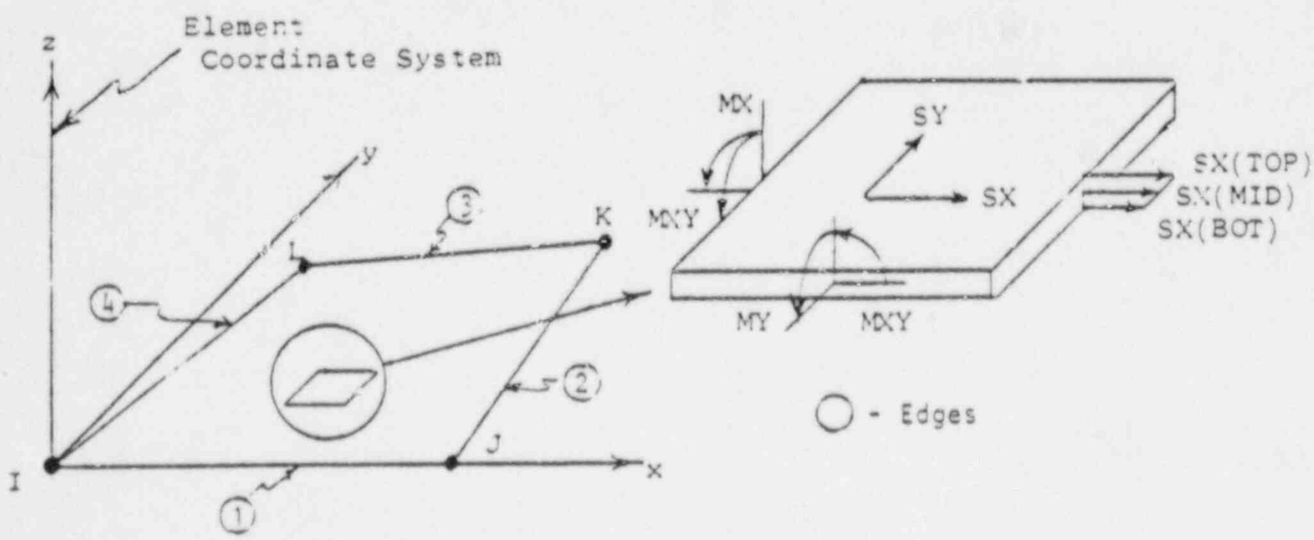


Figure 4.63.2 Quadrilateral Shell Output

TABLE 4.63.2

QUADRILATERAL SHELL

ELEMENT PRINTOUT EXPLANATIONS

LABEL	NUMBER OF CONSTANTS	EXPLANATION
LINE 1		
EL	1	ELEMENT NUMBER
NODES	4	NODES - I,J,K,L
MAT	1	MATERIAL NUMBER
AREA	1	AREA
TTOP,TBOT	2	SURFACE TEMPERATURES - TOP,BOTTOM
LINE 2		
XC,YC,ZC	3	GLOBAL X,Y,Z LOCATION OF CENTROID
PRESS	2	PRESSURES - P1,P2
LINE 3		
MX,MY,MXY	3	MOMENTS IN ELEMENT X AND Y DIRECTIONS
LINE 4 (LINES 4 AND 5 REPEAT FOR EACH LOCATION)		
LOC		TOP, MIDDLE, OR BOTTOM
SX,SY,SXY,SZ	4	COMBINED MEMBRANE AND BENDING STRESSES (ELEMENT COORDINATES)
SIG1,SIG2,SIG3	3	PRINCIPAL STRESSES
LINE 5		
S.I.	1	STRESS INTENSITY
SIGE	1	EQUIVALENT STRESS
EDGE PRINTOUT (PRINTED ONLY IF KEYOPT(4) = 1,2,3,4, OR 5)		
LOC	2	EDGE NODES
FORCES/LENGTH	6	TX,TY,TXY AT EDGE (TX = SX * THICKNESS, ETC.) MX,MY,MXY AT EDGE
STRESSES	6	SX,SY,SXY AT EDGE, (MX,MY,MXY AT EDGE) * (6/(THICKNESS**2)).

NODAL STRESS SOLUTION (PRINTED ONLY IF KEYOPT(5) = 2. REPEATS EACH LOC.)

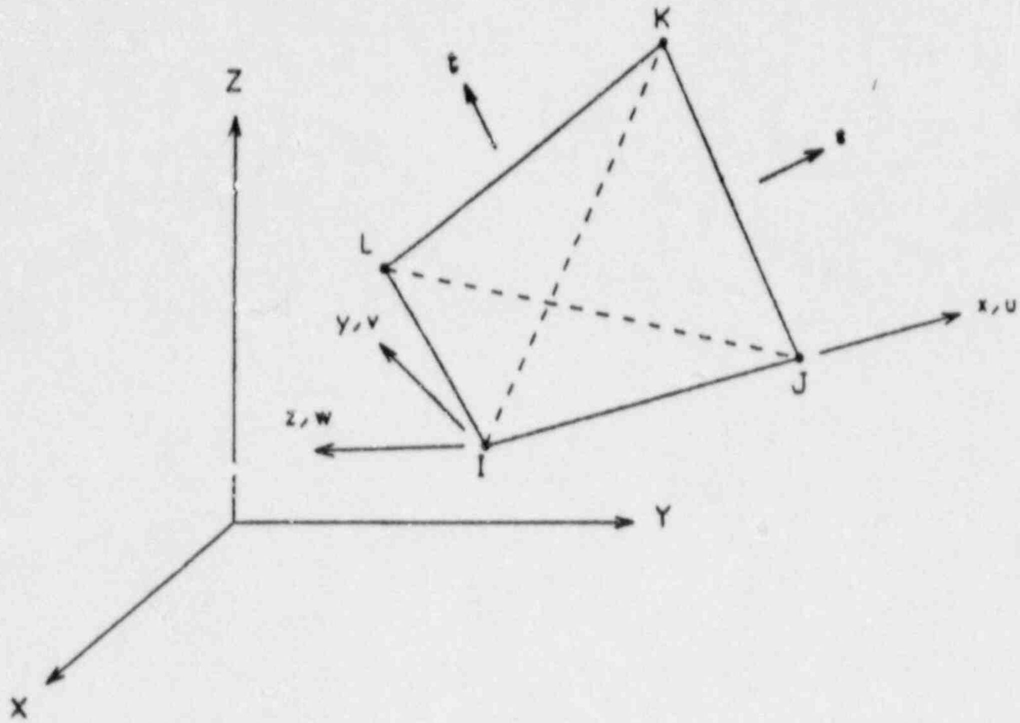
(GIVES SX,SY,SXY,SZ,SIG1,SIG2,SIG3,S.I.,SIGE AT EACH NODE)

****

APPENDIX B

QUADRILATERAL SHELL (STIF63)
THEORETICAL DOCUMENTATION

2.63 STIF63 - QUADRILATERAL SHELL ELEMENT



		SHAPE FUNCTIONS	INTEGRATION POINTS
Stiffness Matrix	Membrane	$u = \frac{1}{4}(u_I(1-s)(1-t)$ $+ u_J(1+s)(1-t)$ $+ u_K(1+s)(1+t)$ $+ u_L(1-s)(1+t))$ (and, if extra shapes are included (KEYOPT(3)=0) and element has 4 unique nodes) $+ u_1(1-s^2) + u_2(1-t^2)$ $v = \frac{1}{4}(v_I(1-s)$ - - - (similar to u) Reference: Wilson(38)	3 x 3

		SHAPE FUNCTIONS	INTEGRATION POINTS
Stiffness Matrix	Bending	Four triangles that are overlaid are used. These triangles connect nodes IJK, IJL, KLI, and KLJ. w is not explicitly defined. (DKT Triangle, Batoz (56)), Razzoque (57))	3 (for each triangle)
Mass Matrix	Membrane	$u = \frac{1}{4}(u_I(1-s)(1-t) + u_J(1+s)(1-t) + u_K(1+s)(1+t) + u_L(1-s)(1+t)$ $v = \frac{1}{4}(v_I(1-s)$ - - - (similar to u)	3 x 3
	Bending	Shape functions are described in the text of this section. Reference: Zienkiewicz (39)	3 (for each triangle)
Stress Stiffness Matrix		Same as mass matrix	Same as mass matrix
Thermal Load Vector		Same as stiffness matrix	Same as stiffness matrix

		SHAPE FUNCTIONS	INTEGRATION POINTS
Pressure Load Vector	Flat Shell Pressure Loading (KEYOPT(6)=0) (Load Vector includes moments)	Same as mass matrix	Same as mass matrix
	Curved Shell Pressure Loading (KEYOPT(6)=1) (Load Vector excludes moments)	One fourth (one third for triangles) of the total pressure times the area is applied to each node normal to the element.	None

Element Temperature Distribution: Linear thru thickness, constant in the plane of the element

Nodal Temperature Distribution: Constant thru thickness, varies bilinearly in the plane of the element

Pressure Distribution: Constant over the plane of the element

Other Applicable Sections

Section 2.0.1 has a complete derivation of the matrices and load vectors of a general stress analysis element.

The basic in-plane stiffness matrix is the same as developed with STIF42 in section 2.42. Membrane (in-plane) rotational stiffness at the nodes is discussed with STIF53 in section 2.53. See section 2.53 also for a discussion of curved shell versus flat shell pressure loading.

Out-of-plane Shape Functions

The shape functions for the out-of-plane displacements are developed next. These are used to develop the element mass, stress stiffness, and pressure load vector. First, various geometrical quantities need to be defined for Figure 2.63.1:

$$a_i = x_j y_k - x_k y_j$$

$$a_j = x_k y_i - x_i y_k \qquad (2.63.1)$$

$$a_k = x_i y_j - x_j y_i$$

$$b_i = y_j - y_k$$

$$b_j = y_k - y_i \qquad (2.63.2)$$

$$b_k = y_i - y_j$$

$$c_i = x_k - x_j$$

$$c_j = x_i - x_k \qquad (2.63.3)$$

$$c_k = x_j - x_i$$

The area of the triangle (Δ) is:

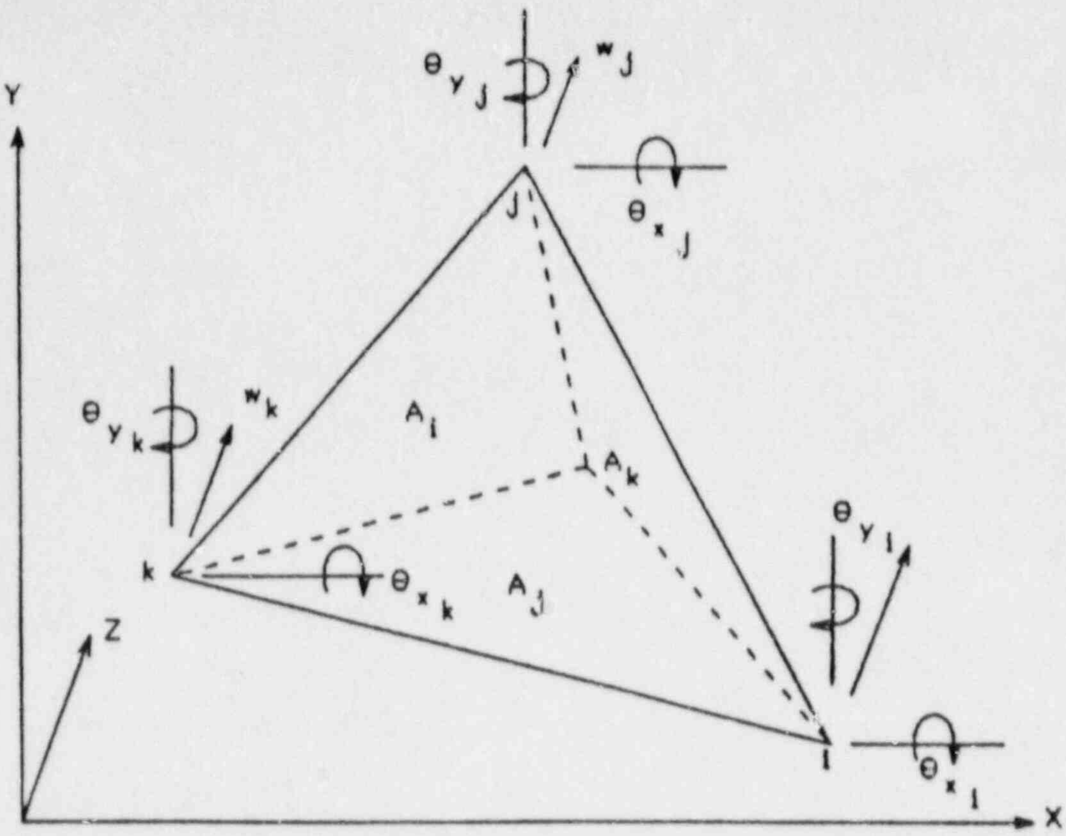


Figure 2.63.1 Triangular Shape Functions

$$\Delta = (a_i + a_j + a_k)/2 \tag{2.63.4}$$

Next, the area coordinates are defined. These locate a point within the triangle, and are given as:

$$\begin{aligned} L_i &= (a_i + b_i x + c_i y)/2\Delta \\ L_j &= (a_j + b_j x + c_j y)/2\Delta \\ L_k &= (a_k + b_k x + c_k y)/2\Delta \end{aligned} \tag{2.63.5}$$

This is equivalent to:

$$\begin{aligned} L_i &= A_i/\Delta \\ L_j &= A_j/\Delta \\ L_k &= A_k/\Delta \end{aligned} \tag{2.63.6}$$

where A_i, A_j, and A_k are given the areas of the subtriangles in Figure 2.63.1. It becomes clear that the three L quantities cannot be independent; specifically,

$$L_i + L_j + L_k = 1. \tag{2.63.7}$$

Next, the shape functions may be set up. This uses the same approach as reported in section 10.6 of Reference 39. The deformation of the element is broken up into two parts:

1. The part that does not cause bending (i.e., the rigid body motion as defined by the nodal translations).
2. The part that causes bending.

Part 1 may be simply characterized as:

$$w_1 = L_i w_i + L_j w_j + L_k w_k \tag{2.63.8}$$

Part 2 is:

$$w_2 = F_{xi}\theta^*_{xi} + F_{yi}\theta^*_{yi} + F_{xj}\theta^*_{xj} + F_{yj}\theta^*_{yj} + F_{xk}\theta^*_{xk} + F_{xk}\theta^*_{xk} \tag{2.63.9}$$

where: θ^*_x = rotation of node about element x-axis away from the rigid body motion plane

θ^*_y = rotation of node about element y-axis away from the rigid body motion plane

$$\theta^*_x = \theta_x + \frac{\partial w_1}{\partial y} \tag{2.63.10}$$

$$\theta_y^* = \theta_y + \frac{\partial w_1}{\partial x} \qquad (2.63.11)$$

θ_x = rotation of node about element x-axis

θ_y = rotation of node about element y-axis

$$\frac{\partial w_1}{\partial x} = (b_i w_i + b_j w_j + b_k w_k)/2\Delta \qquad (2.63.12)$$

$$\frac{\partial w_1}{\partial y} = (c_i w_i + c_j w_j + c_k w_k)/2\Delta \qquad (2.63.13)$$

and:

$$F_{xi} = b_k \; \psi_{ij} + b_j \; \psi_{ik} \qquad (2.63.14)$$

$$F_{yi} = -c_k \; \psi_{ij} + c_j \; \psi_{ik} \qquad (2.63.15)$$

where:

$$\psi_{ij} = L_i^2 \; L_j + \frac{1}{2} \; L_i \; L_j \; L_k \qquad (2.63.16)$$

$$\psi_{ik} = L_i^2 \; L_k + \frac{1}{2} \; L_i \; L_j \; L_k \qquad (2.63.17)$$

The other terms may be arrived at by cyclical permutation of the indices.

Foundation Stiffness

If K_f, the foundation stiffness, is input, the out of plane stiffness matrix is augmented by three or four springs to ground. The number of springs is equal to the number of distinct nodes, and their direction is normal to the plane of the element. The value of each spring is:

$$K_{f,i} = \frac{\Delta K_f}{N_d}, \tag{2.63.18}$$

where: $K_{f,i}$ = normal stiffness at node i

Δ = element area

K_f = foundation stiffness (input quantity EFS)

N_d = number of distinct nodes

Warping

If all four nodes are not defined to be in the same flat plane (or if an initially flat element looses its flatness due to large displacements (KAY,6,1)), additional calculations are performed in STIF63. The purpose of the additional calculations is to convert the matrices and load vectors of the element from the points on the flat plane in which the element is derived to the actual node points. Physically, this may be thought of as adding short rigid offsets between the flat plane of the element and the actual nodes. When these offsets are required, it implies that the element is not flat, but rather it is "warped". To account for the warping, the following procedure is used: First, the normal to element is computed by taking the vector cross-product (the common normal) between the vector from node I to node K and the vector from node J to node L. Then, the check can be made to see if extra calculations are needed to account for warped elements. This check consists of comparing the normal to each of the four element corners with the element normal as defined above. The corner normals are computed by taking the vector cross-product of vectors representing the two adjacent edges. All vectors are normalized to 1.0. If any of the three global

Cartesian components of each corner normal differs from the equivalent component of the element normal by more than .00001, then the element is considered to be warped.

A warping factor is computed as:

$$\phi = \frac{D}{t} \tag{2.63.19}$$

where: D = distance from the first node to the fourth node parallel to the element normal.

t = average thickness of the element

If: $\phi \leq .10$ no warning message is printed

$.10 \leq \phi \leq 1.0$ a warning message is printed

$1.0 < \phi$ a message suggesting the use of triangles is printed and the run terminates

To account for the warping, the following matrix is developed:

$$[W] = \left[\begin{array}{c|c|c|c} w_1 & 0 & 0 & 0 \\ \hline 0 & w_2 & 0 & 0 \\ \hline 0 & 0 & w_3 & 0 \\ \hline 0 & 0 & 0 & w_4 \end{array}\right] \tag{2.63.20}$$

where:

$$[w_i] = \begin{bmatrix} 1 & 0 & 0 & 0 & DZ_i & 0 \\ 0 & 1 & 0 & DZ_i & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \qquad (2.63.21)$$

where: DZ_i = offset from the average plane at node i

and the degrees of freedom are in the usual order of UX, UY, UZ, ROTX, ROTY, and ROTZ. To ensure the location of the average plane goes through the middle of the element, the following condition is met.

$$DZ_1 + DZ_2 + DZ_3 + DZ_4 = 0 \qquad (2.63.22)$$

Finally, the element matrices are converted in the usual way to global Cartesian coordinates:

$$[K_e] = [T_R]^T \, [W]^T \, [K_\ell] \, [W] \, [T_R] \qquad (2.63.23)$$

$$\{F_e^{th}\} = [T_R]^T \, [W]^T \, \{F_\ell^{th}\} \qquad (2.63.24)$$

where: $[K_e]$ = element matrix in global Cartesian coordinates

$[T_R]$ = local to global conversion matrix (uses average normal previously computed)

$[K_\ell]$ = element matrix in element coordinate system as derived directly from the shape functions

$\{F_e^{th}\}$ = element thermal load vector in global Cartesian coordinates

$\{F_\ell^{th}\}$ = element thermal load vector in element coordinate systems as derived from the shape functions.

The mass matrix, the stress stiffness matrix, and the element pressure vector are handled similarly. For elements with no warping, the same procedure is used except that [W] is not included.

Mass Matrix

The element mass matrix is computed as illustrated in Section 2.0. If the reduced mass matrix option (KEYOPT(7)=1) is requested, all rotational terms are set to zero.

Stress Output

For the centroidal stress output, the MX, MY, and MXY quantities are computed from the bending stresses.

If edge stresses are requested (KEYOPT(4)>0) the following logic is used: First, the element reaction forces for the two nodes on the edge are converted from global coordinates to edge coordinates. x' and y' represent the edge coordinate system. Then,

$$TX = \frac{(FX_N - FX_M)\ L}{A} \qquad (2.63.25)$$

$$TY = -\frac{FY_N + FY_M}{L} \qquad (2.63.26)$$

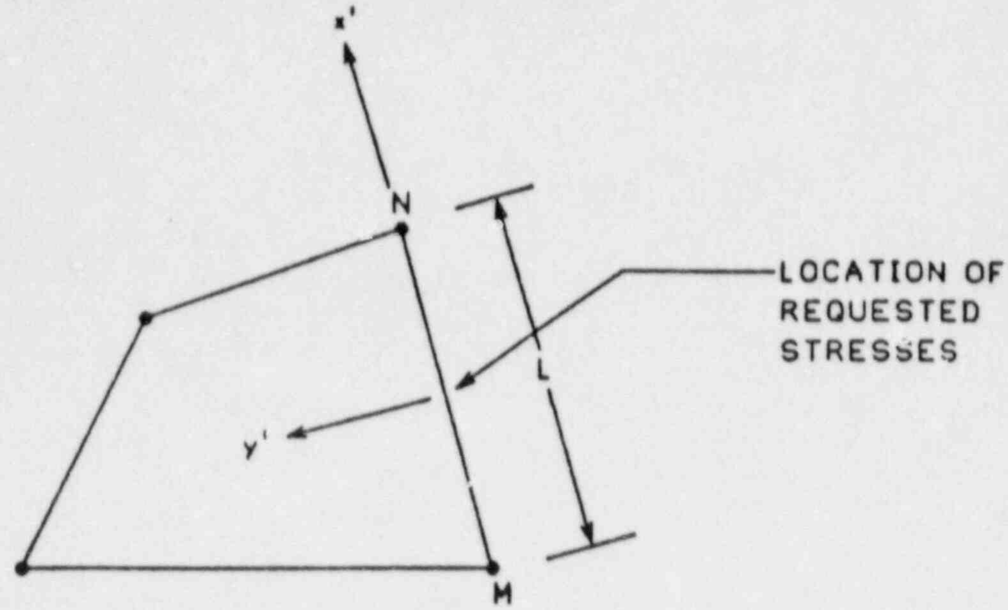


Figure 2.63.2 Edge Stress Diagram

$$TXY = -\frac{FX_N + FX_M}{L} \tag{2.63.27}$$

$$MX = \frac{(MY_N - MY_M)\ L}{A} \tag{2.63.28}$$

$$MY = \frac{MX_N + MX_M}{L} \tag{2.63.29}$$

$$MXY = \frac{FZ_N - FZ_M}{4} - \frac{MY_N + MY_M}{2L} \tag{2.63.30}$$

where: L = length, as shown

A = element area

TX = force per unit length in shell in x direction of edge coordinate system

etc.

FX_N = force at node N in x direction of edge coordinate system

etc.

Note the use of two different conventions in the above six equations. MX on the left side of the equal sign refers to the shell sign convention, that is, moment caused by stresses in the y-direction which vary thru the thickness. MX_N on the right side of the equal sign refers to the ANSYS sign convention on forces and moments, that is, the moment at node N in the direction of one's fingers of the right hand if the thumb is pointing in the positive x-direction.

It may be observed that two of the above quantities (TX and MX) are not as reliable as the others because they involve the area of the element, which is a function of more than the one edge being studied. These three tend to be most reliable for rectangular elements when the stress field is not changing rapidly.

Finally, these force quantities are converted to stress quantities by the following:

$$SX = \frac{TX}{t} \tag{2.63.31}$$

$$SY = \frac{TY}{t} \tag{2.63.32}$$

$$SXY = \frac{TXY}{t} \tag{2.63.33}$$

$$\sigma_x^B = \frac{6\ MX}{t^2} \tag{2.63.34}$$

$$\sigma_y^B = \frac{6\ MY}{t^2} \tag{2.63.35}$$

$$\sigma_{xy}^B = \frac{6\ MXY}{t^2} \tag{2.63.36}$$

where: t = average thickness of element at nodes M and N

σ_i^B = stress due to bending only

APPENDIX C

DEADWEIGHT ANALYSIS BEAM
MODEL RESULTS OUTPUT

TABLE 5.4.2

THREE-DIMENSIONAL ELASTIC BEAM

ELEMENT PRINTOUT EXPLANATIONS

LABEL	NUMBER OF CONSTANTS	EXPLANATION
LINE 1		
EL	1	ELEMENT NUMBER
NODES	2	NODES - I,J
MAT	1	MATERIAL NUMBER
LINE 2		
TCENT,TTOPZ, TBOTY	3	TEMPERATURES - CENTER,TOP-Z,BOT-Y
QZ,QY	2	PRESSURES ON FACES 1 AND 2
LINE 3		
END I		RESULTS AT END I
SDIR	1	AXIAL (DIRECT) STRESS AT END
SBZ	1	BENDING STRESS ON ELEMENT +Z SIDE OF BEAM AT END
SBY	1	BENDING STRESS ON ELEMENT -Y SIDE OF BEAM AT END
SIG1	1	MAXIMUM STRESS IN RECTANGULAR BEAM AT END
SIG3	1	MINIMUM STRESS IN RECTANGULAR BEAM AT END

LINE 5 SAME AS LINE 2 EXCEPT AT END J

LINE 6-8 MEMBER FORCES AND MOMENTS (PRINTED ONLY IF KEYOPT(6) = 1)

THE SIX MEMBER FORCE AND MOMENT COMPONENTS (FX,FY,FZ,MX,MY,MZ) ARE PRINTED FOR EACH NODE (IN THE ELEMENT COORDINATE SYSTEM)

****

```
 ***** ELEMENT STRESSES *****  TIME =  0.000000E+00   LOAD STEP=    1   ITERATION=     1   CUM. ITER.=     1

EL=    1  NODES=     1     2  MAT=  1  TCENT,TTOPZ,TBOTY=    0.0     0.0     0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
END J     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
 STATIC FORCES ON NODE          1  -237.494       0.318909E-10 -0.270768E-10  0.521009E-10   13774.2        13435.0
 STATIC FORCES ON NODE          2   237.494      -0.226900E-08 -0.119680E-08 -0.203098E-06  -13774.2       -13435.0

EL=    2  NODES=     2     3  MAT=  1  TCENT,TTOPZ,TBOTY=    0.0     0.0     0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
END J     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
 STATIC FORCES ON NODE          2  -237.494       0.227818E-08  0.115299E-08  0.202887E-06   13774.2        13435.0
 STATIC FORCES ON NODE          3   237.494      -0.698088E-08 -0.455184E-09 -0.439241E-06  -13774.2       -13435.0

EL=    3  NODES=     3     4  MAT=  1  TCENT,TTOPZ,TBOTY=    0.0     0.0     0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
END J     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
 STATIC FORCES ON NODE          3  -237.494       0.694571E-08  0.515904E-09  0.439501E-06   13774.2        13435.0
 STATIC FORCES ON NODE          4   237.494      -0.124332E-07 -0.936232E-09 -0.778830E-06  -13774.2       -13435.0

EL=    4  NODES=     4     5  MAT=  1  TCENT,TTOPZ,TBOTY=    0.0     0.0     0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
END J     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
 STATIC FORCES ON NODE          4  -237.494       0.123994E-07  0.828759E-09  0.778648E-06   13774.2        13435.0
 STATIC FORCES ON NODE          5   237.494       0.181449E-07 -0.133155E-08  0.964994E-06  -13774.2       -13435.0

EL=    5  NODES=     5     6  MAT=  1  TCENT,TTOPZ,TBOTY=    0.0     0.0     0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
END J     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
 STATIC FORCES ON NODE          5  -237.494      -0.181714E-07  0.143584E-06 -0.965164E-06   13774.2        13435.0
 STATIC FORCES ON NODE          6   237.494       0.127756E-07  0.174463E-07  0.172418E-05  -13774.2       -13435.0

EL=    6  NODES=     6     7  MAT=  1  TCENT,TTOPZ,TBOTY=    0.0     0.0     0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
END J     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
 STATIC FORCES ON NODE          6  -237.494      -0.127416E-07 -0.172383E-07 -0.172442E-05   13774.2        13435.0
 STATIC FORCES ON NODE          7   237.494       0.918063E-08 -0.302655E-08  0.368974E-06  -13774.2       -13435.0

EL=    7  NODES=     7     8  MAT=  1  TCENT,TTOPZ,TBOTY=    0.0     0.0     0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
END J     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
 STATIC FORCES ON NODE          7  -237.494      -0.916305E-08  0.267640E-08 -0.369073E-06   13774.2        13435.0
 STATIC FORCES ON NODE          8   237.494       0.132101E-08  0.242104E-07  0.142914E-05  -13774.2       -13435.0

EL=    8  NODES=     8     9  MAT=  1  TCENT,TTOPZ,TBOTY=    0.0     0.0     0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
END J     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
 STATIC FORCES ON NODE          8  -237.494      -0.151919E-08 -0.240483E-07 -0.142915E-05   13774.2        13435.0
 STATIC FORCES ON NODE          9   237.494      -0.659227E-08 -0.854641E-09 -0.445307E-06  -13774.2       -13435.0

EL=    9  NODES=     9    10  MAT=  1  TCENT,TTOPZ,TBOTY=    0.0     0.0     0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
END J     SDIR= -2.1358        SBZ= -15.524        SBY= -15.142       SIG1=  28.530       SIG3= -32.802
 STATIC FORCES ON NODE          9  -237.494       0.694779E-08  0.613208E-09  0.445308E-06   13774.2        13435.0
 STATIC FORCES ON NODE         10  -63.6015      -0.480864E-08   228.819        12344.0       -13774.4        3597.86

EL=   10  NODES=    10    11  MAT=  1  TCENT,TTOPZ,TBOTY=    0.0     0.0     0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -2.5029        SBZ=  1.7058        SBY=  73.216       SIG1=  72.419       SIG3= -77.425
END J     SDIR= -2.5029        SBZ=-0.51253        SBY=  55.191       SIG1=  53.201       SIG3= -58.206
 STATIC FORCES ON NODE         10  -111.398       -1568.00       -319.819        71284.6       -1513.56       -13881.9
 STATIC FORCES ON NODE         11   111.398        1568.00        319.819       -55874.6       -454.753        18164.7
```

```
EL=   11  NODES=    11    12  MAT=  1  TCENT.TTOPZ.TBOTY=     0.0      0.0      0.0  QZ.QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -2.5029        SBZ=-0.51253        SBY=  55.181       SIG1=  53.201      SIG3= -58.206
END J     SDIR= -2.5029        SBZ= -2.7309        SBY=  37.166       SIG1=  37.394      SIG3= -42.399
 STATIC FORCES ON NODE        11  -111.398       -1568.00        -319.819        55874.6         454.753        -18164.7
 STATIC FORCES ON NODE        12   111.398        1568.00         319.819       -40465.3       -2423.06         22447.5

EL=   12  NODES=    12    13  MAT=  1  TCENT,TTOPZ,TBOTY=     0.0      0.0      0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -2.5029        SBZ= -2.7309        SBY=  37.166       SIG1=  37.394      SIG3= -42.399
END J     SDIR= -2.5029        SBZ= -4.9493        SBY=  19.140       SIG1=  21.587      SIG3= -26.592
 STATIC FORCES ON NODE        12  -111.398       -1568.00        -319.819        40465.3        2423.06        -22447.5
 STATIC FORCES ON NODE        13   111.398        1568.00         319.819       -25055.9       -4391.38         26730.4

EL=   13  NODES=    13    14  MAT=  1  TCENT,TTOPZ,TBOTY=     0.0      0.0      0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -2.5029        SBZ= -4.9493        SBY=  19.140       SIG1=  21.587      SIG3= -26.592
END J     SDIR= -2.5029        SBZ= -7.1676        SBY=  1.1150       SIG1=  5.7798      SIG3= -10.786
 STATIC FORCES ON NODE        13  -111.398       -1568.00        -319.819        25055.9        4391.38        -26730.4
 STATIC FORCES ON NODE        14   111.398        1568.00         319.819       -9646.53       -6359.69         31013.2

EL=   14  NODES=    14    15  MAT=  1  TCENT,TTOPZ,TBOTY=     0.0      0.0      0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -2.5029        SBZ= -7.1676        SBY=  1.1150       SIG1=  5.7798      SIG3= -10.786
END J     SDIR= -2.5029        SBZ= -9.3860        SBY= -16.910       SIG1=  23.793      SIG3= -28.799
 STATIC FORCES ON NODE        14  -111.398       -1568.00        -319.819        9646.53        6359.69        -31013.2
 STATIC FORCES ON NODE        15   111.398        1568.00         319.819        5762.83       -8328.00         35296.0

EL=   15  NODES=    15    16  MAT=  1  TCENT,TTOPZ,TBOTY=     0.0      0.0      0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -39.579        SBZ= -5.1361        SBY= -77.892       SIG1=  43.449      SIG3= -122.61
END J     SDIR= -39.579        SBZ= -9.8588        SBY= -81.185       SIG1=  51.464      SIG3= -130.62
 STATIC FORCES ON NODE        15  -286.398       -4401.00        -410.819        4557.17     -0.144524E-10   -69112.0
 STATIC FORCES ON NODE        16   286.398        4401.00         410.819       -8747.52      0.144524E-10    72033.3

EL=   16  NODES=    16    17  MAT=  1  TCENT,TTOPZ,TBOTY=     0.0      0.0      0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -39.579        SBZ= -9.8588        SBY= -81.185       SIG1=  51.464      SIG3= -130.62
END J     SDIR= -39.579        SBZ= -14.582        SBY= -84.477       SIG1=  59.479      SIG3= -138.64
 STATIC FORCES ON NODE        16  -286.398       -4401.00        -410.819        8747.52     -0.144524E-10   -72033.3
 STATIC FORCES ON NODE        17   286.398        4401.00         410.819       -12937.9      0.144524E-10    74954.6

EL=   17  NODES=    17    18  MAT=  1  TCENT,TTOPZ,TBOTY=     0.0      0.0      0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -39.579        SBZ= -14.582        SBY= -84.477       SIG1=  59.479      SIG3= -138.64
END J     SDIR= -39.579        SBZ= -19.304        SBY= -87.769       SIG1=  67.495      SIG3= -146.65
 STATIC FORCES ON NODE        17  -286.398       -4401.00        -410.819        12937.9     -0.144595E-10   -74954.6
 STATIC FORCES ON NODE        18   286.398        4401.00         410.819       -17128.2      0.144595E-10    77875.8

EL=   18  NODES=    18    19  MAT=  1  TCENT,TTOPZ,TBOTY=     0.0      0.0      0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -39.579        SBZ= -19.304        SBY= -87.769       SIG1=  67.495      SIG3= -146.65
END J     SDIR= -39.579        SBZ= -24.027        SBY= -91.062       SIG1=  75.510      SIG3= -154.67
 STATIC FORCES ON NODE        18  -286.398       -4401.00        -410.819        17128.2      0.144595E-10   -77875.8
 STATIC FORCES ON NODE        19   286.398        4401.00         410.819       -21318.6     -0.144595E-10    80797.1

EL=   19  NODES=    19    20  MAT=  1  TCENT,TTOPZ,TBOTY=     0.0      0.0      0.0  QZ,QY=  0.00000E+00  0.00000E+00        3-D BEAM    4
END I     SDIR= -39.579        SBZ= -24.027        SBY= -91.062       SIG1=  75.510      SIG3= -154.67
END J     SDIR= -39.579        SBZ= -28.750        SBY= -94.354       SIG1=  83.525      SIG3= -162.68
 STATIC FORCES ON NODE        19  -286.398       -4401.00        -410.819        21318.6      0.000000E+00   -80797.1
 STATIC FORCES ON NODE        20   286.398        4401.00         410.819       -25508.9      0.000000E+00    83718.4
```

ATTACHMENT 3

April 4, 1988

To: M. J. Gahan, III

From: R. B. Pond, Jr.

Subject: Main Steam Piping Flaw at Calvert Cliffs Unit 1

In the Fall outage of 1986 a flaw was discovered adjacent a weld in the main steam piping at No. 12 steam generator. The flaw was found using ultrasonic examination, and it consisted of a circumferential area of reduced wall thickness in the base metal which at the most is 0.1 inch below the minimum wall thickness of 0.95 inch. The width of the flaw is about 0.50 inch and it extends for about 24 inches circumferentially. In December 1986 our Level III's made a comparison of the UT data with the original radiograph and resolved that the indication is an area of excessive weld preparation in the original pipe joint. Subsequent radiography by BG&E Co. confirmed that the flaw area has exactly the same boundary as in the preservice condition.

The flaw has no planar character in the material and we understand that a very conservative finite element analysis has clearly established that stresses in this area are adequately low for continued operation.

An option to pad weld this area to meet code compliance has been discussed. It is our practice to use pad welding in relatively thin wall piping, and only as a temporary expedient to avoid problems where there is an ongoing degradation mechanism like erosion-corrosion. The main steam piping flaw meets neither requirement. In this case we have the opinion that the system is adequate for continued operation. A pad weld may induce stresses that will be hard to quantify and which may decrease the margins for safe operation.

We recommend that pad welding is not needed and should not be done.

We recommend that the system be used as is.

R. B. Pond, Jr.
Principal Metallurgist

RBPjr/lrs

cc: W. C. Holston
T. N. Pritchett
B. C. Rudell
D. A. Wright

Job Card No.: 88-30-76