114
BNWL-1397
UC-80

2-
6-70

CSE PRESSURE VESSEL DEFECT STUDY

V. B. Watwood, Jr. and T. L. Anderson

June 1970

AEC RESEARCH &
DEVELOPMENT REPORT

BNWL-1397

LEGAL NOTICE

This report was prepared as an account of Government sponsored work. Neither the United States, nor the Commission, nor any person acting on behalf of the Commission:

A. Makes any warranty or representation, expressed or implied, with respect to the accuracy, completeness, or usefulness of the information contained in this report, or that the use of any information, apparatus, method, or process disclosed in this report may not infringe privately owned rights; or

B. Assumes any liabilities with respect to the use of, or for damages resulting from the use of any information, apparatus, method, or process disclosed in this report.

As used in the above, "person acting on behalf of the Commission" includes any employee or contractor of the Commission, or employee of such contractor, to the extent that such employee or contractor of the Commission, or employee of such contractor prepares, disseminates, or provides access to, any information pursuant to his employment or contract with the Commission, or his employment with such contractor.

PACIFIC NORTHWEST LABORATORY

RICHLAND, WASHINGTON

operated by

BATTELLE MEMORIAL INSTITUTE

for the

UNITED STATES ATOMIC ENERGY COMMISSION UNDER CONTRACT AT(45-1)-1830

3 3679 00061 6856

BNWL-1397
UC-80, Reactor
Technology

CSE PRESSURE VESSEL DEFECT STUDY

By

V. B. Watwood, Jr. and T. L. Anderson

Systems Engineering Department
Physics and Engineering Division

June 1970

FIRST UNRESTRICTED
DISTRIBUTION MADE

JUL 9'70

BATTELLE MEMORIAL INSTITUTE
PACIFIC NORTHWEST LABORATORIES
RICHLAND, WASHINGTON 99352

BNWL-1397

Printed in the United States of America
Available from
Clearinghouse for Federal Scientific and Technical Information
National Bureau of Standards, U.S. Department of Commerce
Springfield, Virginia 22151
Price: Printed Copy \$3.00; Microfiche \$0.65

CSE PRESSURE VESSEL DEFECT STUDY

V. B. Watwood, Jr. and T. L. Anderson

ABSTRACT

Finite element methods are used to predict the behavior of a laminar crack in a thick-walled pressure vessel. An analytical method employing strain energy is developed to evaluate the stress intensity factor for a crack in a body of arbitrary shape. The method is illustrated by several examples, and bounds to exact solutions are discussed. Modification of an existing finite element computer code to include calculation of stored elastic strain energy is described and the program is employed to obtain stress resultants in a pressure vessel under pressure loads and transient thermal shock. Fracture mechanics methods are then employed to predict possible unstable crack growth and estimated growth rates for the pressure vessel defect.

TABLE OF CONTENTS

LIST OF FIGURES	vii
LIST OF TABLES.	ix
1. INTRODUCTION	1
1.1 Finite Element Method.	2
1.2 Fracture Mechanics Approach.	6
2. SUMMARY AND CONCLUSIONS	8
3. STRESS ANALYSIS METHOD.	10
3.1 General Theory	10
3.2 Strain Energy	17
3.3 Description of AXISOL.	19
3.4 Added Features of AXISOL.	19
3.4.1 Plotting	19
3.4.2 Mesh Size Change Element	20
3.4.3 ALTER Option	21
3.4.4 Temperature Transfer Option	21
4. THERMAL ANALYSIS METHOD	22
4.1 General Theory	22
4.2 Computer Code Description	24
5. STRESS INTENSITY FACTOR PREDICTION.	25
5.1 Fracture Mechanics and Crack Propagation	25
5.2 Application	28
5.2.1 Direct Method.	28
5.2.2 Energy or Compliance Method	35
5.3 Bounds to the Exact Solution	38
5.3.1 Lower Bound on G.	39
5.3.2 Upper Bound on G.	41
5.4 Generalization of the Method	43
6. STRESS ANALYSIS - CSE VESSEL.	45
6.1 Assumed Defect Size	45
6.2 Pressure Loading	46

7.	THERMAL ANALYSIS - CSE VESSEL	58
7.1	Thermal Shock Boundary Conditions.	58
7.2	Thermal Analyses	61
7.3	Critical Time Selection	65
7.4	Thermal Stresses	65
8.	DEFECTBEHAVIOR	70
8.1	Fracture Toughness.	70
8.2	Crack Growth.	71
	NOTATION.	73
	REFERENCES	75
	BIBLIOGRAPHY	77

LIST OF FIGURES

1	Nozzle C Geometry	3
2	Element Idealization of Nozzle C-Vessel Intersection with No Defect Present	5
3	Trans Element Introduced Into AXISOL for Mesh Size Changes	20
4	Finite Element Idealization for Center Slotted Panel and Side Slotted Panel	30
5	Boundary Conditions for Center Cracked and Side Cracked Panels	31
6	Elements Near Tip of Crack for Center Slotted Panel	32
7	Center Cracked Panel with Center Wedge Loads	42
8	Stress Components Occurring on Failure Plane - Undefected Structure; 3000 psi Internal Pressure Loading	47
9	Element Idealization of Nozzle C - Vessel Intersection with One-Half Inch Defect Present	49
10	Element Idealization of Nozzle C - Vessel Intersection with 4-in. Defect Present	50
11	Stress Components in Elements Near Defect - 1/2-in. Crack	51
12	Stress Components in Elements Near Defect - 4-in. Crack	52
13	Element Meshes Used in Vicinity of Defect for Nominal 1/2-in. Crack	53
14	Consequences of Crack "Turning" and Extreme Growth	55
15	Element Idealization in Vicinity of Defect Used for Crack "Turning" Study	56
16	Comparison of Analytical and Experimental Temperature at Outer Surface of Shell, Away from Nozzle C; Thermal Solution A and Average of Thermocouples TB06, TB09 and TB013, Run B-19-B	63

17	Comparison of Experimental and Analytical Thermal Gradient in Bottom Head of CSE Pressure Vessel	64
18	Stress Components on Failure Plane, Undefected Structure. Thermal Loading A.	66
19	Stress Components on Failure Plane with 4-in. Defect Present	68

LIST OF TABLES

1	Predicted Values of K_I for Center Slotted Panel Based on Finite Element Stress Solution	32
2	Values of K_I by Finite Element Energy Method for Center-Cracked Panel with Evenly Distributed End Loads a	37
3	Values of K_I/σ by Finite Element Method for Side-Cracked Panel with Evenly Distributed End Loads a	38
4	Values of K_I/P for Cracked Infinite Strip with Center Wedge Loads	42
5	Thermal Boundary Condition Data for Thermal Analysis	61

CSE PRESSURE VESSEL DEFECT STUDY

V. B. Watwood, Jr. and T. L. Anderson

1. INTRODUCTION

As part of the Containment Systems Experiments (CSE), a reactor simulator vessel was constructed and used in experiments to study loss-of-coolant accidents. Dummy core arrangements were placed in the CSE reactor simulator vessel which was then filled with heated coolant and pressurized. In order to simulate loss of coolant in a reactor discharge nozzles located in the CSE pressure vessel were opened permitting a rapid escape of the pressurized fluid. This rapid discharge of the vessel is called blowdown.

During a blowdown run failure of a vessel support occurred which permitted the simulator vessel to move and impact against surrounding structural supports. Although no visible damage to the vessel resulted from the accident, ultrasonic tests revealed a laminar defect in the bottom head of the vessel near the blowdown Nozzle C. It was decided to perform extensive tests and investigations on all portions of the vessel subject to large stresses during previous blowdown tests. This report is part of the results of such investigations.

The primary objective of this study was to predict the behavior of the defect near Nozzle C in the CSE pressure vessel induced by predicted and measured pressure loads and thermal shock, and to attempt to understand why the defect occurred.

Because of the serious consequences of catastrophic failure, and more importantly because of the research nature of this study, an extensive analytical investigation was undertaken in an attempt to model the actual defect and its loadings as accurately as possible within the framework of the present state-of-the-art and time limitations.

Since the defect lies in the immediate vicinity of a nozzle, the stress state is a complex one. Therefore, in order to provide an adequate model of the stress situation for use in a fracture mechanics analysis it was necessary to resort to numerical methods. In particular the finite element method was utilized in this regard for both the elastic and thermal portions of the study.

The balance of this section of the report contains a brief summary of the methods and concepts used to analyze the defect problem. Subsequent sections consider the stress analysis and thermal analysis methods employed. Finally, a section on numerical calculation of stress intensity factor is presented preceding the final sections which deal directly with the thermal and stress analyses of the CSE vessel.

1.1 FINITE ELEMENT METHOD

The finite element method is a very versatile numerical procedure and analysis tool in solid mechanics. Much literature (e.g., Reference 6) has been devoted to the subject. Great strides have been made to increase the accuracy of the method by use of more refined elements while at the same time more efficient equation-solving numerical schemes permit the use of more degrees of freedom in the simulation. Another important aspect to the user is the continual improvement and automation of data input routines. Clearly, it is becoming easier to obtain linearly elastic solutions to geometrically complex structures such as occur in real life.

However, even the finite element method has practical limitations with regard to the geometries it can treat and in particular 3-dimensional simulations are difficult. The complete problem here is clearly 3-dimensional but it can be seen that the vessel-nozzle intersection can be approximated rather well if considered as axisymmetric about the nozzle centerline

since the bottom head is spherical. The defect, of course, does not extend completely around the nozzle centerline although it shows extension in that direction. Also, one of the thermal loading situations is clearly not axisymmetric. Nevertheless it was deemed necessary and considered to be the best approach possible to consider the vessel as an axisymmetric 3-dimensional solid with comparable loading. The cross section of the nozzle-vessel intersection then appears as in Figure 1 with appropriate dimensions shown.

FIGURE 1. Nozzle C Geometry

The particular finite element approach and theory for the 3-dimensional axisymmetric solid elastic behavior is given in Section 3 of this report. Similarly, the thermal code is described in Section 4. A brief summary of the capabilities and limitations of the procedure follow.

First of all, the geometrical configuration is approximated by an assemblage of rings. This can be visualized most easily by considering only a cross section of the axisymmetric body created by passing a plane through the axis of symmetry. Only one "side" of the cross section need be considered, however, as for example in Figure 1. This cross section then is approximated by an assemblage of triangles and quadrilaterals as shown in Figure 2. Each triangular or quadrilateral "element" is in reality a ring which is itself axisymmetric about the axis of symmetry of the complete solid.

Since complete axisymmetry is assumed, then the displacements, stresses, temperatures, and similar parameters show no variation in the θ direction. In the r and z directions, however, (in the plane of the previous cross section) the displacement or temperature field is free to vary linearly within each element. For the elasticity problem this produces a state of constant strain except for ϵ_θ which can vary as $1/r$ with comparable stresses within each element.

In theory, the error associated with a given problem may be reduced as much as desired by reducing the element size appropriately. In practice, there are practical limits imposed both by the computer code and time available for data preparation. In general the element networks employed in this study were considered to be small enough to produce stresses within 10% of the correct value except at points of singularity such as at crack tips and these problems are met in a different way as will be seen in the discussion on fracture mechanics.

FIGURE 2 Element Idealization of Nozzle C-Vessel Intersection with No
Defect Present

For either the elastic or the thermal code the various boundary conditions that can be treated are numerous and will not be detailed here. The particular boundary conditions employed are described in the appropriate sections of this report.

For the thermal shock portion of this analysis, the thermal problem was solved first and temperature distributions at various times were written on tape. After some study only a selected few were utilized in the stress analysis. The element nets were generally different between the thermal and elastic problem in which case suitable interpolation was used to transform the data from one code to another.

1.2 FRACTURE MECHANICS APPROACH

To make use of linear fracture mechanics as presently formulated, it is necessary to deal with a singularity in the elastic stress solution which appears at the crack tip. Since the method of stress analysis is basically a Rayleigh-Ritz approximation, the accuracy of the solution is highly dependent upon the closeness of fit of the trial solution versus the actual solution. Also the singularity is not, of course, modeled directly in the present formulation. These factors result in a very poor characterization of the stress state at the crack tip. This problem led to the development of an energy method for computing the stress intensity factor rather than making direct use of the stress or displacement field predictions in the vicinity of the crack tip. This procedure is given in detail in Section 5.

Briefly, however, the stored strain energy is computed for two slightly different crack lengths and these results are numerically differentiated to produce the strain-energy release rate, G , which is directly related to the stress intensity factor, K .

The usage of the stress intensity factor, once it is obtained, is the essence of linear fracture mechanics and no attempt will be made here to present the entire subject. The aspects of fracture mechanics which will be pursued here are critical crack propagation and, more importantly, crack growth rate. Reference 9 is suggested for further reading in this area.

Subsequent sections of this report will deal with the problem of obtaining the stress intensity factor, K , and the consequences of the magnitude of this factor as predicted by linear elastic fracture mechanics.

2. SUMMARY AND CONCLUSIONS

The behavior of the defect near Nozzle C in the CSE reactor simulator vessel was investigated by means of extensive analytical effort. Two existing finite element computer codes, AXISOL and AMG065, were modified and expanded in order to automate the investigation of the heat conduction-stress analysis problems encountered.

The essential problem--to provide estimates of stress intensity factor K ,-- is shown to be much easier and more general by the use of finite element methods. By a straightforward procedure utilizing existing codes, accurate estimates of this factor are made. The procedure consists of simply calculating the strain energy at several crack lengths and numerically differentiating the resulting strain energy-crack length curve. Indications are given of the direction of the error involved and it is suggested that use of both equilibrium and displacement finite element idealization will result in estimates on both sides of the exact result.

Stress intensity factors associated with 1/2-in. and 4-in. laminar axisymmetric cracks in the vicinity of Nozzle C are determined for pressure and thermal loading on the vessel. Comparison of these values with critical values for unstable crack behavior shows that there is negligible chance of unstable crack growth behavior for the conditions investigated assuming that the metal surrounding the defect is sound. Crack growth rates are also shown to be so small that subsequent detectable growth into presently sound metal is very unlikely. If additional weakened areas exist, then additional growth could occur. However, since there is no way of estimating any material properties for such a weakened zone, neither stability nor growth estimates could be made.

Of the two crack lengths studied in this report (1/2-in. and 4-in.), the 1/2-in. defect size is probably much closer to reality than the larger defect. Tensile tests on the removed cores indicated significant strength remaining in the defect zone itself so that gross weakening of the area leading to pseudo large defect behavior seems unrealistic. If this can be accepted, then it seems probable that the only possibility for laminar crack extension is away from Nozzle C even if significant weakened material exists. Intuitively this does not seem particularly harmful since laminated pressure vessels are sometimes built intentionally.

3. STRESS ANALYSIS METHOD

The purpose of this section is to present a brief description of the method used to solve the various linear elastic problems throughout this report. The method is generally known as the finite element method and is basically a Rayleigh-Ritz procedure which consists of solving for the best approximation out of a trial set of solutions.

The particular ideas employed here are presented in References 1, 2, and 3. Nevertheless, for the sake of completeness and for use in later discussions, a brief but complete summary of the theory involved will be given.

3.1 GENERAL THEORY

The presentation begins with a statement of the principal of virtual displacements, i.e., (see Reference 21 for proof of this theorem; body forces are left out for simplicity),

$$\int_V \epsilon_{ij}^* \sigma_{ij} dV = \int_{S_\tau} \bar{T}_i u_i^* dS \quad (1)$$

where: $\epsilon_{ij} = \frac{1}{2} (u_{i,j} + u_{j,i})$ are the components of the strain tensor,

u_i are the components of the trial displacement vector which must be continuous throughout the body and must meet the prescribed displacement boundary conditions,

$\bar{T}_i$ are the components of the prescribed stress vector tractions,

C_{ijmn} are the elastic constants, i.e., $a_{ij} = C_{ijmn} (\epsilon_{mn} - \epsilon_{mn}^T)$,

ϵ_{mn}^T are the thermal strains,

V indicates the volume of the body,

S indicates the surface of the body and

S_T indicates the surface of the body where tractions are prescribed,

u_i^* and ϵ_{ij}^* are the virtual displacements and associated strains. u_i^* must vanish on S_T .

First it is convenient to rewrite Equation (1) in a slightly different notation. A strain vector, $\underline{\epsilon}$, is defined by "stacking up" the strain components, i.e.,

for plane stress or strain

$$\underline{\epsilon} = \begin{Bmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{xy} \end{Bmatrix} \quad \underline{\tau} = T \begin{Bmatrix} \alpha \\ \alpha \\ 0 \end{Bmatrix} \quad \text{where } T = \text{temperature}$$

for an axisymmetric solid

$$\underline{\epsilon} = \begin{Bmatrix} \epsilon_{rr} \\ \epsilon_{\theta\theta} \\ \epsilon_{zz} \\ \epsilon_{rz} \end{Bmatrix} \quad \underline{\tau} = T \begin{Bmatrix} \alpha \\ \alpha \\ \alpha \\ 0 \end{Bmatrix}$$

Displacement, stress and traction vectors are similarly defined, i.e., for plane stress or strain

$$\underline{u} = \begin{Bmatrix} u_x \\ u_y \end{Bmatrix}, \quad \underline{\sigma} = \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{Bmatrix}, \quad \underline{T} = \begin{Bmatrix} T_x \\ T_y \end{Bmatrix}$$

The elastic constants must then be rewritten as a matrix to be compatible notationally

$$\underline{\sigma} = \underline{C}^T (\underline{\epsilon} - \underline{\tau}) \quad (2)$$

The superscript T indicates the transpose.

The basic concept in finite elements is that the body is divided into a finite number of geometrically simple subdomains called elements. Equation (1) is then rewritten and extended slightly as follows:

$$\sum_{i=1}^E \int_{V_i} \underline{\varepsilon}^{*T} \underline{\sigma} dV = \sum_{i=1}^E \sum_{j=1}^{n_i} \int_{S_{ij}} \underline{u}^* \Delta \bar{T} dS \quad (3)$$

where E = number of elements,
 V_i = volume of i^{th} element,
 n_i = number of sides for i^{th} element,
 S_{ij} = j^{th} side of i^{th} element, and
 $\Delta \bar{T}$ = prescribed jump in stress vector if side is internal and prescribed traction if side is external.

Note that the capability to approximate jumps in the stress vector from element to element has been introduced. Such a capability could have been included in Equation (1) but for simplicity it was not.

Within each element a simple displacement state is taken (subject to completeness considerations not discussed here), e.g., for plane stress:

$$\underline{u} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y \end{bmatrix} \begin{Bmatrix} b_1 \\ b_2 \\ \cdot \\ \cdot \\ b_6 \end{Bmatrix} = \underline{e} \underline{b} \quad (4)$$

and for axisymmetric solids r and z are substituted for x and y respectively. The array of undetermined coefficients $\underline{b}$ in Equation (4) will be chosen in such a manner as to satisfy Equation (3). However, the compatibility requirement that the displacement vector must be continuous throughout the body demands continuity from element to element which must be satisfied. Therefore, Equation (4) will be rewritten writing the $\underline{b}$ vector in terms of the nodal (corners of the elements) displacements.

The triangular element is the basic element for this formulation so Equation (4) is evaluated at each node in turn to form

$$\underline{\underline{s}}^j = \begin{bmatrix} 1 & x_1 & y_1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x_1 & y_1 \\ 1 & x_2 & y_2 & 0 & 0 & 0 \\ \text{etc.} \end{bmatrix} \{b\} \quad (5)$$

where

$$\underline{\underline{s}}^j = \begin{Bmatrix} 1 \\ u_x \\ 1 \\ u_y \\ 2 \\ u_x \\ 2 \\ u_y \\ 3 \\ u_x \\ 3 \\ u_y \end{Bmatrix} \quad \text{and } u_x^i \text{ is the displacement of the } i^{\text{th}} \text{ node in the x direction, etc., for the } j^{\text{th}} \text{ element.}$$

The coefficient matrix in Equation (5) can be inverted

$$\underline{\underline{b}} = \underline{\underline{h}} \underline{\underline{s}}^j$$

Substituting in Equation (4) gives the displacement field for the j^{th} element

$$\underline{\underline{u}} = \underline{\underline{e}} \underline{\underline{h}} \underline{\underline{s}}^j \quad (6)$$

The strains within the j^{th} element then are found from the displacement field in the usual manner:

$$\underline{\underline{\varepsilon}} = \underline{\underline{D}} \underline{\underline{u}} = \underline{\underline{D}} \underline{\underline{e}} \underline{\underline{h}} \underline{\underline{s}}^j = \underline{\underline{g}} \underline{\underline{h}} \underline{\underline{s}}^j \quad (7)$$

where, e.g., for plane stress

$$D = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix}$$

as for axisymmetric solids:

$$D = \begin{bmatrix} \frac{\partial}{\partial r} & 0 \\ 0 & \frac{\partial}{\partial z} \\ \frac{1}{r} & 0 \\ \frac{\partial}{\partial z} & \frac{\partial}{\partial r} \end{bmatrix}$$

Recall that the above Equations (4) through (7) have applied to any arbitrary element. The requirement for complete continuity of the displacement vector is satisfied by noting that the displacement field as expressed in Equation (6) is linear. Therefore, if common nodes of adjoining elements are constrained to move together, the displacements all along the common boundary will be compatible. This constraint is expressed by:

$$\underline{s}_{\sim} = \underline{a}_{\sim} \underline{r}_{\sim} \quad (8)$$

where $\underline{s}_{\sim}$ now is the complete set of element generalized coordinates, i.e.,

$$\underline{s}_{\sim} = \left\{ \begin{bmatrix} 1 \\ s_{\sim} \\ 2 \\ s_{\sim} \\ \cdot \\ \cdot \\ \cdot \\ E \\ s_{\sim} \end{bmatrix} \right\} \quad (9)$$

and where $\tilde{s}^i$ has the definition as used before in Equation (5). $\underline{r}$ is the set of external generalized coordinates and consists of the set of displacements of the nodes

$$\underline{r} = \begin{bmatrix} 1 \\ u_x \\ 1 \\ u_y \\ 2 \\ u_x \\ \vdots \\ \vdots \\ \vdots \\ N \\ u_y \end{bmatrix} \quad \text{where there are } N \text{ nodes.} \quad (10)$$

Clearly $\underline{a}$ is a very simple, although large, matrix which has the important property that, provided the displacement components in Equations (9) and (10) are orthogonal and aligned at each node, there will exist only a single nonzero component in $\underline{a}$ in each row and this component will be a one. Of course each column of $\underline{a}$ may have more than one nonzero component which is, in fact, the way the nodes of different elements are constrained to move together. This property (a maximum of one nonzero element per row) leads to a very simple algorithms for the matrix multiplications involving $\underline{a}$ which follow. These operations are therefore handled in a special way in the computer and are referred to as the "merge" operation.

Substituting Equations (6) and (7) and the constitutive relation into Equation (3) and utilizing the same family of displacement fields for the virtual displacements as the trial solution results in:

$$\sum_{\tilde{s}} \tilde{s}^{*T} \underline{k} \tilde{s}^i - \sum_{i=1} \tilde{s}^{*T} \underline{p}_\tau^i = \sum_{i=1} \tilde{s}^{*T} \underline{p}_m^i \quad (11)$$

$$\text{where } \underline{k}^i = \underline{h}^T \int_{V_i} \underline{g}^T \underline{C} \underline{g} dV \underline{h} \quad (11a)$$

$$\underline{p}_\tau^i = \underline{h}^T \int_{V_i} \underline{g}^T \underline{C} \underline{\epsilon}^\tau dV \quad (11b)$$

$$\underline{p}_m^i = \underline{h}^T \sum_{j=1}^{n_i} \int_{S_{ij}} \underline{e}^T \Delta \bar{T} dS \quad (11c)$$

Utilizing the notation expressed by Equation (9) and

$$\underline{k} = \begin{bmatrix} 1 & & & & & \\ k & 0 & . & . & . & 0 \\ & 2 & & & & \\ 0 & k & 0 & . & . & 0 \\ & & . & & & \\ & & & . & & \\ . & . & . & . & . & E \\ & & & & & k \\ 0 & . & . & . & . & . \end{bmatrix} \quad \underline{p}_\tau = \begin{bmatrix} 1 \\ p_\tau \\ . \\ . \\ . \\ N \\ p_\tau \end{bmatrix}, \quad \underline{p}_m = \begin{bmatrix} 1 \\ p_m \\ . \\ . \\ . \\ N \\ p_m \end{bmatrix}$$

$$\text{gives } \underline{\tilde{s}}^{*T} \underline{k} \underline{\tilde{s}} = \underline{\tilde{s}}^{*T} \underline{p}_\tau + \underline{\tilde{s}}^{*T} \underline{p}_m \quad (12)$$

To proceed the constraint matrix of Equation (8) is substituted to give

$$\underline{x}^{*T} \underline{K} \underline{x} = \underline{x}^{*T} \underline{p}_\tau + \underline{x}^{*T} \underline{p}_m = \underline{x}^{*T} \underline{p} \quad (13)$$

$$\text{where } \underline{K} = \underline{a}^T \underline{k} \underline{a} \\ \underline{p} = \underline{p}_\tau + \underline{p}_m = \underline{a}^T (\underline{p}_{\tau} + \underline{p}_m)$$

The array of undetermined coefficients in $\underline{x}$ are independent which infers

$$\underline{K} \underline{x} = \underline{p} \quad (14)$$

This equation then must be solved for $\mathbf{r}$. Note that if displacement boundary conditions are applied, this can be directly reflected in the components of $\mathbf{r}$. However, in this case, those affected components are not independent and this modifies Equation (14).

The finite element code AXISOL, Reference 1, solves the above equations automatically; finally solving Equation (14) for $\mathbf{x}$ which in turn give ξ from Equation (8) from which the strain ξ and the stresses are computed for each element using Equations (7) and (2).

As mentioned previously the formulation herein, which is the one used in AXISOL, utilizes a basic triangular element. For many applications a quadrilateral element is much easier to use. For that reason a quadrilateral element is available for use in AXISOL which is made up of four of the basic triangles. The code automatically breaks each quadrilateral up into four triangles and reduces out the extra center node so that it is not carried along as an extra two degrees of freedom.

For the purpose of displaying stress distributions the resulting stress tensor for each element is assumed to exist at the geometric center of the element. The only justification for this is intuition which tends to be supported by comparison of results with exact solutions for example problems. The variation of stress between element centers is then assumed to be linear. In the case of the quadrilateral element the stresses of the four triangles are simply averaged in the usual arithmetic sense and those values are assigned to the geometric center of the quadrilateral.

3.2 STRAIN ENERGY

Finally, it is convenient at this time to define the strain energy associated with a particular deformation and

thermal state. The strain energy is defined as:

$$U = \frac{1}{2} \int_V \sigma_{ij} \epsilon_{ij}^e dV \quad (15)$$

where the "elastic" strain ϵ_{ij}^e are defined as $\epsilon_{ij} - \epsilon_{ij}^\tau$. Dividing the integral into element integrals and substituting Equation (2) gives:

$$U = \frac{1}{2} \sum_{i=1}^E (\epsilon - \epsilon^\tau)^T \underline{C} (\epsilon - \epsilon^\tau) dV_i$$

or after substituting Equations (7), (11a) and (11b)

$$U = \frac{1}{2} \sum_{i=1}^E (\underline{s}_i^T \underline{k}_i \underline{s}_i - 2 \underline{s}_i^T \underline{p}_{i\tau} + \underline{Q}_i) \quad (16)$$

where
$$\underline{Q}_i = \int_{V_i} \epsilon^\tau C \epsilon^\tau dV$$

Equation (16) is the most useful form in which to express the strain energy. However, another form which is also sometimes useful can be found by substituting Equation (8) into (16)

$$U = \frac{1}{2} \underline{r}_\tau^T \underline{K}_\tau \underline{r}_\tau - \underline{r}_\tau^T \underline{p}_{\tau\tau} + \frac{1}{2} Q$$

where
$$Q = \sum_{i=1}^E \underline{Q}_i$$

Using Equation (14) produces:

$$U = \frac{1}{2} \underline{r}_\tau^T (\underline{P}_m - \underline{P}_\tau) + \frac{1}{2} Q \quad (17)$$

The capability to compute the strain energy was added to AXISOL by programming Equations (16) and (17).

3.3 DESCRIPTION OF AXISOL

The finite element computer code AXISOL used in the study is fairly typical of the codes currently in widespread use to solve 2-dimensional problems in plane and axisymmetric elasticity.

For both the axisymmetric and plane cases the basic element is triangular and within the element the displacement field is constrained to be a linear function of the coordinates. Thus, the resulting strain field is constant within each element except for ε_{θ} as noted previously. Complete continuity of displacements is achieved at the element boundaries by demanding continuity at the element vertices.

The quadrilateral element in the code is made up of an assembly of four triangles with an additional node introduced at the geometrical center of the quadrilateral. This node is reduced out by considering the subassembly of four elements as a substructure.

Element stresses are computed directly from the displacement field for each triangle. For the quadrilateral these stresses are merely averaged and the resulting state of stress is said to exist at the geometric center of the quadrilateral.

3.4 ADDED FEATURES OF AXISOL

In addition to the modification mentioned previously to give AXISOL the capability to compute the stored elastic strain energy (with or without thermal loading) several other modifications were implemented in the course of this work.

3.4.1 Plotting

A plotting routine existed in the code when it was acquired but, unfortunately, it was for a different type plotter than the one available at PNL. Therefore, it was necessary to start over with the plotting routines.. The final result is that the

complete structure or any part of it can be plotted at any scale. For example, the most useful feature here was to plot the entire structure to a small scale and then provide a "blow-up" of the elements near the defect.

Plotting serves two purposes. The first and most important is data checking and the second is figure preparation for reports, such as Figure 2 of this report.

3.4.2 Mesh Size Change Element

There are many occasions where it is desirable to pass from a coarse to a fine element mesh or vice versa. This is particularly true when working with defects where a fine-element mesh is needed around the defect tip. Such a change necessarily involves many triangles and irregular numbering. The automatic element numbering scheme provided in AXISOL breaks down and it is necessary to provide all such data in complete detail by card input.

To overcome this time-consuming, error-prone work a new quadrilateral element was introduced with five nodes as shown in Figure 3. In reality it is made up of three triangles as indicated in the figure. This element can be input in pairs to automatically fill in a complete line of five node elements just as AXISOL presently does with the normal quadrilateral.

FIGURE 3. Trans Element Introduced Into AXISOL for Mesh Size Changes

3.4.3 ALTER Option

Often one wishes to run several problems where only a very few cards of the data input are different for each run. This is particularly true when working with some defect configurations. Normally this would require resubmitting the entire deck for each run, which would be time consuming. The ALTER modification allows the analyst to rerun the same problem with any number of changes to the data. In this case it is necessary to supply only the changes and the problems are run back-to-back.

3.4.4 Temperature Transfer Option

Thermal loading problems require the input of the temperature at each node of the problem. Supplying such information through the input cards requires not only extensive error-prone handling of the data such as key punching, but also destroys much of the usefulness of automatic node generation capability in AXISOL since each node point will in general have a different temperature.

Therefore, a subroutine was added which enables the code to read in, off tape, temperature data which have been generated earlier by AMG065 heat conduction program (see Section 4). It is neither necessary nor desirable that the nodal mesh be the same in each code. The subroutine takes each node of the AXISOL stress problem and determines where that node lies in the thermal data (which of course must cover the same domain). Least square interpolation is then used to compute the temperature at the node.

4. THERMAL ANALYSIS METHOD

The purpose of this section is to present a brief description of the method used to solve the heat conduction or thermal shock problem of the CSE pressure vessel during blowdown. The problem is treated using a numerical method for the solution of the heat conduction phenomenon in a body of complex geometry. The finite element method is used to discretize the body which reduces the problem formulation to the solution of a matrix equation for the nodal point temperatures of the assembly of finite elements.

4.1 GENERAL THEORY

The formulation applies a variational method, along with the finite element technique, to the numerical analysis of heat conduction in thermally anisotropic, nonhomogeneous bodies. The development which follows is taken from Reference 4.

First, a functional of the temperature field $U(x,y,z)$ and of its first time derivative $\dot{U}(x,y,z)$ is introduced which is shown to satisfy the heat conduction equation throughout the body. It is also shown to satisfy the general heat flux boundary conditions where the temperature is not specified on the boundary.

Next the problem is discretized by assuming a piecewise linear temperature distribution in a finite array of quadrilateral elements covering the body, although each element is made up of four triangular elements (each with linear temperature distributions), the point (node) common to the triangles in the interior of the quadrilateral is reduced out of the expressions.

On the basis of the linear temperature distribution the variational principle (see References 4 and 5 for detailed explanation) is employed and with suitable matrix operations

the vector of nodal-point temperatures $\underline{u}$ is found to satisfy the first degree, initial-value matrix equation

$$\underline{C} \dot{\underline{u}} + \underline{K} \underline{u} = \underline{f} \quad (18)$$

where the dot denotes differentiation with respect to time, t , and

$$\underline{C} = \underline{A}^T \int_V \rho c \underline{\phi}^T \underline{\psi} \underline{B} dV,$$

$$\underline{K} = \underline{\kappa} - \underline{H},$$

$$\underline{\kappa} = \underline{A}^T \int_V \underline{\phi}^{1T} \underline{k} \underline{\phi}^1 \underline{A} dV,$$

$$\underline{H} = h \int_{S_3} \underline{A}^T \underline{\phi}^T \underline{\$A} dS \text{ and}$$

$$\underline{f} = \int_{S_2} \underline{A}^T \underline{\phi}^T \underline{n} \underline{\bar{q}} dS - g u_o \int_{S_3} \underline{A}^T \underline{\phi}^T dS$$

The above terms are defined further by the definition of temperature field U_o and time rate of change of the temperature field $\dot{U}_o$ in an element

$$U_o(x,y,z,t) = \underline{\phi}(x,y,z) \underline{A} \underline{u}(t)$$

and
$$\dot{U}_o(x,y,z,t) = \underline{\psi}(x,y,z) \underline{B} \dot{\underline{u}}(t)$$

where $\underline{\phi}$ and $\underline{\psi}$ are vectors which specify the spatial dependence of U_o and $\dot{U}_o$, and $\underline{A}$ and $\underline{B}$ are matrices of constants defined above. Also

ρ represents the density,

c is the specific heat

$\underline{k}$ is the thermal conductivity tensor,

g is the film coefficient (convective boundary condition) on the element segment S_3 ,

$\underline{n}$ represent a unit normal vector on the element boundary S_2 and $\underline{\bar{q}}$ represents a specified flux vector on S_2 .

Computer Code AMG065, Reference 4, solves Equation (18) for the time dependent temperature, u , of selected nodes throughout the body. A numerical integration formula of order $(\Delta t)^3$ ($\ddot{u}$ assumed constant in each Δt time step) is employed. The resulting transient and steady-state solutions are shown to be numerically stable for any time step.

4.2 COMPUTER CODE DESCRIPTION

Code AMG065 relies heavily upon concepts developed for the finite element method of analysis of 2-dimensional problems in plane and axisymmetric elasticity (see Section 3.3).

AMG065 permits material properties to vary from element to element and adiabatic, constant flux, convective and temperature boundary conditions can be applied. The coefficients of the complete matrix equation are assembled from the coefficients of each quadrilateral element. The code, through the use of a suitable mesh-generating scheme, permits nodes, not specified as input, to be automatically inserted between specified input coordinates.

The accuracy of AMG065 in solving a variety of heat conduction problems is demonstrated (Reference 4) by comparison of the finite element solution with analytical, formal solutions. In all cases the correlation is quite good; nearly exact for specific problems using a fine mesh.

5. STRESS INTENSITY FACTOR PREDICTION

In order to predict the CSE defect growth behavior (to be discussed in detail in later sections) the methods of linear fracture mechanics were employed. Simply stated, crack growth in a body is related to the intensity of the local crack tip stress field which in turn is measured by a single parameter. Unfortunately, all of the solutions available in the literature deal with rather elementary body geometry. Although these elementary formulations may be used to obtain a preliminary estimate of the cracking behavior, the catastrophic nature of a failure in the CSE pressure vessel dictated the need for a more refined analysis.

This section describes a direct method of computation of the stress intensity, K , of linear fracture mechanics. Poor accuracy of the method, unless extremely small elements are used near the crack tip, leads to its abandonment in favor of direct computation of the strain energy release rate which is equivalent to K . This procedure consists of computing the strain energy for two slightly different crack lengths and employing numerical differentiation to determine the strain energy release rate. The accuracy is quite good for a relatively coarse finite element mesh for the example problems presented. Finally the question of bounds to the true solution is considered and it is suggested, but not proven, that the method will always bound the solution in a manner depending upon whether the stress intensity factor increases or decreases with crack length growth.

The material in this section was documented by the senior author elsewhere (Reference 17).

5.1 FRACTURE MECHANICS AND CRACK PROPAGATION

A complete summary of the fields of brittle fracture and fatigue crack propagation is, beyond the scope of this paper.

For such a discussion the reader is referred to References 7 and 8. However, it is desirable, if not necessary, to briefly describe the ideas of what is referred to here as "classical elastic fracture mechanics and crack propagation" since that field is also evolving rapidly.

First of all, both subjects make use of the phenomenon that the limiting solution for the elastic state of stress around a sharp crack as the distance from the crack tip becomes small is of the form

$$\sigma = \frac{K}{r^{1/2}} f(\theta)$$

where σ = one of the stress components, say σ_r ,

r, θ = polar coordinates with origin at the crack tip and

f = a trigonometric function of θ only which depends on the stress components considered and symmetry of the problem.

K is a constant dependent on the geometry such as crack length and loading. Since this solution is based on linear elasticity, K is linearly dependent on the magnitude of the loading. It is important to note that this general form of the solution holds for any arbitrary structure in which the crack is located. The K is referred to as the stress intensity factor since it is some kind of measure of the intensity of the local crack tip stress field.

The hypothesis is made that for a given material the behavior of the crack, whether it remains stable in length, grows without bound or grows at some definite rate, depends only upon the magnitude of this parameter. Thus, for a given material it is possible to test a single specimen for which K can be analytically determined in terms of the load and crack length, and resolve its behavior as a function of K . This same

behavior as a function of K is assumed to hold for any structure. For example, if unstable crack growth occurs at a value of loading and crack length for which K is $100 \text{ ksi-}\sqrt{\text{in.}}$, then K_{cr} for that material is $100 \text{ ksi-}\sqrt{\text{in.}}$. Then for any other structural configuration of the same material in the same environment if K reaches the value of $100 \text{ ksi-}\sqrt{\text{in.}}$, unstable crack growth is predicted to occur. In the same manner data for crack growth are empirically related to K , e.g., from Reference 9:

$$\frac{da}{dN} = C(AK)^4$$

where a = crack length,
 N = number of cycles of loading,
 C = constant depending on environment and
 AK = change in stress intensity factor K during a load cycle

Obviously, this explanation has been oversimplified as attempts to make the hypothesis fit real life have demanded some compromises, e.g., thickness corrections. Nevertheless, it can be seen that if K can be determined for a given structural configuration and crack length, the behavior of the crack can be predicted using previously catalogued material data.

Before proceeding, one final statement on fracture mechanics theory is necessary. Another quantity generally referred to as the strain energy release rate, G , is directly related to K through an equation involving only the elastic constants of the material. This equation is (Reference 8)

$$G = \frac{K^2}{E} (1 - \nu^2) \text{ (plane strain) or}$$

$$G = \frac{K^2}{E} \text{ (plane stress)}$$

where E = Young's modulus
 ν = Poisson's ratio

G conveys more physical meaning than K because, in addition to the strain energy release rate, it can be viewed as the portion of the work performed by the external loads during crack extension that goes into failure of the material per unit area of crack growth (one-half of the total new crack surface area).

5.2 APPLICATION

To utilize existing elastic crack propagation theory it is necessary to determine the appropriate value of the stress intensity factor, K , or equivalently the strain energy release rate, G . In the past, use was made of several closed form solutions, Reference 7, or a collocation technique, Reference 10, to determine these parameters. These methods were, however, not general enough or convenient enough for most applications although they have performed quite well in analyzing test specimens.

Finite element methods, on the other hand, are well known for their generality particularly with respect to geometric irregularities. In addition, they are becoming increasingly easier and more efficient to use. Therefore, the decision was made to attempt to use existing finite element codes to determine the appropriate value of the stress intensity factor.

5.2.1 Direct Method

The first and most obvious attempt was to utilize the stress distribution associated with the tip of a crack. For the 2-dimensional case this is (Reference 7)

$$\begin{aligned} \sigma_x = \frac{K_I}{(2\pi r)^{1/2}} \cos \frac{\theta}{2} \left\{ 1 - \sin \frac{\theta}{2} \sin \frac{3\theta}{2} \right\} \\ + \frac{K_{II}}{(2\pi r)^{1/2}} \sin \frac{\theta}{2} \left\{ 2 + \cos \frac{\theta}{2} \cos \frac{3\theta}{2} \right\} \end{aligned} \quad (19)$$

plus similar equations for σ_y and τ_{xy} . The origin here is at the crack tip and the x direction is parallel to the crack axis and into the uncracked material with r and θ as the usual polar coordinates. Since a finite element solution provides a stress distribution it is a simple matter to solve Equation (19) and the other component equations for K_I and K_{II} utilizing the finite element solution at discrete points. One would expect, of course, to utilize only those elements near the tip of the crack because of the local nature of Equation (19).

To verify this process a problem was solved for which the solution is known. This was the center slotted panel under tension shown in Figure 4. The element idealization is indicated in the figure and consists of 478 nodes and 470 elements. The stress distribution is symmetrical about the axis through the crack and therefore K_{II} is zero. It is also symmetrical about the vertical plate centerline so that only one fourth of the plate need be analyzed. The appropriate geometrical boundary conditions are shown in Figure 5.

Each element furnishes an estimate of the three stress components. Making use of the appropriate values of r and θ one can use Equation (19) to solve for K_I for each stress component. Therefore, it can be seen that each element offers three possible values for K_I (actually more since similar computations can be made for the predicted displacements).

A blowup of the elements near the tip is shown in Figure 6 and the results are tabulated in Table 1. Since these results range from a low of 1.5 to a high of 18.5 (correct value $K_I = 5.82$) it becomes apparent that this procedure lacks consistency. By choosing only those values derived from σ_y where θ remains less than 90 degrees, much better results are obtained. This is perhaps a logical choice for Mode I and perhaps those associated with τ_{xy} would be appropriate for Mode II. There still remains an annoying degree of

FIGURE 4. Finite Element Idealization for Center Slotted Panel and Side Slotted Panel. $L = 85$, $b = 40$, a is variable. Number of elements = 470, number of nodal points = 478.

FIGURE 5. Boundary Conditions for Center Cracked and Side Cracked Panels

FIGURE 6. Elements Near Tip of Crack for Center Slotted Panel. Element numbers for use with Table 1.

TABLE 1. Predicted Values of K_I for Center Slotted Panel Based on Finite Element Stress Solution

Element Idealization is Shown in Figures 4, 5 and 6									
Element Number	σ_x psi	Factor 1/ $\sqrt{\text{in.}}$	$\frac{I}{\text{psi} \cdot \sqrt{\text{in.}}}$	σ_y psi	Factor 1/ $\sqrt{\text{in.}}$	$\frac{K_I}{\text{psi} \cdot \sqrt{\text{in.}}}$	τ_{xy} psi	Factor 1/ $\sqrt{\text{in.}}$	$\frac{K_I}{\text{psi} \cdot \sqrt{\text{in.}}}$
1	0.469	5.05	2.37	2.010	6.24	12.5	-1.130	-7.43	8.40
2	1.016	3.35	3.40	3.156	1.76	5.55	0.349	53.1	18.5
3	0.743	3.50	2.60	2.000	2.94	5.88	0.207	24.4	5.05
4	0.548	4.16	2.28	1.680	3.89	6.54	0.0974	44.0	4.28
5	-0.0846	7.10	X	1.673	4.35	7.28	-0.858	-7.22	6.19
6	-0.115	8.40	X	2.324	2.53	5.88	-0.172	-22.4	3.85
7	0.256	6.08	1.56	2.112	2.93	6.19	0.195	28.9	5.64

$$K_I = \frac{\sigma}{\text{Factor}}$$

Correct $K_I = 5.82$

arbitrariness, and considering the apparent consequences of a poor choice (when the answer is not known), a better method should be sought. It should be noted that use of the displacement field in the same manner produced about the same results as shown above.

Some care must be exercised in regard to the interpretation of stresses for use in computing K_I from Equation (19). Two choices exist. First, the values of r and θ may be used corresponding to the geometric center of the element and compute K_I from (K_{II} is here assumed to be zero).

$$K_I = \frac{\sqrt{2\pi} r_c}{F(\theta_c)} \sigma$$

where r_c and θ_c are the value of the coordinates of the geometric center of the quadrilateral, σ is the average stress of the four surrounding triangular elements and F is the appropriate function depending upon which stress component is under consideration.

The other method is to make use of the definition of the average stress, and use the values of r and θ for the geometric center of each of the four triangles, i.e.,

$$\sigma = 1/4 \sum_{i=1}^4 \sigma_i = \frac{K_I}{4\sqrt{2\pi}} \sum_{i=1}^4 \frac{F(\theta_i)}{\sqrt{r_i}}$$

or

$$K_I = \frac{4 \sqrt{2\pi} \sigma}{\sum_{i=1}^4 \frac{F(\theta_i)}{\sqrt{r_i}}}$$

where r_i and θ_i are the coordinates of the geometric center of i^{th} element. The second alternative was considered to be the most accurate and, therefore, was the one used to generate Table 1.

Chan, et al. (Reference 11) utilized the above approach with somewhat more success by use of an automatic mesh generating scheme which enabled very small element sizes to be used in the vicinity of the crack; the smallest elements were on the order of $\frac{1}{100}$ to $\frac{1}{500}$ the size (relative to overall plate dimensions) of the smallest elements used in the example above. In their approach the writers of Reference 11 select a particular value of θ in Equation (19) and plot K_I (or K_{II}) as computed from one of several ways for various values of r . Three methods are used. The first two of these make use of the stress components or the displacement components as mentioned previously. The third makes use of a line integral which is proportional to the square of the stress intensity factor (Reference 12). The methods are equally accurate if the appropriate stress or displacement component is chosen. The resulting K - r plot is approximately straight (for the problems worked) for a range of r (it goes awry as $r \rightarrow 0$ and at "large" values of r). That slope is projected into $r = 0$ to get the final estimate. Even so, the question of which stress component or displacement to use is open to question (particularly when the correct answer is unknown). Undoubtedly the use of such small elements leads to long computer times and high costs. Applying the above method to the results in Table 1 using the σ_y component and $\theta = 45^\circ$ results in a prediction for K_I of 5.23 which is greater than 10% in error. Refinement of element size would undoubtedly improve the estimate but at extensive increase in data preparation and computer time. If one has the automatic element refining scheme built into his finite element code such as utilized in Reference 11, then only computer time would be lengthened. However, it is most desirable to be able to use an existing code without such extensive modification.

5.2.2 Energy or Compliance Method

Since the above procedure lacks sufficient accuracy without resorting to very small element nets, it was considered desirable to seek a better procedure. The approach that finally evolved was basically an analog to the so-called compliance method of experimentally determining the K or G value. The method is quite simple and consists of computing the strain energy stored for two or more slightly different crack lengths and making use of the definition of G, viz.,

$$G = (\pm) * \frac{dU}{dA} \quad \begin{array}{l} * + \text{ if constant load} \\ - \text{ if constant displacement} \end{array} \quad (20)$$

where U = strain energy stored and
 A = crack opening area, i.e., the crack length for a plate of unit thickness.

The easiest way to calculate the strain energy when only mechanical loads are involved is to make use of Clapeyron's Theorem, i.e., that the strain energy stored for an elastic body is equal to one-half the work that would be done by the applied forces (of the equilibrium state) acting through their total displacements. For the finite element method this means a multiplication of the generalized forces at the nodes by one-half the nodal displacements. Such a procedure is simple, both in concept and in implementation. For those problems with a small number of external loads, it can be done by hand from the existing output of most finite element codes. Only a few additional FORTRAN statements were necessary to modify an existing finite element code to accomplish this objective (see Section 3.2). Alternatively, the strain energy in each element may be calculated directly from the nodal displacements and these summed. This procedure is a little more complicated, but it is sometimes more convenient since the summation can proceed as stresses are being evaluated. Furthermore, no

special manipulations are required when displacement boundary conditions are applied. The computation of the strain energy when thermal loads are involved is covered in Section 4.2 of this report.

About the only word of caution necessary here is that the basic element mesh and nodal point coordinates must remain fixed from one crack length to another. This is required because the quantity to be measured is the change in stored strain energy as the crack changes in length. Therefore, the only changes that will occur in the strain energy estimate will be related to change in crack length and not to change in the element mesh. In order to eliminate possible confusion in this matter the recommended procedure for extending the crack is simply to add another node at the crack tip (with the same coordinates as the original one) and renumber the elements in such a way as to separate the two elements just inside of the crack tip. Thus, the crack is extended in length by an amount equal to one element length.

After the strain energy is calculated for several crack lengths, numerical differentiation may be used to obtain dU/dA . For this work an elementary numerical differentiation was utilized which was simply to subtract the strain energy estimate for adjacent crack lengths and then divide by the increase in area. The resulting derivative is said to apply for a crack length half way between the two used for the strain energy difference.

To illustrate this procedure consider the identical problem as discussed in Section 5.2.1. Several crack lengths with the same element idealization were considered. The strain energy for each configuration was computed and these are given in Table 2. The differencing scheme is illustrated in the presentation of data and a comparison with the work of Isida (see Reference 7) is given. Isida's work is also numerical but it

is generally considered the best solution available. The agreement is excellent with the error less than 2%. There is no ambiguity since only a single estimate is produced.

TABLE 2. Values of K_I by Finite Element Energy Method for Center-Cracked Panel with Evenly Distributed End Loads o. a as shown; b = 40 in.; L = 85 in.; unit thickness.

a, in.	Plane Strain, $\nu = 0.3$			a, in.	K_I/σ from Ref 7 $\sqrt{\text{in.}}$	diff %
	UE/σ^2 in. ²	$\frac{E}{\sigma^2} G$, in.	K_I/σ $\sqrt{\text{in.}}$			
7	3160.04					
		21.50	4.86	7.5	4.96	2.0
8	3181.54					
		24.76	5.22	8.5	5.32	1.9
9	3206.30					
		28.14	5.56	9.5	5.66	1.8
10	3234.44					
		31.66	5.90	10.5	6.00	1.7
11	3266.10					
		35.38	6.23	11.5	6.34	1.8
12	3301.48					
		39.26	6.57	12.5	6.67	1.5
13	3340.74					
		43.36	6.90	13.5	7.01	1.6
14	3384.10					

Another example, the side-cracked plate, was solved and the results are given in Table 3. Again the element idealization of Figure 4 was utilized but with the appropriate boundary conditions as shown in Figure 5. Again the results are excellent when compared to another numerical scheme.

TABLE 3. Values of K_I/σ by Finite Element Energy Method for Side-Cracked Panel with Evenly Distributed End Loads
a. a as shown; b = 40 in.; L = 85 in.; unit thickness.

a, in.	Plane Strain, v = 0.3			a, in.	K_I/σ from Ref 10	diff %
	UE/σ^2 , in. ²	$\frac{E}{\sigma^2} G$, in.	K_I/σ $\sqrt{\text{in.}}$			
9	3271.28					
		55.10	7.78	9.5	8.00	2.8
10	3326.38					
		67.10	8.58	10.5	8.76	2.1
11	3393.54					

Note that for both cases the finite element procedure consistently underestimates the stress intensity factor.

It should be apparent that the finite element strain energy method used above to calculate K or G will work equally well for any elastic stress field within the solid. Hence, this scheme also serves as the method to examine cracking behavior under thermal gradients. AMG065 output of nodal-point temperatures during thermal transients provides the input to **AXISOL** as thermal load. **AXISOL** then performs the stress analysis and computes the strain energy stored in the body for a specified crack length and thermal load. Numerical differentiation, as described above, provides the stress intensity factor.

5.3 BOUNDS TO THE EXACT SOLUTION

Since bounds are of importance both from an academic and a practical point of view the following discussion is directed toward the problem of determining when and in what direction the procedure outlined above could be expected to bound the true solution.

As mentioned previously the finite element method can and in fact is most easily viewed as a generalization of the Raleigh-Ritz method of approximate analysis. The function which is minimized is the potential energy (for the compatible displacement model finite element). The fact that the potential energy is directly related to the strain energy is probably the primary reason for the remarkable accuracy of the method presented herein.

5.3.1 Lower Bound on G

It is well known that the predicted strain energy is a lower bound to the true solution for the compatible displacement element model. However, since the stress intensity factor derived herein is the consequence of the subtraction of the strain energy estimates for two slightly different configurations, little can be said directly regarding bounds. What determines the bound on G is the way in which the quality of the strain energy estimate changes in moving from one configuration to the other. Since the strain within each element is constrained by the assumed deformation patterns within the element, the greater the strain gradient the more inaccurate the approximation will be. Therefore, since the stress intensity factor, K (or G), is directly related to the "magnitude" of the stress concentration and hence to the "steepness" of the strain gradient, it seems reasonable that if K is increasing with crack length then the strain energy estimate quality will be decreasing resulting in a lower bound on the predicted K (or G).

To make this idea a little more precise let the strain energy estimate for the initial crack length be given by

$$U_1 = \bar{U}_1 + \varepsilon_1$$

where U_1 = strain energy estimate of Configuration 1,
 $\bar{U}_1$ = exact strain energy for Configuration 1 and
 ϵ_1 = error associated with the strain energy estimate
for Configuration 1.

A similar relation holds for the case where the crack length has grown slightly, and is referred to as Configuration 2. Substituting such equations into Equation (20) for a plate of unit thickness yields

$$G = \frac{\bar{U}_2 - \bar{U}_1 + \epsilon_2 - \epsilon_1}{a_2 - a_1} \quad (21)$$

which reduces in the limit to

$$G = \bar{G} + \epsilon_G$$

where G = estimated value of the strain energy release rate,
 $\bar{G}$ = exact value of strain energy release rate and
 $\epsilon_G = \frac{\epsilon_2 - \epsilon_1}{\Delta a}$ = error associated with estimate.

Since U is underestimated using compatible elements, then ϵ_1 and ϵ_2 are negative. Then, as before, if G is increasing with crack extensions, ϵ_2 should be greater in magnitude than ϵ_1 so that ϵ_G will be negative. The validity of this argument rests heavily on the assumption that the element net changes associated with the crack extension, affect only the strain energy estimate because of the crack length change and do not produce significant side effects.

In the above examples the center-slotted and side-slotted panels under tension are examples where G increases with crack length and, as pointed out earlier, the predictions herein are less than the values given in the literature. Also note that since the other methods may well err in the opposite direction,

the error of the finite element procedure with respect to the true solution may even be smaller than that indicated.

5.3.2 Upper Bound on G

If G is decreasing with crack extension, then ϵ_1 exceeds ϵ_2 in absolute value, although both are still negative, and ϵ_G is positive thus producing an upper bound on G .

To check for the other bound the center-slotted infinite length panel with a center wedge load was solved using the finite element scheme (see Figure 7). The element layout and displacement boundary conditions were similar to Figure 4 except that the quarter-plate was taken longer (210 versus 85) to better simulate the infinite length situation. Also the loading was changed from an end tension to a vertical concentrated load applied at the left edge (which places it at the center of the full simulation, of course) of the plate. The results are given in Table 4 with a comparison to an approximate closed form solution taken from Reference 7.

Note that the finite element values are greater than the reference values. It can be seen then that the limited results presented herein bear out the bounds concept outlined before. It should be made clear that no claim is being made that the statements on bounds have been proven. Clearly this is not so and a much more precise argument involving numerical differentiation as well as a more definitive relationship between K and the quality of the finite element estimate of the strain energy is required.

The converse argument applies to an element model that satisfies equilibrium throughout the structure. (For a discussion of equilibrium models see Reference 13.) The combination of upper and lower bounds would provide a much needed assurance of accuracy without resorting to finer gridworks.

FIGURE 7. Center Cracked Panel with Center Wedge Loads.
 $L = 210$, $b = 40$, a is Variable.

TABLE 4. Values of K_I/P for Cracked Infinite Strip with Center Wedge Loads. Length to width ratio of finite element simulation = 5.25. a as indicated, $b = 50.0$ in.

a , in.	Plane Strain, $\nu = 0.3$		$\frac{K_I}{P}$, $\frac{1}{\sqrt{\text{in.}}}$	$\frac{K_I}{P}$, $\frac{1}{\sqrt{\text{in.}}}$	Diff %
	$\frac{UE}{P^2}$	$\frac{E}{P^2} G$			
9.0	1.43052				
9.5		0.03539	0.1972	0.1919	2.8
10.0	1.46591				
10.5		0.03293	0.1902	0.1845	3.1
11.0	1.49884				

$$a. \quad \frac{K_I}{P} = \frac{1}{\sqrt{\pi a}} \left[\frac{E}{\pi a} \sin \frac{\pi a}{b} \right]^{-1/2} \quad \text{from Reference 7.}$$

5.4 GENERALIZATION OF THE METHOD

The procedures outlined above may be applied directly to any geometry desired which enables one to apply the method to real engineering problems as well as to specimen analysis. The finite element code used herein was for 2-dimensional problems only, either axisymmetric or plane, but the method applies equally well to the 3-dimensional case. However, a problem which appears immediately is that in the general case the distinction between crack surface displacement Modes I and II is lost. This at first appears disturbing, since almost all material fracture data are classified by mode. However, this apparent problem becomes less significant when one considers that nearly all such data is for Mode I. The writers know of only two apparently successful attempts to measure K_{IIc} (References 14 and 15). Erdogan and Sih (Reference 14) find that K_{Ic} and K_{IIc} are about equal in value for plexiglass ($K_{Ic} = 400 \text{ psi}\sqrt{\text{in.}}$ and $K_{IIc} = 410 \text{ psi}\sqrt{\text{in.}}$). However, Corten (Reference 15) states that $G_{IIc} \gg G_{Ic}$ and gives more experimental data where K_{IIc} is about four times K_{Ic} for a glass fiber in epoxy-resin matrix material. The presence of fibers in the material may well influence the relationship between K_I and K_{II} .

Mode II data are completely lacking for most materials of interest. Hence, for most practical purposes it is a justified and conservative procedure to say that $G_{IIc} = G_{Ic}$ which is to simply ignore the mode concept. The same conclusion can be reached for crack propagation. It should be pointed out, however, that with the above finite element procedure the modes could be separated with acceptable accuracy by applying the proper constraints in the vicinity of the crack. For example, the Mode II contribution could be suppressed by forcing the nodal points on opposite sides of the crack opening to move together in a direction parallel to the crack surface. In

a similar manner the Mode I contribution could be suppressed by forcing these same nodes to move together in a direction normal to the crack surface.

It can be seen that the proposed finite element method above offers a convenient, economical and suitably accurate method of determining stress intensity factors. Probably the most glaring weakness is one of knowing when the element mesh is refined enough to provide suitable accuracy. The problem would be met best by utilizing more than one mesh size and observing the sensitivity of the results or by providing a dual equilibrium model analysis and examining the size of the "gap" between the two solutions. The severity of this problem is diminished, however, when the overall accuracy of classical fracture mechanics is considered.

6. STRESS ANALYSIS - CSE VESSEL

Two types of loading are immediately evident which could possibly result in defect growth. First of all there is the presence of the internal pressure loading. Preliminary estimates (Reference 16) of this form of loading indicate a low undefected structure stress level in the vicinity of the defect. Nevertheless, this type of loading was considered first because it is of a nonself-relieving type and could lead to catastrophic failure.

A second type of loading considered was that created by thermal gradients set up because of the rapid drop of the fluid temperature from an initial temperature of 600 °F down to 212 °F after completion of the blowdown. This loading will be referred to herein as thermal shock loading. Depending on the blowdown nozzle size, placement of blowdown nozzle (bottom, middle, or top) and assumptions used on the heat transfer coefficient, different thermal loading conditions exist. These are detailed in Section 7 (Thermal Analysis).

A final loading not considered in this report is one caused by dynamic loads of the blowdown extension nozzles on the vessel (applies only when blowdown nozzle is Nozzle C). The primary reason for not considering this loading, is the degree of complexity and corresponding analysis time required. On the other hand, the initial estimates (Reference 16) indicated that the stress level associated with this dynamic loading were comparable to the thermal loading condition so the decision was made to consider only the thermal condition at this time.

6.1 ASSUMED DEFECT SIZE

An examination of the ultrasonic mapping of the defect indicated that the overall dimensions of an ellipse which would encompass all the area was about 16 in. by 5 in. with

the longer dimension "wrapping" itself around Nozzle C or in terms of the previously mentioned coordinate system, the long dimension is the θ direction. The plane of the defect was generally laminar, i.e., parallel to the shell mid-plane and at a depth of about 1.5 in. Within the defect ellipse defined above the actual defect areas were "spotted" with the largest single area of dimensions 12 in. by 3 in.

When the defect was "cored," however, it was discovered that the apparently "solid defect" regions were actually made up of many small defects still generally laminar but none (in the core samples) exceeding a maximum dimension of about 1/8 in. This would indicate that none of the actual small defects exceeded 1/2 in. in length in any direction. Although the small defects were generally laminar, they were scattered over a depth of about 1/2 in.

After some consideration it was decided that two situations would suffice for the study. First on the assumption that the general area was weakened by the presence of the small defects, an axisymmetric, viz., extending completely around the nozzle, crack was studied, parallel to, but 1.5 in. in from the outside surface, and 4-in. in length. Second, a similar crack but only 1/2 in. in length was considered. This latter configuration is probably the most realistic.

6.2 PRESSURE LOADING

Consider first the unperturbed stress field induced by pressure loading. A pressure of 3000 psi was taken as nominal although operating pressure rarely exceeds 2400 psi. The finite element net used for the undefected configuration is shown in Figure 2. The stress components along the failure plane located 1.5 in. in from the outer surface, are shown in Figure 8. The primary conclusions to be drawn from these results are first that it is very unlikely that pressure type

FIGURE 8. Stress Components Occurring on Failure Plane -
 Undefected Structure; 3000 psi Internal Pressure
 Loading

loading caused the defect originally because of the very low shear stress and the compressive stress normal to the failure plane. Second, because of the large tensile membrane type stress, flaw growth would be expected to occur radially, i.e., normal to the defect plane, provided the introduction of the defect does not drastically change the overall stress field. This, of course, contradicts the present defect shape unless it has always existed in its present form or else there exists a "weakened" plane in that area. If growth does occur in the laminar direction because of this pressure loading, it most certainly is associated with a Mode II type stress field.

Consider now the situation where the defect is introduced. The element idealization for the 1/2 in. and 4-in. defects are shown in Figures 9 and 10. Before discussing the stress intensity factors involved, consider the stress state along and at each end of the defect. The results for the 1/2 in. and 4-in. defects are shown in Figures 11 and 12.

First of all it should be noted that the stress values near the crack tips must be interpreted carefully because of the singularities present in the exact elastic solution at those points. However, there is some useful information to be extracted from this data.

For both cracks the stress perturbations show indications of Mode II type stress field. This is most evident from the large jump in magnitude of the σ_s component that occurs across the defect face near the crack tip. The 1/2-in. crack indicates that growth, if it occurs in a laminar direction, will most likely occur in a direction away from Nozzle C although there is little difference. The 4-in. defect shows a definite preference for growth toward Nozzle C.

Growth of the defect into the nozzle appears to be the more serious situation. Therefore, stress intensity values associated with growth toward the nozzle were examined.

FIGURE 9. Element Idealization of Nozzle C - Vessel Intersection with One-Half Inch Defect Present

FIGURE 10. Element Idealization of Nozzle C - Vessel Intersection with 4-in. Defect Present

FIGURE 11. Stress Components in Elements Near Defect - 1/2-in. Crack

FIGURE 12. Stress Components in Elements Near Defect - 4-in. Crack

Let us first consider the 1/2-in. crack. The element mesh was altered in the region near the crack tip as shown in Figures 13B and 13C. Following the procedures outlined in Section 5 the following values for the stress intensity factor were found:

$$\begin{array}{ll} 7/16\text{-in. crack} & K = 0.56 \text{ ksi-}\sqrt{\text{in.}} \\ 17/32\text{-in. crack} & K = 0.61 \text{ ksi-}\sqrt{\text{in.}} \end{array}$$

Although the crack behavior under these conditions will not be discussed until later (Section 8) it is apparent that, although growing with crack extension, these are very low values.

a 112 Inch Crack Element Mesh

b. 3/8 Inch Crack Element Mesh

c. 916 Inch Crack Element Mesh

FIGURE 13. Element Meshes Used in Vicinity of Defect for Nominal 1/2-in. Crack

The same procedure for the 4-in. crack gives:

$$3\text{-}15/16\text{-in. crack} \quad K = 3.89 \text{ ksi-}\sqrt{\text{in.}}$$

$$4\text{-}1/32\text{-in. crack} \quad K = 4.00 \text{ ksi-}\sqrt{\text{in.}}$$

where the extended-crack meshes for the 4-in. crack size are similar to those in Figures 13B and 13C. Comparison of these results with the formula for K of an infinite cracked plate under shear at infinity, $K = \tau \sqrt{\pi a}$, where τ is taken from Figure 8 as 0.60 ksi, shows that the K 's for the 1/2-in. crack compare well, whereas, the 4-in. results are underestimated by a factor of three. In all fairness to the approximate method it should be pointed out that there is a contribution to K in the finite element method from the compressive stresses across the failure zone. This contribution could have been eliminated, but to do so was not in the interest of conservatism. It is possible that crack growth may indeed be influenced by compressive stress.

There is, of course, the possibility of the crack "turning" and growing towards either the inner or outer surface. Inspection of the two possibilities reveals that the worst situation is growing into the inside since, upon penetration, the nozzle would be "held on" only by a thin portion which would be highly vulnerable to complete failure by tearing (see Figure 14). In addition, inspection of the stress field associated with the flaws indicates that the highest tensile stress occurs on the surface towards the inside of the shell at the end of the crack toward Nozzle C (for the 4-in. crack). This stress is seen to be about 10 ksi at the center of the first element beyond the crack tip.

For the turning situation, a slightly different finite element net was employed to gain "room" for the crack extension. Figure 15 shows the element net employed in the crack region.

FIGURE 14. Consequences of Crack "Turning" and Extreme Growth

FIGURE 15 Element Idealization in Vicinity of Defect Used for Crack "Turning" Study

The resulting stress intensity factors are as follows (loading is 3000 psi internal pressure; basic 4-in. laminar crack);

<u>Length of 90° turn extension. in.</u>	<u>K k s i -</u>	<u>$a\sqrt{\pi a}$ ksi-$\sqrt{\text{in.}}$</u>
1/16	3.90	1.9
3/16	6.19	3.3
5/16	7.66	4.2

It is interesting to note that if the membrane stress existing at this point in the shell (-6.0 ksi) and the "turned" length (not the total crack length) are used in conjunction with the usual approximate formula (exact for an infinite cracked plate), $K = \sigma\sqrt{\pi a}$, where a is one-half the crack length, fair agreement results with the finite element procedure. The results from such a calculation are shown in the third column above. Although somewhat low, these approximate predictions are not grossly so, which says that in this situation the presence of the laminar crack has only a moderate affect on crack behavior at 90 degrees. In addition to being of interest in itself this also obviates the need to examine the tendency of the 1/2-in. crack to turn since the results will be very similar and somewhat less than the above.

7. THERMAL ANALYSIS - CSE VESSEL

7.1 THERMAL SHOCK BOUNDARY CONDITIONS

At the initiation of any given blowdown, the vessel temperature is close to uniform throughout and is approximately the same as the fluid. For the nominal hot blowdown this is about 600 °F. As soon as blowdown commences and the pressure drops to saturation, a rapid fluid temperature drop begins where, by the mechanism of flashing, the fluid temperature remains at the saturation point while the pressure drops to atmospheric in 8 to 100 sec depending upon the nozzle break size. This, of course, means that the vessel experiences a severe thermal shock as the fluid quickly drops to 212 °F.

The phenomenon of heat transfer across the vessel-fluid interface is a complicated one. Among others, temperature drop across the interface, pressure, surface character and orientation, type of fluid and metal involved are parameters that enter into the situation. A rather common model which was utilized herein was to employ the approximation that the heat flux across the interface is linearly proportional to the temperature drop across the interface, viz., $\frac{q}{A} = h\Delta T$

where $\frac{q}{A}$ = heat flux $\frac{\text{Btu}}{\text{ft}^2\text{hr}}$,

h = heat transfer coefficient and

ΔT = temperature drop across the interface

The constant of proportionality, the heat transfer coefficient, is then treated as a constant with respect to time although it may be changed at each time increment in the step-by-step solution of the thermal transient.

A simplified formula, developed by Jakob and Hawkins (Reference 18) was used to estimate h as follows:

$$h = 0.64 (q/A)^{\frac{3}{4}} \left(\frac{P}{P_{\text{atm}}} \right)^{0.4}$$

$$5000 < q/A > 75,000$$

$$h \sim \frac{\text{Btu}}{\text{hr ft}^2 \text{ } ^\circ\text{F}}$$

P = absolute pressure

P_{atm} = standard atmospheric pressure

Using the upper limit of applicability of this formula, $q/A = 75,000 \frac{\text{Btu}}{\text{hr-ft}^2}$ gives an h of 2900 Btu/hr-ft² °F for a pressure of one atmosphere. This value is probably much too high since the maximum flux rate possible under these conditions according to Zuber (Reference 19) is approximately 35,000 Btu/hr-ft². On the other hand, the presence of about 1/2-in. of stainless steel cladding on the CSE pressure vessel inner wall in the vicinity of Nozzle C provides a pseudo poor conductor skin relative to the carbon steel. This effect acts in series with the interface coefficients and results in a low sensitivity of the bulk of the vessel to changes in h provided it remains above about 500 Btu/hr-ft². Also, from a stress analysis point of view, maximum stresses are usually associated with maximum thermal gradients, so that using the larger value for h is conservative.

Since the heat transfer coefficient and fluid temperature are highly dependent upon pressure, it was considered necessary to adjust the value of h in the initial portion of the blow-down. The time between the start and end of the active blow-down is therefore divided into several periods called time sequences in which the value of h and environmental temperature are based on the fluid pressure and temperature existing at the midpoint of the time sequence. Each time sequence is in turn made up of several time increments. This provides a

"stepping down" of the coefficient and environmental temperature. This procedure is considered to produce satisfactory accuracy.

After considering all of the possibilities of various combinations of nozzle sizes, nozzle locations, initial conditions, etc., two situations were isolated as being the most severe and were modeled. The first of these, referred to as Thermal Case A, corresponds physically to the blowdown where a small nozzle (2-in.) located in the top or middle of the vessel is used (so that water remains in the bottom of the vessel completely covering the Nozzle C area) and where the initial temperature is maximum (600 °F). The reason for the small nozzle is to provide maximum time with the higher values of h . This condition also represents one bound on the bottom blowdown situation. Since the bottom blowdown requires the longest time to complete, a 2-in. bottom blowdown was used as the model in this case.

The other thermal condition considered, referred to as Thermal Case B, corresponds to the situation where a bottom nozzle is used and the blowdown nozzle diameter is maximum (8 in.). Again an initial temperature of 600 °F is used. In this case it is assumed that after the active portion of the blowdown is over then only dry steam is in contact with the inner nozzle area, whereas, the bottom of the vessel remains in contact with water. Since the analysis is constrained to be axisymmetric, this, in effect, puts all of the remainder of the vessel interior in contact with water. Also, since the value of the heat transfer to steam is much smaller than the transfer to water the assumption is made that the inner portion of the nozzle becomes insulated after active blowdown ceases. This leads to higher axial gradients than existed in Case A. In reality this case represents one extreme for the bottom blowdown as actual thermocouple data showed some cooling in the nozzle area.

The actual boundary condition data used are summarized in Table 5.

TABLE 5. Thermal Boundary Condition Data for Thermal Analysis

<u>Sequence Number</u>	<u>Time at End of Sequence, sec</u>	<u>Gage Pressure at Midpoint of Sequence, psi</u>	<u>Fluid Temperature During Sequence, °F</u>	<u>h Used During Sequence, Btu/ft²-hr °F</u>
<u>Thermal (a)</u>				
1	30	1290	580	17,400
2	60	1010	550	15,800
3	70	620	490	13,100
4	80	245	400	9,100
5	95	50	300	5,200
6	500	0	212	2,900
<u>Thermal (b)</u>				
1	2	1420	590	18,100
2	4	1050	552	16,100
3	6	540	478	12,300
4	8	65	312	5,700
5	500	0	212	2,900

Thermocouple data for Run B-19-B showed that complete boil-off occurred at 420 sec after blowdown initiation or about 325 sec after the active blowdown ceased. Nevertheless, the thermal simulations were carried out to 500 sec as though water was still present.

7.2 THERMAL ANALYSES

A finite element formulation of the heat conduction equations and associated computer code, Section 4, was used to solve the thermal portion of the problem. The element mesh used was almost identical with that used for the solid shell stress solution, Figure 2, except that some variation in the

location of the interior nodes was made to account for the 1/2-in. of stainless cladding. The first two elements made up the cladding leaving the other elements for the remaining 3-1/2 in. of carbon steel (in the 4-in. section, of course). This provided for elements of dimensions 1/4-in. in the direction of maximum gradient for the cladding and a nominal 0.6-in. thick element for the rest of the vessel. This, of course, gave minimum element size where the temperature gradients were maximum which is desirable.

To gain some confidence in the boundary conditions selected for the analysis a comparison of predicted temperature on the outside surface of the shell versus time is compared with some actual thermocouple data. This particular thermocouple data was obtained by averaging data from three thermocouples identified as TB06, TB09 and TB013, all exterior surface gages, which are far enough away from the nozzle to be virtually unaffected by the presence of the nozzle. The comparison is shown in Figure 16.

Even though the initial fluid temperature for this run was close to 600 °F, the outer shell temperature was initially only about 580 °F because of nonequilibrium or thermal leakage. After about 180 sec the thermocouple data show an abrupt downward turn and from then on it is quite close to the predicted temperature. The temperature-time gradient is also shown, and even though the thermocouple data show considerable scatter, the correlation is quite good after about 200 sec.

Finally a comparison of the analytical and experimentally observed thermal profiles through the thickness of the shell is shown in Figure 17. These profiles are given at 180 sec after blowdown initiation which turns out to be the critical one. The correlation is seen to be good.

FIGURE 16. Comparison of Analytical and Experimental Temperature at Outer Surface of Shell, Away from Nozzle C; Thermal Solution A and Average of Thermocouples TB06, TB09 and TB013, Run B-19-B

FIGURE 17, Comparison of Experimental and Analytical Thermal Gradient in Bottom Head of CSE Pressure Vessel

7.3 CRITICAL TIME SELECTION

By looking at such parameters as the root mean square of the temperature variance from uniform temperature gradient radially across the failure zone or temperature gradient axially in the failure zone, several critical temperatures were selected. Stress analyses of the undefected shell were then run using these several temperature distributions. Using stored elastic strain energy and the normal or shear stress components at the failure surface as guides, a final critical temperature was selected. For thermal Case A this occurred at a time of 180 sec after blowdown commenced.

The critical situation for thermal Case B was a little more difficult. First of all the maximum axial gradient had not been reached at $t = 500$ sec. Continuation of boil off after 500 sec was considered so unrealistic that it was not considered worth the computer time to continue any further. In addition the axial gradient was apparently nearing the maximum. Comparison of the same parameters as above then led to the conclusion that the critical temperature gradient occurred at $t = 140$ sec.

7.4 THERMAL STRESSES

The finite element nets used in the thermal stress analyses are identical with those used previously. The resulting stresses in the undefected shell along the failure zone for Thermal Case A are shown in Figure 18. The stresses associated with Thermal Case B were similar in shape and almost everywhere less severe. Several observations can be made based on this figure. First, it is noted that the dominant cracking mechanism possible is the tensile stress across the failure plane and that this stress is dropping to zero and going negative at the end of the failure zone toward the nozzle. Secondly, one notes that the stress component parallel to the

FIGURE 18. Stress Components on Failure Plane, Undefected Structure. Thermal loading A.

defect plane is compressive and, therefore, there is no need to even consider the turning phenomenon discussed earlier in connection with the pressure loading. Finally, it can be concluded that there is no need to consider Thermal Case B any further as it is less severe than Case A. This final observation was confirmed by checking the stress intensity factors for the 4-in. crack.

Introduction of the 1/2-in. crack in the same position as before (see Figure 9) puts the end toward Nozzle C into a zone of compressive stress as can be seen in Figure 18. The 1/2-in. size is small enough not to disturb the gross field much and, therefore, the possibility of cracking in the direction toward Nozzle C becomes remote. The question of crack propagation under alternating compressive stress is not well understood. Therefore, for the 1/2-in. defect it suffices to say that there is a greater tendency for extension to occur in a direction away from Nozzle C and to obtain an estimate of the stress intensity factor from a straight forward application of $K = \sigma\sqrt{\pi a}$ using the maximum value of σ (within the failure zone) from Figure 18. The resulting K for the 1/2-in. defect is 2.3 ksi- $\sqrt{\text{in.}}$.

On the other hand, when the 4-in. defect is introduced into the failure zone as before, tensile stresses develop ahead of the crack toward Nozzle C. This can be seen in Figure 19 where the stress components in the defect vicinity are given. In addition it can be seen that there is a substantial Mode II shear contribution in the end towards Nozzle C. This is evidenced by the rising value of τ_{sn} as it approaches the crack tip from ahead and by the large jump in σ_s as one moves across the defect on the other side of the crack tip.

As before, application of the finite element technique results in the following values when the crack extension is toward Nozzle C:

FIGURE 19. Stress Components on Failure Plane with 4-in. Defect Present. Thermal Case A, $t = 180$ sec.

<u>Crack Length,</u> <u>in.</u>	<u>K,</u> <u>ksi-$\sqrt{\text{in.}}$</u>
3 15/16	7.97
4 1/32	7.51

Note that the stress intensity factor is reducing, in this case, as crack extension proceeds.

Consideration of crack extension in the direction away from Nozzle C results in significantly larger values for K which are seen to increase with crack extension

<u>Crack Length,</u> <u>in.</u>	<u>K,</u> <u>ksi-$\sqrt{\text{in.}}$</u>
3-15/16	10.20
4- 1/16	10.56

These values represent the largest stress intensity factors found in the study.

8. DEFECT BEHAVIOR

This section investigates the manner in which the subject defect responds to pressure and thermal shock loads whose stress analyses were detailed in previous sections.

8.1 FRACTURE TOUGHNESS

First of all we consider the possibility of unstable crack growth. Because of the axisymmetric nature of this structure, the crack tip area is under a high degree of constraint in such a manner that a condition of plane strain normal to the crack leading edge is much more realistic than plane stress. Reference 9 gives estimates of the minimum fracture toughness of the material in question which is Type A212B steel. However, this property is highly temperature-dependent and estimates are given only for the 20 to 140 °F temperature range. In reality this presents no problem since the lowest numerical value given, corresponding to a temperature of 20 °F, is $K_{Ic} = 47 \text{ ksi}\sqrt{\text{in.}}$ which is significantly greater than any of the stress intensities found in this study. One word of caution here is that all presently available experimental data on critical stress intensity values are based on Mode I behavior whereas the K estimates found in this study sometimes include K_{II} contributions. However, this is not considered cause for concern because of the wide margins of safety indicated here.

Perhaps the most worrisome situation is the possibility of the defect turning and growing as a result of internal pressure at high temperature (which accelerates growth) after which pressure could be imposed (and in fact has been) at low (40 to 60 °F) temperatures where the possibility of unstable growth is maximum. However, projecting the K data found herein for the turning situation on a $\sigma\sqrt{a}$ and net section stress basis indicates that the crack would have to grow to a length of about 2.3 in. normal to the failure plane before unstable growth would occur.

8.2 CRACK GROWTH

Let us next consider the crack growth possibilities. Reference 20 shows that elevated temperature increases the crack growth rate for Type A212B steel and, therefore, the data given in Reference 20 for that metal and the 600 to 650 °F temperature range are utilized. The stress intensity ranges for the data are higher than were needed here so a variation of form

$$\frac{da}{dN} = C (\Delta K)^n$$

was utilized to project the data from Reference 20 back into the desired stress range. Such extrapolation could lead to error, but it is considered acceptable in this case, since the extension is to be used for smaller values of AK. This procedure yields the formula:

$$\frac{da}{dN} = 6.9 \times 10^{-7} (AK)^{4.95}$$

where $\frac{da}{dN}$ = growth rate in μ in./cycle and

AK = fluctuation range of stress intensity factor.

To apply this formula to the 90 degree turn situation for the pressure loading, it is first necessary to estimate the initial flaw size. Since the core samples indicated laminar flaws spread over a depth of about 1/2 in., a reasonable initial flaw size may be taken as 1/4 in. Again using the previous data for the 4-in. laminar crack with a "turn" of 5/16 in. to provide a base point on which to project K, estimates may be made of the growth behavior of the crack. This produces a prediction of 9×10^6 cycles necessary for the flaw to grow from 1/4 to 1/2 in. long. Using the same procedure, the estimate for growth from 1/2 to 1 in. is 2×10^6 cycles. Since the maximum frequency that could be expected is about 100 cycles per year, the above margins appear safe.

Consider next, laminar growth of the 4-in. defect caused by the most severe thermal shock loading, Thermal Case A. Using the stress intensity values given previously (one cycle is taken here to be equivalent to one blowdown; heat-up is assumed to be slow enough that significant gradients do not develop) with the above crack propagation law gives the following estimates for the number of cycles required for the defect to grow 1/4 in. in each direction:

toward Nozzle C	17×10^6 cycles
away from Nozzle C	2×10^6 cycles

As can be seen, these are very low growth rates and for that reason the less severe 1/2-in. crack is not considered.

NOTATION

A	= Crack opening area
a	= Crack length
C_{ijmn}	= Elastic constants
c	= Specific heat
E	= Young's modulus
G	= Strain energy release rate, general
G_I, G_{II}	= Strain energy release rate, Mode I or II
g	= Film coefficient for convective boundary condition
h	= Heat transfer coefficient
K	= Stress intensity factor, general
K_{Icr}, K_{IIcr}	= Stress intensity factor, critical
K_{cr}	= Stress intensity factor, critical
k	= Thermal conductivity tensor
N	= Number of load cycles
n_i	= Number of sides for i th element
$\hat{n}$	= Unit normal vector on element boundary
$\bar{q}$	= Specified flux vector
q/A	= Heat flux
r, θ	= Polar coordinates
$\underline{r}$	= Matrix of nodal displacements
S	= Surface of body
S_T	= Surface of body where tractions are prescribed
S_{ij}	= j th side of i th element
$\underline{s}^j$	= Column vector of nodal displacements for j th element
ΔT	= Temperature drop
$\bar{T}_i$	= Components of prescribed stress vector tractions

- T = Temperature
 $\Delta \bar{T}$ = Prescribed jump in stress vector
 U = Strain energy
 $U_o, \dot{U}_o$ = Temperature field and time rate of change of temperature field respectively in an element
 u = Vector of nodal-point temperatures
 $\dot{u}$ = First derivative of u with respect to time, t
 u_i^* = Virtual displacement
 u_i = Components of a trial displacement vector
 V_i = Volume of i th element
 V = Volume of body
 x, y, z = Cartesian coordinates
 a = Coefficient of linear thermal expansion
 ϵ_{ij} = Components of strain tensor
 ϵ_{im}^T = Components of thermal strain tensor
 ϵ_{ij}^* = Strain associated with u_i^*
 p = Density
 $\sigma, \sigma_r, \sigma_x$, etc = Stress component, subscript indicates direction
 σ_{ij} = Stress tensor components
 v = Poisson's ratio

REFERENCES

1. E. L. Wilson. "Structural Analysis of Axisymmetric Solids," AIAA Journal, vol 3, no. 12. December 1965.
2. R. W. Clough and Y. Rashid. "Finite Element Analysis of Axisymmetric Solids," J. Engineering Mechanics Division, ASCE, vol 91, no. EM1, February 1965.
3. M. J. Turner, R. W. Clough, H. C. Martin and L. J. Topp. "Stiffness and Deflection Analysis of Complex Structures," J. Aeronautical Sciences, vol 23, no. 9. 1956.
4. E. B. Becker and C. H. Parr. Application of the Finite Element Method to Heat Conduction in Solids, T. R. S-117. U.S. A. Missile Command, Redstone Arsenal, Alabama, November 1967.
5. P. Rosen. "On Variational Principles for Irreversible Processes," J. of Chemical Physics, vol 21, p. 1220. 1953.
6. O. C. Zienkiewicz and Y. K. Cheung. The Finite Element Method in Structural and Continuum Mechanics. McGraw-Hill Co., N.Y., 1967.
7. P. C. Paris and G. C. M. Sih. "Stress Analysis of Cracks," Fracture Toughness Testing and its Application, ASTM Special Tech. Pub. No. 381. 1965.
8. G. R. Irwin. Encyclopedia of Physics, Elasticity and Plasticity. 1958. Vol 6.
9. G. R. Irwin, J. M. Krafft, P. C. Paris and A. A. Wells. Basic Aspects of Crack-Growth and Fracture, NRL-6598. Naval Research Laboratory, Washington, D.C., November 1967.
10. B. Gross, J. E. Srawley and W. F. Brown, Jr. Stress-Intensity Factors for a Single-Edge-Notch Tension Specimen by Boundary Collocation of a Stress Function, NASA-TN-D-2395. National Aeronautics and Space Administration, Washington, D.C., August 1964.
11. S. K. Chan, I. S. Tuba, and W. K. Wilson. On the Finite Element Method in Linear Fracture Mechanics. Presented at the 2nd National Fracture Mechanics Symposium, Lehigh U., June 17-19, 1968.

12. J. R. Rice. A Path Independent Integral and the Approximate Analysis of Strain by Notches and Cracks. Brown Engineering Report No. E39, May 1967.
13. V. B. Watwood and B. J. Hartz. "An Equilibrium Stress Field Model for Finite Element Solutions of Two-Dimensional Elastostatic Problems," Int. J. of Solids Structures, vol 4, p. 857. 1968.
14. F. Erdogan and G. C. Sih. "On the Crack Extension in Plates Under Plane Loading and Transverse Shear," J. Basic Engineering, ASME, December 1963.
15. H. T. Corten. "Influence of Fracture Toughness and Flaws on the Interlaminar Shear Strength of Fibrous Composites," Fundamental Aspects of Fiber Reinforced Plastic Composites; edited by R. T. Schwartz and H. S. Schwartz, Interscience Publishers, New York. Chapter 6.
16. Nuclear Safety Quarterly Report, February-March-April, 1968. BNWL-885, pp 2-26. Battelle-Northwest, Richland, Washington, October 1968.
17. V. B. Watwood. The Finite Element Method for Prediction of Crack Behavior, BNWL-SA-2383. Battelle-Northwest, Richland, Washington, February 18, 1969.
18. M. Jakob and G. Hawkins. Elements of Heat Transfer, John Wiley and Sons, New York, 1957. 3rd Ed.
19. N. Zuber. "On the Stability of Boiling Heat Transfer," Trans. Am. Soc. Mech. Eng., vol 80, p. 711. 1958.
20. A. J. Brothers. Fatigue Crack Growth in Nuclear Reactor Piping Steels, GEAP-5607. General Electric Company, Schenectady, N.Y., March 1968.
21. C. E. Pearson. Theoretical Elasticity, Harvard Monographs in Applied Science. No. 6. 1959.

BIBLIOGRAPHY

M. B. Reynolds. Fracture Mechanics and the Stability of Engineering Structures, GEAP-4678. Vallecitos Atomic Lab., General Electric Company, Pleasanton, California, September 1965.

G. R. Irwin. Structural Mechanics. Proceeding of the First Symposium of Naval Structural Mechanics, Pergamon Press, Long Island, N.Y., 1960.

W. F. Brown and J. E. Srawley. Plane Strain Crack Toughness of High Strength Metallic Materials, ASTM Special Tech. Pub. No. 410. 1966.

B. Gross and J. E. Srawley. Stress Intensity Factors for Single-Edge-Notch Specimens in Bending or Combined Bending and Tension by Boundary Collocation of a Stress Function, NASA-TND-2603. National Aeronautics and Space Administration, Washington, D.C., January 1965.

Stress Analysis. Edited by O. C. Zienkiewicz and G. S. Holister. John Wiley and Sons, New York, N.Y., 1965.

DISTRIBUTION

No. of
Copies

OFFSITE

1 AEC Chicago Patent Group
 G. H. Lee

215 AEC Division of Technical Information Extension

ONSITE-HANFORD

1 AEC Chicago Patent Group
 R. K. Sharp (Richland)

1 AEC Richland Operations Office
 C. L. Robinson

3 Battelle Memorial Institute

2 RDT Assistant Director for Pacific Northwest Programs

17 Battelle-Northwest
 W. E. Anderson
 T. L. Anderson
 F. G. Dawson
 J. C. Fox
 L. T. Pedersen
 G. J. Rogers
 R. A. Selby
 Engineering Mechanics Section File (3)
 Technical Information (5)
 Technical Publications (2)