The Application of Limit Analysis to Shell Structures Using Existing Finite Element Analysis Codes

1977

George G. Myers

University of Central Florida

Find similar works at: https://stars.library.ucf.edu/rtd

University of Central Florida Libraries http://library.ucf.edu

Part of the Engineering Commons

STARS Citation


https://stars.library.ucf.edu/rtd/362

This Masters Thesis (Open Access) is brought to you for free and open access by STARS. It has been accepted for inclusion in Retrospective Theses and Dissertations by an authorized administrator of STARS. For more information, please contact lee.dotson@ucf.edu.
THE APPLICATION OF LIMIT ANALYSIS TO SHELL STRUCTURES USING EXISTING FINITE ELEMENT ANALYSIS CODES

BY

GEORGE G. MYERS
B.S.A.E., Northrop University, 1962

THESIS

Submitted in partial fulfillment of the requirements for the degree of Master of Science in the Graduate Studies Program of the College of Engineering of Florida Technological University

Orlando, Florida
1977
THE APPLICATION OF LIMIT ANALYSIS TO SHELL STRUCTURES
USING EXISTING FINITE ELEMENT ANALYSIS CODES

by

George G. Myers

ABSTRACT

A ramjet inlet duct structure has been designed using nonlinear structural analysis to allow local plastic deformations which result in structural weight reduction. To establish the feasibility of reduced shell thickness, the Lower-Bound Theorem of limit analysis has been applied using an existing finite element analysis computer code intended for elastic analysis. The results of the analyses were verified by structural testing. Predicted stresses and deflections showed good correlation with measured values up to the point of failure. This work was supported by the U.S. Air Force Aero Propulsion Laboratory, Wright-Patterson Air Force Base, Ohio, under Contract No. F33615-72-C-1366.
ACKNOWLEDGMENT

The author would like to express his appreciation for the contributions made to the successful completion of this program provided by Russell Lenz, structural analysis and data reduction; James Luksa, test engineering; William Stimmel, Jr., structural test laboratory; John A. Berndobler, manufacturing; Dr. Ramon Mayor, failure analysis and weld evaluation; Mr. Franklin C. Moore for technical guidance; and Dr. David Jenkins for his guidance and patience.
CONTENTS

ABSTRACT ........................................ ii
ACKNOWLEDGMENT ................................ iii
NOTATION ........................................ vi
INTRODUCTION ................................. 1
Chapter
   I. ANALYTICAL APPROACH ................. 3
      Idealized Plastic Behavior .......... 3
      Limit Analysis Theory .............. 7
      Yield Criteria ....................... 12
      Finite Element Analysis of Elastic Structures 15
      Application of Limit Analysis Using Elastic
      Finite Element Codes ................ 23
   II. SUBSTANTIATION OF ANALYTICAL TECHNIQUE 29
      Finite Element Analysis of Two
      Dimensional Structures ............ 32
      Test of Two Dimensional Structures .... 36
      Correlation of Test and Analytical Results
         for Two Dimensional Model .......... 40
      Three Dimensional Structure .......... 44
<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Finite Element Analysis of Three Dimensional Structures</td>
<td>v</td>
</tr>
<tr>
<td>Test of a Three Dimensional Structure</td>
<td>46</td>
</tr>
<tr>
<td>Correlation of Test and Analytical Results for a Three Dimensional Model</td>
<td>51</td>
</tr>
<tr>
<td>III. CONCLUSIONS</td>
<td>54</td>
</tr>
<tr>
<td>REFERENCES</td>
<td>67</td>
</tr>
</tbody>
</table>

REFERENCES: 69
NOTATION

D  Flexural rigidity, in-lb (m-N)
E  Young's modulus, psi (Pa)
F  Membrane force stress function
h  Thickness, in (mm)
M  Bending moment per unit width, lb (N)
M_P  Fully plastic bending moment, lb (N)
m  Normalized bending moment
Nx, Ny, Nxy  Membrane forces, lb/in (N/m)
n  Normalized membrane force
p  Normal surface load, psi (Pa)
P^-  Lower bound yield point load
Po  Yield point load
P^+  Upper bound yield point load
Pc  Collapse load
R  Radius of curvature, in (mm)
u  Plate parameter
w  Normal deflection, in (mm)
x, y, z  Cartesian coordinates, in (mm)
\nabla^4  Biharmonic operator
\sigma  Strain, in/in (mm/mm)
\sigma_y  Yield strain
\sigma'_y  Yield stress, psi (Pa)
INTRODUCTION

The classical approach to the analysis of missile structures is to assure that all structural elements are maintained in an elastic state. Elasticity is assured in peak load areas by providing structural support through the addition of doublers that are mechanically attached or welded to the structure in question. Stiffened areas are also achieved through chemical milling or machining of thicker material such that the unmilled area provides the stiffened areas. In some smaller missile structures, manufacturing costs are reduced by selecting a constant shell thickness which prevents yielding at the expense of unnecessary structural weight. A viable alternative to the above approach is to maintain the structural shell at a constant thickness and allow yielding to a point short of structural collapse. Structural collapse occurs when the load is increased to a level where plastic hinges form and static stability gives way to kinematic behavior. For structures which are used only once, as in the case of most missiles, structural deformation can be allowed so long as aerodynamic and structural performance are not degraded. This thesis demonstrates a lower bound theorem limit analysis approach for local yielding
of thin shell structures. Limit analysis is conducted through the use of existing elastic finite element analysis codes and the analytical results are verified through structural testing.

The lower bound limit analysis approach is relatively easy to use in conjunction with elastic finite element analysis codes and provides good correlation with test results.
CHAPTER I

ANALYTICAL APPROACH

Idealized Plastic Behavior

Elastic deformations are those characterized by complete recovery once the applied loads are removed from a structural element. Any deformation of a continuum that does not obey the constitutive laws of classical elasticity, neglecting instabilities, is defined as an inelastic, or plastic, deformation. Irreversible plastic deformations result from the mechanism of slip through dislocation, motion at the atomic level, and occur above the elastic limit of the material under loading.

Mase (1)\(^1\) states that in the theory of plasticity, the primary concerns are with the mathematical formulation of stress-strain relationships suitable for the phenomenological description of plastic deformations and with the establishment of appropriate yield criteria for predicting the onset of plastic behavior. Plastic flow is defined as the ongoing plastic deformation which may be related to the amount and rate of deformation. A solid in the plastic state can sustain shear stresses even when at rest.

\(^1\)Numbers in parentheses refer to references which are listed at the end of the thesis.
It is extremely difficult to consider realistically in equation form the effects of instantaneous elastic response, hysteresis, creep, and plastic flow under combined stresses. To make the engineering problems solvable, idealized stress-strain laws are defined.

In the use of the Lower Bound Theorem of limit analysis, the elastic-perfectly plastic stress-strain law is applied. This law assumes a constant strain rate until yielding occurs and then perfectly plastic deformation begins as shown in Figure 1. For a structural element in bending the elastic perfectly plastic stress-strain distribution is shown in Figure 2a. Load is applied until yielding occurs (Point A) and then additional load cannot be carried even though strain continues. The segment AB corresponds to plastic flow under constant tensile stress; the segment BC to complete elastic unloading and subsequent loading in compression; and segment CD to plastic flow under constant compressive stress. The slope of segment OA corresponds to Young's Modulus. The stress distribution as a function of shell thickness is assumed to be trapezoidal as shown in Figure 2b. The elastic-perfectly plastic material characteristic lends itself well to the limit analysis of some ductile metallic structures and is a good approximation for many other metals which are strain hardenable.
Figure 1  Elastic-Perfectly Plastic Stress-Strain Law
a. Stress-Strain Distribution

![Stress-Strain Distribution Diagram](image)

b. Stress Distribution

![Stress Distribution Diagram](image)

Figure 2  Elastic-Perfectly Plastic Element in Bending
Limit Analysis Theory

Limit analysis methods define the load carrying capability of rigid perfectly plastic continua of one, two and three dimensional structures, Prager (2). Rods, beams, arches, and rings represent a one dimensional state of stress of limit analysis. Membranes, flat sheets with inplane stress, plates and shells represent two dimensional states of stress. The design of shell structures based on elasticity fails to take advantage of the ability of the structure to support loads above the initial yield capability. The ductility of redundant structures permits a redistribution of loads beyond the elastic limit state of stress. This redistribution can often result in carriage of additional loads without total structural failure. Lower bound limit analysis of three dimensional shells is feasible using one dimensional state of stress and finite element analysis. These methods combine to allow progressive plastic hinge formation with increases in loading until the shell structure becomes a mechanism and structural collapse occurs. This method of analysis and its application to shell structures is the subject of this thesis.

Two fundamental theorems of limit analysis provide bounds on the load which permit load redistribution in redundant structures and are concerned with the question of whether the given loads are or are not capable of producing unconstrained deformations in a rigid, perfect plastic continuum, Hodge (3). In many problems the yield point load is difficult to define but in the case of
these two theorems of limit analysis, the yield load can be bounded. The Lower Bound Theorem is stated in terms of static admissibility. A statically admissible field is defined by Prager and Hodge (4) as a stress field which satisfies all equilibrium conditions throughout the body, whose stresses are below yielding throughout the body, and satisfies the boundary conditions. The Lower Bound Theorem states that if there exists any statically admissible stress field for the load $P^-$, then $P^-$ is the lower bound on the yield point load.

$$P^- \leq P_0$$

The statement of the Upper Bound Theorem begins with any velocity field which satisfies the velocity boundary conditions. The theorem states that if there exists any kinematically admissible field for the load $P^+$, then $P^+$ is the upper bound on the yield point load:

$$P_0 \leq P^+ \quad P_0 \leq P^+$$

A kinematically admissible field is defined by Prager and Hodge (4) as a strain rate field derived from a velocity field which satisfies the boundary conditions.

The complete solution of limit analysis can only be found if the yield point is both statically and kinematically admissible. This thesis will deal with approximate yield conditions which are statically admissible, i.e., the Lower Bound Theorem.
The fixed end beam shown in Figure 3 illustrates the application of limit analysis to structural design when the load is allowed to exceed the elastic limit load, \( P_0 \). Part a of the figure illustrates the formation of plastic hinges where the bending moment reaches the critical value of plastic moment, \( M_p \), during a monotonic application of load. The plastic moment occurs when the entire cross section of the beam has reached the yield stress in tension or compression, and the strains are assumed, as in elastic beams, to be linear through the thickness. The deflection of the beam is linear up to the elastic limit load, \( P_0 \), where the first plastic hinge forms.

For loads above this value the rate of deflection increases until the collapse load, \( P_c \), is reached; at which time the deflections become unbounded and the beam forms a kinematically unstable mechanism. Inspection of the moment distribution along the beam at the elastic limit load and for the collapse load shows that the bending moment is reduced over portions of the beam near the right support for loads above the elastic limit load. Also, it should be noted that the deflections of the beam are in excess of those predicted by linear elastic analysis for loads above \( P_0 \).

The plastic analysis provides an increase in working load for the structure with local yielding occurring only at the left support. For the majority of its life, the structure will operate below the elastic limit, assuming that during its life the load increases monotonically.
Figure 3  Limit Analysis Load-Deflection
The application of limit analysis to the design of actual structures must include the following considerations for establishing the maximum permissible limit loading in addition to the collapse load:

1. The deflections of the structure must not impair the performance of the shell structure;
2. The local strains at plastic hinges must not exceed the ductility of the materials;
3. Cyclic loading must be such as to preclude fatigue failure; and
4. Manufacturing and processing flaws must not cause brittle fracture.
Yield Criteria

The most important quantity required for an accurate limit analysis is the value of the plastic moment, $M_p$. In the case of a beam, the plastic moment is calculated by a simple rotation of the cross section about the neutral axis. For more complex structures, e.g., the inlet/duct, strain in the middle surface of the shell must be included. Therefore, a yield criterion involving all the stress components is required. Hodge (3) presents the following criterion for combined bending and normal forces:

$$ m + n^2 \leq 1 $$

where $m$ is the normalized bending moment and $n$ is the normalized normal force. These quantities are defined as:

$$ m = M/M_p \quad \text{and} \quad n = N/N_p $$

where $M$ and $N$ are the applied forces in the structure and the limit values are given in terms of the yield stress, $\sigma_Y$, and thickness, $h$, as:

$$ M_p = \frac{1}{4} \sigma_Y h^2 \quad \text{and} \quad N_p = \sigma_Y h. $$

When the loading combination at a point in the structure becomes such that the yield criterion in terms of the normalized forces equals unity, a plastic hinge forms and the structure cannot transmit any additional load at that point.

Since the ductility requirements at a plastic hinge are also of interest, the relationship between the strain and the forces is required. Figure 4 gives this relationship in terms of the yield
Strain/load relationships were determined in terms of the yield strain for an elastic-perfectly plastic materials.
strain, $\varepsilon_Y$, for an elastic-perfectly plastic material. The curves show that the moment, $M$, becomes asymptotic to the plastic moment, $M_p$, for maximum strains less than four times the yield strain. Most materials of interest have a capability in excess of 15 to 20 times the yield strain. This situation allows the plastic moment to be approximated by simple rectangular stress distribution neglecting the actual stress, and provides confidence that the resulting strains calculated by limit analysis procedures will be finite.
Finite Element Analysis of Elastic Structures

In structural design, the stress analyst oftentimes must determine stress fields and deflections of large redundant structures. The finite element analysis method provides a system for representing the structural stiffness or flexibility through the mathematical assembly of finite elements which are interconnected in a way to accurately represent the redundant structure. General purpose computer codes created for structural analysis usually have a variety of elements which permit mathematical modeling of the structures. These elements include but are not limited to framework, plane stress, solid, flat plate bending, axisymmetrical solid elements. The element stiffness or flexibility matrices are derived through strain energy equations. Details for derivation of element stiffness and flexibility matrices and finite element analysis methods are beyond the intended scope of this thesis and are presented in many textbooks and reports including those by Gallagher (5) and Rubinstein (6).

The Martin-Marietta Space Frame computer code used in this study incorporates the stiffness (deflection) method for structural analysis. The procedure for the stiffness method, as given by Rubinstein (6), is as follows:

1. Define the system coordinates at the intersections of elements where displacement can take place. These displacements are to be independent so that a corresponding stiffness matrix $[K]_s$ exist such that

$$\{F\}_s = [K]_s \{u\}_s$$
2. Select elements so that the system coordinates of Step 1 occur only at their ends. Select for each element \( s \) element coordinates for which a stiffness matrix \([k]\) exists, so that

\[
\{p\}_s = [k]_s \{d\}_s
\]

or

\[
\{p\} = [k] [d]
\]

for all elements. Use the element coordinates to account for all significant energy forms.

3. Construct a transformation matrix \([\beta]\) which insures compatibility by relating element displacements \([d]\) to system displacements \([u]\)

\[
\{d\} = [\beta] \{u\}
\]

4. Synthesize the system stiffness matrix \([k]\) using

\[
[k] = [\beta]^T [k] [\beta]
\]

5. Apply to the structure a superposition of forces followed by a superposition of displacements. The fixed coordinate forces \([F]^o\) are computed from the corresponding forces \([F]_s^o\) at the element coordinates such that

\[
[F]^o = [\beta]^T [P]^o
\]

This equation is an expression of equilibrium at the joints of the structure.

6. Compute the displacements \(u\) at the system coordinates such that

\[
\{u\} = [k]^{-1} \left( \{F\}^f - \{F\}^o \right)
\]
The forces designated as \( \{ F \}^f \) are defined as forces acting at the coordinates. The forces \( \{ F \}^0 \) are forces required to set the displacements at the coordinates to zero due to forces acting away from the coordinates (fixed end moments).

7. Using the results in Step 6,
\[
[\delta] = [\Theta] \{ u \} = [\Theta] [K]^{-1} (\{ F \}^f - \{ F \}^0)
\]
and therefore the superposition of internal forces \( \{ P \} \) is given by
\[
\{ P \} = [K] [\delta] = [K] [\Theta] [K]^{-1} (\{ F \}^f - \{ F \}^0)
\]

8. The superposition of forces and displacements give the corresponding forces and displacements at any point in the structure. Therefore, using the results from Steps 5 and 7, the final value of the forces \( \{ P \}^f \) at the element coordinates is
\[
\{ P \}^f = \{ P \}^0 + [K] [\Theta] [K]^{-1} (\{ F \}^f - \{ F \}^0)
\]
If \( \{ F \}^0 \) is substituted in the above equation, in terms of \( \{ P \}^0 \),
\[
\{ F \}^0 = [\Theta]^T \{ P \}^0
\]
and the final form of the equation is
\[
\{ P \}^f = \{ P \}^0 + [K] [\Theta] [K]^{-1} (\{ F \}^f - [\Theta]^T \{ P \}^0)
\]
\[
= [K] [\Theta] [K]^{-1} \{ F \}^f + ([I] - [K] [\Theta] [K]^{-1} [\Theta]^T ) \{ P \}^0
\]
in which \([ I ]\) is the identity matrix.
With forces $\{P\}$ known, each element can be analyzed as a statically determinate structure to compute displacements and internal forces in the structure.

Many preprogrammed computer codes such as NASTRAN, MAGIC III, FORMAT III and Martin-Marietta's Space Frame Program eliminate the need for the analyst to personally follow the above steps. They require only that the structural elements be defined along with the loading environment and the computer performs the linear algebra to automatically generate the internal forces, stresses and deflections.

The Martin-Marietta Space Frame Analysis codes uses linear algebra for structural analysis to calculate deflections, influence coefficients, internal forces and stresses in any elastic structure due to externally applied loads, thermal loads and displacements.

The structure's stiffness is represented by a system of elastic finite elements which are interconnected at node points. The individual element stiffnesses are assembled into a stiffness matrix which is inverted by the choleski method and multiplied by an external load matrix to produce nodal displacements. The product of the displacement and the stiffness matrices produces element forces which are used to determine stresses. The computer output consists of element internal forces and moments, nodal displacements, and element stresses. Some of the optional output consists of computer plots of the mathematical model, the assembled stiffness matrix, and a flexibility matrix. The code is capable of handling up to
1000 node points of 6-DOF each, and utilizes the IBM 360/370 computer with double precision accuracy.

The Space Frame Program is used to determine thermoelastic and static stresses, deflection and influence coefficients for any complex elastic structure. Through an iteration process, the code can also be used in plastic limit and large deflection analyses. Figure 5 presents the code's main features and uses while Figure 6 represents the code's input requirements and final output.
<table>
<thead>
<tr>
<th>FEATURES</th>
<th>USES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Uses matrix algebra and the stiffness method to determine internal loads, stresses and displacements and influence coefficients in complex structures.</td>
<td>Thermoelastic, static stress and limit analysis of any complex structure that can be defined by interconnected finite elements and less than 1000 node points.</td>
</tr>
</tbody>
</table>

Figure 5 Martin-Marietta Space Frame Code Features and Uses
INPUTS REQUIRED

NODAL COORDINATES
  CYLINDRICAL
  CARTESIAN
  ROTATIONAL TRANSFORMATION OPTION

NODAL APPLIED LOADS
  NODAL FORCES AND MOMENTS
  NODAL TEMPERATURES
  COMBINED LOADING OPTION

NODAL SPECIFIED DISPLACEMENTS
  TRANSLATION AND ROTATION
  FIXITY

ELEMENT TYPE
  BAR (TAPER, PIN AND OFFSET OPTIONS)
  TRIANGULAR PLATE (IN-PLANE AND OUT-OF-PLANE FORCES)
  RECTANGULAR PLATE (IN-PLANE AND OUT-OF-PLANE FORCES)
  QUADRILATERAL SHEAR PANEL (CONSTANT SHEAR FLOW)

ELEMENT STIFFNESS DATA
  MATERIAL PROPERTIES
  BEAM MOMENT OF INERTIA, POLAR MOMENT OF INERTIA,
  CROSS SECTION AREA AND DISTANCE TO EXTREME FIBERS
  PLATE OR SHEAR PANEL THICKNESS

Figure 6  Martin-Marietta Space Frame Computer Code
PRELIMINARY CALCULATIONS

DIRECTION COSINES
ELEMENT THERMAL FORCES
STIFFNESS MATRIX
SOLUTION MATRIX

OUTPUT

PLOT OF THE ANALYTICAL MODEL
NODAL TRANSLATIONS AND ROTATIONS
STIFFNESS MATRIX
FLEXIBILITY MATRIX
ELEMENT INTERNAL FORCES AND MOMENTS
ELEMENT STRESSES

Figure 6  (continued)
Application of Limit Analysis
Using Elastic Finite Element Programs

The initial steps to modeling the limit finite element analysis using an elastic analysis computer code is identical to that of an elastic model. The shells are modeled as a gridwork structures using framework bar elements to carry axial and out-of-plane shear loads and their associated bending moments. The gridwork is interconnected with inplane shear elements to represent inplane shear stiffness. The framework bar elements have twelve degrees of freedom while the inplane shear elements have eight degrees of freedom. These elements are shown in Figure 7. A representative finite element mathematical model is shown in Figure 8. After completing the elastic analysis, areas in the structure which have exceeded the elastic bending limit are located. A plastic hinge is placed at the point of maximum stress and the problem is computed again and reviewed for additional bending stresses which exceed the elastic limit and so on until either a mechanism is formed or no additional plastic hinges are required. For the thesis subject shell structures studies thus far, no more than three iterations have been required.

The first structure modeled was that of a two dimensional frame where only bar elements were required. Later problems involving three dimensional shell structures required bar and inplane shear elements interconnected to form a gridwork to represent the shell structure stiffness.
Figure 7 Finite Structural Elements
Figure 8  Gridwork Finite Element Math Model
The steps required in the limit analysis of shell structures are as follows:

1. Create a finite element mathematical model which represents the elastic stiffness of the structure being analyzed. Subject the structure to elastic analysis to determine stress distribution.

2. Find the point of maximum bending stress which exceeds the plastic limit.

3. Introduce an additional node point near the point of maximum bending stress as shown in Figure 9b. The new element which is introduced between the point of maximum stress and the new node point should be above five per cent of the length of the adjacent element. The new element should have pinned fixity at the point of maximum bending stress.

4. Apply bending moments equivalent to the maximum bending moment to the new element at the point of plastic bending and to the adjacent element as shown in Figure 9b and perform the analysis again.

5. Review the results of the second run for additional plastic stresses and repeat steps 2 through 5 until either structural collapse occurs or no additional plastic stress areas occur. In this study of the
inlet duct structure, structural collapse is determined by the presence of large deflections.

Figure 9 shows the steps for limit analysis of a beam which is fixed at each end and loaded by a point load at node point 3 (not at the center of the beam). Figure 9a shows the elastic model with six nodes. The additional node from the first iteration is shown in Figure 9b. Figure 9c shows the plastic resisting moment at node point 1 which resulted from the second iteration. The bar, of course, is pinned at node 1 after the second iteration. The structure does not collapse until the plastic moment is reached as shown in Figure 9d.

The application of the Lower Bound Theorem to Limit Analysis is no more difficult than elastic analysis of structures using finite element codes except that more than one solution is necessary. The method provides a workable solution to the problem for the structures engineer who has access to a finite element analysis program.
Figure 9

a. Elastic Model

b. Plastic Model

c. Plastic Model with Plastic Hinge at Node 1

d. Moment Distribution
CHAPTER II

SUBSTANTIATION OF ANALYTICAL TECHNIQUE

The limit analysis approach using finite elements was verified through two series of analyses and structural tests; 1) Phase A which analyzed and tested six two dimensional pressure vessels like that shown in Figure 10 and 2) Phase B which analyzed and tested a three dimensional vessel as shown in Figure 16. The pressure vessels represent a missile inlet and air transfer duct for a ramjet powered, air launched missile. Phase A consisted of the evaluation of one dimensional stress distributions in the constant cross section duct free of end constraints. The data obtained in this series of tests were used in developing the analytical model for design of a representative full scale duct structure for Phase B testing. Phase A test results show that geometric non-linearities from large elastic deformations do not play a significant role in the deflection or stress response. Test results showed good correlation with predictions based on limit analysis procedures using finite element stress analysis techniques.

Phase B portion of the program consisted of an inlet duct configuration representative of the actual flight weight structure and included the effects of the cowl lip and finite length. The Phase B inlet test structure was fabricated with 17-4 PH-H1025
Figure 10. The Phase A plexiglas inlet assembly permitted large deflections without material yielding.
stainless steel. The structure was instrumented with strain gages and deflection transducers at four stations, and internal pressure was applied until failure occurred. The inlet failed at 120 psig (0.83 MPa) by fracture in a weld. There is good correlation between calculated and measured stresses and deflections up to the time of failure.

The results show that the structure performed well into the plastic regime, and that failure prior to the complete formulation of plastic hinges was a result of lack of ductility in the weld. The data derived during this task has provided confidence in the procedures established for predictive analysis to establish the structural integrity of the air induction system with minimum weight and volume, and design criteria has been gained that will ensure that the structure has the necessary ductility to enhance the feasibility of the limit analysis approach.
Finite Element Analysis of Two Dimensional Structures

The first phase of the program consisted of six individual analyses and tests; three plexiglas models to permit large deflections without material yielding, and three aluminum structures to provide stress, strain, and deflection data for plastically deformed structures. The finite element analysis technique was used to predict the behavior of each specimen with an iterative procedure used to account for the plastic deformations.

The finite element model of a typical cross section shown in Figure 11 consists of seven bar elements for the inlet structure and seven bar elements for the aluminum cover plate. Due to symmetry, only one half of the structure was modeled. The uniform pressure load was applied as force components in the X and Y direction at each node point. The element stiffness properties were based on a 1-in (25.4 mm) wide shell element, and the properties for the six specimens are shown in Table 1. These models were used to calculate the linear elastic response of the structures to uniform internal pressure. Several variations were considered during the course of the study to better approximate the relative stiffness at node point 8, which represents the joint between the specimen and the cover. Since that joint was a preloaded bolted connection with a layer of cast-in-place RTV rubber to provide a pressure seal, an accurate approximation was not achieved for all tests. However, an absolute correlation with the experimental data was not required to assess the nonlinear response of the structures, as shown later, the most significant tests provided good agreement.
Figure 11 A uniform pressure load was applied as force components in the X and Y direction at each node point shown on the finite element math model for the inlet/test fixture.

<table>
<thead>
<tr>
<th>Thickness h (in)</th>
<th>$I_{\text{Plastic}}$ (in$^4$/in)</th>
<th>$I_{\text{Aluminum}}$ (in$^4$/in)</th>
<th>Area (in$^2$/in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.063</td>
<td>-</td>
<td>0.000023</td>
<td>0.063</td>
</tr>
<tr>
<td>0.125</td>
<td>0.000189</td>
<td>0.000181</td>
<td>0.125</td>
</tr>
<tr>
<td>0.188</td>
<td>0.000642</td>
<td>0.000617</td>
<td>0.188</td>
</tr>
<tr>
<td>0.250</td>
<td>0.001509</td>
<td>-</td>
<td>0.250</td>
</tr>
</tbody>
</table>

\[
I = \frac{h^3}{12} \left( \frac{1}{1-\mu^2} \right)
\]

where $\mu$ = Poisson's Ratio

$h$ = Material Thickness

$\mu_{\text{Plastic}} = 0.37$

$\mu_{\text{Aluminum}} = 0.32$
Limit analysis was used to predict internal loads and deflections for the yielded aluminum structures. The finite element modified model for this analysis is shown in Figure 12. The model is identical to the preliminary analysis model except that provisions were made for inelastic hinges at nodes 7 and 10. Bar elements 8 and 9 were necessary to permit control of the moment at node points 7 and 10. These were very stiff bars that were pinned at node 7 and 10 but transferred bending through node 9. Bar 7 was pinned at node points 8 and 10.

The procedure for limit analysis of phase structures was:

1. Apply the node point pressure loads to the structure and determine the node where yielding first takes place.

2. Calculate the resisting moment of the duct wall based on the bending modulus of rupture; apply the resisting moment at hinged joint where yielding was indicated and analyze again.

3. Repeat step 2 with increasing loads until the structure becomes unstable. The run previous to the unstable run will determine the maximum permissible applied pressure to the structure if the plastic joint which causes instability is loaded to its maximum allowable stress.
Figure 12 A limit analysis math model provides means for inserting a plastic hinge to predict internal loads and deflections for the yielded aluminum structures.
Test of Two Dimensional Structures

The plexiglas tank and duct walls were formed under heat and pressure and then bonded together. The aluminum tanks were rolled, welded, and aged. Slots were cut in the duct portion of the structure to form a 12-in (304.8 mm) wide test section that deflected unrestrained by the end plates. Strain-gages were attached and sealed to protect them from moisture. The entire test section and adjoining fixture were lined with Dow Corning DC93-072 silicone. After a 24-hour curing period for the silicone, the aluminum pressure lid was bolted into place. Just prior to the installation of the lid, an uncured layer of DC93-072 silicone rubber was brushed onto the mating surfaces between the lid and the test structure to ensure a leak-tight seal. Corner gussets were added to the third test structure to relieve the loads around the stress risers at the end of the slits used to separate the plexiglas duct from the test fixture. This modification was deemed necessary after failure of the second plexiglas specimen.

Figure 13 shows the instrumentation locations for each of the six tests. Due to symmetry, the strain gages were located on one side of the inlet center line and the deflection gages were located on the opposite side. Duct pressure was measured at the fluid drain hole only since fluid flow rate through the duct was low. The strain gages were placed approximately 20 degrees apart on the curved portion of the duct and 1/2-in (12.7 mm) from the corners as shown. One biaxial arrangement of the gages was used
Symbols:
- Uniaxial Strain Gages
- Biaxial Strain Gages
- LVDT

Instrumentation Used in Each Test

<table>
<thead>
<tr>
<th>Test No.</th>
<th>S1</th>
<th>S2</th>
<th>S3</th>
<th>S4</th>
<th>S5</th>
<th>S6</th>
<th>D1</th>
<th>D2</th>
<th>D3</th>
<th>D4</th>
<th>D5</th>
<th>D6</th>
<th>D7</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>2</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
<td></td>
<td></td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>3</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
<td>X</td>
<td>X</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
<td>X</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
<td>X</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
<td>X</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Note: All strain gages and LVDT potentiometers are located on the center of the test duct.

Figure 13 Instrumentation and locations used in Phase A testing provided adequate test data.
on the center line of the duct, but all other gage arrangements were uniaxial. For the plexiglas structures, where large strains were expected, Micro-measurements, Incorporated No. EA-05-250BF-350 strain gages were used; and No. EA-13-250BF-350 strain gages were used on the aluminum structures. The strain gages provided 5 per cent strain capability necessary for the large deflections in the test. The gages were placed on both the inner and outer surfaces at all locations to allow calculation of the bending moment and axial forces. Deflections were measured with LVDT (Linear Variable Differential Transformer) deflection gages attached by bonded tabs. These bonded tabs caused some problems, especially with the larger deflections due to movements transverse to the LVDT and failure of the bond. Phase B used spot-welded tabs attached to the LVDT's by wires.

After the instrumentation had been installed, wired and checked-out, the tank was covered and sealed. It was then filled with water until overflow occurred at the bleed line, indicating a minimum of trapped air. After the fill and bleed line valves had been closed, the strain and deflection instrumentation was read as test "zero". Reading and recording was automatic and was accomplished with an SEL 600 Data Acquisition System, a digital recorder having an accuracy of 0.1 per cent.
Strain and deflection readings were recorded at various pressure intervals for each test depending on expected failure levels. Pressure was held constant at the end of each loading increment so that strains and deflections could be recorded. The recording period was 10 to 15 seconds in all cases. Table 2 is a summary of the Phase A test program.

### TABLE 2  INLET STRUCTURAL TEST SUMMARY

<table>
<thead>
<tr>
<th>Run No.</th>
<th>Material</th>
<th>Skin Thickness (in)</th>
<th>Maximum Expected Pressure (psig)</th>
<th>Pressure Rate (psig/min)</th>
<th>Failure Pressure</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Plexiglas sheet</td>
<td>0.125</td>
<td>5</td>
<td>0.5</td>
<td>Failed at 5.5 psig</td>
</tr>
<tr>
<td>2</td>
<td>Plexiglas sheet</td>
<td>0.188</td>
<td>7</td>
<td>0.5</td>
<td>Failed at 2.5 psig*</td>
</tr>
<tr>
<td>3</td>
<td>Plexiglas sheet</td>
<td>0.250</td>
<td>9</td>
<td>1.0</td>
<td>Seal leak at 12 psig</td>
</tr>
<tr>
<td>4</td>
<td>6061-T6 aluminum alloy sheet</td>
<td>0.063</td>
<td>10</td>
<td>2.0</td>
<td>Seal leak at 10 psig</td>
</tr>
<tr>
<td>5</td>
<td>6061-T6 aluminum alloy sheet</td>
<td>0.125</td>
<td>17</td>
<td>6.0</td>
<td>Seal leak at 22 psig</td>
</tr>
<tr>
<td>6</td>
<td>6061-T6 aluminum alloy sheet</td>
<td>0.188</td>
<td>38</td>
<td>10.0</td>
<td>Seal leak at 36 psig</td>
</tr>
</tbody>
</table>

*Premature failure at stress concentration.*
Correlation of Tests and Analytical Results

for Two Dimensional Model

The center line deflections for the six tests are shown as a function of pressure in Figure 14. The normalized load and deflection parameters employed by Timoshenko (7) are used to evaluate the degree and source of nonlinear response. Geometric nonlinearity would be indicated by a reduction in the rate of deflection with increasing load, while material yielding results in an increased rate of deflection. The data indicate that the only source of nonlinearity for the configurations tested comes from local yielding in the aluminum specimens.

All tests, except the first two plexiglas specimens, were terminated by loss of the pressure seal due to excessive deflections. The differences in the initial slopes of the deflection curves is caused in part by the variation in the relative stiffness at the specimen's attachment joints. Failure of the LVDT attachment and the large deflections of the 1/16-in (1.6 mm) thick aluminum specimen (No. 4) resulted in very little experimental data for that test. The remaining two aluminum structures, however, performed well and provide good strain and deflection data.

A comparison of analytical with experimental response of the aluminum structures is shown in Figure 15. These results show that the limit analysis method employed provides an excellent lower bound of 17 psi (117 KPa) and 38 psi (262 KPa) for the collapse pressure. Again, loss of the LVDT's at the peak measured pressure did not allow
Figure 14 Phase A center line deflections as a function of pressure indicate that the only source of nonlinearity comes from local yielding in the aluminum specimens.
Figure 15  A comparison of analytical with experimental response of the Phase A aluminum structures indicates that improved approximations of joint stiffness would increase the analytical correlation.
an accurate measurement of the maximum deflections. The data indicates that improved approximations of the joint stiffness would increase the analytical correlation. The success of the Phase A study provides confidence that a full scale model of the inlet could be designed using the finite element limit analysis approach.
Three Dimensional Structure

The test structure shown in Figure 16 was made from 17-4 PH stainless steel and heat treated to the H1025 condition after welding. The configuration of the test specimen was designed to simulate a ramjet inlet while providing detail parts that would be rolled or machined. The cowl lip was simulated with a conical ring machined to provide a tapered thickness. The duct and lower tank wall were 0.125-in (3.2 mm) thick, and the upper tank wall and duct splitter plates, as seen more clearly in Figure 18, were 0.062-in (1.6 mm) thick. Ring frames were provided at the forward and aft ends of the cylindrical tank for stiffness. Gussets were welded to the ring frames along the lower edge to help support the ends from the duct pressure loads. Three-quarter-inch (19 mm) thick aluminum end plates were bolted to the tank frames to close off the duct and restrain the pressure sealing system.
Figure 16 The Phase B test structure was designed to simulate the baseline inlet elastic-plastic response.
Finite Element Analysis of Three Dimensional Structures

The analysis of the three dimensional duct structure was accomplished using the Martin Marietta Space Frame Finite Element computer code. The model consisted of 200 nodes composed of bar elements and shear panels which represent the stiffness of the structure. Simulation of the structure in the plastic range was an iterative process that required the addition of load limiting hinges to the elastic model. These hinges allow adjacent elements to carry loads while maintaining the bending moment at the hinge at the plastic limit level. Tensile specimens made from the same material lot used to fabricate the test inlet were heat treated and tested to determine the yield stress, strain, and ultimate elongation for the actual structural material. Both parent metal and welded specimens were tested. The average values from these tests are presented in Figure 17. The 0.2 per cent offset yield strength was used to establish the plastic bending moment for the limit analysis calculations. The estimated plastic moments are: parent metal, \( M_p = 703 \text{ in-lb/in} (3120 \text{ N}) \), and weld, \( M_p = 656 \text{ in-lb/in} (2920 \text{ N}) \). These values were used in the limit analysis of the structure to estimate the collapse pressure and the deflections.

The first step in the analytical solution was to perform an elastic analysis without hinges. From this solution, the location and pressure at which the yield condition is initially reached is then determined. Next, a hinge is inserted in the structural
Figure 17  Raw data averages of parent metal and welded specimens were used (with 0.2 per cent offset yield strength) to establish the elastic-plastic model material for limit analysis calculations.
model at these yield locations and the value of the plastic moment is specified as an input to the program. Another elastic solution is obtained at higher pressures with hinges specified. Using this solution, the locations of additional hinges are determined.

Figure 18 shows the progression of the plastic hinges and the pressures at which they are formed. The iterative process is continued until the maximum rate of deflection becomes unbounded. The pressure at which this occurs is defined as the collapse pressure of the structure. The pressure-deflection predictions of the test structure is shown in Figure 19 for four stations along the inlet as designated in Figure 20. The curves indicate that the peak deflection becomes unbounded at approximately 175 psig (1.2 MPa). The comparison with two measured deflections shows good correlation near the cowl lip and indicates that the analysis over-estimates the deflection at the center of the structure.
An iterative process determined the locations and progression of plastic hinges on the test specimen.
Pressure/deflection predictions of four stations along the inlet test structure indicate that the peak deflection becomes unbounded at approximately 175 psig (1.2 MPa).
Test of Three Dimensional Structures

The three dimensional inlet duct structure test, Phase B, was accomplished in two stages: 1) two initial tests within the elastic range of the material provided a complete description of the structure's linear response, and allowed for optimum selection of the strain gages that were monitored during the second stage; 2) the third test pressurized the structure to failure. Because of symmetry, strain gages were mounted on one side of the structure and deflection measuring devices were attached to the other side. Two element strain rosettes (BLH type FAET-12D-1259-E5) provided a ±3 per cent strain capability, and deflections were obtained with an LVDT connected by wires to minimize the effects of tangential movement on the normal surface deflection measurements. The pressure was measured by a transducer mounted in the duct and visual observation was made with a dial gage on the inlet line. Data were recorded at selected pressure levels using a 50-channel SEL 600 Data Acquisition System that has an accuracy of 0.1 per cent.

Placement of instrumentation on the specimen is shown in Figure 20. The solid indicators show those gages selected for monitoring the final test. The relationship between hoop and meridional strains at each location was determined during the first two tests. Since the magnitudes of the meridional strains were
Figure 20  Instrumentation locations were chosen to specifically monitor those areas pertinent to the two-stage Phase B program.
relatively small, only two meridional gages were monitored during the final test. It was assumed that the strain ratios measured during the first two tests were the same as for the final test. This allowed the stresses to be calculated with one strain component.

Pressurization of the duct was accomplished during the initial two tests with a cast-in-place silicone rubber seal. During these tests, excessive leaking was experienced. Although it was possible to accomplish the objective of these tests by providing adequate flow to maintain the desired pressure, it was not satisfactory for the later test. After completion of the first stage of data acquisition, nitrile rubber bladders were installed in the duct to pressurize the structure to the required levels for failure testing without leakage. The effects of this cyclic loading and disassembly of the structure on the strain measurements was evaluated by monitoring several strain gages during the above operations. This showed that the structure remained linear and reproducible.

The basic loading procedure followed was to: 1) vacuum fill the duct with water to minimize trapped air, 2) record output at zero pressure, 3) pump water at predetermined rate to specified pressure level, 4) record data at constant pressure levels, 5) decrease pressure and record at specified levels, and 6) measure output at zero pressure.
Correlation of Tests and Analytical Results
for Three Dimensional Model

During the Test 3 loading, the structure failed by fracturing of the corner weld. The fracture initiated at the aft end and progressed forward. Observation of the pressure gage and subsequent correlation with the pressure transducer recordings indicate that the pressure at failure was approximately 120 psig (0.82 MPa). Some measured strains are presented in Figures 21 and 22 where it is shown that the last recording occurred at 110 psig (0.78 MPa).

Inspection of the strain data confirms that the structure began to yield locally at the forward and aft duct attachment for pressures between 60 and 70 psig (0.41 and 0.48 MPa). The nonlinear response of the strains at sections A-A and D-D reflect the influence of the plastic hinge forming. These measured strains are generally below the yield strain of the material because they are located some distance from the plastic hinge. Also, the strains at the forward end are higher than those at the aft end because the gages were applied 0.5-in (13 mm) and 1.0-in (25 mm) from the edge, respectively. The measured deflections are generally well behaved and linear for the pressures recorded.
Figure 21 Strain/pressure distributions during test 3 show plastic response prior to the failure pressure of 120 psi (0.82 MPa).
Figure 22  Strain/pressure distributions during test 3 show plastic response prior to the failure pressure of 120 psi (0.82 MPa).
The prime objective of this test was to establish the validity of limit analysis procedure for the inlet duct structure design; therefore, correlation of experimental data with analysis predictions at as high a pressure as possible was desired. Thus, the measured strain data was extrapolated to 120 psid (0.82 MPa) in order to estimate the stress distributions throughout the specimen at the instant of failure. The strain data was used to estimate the stresses and the bending moments to compare with the limit analysis predictions. Since the strains must be measured on both the inside and outside surfaces to determine the experimental bending moment, measured moment data is possible only at sections A-A and D-D because these were the only locations where back-to-back gages could be placed.

The moment correlation is restricted to the uniform thickness aft section D-D. This is because the tapered thickness at the cowl lip A-A makes it very difficult to estimate the measured bending moments or to extrapolate the predicted values between node points. All other correlations are based on surface stresses as determined by the measured strains and the modulus of the material. The bending moment distribution in the duct at section D-D is shown in Figure 23. The predicted plastic solution at 120 psid (0.82 MPa) pressure is based on hinge moments at locations B and C near the forward and aft ends. The dashed curve shows the moment distribution at 63 psid
Figure 23  The section D-D bending moment distribution prediction at 120 psig (0.82 MPa) was based on hinge locations B and C near the forward and aft ends.
(0.43 MPa) pressure as a comparison when the structure is constrained to work within the elastic limit of the material. The overall correlation with the measured data was good except at the center of the duct.

Figures 24 through 29 present the predicted stress distributions and test results in the duct structure at an internal pressure of 120 psid. The predicted stresses were obtained by finite element solutions with plastic hinges at the forward and aft sections of the inlet duct, and are based on the modulus of rupture of the material. For the parent metal this value is 270 ksi, and for the weld it is 252 ksi.

The experimental stresses are calculated using the properties of the material and the measured strains. The stress is given as:

\[ \sigma_{H} = \frac{E \epsilon_{H}}{1 - v^2} \]

Above strains of 0.0063 in/in, the secant modulus, obtained from the stress-strain curve given in Figure 17 is used to calculate the stress, and Poisson's ratio, \( v \), is assumed to equal 0.5. Figure 30 presents the predicted and test deflections of the duct at section C-C.

This data, together with that presented in the evaluation of two dimensional structures, indicates that the finite element analysis solution provides an accurate prediction of structural response well into the plastic regime.
Figure 24  Duct Hoop Stress, Section A-A, 120 psid
Figure 25  Duct Hoop Stress, Section B-B, 120 psid
Figure 26  Duct Hoop Stress, Section C-C, 120 psid
Figure 27  Duct Hoop Stress, Section D-D, 120 psid
Figure 28  Tank Hoop Stress, Section C-C, 120 psid
Figure 29  Tank Hoop Stress, Section D-D, 120 psid
Figure 30  Duct and Tank Deflections at Section C-C
CHAPTER III

CONCLUSIONS

The application of the Lower Bound Theorem limit analysis to shell structures using elastic finite element computer codes has been successfully demonstrated through this analysis and test program. Structural failures of the pressurization seals during phase A testing resulted from large deflections caused by structural collapse at or near the predicted failure pressures. Also, phase A deflection data indicates that yielding and structural collapse occurs as predicted, particularly in test numbers 5 and 6. Phase A testing provided the confidence needed to continue with analysis and testing of the more complex phase B inlet structure.

Phase B test data correlated well with the limit analysis up to the point of premature weld failure. Plasticity began at 40 per cent of the expected collapse pressure as analytically predicted but failure occurred at 70 per cent of expected collapse pressure. The weld failure resulted in a redesign of the ramjet duct to eliminate welds in the peak stress areas and addition of weld fillets to force yielding to occur adjacent to the welds. Limit analysis applied to a flight vehicle ramjet duct resulted in a duct structural weight savings of 25 per cent without the need for expensive milling of excess material.
The method of limit analysis using elastic finite element computer codes is relatively easy to apply and gives excellent results. In applying the method to actual structure, care must be taken to assure ductility where plasticity is expected to occur.
REFERENCES


