THREE DIMENSIONAL STRESS ANALYSIS OF AN ARTILLERY PROJECTILE JOINT

Tien-Yu Tsui
Army Materials and Mechanics Research Center
Watertown, Ma. 02172

M.L. Chiesa and M.L. Callabresi
Sandia National Laboratories
Livermore, CA. 94550

ABSTRACT

This study presents the results of a three-dimensional elastic-plastic dynamic stress analysis of one of the structural joints encountered in artillery projectiles. The particular spline joint analyzed has equally spaced set screws around the surface of the projectile. In general, these types of structural joint problems are variable contact problems in that the interaction between the set screws and their bearing surface along with the interaction between the interfaces of the joint are nonlinear in nature. Due to the complexity of the structural configuration and loadings of the joint, the finite element method has been used to solve the problem. The numerical analysis covers the time from initial launching to barrel exit. Stresses and deformations in the joint are determined at various stages of loading. The effect of the set screws and set screw holes on the stress distributions in the joint is examined in detail.

INTRODUCTION:

Artillery projectiles are subjected to extremely high loads during firing. At present, the design of artillery projectiles is greatly facilitated by the exploitation of the finite element method. But the application of the method has been limited to the simplified two dimensional or axisymmetric analysis. This is due to the complexity of the geometries of the projectiles thus requiring very long computer analysis. Although successes were achieved in the previous designs, there have always been concerns of the structural integrity of the projectiles with the presence of high local stresses. In the case of XM753 projectiles, high local stresses appear in the region of the pinned joint. In the case of XM785 projectiles the use of spline joint with set screws in the design also leads to high local stresses in the joint area.

After firing, an artillery projectile is subjected to various loads continuously changing with time. It experiences first a very high compressive load in the axial direction during in bore flight and then a high tensile load at barrel exit. Such a loading history causes the projectile to undergo a stress reversal i.e. from a stress state of compression to that of tension. However it has been a common practice in the design of artillery projectiles to perform either one or two simple independent stress analysis. The one which is generally carried out is a two dimensional quasi-static analysis in which the loads at the peak linear acceleration during in bore flight are used to compute the stresses and deformations in the projectile.

Sometimes a two dimensional dynamic analysis is also conducted to determine the stresses and deformations in the projectile caused by the sudden drop off of pressure of the propellant gas at barrel exit.

In the present investigation a detailed three dimensional elastic-plastic dynamic stress analysis of a spline joint of XM785 projectile is performed. The purposes of the study are.

(1) to verify the structural adequacy of a proposed design.

(2) to asses the effect of set screws and set screw holes on the stress distributions in the joint region of the projectile.

In addition the present work also serves as an initial effort to determine the extent of the influence of the stress reversal or Bauschinger effect on the structural integrity of the projectile.

DESCRIPTION OF LOADING CONDITIONS

After a projectile is fired and before it departs the gun barrel it is subjected to a combination of the following loads:

(1) axial compressive load due to linear acceleration of the projectile

(2) centrifugal load due to angular rotation of the projectile.

(3) torsional load due to angular acceleration of the projectile.

(4) internal load due to interaction of interior components and projectile.

(5) external load due to gun tube constraint, rotating band pressure and balloting.

As the projectile departs the gun barrel, it experiences a tensile load or a negative set-back load (elastic release) in the axial direction resulting from the sudden drop off of propellant pressure at the barrel exit. Among the loads the axial load is the dominating one.

It has been a general practice to omit the effect of the torsional load induced by the angular acceleration in the design of the projectiles. Analysis has shown that as a result of such an omission, the magnitude of the effective stresses in the projectiles are about 2% - 4% lower (1). Since the present investigation concerns the determination of the stress distributions in the region of the joint, only the portion of the projectile in the neighborhood of the joint is considered. The area where the rotating band is located is excluded. Thus the load due to rotating band pressure is not included in the analysis. Only the axial, centrifugal and internal loads during in bore flight and the negative set back load at barrel exit are considered in the three dimensional dynamic stress analysis of the joint. Fig. 1 shows the linear acceleration of the projectile used in this analysis for the calculation of the axial load. The projectile reaches an acceleration of 17,000g in about 6 milliseconds, zero acceleration at barrel exit and immediately is subjected to a negative acceleration of 2000g.

METHOD OF STRESS ANALYSIS

Figs. 2 and 3 shows respectively the geometric configuration of an artillery projectile and a typical section of its joint. The latter is formed by passing a plane at the midpoint of two neighboring set screws and another plane through the center of one of the set screws. The two planes are parallel to the axis of the projectile.

In the initial phase of the analysis, the finite element model of the joint (see Fig. 4) was created by using the computer program PATRAN-G (2). It employs sophisticated interactive color graphics and a powerful geometry-based language for geometry construction and finite element modeling during the first phase of the program. Then nodal point and finite element generation, assignment of physical properties, application of external loads and definition of sliding interfaces are accomplished in the second phase of the program.

The explicit three dimensional finite element code DYNA3D (3) was used to compute the stresses and deformations in the finite element model of the joint. DYNA3D is designed to analyze the large deformation dynamic response of inelastic solids. It has a contact algorithm that can model gaps and sliding materials interfaces. It uses a 8-node constant stress solid element and one point integration in element stiffness calculations. It is programmed to take full advantage of vector optimization on the CRAY-1 (a class VI machine) and can execute at less than 0.67CPU (central processor Units) minutes per million mesh cycles. A symmetric, penalty based, contact-impact algorithm was implemented that not only reduced hourglassing problems, but also was considerably faster in execution speed and exceedingly reliable.

There were 1773 eight node brick elements and 2611 nodes in the finite element model of the joint. Four sets of gap or sliding interfaces were required in the model. Three sets of sliding interfaces were used to model the contact between the case structure and the wedge. A sliding interface with small initial gap was used to model the interaction between the set screw and the hole.

The computations were performed on a CRAY-1 computer at Sandia National Laboratories. The computer time required to complete the dynamic analysis was about 2.8 CPU hours.

DISCUSSION OF RESULTS

The post processor program GRAPE (4) was used to obtain plots of stress contours on the surfaces of the joint and the deformed shapes of the joint at the time when the maximum linear acceleration is reached and also at barrel exit. These are shown in Figs. 5 to 11. Examination of the results of the analysis leads to the following findings:

(1) Due to the presence of set screws, a larger portion of the axial compressive load is transmitted, during in-bore flight, through the center sections of the regions bounded by each pair of set screws. Consequently, the center

sections experience higher stresses. This is different from the uniform stress distribution in the circumferential direction found in the two dimensional stress analysis.

2. Higher stresses occur on the outer surfaces of the case structure and the wedge at both peak axial compression and tension at barrel exit due to bending effect.

3. Local yielding occurs in the set screws, set screw holes and other areas in the case structure and the wedge at time of peak axial compression and that of tension at barrel exit. Initial examination of the plastic strains (or Bauschinger effect) at peak axial compression leads one to believe that they have a negligible effect on the level of stresses in the joint at barrel exit. This is because their magnitudes are small (.001) and their locations are also different from the plastic strains induced by the tensile load or negative set-back at barrel exit.

4. Similar stress distributions were obtained when a wedge made of either steel or titanium was used with a titanium case structure.

In summary, a detailed three dimensional elastic-plastic dynamic stress analysis of an artillery projectile was obtained to determine the adequacy of a proposed design. The analysis was performed by using the DYNA3D finite element program. The results of the present analysis will be verified by laboratory test currently under preparation at AMMRC. Future analytical work will include application of fracture mechanics to the problem and accurate determination of the Bauschinger effect by using actual stress strain curves of the materials. The present analysis employed isotropic strain hardening which does not accurately represent the material behavior in a loading condition where there is a reversal of applied loads.

ACKNOWLEDGEMENT

The support and encouragement of Mr. C. Spinelli, Development and Management Officer for XM785 projectile, Nuclear Projectile Branch at ARRADCOM, Dover, N.J. and Dr. G.A. Benedetti, Chief, Applied Mechanics Division, Sandia National Laboratories, are gratefully acknowledged.

REFERENCES

1. T. Tsui, "On The Magnitude Of Stresses Due To Angular Acceleration Of Projectile", unpublished notes.

2. PATRAN-G, PDA Engineering, 1560 Brookhollow Drive, Santa Ana, California 92705

3. J.O. Hallquist, "DYNA3D-Nonlinear Dynamic Analysis Of Solids In Three Dimensions", Lawrence Livermore, Rept. UCID-19592, Nov. 1982.

4. B.E. Brown, "Displacing The Results Of Three Dimensional Analysis Using GRAPE", Lawrence Livermore National Laboratory, Rept. UCID-18507, Oct. 1979.

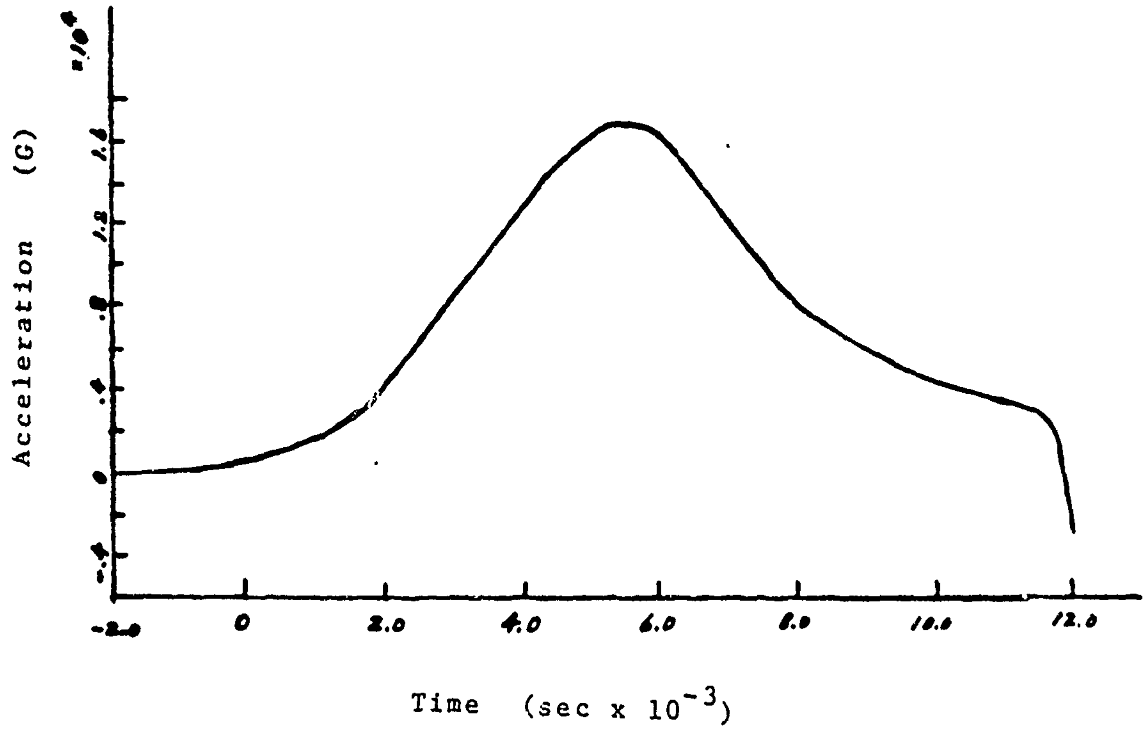


Fig. 1 Linear Acceleration of Projectile

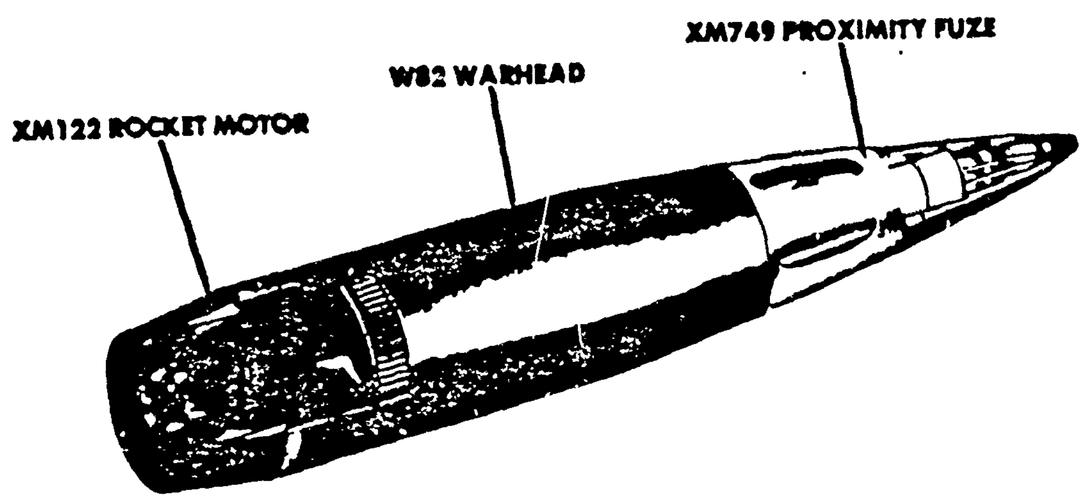


Fig. 2 Projectile, Atomic, 155MM: XM785

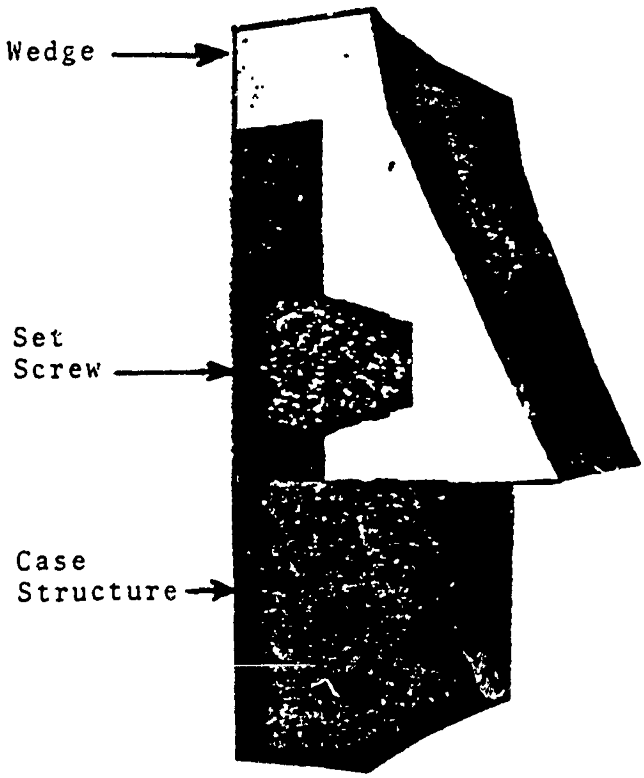


Fig. 3 Typical Joint Section

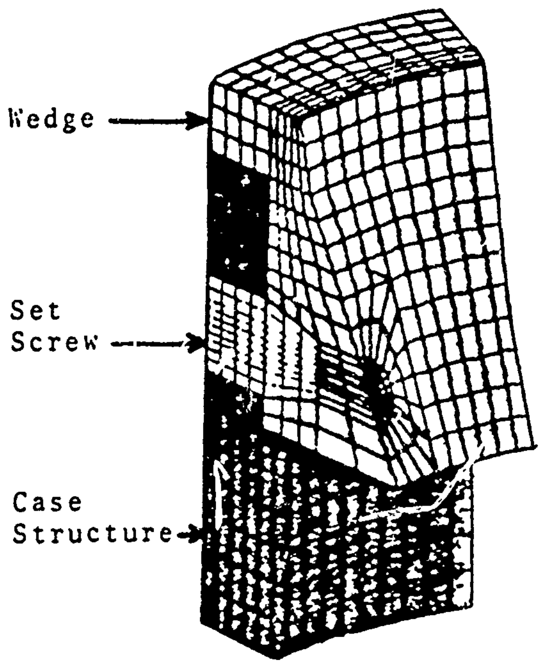
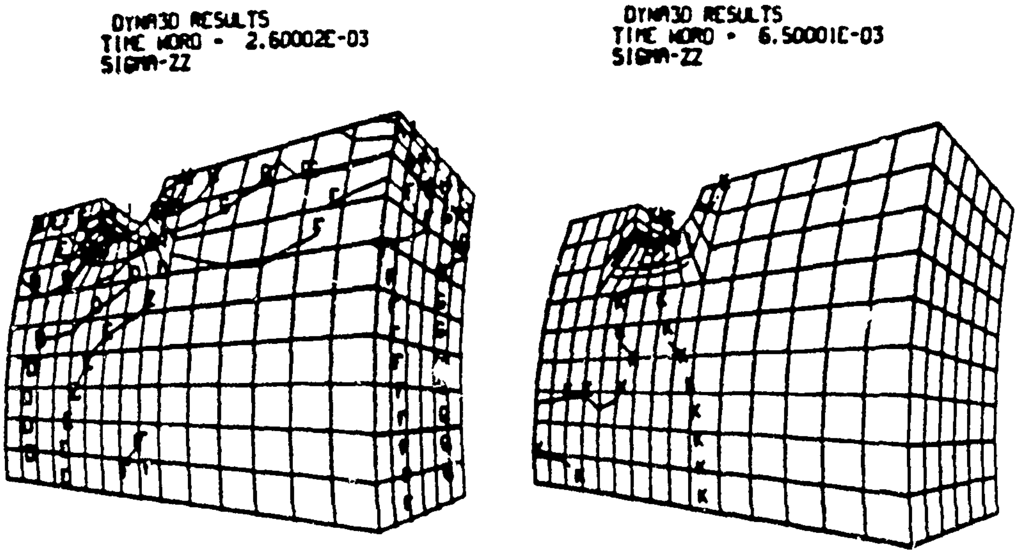


Fig. 4 Finite Element Model

See Table 1 for Values of Symbols

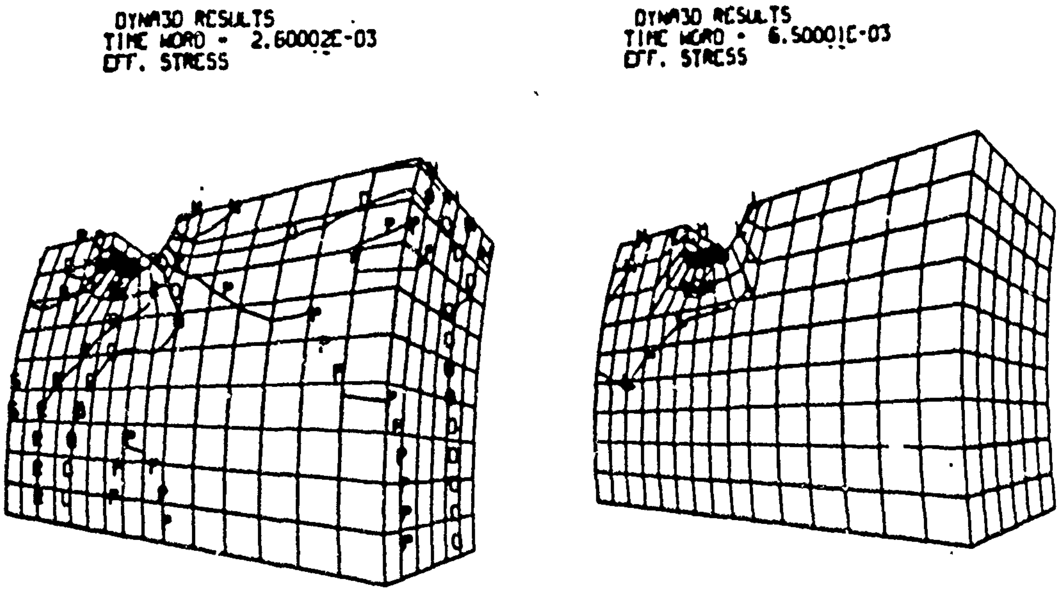


Case 1 Case 2

Fig. 5 Axial Stresses in the Case Structure

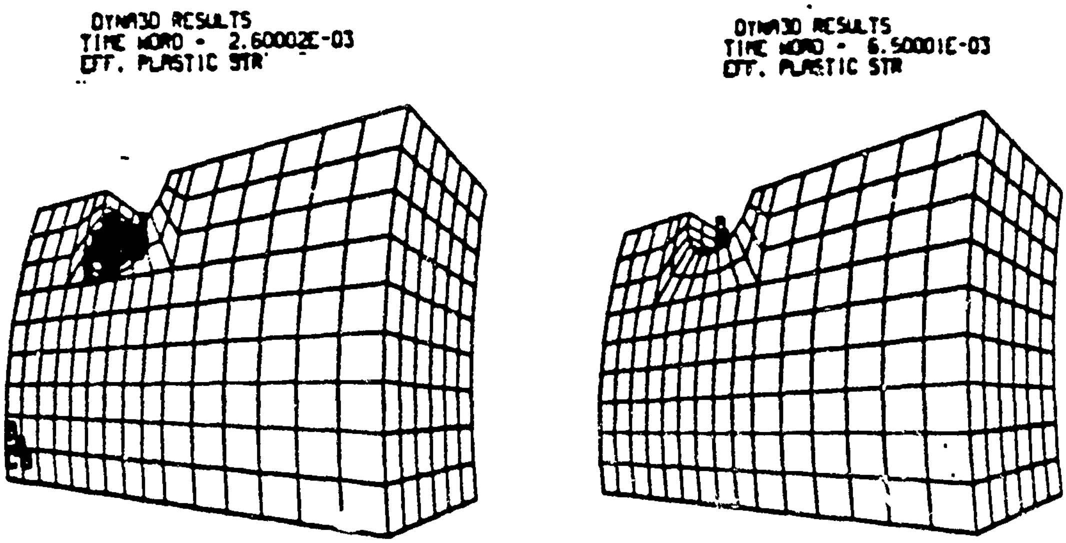
Case 1: At time of maximum compression

Case 2: At time of barrel exit



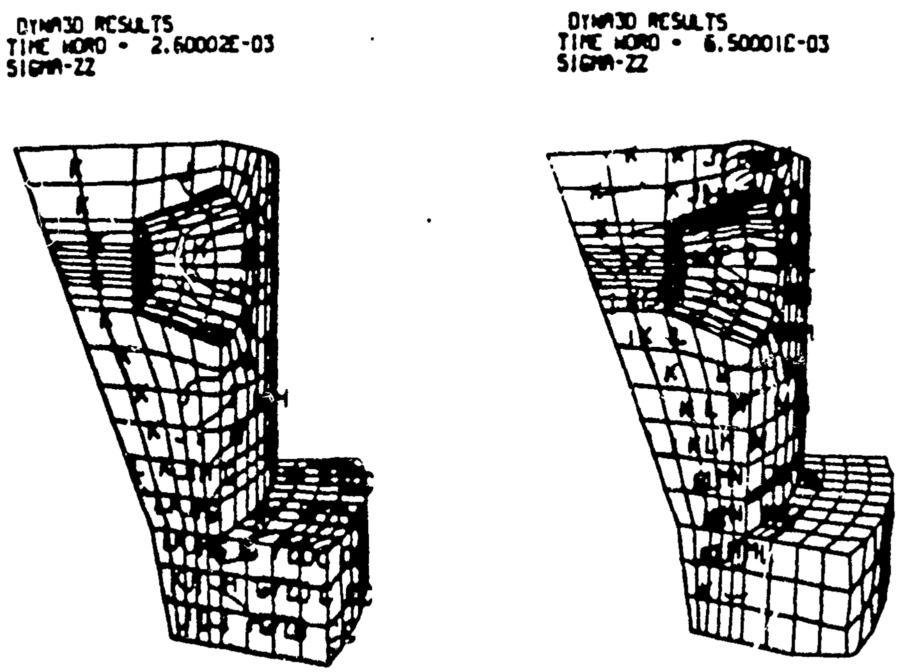
Case 1 Case 2

Fig. 6 Effective Stresses in the Case Structure



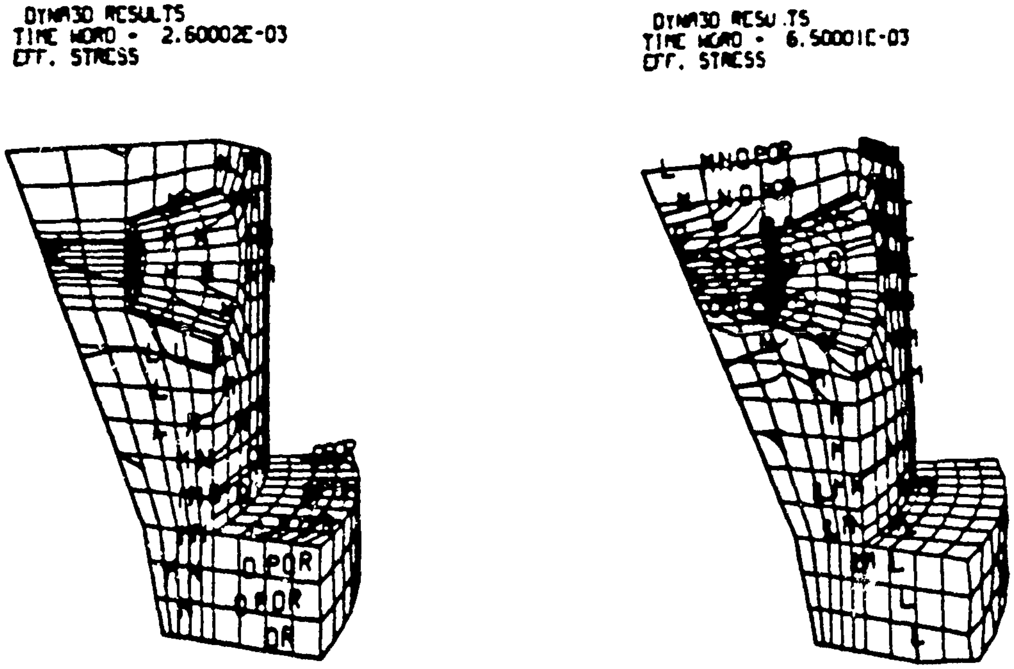
Case 1 Case 2

Fig. 7 Effective Plastic Strains in the Case Structure



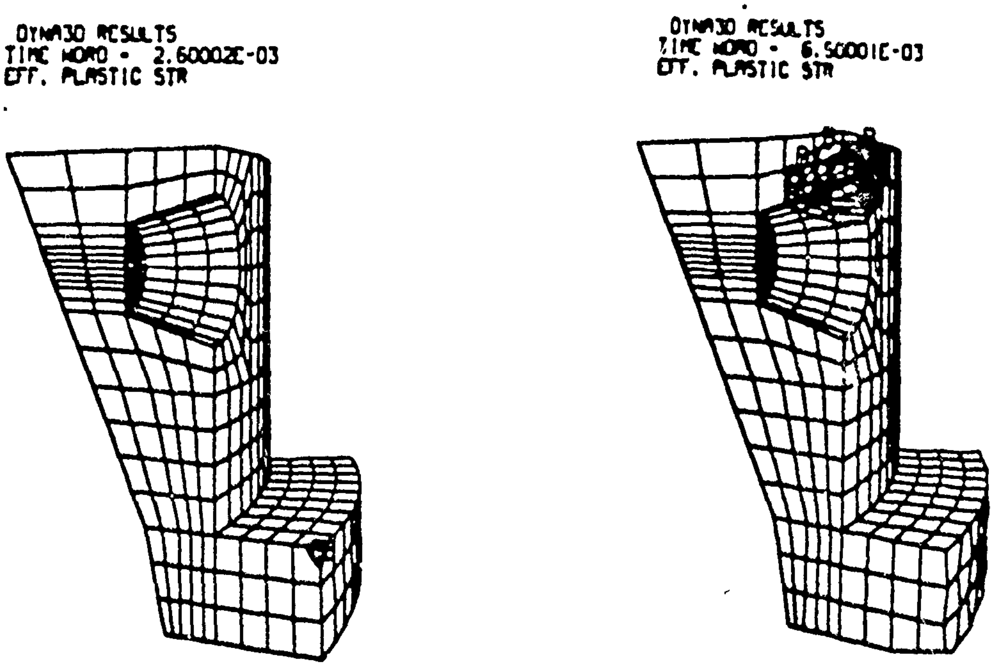
Case 1 Case 2

Fig. 8 Axial Stresses in the Wedge



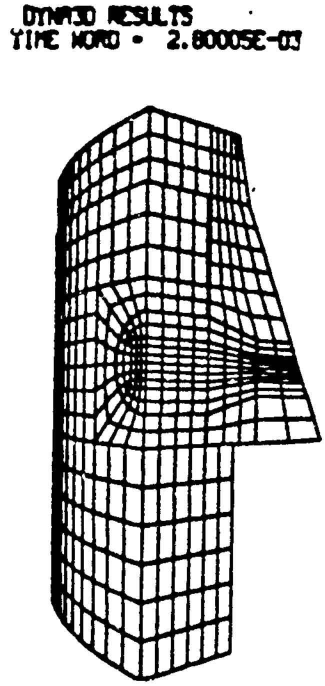
Case 1 Case 2

Fig. 9 Effective Stresses in the Wedge

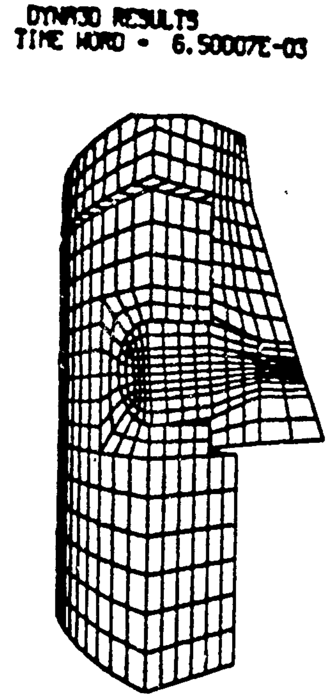


Case 1 Case 2

Fig. 10 Effective Plastic Strains in the Wedge



Case 1



Case 2

Fig. 11 Deformed Shape of Finite Element Model

Table 1 - Levels Of Stresses and Strains

	Stress (ksi)	Strain (case 1)	Strain (case 2)
A	-200	0	0
B	-180	.001	.01
C	-160	.002	.02
D	-140	.003	.03
E	-120		
F	-100		
G	-80		
H	-60		
I	-40		
J	-20		
K	0		
L	20		
M	40		
N	60		
O	80		
P	100		
Q	120		
R	140		
S	160		
T	180		
U	200		