General Disclaimer

One or more of the Following Statements may affect this Document

- This document has been reproduced from the best copy furnished by the organizational source. It is being released in the interest of making available as much information as possible.

- This document may contain data, which exceeds the sheet parameters. It was furnished in this condition by the organizational source and is the best copy available.

- This document may contain tone-on-tone or color graphs, charts and/or pictures, which have been reproduced in black and white.

- This document is paginated as submitted by the original source.

- Portions of this document are not fully legible due to the historical nature of some of the material. However, it is the best reproduction available from the original submission.

Produced by the NASA Center for Aerospace Information (CASI)

NASA Technical Memorandum 7904

(NASA-TM-79043) CELFE/NASTRAN CODE FOR THE ANALYSIS OF STRUCTURES SUBJECTED TO HIGH VELOCITY IMPACT (NASA) 25 p HC A02/MF A01 CSCL 20K
N79-15325
Unclas
G3/39 42883

CELFE/NASTRAN CODE FOR THE ANALYSIS OF STRUCTURES SUBJECTED TO HIGH VELOCITY IMPACT

C. C. Chamis
Lewis Research Center
Cleveland, Ohio

TECHNICAL PAPER presented at the
Seventh NASTRAN User's Colloquium
NASA Marshall Space Flight Center
Huntsville, Alabama, October 4-6, 1978

FEB 1979
RECEIVED
NASA STI FACILITY

CELFE/NASTRAN CODE FOR THE ANALYSIS OF STRUCTURES SUBJECTED TO HIGH VELOCITY IMPACT

by C. C. Chamis

National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio

E-9857

ABSTRACT

The CELFE (Coupled Eulerian Lagrangian Finite Element)/NASTRAN Code three-dimensional finite element code has the capability for analyzing of structures subjected to high velocity impact. The local response is predicted by CELFE and, for large problems, the far-field impact response is predicted by NASTRAN. The coupling of the CELFE code with NASTRAN (CELFE/NASTRAN code) and the application of the code to selected three-dimensional high velocity impact problems are described.

INTRODUCTION

A three-dimensional finite element computer code was recently developed for determining the response of components and structures which are subjected to high velocity impact. The schematic in figure 1 depicts a general class of high velocity impact problems for which a three-dimensional finite element code is required. This code is based on the coupled Eulerian-Lagrangian formulations (mode) and has been given the acronym CELFE (Coupled Eulerian Lagrangian Finite Element). CELFE is structured to handle high velocity impact structural response in the presence of geometric and material nonlinearities, material flow, and anisotropic material behavior. CELFE has been coupled with NASTRAN in order to: (1) take advantage of already available capabilities in NASTRAN to solve large systems of equations which require out-of-core storage, and (2) to determine the far-field structural response of large structures subjected to local high velocity impact.

The code was developed by the Lockheed Missiles and Space Company, Huntsville Research and Engineering Center, Huntsville, Alabama, under contract NAS3-18908 to NASA Lewis Research Center (LeRC). It is completely documented in references 1 to 3. The code was installed on the LeRC computer (UNIVAC 1110) in late January 1978. Presently, our effort has been devoted mainly to becoming familiar with the code capabilities and to correcting system-associated errors. It is anticipated that the code will be available for public distribution through COSMIC (University of Georgia, Athens, Georgia 30601) by early 1979.

The objective of the present paper is to provide a brief description of the CELFE/NASTRAN Code and its capabilities, and discuss selected problems which have been solved using the coupled computer program. Detailed descriptions of all aspects, that is, formulation, programming and user's manual, are given in references 1 to 3. The notation used in the discussion is defined when it first appears and it is also summarized in the appendix for convenience.

COUPLED EULERIAN-LAGRANGIAN FINITE ELEMENT (CELFE) CODE

The CELFE code used in the CELFE/NASTRAN program has the following general capabilities:

First, it simulates the transient state during the impact process and right after the process is completed; second, it computes the dynamic response of the entire structure for specified times after the impact process is completed; and third, it predicts various phenomena which occur during the impact process such as rebounding, sliding, and penetration. Also, CELFE traces the movement of the free surface, and the failure front. The specific capabilities of CELFE are summarized in table I.

The CELFE code is divided into three parts: the first part contains a finite element module to simulate the dynamic behavior in the impact Eulerian zone; the second part consists of a regular finite element procedure to analyze the dynamic response in the near field (Lagrangian) zone; and the third part consists of a coupling (interfacing) procedure to integrate the above two separate parts to form a complete system, that is, the CELFE system. The CELFE system has two options for computational purposes. The in-core CELFE code option (for small problems) and the coupled CELFE/NASTRAN code option (for large problems). The in-core option is a subset of the coupled option to allow more flexibility in utilizing the computer program.

STRUCTURAL MODELING AND FORMULATION

The CELFE Code treats the entire structure as partitioned into an impact zone and a structural (lagrangian) zone. The impact zone consists of an Eulerian zone (E zone) and a transition zone (E-L zone). The Lagrangian zone (L zone) can be divided into an in-core and a NASTRAN zone according to the options of the in-core CELFE code and the coupled CELFE/NASTRAN code. The zones are denoted L_c and L_N, respectively, as depicted schematically in figure 2. Since the in-core option is a subset of the coupled option, the Lagrangian zone (L zone) may be composed solely of L_c when the in-core option is used; or composed solely of L_N or mixed L_c and L_N when the coupled option is used. This allows the user to arrange his problem to optimize computational efficiency.

In the entire impact zone, together with the L_c zore, three-dimensional solid elements are used. In the remaining portion of the structure, that is, the L_N zone, any appropriate elements (depending on the nature of the structure) can be applied. In the impact zone (E zone and E-L zone), the dynamic behavior is handled by the three-dimensional impact element using a hydro-elasto-viscoplastic model with a general moving coordinate system. The primary variables in this zone consist of the density, ρ, momentum, ρV_j, $j = 1, 2, 3$, total energy, ρe, and deviatoric stresses, S_{ij}, $i, j = 1, 2, 3$. The remaining variables (pressure P, stresses σ_{ij}, and displacements u_j) belong to the secondary set. In the Lagrangian zone (L_c and L_N zones) conventional structural dynamic analyses are applied. In this study, the displacement method was adopted, and thus the displacements (both the lateral and rotational displacements) were chosen to be the primary variables. The governing equations, coordinate systems, and types of elements in the various zones are different. To accommodate these, the coupling procedure provides two mechanisms; namely, an in-core procedure to couple the E-L and L_c zones (or L_N zone in case of empty L_c); and an interfacing procedure to couple the main CELFE program with NASTRAN. The coupling variables for the present procedure contain lateral displacement and velocity fields.

The governing equations of the finite element formulation are summarized in table II. The associated variables, types of elements, and related aspects concerning the above discussions for various zones are summarized in table III.

SOLUTION PROCEDURE USED BY THE COMPLETE CELFE SYSTEM

The CELFE system employs a generalized two-step time integration scheme, integrated with respect to time. For the coupled CELFE/NASTRAN option, it may not be necessary to include the NASTRAN part. In the present code, it is included for the sake of simplicity in coding the interface procedure. The solution procedure includes both the in-core CELFE option and the coupled CELFE/NASTRAN option. The solution procedure is summarized in the flow chart shown in figure 3. The key computations in this flow chart are as follows:

1. Read in parameters defining the problem and start integration with respect to time (boxes 1, 2, and 3).

2. For each time step:

a. Predict the solutions of primary variables in various zones using the results obtained in the previous time step (box 5).

b. Construct the global system matrix for the entire structure in terms of coupling variables, and solve the resulting equation using the solution obtained in the previous time step (box 8).

c. Update the secondary variables in various zones using the above predicted values of the primary variables (box 9) and repeat steps a and b.

d. Update the mesh in the E-L zone and/or L_c zone (box 9).

e. Correct the results by repeating steps a to d using the above predicted solutions together with the results obtained in the previous time step.

f. Test the corrected results for the criteria governing projectile rebounding, sliding, and/or penetration (box 10).

3. If the test shows the penetration continues, repeat the steps stated in item 2 above for the next time step. Otherwise,

a. If the projectile rebounds from the target, and/or if disturbances in the structure are damped to a certain user selected fraction of the original values, the program regards the entire structure as a Lagrangian zone and the dynamic response is handled accordingly (box 7).

b. If the projectile slides over the target due to oblique impact, the program considers the mesh for the entire structure to be Lagrangian (box 7). The steps described in item 2 above must be repeated.

In order to maintain the accuracy of computations in the vicinity of the impact point, as well as to have a manageable in-core storage, the dimensions of the impact zones in this code are appropriately chosen a priori. At $t = 0$, the entire impact zones are identical with the E-L zone, where the zone is initially discretized into Lagrangian mesh (fig. 2). For $t > 0$, when failure is detected at certain nodes, the mesh containing these nodes is turned to Eulerian. Hence, the impact zone, whose dimension has to be specified a priori, must be sufficiently large to contain the failure region up to the impact process completion. If excessively large impact zones are assigned, on the other hand, it will require unnecessary computer storage and computational time.

PROCEDURE FOR COUPLING CELFE WITH NASTRAN

(CELFE/NASTRAN)

The coupled CELFE/NASTRAN code is organized to solve large structural systems as shown in the flow chart in figure 4 in order to handle either or both of the following:

1. Utilize the matrix equation solver SOLVE of NASTRAN to obtain the solution in the impact zone; and/or

2. Couple the Eulerian and Lagrangian modes for the entire structural system.

Utilization of NASTRAN Subroutine SOLVE

Symbolically, the governing equation (first equation, table II) can be rewritten for each element in the impact zone as

$$A\varphi = B \tag{1}$$

where φ is a vector of the 14 primary variables (density, internal energy, three velocities, and nine stresses); A represents geometry, material, inertia and dynamic relationships (Eulerian formulation) between the primary variables at the current time step; and B contains A and φ from the previous time step (inertial forces) and the associated dynamic forces.

Equation (1) can be solved using the NASTRAN SOLVE module for each primary variable. In order to optimize the time required for the interfacing procedure, the following scheme is used:

1. At each time step, individually compute A and B for each element, and for each primary variable. These element matrices are stored on a disk file in INPUTT2 matrix format (boxes 6, 7, and 9).

2. Store all relevant data, including the solutions of the previous time step, and those of the secondary variables, etc. into disk files (D_2, D_3, and P_1) before calling EXIT from CELFE.

3. Using NASTRAN, read (box 11) all the data of element matrices A and B (box 12), and sum the element matrices for each primary variable (box 14). Then solve equation (1) using SOLVE (box 14).

4. Write the solutions for φ on a disk file using OUTPUT2 matrix format (D_4).

5. Using CELFE, read the output disk and make further computations (box 5).

Coupling CELFE with NASTRAN (CELFE/NASTRAN)

The governing equations for this part are:

1. CELFE (impact) zone for each element (denoted by superscript)

$$[K]^{(e)}_{E-L}\{u\}^{(e)}_{E-L} = \{G\}^{(e)}_{E-L} \tag{2}$$

where K is the equivalent stiffness matrix, u the displacement vector, and G the equivalent forces all for the current time step.

2. Lagrangian and/or NASTRAN zones

$$\left\{[p^2M]_L + [pB]_L + [K]_L\right\}^{(e)}\{\delta\}^{(e)}_L = \{G\}^{(e)}_L \tag{3}$$

Summing equations (2) and (3) over $e = 1, 2, \ldots, N$, yields

$$\left\{[p^2M] + [pB] + [K]\right\}\{\delta\} = \{G\} \tag{4}$$

N is the total number of elements in the entire structure (including the projectile), p denotes time derivative (d/dt), and δ denotes global displacements.

The actual coupling variables consist of lateral displacements and velocities, u_j and v_j, $j = 1, 2, 3$, in the impact zone. The lateral displacement field consists of six degrees of freedom (three lateral and three angular displacements).

The procedure for the solution of this part is similar to the in-core version with some additional steps to couple CELFE with the NASTRAN as follows:

1. Compute all element matrices (eq. (2)) at each time step and store them in mass storage formated for the NASTRAN module INPUTT2 (box 7 and D_2, flow chart in fig. 4).

2. Compute the load vector G and store it in disk file (box 9; P_1). Store all relevant data in disk files (D_1 and D_3) before calling EXIT from CELFE (box 10).

3. Call NASTRAN (box 11)

   a. Read all element stiffness matrices from the disk file and add the matrices into NASTRAN structural stiffness matrix K in equation (3) to form the coupled system matrix K (box 13).

   b. Add the inertia and damping terms, together with the load vector (box 12) composed of those read from the breakpointed disk file for CELFE grids and form equation (4) (box 13). Solve the system and store the results in OUTPUT2 format on a disk file (box 13).

   c. Return to CELFE for further computations (boxes 15 and 4).

The procedures utilizing the equation solver module SOLVE in NASTRAN for the impact zone, and the coupling of CELFE with NASTRAN can be used either separately or simultaneously. The choice depends on the specific problem under consideration.

ANALYSIS OF A THREE-DIMENSIONAL IMPACT PROBLEM

The results of a three-dimensional impact problem are discussed in this section to illustrate the use of the CELFE/NASTRAN program. The given conditions of the example are as follows:

Target: boron/epoxy (AVCO 5505) plate with dimensions 9.75 by 8.75 by 0.083 inch

Projectile: silastic with dimensions $(0.30)^3$ in^3

Impact velocity: 630 ft/sec normal to the plate

The geometric configuration is depicted in figure 5, and material properties are listed in table IV. The boron/epoxy target was unidirectional with fibers parallel to the y-axis.

Due to the symmetry, only one quadron of the plate needed to be considered. The finite element mesh of the model is shown in figure 6. The impact zone consisted of the first 33 elements (nodes 1 to 76). Elements 30 to 33 (nodes 62 to 76) constituted the transitions zone (E-L zone).

In the Lagrangian zone all elements are NASTRAN general quadralateral elements (L_N zone). The L_c zone was assigned to be empty for simplicity. The mesh is illustrated in figure 7. To ensure displacement compatibility across the boundary, multipoint constraints (MPC) were employed to connect the CELFE and NASTRAN zones. A NASTRAN grid point was added to each CELFE midpoint grid lying on the E-L zone. This E-L zone has now become the interface of the CELFE and NASTRAN zones due to the empty L_c zone. The NASTRAN grid points added were 163, 166, 169, 172, and 175, respectively, to the CELFE grids 63, 66, 69, 72, and 75 (fig. 7). The midpoint grids were coupled by MPC's for the three translational degrees of freedom. Rotational displacements were transmitted from the quadralateral plate elements to the CELFE isoparametric elements by using rigid bars connected between the midpoint node (node 163, fig. 8) and the top and bottom nodes of the solid (nodes 64 and 62, respectively).

Results for the example problem are illustrated in figures 9 to 11. In figure 9 the pressures developed in the plate midplane nodes (5, 8, and 11, fig. 6) are plotted as a function of time. The pressures build up rapidly and are approximately the same for all three nodes. The three normal stresses (σ_{xx}, σ_{yy}, and σ_{zz}) are plotted along an axis passing through the impact center after

0.07 microsecond (fig. 10). All the stresses are compressive, increase rapidly in the contact region, and decrease very rapidly away from this region. In figure 11 the failure propagation is shown as a function of time. Failure occurs almost at the initial stages of the impact event. The results obtained from this example problem demonstrate how the CELFE/NASTRAN code can be used to describe the local and far field response of structural parts subjected to high-velocity impact.

The computer CPU times (UNIVAC 1108) for this problem were 6 minutes per time step in the impact zone and 30 minutes per time step for CELFE/NASTRAN. These times are relatively large and speed improvements are required to make the code practical for parametric and sensitivity studies.

SUMMARY

The important characteristics of the CELFE/NASTRAN code are summarized below:

1. A three-dimensional computerized capability has been developed (CELFE/NASTRAN) which can be used to describe the local and far field responses of structures subjected to high velocity impact.

2. The CELFE part of the capability accounts for local geometry and material nonlinearities, anisotropic and viscoplastic material behaviors, material flow and penetration, normal, oblique and sliding impact.

3. The coupled CELFE/NASTRAN code can handle far field material and geometry nonlinearities.

REFERENCES

1. Chan, S. T. K.; Lee, C. H.; and Brashears, M. R.: Three-Dimensional Finite Element Analysis for High Velocity Impact. (LMSC-HREC-TR-D390900, Lockheed Missiles and Space Company; NASA Contract NAS3-18908.) NASA CR-134 933, 1975.

2. Lee, C. H.: CELFE: Coupled Eulerian-Lagrangian Finite Element Program for High Velocity Impact, Part I - Theory and Formulation. (LMSC-HREC-TR-D497204-1, Lockheed Missile and Space Company; NASA Contract NAS3-18908.) NASA CR-159395, 1978.

3. Lee, C. H.: CELFE: Coupled Eulerian-Lagrangian Finite Element Program for High Velocity Impact, Part II - Program User's Manual. (LMSC-HREC-TR-D497204-11. Lockheed Missile and Space Company; NASA Contract NAS3-18908.) NASA CR-159396, 1978.

APPENDIX - SYMBOLS

A	array of coefficients (geometric, material, inertial, flow) of interrelationships between the primary variables in the Eulerian mode (eq. (1))
A_{rs}	array of displacement shape-function relationships
B	vector of inertial and dynamic forces in the Eulerian mode (eq. (1))
B_{rs}	array of relationships between the derivatives of the displacement shape functions and those of the primary variables
CELFE	Coupled Eulerian Lagrangian Finite Element
C_r	vector of relationships between velocity and displacement shape functions
c	subscript - compression
D	mass storage disk files used in code for storing arrays
E	Eulerian zone
E-L	Eulerian-Lagrangian coupling zone
E_ℓ	modulus of elasticity, subscripts denote direction
e	internal energy
G	equivalent element forces for the current time step (eq. (2))
G_ℓ	shear modulus, subscripts denote direction
INPUTT2	NASTRAN module for input data
i	index
j	index
K	equivalent element stiffness matrix for the current time step (eq. (2))
L	Lagrangian zone
L_e	CELFE zone
L_N	NASTRAN zone
M	equivalent element mass matrix (eqs. (3) and (4))
NASTRAN	NASA STRUCTURAL ANALYSIS
n	superscript - current iteration cycle

OUTPUT2	NASTRAN module for NASTRAN output data
P	pressure; mass storage disk files for load vectors
p	derivative with respect to time (d/dt)
S	subscript, shear
S_{ij}	deviatoric stress tensor
S_{ℓ}	[illegible] composite strength (subscripts denote direction and sense)
T	subscript - tension
t	time
Δt	time step
U, u	displacement
V, v	velocity
x, y, z	orthogonal coordinate reference system; subscripts denote coordinate directions
1, 2, 3	subscripts where 1 is along the fiber direction, 2 transverse, and 3 through the thickness
3-D	three-dimensional
δ	global displacement
ν_{ℓ}	Poisson's ratio, subscripts denote direction
ρ	density
ρe	total energy
ρv	momentum
σ_{ij}	stress field
φ	vector of primary variables in the Eulerian mode (eq. (1))

TABLE I. - SPECIFIC CAPABILITIES OF THE CELFE CODE

1. Predict normal and oblique impact
   Penetration process
   Sliding process
   Rebound process
2. Penetration process
   Dynamic analysis during penetration process
   Dynamic response of the structure after penetration process is completed
3. Relevant criteria in CELFE system
   Failure criterion for composites
   Plastic and viscoplastic yielding for isotropic materials
   Sliding criterion - modified coulomb's law
   Rebound criterion
   Degeneracy due to damping
4. Large structure impact via CELFE/NASTRAN coupling

TABLE II. - GOVERNING EQUATIONS OF THE

FINITE ELEMENT FORMULATION

[See appendix for symbol definitions.]

Finite element analog of impact zone:

$$\left([A_{rs}] - \frac{\Delta t}{3}[B_{rs}]\right)\left\{u_{js}^{(n+1)}\right\} = \left([A_{rs}] + \frac{2\,\Delta t}{3}[B_{rs}]\right)\left\{u_{js}^{(n)}\right\} + \Delta t\{C_r\}$$

Finite element analog of Lagrangian zone:

$$\left\{[p^2M] + [pB] + [K]\right\}\{\delta\} = \{G\}$$

Coupling variables:

$u_j, v_j;\ j = 1,\ 2,\ 3$

TABLE III. - CLASSIFICATIONS OF VARIOUS ZONES IN HIGH VELOCITY IMPACT

[See appendix for symbol definitions.]

Classification of zones	Impact zone		Lagrangian zone	
	E-zone	E-L zone	L_c zone	L_N zone
	Eulerian zone (empty at $t = 0$)	Transition zone	In-core Lagrangian zone (may be empty)	NASTRAN zone (may be empty)
Primary variables	$\rho, \rho v_j, \rho e, S_{ij}$; $i, j = 1, 2, 3$		u_j, θ_j; $j = 1, 2, 3$	
Secondary variables	u_j, P, σ_{ij}; $i, j = 1, 2, 3$		$\dot{u}_j, \dot{\theta}_j$; $j = 1, 2, 3$ $(\dot{u}_j = v_j)$	
Type of elements	3-D isoparametric elements		3-D isoparametric elements	Any compatible elements
Coupling variables	v_j, u_j; $j = 1, 2, 3$			

TABLE IV. - PROPERTIES FOR A BORON AVCO 5505 PLATE

AND A SILASTIC PROJECTILE

[Subscripts defined in the appendix.]

	Target	Projectile
Material	Boron AVCO 5505	Silastic
Density, lb/in^3	0.073	0.0512
Elastic modulus, psi	$E_{\ell 11} = 29.2\times10^6$ $E_{\ell 22} = E_{\ell 33} = 3.15\times10^6$	1.79×10^6
Shear modulus, psi	$G_{\ell 12} = G_{\ell 13} = 0.78\times10^6$ $G_{\ell 23} = 0.6\times10^6$	0.6×10^6
Poisson's ratio	$\nu_{\ell 12} = \nu_{\ell 13} = 0.17$ $\nu_{\ell 23} = 0.53$	0.49
Uniaxial failure stresses, ksi	$S_{\ell 11T} = 199.0$ $S_{\ell 11C} = 232.0$ $S_{\ell 22T} = S_{\ell 33T} = 8.1$ $S_{\ell 22C} = S_{\ell 33C} = 17.9$ $S_{\ell 12S} = S_{\ell 13S} = 9.1$ $S_{\ell 23S} = 8.9$	Yield: 27027

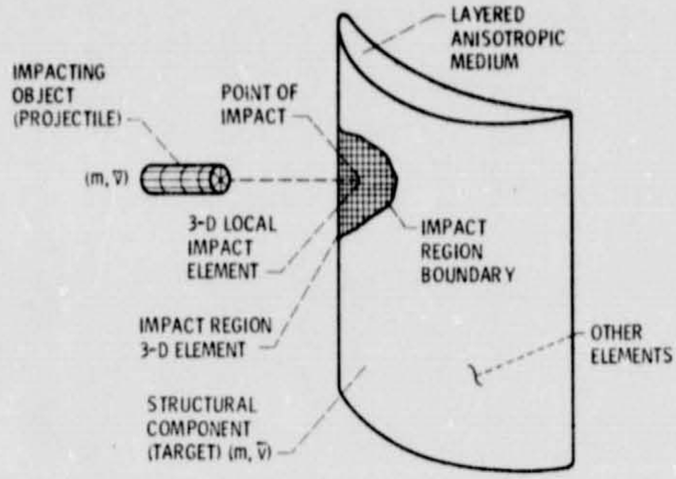

Figure 1. - Impact conditions and idealizations.

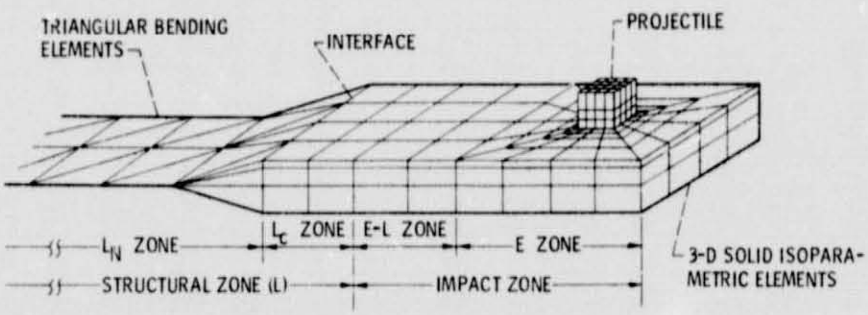

Figure 2. - Typical CELFE/NASTRAN finite element model.

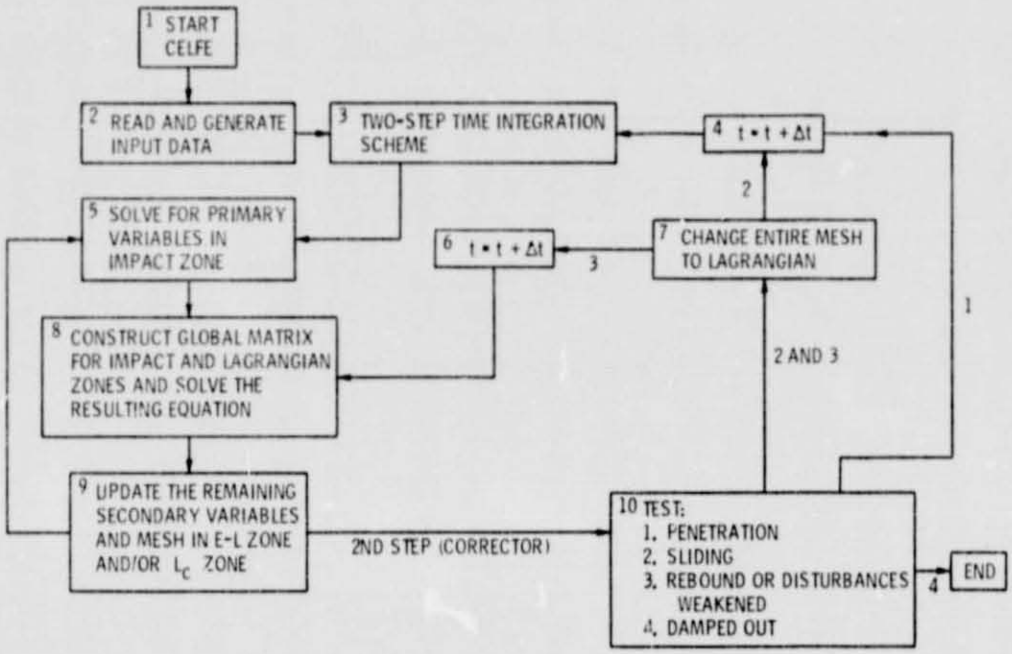

Figure 3. - Flow chart of CELFE solution procedure.

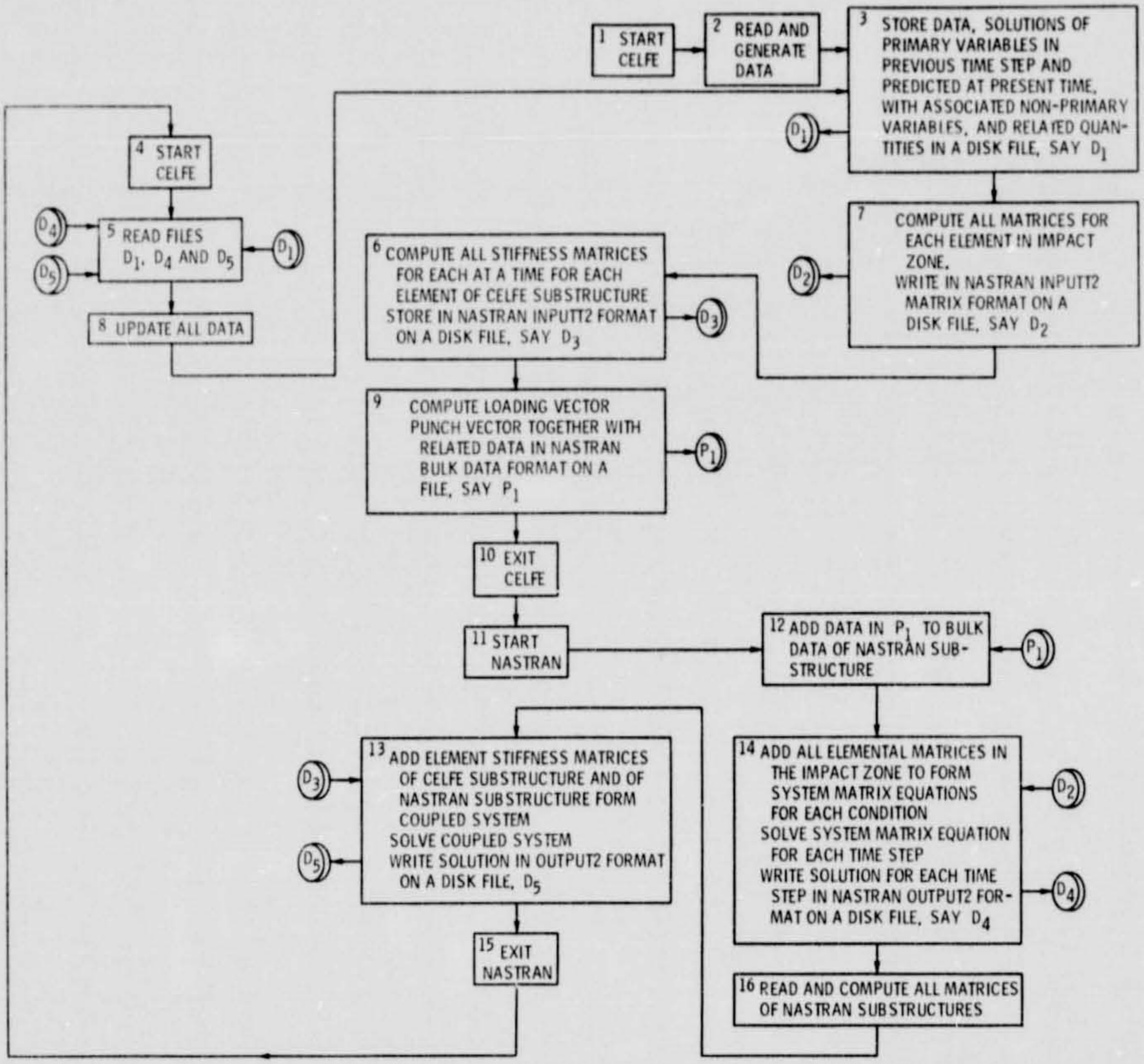

Figure 4. - Flow chart interfacing procedure for CELFE/NASTRAN.

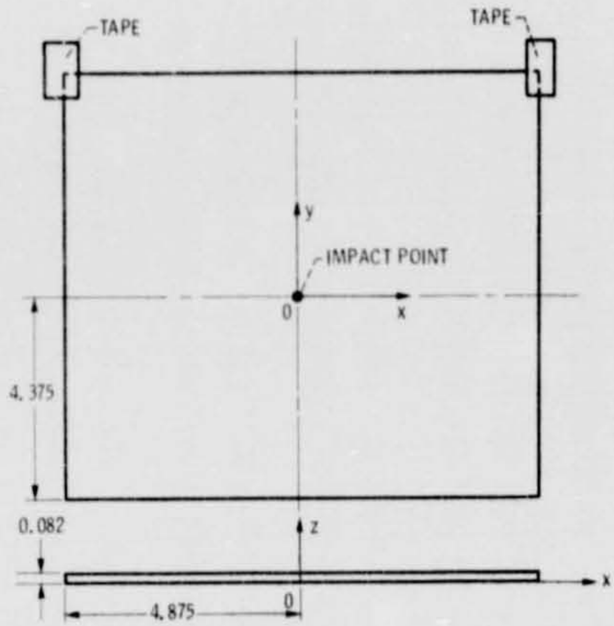

Figure 5. - Sketch of three-dimensional impact problem of a unidirectional boron AVCO 5505 composite by a silastic particle (dimensions in inches).

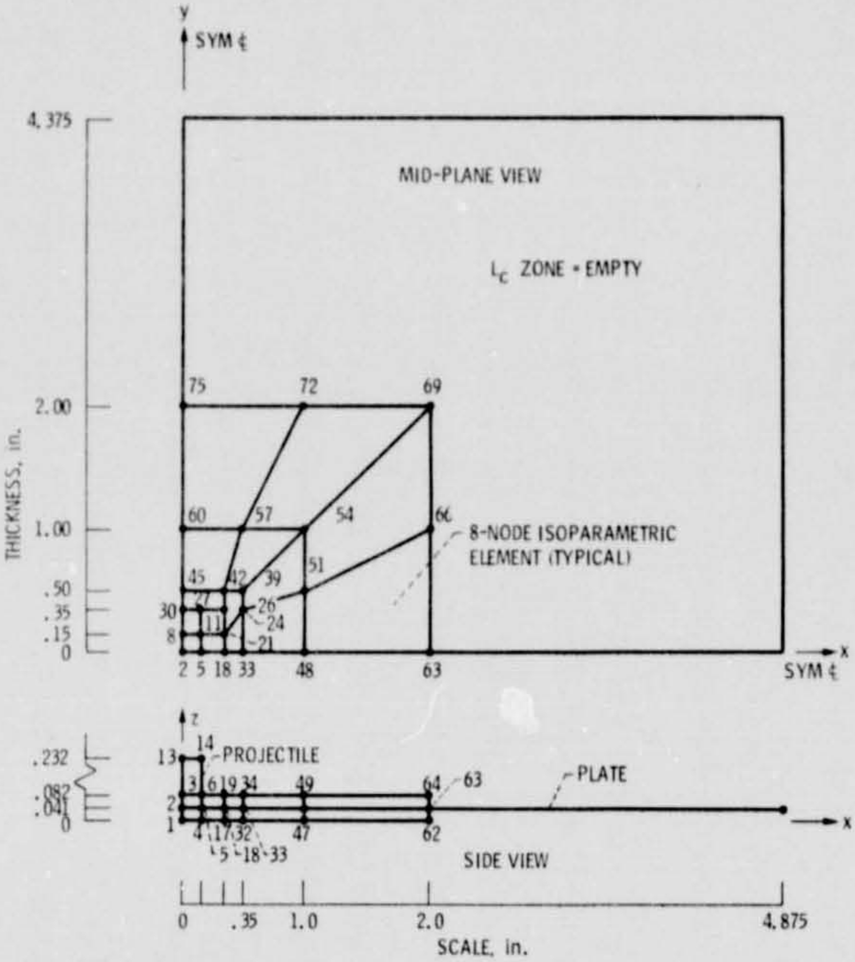

Figure 6. - Finite element model of panel of figure 5 for the CELFE/NASTRAN code. (Dimensions in inches.)

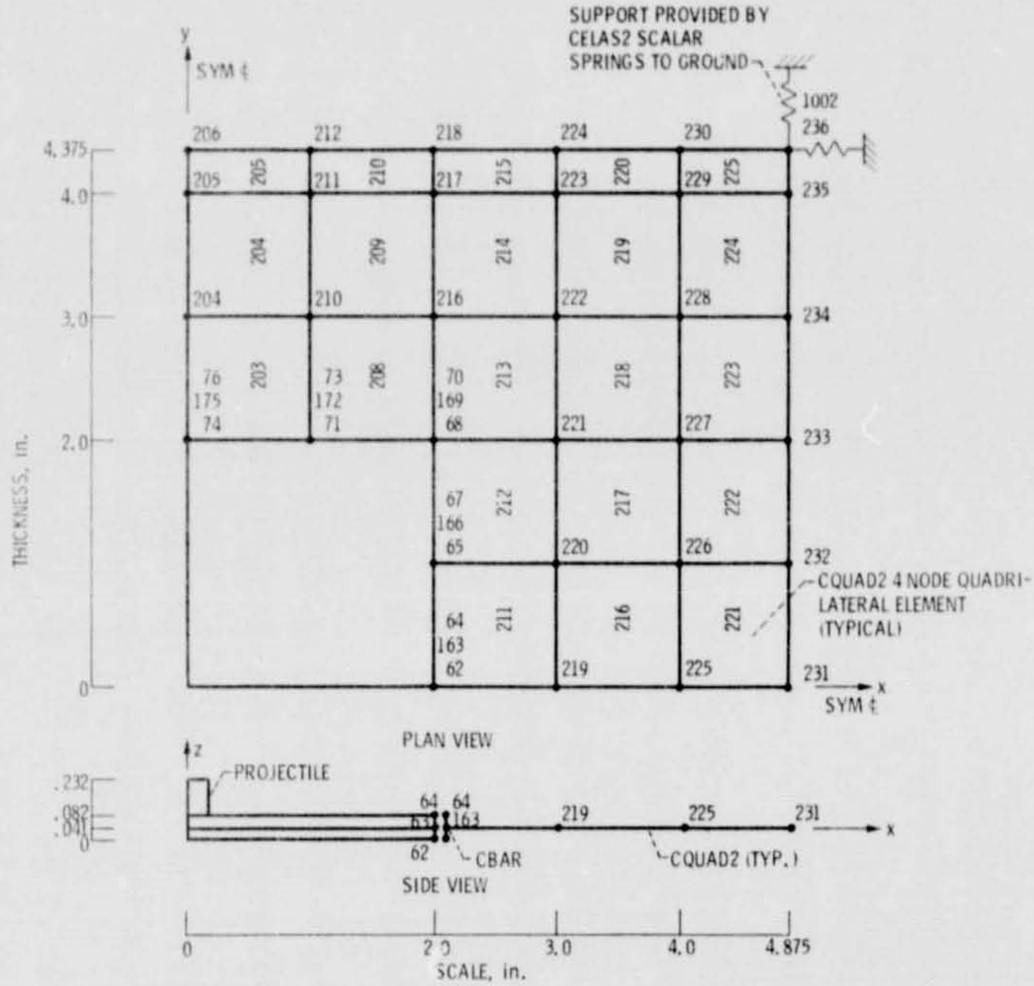

Figure 7. - NASTRAN finite element model of panel of figure 5 in CELFE/NASTRAN code.

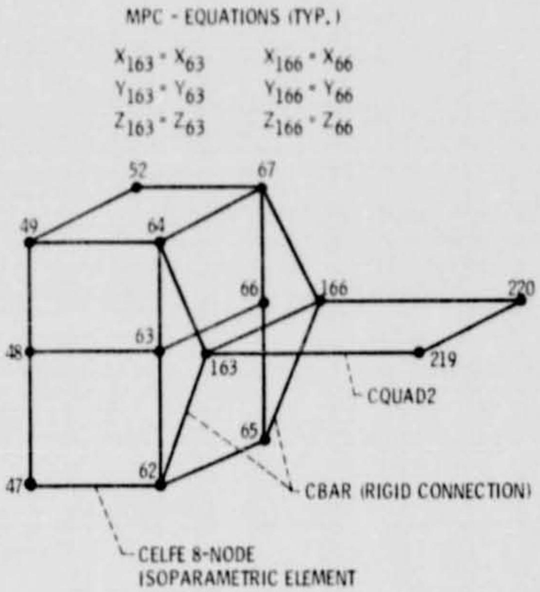

Figure 8. - CELFE/NASTRAN element interfacing.

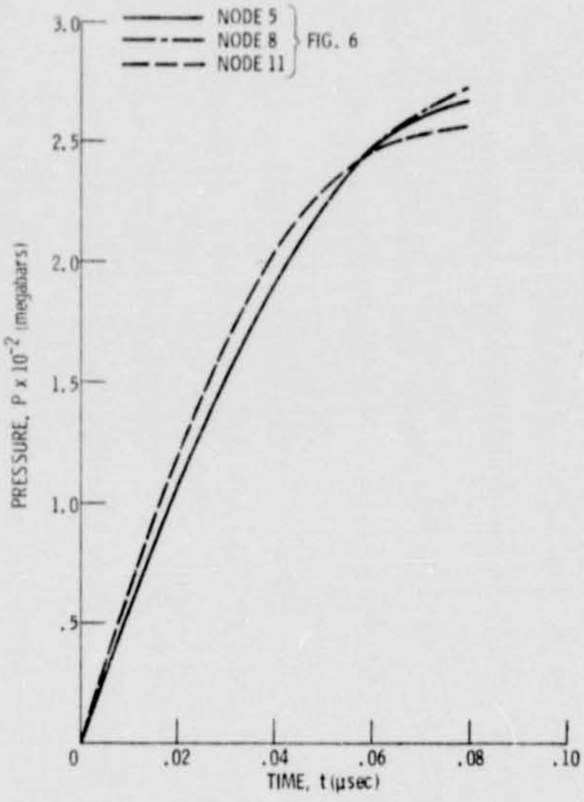

Figure 9. - Pressure developments at various nodal points.

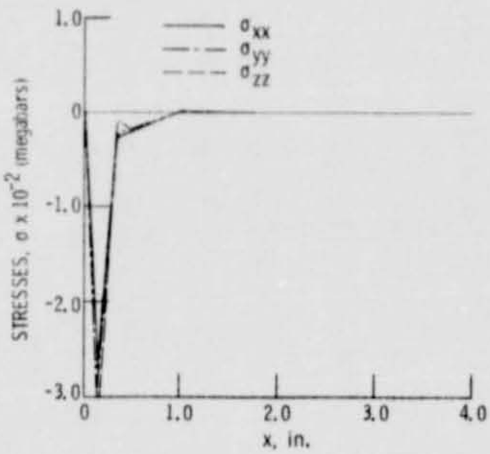

Figure 10. - Stress distributions along x-direction with y = 0.0, z = 0.041 at t = 0.07 μsec (fig. 6).

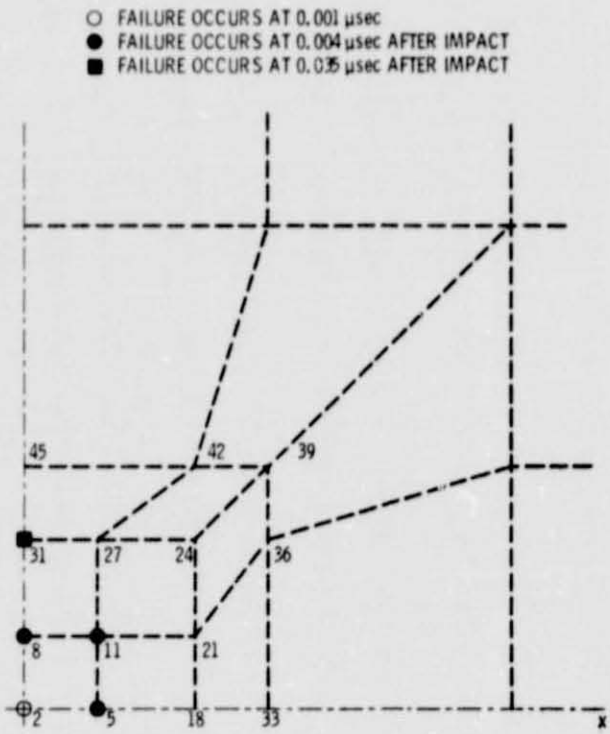

Figure 11. - Propagation of failure front with respect to time.