LA-8121-MS
Informal Report
UC-71
Issued: January 1980

**CRASHC — A Two-Dimensional Code
to Compute the Response of Axisymmetric
Shipping Containers to End-On Impacts**

Thomas A. Butler
Elton G. Endebrock
James B. Payne

CRASHC -- A TWO-DIMENSIONAL CODE TO COMPUTE THE RESPONSE OF AXISYMMETRIC SHIPPING CONTAINERS TO END-ON IMPACTS

by

Thomas A. Butler
Elton G. Endebrock
James B. Payne

ABSTRACT

Safety evaluation of radioactive material shipping containers entails determining their response to severe impact conditions. In this paper a computer code for obtaining the nonlinear response of axisymmetric shipping containers to end-on impact is described. The nonlinear equations of motion are derived with the finite element method. Large displacements and nonlinear strain and material properties are considered. The resulting computer code, CRASHC, is then used to simulate several impact tests. Results from these analyses indicate that the code can be successfully used for simulating these impact conditions. Based on the simulations, several recommendations are made for improving these kinds of analyses and for interpreting the results.

I. INTRODUCTION

This report describes a two-dimensional computer code (CRASHC) that is designed to predict the response of axisymmetric shipping containers that strike targets in end-on orientations. CRASHC is a dynamic response code that uses the finite element method to assemble the equations of motion. Because CRASHC is to be used for predicting the response of containers involved in severe impacts that can cause component failure, it considers large displacements and strains and nonlinear material properties.

Previous work at the Los Alamos Scientific Laboratory (LASL) resulted in the development of the one-dimensional computer code IMPAC2.¹ It predicts longitudinal parameters reasonably well,² but because of limitations inherent

in the one-dimensional assumptions,¹ it cannot be used to make meaningful predictions of such important parameters as shell hoop strains and two-dimensional shielding deformations. CRASHC is designed to account for these and other two-dimensional effects.

The basic numerical solution routines in CRASHC were derived from the NONSAP computer code.³ These routines were altered somewhat to make them more efficient for solution of the container impact problem. We also altered the NONSAP routines for the two-dimensional continuum finite element and the truss finite element for use in CRASHC. Fluid and shell finite elements were independently developed and added to the program to efficiently model specific container components.

In the following sections the container impact problem is described, and our approach to solving it is presented. Use of the resulting computer code CRASHC is described along with its limitations. The section that compares CRASHC results with experimental data is of particular importance because it shows the code's capability to provide realistic predictions of container response.

II. DESCRIPTION OF PROBLEM

A schematic of a typical type B radioactive material shipping container appears in Fig. 1. Radioactive material is held within an inner containment shell. This shell is encapsulated by a shielding material, usually lead or depleted uranium (D38). An outer shell serves to hold the container together and protect the shielding material. At one end of the outer shell is a closure bolted to the container with a seal or set of redundant seals to prevent leakage of material. Many containers have impact-limiting devices at both ends and on the sides. These devices may consist of metal fins (which also could serve to dissipate heat), balsa wood, metallic cones, or any of a number of other energy-mitigating materials in various geometries.

Several response measures must be calculated to show that a particular design is adequate to minimize environmental and safety risks during impact accident conditions. Deformation of the shielding material and strains in the inner and outer shells are the basic measures. Also important are the loads

Fig. 1.

Schematic of a typical Type B radioactive material shipping container.

experienced in the closure area, deformations near seals, shielding stresses and elongation in the case of depleted uranium, and the response of the radioactive contents.

CRASHC has been designed to calculate deformations and strains so that all of these measures except response of the radioactive contents may be evaluated. Response of the contents may be obtained if they can be represented axisymmetrically. Data from CRASHC can be used as input to more detailed models for more accurate prediction of response of the container contents and seal area.

Often as important as the container configuration in predicting its response when striking realistic targets is the flexibility of the target itself. CRASHC has been designed so that targets can be included in the calculations in at least an approximate sense. Details of implementing this particular feature are discussed in Appendix A where other computer code input is described.

Several aspects of container response to severe impact conditions are not amenable to normal dynamic structural analysis techniques. First, the problem is highly nonlinear. Nonlinearities occur in three distinct areas. The material properties for all container components are highly nonlinear for the large displacements normally expected. Also, the large displacements and associated large strains must be treated nonlinearly. The interface between the outer shell and shielding material is very complex during impact and is in some cases highly nonlinear. This interface is purposely welded in some containers and is therefore relatively easy to handle. However, in most containers the shielding can slide relative to the outer shell during large deformations. For lead shielding the interface will change after each impact that a container experiences (even when dropped through small distances) because of plastic flow of lead at low stress levels. Our method of handling each of these aspects is detailed in the following sections of the report.

III. THEORY

The finite element method is used to determine the nonlinear equations of motion for the shipping container. A two-dimensional axisymmetric continuum element is employed to describe the shielding material, solid impact limiters, and target. Its formulation allows for large displacements and strains and nonlinear material properties with strain hardening. The element is an updated version of that developed by Bathe⁴ and used in the NONSAP computer code.³ A specialized version of the element is used to meet the needs of the container problem. We started with a special formulation of the element from a LASL version of the NONSAP code. Large strain difficulties present in the original element were eliminated by Browning⁵ when he replaced the Almansi strain measure with a logarithmic strain measure. This strain measure is coupled with the Cauchy, or true, stress measure. Both stress and strain are converted to a material reference frame in the constitutive relations. Therefore, additional terms that account for large strain are introduced. We used only the updated Lagrangian formulation of the element in CRASHC to take advantage of Browning's changes.

Material nonlinearities are approximated with a bilinear elastic-plastic model with isotropic strain hardening. The von Mises criterion describes the

yield surface. Isotopic strain hardening is generally considered inadequate for cyclic loading, but suffices for the impact loading considered in CRASHC. Details of the material model are in Ref. 4.

An axisymmetric shell finite element was developed to represent container shell components.⁶ An isoparametric formulation is used. Because the shell components interface with the components represented with the two-dimensional continuum element, both two and three-node versions were coded to provide compatible displacement functions. The shell element considers large displacements and strains and includes shear deformation, which is important for thick shells. Each node point associated with the shell element can translate in two degrees of freedom and rotate in one. Therefore, bending as well as membrane loads are transmitted.

The nonlinear material model used with the shell element is similar to the one used for the two-dimensional continuum element. Details of its derivation appear in Appendix B. The yield function and flow rules are calculated based on membrane stresses at each integration point along the element length and through its thickness.

A fluid finite element was developed to include the effects of liquid in the container cavity. The element was derived by modifying the two-dimensional continuum finite element in the NONSAP code. The changes consisted primarily of replacing the stress-strain relations for a solid with those for a fluid. Input information was changed by substituting the bulk modulus of the fluid for the Poisson's ratio and the modulus of elasticity. The method for obtaining the fluid element was suggested in Ref. 7.

Changes were made in the equation-numbering scheme in the NONSAP code to provide for sliding interfaces between the container shielding material and inner and outer shells. This feature is appropriate only for vertical interfaces. Coincident node points are put on the shielding material and shell along the interface. These points are made dependent radially and independent vertically. The nonlinear truss element in NONSAP was retained for use in connecting these points vertically. By properly specifying nonlinear curve-description material models for the trusses, a soft connection or a stick-slip boundary between these two container components can be simulated. Details of an application of these features are discussed in Sec. IV of this report.

Once the stiffness and mass matrices are developed, the solution scheme is similar to that used in the NONSAP computer code. Some streamlining is made

possible by specializing the logic for the container impact problem. An implicit integration scheme is used to solve the nonlinear equations of motion. With this method the stiffness matrix is inverted at each time step, and displacements are determined directly. Because most containers are long and slender, the band width of the stiffness matrix is relatively small. Inverting this matrix once for each time step is not excessively time consuming because the code's inverter takes advantage of the small bandwidth. The implicit integration method is developed using the Newmark- β difference formulas. Before computation for each time step, the nonlinear stiffness matrix is updated depending on the displacement and stress fields at the beginning of that step. Within each time step an equilibrium iteration procedure is used to assure an energy balance. Details of the solution method are in Ref. 4.

IV. COMPARISONS WITH EXPERIMENTS

We used the results of several experiments performed by Battelle Columbus Laboratories (BCL) and LASL to investigate the accuracy of the analytical model. An early set of experiments performed during 1977 was first used to check linear displacement and strain features of the code and its nonlinear material models. These involved dropping a stainless steel tube filled with lead from heights up to 12.0 in. Data from other experiments performed during 1979 with scale model containers were used to further check the code including large displacement and strain features.

The stainless steel tube and lead specimen used in the BCL experiments is shown in Fig. 2. It is a 304L stainless steel tube welded to a solid steel cone. The cone acts as a waveguide so that stress waves generated at the impact surface will be nearly axisymmetric when they reach the tube. The steel tube was filled with commercial-grade lead.

The test specimen was not intended to model any actual shipping container. Rather, it is an experimental tool that simulates several of the important dynamic phenomena associated with a shipping container's response to impacts. As such, it can be used to obtain data for substantiating the analytical computer code described in this report. Data from four strain gages attached to the specimen at the locations shown in Fig. 2 were used to compare with the analytical results. Two of these gages were oriented in the longitudinal direction and were located 0.6 in. above the bottom (cone-end) of the

Fig. 2
Simplified test specimen for duplicating
container impact mechanics.

lead-filled tube. These two gages were placed 180° apart. A third gage was oriented in the hoop direction and was located next to one of the longitudinal gages near the base of the tube. The fourth gage was located one half the length of the tube from the bottom and was oriented in the longitudinal direction.

A second test specimen, similar to the one used by BCL, was used at LASL to generate additional data and to study phenomena that may occur during impacts. The dimensions of this specimen are the same as the BCL specimen except that the cone is 2.5 in. long and has a diameter of 0.5 in. at the impact plane. The BCL specimen was dropped onto a 4-in.-thick steel target, and the LASL specimen was dropped onto a 1-in.-thick aluminum target. Data were taken from the same locations for both specimens.

The BCL test specimen was dropped several times from a low level (less than 6 in.) to assure that the lead was slumped to fill edge voids next to the steel tube. These drops were repeated until no further lead slump could be measured. The LASL specimen was dropped a total of 12 times. Table I gives the drop heights for each test and lead slump data for several of the tests. As the slump data indicate, the condition of the interface between the lead and the steel was probably different for most of the drops. These lead-steel interface differences also occur in actual shipping containers making the problem difficult to treat.

When the LASL test specimen was filled with lead, a second stainless steel tube was filled with lead. Material from this tube was used to determine the quasi-static stress-strain curve for the lead in the test specimen. This

TABLE I

DROP HEIGHT FOR TESTS

<u>Drop</u>	<u>Height (in.)</u>	<u>Δ (top of steel - top of lead)(in.)</u>
1	0.5	not measured
2	0.5	not measured
3	1.0	not measured
4	1.0	not measured
5	2.0	not measured
6	2.0	0.078
7	3.3	0.105
8	1.0	0.105
9	6.0	0.140
10	12.0	0.157
11	12.0	0.157
12	1.0	

stress-strain curve is shown in Fig. 3 along with its bilinear representation used in the analysis. The experimental curve shows that the lead modulus is high for very low values of strain. However, the lead gradually yields and becomes quite soft for high strain values.

We modeled both the BCL and LASL test specimens with CRASHC to analytically determine their response to various impact conditions. The finite element meshes used for modeling the test specimens are shown in Fig. 4. For both test specimens, two-dimensional continuum elements were used to model the target plate, steel cone, and lead. Shell finite elements were used to model the tube.

Material properties used for modeling the test specimen components and target plates are shown in Tables II and III. All materials properties were represented with a bilinear stress-strain curve and von Mises yield criterion. Results from CRASHC are compared with test results using the BCL test specimen in Figs. 5-7 for a 6-in. drop. From the figures it is evident that little or

Fig. 3.
Quasi-static stress-strain curve for lead.

it is experimentally. The most likely cause of this phenomenon is the interface between the lead and stainless steel. Most of the weight is in the lead, whereas most of the stiffness and strength are in the steel. Therefore, this interface is very important. In the analytical model the steel-lead interface is welded (no slippage allowed). Figure 8 shows the result of simulating the interfaces with the stick-slip condition. This example shows that both the wave rise time and its amplitude are approximated more closely with such a model.

A third point to be considered from the data in Figs. 5-7 is that CRASHC predicts peak strains very accurately near the bottom of the tube. Peak strains halfway up the tube are overpredicted. This is probably caused by using a welded interface in the analytical model. The stick-slip interface causes the peak strains halfway up the tube to match quite well.

Figures 9-11 compare CRASHC results for a 12-in. drop with data obtained using the LASL test specimen. The most important observation from these data is that considerable plastic deformation occurred, and its value is accurately

no plastic deformation occurred in the stainless steel tube during this drop. Therefore, these data can be used to substantiate CRASHC only for linear shell response.

Several observations can be made from the data. First, CRASHC accurately predicts the overall motion of the stress wave as it passes through the specimen. When the wave reflects from the bottom of the specimen at approximately 1.0 ms, the model is no longer valid because boundary conditions at the impact surface are no longer correct. However, maximum strains will normally occur before this happens. A second observation is that the rise time for the wave is somewhat faster analytically than

Fig. 4.
Finite element meshes for test specimen.

predicted by CRASHC. In Fig. 9 experimental strain histories from the two longitudinal gages near the base of the specimen are shown. Note that these data show the drop was somewhat asymmetric. However, the final plastic strain predicted by CRASHC is very near the average of the actual plastic strain at these two locations.

The observations made from the 6-in. drop data hold for these data also. Wave rise time predictions made using CRASHC are faster here relative to the experimental data than they are for the 6-in. drops. This is because the lead-steel interface was again modeled as being welded, and, because loads are higher for a 12-in. drop, slippage is more likely to occur at the interface.

TABLE II

SIX INCH DROP (BCL SPECIMEN)

	Modulus of Elasticity <u>(psi)</u>	<u>Poisson's Ratio</u>	Yield Stress <u>(psi)</u>	Hardening Modulus <u>(psi)</u>
Target plate	30×10^6	0.30	0.45×10^5	30×10^4
Core	30×10^6	0.30	0.45×10^5	30×10^4
Shell	30×10^6	0.30	0.36×10^5	30×40^4
Lead	10×10^4	0.43	2000.	30×10^3

An additional factor that has a considerable influence on the analytical results is the effect of strain rate on the component material properties. Stress-strain curves for all of the materials are somewhat dependent on the strain rate. However, there is little experimental data that can be used to identify strain-rate effects. From Ref. 8 we do know what lead materials properties are affected the most.

TABLE III

TWELVE INCH DROP (LASL SPECIMEN)

	Modulus of Elasticity <u>(psi)</u>	<u>Poisson's Ratio</u>	Yield Stress <u>(psi)</u>	Hardening Modulus <u>(psi)</u>
Target plate	10×10^6	0.33	$.35 \times 10^5$	10×10^4
Cone	30×10^6	0.30	$.36 \times 10^5$	30×10^4
Shell	30×10^6	0.30	$.36 \times 10^5$	30×40^4
Lead	15×10^4	0.43	620.	20×10^3

Fig. 5.
Longitudinal strain at base of BCL test specimen during 6-in. drop.

Fig. 6.
Hoop strain at base of BCL test specimen.

Further insights into strain-rate effects on materials properties and other container impact phenomena were gained from simulating scale-model container tests performed during May 1979. In particular, CRASHC was used to simulate a test in which a one-quarter scale model of a truck container was dropped 70 ft in an end-on orientation. The specimen, built by BCL, was dropped at LASL during the test series reported in Ref. 9. It weighed approximately 580 lbs and was cylindrical (48 in. long and 7.3 in. diam).

Fig. 7.
Longitudinal strain at L/2 for BCL test specimen during 6-in. drop.

The CRASHC model used to simulate the test had 158 node points and 109 two-dimensional continuum elements. We first assembled the model using continuum elements for the lead shielding and the shell finite element for the shell components. Because the problem did not converge well with the shell element, we switched to two-dimensional continuum elements throughout. Both inner and outer stainless steel shells were included in the model. The interface between the lead and outer shell was allowed

Fig. 8.
Longitudinal strain at base of BCL
test specimen using a stick-slip
lead-stainless steel interface.

Fig. 9.
Longitudinal strain at base of LASL
test specimen during 12-in. drop.

Fig. 10.
Hoop strain at base of LASL test
specimen during 12-in. drop.

Fig. 11.
Longitudinal strain at L/2 for LASL
test specimen during 12-in. drop.

to slip longitudinally whereas the lead interface with the inner shell was modeled as being welded. Because the weld holding the bottom steel plate to the outer shell failed at impact, we eliminated this plate from the model. The impact boundary was not allowed to move radially to simulate frictional effects between the specimen and the impact pad. The finite element grid for the lower portion of the model near the impact plane is shown in Fig. 12.

Because no active instruments were used on the test,⁹ the measured final deformation of the container is compared with the analytical predictions. Figure 13 presents the results of this study. Several different values for the yield stress of stainless steel were used because the yield stress varies considerably with strain rate. Reference 8 reports this value to be as high as 80 000 psi for strain rates of 1000 strain/s at strain values of .04. Strain

Fig. 12.
Finite element mesh for BCL quarter-scale model lead-shielded container.

Fig. 13.
Final deformation of BCL quarter-scale model subjected to a 70-ft drop.

rates near the points of maximum deflection were calculated to be a maximum of 200 strain/s maximum. Of course the strain rate is different at each point in the container for each time. However, the model cannot handle all these variations, so the problem was parameterized using several yield values but the same value for all points during individual runs. Several values were also used for the lead yield stress. Except for the lead slump, container response is relatively insensitive to changing the value from 620 psi to 2000 psi.

Lead slump at the top of the container was measured to be 1.7 in. The analytical value varies from 0.45 in. with a lead yield stress of 620 psi to 0.30 in. for a yield stress of 2000 psi. Part of the difference between test and analysis is due to unconstrained lead motion through the void left between the lead and outer shell when the lead is cast. This void is caused by the differences in the thermal coefficient of expansion between lead and stainless steel. A nominal value for lead slump needed to fill this void is 0.55 in. A portion of the remaining 0.50 in. difference between analysis and test could be from unknown lead porosity. A change in porosity of approximately 1% would account for the total difference. Reference 9 reports porosity changes of this magnitude.

The curves in Fig. 13 show final deflection shapes for various stainless steel yield stress values. Near the base of the container where strain rates are maximum, the higher yield stress gives better agreement with test data. Further up where strain rates are lower, the displacements for lower yield stress show better agreement. Obviously, a material model is needed that sets instantaneous yield stress values based on calculated strain rates.

We used the same finite element representation of the quarter-scale container model to check our fluid finite element. The same materials properties, geometry, drop height, and time step were used to simulate a container with and without fluid (water) in the cavity. The vertical and radial displacements obtained from the two solutions are given in Table IV. Near the impact end, the radial displacements in the container with the fluid are 10-12% greater than the radial displacements of the container without the fluid. At the end away from the impact, the radial displacements from the two cases are about the same. The vertical displacements of the two cases are only slightly different.

TABLE IV

DISPLACEMENTS OF CONTAINERS WITH AND WITHOUT A FLUID

Point <u>Coordinates</u>		<u>Container Without Fluid</u>		<u>Container With Fluid</u>	
<u>Height (in.)</u>	<u>Radius (in.)</u>	<u>Radial Disp.(in.)</u>	<u>Vertical Disp.(in.)</u>	<u>Radial Disp.(in.)</u>	<u>Vertical Disp.(in.)</u>
0.25	3.52	0.0245	-0.0019	0.0288	-0.0021
0.50	3.52	0.0531	-0.0089	0.0607	-0.0085
0.75	3.52	0.0828	-0.0158	0.0934	-0.0157
1.25	3.52	0.1221	-0.0322	0.1379	-0.0336
6.00	3.52	0.0001	-0.2585	0.0025	-0.2569
22.00	3.52	0.0001	-0.2578	0.0001	-0.2577

V. COMPUTER CODE USE AND LIMITATIONS

Like most complex computer codes, CRASHC can be used effectively only if its limitations are understood. Perhaps the most important aspect to consider when using CRASHC is that its results cannot be considered accurate unless the user has investigated both temporal and spatial convergence. The implicit integration scheme used in CRASHC is unconditionally stable. However, that does not mean it is unconditionally convergent.

A good guideline to use in choosing a time step for good temporal convergence is to satisfy the formula

$$C = \frac{E}{\rho} \left(\frac{\Delta t}{\Delta x} \right)^2 \leq 25 \quad ,$$

where E is the modulus of elasticity, ρ is the mass density, Δt is the time increment, and Δx is the smallest dimension of the smallest element. When several materials are involved, each should be checked to determine the smallest suggested value for Δt . Note that this criterion is approximate. It is based on using five time increments to describe the fundamental waveform within the

smallest element. Once this time step is defined, it should be tried and the results compared with results using a smaller time step. If the results are essentially the same, then the time step chosen is sufficient for temporal convergence. If not, successively smaller time steps should be used until temporal convergence is achieved.

Spatial convergence is usually more difficult to prove than temporal convergence. For simple structures, specific guidelines can be used to determine the number of elements needed to sufficiently define the problem. However, for complicated container structures, a specific criterion is very difficult to define. Therefore, we will present an example to show the importance of considering spatial convergence and some general guidelines. The LASL test specimen described previously was analyzed using two different meshes. A portion of each of the meshes is shown in Fig. 4. Results of the analyses are shown in Figs. 14 and 15. The coarse mesh obviously gives erroneous results. For hoop response a high amplitude false wave pattern is introduced. The finer mesh eliminates this false wave pattern. If an analyst has prior knowledge that these false waves come from too coarse a mesh, he could artificially eliminate them by filtering the signal, but this is usually not the case. Also, the longitudinal response shows a different characteristic in that the strain is merely overpredicted and no false wave pattern appears. One way to overcome this problem is to try successively finer meshes until all important parameters converge. This could be very costly in man hours and computer time. A better method is to make sure that there are several elements across each region of interest. For instance, in our CRASHC model several elements should be present

Fig. 14.
Longitudinal strain at base of test specimen for different mesh densities.

Fig. 15.
Hoop strain at base of test specimen for different mesh densities.

in both the longitudinal and radial directions in this lead continuum. A second suggestion is that the analyst should always look for suspicious wave patterns in computed results. If they cannot be logically explained, a finer mesh should be used.

One of the more important factors affecting overall response of shipping containers is the representation of the interfaces between component materials that are not welded. Currently, in CRASHC an interface can be modeled as welded, slipping, or stick-slip. Most response parameters will probably be bounded by first modeling a container with sliding interfaces and then modeling it with welded interfaces. Depending on the parameter of interest, it should be maximized with one of these two cases. For instance, for the test specimen discussed in the previous section, the longitudinal strains will be maximized with a welded interface between the lead and tube, and the hoop strain at the base will be maximized with a sliding interface.

From our results the stick-slip interface seems to provide the most accurate model. However, the analyst needs a logical method for choosing the stick-slip model parameters. One possible way to treat this is to use a friction model and calculate normal loads at interfaces based on system deflections.

VI. CONCLUSIONS AND RECOMMENDATIONS

Even though CRASHC is limited to analyzing end-on impacts of axisymmetric containers, it is a valuable tool for studying the container impact problem. Because it is a specialized two-dimensional code, it is much more economical to use than general three-dimensional codes that would have to be used for studying more complex container impact problems. CRASHC can handle most mechanical phenomena present in these more general impact problems (i.e., the material models, interface problems, and other phenomena are the same or very similar). Therefore, we consider it a valuable tool for studying the shipping container impact problem, especially because it does have considerable experimental verification.

Based on the studies presented in this report, we have found that two especially critical areas to consider when analyzing container impact are the mechanical interfaces between materials and the materials properties representations. From our study of the BCL one-quarter-scale container model, we conclude that for lead-shielded containers fabricated as this model was and

dropped from heights above 30 ft, the lead-stainless steel interfaces should be modeled as either sliding or welded. The inner cavity is essentially welded to the lead, and the outer shell is free to slide relative to the lead because of the manner in which the lead shrinks during the casting procedures. For other fabrication methods and for drops from lower heights, a stick-slip interface may be more appropriate between the stainless steel and lead.

From studying the BCL quarter-scale model's response to impact, we conclude that materials properties, in particular strain-rate effects, of the stainless steel are very important. We recommend that future analytical efforts include developing a material model that will adjust materials parameters depending on instantaneous strain-rates. Because the elastic modulus and strength of the steel are so much higher than those for lead, the total container response is not nearly as sensitive to lead properties as to steel. One exception may be Poisson's ratio, which is not known precisely for lead. We used a value of 0.43 for all of our analyses. Published values range from 0.40 to 0.45.

We also found from analyzing the BCL quarter-scale model that only about one third of the total lead slump is predicted by the analysis. The other two thirds appears to be divided between the effects of filling void spaces left when casting the lead and changing lead porosity.

Certain improvements besides those mentioned above should be made to CRASHC as funds become available. One is to develop a graphics package for use with the code or to change the code for saving output for use in existing graphics packages. Another needed improvement is to change the convergence criteria so that the code runs more efficiently. This could possibly include a variable time step feature. Currently, the maximum integration time step is driven by the first few μ s of response after impact as stress waves pass through the smallest elements in the model. Improvements in the convergence criteria could also be designed to improve the convergence properties of the shell finite element.

APPENDIX A

DATA INPUT TO CRASHC

Data input to CRASHC is similar to input for the NONSAP code. If additional details are needed, refer to Ref. 3.

INPUT SUMMARY

- I. Heading Card
- II. Master Control Cards
- III. Nodal Point Data
- IV. Applied Loads Data
  - 1. Control card
  - 2. Load function data
  - 3. Nodal loads data
- V. Initial Conditions
- VI. Truss Elements
  - 1. Element group control card
  - 2. Linear elastic material/section property data
  - 3. Nonlinear elastic material/section property data
  - 4. Element data cards
- VII. Two-Dimensional Continuum Elements
  - 1. Element group control card
  - 2. Material property data
  - 3. Element data cards
- VIII. Axisymmetric Shell Element
  - 1. Element group control card
  - 2. Material property data
  - 3. Stress output table cards
  - 4. Element data cards
- IX. Fluid Elements
  - 1. Element group control card
  - 2. Material property data
  - 3. Stress output table cards
  - 4. Element data cards

I. HEADING CARD (12A6)

Columns	Variable	Entry
1 - 72	HED(12)	Enter the master heading information for use in labeling the output

II. MASTER CONTROL CARDS

Card 1 (15,6I1,4X,15,5X,15,2F10.0,15)

Columns	Variable	Entry
1 - 5	NUMNP	Total number of nodal points; EQ.0; program stop
6	IDOF(1)	Master X-translation code; EQ.0; admissible EQ.1; deleted
7	IDOF(2)	Master Y-translation code
8	IDOF(3)	Master Z-translation code
9	IDOF(4)	Master X-rotation code
10	IDOF(6)	Master Y-rotation code
11	IDOF(6)	Master Z-rotation code
16 - 20	NEGNL	Number of nonlinear element groups
26 - 30	NSTE	Number of solution time steps
31 - 40	DT	Time step increment
41 - 50	TSTART	Time at solution start
51 - 55	IPRI	Output printing interval; EQ.0; default set to "1"

Card 2 (4I5)

Columns	Variable	Entry
1 - 5	IMASS	Control flag indicating static or dynamic analysis; EQ.0; static analysis GT.0; dynamic analysis If IMASS.GT.0, a lumped (diagonal) mass matrix is used.

Columns	Variable	Entry
16 - 20	JPRINT	Control flag for printing input information; EQ.0, no input information printed GT.0; input information is printed

Card 3 (4I5,E10.4)

Columns	Variable	Entry
1 - 5	ISREF	Number of time steps between reforming effective stiffness matrix EQ.0; default set to "1"
6 - 10	NUMREF	Number of allowable stiffness reformations in each time step
11 - 15	IEQUIT	Number of time steps between equilibrium iterations EQ.0; default set to "1"
16 - 20	ITEMAX	Maximum number of equilibrium iterations permitted EQ.0; default set to "15"
21 - 30	RTOL	Relative tolerance used to measure equilibrium convergence EQ.0; default set to "1.E-3"

Card 4 (4I5)

Columns	Variable	Entry
1 - 5	NPB	Number of blocks of displacement/velocity/acceleration printout EQ.0; print all nodal point components
6 - 10	IDC	Displacement printout code EQ.0; no displacement printout EQ.1; print displacements
11 - 15	IVC	Velocity printout code EQ.0; no velocity printout EQ.1; print velocities
16 - 20	IAC	Acceleration printout code EQ.0; no acceleration printout EQ.1; print accelerations

NOTES

If NPB.EQ.0, all nodal quantities are printed regardless of the values of IDC, IVC, and IAC.

NOTES/CONT

The displacement solution at the nodes within the blocks is printed if IDC.EQ.1. In dynamic problems the velocity and/or acceleration solutions also are printed if IVC.EQ.1 and/or IAC.EQ.1.

Card 5 (16I5)

Columns	Variable	Entry
1 - 5	IPNODE(1,1)	First node of printout block no. 1.
6 - 10	IPNODE(2,1)	Last node of printout block no. 1.
11 - 15	IPNODE(1,2)	First node of printout block no. 2.
		etc.

NOTES

Two entries are expected for each printout block, namely, the first node of the block and the last node of the block. All nodal points between these two nodes will be included in the printout block.

In NPB.EQ.0, leave this card blank.

III. NODAL POINT DATA (A1,I4,A1,I4,5I5,3F10.0,I5,F10.0)

Columns	Variable	Entry
1	CT	Symbol describing the coordinate system for this node; EQ.; (blank) cartesian (X,Y,Z) EQ.X; X-cylindrical
2 - 5	N	Node (joint) number; GE.1 and LE.NUMNP
6	PSF	Print suppression flag (ignored unless N.EQ.1); EQ.; (blank) no suppression EQ.A; suppress ordered list of node coordinates EQ.B; suppress list of equation numbers EQ.C both A and B, above
7 - 10	ID(1,N)	X-translation boundary code
11 - 15	ID(2,N)	Y-translation boundary code
16 - 20	ID(3,N)	Z-translation boundary code

Columns	Variable	Entry
21 - 25	ID(4,N)	X-rotation boundary code
26 - 30	ID(5,N)	Y-rotation boundary code
31 - 35	ID(6,N)	Z-rotation boundary code
36 - 45	X(N)	X (or z)-coordinate
46 - 55	Y(N)	Y (or r)-coordinate
56 - 65	Z(N)	Z (or θ)-coordinate (degrees)
66 - 70	KN	Node number increment for node data generation; EQ.0; no generation
71 - 80	TEMP(N)	Nodal point temperature

NOTES

Two nodal points will have the same translation/rotation when a negative boundary code is used. For example, if ID(2,122) is -121, then nodal points 122 and 121 will have the same displacement in the Y-direction.

IV. APPLIED LOADS DATA

1. Control card (3I5)

Columns	Variable	Entry
1 - 5	NLOAD	Number of cards used to prescribe loads acting at the nodes
6 - 10	NLCUR	Number of load curves (time functions)
11 - 15	NPTM	Maximum number of points used to describe any one of the load curves

2. Load function data

Input NLCUR sets of the following data cards in order of increasing load function number.

a. Control data (2I5)

Columns	Variable	Entry
1 - 5	NFT	Time function number; GE.1 and LE.NLCUR

Columns	Variable	Entry
5 - 10	NPTS	Number of points (i.e., $f(t)$, t pairs) used to input this time function; GE.2 and LE.NPTM

b. [$f(t)$, t] data (8F10.0)

Columns	Variable	Entry
1 - 10	TIMV(1)	Time at point 1, t_1
11 - 20	RV (1)	Function value at point 1, $f(t_1)$
21 - 30	TIMV(2)	Time at point 2, t_2
31 - 40	RV (2)	Function value at point 2, $f(t_2)$
. . .	. . .	. . .
71 - 80	RV (4)	Function value at point 4, $f(t_4)$

Next card (if required)

1 - 10	TIMV(5)	Time at point 5, t_5
11 - 20	RV (5)	Function value at point 5, $f(t_5)$
. . .	. . .	. . .

3. Nodal loads data (3I5,F10.0)

Skip this section if NLOAD.EQ.0; otherwise input NLOAD cards in this section.

Columns	Variable	Entry
1 - 5	NOD	Node number to which this load is applied; GE.1 and LE.NUMNP
6 - 10	IDIRN	Degree-of-freedom number for this load component; EQ.0; solution terminated EQ.1; X-translation EQ.2; Y-translation EQ.3; Z-translation EQ.4; X-rotation EQ.5; Y-rotation EQ.6; Z-rotation
11 - 15	NCUR	Load curve number that describes the time dependence of the load; GE.1 and LE.NLCUR

Columns	Variable	Entry
16 - 25	FAC	Function multiplier used to scale f(t) for the load at "t"; EQ.0; default set to "1.0"

V. INITIAL CONDITIONS

Card 1 (I5)

Columns	Variable	Entry
1 - 5	ITCON	First nodal point that has a non-zero velocity

Card 2 (3E10.0)

Columns	Variable	Entry
1 - 10	V(1)	Initial velocity in the Y direction
11 - 20	V(2)	Initial velocity in the Z direction
21 - 30	V(3)	Initial rotational velocity (θ_x)

VI. TRUSS ELEMENTS

TRUSS elements are two-node members that transmit axial forces only.

1. Element control card (20I4)

Columns	Variable	Entry
1 - 4	NPAR(1)	Enter the number "1"
5 - 8	NPAR(2)	Number of TRUSS elements in this group GE.1
9 - 12	NPAR(3)	Type of nonlinear analysis EQ.0; default set to "1" EQ.1; materially nonlinear only EQ.2; updated Lagrangian formulation
56 - 60	NPAR(15)	Material model number EQ.1; linear elastic EQ.2; nonlinear elastic

Columns	Variable	Entry
61 - 64	NPAR(16)	Number of different sets of section/ material properties GE.1 EQ.0; default set to "1"
65 - 68	NPAR(17)	Number of material model constants per set EQ.0; if NPAR(15).EQ.1 GE.4; if NPAR(15).EQ.2

2. Linear elastic material/section property data

Skip this set of cards if NPAR(15).NE.1. Otherwise read NPAR(16) sets of cards.

a. Material number card (I5)

Columns	Variable	Entry
1 - 5	N	Material/section number GE.1 and LE.NPAR(16)

b. Property card (4F10.0)

Columns	Variable	Entry
1 - 10	E(N)	Young's modulus
11 - 20	AREA(N)	Cross-sectional area
21 - 30	DEN(N)	Mass density (for dynamic analysis)
31 - 40	STRAI(N)	Initial strain (axial)

3. Nonlinear elastic material/section property data

Skip this set of cards if NPAR(15).NE.2, otherwise read NPAR(16) sets of cards.

a. Material number card (I5)

Columns	Variable	Entry
1 - 5	N	Material/section number GE.1 and LE.NPAR(16)

b. Section property card (3F10.0)

Columns	Variable	Entry
1 - 10	AREA(N)	Cross-sectional area

Columns	Variable	Entry
11 - 20	DEN(N)	Mass density for dynamic analysis
21 - 30	STRAI(N)	Initial axial strain

c. Stress-strain curve card (8F10.0)

Columns	Variable	Entry
1 - 10	PROP(1,N)	Strain at point 1, e^1
11 - 20	PROP(2,N)	Strain at point 2, e^2
.	.	.
.	.	.
.	.	.
	PROP(NCON/2,N)	Strain at point NCON/2
	PROP(NCON/2+1,N)	Stress at point 1, σ^1
	PROP(NCON/2+2,N)	Stress at point 2, σ^2
.	.	.
.	.	.
.	.	.
	PROP(NCON,N)	Stress at point NCON/2

4. Element data cards (4I5,F10.0,2I5)

NPAR(2) elements must be input and/or generated in this section in ascending sequence beginning with "1"

Columns	Variable	Entry
1 - 5	M	TRUSS element number; GE.1 and LE.NPAR(2)
6 - 10	II	Node number at one end
11 - 15	JJ	Node number at other end GE.1 and LE.NUMNP
16 - 20	MTYP	Material property set number; GE.1 and LE.NPAR(16)
21 - 30	PINIT	Initial axial force in the TRUSS; GT.0, tension

Column	Variable	Entry
31 - 35	IPS	Flag for printing axial stress in TRUSS element; EQ.0; no printing EQ.1; print element stress
36 - 40	KG	Node generation increment used to compute node numbers for missing elements; EQ.0; default set to "1"

VII. TWO DIMENSIONAL CONTINUUM ELEMENTS

1. Element group control card (1614)

Columns	Variable	Entry
1 - 4	NPAR(1)	Enter the number "2"
5 - 8	NPAR(2)	Number of 2-D CONTINUUM elements in this group; GE.1
9 - 12	NPAR(3)	Code indicating type of nonlinear analysis; EQ.3; updated Lagrangian
17 - 20	NPAR(5)	Element type code; EQ.0; axisymmetric
25 - 28	NPAR(7)	Maximum number of nodes used to describe any one element; GE.4 and LE.8 EQ.0; default set to "8"
37 - 40	NPAR(10)	Numerical integration order to be used in Gauss quadrature formulae; EQ.0; default set to "2" GE.2 and LE.4
49 - 52	NPAR(13)	EQ.0, print stresses at integration points
57 - 60	NPAR(15)	Material model number; EQ.2; Elasto-plastic (von Mises)
61 - 64	NPAR(16)	Number of different sets of material properties; GE.1

2. Material property data

NPARG(16) sets of cards must be input in this section. Card "a" (material number card) is the same for all material models, but card(s) "b" (material property card(s)) depend on the material model number (NPARG(15)).

a. Material number card (15,F10.0)

Columns	Variable	Entry
1 - 5	N	Material property set numbers; GE.1 and LE.NPARG(16)
6 - 15	DEN(N)	Mass density of the material used in calculation of the mass matrix, ρ ; GE.0.0

b. Material property card(s) (8F10.0)

For MODEL "2" (NPARG(15)).EQ.2 [elastic-plastic material,
von Mises yield conditions]

Columns	Variable	Entry
1 - 10	PROP(1,N)	Young's modulus, E
11 - 20	PROP(2,N)	Poisson's ratio, ν
21 - 30	PROP(3,N)	Yield stress in simple tension, σ_y
31 - 40	PROP(4,N)	Strain hardening modulus, E_t

3. Element data cards (15,I3,I2,20X,10I5)

Columns	Variable	Entry
1 - 5	M	2-D CONTINUUM element number; GE.1 and LE.NPARG(2)
6 - 8	IEL	Number of nodes used to describe this element EQ.0; default set to "NPARG(7)" LE.NPARG(7)
9 - 10	IPS	GT.0 print stresses for this element EQ.0; no stress output for this element
31 - 35	MTYP	Material property set number assigned to this element; GE.1 and LE.NPARG(16)

Columns	Variable	Entry
36 - 40	KG	Node generation parameter used to compute node numbers for missing elements (given on first card of a sequence); EQ.0; default set to "1"
41 - 45	NOD(1)	Global node number of element nodal point 1
46 - 50	NOD(2)	Global node number of element nodal point 2
51 - 55	NOD(3)	Global node number of element nodal point 3
56 - 60	NOD(4)	Global node number of element nodal point 4
61 - 65	NOD(5)	Global node number of element nodal point 5
66 - 70	NOD(6)	Global node number of element nodal point 6
71 - 75	NOD(7)	Global node number of element nodal point 7
76 - 80	NOD(8)	Global node number of element nodal point 8

VIII. AXISYMMETRIC SHELL ELEMENT

1. Element group control card (2014)

Columns	Variable	Entry
1 - 4	NPAR(1)	Enter the number "3"
5 - 8	NPAR(2)	Number of 2-D CONTINUUM elements in this group GE.1
9 - 12	NPAR(3)	Code indicating type of nonlinear analysis; EQ.0; default set to "1" EQ.1; material nonlinear analysis only EQ.2; geometric nonlinear shell

Columns	Variable	Entry
17 - 20	NPAR(5)	Element type code; EQ.0; axisymmetric
25 - 28	NPAR(7)	Maximum number of nodes used to describe any one element; GE.4 and LE.3 EQ.0; default set to "8"
37 - 40	NPAR(10)	Numerical integration order to be used in Gauss quadrature formulae in meridional direction; EQ.0; default set to "2" GE.2; and LE.4
49 - 52	NPAR(13)	Number of stress output location tables; EQ.0, print stresses at integration points
57 - 60	NPAR(15)	Material model number; EQ.1; linear isotropic EQ.2; elastic-plastic (von Mises)
61 - 64	NPAR(16)	Number of different sets of material properties; GE.1
73 - 76	NPAR(19)	Numerical integration order to be used in Gauss quadrature formulae through the element thickness; EQ.0; default set to "1" GE.1 and LE.5 (must be an odd integer)

2. Material property data

NPAR(16) sets of cards must be input in this section. Card "a" ("material number card") is the same for all material models, but card(s) "b" ("material property card(s)") depend on the material model number (NPAR(15)).

a. Material number card (I5,F10.0)

Columns	Variable	Entry
1 - 5	N	Material property set number; GE.1 and LE.NPAR(16)
6 - 15	DEN(N)	Mass density of the material used in calculation of the mass matrix, ρ ; GE.0.0

b. Material property card(s) (8F10.0)

For MODEL "1" (NPAR(15).EQ.1)   linear isotropic

Columns	Variable	Entry
1 - 10	PROP(1,N)	Young's modulus, E
11 - 20	PROP(2,N)	Poisson's ratio, ν
21 - 30	ALPHA(1)	The meridional coefficient of thermal expansion for this element group
31 - 40	ALPHA(2)	The hoop coefficient of thermal expansion for this element group

For MODEL"2" (NPAR(15).EQ.2) elastic-plastic material, von Mises yield condition

Columns	Variable	Entry
1 - 10	PROP(1,N)	Young's modulus, E
11 - 20	PROP(2,N)	Poisson's ratio, ν
21 - 30	PROP(3,n)	Yield stress in simple tension, σ_y
31 - 40	PROP(4,N)	Strain hardening modulus, E_t
41 - 50	ALPHA(1)	The meridional coefficient of thermal expansion for this element group
51 - 60	ALPHA(2)	The hoop coefficient of thermal expansion for this element group

3. Stress output table cards (4I5)

Skip this section if stresses at integration points are to be printed, i.e., NPAR(13).EQ.0; otherwise supply NPAR(13) cards.

Columns	Variable	Entry
1 - 5	ITABLE(N,1)	Stress output location 1
6 - 10	ITABLE(N,2)	Stress output location 2
11 - 15	ITABLE(N,3)	Stress output location 3
16 - 20	ITABLE(N,4)	Stress output location 4

NOTES

Stress tables are defined to provide flexibility in element stress output requests. Each element can refer to one of the tables defined, and the element stresses are then calculated at the points specified in the table. Stresses can be printed at up to 4 integration points along the element center line.

NOTES CONT

The first "0" entry in a table will terminate that table.
For example, if ITABLE(N,1).EQ.3 and ITABLE(N,2).EQ.0, then stresses will be printed at point 3 only whenever this stress table is referred to.

4. Element data cards (I5,I3,I2,I0X,F10.,6I5,2F10.)

Columns	Variable	Entry
1 - 5	M	AXISYMMETRIC SHELL element number; GE.1 and LE.NPAR(2)
5 - 8	IEL	Number of nodes used to describe this element; EQ.0; default set to "NPAR(7)" LE.NPAR(7)
9 - 10	IPS	Number of the stress table to be used for stress calculations; EQ.0; no stress output for this element
21 - 30	THIC(1)	Shell thickness at NOD(1). If this is zero the thickness will be the same as the last element.
31 - 35	MTYP	Material property set number assigned to this element; GE.1 and LE.NPAR(16)
36 - 40	KG	Node generation parameter used to compute node numbers for missing elements (given on first card of a sequence); EQ.0; default set to "1"
41 - 45	NOD(1)	Global node number of element nodal point 1
46 - 50	NOD(2)	Global node number of element nodal point 2
51 - 55	NOD(3)	Global node number of element nodal point 3 NOD(3) is the middle nodal point
61 - 70	THIC(2)	Shell thickness at NOD(2)
71 - 80	THIC(3)	Shell thickness at NOD(3)

NOTES

If THIC(1) is not zero and THIC(2) is zero, then THIC(2) and THIC(3) are set equal to THIC(1). If THIC(1) is zero, then THIC(1), THIC(2), and THIC(3) are all set equal to the corresponding thicknesses of the last element.

IX. FLUID ELEMENTS

1. Element group control card (2014)

Columns	Variable	Entry
1 - 4	NPAP(1)	Enter the number "4"
5 - 8	NPAP(2)	Number of 2-D CONTINUUM elements in this group GE.1
9 - 12	NPAP(3)	Code indicating type of nonlinear analysis; EQ.3; updated Lagrangian
17 - 20	NPAP(5)	Element type code; EQ.0; axisymmetric
25 - 28	NPAP(7)	Maximum number of nodes used to describe any one element; GE.4 and LE.8 EQ.0; default set to "8"
37 - 40	NPAP(10)	Numerical integration order to be used in Gauss quadrature formulae; EQ.0; default set to "2" GE.2 and LE.4
49 - 52	NPAP(13)	Number of stress output location tables; EQ.0; print stresses at integration points
57 - 60	NPAP(15)	Material model number; EQ.1; linear isotropic
61 - 64	NPAP(16)	Number of different sets of material properties; GE.1

2. Material property data

NPAP(16) sets of cards must be input in this section. Card "a" (material number card) is the same for all material models, but card(s) "b" (material property card(s)) depend on the material model number (NPAP(15)).

a. Material number card (15,F10.0)

Columns	Variable	Entry
1 - 5	N	Material property set number; GE.1 and LE.NPAR(16)
6 - 15	DEN(N)	Mass density of the material used in calculation of the mass matrix, ρ ; GE.0.0

b. Material property card(s) (8F10.0)

For MODEL "1" (NPAR(15).EQ.1) linear isotropic

Columns	Variable	Entry
1 - 10	PROP(1,N)	Bulk Modulus, K

3. Stress output table cards (9I5)

Skip this section if stresses at integration points are to be printed, i.e., NPAR(13).EQ.0; otherwise supply NPAR(13) cards.

Columns	Variable	Entry
1 - 5	ITABLE(N,1)	Stress output location 1
6 - 10	ITABLE(N,2)	Stress output location 2
11 - 15	ITABLE(N,3)	Stress output location 3
16 - 20	ITABLE(N,4)	Stress output location 4
21 - 25	ITABLE(N,5)	Stress output location 5
26 - 30	ITABLE(N,6)	Stress output location 6
31 - 35	ITABLE(N,7)	Stress output location 7
36 - 40	ITABLE(N,8)	Stress output location 8
41 - 45	ITABLE(N,9)	Stress output location 9

NOTES

Stress tables are defined to provide flexibility in element stress output requests. Each element can refer to one of the tables defined, and the element stresses are then calculated at the points specified in the table. Refer to Ref. 4 for selection of the stress calculation points. The first "0" entry in a table will terminate that table. For example, if ITABLE(N,1).EQ.7 and ITABLE(N,2).EQ.0, then stresses will be printed at point 7 only when this stress table is referred to.

4. Element data cards (I5,I3,I2,I0X,F10.0,I0I5)

Columns	Variable	Entry
1 - 5	M	2-D CONTINUUM element number; GE.1 and LE.NPAR(2)
6 - 8	IEL	Number of nodes used to describe this element; EQ.0; default set to "NPAR(7)" LE.NPAR(7)
9 - 10	IPS	Number of the stress table to be used for stress calculations; EQ.0; no stress output for this element
31 - 35	MTYP	Material property set number assigned to this element; GE.1 and LE.NPAR(16)
36 - 40	KG	Node generation parameter used to compute node numbers for missing elements (given on first card of a sequence); EQ.0; default set to "1"
41 - 45	NOD(1)	Global node number of element nodal point 1
46 - 50	NOD(2)	Global node number of element nodal point 2
51 - 55	NOD(3)	Global node number of element nodal point 3
56 - 60	NOD(4)	Global node number of element nodal point 4
61 - 65	NOD(5)	Global node number of element nodal point 5
66 - 70	NOD(6)	Global node number of element nodal point 6
71 - 75	NOD(7)	Global node number of element nodal point 7
76 - 80	NOD(8)	Global node number of element nodal point 8

APPENDIX B

NONLINEAR MATERIAL MODEL FOR SHELL FINITE ELEMENT

This appendix presents the nonlinear material model used with the shell finite element. A bilinear approximation of the material stress-strain curve is used with von Mises yield criterion.¹⁰ A brief summary of the yield criterion is given followed by a description of the plastic stress-strain relations.

The von Mises yield condition is defined by the function

$$f(\sigma, \epsilon^p, \kappa) = J_2 - \kappa^2, \quad (B-1)$$

where J_2 is the second invariant of the stress deviation.¹⁰ The resulting yield surface is shown in Fig. B-1. If we use the yield stress in simple tension, $\kappa^2 = \frac{\sigma_y^2}{3}$, where σ_y is the yield stress. Isotropic hardening is assumed, so σ_y is a function of the plastic deformation. The total plastic strain is ϵ^p , and σ is the stress.

For principle stresses in a biaxial stress state,

$$J_2 = 1/3(\sigma_{11}^2 + \sigma_{22}^2 - \sigma_{11}\sigma_{22}), \text{ and} \quad (B-2)$$

$$f(\sigma, \epsilon^p, \kappa) = 1/3(\sigma_{11}^2 + \sigma_{22}^2 - \sigma_{11}\sigma_{22} - \sigma_y^2).$$

When the material is yielding, $f(\sigma, \epsilon^p, \kappa) = 0$. If $f(\sigma, \epsilon^p, \kappa) < 0$, then an elastic state exists, and $f(\sigma, \epsilon^p, \kappa) > 0$ is not physically possible.

For the shell finite element, $\sigma_{11} = \sigma_s$, or meridional stress, and $\sigma_{22} = \sigma_\theta$, or hoop stress. Substituting into (B-2),

$$f(\sigma, \epsilon^p, \kappa) = 1/3(\sigma_s^2 + \sigma_\theta^2 - \sigma_s\sigma_\theta - \sigma_y^2) \quad (B-3)$$

Fig. B-1.

Expansion of von Mises yield surface during isotropic hardening.

Equation (B-3) is the plastic potential function to which the normality rule can be applied, or, in matrix notation,

$$\begin{Bmatrix} d\epsilon_s^p \\ d\epsilon_\theta^p \end{Bmatrix} = d\lambda \begin{Bmatrix} \frac{\partial f}{\partial \sigma_s} \\ \frac{\partial f}{\partial \sigma_\theta} \end{Bmatrix},$$

where $d\lambda$ is a scalar to be determined. Figure B-1 shows the yield surface defined by Eq. (B-3). The normality rule states that increments in plastic strain must be normal to this surface (see Fig. B-1).

Because $f(\sigma, \epsilon^p, \kappa) = 0$ during plastic deformation, the total derivative

$$df = \begin{bmatrix} \frac{\partial f}{\partial \sigma_s} & \frac{\partial f}{\partial \sigma_\theta} \end{bmatrix} \begin{Bmatrix} d\sigma_s \\ d\sigma_\theta \end{Bmatrix} + \begin{bmatrix} \frac{\partial f}{\partial \epsilon_s^p} & \frac{\partial f}{\partial \epsilon_\theta^p} \end{bmatrix} \begin{Bmatrix} d\epsilon_s^p \\ d\epsilon_\theta^p \end{Bmatrix} = 0,$$

or,

$$\begin{bmatrix} \frac{\partial f}{\partial \sigma_s} & \frac{\partial f}{\partial \sigma_\theta} \end{bmatrix} \begin{Bmatrix} d\sigma_s \\ d\sigma_\theta \end{Bmatrix} = - \begin{bmatrix} \frac{\partial f}{\partial \epsilon_s^p} & \frac{\partial f}{\partial \epsilon_\theta^p} \end{bmatrix} \begin{Bmatrix} d\epsilon_s^p \\ d\epsilon_\theta^p \end{Bmatrix}.$$

Use the notation

$$d\sigma = \begin{Bmatrix} d\sigma_s \\ d\sigma_\theta \end{Bmatrix}, \quad \frac{\partial f}{\partial \sigma} = \begin{Bmatrix} \frac{\partial f}{\partial \sigma_s} \\ \frac{\partial f}{\partial \sigma_\theta} \end{Bmatrix}, \quad \sigma = \begin{Bmatrix} \sigma_s \\ \sigma_\theta \end{Bmatrix},$$

$$d\epsilon^p = \begin{Bmatrix} d\epsilon_s^p \\ d\epsilon_\theta^p \end{Bmatrix}, \quad \frac{\partial f}{\partial \epsilon^p} = \begin{Bmatrix} \frac{\partial f}{\partial \epsilon_s^p} \\ \frac{\partial f}{\partial \epsilon_\theta^p} \end{Bmatrix}, \quad \text{and}$$

$$d\epsilon = \begin{Bmatrix} d\epsilon_s \\ d\epsilon_\theta \end{Bmatrix} \quad (\text{total strain increment}).$$

Then

$$\left(\frac{\partial f}{\partial \sigma}\right)^T d\sigma = -\left(\frac{\partial f}{\partial \epsilon^p}\right)^T d\epsilon^p .$$

The stress increments are calculated from the relation

$$d\sigma = C^E (d\epsilon - d\epsilon^p) ,$$

where C^E is the 2×2 constitutive matrix. We can eliminate $d\epsilon^p$ and $d\sigma$ to obtain

$$d\lambda = \frac{\left(\frac{\partial f}{\partial \sigma}\right)^T C^E d\epsilon}{-\left(\frac{\partial f}{\partial \epsilon^p}\right)^T \frac{\partial f}{\partial \sigma} + \left(\frac{\partial f}{\partial \sigma}\right)^T C^E \frac{\partial f}{\partial \sigma}} . \quad (B-4)$$

The incremental elastic-plastic material law becomes

$$d\sigma = C^{EP} d\epsilon ,$$

where

$$C^{EP} = C^E - \frac{C^E \frac{\partial f}{\partial \sigma} \left(C^E \frac{\partial f}{\partial \sigma}\right)^T}{-\left(\frac{\partial f}{\partial \epsilon^p}\right)^T \frac{\partial f}{\partial \sigma} + \left(\frac{\partial f}{\partial \sigma}\right)^T C^E \frac{\partial f}{\partial \sigma}} . \quad (B-5)$$

The vector $\partial f / \partial \sigma$ can be directly evaluated from Eq. (B-2) as

$$\frac{\partial f}{\partial \sigma} = \begin{Bmatrix} \frac{\partial f}{\partial \sigma_s} \\ \frac{\partial f}{\partial \sigma_\theta} \end{Bmatrix} = \frac{2}{3} \begin{Bmatrix} \sigma_s - .5 \sigma_\theta \\ \sigma_\theta - .5 \sigma_s \end{Bmatrix} .$$

The increment in plastic work per unit volume can be used to determine $\partial f / \partial \epsilon^p$.
The increment in plastic work is

$$dW^p = \sigma d\epsilon^p .$$

From Eq. (B-1)

$$\frac{\partial f}{\partial \epsilon^p} = \frac{\partial J_2}{\partial \epsilon^p} - \frac{\partial \kappa^2}{\partial \epsilon^p} ; \quad \frac{\partial J_2}{\partial \epsilon^p} = 0 ,$$

because

$$\kappa^2 = \frac{\sigma_y^2}{3} \text{ and } \frac{\partial f}{\partial \epsilon^p} = - \frac{\partial \kappa^2}{\partial \epsilon^p} = - \frac{2}{3} \sigma_y \frac{\partial \sigma_y}{\partial \epsilon^p}$$

or

$$\frac{\partial f}{\partial \epsilon^p} = - \frac{2}{3} \sigma_y \frac{d\sigma_y}{dW^p} \frac{\partial W^p}{\partial \epsilon^p} = - \frac{2}{3} \sigma_y \frac{d\sigma_y}{dW^p} \sigma .$$

Finally,

$$\frac{\partial f}{\partial \epsilon^p} = \frac{2}{3} \sigma_y \frac{d\sigma_y}{dW^p} \begin{Bmatrix} \sigma_s \\ \sigma_\theta \end{Bmatrix} . \quad (B-6)$$

While the material is yielding, the equivalent uniaxial stress is equal to σ_y (i.e., $f(\sigma, \epsilon^p, \kappa) = 0$), which varies linearly for the bilinear case. So the plastic work done is

$$W^p = \int_{\text{volume}} \sigma_y d\epsilon^p ,$$

where

$d\epsilon^p$ is the increment of equivalent uniaxial plastic strain

and

$$d\bar{\epsilon}^p = \left\{ 2/3 \left((d\epsilon_s^p)^2 - d\epsilon_s^p d\epsilon_{xy}^p + (d\epsilon_{xy}^p)^2 \right) \right\}^{1/2}$$

But

$$d\bar{\epsilon} = d\bar{\epsilon}^e + d\bar{\epsilon}^p ,$$

or

$$d\bar{\epsilon}^p = \frac{d\sigma_y}{E_H} - \frac{d\sigma_y}{E} = d\sigma_y \left(\frac{E - E_H}{EE_H} \right) ,$$

where

E = elastic modulus and

E_H = hardening modulus.

Then

$$W^p = \int \sigma_y \left(\frac{E - E_H}{EE_H} \right) d\sigma_y$$

or

$$dW^p = \sigma_y \left(\frac{E - E_H}{EE_H} \right) d\sigma_y ,$$

and

$$\frac{EE_H}{E - E_H} = \sigma_y \frac{d\sigma_y}{dW^p} . \quad (B-7)$$

Combining Eqs. (B-6) and (B-7) gives

$$\frac{\partial f}{\partial \sigma^p} = - \frac{2}{3} \frac{E E_H}{E - E_H} \begin{Bmatrix} \sigma_s \\ \sigma_\theta \end{Bmatrix} \quad . \quad (B-8)$$

Substituting Eq. (B-8) into (B-5) gives the final elastic-plastic constitutive matrix

$$C^{EP} = C^E \left(1 - \frac{2}{3} \frac{E E_H}{E - E_H} (\sigma)^T \left(\frac{\partial f}{\partial \sigma} \right)^T + \left(\frac{\partial f}{\partial \sigma} \right)^T C^E \frac{\partial f}{\partial \sigma} \right) ,$$

where

$$\frac{\partial f}{\partial \sigma} = \frac{2}{3} \begin{Bmatrix} \sigma_s - .5\sigma_\theta \\ \sigma_\theta - .5\sigma_s \end{Bmatrix} \equiv \text{deviatoric stresses.}$$

ACKNOWLEDGMENTS

The authors acknowledge the use of a LASL version of the NONSAP computer code as the basis for establishing computer code CRASHC. NONSAP was originally written by K. J. Bathe, now of the Massachusetts Institute of Technology. The LASL version has several improvements developed by R. V. Browning of LASL Group WX-3. We also wish to acknowledge the work of R. C. Dove and R. L. Rhorer in obtaining the response data for the LASL test specimen discussed in Sec. IV.

REFERENCES

1. J. Counts and J. B. Payne, "IMPAC2 - A Lumped-Mass Nonlinear Spring Computer Program to Analyze Type B Shipping Container Impact Problems," Los Alamos Scientific Laboratory report LA-6643-MS (January 1977).
2. J. Counts and J. B. Payne, "Comparison of Predicted and Measured Drop Test Responses of Type B Shipping Containers," Los Alamos Scientific Laboratory report LA-6642-MS (January 1977).
3. K. J. Bathe, E. L. Wilson, and R. H. Iding, "NONSAP - A Structural Analysis Program for Static and Dynamic Response of Nonlinear Systems," University of California report SESM 74-3 (1974).
4. K. J. Bathe, H. Ozdemir, and E. L. Wilson, "Static and Dynamic Geometric and Material Nonlinear Analysis," University of California report SESM 74-4 (1974).
5. R. V. Browning, "Notes on Logarithmic Strain Updates for NONSAP," Los Alamos Scientific Laboratory unpublished notes (November 1976).
6. W. A. Cook, "A Finite Element Model for Nonlinear Shells of Revolution," Los Alamos Scientific Laboratory report LA-8138-MS (November 1979).
7. D. Shantaram, D. R. J. Owens, and O. C. Zienkiewicz, "Dynamic Transient Behavior of Two- and Three-Dimensional Structures Including Plasticity, Large Deformation Effects and Fluid Interaction," *Earthquake Engineering and Structural Dynamics*, 4, 561-578 (1976).
8. R. A. Robinson, W. J. Zielenbach, and A. A. Lawrence, "A Survey of Strain-Rate Effects for Some Common Structural Materials Used in Radioactive Material Packaging and Transportation Systems," Battelle Columbus Laboratories (February 1976).
9. R. J. Burian, W. E. Black, A. A. Lawrence, and M. E. Balmert, "The Validity of Scale Modeling for Large Deformations in Shipping Containers," Battelle Columbus Laboratories report BMI-2040 (October 1979).
10. Y. C. Fang, Foundations of Solid Mechanics (Prentice-Hall, Inc., Englewood Cliffs, New Jersey, 1965), pp. 137-152.