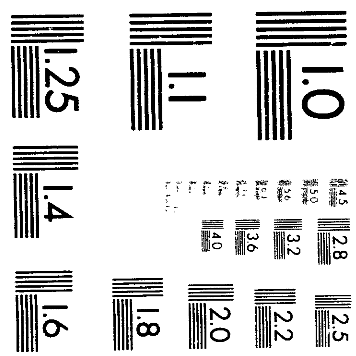

1.0
1.1
1.25
1.4
1.6
1.8
2.0
2.2
2.5
2.8
3.2
3.6
4.0

1 of 1

Conf-931121--30

UCRL-JC-114579
PREPRINT

CRASHWORTHINESS ANALYSIS USING ADVANCED MATERIAL MODELS IN DYNA3D

Roger W. Logan
Michael J. Burger
Larry D. McMichael
Ray D. Parkinson

This paper was prepared for presentation at the
1993 ASME Winter Annual Meeting
SERAD Emerging Technology Session
New Orleans, LA
November 29-30, 1993

October 22, 1993

Lawrence Livermore National Laboratory

CRASHWORTHINESS ANALYSIS USING ADVANCED MATERIAL MODELS IN DYNA3D

Roger W. Logan
Michael J. Burger
Larry D. McMichael
University of California
Lawrence Livermore National Laboratory
Livermore, CA

Ray D. Parkinson
Kaiser Aluminum & Chemical Corporation
Center for Technology
Pleasanton, CA

ABSTRACT

As part of an electric vehicle consortium, LLNL and Kaiser Aluminum are conducting experimental and numerical studies on crashworthy aluminum spaceframe designs. We have jointly explored the effect of heat treat on crush behavior and duplicated the experimental behavior with finite-element simulations. The major technical contributions to the state of the art in numerical simulation arise from the development and use of advanced material model descriptions for LLNL's DYNA3D code. Constitutive model enhancements in both flow and failure have been employed for conventional materials such as low-carbon steels, and also for lighter weight materials such as aluminum and fiber composites being considered for future vehicles. The constitutive model enhancements are developed as extensions from LLNL's work in anisotropic flow and multiaxial failure modeling. Analysis quality as a function of level of simplification of material behavior and mesh is explored, as well as the penalty in computation cost that must be paid for using more complex models and meshes. The lightweight material modeling technology is being used at the vehicle component level to explore the safety implications of small neighborhood electric vehicles manufactured almost exclusively from these materials.

INTRODUCTION AND BACKGROUND

Lawrence Livermore National Laboratory (LLNL) is developing technology for an integrated package for the analysis of vehicle handling and impact into roadside features and other vehicles. The program involves the development and use of rigid-body algorithms and LLNL's DYNA and NIKE finite-element codes. The goal is a tool for use by highway engineers at Federal Highway Administration (FHWA) and state DOTs allowing good quantitative results at the workstation level. The work has involved integration of handling and deformation codes, development of material and tire models, and comparisons to test data. In this work, LLNL and Kaiser Aluminum's Center For Technology (CFT) have conducted a cooperative study keyed toward the use of aluminum extrusions in a space-frame vehicle design. The technology advances are synergistic with some of LLNL's other missions as well, with a key example being the spaceframe shipping container for weapon components, as described below.

One goal of LLNL's work in crashworthiness analysis is to develop the technical capability to accurately model vehicle/barrier crash and post-crash behavior. An improved analysis capability will improve highway barrier (and possibly vehicle) designs to minimize risk to occupants, and minimize the hazards due to post-crash vehicle motion. These technical developments will form an integral part of the VISTA (Vehicle Impact Simulation Technology Advancement) program in conjunction with the FHWA, National Highway Traffic Safety Administration (NHTSA), and others. One goal of VISTA is to integrate the entire state of technology, including DYNA3D (Whirley, 1991), NIKE3D (Maker, 1991), and subsequent work into a user-friendly highway design tool useful at various levels of expertise.

The current state-of-the-art in barrier design and post-crash dynamics involves a mixture of actual testing using instrumented vehicles, and empirical/numerical modeling using small personal-computer based

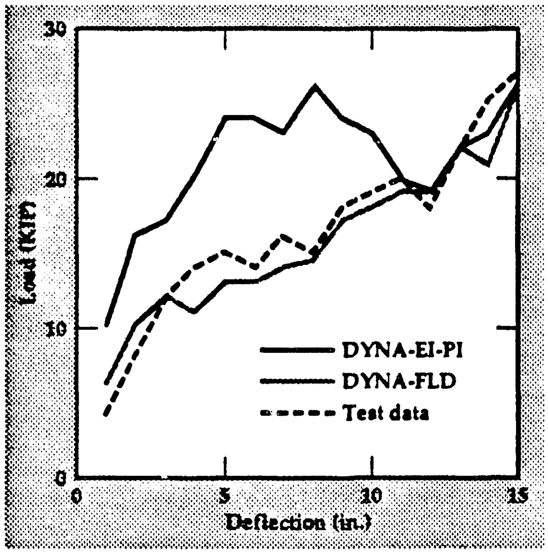
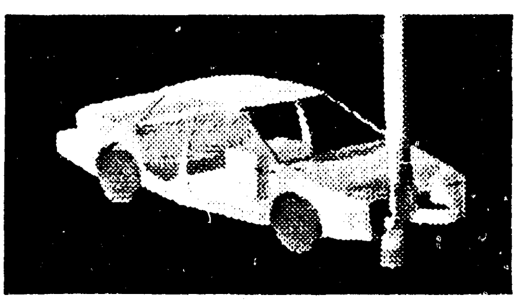


Fig. 2: Load-deflection for bogey crush into pole. Static test data matches DYNA3D if FLD failure model is used.

VEHICLE MODELS AND INTEGRATED ANALYSIS

A forerunner of analyses to follow was performed this year in a joint effort involving LLNL, the Federal Highway Administration, and University of Alaska faculty (Wekezer et al., 1993). We developed a working model of a 1991 domestic sedan. The goal of the study was to define the car in sufficient detail to capture its pre-crash, impact, and post-crash behavior, and yet keep the model simple enough that analyses could run overnight on a workstation. In the light pole impact example (Fig. 3), the car model is rigid material aft of the firewall. Underhood features are modeled as simple rigid bodies. The vehicle model consists of 20 parts, 2406 nodes (with 6 d.o.f's at each node), 10 beam-, 1575 plate- and 224 solid- elements. This is an example of problems we hope to eventually run routinely: large deformations of both vehicle and roadside feature, possible coupling to vehicle handling, in a workstation environment with overnight turnaround.



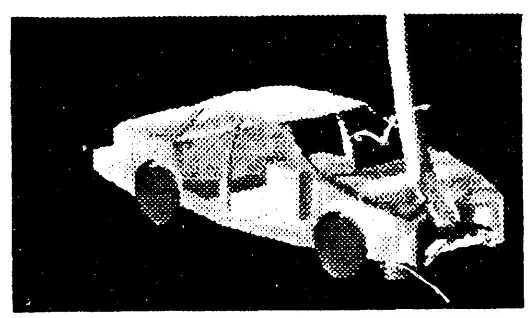


Fig. 3: Domestic sedan impacting luminaire support. Pole failure is modeled with LLNL's SAND technology.

The model above was then used in post-predictive mode to demonstrate DYNA3D's crash modeling capabilities. These analytical predictions are compared with crash test results obtained from NHTSA, where this 1991 domestic sedan was impacted against a rigid wall at a velocity of 57.5 km/h. Although all major structural components of the car were accounted for, the soft crush characteristics of the bumper area were not accounted for in the vehicle model used here and in Fig. 3. To compensate for that, a clear distance of 0.5 m between the structural bumper and the rigid wall was allowed. Fig. 4 compares DYNA3D's prediction and crash test results of the time - acceleration history of the engine block (upper) and rear seat area (lower). Given the coarse finite element mesh of the model, and the first-run non-tuned conditions, the agreement is remarkably good.

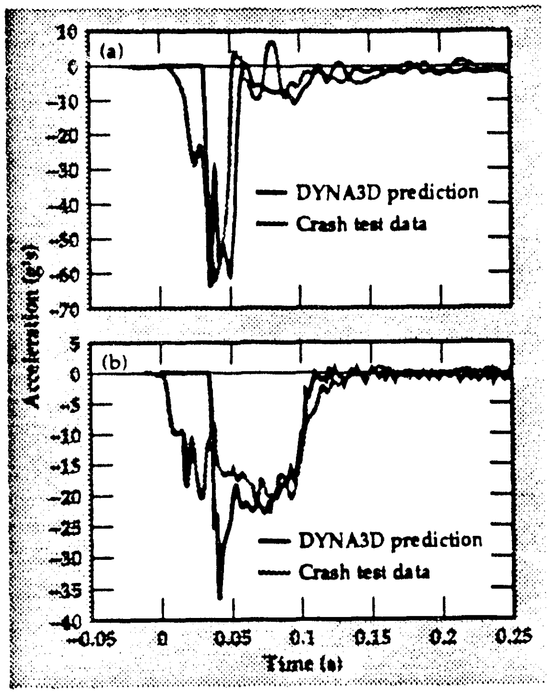
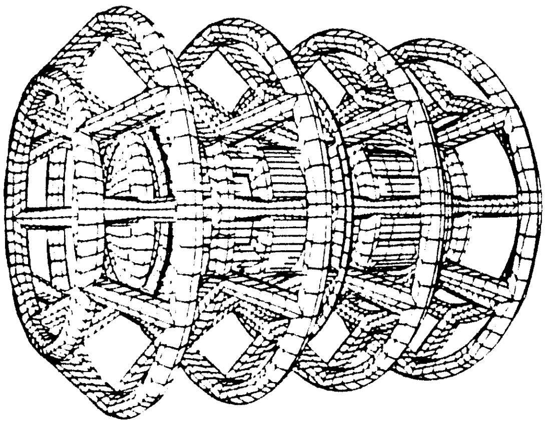


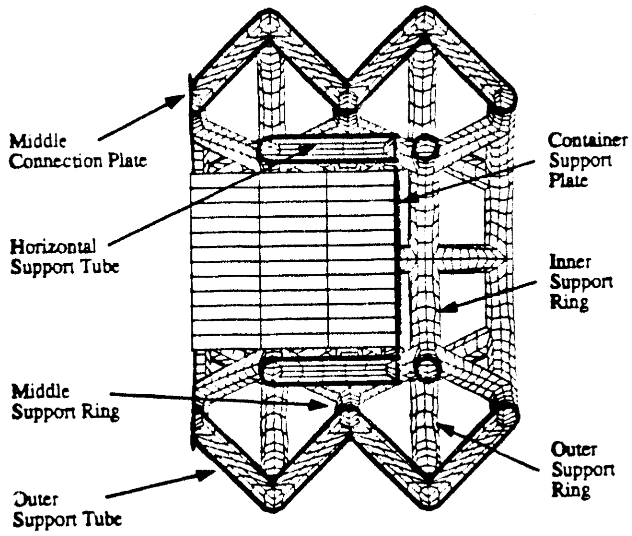
Fig. 4: Acceleration history of 1991 domestic sedan for a 57.5km/h rigid wall impact. Agreement is good for only a 2000-node vehicle.

SPACEFRAME SHIPPING CONTAINER

Impact simulations performed at LLNL are often characterized by large, nonlinear deformations of complex geometrical bodies composed of several types of material. The features and capabilities available in DYNA3D have made the code a valuable tool in determining the dynamic response of impacting bodies. These features include sliding interfaces capable of predicting relative movement and gap formation between material boundaries, and an isotropic power law plasticity material model. Sliding interfaces are useful for defining contact regions between the impacting body and its target, or regions within a body which are initially separate but come into contact as the body deforms during impact. Power law plasticity is capable of representing the post-yield stress-strain behavior of ductile materials, such as steel and aluminum. An example application of DYNA3D to predict the impact behavior of ductile bodies is an analysis performed at LLNL to determine the feasibility of using an aluminum spaceframe, impact cage to transport nuclear weapons (McMichael, 1993).

The concept of using an impact cage constructed of cylindrical tubes and rings (toroids), and circular plates [see Figure 5] was originated by G. Dittman at LLNL. The lightweight, high strength aluminum impact cage offers enhanced protection of the weapon in a combined accident environment by dissipating the impact kinetic energy through plastic deformation of the tubes and rings, by eliminating the fuel source provided by the combustible packaging materials typically used to mitigate impact, and by providing a fire-retardant coating for the weapon container to protect the weapon from a fuel fire. The impact cage was formed from two halves which could be bolted together around the middle connection plate at the center of the cage. The weapon container was supported within the impact cage by horizontal tubes connecting the inner support rings of the cage to the half-rings on the circular support plates located at each end of the cylindrical weapon container. The arrangement of the aluminum tubes and rings was intended to provide a layered crush resistance which would use the plastic deformation of the middle and outer rings and the cylindrical tubes to dissipate the impact kinetic energy and minimize damage to the weapon container. The project evaluated the structural response of the impact cage due to an impact with a rigid wall; a rigid wall in DYNA3D is an entity which represents an infinitely rigid, unyielding surface. The velocity of the cage and weapon container at impact was 76.2 m/sec.

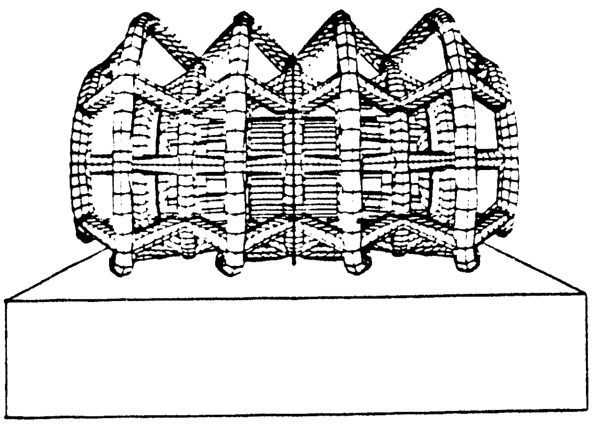

A lightweight, high strength aluminum impact cage.

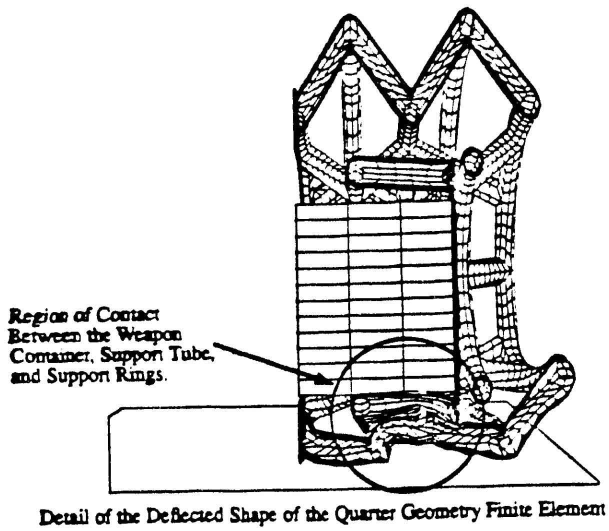


Detail of Quarter Geometry Finite Element Model.

Fig. 5: Mesh of the spaceframe shipping container, using 8-node brick elements.

Due to the mass and high impact velocity, the impact cage was required to dissipate a large amount of kinetic energy, resulting in buckling of the cylindrical tubes and large plastic deformations of the aluminum tubes and rings along the entire length of the impacting side. The deformed geometry of a model simulating the side impact of the cage is shown in Figure 6. The radial deformation of the impact cage in this scenario was over 38cm. As the impact cage deflects during a side impact, the outer cylinders connecting the second middle support ring from the end and the first and second outer support rings from the end are stiff enough to force the middle support ring to deform radially inward to contact the horizontal container support tube. The horizontal support tube is then forced by the middle support ring into contact with the weapon container. Modeling of this intricate series of material deformation and surface contact is accomplished through the use of material boundary slidelines to define the surfaces of potential contact. DYNA3D searches for contact between the defined surfaces, and implements a penalty function to prevent nodal interpenetration along the interfaces.


Deflected Shape of the Impact Cage.



Detail of the Deflected Shape of the Quarter Geometry Finite Element Model.

Fig. 6: The spaceframe shipping container design after impact simulation with DYNA3D.

The underlying concepts described in the modeling of the impact behavior of the weapon transport cage are similar to the modeling techniques which will be implemented to analyze the crash behavior of vehicles. The structural concept of the weapon transport cage is itself similar in nature to the lightweight, spaceframe vehicle design proposed for Calstart's Neighborhood Electric Vehicle. The same philosophy used to design the impact cage to transfer loads away from the weapon container is evident in the design of vehicles codes. These codes have been developed over many years, and their empirical aspects tuned against crash test data. They are useful tools, but their relative lack of physics leaves them open to technical or legal doubt when extrapolation is involved. As an alternative, LLNL's 3-D NIKE and DYNA codes could be used separately or coupled to analyze the vehicle/barrier crash interaction. Recently, the FHWA concluded that DYNA3D is the code of choice on which to form an enhanced roadside design analysis program. The incentive for improved roadside design is quite large, since about 40,000 traffic deaths occur each year in this country. As a direct consequence, there are about 40 billion dollars worth of lawsuits circulating at any given time. Often state Departments of Transportation (DOTs) are the target of these lawsuits. More than half of these fatal accidents typically involve only one vehicle. Thus, the ability to model and analyze barrier crash and post-crash motion with physics-based tools like NIKE, DYNA, and an integrated Real-Time Handling (RTH) capability could provide a strong supplemental tool for sorting out legitimate and erroneous scenarios of responsibility, in addition to its primary role as a highway design aid.

LLNL's research complements programs such as VISTA through technical efforts which are organized into four overlapping technical effort areas (Logan, 1992). These include (1) Vehicle handling and interfacing, (2) Roadside features and component modeling, (3) Vehicle models and integrated analysis, and (4) Static and dynamic test and validation. The emphasis in this paper is on topics (2) and (3) leading to a particular instance of test and validation in a collaborative effort between LLNL and Kaiser CFT.

PRECURSOR WORK IN COMPONENT MODELING

Before embarking on a 'big picture' analysis of vehicle and roadside barrier under linked handling and impact conditions, it is necessary to consider the FEM deformation analysis of smaller components. This work was begun with an analysis of a rigid bogey developed under a UC Davis / Caltrans collaboration. The bogey has a crushable steel box-structure front end resembling a coarse honeycomb structure, as modeled with DYNA3D in Fig. 1. This analysis was run at slow velocities to approximate the static crush test conducted on the actual structure. The mesh was kept coarse in the spirit of workstation level models, and the load-deflection was compared against test data as shown in Fig. 2. The first runs with DYNA3D used an elastic perfect-plastic material model. This type of material behavior gives a numerically well-posed problem which is not too dependent on the mesh size. However, the calculated load-deflection (DYNA-El-Pl line) is too stiff during early stages of the crushing process. Use of the augmented Forming Limit Diagram concept (Logan, 1993) with rate-dependent flow and failure allows a match to be achieved (DYNA-FLD) with the test data. The effectiveness of advanced material models under development at LLNL is demonstrated here for isotropic flow and failure. Related studies involve the integration of anisotropic flow and failure theories for analysis of metallic and non-metallic materials such as deep drawing steels or chopped fiber composites. The type of simulation in Fig. 1-2 is neither predictive nor post-predictive (i.e. post-mortem but first-run), but is still of value in learning the techniques and meshing required to match real tests.

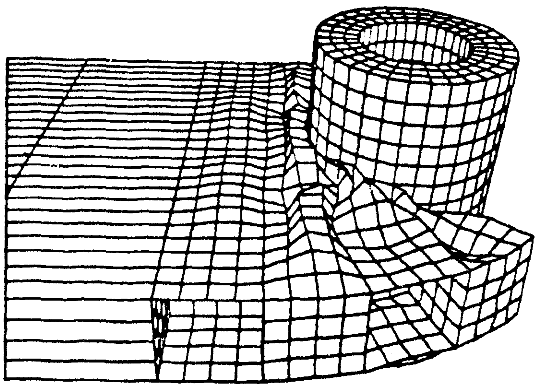


Fig. 1: Workstation level DYNA3D mesh of bogey front crush area on impacting rigid pole.

to use the frame and structural components of the vehicle body to protect the passenger compartment during a crash. The resources available at LLNL to implement the numerous material models and contact interface algorithms available in DYNA3D are a powerful asset in modeling the complex geometries and material behaviors associated with simulating the dynamic, nonlinear, large deformation response of vehicle impacts.

DESIGN AND TESTING OF AUTOMOTIVE SPACEFRAME COMPONENTS

In the first stage of preparation for the design of a spaceframe aluminum chassis for lightweight electric vehicle designs, we conducted a series of crush tests on extrusion sections, complemented by finite-element calculations of the crush behavior. The goal of this experimental/numerical study is to improve both the material behavior (strength with energy absorption) and the numerical modeling technology to allow scale-up to a full-vehicle finite-element model. Since one goal of the finite-element technology advancements is to understand the limits of coarse meshing for workstation applications, the vehicle models might well be of the 2000-3000 node level as in the previous section. Thus, ability (or inability) to capture the experimental crush is a critical issue.

The quasi static axial crush tests were carried out on a Baldwin Universal Testing Machine with a 2300 kN capacity. 51mm square tubing was extruded in a 6XXX series Kaiser alloy. By varying process parameters and heat treatment two distinct failure modes can be obtained. In each case the samples were 200mm in length prior to the test. Four different processing and heat treatment variations were performed on the extruded material and the results of the axial crush tests are shown in Figure 7.

Figure 7a is the crushed extrusion which has been completely annealed, designated 6XXXA0. The material in this case has been completely annealed and demonstrates a buckle failure mode. The crush load [Pmax] for the sample was 30 kN and would clearly not be suitable for use in automotive structural applications. Figure 7b shows the extrusion in a partially heat treated condition, 6XXXAP, where the buckling failure mode is retained yet the crush load [Max] of 58 kN is still not suitable for structural application.

Figure 7d shows the deformation of the tube, designated 6XXXAS, that has a reasonable crush load [Max] of about 120 kN, but the crush failure mode is to bulge and eventually split. Figure 8 shows the load vs. crush curve for this material (AS) where the load reaches the peak very quickly and drops off rapidly. The energy absorbed in this case is 15.6 kJ/ kg. In the application of automotive spaceframes the structure is required to withstand operational loads and have high impact strength and energy absorbing characteristics. High strength combined with high energy absorption is essential for reducing the acceleration of the passenger in a forward impact accident. The material in this case, although of high strength, would not be suitable for impact absorbing parts of the structure.

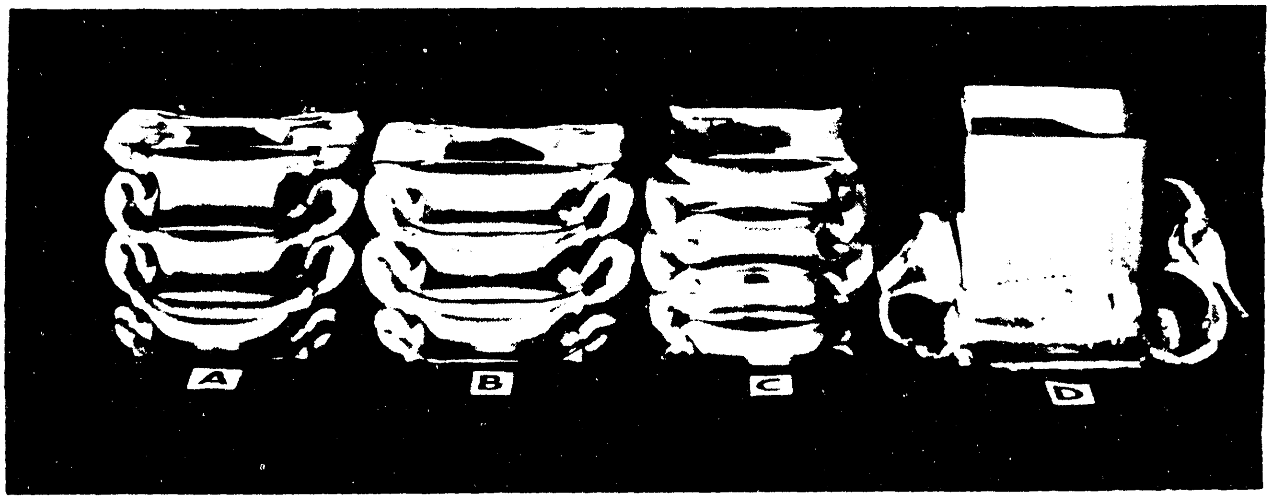


Fig. 7: Experimental crush behavior of 6XXX aluminum extrusions with different heat treats; note increased number of buckles in (c), splitting in (d).

Figure 7c shows the performance of the tube material designated 6XXXAC, which is more suitable for the energy absorbing sections of an automotive spaceframe. In this case the material processing parameters and heat treatment were changed. The crush load [Max] is similar to the first sample [120 kN], but the failure mode is changed and a succession of buckles are seen. Figure 8 shows the load vs. crush curve in this case (AC) where the total energy absorption is approximately 35 kJ / kg, more than twice the value seen in the first sample. After the initial peak, the curve tends to flatten except for the series of smaller peaks which correspond to each individual buckle. This material is more suited to the crush areas of the structure.

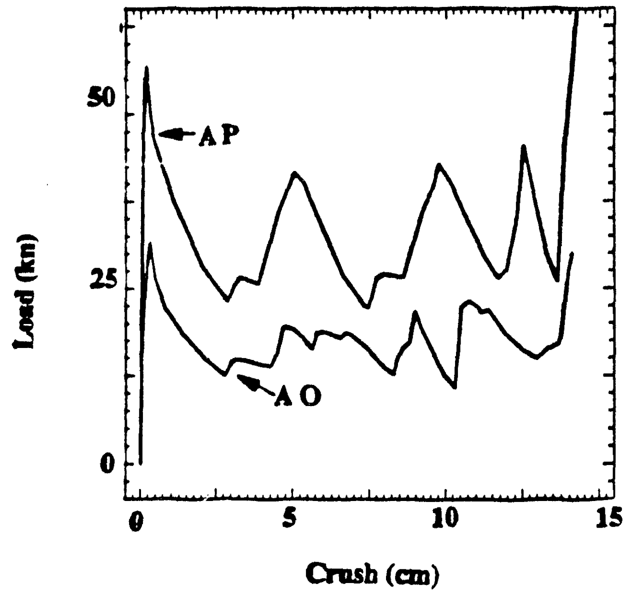


Fig. 8: Load vs. crush for aluminum extrusions. Splitting in (AS), high energy absorption in (AC).

It is preferred that an optimum alloy is used in the structure rather than several for different parts of the vehicle. In this way manufacturing costs can be maintained to a minimum. One single failure mode will also improve the accuracy of the modeling. The failure mode of buckling is desired, to improve energy absorption, and the geometric perturbations can be designed in using the DYNA analysis, seen below. In this way the average crush force can be retained but the curve will be even flatter than shown in Fig. 8.

AXIAL CRUSH MODELING

The aluminum extrusions were modeled in axial crush with DYNA3D. The goals were to determine the finite-element mesh size necessary to match the experimentally observed behavior in the quasi-static tests described above, and to match the load vs. crush behavior with stress-strain behavior as in static tension tests conducted as part of the study.

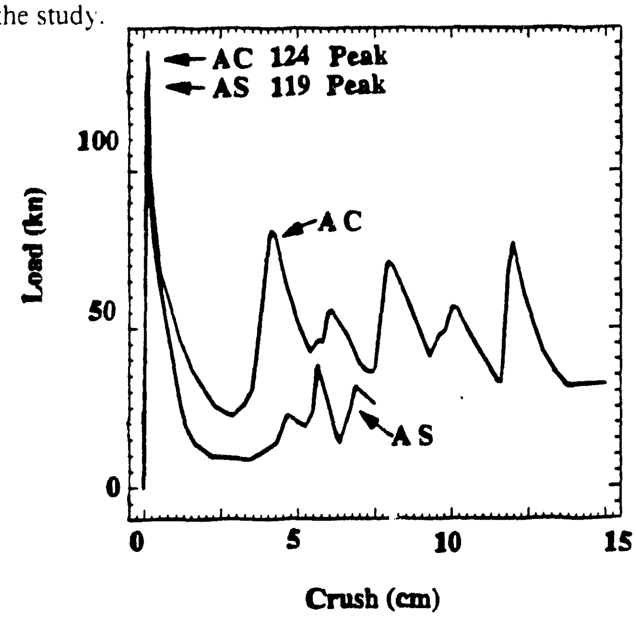


Fig. 9: Load vs. crush for axial compression of 6XXX-A0 and -AP heat treat aluminum extrusions.

Fig. 9 shows the crush force vs. distance curves for the 6XXX-A0 and -AP samples. Except for the high ratio of early peak to average crush level (which may be designed out if desired), the (A0) curve is smooth and flat except for the perturbations where each of 3 successive buckles forms during crush. However, the average crush force of about 16 kN is too low for an optimized design. There is an improved situation in the force vs. distance for the 6XXX-AP heat treat, with average crush force now up to about 31 kN, but again with a good flat curve except for buckle formation.

Fig. 10 shows a sequence of the DYNA3D analysis of the crush of the crush test of the (AP) material. The sequence of formation of the buckles (location is random unless seeded) and the number and nature of the buckles agrees well with the test.

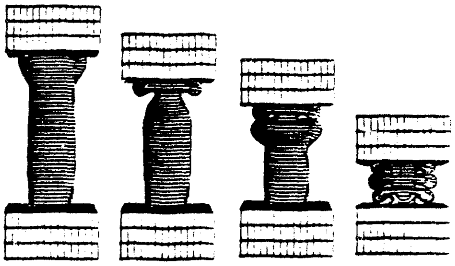


Fig. 10: DYNA3D calculation of crush of the 6XXX-AP extrusion.

In Fig11a-11c, we explore the effect of mesh size (3mm, 8mm, and 12mm squares) on the nature of the force-distance in crush. For the 3mm and 8mm meshes, we achieve a good estimate of crush force, with some differences in the perturbations due to different patterns in predicted buckle formation.

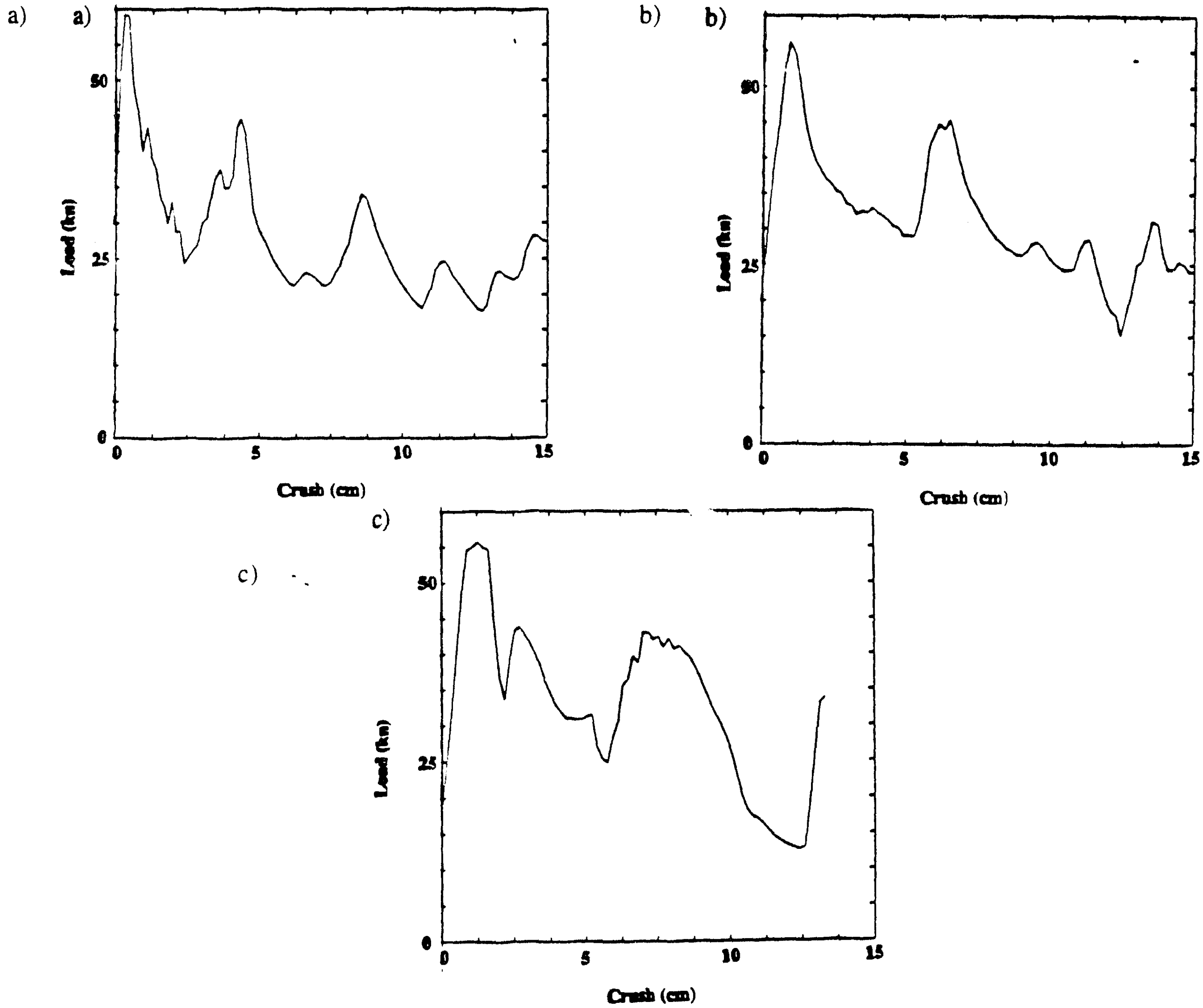


Fig. 11: DYNA3D calculation of load vs. crush of the 6XXX-AP extrusion. Mesh size is (a) 3mm squares, (b) 8mm, (c) 12mm.

The 12mm mesh, although peak force is good, is of marginal use for design of the section. It may be usable in a coarse-mesh vehicle model even without tuning of load vs. crush, but it is near the limit of realistic modeling results to capture the onset and progression of buckles. However, since there is a factor of approximately 4**3 or 64x in CPU time from 3mm to 12mm squares, it is important to find this optimum mesh for extension to full vehicle studies. Mesh sizes of 12mm (or even 24mm) are about the minimum size that can be used throughout a full vehicle model and still retain a workstation-level capability. This is because of the increase in degrees of freedom with the simultaneous decrease in time step size due to the smaller elements.

Fig. 12 provides a good illustration of the difficulties in capturing buckling behavior as the mesh is coarsened. On the left is the starting mesh (with 4-fold symmetry) for crush of a 6XXX-AP tube, and on the right is the crushed geometry with 8mm elements. The buckles are forming correctly, but it is clear that we are approaching the coarsest mesh within reason to capture the full buckling during crush. Beyond this level, pseudo-methods may be more effective in a full vehicle model. However, even at this level, Fig. 13 shows that once again a good approximation of the force-displacement curve in crush is achieved. For the AP material, a full piecewise plasticity model is needed to capture the high post-yield peak as the first buckle forms.

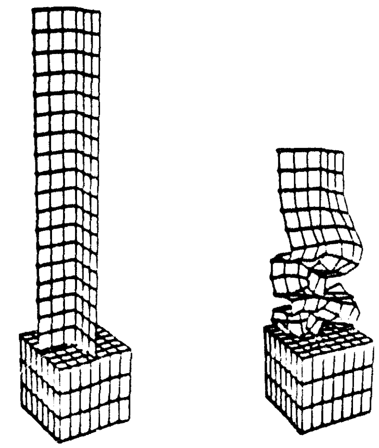


Fig. 12: Calculated crush of 6XXX-A0 tube using mesh size of 8mm ; near upper limit.

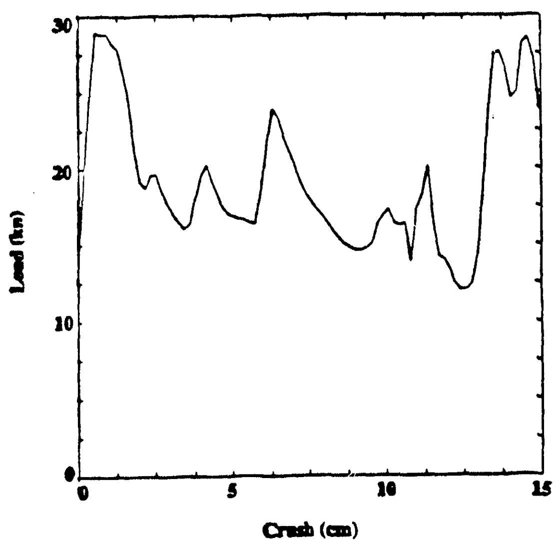


Fig. 13: DYNA3D calculation of load vs. crush of the 6XXX-A0 extrusion. Mesh size is 8mm.

SUMMARY AND FUTURE WORK

We have demonstrated the effectiveness of integrated vehicle/barrier impact analysis at the workstation level, and identified the needs for additions and refinements leading toward a complete package. Goals for the future in LLNL's internal research will again focus around the four technology areas established. We will work toward full linkage of vehicle handling (and occupant dynamics) to NIKE and DYNA, and development of compatible tire models for all the codes. Study of both roadside and vehicle structural sections will continue at the component level to assure that model simplification is efficient yet accurate compared to more refined meshes. A more complete suite of vehicle models and roadside hardware will be developed as part of the VISTA program. These will make use of material model developments for improved flow and crush of aluminum and fiber composite materials (anisotropy, forming limit, composite damage). These will be used in future lightweight designs such as Calstart's Neighborhood Electric Vehicle (NEV). LLNL and Kaiser anticipate the furthering of this cooperative study as part of the Calstart consortium. Improved understanding of aluminum spaceframe design, energy absorption, and manufacturing economics will benefit concurrent LLNL programs such as the shipping container design, while leading to an enhanced ability for materials suppliers such as Kaiser to help vehicle manufacturers design and manufacture for crashworthiness with less tooling for prototype extrusions and vehicles.

ACKNOWLEDGMENTS

The authors wish to acknowledge the contributions of Prof. A. Frank at UC Davis (Figs. 1-2), and Prof. J. Wekezer of U. Alaska on the domestic sedan model (Fig. 3-4), G. Dittman of LLNL for container design (Fig. 5-6), and FHWA and NHTSA for their collaborative support.

This work was performed under the auspices of the U. S. Department of Energy by the Lawrence Livermore National Laboratory under contract W-7405-Eng-48.

REFERENCES

Logan, R.W., "Tire, Accident, Handling, and Roadway Safety," *Engineering Research and Development Thrust Area Report FY92,*, ed. S.Y. Lu, University of California, Lawrence Livermore National Laboratory, UCRL-53868-92 (1992).

Logan, R.W., *Implementation of a Pressure and Rate Dependent Forming-Limit Diagram Model Into NIKE and DYNA*, University of California, Lawrence Livermore National Laboratory, UCRL-ID-105760 (1993).

Maker, B.N., "NIKE3D, A Nonlinear, Implicit, Three-Dimensional Finite Element Code for Solid and Structural Mechanics - User's Manual", UCRL-MA-105268, (1991).

McMichael, L.D., "Evaluation of a Space Frame Impact Cage for Weapon Transport," University of California, Lawrence Livermore National Laboratory, UCRL-ID-113094, (1993).

Wekezer, J.W., Oskard, M.S., Logan, R.W., and Zywicz, E., "Vehicle Impact Simulation", J. Transp. Engr., in press (1993).

Whirley, R.G., "DYNA3D, A Nonlinear, Explicit, Three-Dimensional Finite Element Code for Solid and Structural Mechanics - User's Manual", UCRL-MA-107254, (1991).

DATE
FILMED
12/27/93

END