NASA Technical Memorandum 102595

(NASA-TM-102595) A REVIEW OF THE ANALYTICAL SIMULATION OF AIRCRAFT CRASH DYNAMICS (NASA) 25 p CSCL 01C

N90-20068

Unclas
G3/03 0272350

A Review of the Analytical Simulation of Aircraft Crash Dynamics

Edwin L. Fasanella, Huey D. Carden,
Richard L. Boitnott, and Robert J. Hayduk

January 1990

NASA
National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23665-5225

A REVIEW OF THE ANALYTICAL SIMULATION OF AIRCRAFT CRASH DYNAMICS

by

Edwin L. Fasanella *, Huey D. Carden **, Richard L. Boitnott ***

and,

Robert J. Hayduk **

*Lockheed Engineering and Sciences Company

**NASA Langley Research Center

***U. S. Army Aviation Research and Technology Activity

Hampton, VA

SUMMARY

The National Aeronautics and Space Administration has been conducting extensive research on the crash dynamics of aircraft for more than fifteen years. A large number of full-scale tests of nearly-complete general aviation aircraft, helicopters, and one unique air-to-ground controlled-impact of a transport aircraft have been performed. Additionally, research has also been conducted on seat dynamic performance, load-limiting seats, restraint systems, load-limiting subfloor designs, and emergency-locator-transmitters (ELTs). Computer programs have been developed to provide designers with methods for predicting accelerations, velocities, and displacements of collapsing structure - airframe or seat - and for estimating the human response to crash loads. The results of full-scale aircraft and component tests have been used to verify and guide the development of analytical simulation tools and to demonstrate impact load attenuating concepts. Several simplified empirical-analytical techniques have been developed by NASA and others to approximate the crash dynamics of aircraft.

In recent years the trend of NASA's research in aircraft crash dynamics has been from metal aircraft structures to composites. It is well recognized that composites will continue to find more extensive applications in future structures. Research on aspects of composites has been on-going for several years, but only recently has attention been given to the crash-dynamic response and characteristics of composite structures. Research has been conducted by NASA and the U. S. Army into the abrasion behavior of composite aircraft skins, the impact response of composite fuselage frames, and scaling of impact-loaded carbon fiber composites. Techniques have been developed for analyzing cylindrical composite panels with internal pressure loading and for predicting the performance of composite subfloors.

This paper will address analytical simulation of metal and composite aircraft crash dynamics. Finite element models are examined to determine their degree of corroboration by experimental data and to reveal deficiencies requiring further development. This status report will indicate areas for future research in aircraft crash dynamics.

INTRODUCTION

The National Aeronautics and Space Administration has been conducting extensive research on the crash dynamics of aircraft for more than fifteen years (Ref. 1). A large number of full-scale tests of nearly-complete general aviation aircraft, helicopters, and one unique air-to-ground controlled-impact of a transport aircraft have been performed. Additionally, research has also been conducted on seat dynamic performance, load-limiting seats, restraint systems, load-limiting subfloor designs, and emergency-locator-transmitters (ELTs). Computer programs have been developed to provide designers with methods for predicting accelerations, velocities, and displacements of collapsing structure - airframe or seat - and for estimating the human response to crash loads. The results of full-scale aircraft and component tests have been used to verify and guide the development of analytical simulation tools and to demonstrate load attenuating concepts (Ref. 2). Several simplified empirical-analytical techniques have been developed by NASA and others to approximate the crash dynamics of aircraft (Ref. 3).

In recent years the trend of NASA's research in aircraft crash dynamics has been from metal aircraft structures to composites (Ref. 4). It is recognized that composites will continue to find more extensive applications in structures of the future. Research on composites has been on-going for several years, but only recently has attention been given to the crash-dynamic response and characteristics of composite structures. Research has been conducted by NASA and the Army into the abrasion behavior of composite aircraft skins (Ref. 5), the impact response of composite fuselage frames (Ref. 6), and scaling of impact-loaded carbon fiber composites (Ref. 7). Techniques have been developed for analyzing cylindrical composite panels with internal pressure loading (Ref. 8) and for predicting the performance of composite subfloors components (Ref. 9).

This paper will review the status of analytical simulation of metal and composite aircraft crash dynamics at the NASA Langley Research Center. This status report will indicate areas for future research in aircraft crash dynamics.

DYCAST

DYCAST (DYnamic Crash Analysis of STructures) is a nonlinear structural dynamic finite element computer code developed by Grumman with principal support from NASA and FAA (Ref. 10). The basic element library consists of the following elements: (1) stringers or rod elements with axial stiffness only; (2) three-dimensional beam elements with 12 fixed cross-sectional shapes typical of aircraft structures with axial, torsional, two shear, and two bending stiffnesses; (3) isotropic and orthotropic membrane skin triangles with in-plane normal and shear stiffnesses; (4) isotropic plate bending triangles with membrane and out-of-plane bending stiffnesses; and, (5) nonlinear translational or rotational spring elements that provide stiffness with user-specified force-displacement or moment-rotation tables (piecewise linear). The spring element can be either elastic or dissipative and is useful to model the crush behavior of components for which data are available and whose behavior may be too complex (or too time consuming) to model otherwise. It may be used to model energy absorbing devices, gap elements with variable contact and rebound characteristics, and contact elements with friction that describe contact/rebound between structure and a arbitrary plane (with or without friction).

Changing stiffnesses in the structure are accounted for by plasticity (material nonlinearity) and very large deflections (geometric nonlinearities). Material nonlinearities are accommodated by one of three options: (1) elastic-perfectly plastic, (2) elastic-linear hardening plastic, or (3) elastic-nonlinear hardening plastic of the Ramberg-Osgood type. The second option has been used exclusively for this modeling effort. Geometric nonlinearities are handled in an updated Lagrangian formulation be reforming the structure into its deformed shape after small time increments while accumulating deformation, strains, and forces. The nonlinearities due to combined loadings (such as beam-column effects) are maintained, and stiffness variation due to structural failures are computed. The failure option is imposed automatically whenever a material failure strain criterion is met, or manually by the user at a restart.

Other features include: multiple time-load history tables to subject the structure to time dependent loading; gravity loading; initial pitch, roll, yaw, and translation of the structural model with respect to the global system; a bandwidth optimizer as a pre-processor; and deformed plots and graphics as post-processors.

Numerical time integrators available are fixed-step central difference, modified Adams, Newmark-beta, and Wilson-theta. The last three have a variable time step capability, which is controlled internally be a solution convergence error measure. Thus, the size of the time step is increased and decreased as required during the simulation. The Newmark-beta time integrator was used exclusively for the models presented in this paper.

APPLICATIONS OF DYCAST

The DYCAST nonlinear finite element computer code has been used in a series of progressively more difficult modeling tasks with the goal of accurately modeling complete transport aircraft crashes such as the Controlled Impact Demonstration (CID). Single aircraft frames and fuselage section vertical drop tests were modeled and analyzed to obtain comparisons with experimental data and to develop hybrid element crush springs for use in the large CID model. Modifications to DYCAST as the research progressed (including a progressive failure criteria) were made during the study.

Controlled Impact Demonstration

The aircraft used in the CID was a Boeing 720, four-engine, intermediate range, jet transport (see Fig. 1) that had entered FAA service in the mid-1960's and was ready for retirement. Even though the Boeing 720 is now considered obsolete, its structural design and construction are still representative of narrow-body transport aircraft currently in use. In preparation for the CID test, several fuselage sections were drop tested. Because of the difficulty in locating a Boeing 720, nearly identical (structurally) 707 fuselage sections were used for test specimens.

As previously mentioned, the aircraft used in the CID was a Boeing 720 jet transport that was retired from FAA service. A plan view of the CID is shown in figure 2 with longitudinal Body Station (BS) locations labeled and compared to the distance (in inches) from the nose of the aircraft (x- coordinate). A study by NASA, the FAA, and three major U. S. airframe manufacturers was conducted to determine a typical "impact survivable" accident. In addition, the FAA required fuel spillage and ignition sources to test the antimisting kerosene fuel (AMK).

The planned CID scenario resulting from these studies is illustrated in figure 3. The Boeing 720 was to follow a 3.3 to 4.0 degree glide slope in a 1 degree nose-up attitude. The aircraft was to have a 17 ft/s sink rate and a longitudinal velocity of approximately 150 knots. After the primary impact, the airplane fuselage was to slide between a corridor of wing openers to cut the wing tanks and insure spillage of 20 to 100 gallons of AMK per second. The structural crashworthy experiment would be completed before the airplane contacted the wing openers.

Although a symmetric impact was planned, in the actual impact the outboard number 1 engine on the left wing contacted the ground first as a result of a 13 degree roll and yaw, and zero degree pitch conditions for the airplane (see Fig. 4). Gross weight of the airplane at impact was estimated to be 192,000 pounds. At the time of wing contact, the airplane CG sink rate was approximately 17 ft/s and the horizontal velocity was 151.5 knots. The impact and drag force on the wing caused the airplane to develop an angular velocity about the pitch axis. Approximately 0.5 second after the left wing impacted the ground, the forward fuselage (nose) impacted the ground behind the nose gear wheel well. The pitch attitude of the airplane at nose impact was about -2.5 degrees (nose down).

The left wing impact and wing crush significantly influenced the crash dynamics of CID and subsequent fire damage obliterated the crush at the rear of the airplane. From an analysis of integrated vertical acceleration traces, it is estimated that the CG velocity was reduced from 17 ft/s at left wing contact to about 12 ft/s at nose impact. The angular pitch rate at nose impact measured about the CG was about -6 degrees per second (-0.1 rad/s). Highest vertical velocities were forward of the CG (18 ft/s near the nose gear contact point) and the lowest were located behind the CG.

Modeling Considerations

Various structural failure mechanisms must be accommodated analytically for accurate modeling of the structural behavior. For example, the lower lobe of the fuselage section resists vertical loading through deformation of the frames, whereas the longitudinal stringers and skin offer little resistance to crush. The lower frames could be expected to fail in bending and/or in shear and to potentially develop points of inflection and "snap through" due to the ground reaction forces. The ground reaction might also impose high transverse shear loads on the frame cross sections. In addition, plastic hinges might develop in the frames between the floor level and the fuselage bottom. If the frames do not rupture while undergoing these types of deformations, large impulsive moments would be applied to the floor and the upper frame. Thus the analytical formulation needs to provide for many basic failure mechanisms.

For the current level of crash analysis development and computer resources, it is necessary to judiciously limit the number of degrees of freedom (nodes) and the number of structural elements in the crash model. As the degrees of freedom increase, model debugging, verification of the dynamic behavior, and interpretation of the results become increasingly difficult. Consequently, it is desirable to understand the behavior of less complicated components prior to formulation of the complete structural model.

Transport Fuselage Sections

A two-frame model with sufficient detail to model the floor, two seats with lumped mass occupants, and the fuselage structure without using nonlinear springs (except for the seat and ground properties) was formulated at the Langley Research Center for research purposes. Although symmetry was lacking for the full section, the two forward frames and seats did exhibit a vertical plane of symmetry. Thus a symmetric, two-frame, half model was used for computations on the section.

The finite element two-frame model is shown schematically in figure 5. Stiff ground springs simulated the concrete pad impact surface. Each frame of the lower fuselage below the floor was modeled using eight beam elements. Floor and seat rails were modeled using appropriate beam elements. Fuselage structure above the floor (not expected to deform plastically) was modeled in less detail to keep the model as small as possible. The triple seat/occupant model consisted of 4 lumped masses connected by horizontal stringer elements supported by 4 nonlinear springs representing the vertical legs. Mass of the three occupants was distributed using a 2 to 1 ratio with inboard seat legs supporting two occupants and outboard legs supporting only one occupant due to asymmetry of the seat pan with respect to the legs.

To simulate end constraints and strengthen the section, motion was not allowed in the fore-and-aft (x-axis) direction. Initially, the time step was allowed to vary, but was later held constant to

250 microseconds to correspond to the sample rate (4000 per second) used to digitize the experimental accelerations. Consequently, the same digital low pass filter used to filter the experimental data could be used to filter the DYCAST calculated accelerations without requiring an interpolation algorithm before filtering. The 250 microsecond time step was conservative for this problem compared to a minimum time step of 500 microseconds when a variable time step was allowed in an earlier run. To run 901 constant time increments (.225 sec total) required 1620 CPU seconds on a CDC Cyber 175 with a maximum field length of 303K.

Figure 6 illustrates the structural behavior/damage experienced by the fuselage section during the vertical impact test at 20 ft/s. The gross structural damage to the fuselage was primarily confined to the lower fuselage below the floor level. All seven frames ruptured near the bottom contact point. Plastic hinges formed in each frame along both sides of the fuselage. Crushing measured from floor level varied from 22-23 inches for the forward end to 18-19 inches for the rear.

Nonlinear material properties used for the critical subfloor aluminum frame beam elements were elastic-plastic with a small amount of linear strain hardening. The yield stress initially chosen was 83,000 psi with a failure strain of 11 percent. However, the DYCAST model was too strong using this yield stress and failure strain and did not predict sufficient crush. When using elements to represent macro-sections of structure, the elongation to failure is difficult to determine due to stress risers, section changes, etc. Although material coupon tests may show 11 - 13 percent elongation, failure generally occurs much earlier when material is fabricated into large structure. In addition, data from tests of large panels of aircraft structure indicate that yield is typically in the 40-50,000 psi range.

As a consequence, the aluminum beam yield stress in the DYCAST model was reduced to 50,000 psi and failure strain was set to 5 percent. Effects of the beam failure strain criteria in DYCAST on the two-frame model are shown in figure 7 for 0.23 seconds after impact. The beam failure criteria for the two-frame model was conservative since the entire beam failed when one Gaussian integration point reached the specified maximum normal strain. With a failure strain of 5 percent, beams at the bottom of the fuselage failed too soon and excessive deformation of the structure was predicted. To overcome the early failure, the maximum strain was increased to 8 per cent for better simulation. As a result of this problem, DYCAST was later modified to include a partial beam failure criteria. Partial element failure is obtained when the specified failure strain is reached locally. The material stiffness corresponding to that integration point is set equal to zero. An element is deleted only when all integration points reach the specified failure strain.

For a yield stress of 50,000 psi and failure strain of 8 percent, good correlation with experiment was achieved; thus these values were used in the two-frame model for all comparisons with experimental data. Figure 8 presents a comparison of vertical experimental and analytical floor accelerations at the wall-floor intersection.

Although a full section model using only finite-elements for the subfloor was desirable, constraints of time and computer resources limited the finite element subfloor model of the forward section to a two-frame model. Comparisons of experimental data with results from the two-frame model indicate that DYCAST can provide excellent correlation with impact behavior of fuselage structure with a minimum of empirical force-deflection data representing structure in the analytical model.

Modeling the CID

Since the CID impact was planned to be symmetric, a half-model of the Boeing 720 was developed. In addition, the wing and the forward cabin models were not as well defined as the rear of the fuselage where initial impact was expected. After the CID test, the model was modified and additional crush springs were added in the forward cabin.

Fuselage contact occurred 0.46 seconds after left wing (engine no. 1) contact. Model predictions begin at fuselage contact with the ground and continue for 0.15 seconds. Initial conditions used in the DYCAST analysis were 12 ft/s vertical velocity at the CG, -0.1 rad/s pitch rate, and an initial pitch of -2.5 degrees at the time of fuselage impact. Yaw and roll conditions were ignored for this analysis. To introduce the pitch rate into DYCAST, the initial velocity of each node was computed using $v = v_{cg} + (w \times R)$ where w is the pitch rate in radians per second, v_{cg} is the velocity vector of the center of gravity, and R is the vector from the CG to the node.

The CID DYCAST model at time zero (initial fuselage contact) is shown in figure 9. The fuselage bottom initial ground contact point at BS 488, behind the rear nose gear bulkhead, was positioned just above the ground. The model consists of 126 beams, 73 structural crush springs, 15 ground springs, 113 concentrated masses, 196 independent degrees of freedom and 68 dependent degrees of freedom (see Fig. 10). External forces acting on the model are gravity, friction (coefficient of 0.4 was used), and time dependent lift and wing crush forces. A vertical wing crush force was needed due to the inadequacy of the simple wing model to predict progressive rupture of the wing structure. The wing was modeled using eight tapered offset beams as shown in figure 10. A typical fuselage cross section is also shown in figure 10. Six offset beam elements constrained to the motion of a reference element on the plane of symmetry at floor level represent the fuselage bending properties. The keel is represented by a beam in the center section and as springs in the remainder of the aircraft. The 24 diagonal springs represent the shear resistance of the fuselage skin. Vertical springs were used to model the lower fuselage crush properties developed from analysis of single frames and section tests.

For the model to run 0.15 seconds in real time required 1641 seconds on a CYBER 175. (A constant time step of 0.0005 seconds was used.) With the airplane pitched as shown, BS 960 at the rear main landing gear bulkhead is about 27 inches off the ground. The time sequence of ground contact was: BS 488 at time zero; BS 388 at 0.01 seconds; BS 302 and BS 568 at 0.035 seconds; BS 600J at 0.075 seconds; and BS 620 at 0.100 seconds. By 0.15 seconds, most of the aircraft vertical energy was absorbed. In the experimental data, BS 960 makes ground contact about 0.4 seconds after the initial ground contact. However, the vertical acceleration at the floor level was only 2 or 3 G's.

Figure 11 shows analytical DYCAST and experimental CID vertical floor peak accelerations as a function of the fuselage x-coordinates. The x-axis origin was located at the aircraft nose (see Fig. 2 for the relation of the x-coordinates to the Body Stations). Both experimental accelerations and DYCAST predictions have been filtered with a 100 Hz digital filter. The comparisons are considered good for the symmetric model used with roll and yaw neglected. Figures 12 compares CID experimental and DYCAST acceleration time histories for the floor location at BS 207 (near the pilot). DYCAST overpredicts the peak accelerations, but the basic waveform of the acceleration matches the experimental acceleration pattern well.

Figure 13 shows a comparison of measured fuselage crush (post test) versus DYCAST predicted crush for forward fuselage locations. The wing cutters ripped out the center section keel beam and post crash fire destroyed the aft end of the aircraft; thus measurements past the center of the aircraft can not be assessed for initial impact. Predictions of crush and acceleration levels from a symmetric CID model agreed well with data from the CID experiment. The building-block analysis approach of using results from detailed models of substructure to form hybrid elements for input to more complex structures (i.e., full airplane models) was successfully used to limit the size of the model. The good comparisons achieved with the DYCAST model of the CID and experimental data indicate the validity of this approach as a useful method for assessing crash dynamics of large transport aircraft.

COMPOSITE FUSELAGE FRAMES

As mentioned earlier, a transition to research on composite structures has been made. As part of the research efforts to generate a data base and achieve an understanding of the response and failure processes in composite structures under impact loads, generic composite structures are being tested. Basic composite structural elements (ie. beams, typical aircraft fuselage frames, and substructures) are being studied both experimentally and analytically to help achieve the desired goals. Composite fuselage frames, as shown in figure 14, have been tested as part of the database and analyzed with DYCAST to assess the capability and deficiency of the current code.

EXPERIMENTS

Test Specimens

Figure 14 shows a photograph of an entire six-foot diameter circular frame. A close-up view of the Z-shaped frame is shown in Figure 15. Dimensions of the frame's cross section are also shown in figure 15 as well as a photograph of the splice plates used to join four 90 degree segments into a complete circular frame. These frames were fabricated from a prepreg of five harness satin weave graphite fabric in a Hercules epoxy matrix designated as 280-5H/3502. The prepregs were draped over a layup tool into a quasi-isotropic layup.

Cross sectional dimensions of the frames are typical of designs often proposed for composite fuselage frames. The six-foot inside diameter of the frames was chosen to reduce test specimen costs and to facilitate testing. A complete graphite-epoxy frame weighed approximately 7.2 pounds without added instrumentation. Addition of strain gages, instrumentation leads, and metallic tape to secure the strain gage leads increased specimen mass to approximately 8.0 pounds. On the first graphite-epoxy frame test a 3/16 inch steel cable was attached across the horizontal diameter of the frame to represent the constraint of a floor on the lateral expansion of an impacting frame. In the second and third composite frame tests a 1-inch diameter by 0.058-inch wall-thickness aluminum tube was used to represent this constraint more accurately. The mass on the specimen was increased in tests 4 and 5 by using a steel bar (3/4" X 6") to represent the floor.

The test conditions for each of the six frame tests are given in figure 16. Frames 1 and 2 were oriented so impact occurred at a splice plate and frames 3, 4, and 5 were rotated 45 degrees so impact occurred between splice plates. A static test was performed on frame 6.

In the first three composite frame tests ten pound masses were attached to the left and right sides of the frame at the frame-floor intersection to represent structural and/or seat/occupant loads on the frame. Two 5 lb masses, on opposite sides of the web were connected to the frame to place the center of gravity in the plane of the frame web.

In the fourth composite frame drop test the applied mass was increased to 100 lbs by replacing the lightweight aluminum bar with an 80 lb rectangular steel bar. This bar measuring 0.75 inch wide by 6 inch deep and slightly less than 72 inches long was rigidly attached to the tube attachment plate to prevent rotation. In the fifth composite frame test, occupant mass was simulated with a 6 inch by 0.75 inch bar approximately 79 inches long attached directly to the frame. In the fourth and fifth frame tests the mass was attached to the frame such that the centroids of the mass and frame were in approximately the same plane.

Drop Tower

The drop tower used for the frame tests is shown in figure 17. Four 18-foot long angle sections make up the main structure of the drop tower which guide the frame during its free fall. The drop tower has two 6-foot by 7-foot fences to provide lateral support to the frame during impact and to constrain deformations to the plane of the frame. The fences simulate the constraint of stringers and skin as connected to a frame in an actual fuselage which reduce twisting and out-of-plane bending. The front fence is transparent plexiglas and the rear fence is metal. The transparent fence allows photographic coverage of the frame against a grid of lines on the rear fence. Grid line spacing is three inches. Front and rear fence separation is 2.50 inches which allowed sufficient clearance for the 2.25-inch wide frame to fall unobstructed. Heavy angle beams stiffen and stabilize the front and rear fences.

TEST RESULTS

The first frame with a steel cable representing the floor failed about 15 degrees from the impact point and the separated ends of the frame slid relative to each other and overlapped about 18 inches. In the next two composite frame tests one-inch diameter aluminum tube replaced the steel cable representing the floor. The tube prevented the two ends of the failed frame from overlapping after failure. The second graphite-epoxy frame impacted at 27.5 fps. This frame suffered a complete cross sectional failure similar to the first frame at the same location. The repeatability of the failures of the two frames is illustrated in figure 18. A two inches long tear of the inside flange of the second frame was probably caused by the two fractured ends being forced together. This tearing was not observed in the first drop test. This second frame showed no evidence of additional partial failure as was observed in the first test.

Since the upper halves of the first three specimens did not have any visual damage and the strains measured in the upper halves of the frames were very low, generally below 0.002; the fourth specimen was constructed from two upper halves from the previously tested specimens. A fifth specimen was obtained from the upper half of specimen 4 since the upper halves appeared to have little influence on the frame response. The attached mass in test 5 was 93 pounds. Figure 19 shows the fifth specimen after impacting the concrete floor with a 20 fps velocity. This specimen had three localized major fractures. Although the initial failure was slightly off center, subsequent failures to the left and right sides of the frame were symmetric. The individual fractures in frame 5 were similar to the fractures of frame 4.

Analysis

DYCAST Model

The nonlinear finite element structural analysis computer program DYCAST was used to model the composite frame. DYCAST models were constructed for both static and dynamic analysis. The DYCAST models used straight Z cross-section beam elements to model the circular frame. Each element for the static model represented approximately a 2.5 degree segment of the circular frame. Only half of the frame was modeled to take advantage of symmetry. The half frame model had 34 beam finite elements. To conserve computer resources and improve turn-around, a less refined model was developed for dynamic analysis with only 18 elements to represent the frame below the loading bar representing the floor. Static analyses showed that the less refined model predictions compared closely with the DYCAST model with elements every 2.5 degrees. Constraints were applied to impose planar vertical motion without twisting of the beam elements. Isotropic properties were used in the beam model of the frame. A maximum strain partial failure criteria, recently made available in the program, was used to specify when failure occurred. DYCAST accounts for partial failure of elements by monitoring the tensile and compressive strain magnitude at Gaussian integration points in the beam cross section. When the strain at an integration point exceeds the failure strain, the material properties (Young's modulus and shear modulus) at that integration point are set to zero. The element can thus carry partial load until the strain at all integration points exceed the failure strain.

RESULTS

Analysis and Experimental Correlation

DYCAST modeling was primarily performed for composite frame tests four and five which had loadings of 100 and 93 pounds, respectively, and impact velocity of 20 fps. The Newmark-beta time integrator was used in DYCAST with an initial time step of .00005 seconds. For dynamic analysis, the failure strain was found to be quite critical in the modeling effort. Figure 20 shows the circular frame experimental velocity measured on the horizontal bar that represents floor level. Although accelerations were the primary measurements used for comparisons of dynamic analysis with experiment, velocity was also used since velocity requires no elaborate filtering.

Experimentally, velocity was obtained from the integration of the accelerometer traces and also from high speed motion picture analysis as a check. DYCAST model predicted velocities are shown (fig. 20) for failure strains of 4000, 5500, and 8000 microinches. As can be seen, failure strain has a significant influence in the model predictions. The value of 5500 microinches failure strain was chosen for the final DYCAST model since it predicted velocity change for the first 25 msec which best matched the experimental velocity. In addition this value was close to the experimentally measured peak strains in the vicinity of failures. Figure 21 shows a comparison of the experimental floor level acceleration with two DYCAST predictions using a failure strain of 5500 microinches. All three traces were digitally filtered with the same 60 Hz low pass filter. Bending out of the frame's centroid plane and twisting were constrained in one DYCAST approximation (In-Plane). In the other DYCAST model (Free) these deformations were not constrained. The "Free" DYCAST analysis agreed best with the experimental values. The DYCAST analysis with deformations prescribed in-plane is much too stiff and over predicts the response. Although the experiment was designed to keep deformations in-plane, a 1/4" clearance gap between the specimen and the plexiglas wall allowed the frame to initially twist and bend out of plane. Once the frame was resting against the wall the twisting and out-of-plane deformations were prevented.

FUTURE DEVELOPMENTS

New DYCAST Capabilities

In previous sections of the paper applications of DYCAST have been primarily to metallic structures for which the program was initially developed. The modelling effort with the six-foot diameter graphite-epoxy circular frame was aimed at an initial evaluation of the program's capability to handle composite structures. Although some encouraging success has been indicated for the composite applications, it is clear that a need exists for efficient, new analysis capabilities for crash loading conditions on composite structures. Two initiatives, a grant to George Washington University (GWU), and a task assignment contract are discussed below for developing improved computational capabilities for analysis of dynamic responses of composite structures.

COMPOSITE STRUCTURES ANALYSIS DEVELOPMENT

A task assignment contract through the NASA Computational Structural Mechanics (CSM) program is underway to add a composite plate element and a curved composite beam element to the DYCAST library. The composite curved beam element is to be developed and implemented into DYCAST through close cooperation between GWU grantees and the contractor under the task assignment contract.

A composite plate element is to be implemented in DYCAST specifically for composite structures analysis. During the course of the initial development of DYCAST, two plate elements were included for elasto-plastic analysis: a very accurate but computationally intense triangular element (TRIP) based on a quintic polynomial shape function, and a simpler Discrete Kirchhoff Theory (DKT) triangular element (TRP2) that is currently in all versions of DYCAST. These elements are being expanded to include the capability to treat composite material behavior as a first step in developing capability for nonlinear analysis of composite structures. To achieve this goal additions to DYCAST will include:

- o Input of composite material properties either as integrated material property matrices or by a layer by layer input of material properties and ply angles
- o Additional integration points through the plate thickness (up to 100) to adequately describe composite layers and material behavior
- o Treatment of nonlinear composite material behavior
- o Treatment of material failure through the implementation of failure criteria

General steps of the development and/or expansion of these capabilities and their inclusion into DYCAST are discussed below. However, more detailed derivations and developments are not included since such is beyond the scope and intent of this paper.

Element Constitutive Relations-Linear Case

In the derivation of the original TRP2 element, the three integrated material matrices [A], [B], and [D] relate the stress and moment resultants to the middle surface strains and curvatures. For a homogeneous material, the [A], [B], and [D] matrices are:

$$[A] = \int_{-h/2}^{h/2} [\bar{Q}]\, dz$$

$$[B] = \int_{-h/2}^{h/2} z[\bar{Q}]\, dz$$

$$[D] = \int_{-h/2}^{h/2} z^2[\bar{Q}]\, dz$$

The $[\bar{Q}]$ matrix is the 3 by 3 matrix of material properties that characterize the material at a point, z is the coordinate normal to the plate mid-surface with its origin on the mid-surface, and h is the plate thickness.

For a laminated composite plate where the material properties in each lamina can be considered uniform in the transverse direction, the matrices become:

$$[A] = \sum_{k=1}^{n} [\overline{Q}^{k}] (h_k - h_{k-1})$$

$$[B] = 1/2 \sum_{k=1}^{n} [\overline{Q}^{k}] (h_k^2 - h_{k-1}^2)$$

$$[D] = 1/3 \sum_{k=1}^{n} [\overline{Q}^{k}] (h_k^3 - h_{k-1}^3)$$

where h_k is the distance to the bottom of the kth ply from the mid-surface of the laminate, and $[\overline{Q}]^k$ is the kth ply material property matrix transformed into the laminate reference material coordinate system which involves the material principal axes and ply orientation angle of individual layer k. The stiffness matrix involves, then, the Young's modulii of the layers in the two principal directions, the shear modulus, and the major Poisson's ratio. The mass matrix is similarly expanded to include the individual mass densities of the layers of the composite material.

Nonlinear Material Capability

Nonlinear material behavior in DYCAST presently is accounted for at every Gauss integration point through the material thickness. Accordingly, the matrix of stiffness coefficients must be recalculated at every Gauss point at every step of the required incremental analysis. Because in practice a composite laminate is likely to be composed of a large number of plies, as a first step in the development, the number of Gauss integration points allowed through the thickness of plates will be increased to 100 (from the current 9). Additionally, these points through the thickness of the plate will now be identified with actual plies or "layers" of the composite material rather than identified with the arbitrarily selected Gauss integration point of the original formulation.

Individual ply nonlinear material properties will be represented by a set of piecewise-linear multi-segment stress-strain curves derived from uniaxial and shear tests. Curves for compression need not be the same as the tension properties.

Failure Criteria

As a first treatment, a failure criteria that will be operationally simple to use, will require limited data, and can be used to predict actual modes of failure will be incorporated into the code. The maximum strain criteria chosen for these above reasons should perform well for laminated composites in which both the fiber and matrix are relatively brittle. It should be especially accurate where nonlinear shear strain components are small and do not interact with the direct strains. Under increased interactions, tensorial failure theories based on strength (Tsai-Wu) may prove to be more accurate.

The layer-by-layer failure algorithm added to DYCAST accounts for failure of continuous filament type reinforced laminates. The maximum strain criterion is implemented within each layer, with different values for the fiber direction, the transverse direction, and the in-plane shear. When a failure strain is reached at one of the strain recovery points in a layer, stiffnesses and stresses assigned to the point are deleted for all succeeding time or load steps. In this way the laminate's total stiffness and stress resultants degrade naturally and progressively as the individual layers fail. The analyst has the option to allow failure in one direction to cause failures in other directions or to keep the failures independent of each other. Further development of practical failure or damage criteria will be continuing.

Evaluation and/or verification of the new elements' capabilities will be coordinated with benchmark test cases and experimental data being developed concurrently through generic composite structures research.

Adaptation to Supercomputers

Under the grant with GWU, the objective of the research is to develop an effective computational strategy for predicting the dynamic response of composite structures during impact. The strategy is based on generating the dynamic response of a complex structure using large perturbations from a model associated with a simpler structure or a simpler mathematical/discrete model of the original structure. The response of the simpler model is used as a predictor and an iterative process (e.g. based on the preconditioned conjugate gradient or the multigrid method) is used to generate the response of the original model. Two general approaches are used for selecting the simpler model and establishing the relations between the original and simpler models, namely, hierarchal modeling; and decomposition or partitioning strategy (see Ref. 11.).

In the hierarchical modeling the simpler model is selected to be a mathematical model of the lower-dimensionality. For example, if the original model is associated with a two-dimensional plate or shell structure, the simpler model is selected to be a thin-walled beam. This is the approach currently being used to numerically simulate the impact response of composite fuselage frames. A curved beam finite element is being developed based on Vlasov's type thin-walled beam theory. The beam element, in addition to the 6-degrees of freedom involving displacements and rotations, accounts for the warping of the cross-section, the transverse shear deformation, and the anisotropic material response.

In the partitioning strategy the unsymmetric dynamic response of the structure is approximated by a linear combination of symmetric and anti-symmetric modes. Each of the modes is generated by using only a portion of the structure, with a fraction of the degrees of freedom of the original model. The simpler model corresponds to a symmetrized structure (see Ref. 12). Initial experience with this approach has demonstrated the effectiveness, particularly when implemented on the new multiprocessor computers (such as CRAY-2, CRAY X-MP and ETA-10, see Ref. 13). The equations of the simpler model, associated with the different symmetric/antisymmetric components of the response, can be solved in parallel on different processors.

Results derived from these approaches to solving composite structures response to crash loading situations will be compared with an experimental data base being developed for generic composite structural components. Additionally, it is planned to use the new curved beam element from this effort as the curved beam element to be implemented in the DYCAST code.

CONCLUDING REMARKS

The DYCAST nonlinear finite element computer code has been used in a series of progressively more difficult modeling tasks with the goal of accurately modeling complete transport aircraft crashes such as the Controlled Impact Demonstration (CID). Single aircraft frames and fuselage section vertical drop tests were modeled and analyzed to obtain comparisons with experimental data and to develop hybrid element crush springs for use in the large CID model. Modifications to DYCAST as the research progressed (including a progressive failure criteria) were made during the study. Predictions of crush and acceleration levels from a symmetric CID model agreed well with data from the CID experiment. The good comparisons achieved with the DYCAST model of the CID indicate the validity of the building-block analysis approach of using results from detailed models of substructure to form hybrid elements for inputs to more complex structures (i.e., full airplane models) to limit the size of the model but still provide a useful prediction tool for crash assessment.

Graphite-epoxy Z cross sectional frames were drop tested at typical vertical crash velocities onto a concrete surface. Floor level masses were attached to these frames to represent structural and occupant weights. Cross sectional shape of these frames was similar to designs often proposed for transport fuselage frames. Floor level acceleration and strain data were measured during the tests.

Lightly loaded frames failed at or near the impact point. More heavily loaded frames failed initially at or near the impact point and subsequently failed at additional locations left and right of the impact point. For graphite-epoxy frames impacted on splice plates, failure occurred approximately 15 degrees from the impact point. For frames impacted between splice plates, failure occurred at or near the impact point. The failures of all graphite-epoxy frames were complete separation across the frame sections. Portions of the frames between failures did not exhibit visible damage.

Initial results from an analysis of the response of the frames using the finite element computer program DYCAST were obtained. Good correlation was obtained between predicted and experimental floor level accelerations. The failure strain used in the DYCAST analysis proved to have a significant influence on the predicted frame response. Additional analytical studies are contemplated to assess further the capabilities and possible improvements needed to analyze composite structural elements using the DYCAST code.

REFERENCES

1. Thomson, R. G.; Carden, H. D.; and, Hayduk, R. J.: Survey of NASA Research on Crash Dynamics. NASA TP 2298, April 1984.

2. Fasanella, E. L.; Widmayer, E.; and, Robinson, M. P.: Structural Analysis of the Controlled Impact Demonstration of a Jet Transport Airplane. Journal of Aircraft, Vol. 24, No. 4, April 1987, pp. 274-280.

3. Carden, H. D.: Correlation and Assessment of Structural Airplane Crash Data with Flight Parameters at Impact. NASA TP 2083, November 1982.

4. Carden, H. D.: Impact Dynamics Research on Composite Transport Structures. NASA TM 83691, March 1985.

5. Jackson, K. E.: Abrasion Behavior of Aluminum and Composite Skin coupons, Stiffened Skins, and Stiffened Panels Representative of Transport Airplane Structures. NASA TP 2520/AVSCOM TP 85-B-7, November 1985.

6. Boitnott, Richard L.; Fasanella, Edwin L.; Calton, Lisa E.; and, Carden, Huey D.: Impact Response of Composite Fuselage Frames. Presented at the General Aviation Aircraft Meeting and Exposition, Wichita, KN, April 28-30, 1987, SAE Paper 871009.

7. Morton, J., "Scaling of Impact Loaded Carbon Fiber Composites," Proceedings of the 28th SDM Conference, Part 1., Monterey, CA., April 1987, pp. 819-826.

8. Boitnott, R. L.; Johnson, E. R.; and, Starnes, J. H., Jr.: A Nonlinear Analysis of Infinitely Long Graphite-Epoxy Cylindrical Panels Loaded with Internal Pressure. Presented at the [illegible]

AIAA/ASME/ASCE/AHS 26th Structures, Structural Dynamics, and Materials Conference, Orlando, FL, April 15-17, 1985, AIAA Paper No. 85-0770-CP.

9. Farley, Gary L.: Energy Absorption of Composite Materials and Structures. Presented at the American Helicopter Society 43rd Annual Forum and Technology Display, St. Louis, MO, May 18-20, 1987.

10. Pifko, A. B.; Winter, R.; and, Ogilvie, P. L.: DYCAST - A Finite Element Program for the Crash Analysis of Structures. NASA CR 4040, January 1987.

11. Noor, Ahmed K.; and, Peters, Jeanne M.: A Computational Strategy for Making Complicated Structural Problems Simple. AIAA/ASME/ASCE/AHS/ASC 29th Structures, Structural Dynamics and Materials Conference, April 18-20, 1988, Williamsburg, VA.

12. Noor, Ahmed K.; and, Peters, Jeanne M.: Model-Size Reduction for the Nonlinear Dynamic Analysis of Quasi-symmetric Structures. Engineering Computations, Vol. 4, Number 3, September 1987.

13. Noor, Ahmed K.; and, Peters, Jeanne M.: Nonlinear Finite Element Dynamic Analysis on Multi-processor Computers. International Journal for Numerical Methods in Engineering (to appear).

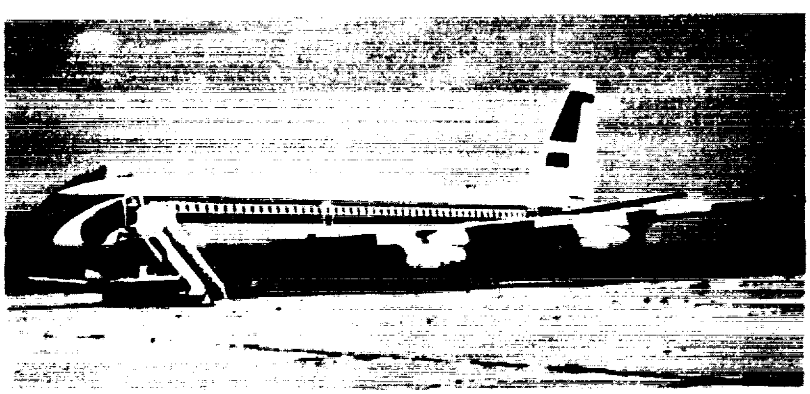


Fig. 1 Boeing 720 transport used in CID.

ORIGINAL PAGE IS
OF POOR QUALITY

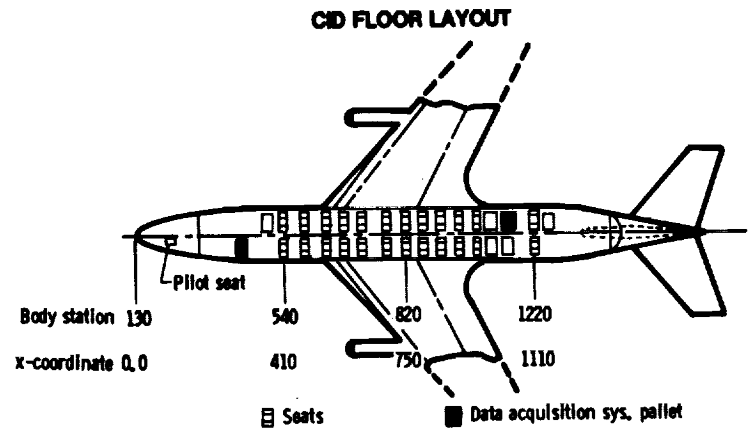


Fig. 2 CID floor layout.

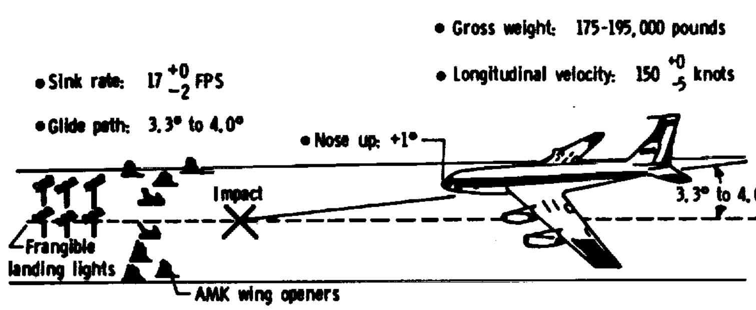


Fig. 3 CID planned impact scenario.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

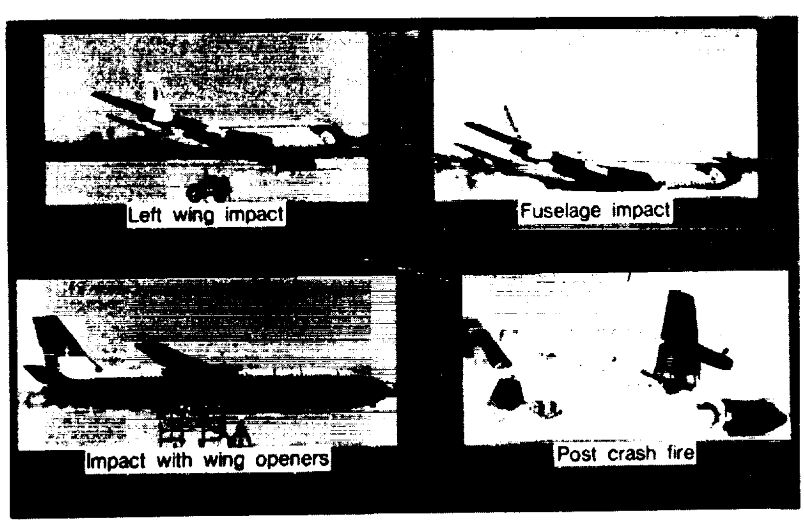


Fig. 4 CID impact scenario.

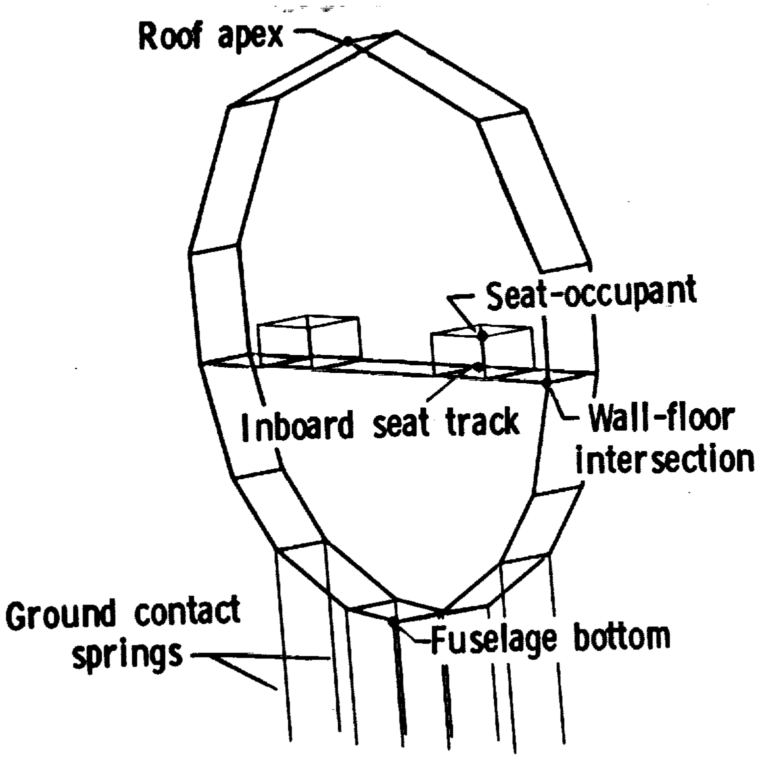
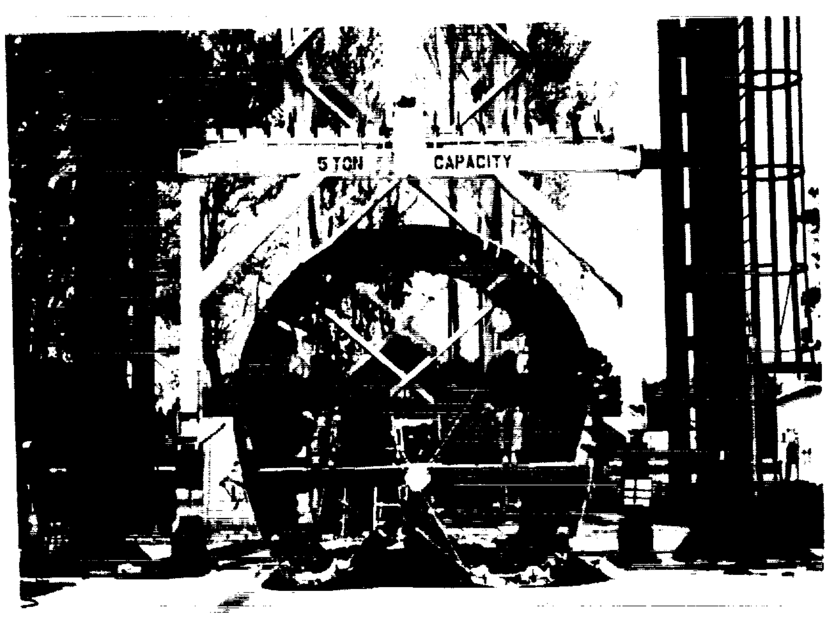


Fig. 5 Two-frame model showing nodes where responses are compared.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH



ORIGINAL PAGE IS
OF POOR QUALITY

Fig. 6 Forward transport section showing post-test damage.

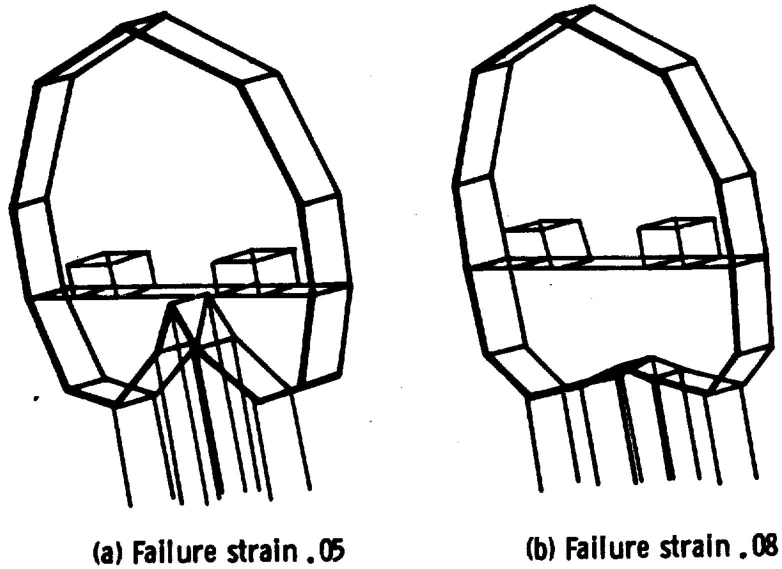


Fig. 7 Computer graphics showing effects of beam failure strain criteria at time 0.23 seconds after impact.

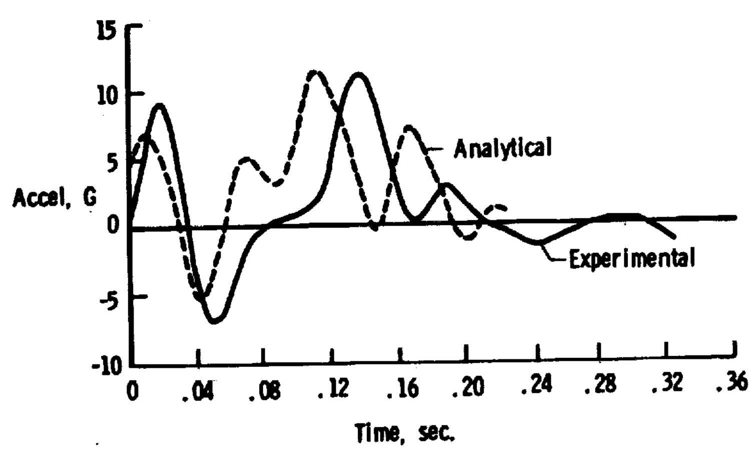


Fig. 8 Comparison of experimental and analytical vertical accelerations at wall/floor interface.

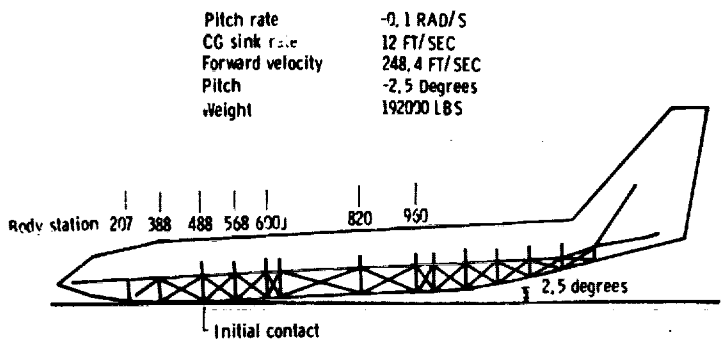


Fig. 9 CID model attitude and conditions at initial fuselage ground impact.

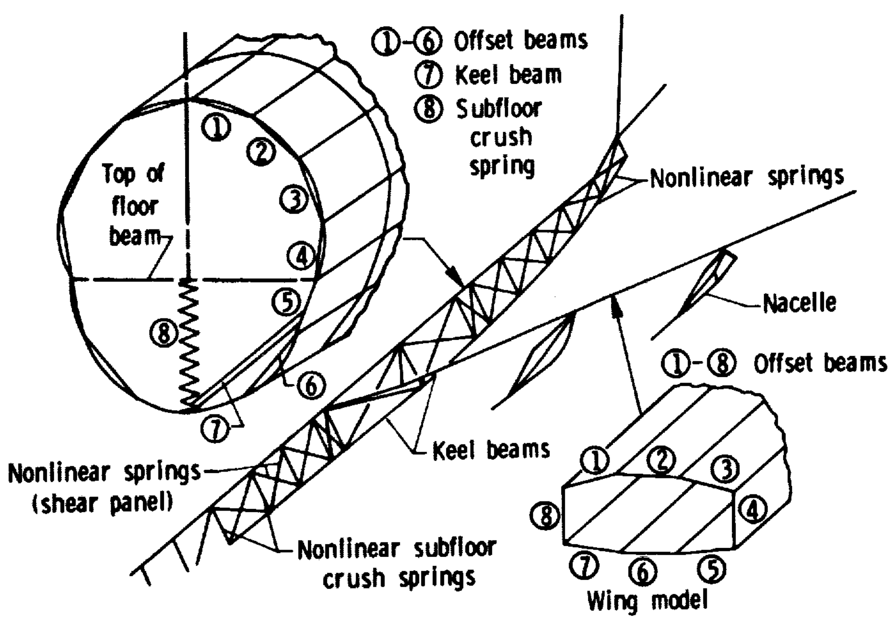


Fig. 10 CID finite element airplane model.

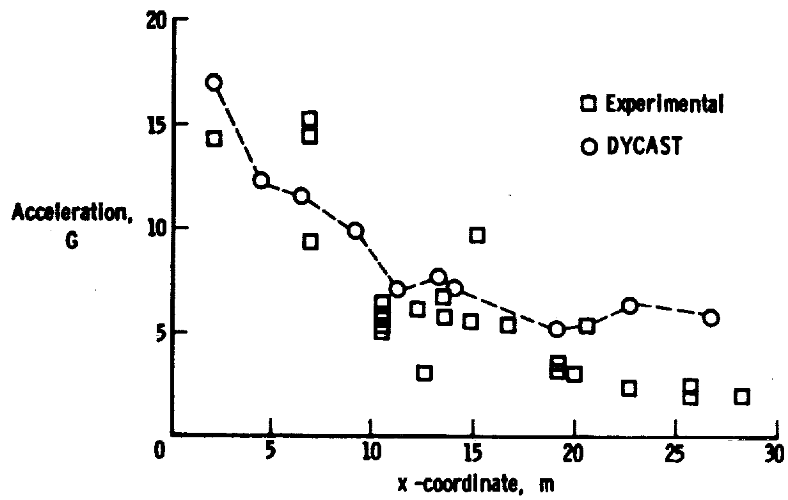


Fig. 11 Comparison of peak vertical floor accelerations.

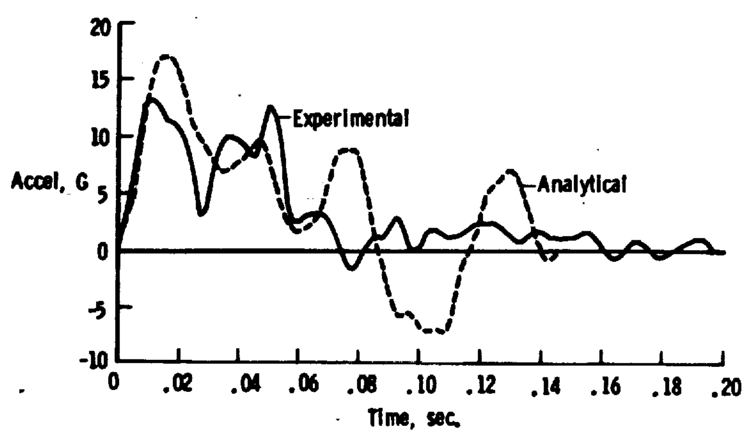


Fig. 12 Comparison of experimental and DYCAST analytical vertical floor accelerations at BS207.

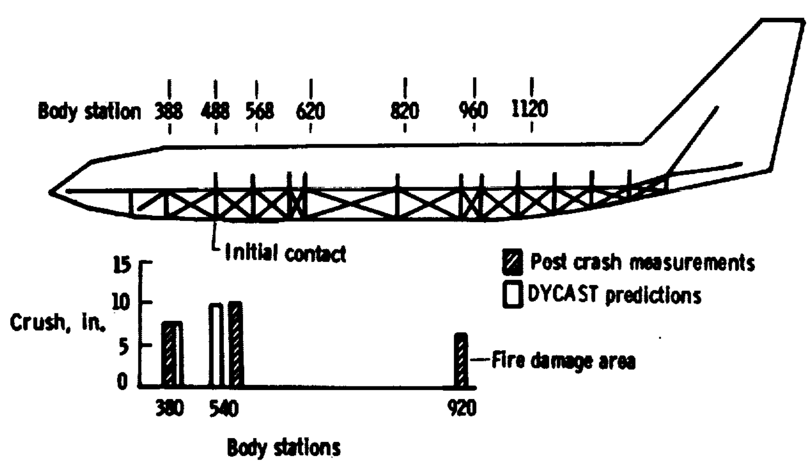


Fig. 13 Comparison of post crush measurements with DYCAST predictions.

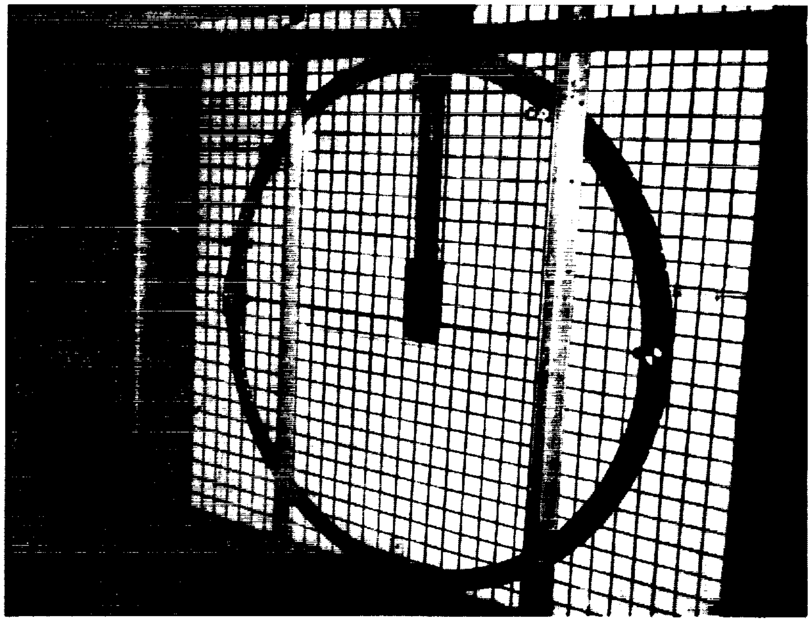


Fig. 14 Graphite-epoxy Z-shaped frame.

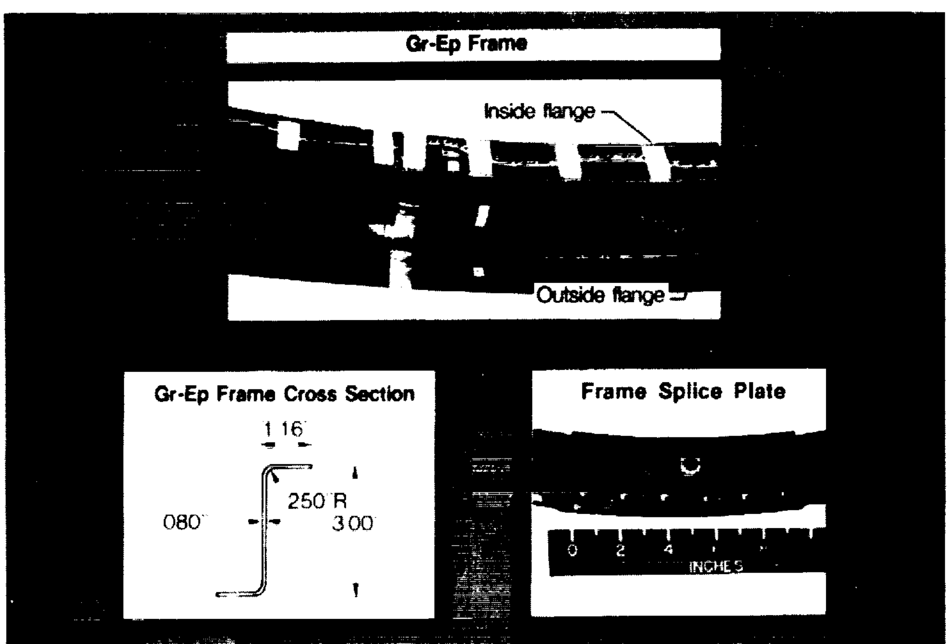


Fig. 15 Graphite-epoxy Z-shaped frame cross section.

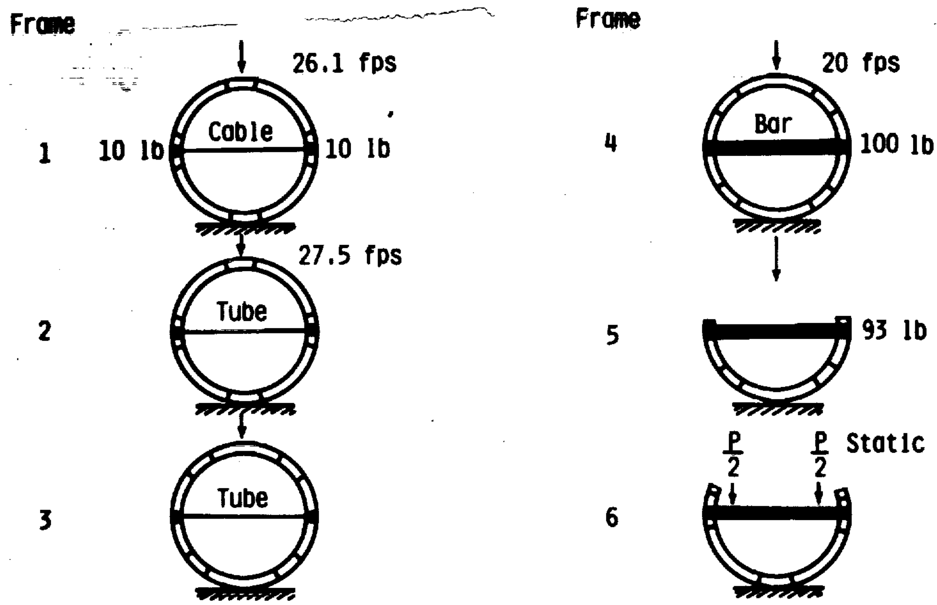


Fig. 16 Composite frame test conditions.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

ORIGINAL PAGE IS
OF POOR QUALITY

Fig. 17 Drop tower with frame in pre-drop position.

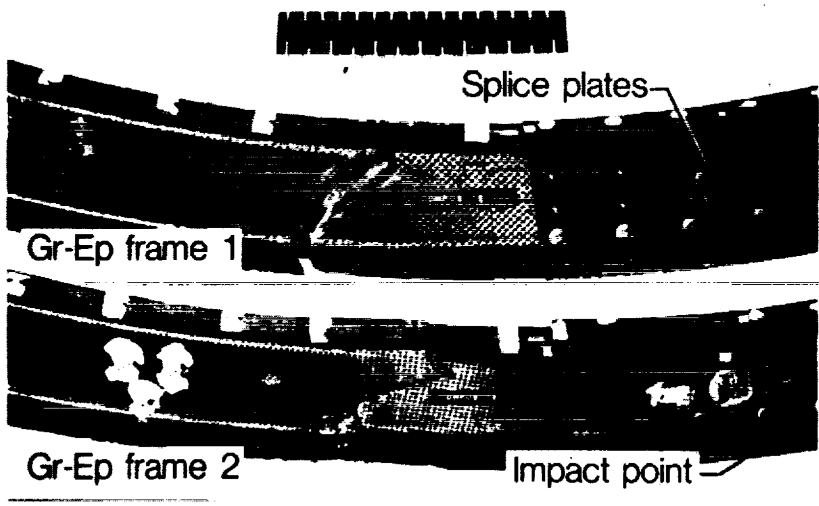


Fig. 18 Failure similarities of frames 1 and 2.

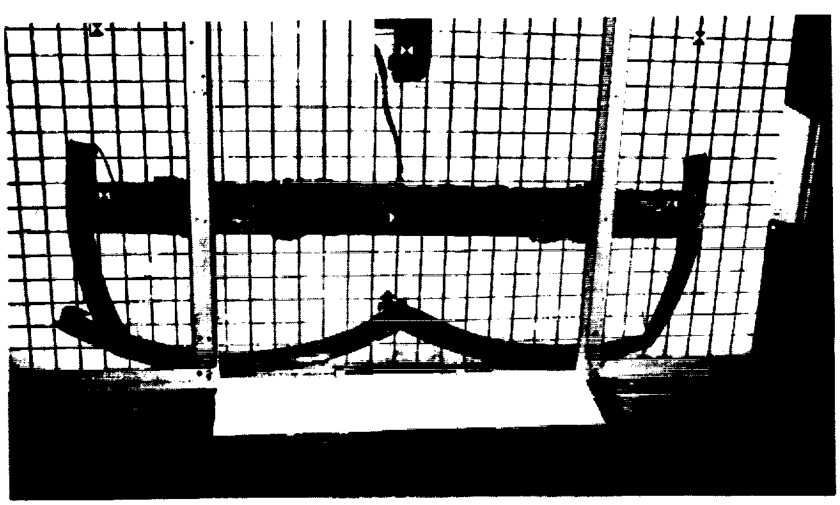


Fig. 19 Frame 5 after impact.

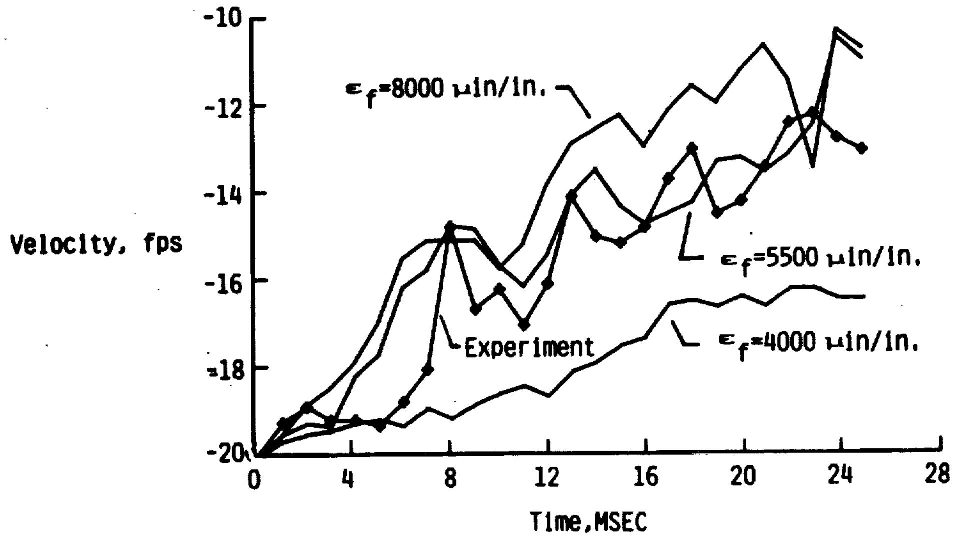


Fig. 20 DYCAST and experimental floor velocities for frame 5. DYCAST results are shown for failure strains of 0.004, 0.0055, and 0.008.

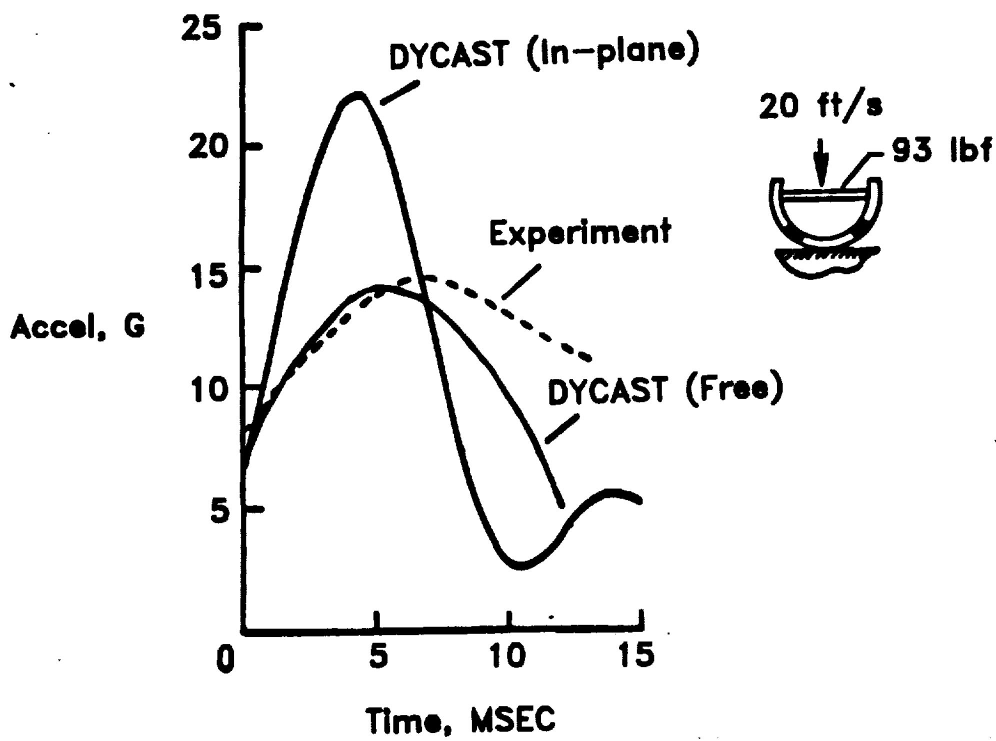


Fig. 21 DYCAST correlation with composite frame floor acceleration.

Report Documentation Page

National Aeronautics and Space Administration

1. Report No. NASA TM-102595	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle A Review of the Analytical Simulation of Aircraft Crash Dynamics		5. Report Date January 1990
		6. Performing Organization Code
7. Author(s) Edwin L. Fasanella, Huey D. Carden, Richard L. Boitnott and Robert J. Hayduk		8. Performing Organization Report No.
		10. Work Unit No. 505-63-01-11
9. Performing Organization Name and Address NASA Langley Research Center Hampton, VA 23665-5225		11. Contract or Grant No.
		13. Type of Report and Period Covered Technical Memorandum
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, DC 20546-0001		14. Sponsoring Agency Code

15. Supplementary Notes

Edwin L. Fasanella: Lockheed Engineering & Sciences Co., Hampton, VA
Huey D. Carden and Robert J. Hayduk: NASA Langley Research Center, Hampton, VA
Richard L. Boitnott: US Army Aviation Research and Technology Activity, Hampton, VA

16. Abstract

The National Aeronautics and Space Administration has been conducting extensive research on the crash dynamics of aircraft for more than fifteen years. A large number of full-scale tests of general aviation aircraft, helicopters, and one unique air-to-ground controlled-impact of a transport aircraft have been performed. Additionally, research has also been conducted on seat dynamic performance, load-limiting seats, load-limiting subfloor designs, and emergency-locator-transmitters (ELTs). Computer programs have been developed to provide designers with methods for predicting accelerations, velocities, and displacements of collapsing structure and for estimating the human response to crash loads. The results of full-scale aircraft and component tests have been used to verify and guide the development of analytical simulation tools and to demonstrate impact load attenuating concepts. In recent years, the trend of NASA's research in aircraft crash dynamics has been directed less on metal aircraft structures and more toward composite structures.

17. Key Words (Suggested by Author(s)) Aircraft crash dynamics Nonlinear structural dynamics Impact dynamics Composite materials	18. Distribution Statement Unclassified - Unlimited Subject Category 03

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of pages	22. Price
Unclassified	Unclassified	24	A03

NASA FORM 1626 OCT 86