Finite Element Simulation of a Full-Scale Crash Test of a Composite Helicopter

Edwin L. Fasanella, Karen E. Jackson, and Karen H. Lyle
US Army Research Laboratory, Vehicle Technology Directorate
e.l.fasanella@larc.nasa.gov, k.e.jackson@larc.nasa.gov, k.h.lyle@larc.nasa.gov
Hampton, VA

Abstract

A finite element model of the Sikorsky Advanced Composite Airframe Program (ACAP) helicopter was developed using the nonlinear, explicit transient dynamic code, MSC.Dytran. Analytical predictions were correlated with experimental data obtained from a full-scale crash test of the Sikorsky ACAP helicopter flight test article that was conducted at the Impact Dynamics Research Facility of NASA Langley Research Center. The helicopter was impacted at 38-ft/s vertical and 32.5-ft/s forward velocity with an attitude of 6.25° pitch (nose up) and 3.5°-left roll. The objective of the crash simulation was to evaluate the capabilities of a commercially available transient dynamic code in predicting the response of a composite airframe subjected to impact loading. The model was developed from an existing MSC.Nastran modal-vibration model of the helicopter. Considerable modifications were made in converting the original modal-vibration model to a model for crash simulation. Following conversion of the model, a two-stage modeling approach was used to generate analytical predictions. Due to the relatively long pulse duration, a rigid structural model containing a fairly complex landing gear model was executed from initial contact through landing gear stroke. Prior to fuselage contact, the nodal displacements and velocities were output to a file. Then, a flexible structural model was executed with the nodal displacements and velocities used as initial conditions. This paper describes the development of the finite element crash model, the two-stage modeling approach, and the correlation of the analytical predictions with the experimental data from the full-scale crash test of the ACAP helicopter.

Introduction

An important aspect of crashworthiness research is the demonstration and validation of analytical/computational tools for accurate simulation of airframe structural response to crash impacts. The “validation of numerical simulations” was identified as one of five key technology shortfalls during the Workshop on Computational Methods for Crashworthiness [1], which was held at NASA Langley Research Center in 1992. Crash simulation codes can be used during the airframe design phase to certify seats and aircraft to dynamic crash loads, to predict seat and occupant response to impact with the probability of injury, and to evaluate numerous crash scenarios not economically feasible with full-scale crash testing.

The US Army has been active in supporting the development and utilization of crash modeling and simulation codes for many decades. More than 25 years ago, the US Army sponsored the initial development of a kinematic crash analysis code, KRASH [2], by the Lockheed-California Company. Kinematic codes employ a semi-empirical modeling approach using lumped-masses, beams, and nonlinear springs to represent the airframe structure. These codes rely heavily on test data for definition of spring properties to characterize the crushing behavior of the subfloor and other structural components. Good correlation between the analytical and experimental data is usually obtained for global parameters, such as engine or landing gear response. However, these codes would be unable to predict localized responses, e.g. the stress level in an airframe component at a particular time during a crash event.

Currently, a new generation of crash analysis codes have been developed that will accurately simulate the nonlinear, transient dynamic response of airframe structures. These finite element codes, such as LS-DYNA [3], MSC.Dytran [4], and PAM-CRASH [5], use an explicit solver.
that eliminates the need to repetitively decompose large global stiffness matrices as is required for implicit codes. Explicit codes require an extremely small time step, typically less than a microsecond, whose duration is controlled by the smallest element in the model. Thus, impact simulations having a pulse duration on the order of 30-40 milliseconds can require several CPU hours to solve on an engineering workstation. Consequently, efficient beam, shell, and solid elements are needed to achieve quick run times for very large models.

The new codes are capable of modeling nonlinear geometric behavior including large structural deformations. In addition, these codes are very effective in modeling materials, such as metals, that deform plastically and that have well known failure mechanisms. However, the use of light weight, high strength composite materials for aircraft construction brings with it difficulties in modeling material response and failure behavior. Structural composite materials can exhibit a wide range of material responses from linear elastic to completely nonlinear anisotropic behavior, depending on the individual fiber and matrix properties and laminate stacking sequence. Also, laminated composite materials exhibit a wide variety of failure modes including matrix cracking, fiber failure, and delamination that can occur singly or in combination. These failure modes can change depending on the type and rate of loading. In general, the initial failure event in a single ply of a composite laminate does not produce catastrophic failure. Consequently, the capability to model the progressive failure of composite materials, from initial damage to ultimate failure, is needed. With the increased application of composite materials in the construction of advanced aircraft and rotorcraft, it is important to build confidence in the computational capabilities of these codes through analytical/experimental validation.

In 1996, the US Army initiated a four-phase research program to evaluate the capabilities of commercial crash simulation codes for modeling the impact response of a composite helicopter. As part of this program, a finite element crash model of the Sikorsky Advanced Composite Airframe Program (ACAP) helicopter [6, 7] was developed. In this effort, an existing MSC.Nastran [8] modal-vibration model of the helicopter was converted to an MSC.Dytran model for the crash simulation. In 1999, a full-scale crash test of an ACAP helicopter was performed at the Impact Dynamics Research Facility (IDRF) [9] of NASA Langley Research Center to generate experimental data for correlation with the simulation. The present paper will describe: (1) the development of the helicopter crash model, (2) modifications made to the crash model to better represent the test article, (3) a summary of the experimental results from the full-scale crash test of the ACAP helicopter, and (4) the validation of the crash simulation through analytical/experimental correlation.

Finite Element Model Development

Description of MSC.Dytran Finite Element Code

The commercial code, MSC.Dytran, was used to perform the crash simulation of the ACAP helicopter. MSC.Dytran is a three-dimensional finite element code for simulating highly nonlinear transient response of solids, structures, and fluids. The MSC.Patran [10] pre- and post-processing code was used with the MSC.Dytran “Preference” to build the finite element crash model and to post-process the results.

Description of the Sikorsky Modal-Vibration Model

An MSC.Nastran model of the ACAP helicopter that was originally developed for correlation with modal vibration data [11] was obtained from Sikorsky Aircraft. The model, shown in Figure 1, had approximately 5000 nodes, 9,500 elements, 219 material models including many different composite materials, and over 700 different property cards. The elements included 5,453 shell elements; 1,956 beam elements; 1,770 rod elements; and 372 concentrated masses. Because this model was originally used for modal analysis, extensive modifications were required to convert it for a crash analysis.

In 1996, the US Army initiated a four-phase research program to evaluate the capabilities of commercial crash simulation codes for modeling the impact response of a composite helicopter. As part of this program, a finite element crash model of the Sikorsky Advanced Composite Airframe Program (ACAP) helicopter [6, 7] was developed. In this effort, an existing MSC.Nastran [8] modal-vibration model of the helicopter was converted to an MSC.Dytran model for the crash simulation. In 1999, a full-scale crash test of an ACAP helicopter was performed at the Impact Dynamics Research Facility (IDRF) [9] of NASA Langley Research Center to generate experimental data for correlation with the simulation. The present paper will describe: (1) the development of the helicopter crash model, (2) modifications made to the crash model to better represent the test article, (3) a summary of the experimental results from the full-scale crash test of the ACAP helicopter, and (4) the validation of the crash simulation through analytical/experimental correlation.

Finite Element Model Development

Description of MSC.Dytran Finite Element Code

The commercial code, MSC.Dytran, was used to perform the crash simulation of the ACAP helicopter. MSC.Dytran is a three-dimensional finite element code for simulating highly nonlinear transient response of solids, structures, and fluids. The MSC.Patran [10] pre- and post-processing code was used with the MSC.Dytran “Preference” to build the finite element crash model and to post-process the results.

Description of the Sikorsky Modal-Vibration Model

An MSC.Nastran model of the ACAP helicopter that was originally developed for correlation with modal vibration data [11] was obtained from Sikorsky Aircraft. The model, shown in Figure 1, had approximately 5000 nodes, 9,500 elements, 219 material models including many different composite materials, and over 700 different property cards. The elements included 5,453 shell elements; 1,956 beam elements; 1,770 rod elements; and 372 concentrated masses. Because this model was originally used for modal analysis, extensive modifications were required to convert it for a crash analysis.

In 1996, the US Army initiated a four-phase research program to evaluate the capabilities of commercial crash simulation codes for modeling the impact response of a composite helicopter. As part of this program, a finite element crash model of the Sikorsky Advanced Composite Airframe Program (ACAP) helicopter [6, 7] was developed. In this effort, an existing MSC.Nastran [8] modal-vibration model of the helicopter was converted to an MSC.Dytran model for the crash simulation. In 1999, a full-scale crash test of an ACAP helicopter was performed at the Impact Dynamics Research Facility (IDRF) [9] of NASA Langley Research Center to generate experimental data for correlation with the simulation. The present paper will describe: (1) the development of the helicopter crash model, (2) modifications made to the crash model to better represent the test article, (3) a summary of the experimental results from the full-scale crash test of the ACAP helicopter, and (4) the validation of the crash simulation through analytical/experimental correlation.

Finite Element Model Development

Description of MSC.Dytran Finite Element Code

The commercial code, MSC.Dytran, was used to perform the crash simulation of the ACAP helicopter. MSC.Dytran is a three-dimensional finite element code for simulating highly nonlinear transient response of solids, structures, and fluids. The MSC.Patran [10] pre- and post-processing code was used with the MSC.Dytran “Preference” to build the finite element crash model and to post-process the results.

Description of the Sikorsky Modal-Vibration Model

An MSC.Nastran model of the ACAP helicopter that was originally developed for correlation with modal vibration data [11] was obtained from Sikorsky Aircraft. The model, shown in Figure 1, had approximately 5000 nodes, 9,500 elements, 219 material models including many different composite materials, and over 700 different property cards. The elements included 5,453 shell elements; 1,956 beam elements; 1,770 rod elements; and 372 concentrated masses. Because this model was originally used for modal analysis, extensive modifications were required to convert it for a crash analysis.

In 1996, the US Army initiated a four-phase research program to evaluate the capabilities of commercial crash simulation codes for modeling the impact response of a composite helicopter. As part of this program, a finite element crash model of the Sikorsky Advanced Composite Airframe Program (ACAP) helicopter [6, 7] was developed. In this effort, an existing MSC.Nastran [8] modal-vibration model of the helicopter was converted to an MSC.Dytran model for the crash simulation. In 1999, a full-scale crash test of an ACAP helicopter was performed at the Impact Dynamics Research Facility (IDRF) [9] of NASA Langley Research Center to generate experimental data for correlation with the simulation. The present paper will describe: (1) the development of the helicopter crash model, (2) modifications made to the crash model to better represent the test article, (3) a summary of the experimental results from the full-scale crash test of the ACAP helicopter, and (4) the validation of the crash simulation through analytical/experimental correlation.

Finite Element Model Development

Description of MSC.Dytran Finite Element Code

The commercial code, MSC.Dytran, was used to perform the crash simulation of the ACAP helicopter. MSC.Dytran is a three-dimensional finite element code for simulating highly nonlinear transient response of solids, structures, and fluids. The MSC.Patran [10] pre- and post-processing code was used with the MSC.Dytran “Preference” to build the finite element crash model and to post-process the results.

Description of the Sikorsky Modal-Vibration Model

An MSC.Nastran model of the ACAP helicopter that was originally developed for correlation with modal vibration data [11] was obtained from Sikorsky Aircraft. The model, shown in Figure 1, had approximately 5000 nodes, 9,500 elements, 219 material models including many different composite materials, and over 700 different property cards. The elements included 5,453 shell elements; 1,956 beam elements; 1,770 rod elements; and 372 concentrated masses. Because this model was originally used for modal analysis, extensive modifications were required to convert it for a crash analysis.

In 1996, the US Army initiated a four-phase research program to evaluate the capabilities of commercial crash simulation codes for modeling the impact response of a composite helicopter. As part of this program, a finite element crash model of the Sikorsky Advanced Composite Airframe Program (ACAP) helicopter [6, 7] was developed. In this effort, an existing MSC.Nastran [8] modal-vibration model of the helicopter was converted to an MSC.Dytran model for the crash simulation. In 1999, a full-scale crash test of an ACAP helicopter was performed at the Impact Dynamics Research Facility (IDRF) [9] of NASA Langley Research Center to generate experimental data for correlation with the simulation. The present paper will describe: (1) the development of the helicopter crash model, (2) modifications made to the crash model to better represent the test article, (3) a summary of the experimental results from the full-scale crash test of the ACAP helicopter, and (4) the validation of the crash simulation through analytical/experimental correlation.

Finite Element Model Development

Description of MSC.Dytran Finite Element Code

The commercial code, MSC.Dytran, was used to perform the crash simulation of the ACAP helicopter. MSC.Dytran is a three-dimensional finite element code for simulating highly nonlinear transient response of solids, structures, and fluids. The MSC.Patran [10] pre- and post-processing code was used with the MSC.Dytran “Preference” to build the finite element crash model and to post-process the results.

Description of the Sikorsky Modal-Vibration Model

An MSC.Nastran model of the ACAP helicopter that was originally developed for correlation with modal vibration data [11] was obtained from Sikorsky Aircraft. The model, shown in Figure 1, had approximately 5000 nodes, 9,500 elements, 219 material models including many different composite materials, and over 700 different property cards. The elements included 5,453 shell elements; 1,956 beam elements; 1,770 rod elements; and 372 concentrated masses. Because this model was originally used for modal analysis, extensive modifications were required to convert it for a crash analysis.

Figure 1. Sikorsky Modal-vibration model of the ACAP helicopter.
Conversion of the modal-vibration model to a crash model

The initial work in converting the modal-vibration model to an MSC.Dytran input deck concentrated on combining elements, deleting unnecessary elements, and rediscretizing the model. The original linear elastic material property cards had to be modified to include yield stress, strain hardening, density, and maximum strain to failure, as appropriate. The tail cone was greatly simplified, and the stabilizer and rudder were both removed. A transition mesh was developed to connect the coarse mesh of the tail cone to the finer mesh of the fuselage cabin. Many of the original triangular elements were combined and converted to quadrilateral elements. Triangular elements are not recommended for models that undergo large deformations as they are typically too stiff.

Most of the original concentrated masses in the modal-vibration model were removed. Other concentrated masses were modified or added to represent actual lumped masses on the helicopter including the rotorcraft transmission, engines, anthropomorphic dummies, seats, fuel, instrumentation boxes, batteries, and cameras, etc.

The elements in the original modal-vibration model representing the main and nose landing gear were removed. An external user subroutine was developed to calculate the main landing gear forces as a function of velocity and stroking distance including both the oleo-pneumatic and the crushable honeycomb stages. The original crashworthy nose gear of the ACAP helicopter had been removed and replaced with a non-crashworthy standard nose gear. Modifications were required to make the existing nose gear more crashworthy. The hydraulic fluid was drained, and a thin-walled aluminum tube with a honeycomb core was inserted inside the gear to provide some energy absorption. The modified nose gear was modeled as a spring having a constant spring force of 8,000-lb. to represent the crush strength of the honeycomb-filled aluminum tube. A more detailed description of the landing gear modeling approach is provided in the next section.

Since the original modal-vibration model was too detailed in some regions and too coarse in other regions, especially near the impact point, some rediscretization of the mesh was made. One of the main energy absorbing devices for the structure was the crushable lower portion of the keel beams and subfloor bulkheads. There were four keel beams beneath the floor, two inner keel beams and two outer keel beams. The keel beams and transverse bulkheads beneath the floor were constructed of two horizontal C-channels, one above the other, with a beaded (or waffle) web geometry. The upper channel was constructed of graphite and the lower 4-inch high beaded web was constructed of Kevlar. The outer keel beams were very thick and did not crush in the test. The lower portion of the inner keel and bulkhead beams was fabricated of a thinner beaded Kevlar construction designed to crush and absorb energy. In the original model, the crushable Kevlar web was modeled using 4-inch-high shell elements with longitudinal beam elements to represent the flanges of the C-sections. The Kevlar shell elements representing the keel beam and the bulkheads were rediscretized by dividing each original shell element vertically into four shell elements. This representation, as shown in Figure 2, allowed crushing to occur [12].

Following the extensive modifications, the ACAP helicopter crash model executed with a time step of approximately 1.9 microseconds. The model was run many times to track down the elements controlling the time step. These elements were then combined with other elements to increase the time step to an appropriate duration to allow a reasonable run time for the model.

Description of the crash model

The final crash model of the ACAP helicopter is shown in Figure 3. The model consists of 4,128 grid points and 7,346 elements, which
include 3,118 beam and rod elements, 3,283 quadrilateral shell elements, 695 triangular shell elements, and 250 solid elements that represent the impact surface. The number of different material property cards was reduced from 219 in the MSC.Nastran modal-vibration model to 34 in the MSC.Dytran crash model. In addition, the number of different property cards for shell elements was significantly reduced by combining many of the PCOMP cards. The PCOMP cards are used to specify the material properties and orientations for each ply in a laminated composite shell element. Also, the total number of concentrated masses in the model was reduced from 372 in the modal-vibration model to 98 in the crash model. The 98 concentrated masses represent actual lumped masses used in the experiment, as indicated in Table 1. In the final crash model, the structural elements weighed 2,838-lb. and the concentrated masses weighed 5,160-lb., for a total weight of 7,998-lb.

A flat plate consisting of 250 solid elements with fixed bottom nodes was added to the model to represent the impact surface. A master surface to slave node contact was defined between the flat impact surface and the nodes in the structural model. The initial conditions including pitch, roll, yaw, and translational and rotational velocities were determined from measurements made from photographs taken from the high-speed video and from motion picture analysis of the high-speed (400 frames/s) film. The coordinate system used in the model was x-axis positive from nose to tail, z-axis positive up, and y-axis position to the right. The nominal impact conditions are -38.5 ft/s vertical velocity, -32 ft/s longitudinal velocity, 9.6 degrees/second angular pitch velocity (rotating backward due to swing), 6.25° pitch (nose up), and 3.5° left roll. For this simulation, it was more expedient to adjust the position of the impact surface, rather than the structural model to account for the roll and pitch attitude. The positions of the center-of gravity for the helicopter model were determined to be x = 203.7-in., y = 0-in., and z = 87-in. Note that these dimensions are given in reference to the manufacturer’s body station (BS), butt-line, and waterline coordinate system.

Modeling Approach

To perform the simulation, a two stage modeling approach was employed in which a rigid structural model of the helicopter was executed during deformation of the landing gear. Approximately 0.05 seconds before fuselage contact, the x, y, and z-locations of all grid points and the corresponding nodal velocities in the rigid model were output to a file. These initial conditions were then input as the starting point of the flexible model simulation. The rigid-to-flexible approach was used to significantly decrease the CPU time required to complete the simulation, and because the rigid model made the introduction of the pitch angular velocity much easier to input. The development of the landing gear model, rigid structural model, and flexible structural model are discussed in the following subsections.
Table 1. Summary of the major concentrated masses in the finite element model.

<table>
<thead>
<tr>
<th>Concentrated Mass</th>
<th>Weight, lb.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pilot dummy and seat</td>
<td>279.3</td>
</tr>
<tr>
<td>Copilot dummy and seat</td>
<td>315.5</td>
</tr>
<tr>
<td>Right and left fuel tank (805 lb. each)</td>
<td>1,610.0</td>
</tr>
<tr>
<td>Nose gear</td>
<td>52.0</td>
</tr>
<tr>
<td>Lifting hard point in the nose</td>
<td>98.0</td>
</tr>
<tr>
<td>Right and left main gear (122 lb. each)</td>
<td>244.0</td>
</tr>
<tr>
<td>Right and left engine (315 lb. each)</td>
<td>630.0</td>
</tr>
<tr>
<td>Main gearbox</td>
<td>832.0</td>
</tr>
<tr>
<td>Tail mass</td>
<td>69.0</td>
</tr>
<tr>
<td>Main rotor hub mass</td>
<td>352.6</td>
</tr>
<tr>
<td>Right and left troops &amp; seats (180 lb. each)</td>
<td>360.0</td>
</tr>
</tbody>
</table>

Landing Gear Model

A landing gear model, which accurately simulates the energy absorbed by the gear without adding substantial complexity to the model, was developed. For a crash model, the landing gear response can be represented as a spring, as shown in Figure 4(a), where the force is computed in a user-written external subroutine. In general, the “spring” force equation can be dependent on the relative displacement and the relative velocity of the connected nodes. This flexibility allows a user-defined spring to simulate the effect of the velocity dependant oleo-pneumatic stage and the crushable honeycomb stage, which depends only on displacement. An assumption is made that the gear is fully deployed, as would be expected in a crash scenario. Alignment of the spring relative to the aircraft can be a challenging modeling problem. A number of rigid joints, such as sliding, rotational, ball, and universal joints, are currently available as standard capabilities in commercial codes. However, these joints may become unstable when large forces such as those experienced by the landing gear during a severe impact are applied. For this reason, these built-in joints were replaced in the analysis with a component containing several nodes and beam elements over which the forces are distributed. Nodal alignment was accomplished by creating four contact surfaces on the outer surface of two perpendicular plates. The beam nodes representing the gear hub and piston were then constrained to remain within the intersecting shaded region, as shown in Figure 4(b).

For simplicity of model development, the initial landing gear models were attached to a simple triangular rigid plate element, which approximated the fuselage. The aircraft mass, center of gravity, and moments of inertia were explicitly specified for the triangular element. Workstation simulations using this simple model are completed in a few minutes. The predicted nodal accelerations, velocities, and displacements at the gear attachment nodes were compared with the corresponding experimental data. These comparisons allowed modifications to be quickly evaluated. Once adequate experimental and analytical correlation was achieved, the simple fuselage representation was replaced by a rigid ACAP fuselage model with accurate geometry and mass moments of inertia about the center-of-gravity.

Rigid Structural Model

The ACAP structural model was made rigid by setting material properties for all elements to MATRIG, except for those elements forming the landing gear. As described in the previous section, landing gear forces were computed using an external subroutine. This procedure allowed quick runs to be made, of approximately one-hour duration, to refine the correlation of the sequence of events, and to ensure that the landing gear subroutine accurately predicted the gear forces. For this model, it was important to ensure that the location of the center-of-gravity and moments of inertia matched the experimental values closely. The rigid model was executed for a duration of 0.2 seconds after left gear contact. The archive files generated by the rigid model were post-processed in MSC.Patran with user-
written commands to create a file containing nodal velocities and a file containing nodal positions at 0.045 seconds. The nodal position file, which had the new GRID locations for each node, was input into the flexible model using the INCLUDE statement. The velocity file was read into the flexible model by an external user-written FORTRAN subroutine.

Flexible Structural Model

The model was transitioned from rigid to flexible at 0.045 seconds after left gear impact, which is just after the right landing gear tire makes contact with the impact surface. The flexible structural model was the same as the rigid model except for the following modifications. The material cards were non-rigid, the initial GRID coordinates were the nodal positions of the rigid model at 0.045 seconds, and the initial velocity of each grid point was read in by the user subroutine from the rigid model at 0.045 seconds. Also, during this latter stage of the analysis, each landing gear force was modeled using the appropriate constant honeycomb crushing force, 8,000 pounds for the nose gear and approximately 26,000 pounds for each of the main gear. The transition occurred without incident except for a slight discontinuity that occurred when the flexible model was initiated due to the sudden onset of the landing gear honeycomb force. Several techniques to minimize this discontinuity (acceleration spike) at the transition point from rigid to flexible were investigated, such as ramping the honeycomb force over multiple time steps. However, an optimum solution to this problem was not found.

Experimental Program

A full-scale crash test of the Sikorsky ACAP helicopter was performed in 1999 to generate experimental data for correlation with the crash simulation. The development of two ACAP helicopters was sponsored in the early 1980’s by the US Army Applied Technology Laboratory. Separate contracts were awarded to Bell Helicopter Textron and Sikorsky Aircraft to design, fabricate, and test a new helicopter manufactured primarily from composite materials as part of the Advanced Composite Airframe Program. The helicopters were intended to demonstrate the potential of advanced composite materials to save weight and cost in airframe structures; while achieving systems compatibility and meeting Army requirements for vulnerability, reliability, maintainability, and survivability. The Sikorsky ACAP helicopter was designed as a composite replacement for the metallic S-76 helicopter. The resulting ACAP airframe consisted of 82% composite materials, and the total weight and cost savings achieved based on the final design were 23% and 24%, respectively. A systems approach was used...
in designing the helicopter for maximum crash protection, including energy absorbing landing gear, crushable subfloor structure, and load-limiting seats. However, the primary energy absorbing elements were the landing gear. The landing gear were designed to remove 80% of the energy of a crash performed at the impact conditions of 38-ft/s, 10° pitch, and 10° roll as specified in Reference [7]. The main landing gear contained an aluminum honeycomb tube that would dissipate kinetic energy during a crash event through stable crushing of up to 18 inches.

In 1987, the Sikorsky ACAP static test article was crash tested at the IDRF at NASA Langley Research Center to demonstrate and verify the crashworthy features of the airframe design [7]. A drop test was performed at 39 ft/s vertical velocity with a 10-degree pitch and a 10-degree roll impact attitude. There was no induced pitch or roll rate. Some of the data obtained from this test could have been used for correlation with the MSC.Dytran crash simulation. However, a more severe impact condition with both vertical and longitudinal velocity components was desired for this correlation. Given that the flight test article was available and in good condition, a second crash test was performed for the specific purpose of generating experimental data for comparison with the MSC.Dytran simulation.

In June of 1999, the second full-scale crash test of the Sikorsky ACAP helicopter was conducted at the IDRF. Prior to the drop test, some modifications were made to the airframe to correct problems that had occurred in the 1987 test [13]. Onboard hardware in the 1999 test included two different energy-absorbing crew seats for the pilot and copilot positions, two energy-absorbing troop seats, three-50th percentile Hybrid II anthropomorphic dummies, a modified 50th percentile dummy with a self-contained data acquisition system in the co-pilot seat, and two MA-16 inertia reels with triaxial locking mechanisms for pilot and copilot restraint systems. The fuel tanks were filled with water and instrumented with pressure transducers to record the hydrodynamic pressure during impact. Finally, an airbag electronic switching unit was mounted to the floor in the rear of the cabin to indicate the time at ignition by turning on a lamp. Approximately 90 channels of data were collected at 10,000 samples per second using a digital data acquisition system, including 62 accelerometers, 11 displacement transducers (string potentiometers), 5 loads cells (one lumbar and 4 seat restraint), 4 pressure transducers, 1 radar, and 1 continuity strain gage. A schematic drawing of the helicopter indicating the locations of selected accelerometers used in the analytical/ experimental correlation is shown in Figure 5.

Figure 5. Selected accelerometer locations on the ACAP helicopter.
A double-harness support was used to raise the helicopter to a height of approximately 40 ft. at the IDRF. The helicopter was dropped in a pendulum fashion, to achieve the impact conditions of 38 ft/s vertical velocity and 32.5 ft/s longitudinal velocity, with a +6.25° (nose up) pitch and a 3.5° roll (left) onto a rigid impact surface.

All acceleration data from the ACAP crash test were analyzed and checked for polarity errors, zero-offsets, and noise. Floor acceleration data in the vertical direction were integrated to obtain the vertical velocity change. The integration also provided a quality check of the data. Those channels in which the integrated velocity change varied greatly from the nominal vertical impact velocity were not used for the correlation with the analysis.

In general, high quality experimental data was obtained from the crash test. However, this test was the last one performed using the umbilical based data acquisition system at the IDRF. In addition to the problems incurred with using a very long umbilical cable of several hundred feet, more noise than usual was noted in this test. In severe tests such as the ACAP helicopter crash test, it is not unusual for wires to be cut and for anomalous electrical transients to be generated in the data. Several acceleration traces were noted to have a number of high-amplitude transients, and to have exhibited a zero offset after the test. The channels that showed large zero offset differentials were not used for comparison with the analysis.

**Analytical and Experimental Correlation**

*Correlation with time sequence of events*

An important correlation between the analytical and experimental data is a comparison of the predicted and measured time sequence of events. The comparison, shown in Table 2, indicates that the simulation correctly predicted the timing of right gear contact, nose gear contact, tail cone failure, and the peak accelerations of four different locations on the helicopter. In most cases, the simulation predicted the event timing within seven milliseconds. It should be noted that events occurring earlier than 0.045 seconds were predicted using the rigid model, while events occurring after 0.045 seconds were predicted using the flexible model.

<table>
<thead>
<tr>
<th>Event</th>
<th>Simulation (seconds)</th>
<th>Measured test result (seconds)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Left main gear contact</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>Right main gear contact</td>
<td>0.016</td>
<td>0.012</td>
</tr>
<tr>
<td>Nose gear contact</td>
<td>0.068</td>
<td>0.069</td>
</tr>
<tr>
<td>Tail breaks</td>
<td>0.075</td>
<td>0.074</td>
</tr>
<tr>
<td>Time of peak accel.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Left troop seat floor</td>
<td>0.110</td>
<td>0.105</td>
</tr>
<tr>
<td>Rt. troop seat floor</td>
<td>0.114</td>
<td>0.107</td>
</tr>
<tr>
<td>Copilot floor</td>
<td>0.115</td>
<td>0.110</td>
</tr>
<tr>
<td>Right engine</td>
<td>0.117</td>
<td>0.116</td>
</tr>
</tbody>
</table>

Comparisons of the predicted helicopter motion with high-speed film pictures of the actual crash scenario are shown in Figure 6 for time equal to 0.0, 0.05, 0.1, and 0.15 seconds after left gear contact. The predicted motion of the helicopter agrees well with the observed crash scenario. In addition, the simulation accurately predicts the location and time of tail cone failure.

*Correlation between model and structural deformation*

Several discrete structural failures were noted to occur during the crash test, including failures of the ceiling beams in the fuselage cabin, support beams for the overhead engines and transmission, and the separation of the tail from the fuselage. In general, the crash simulation predicted the location of these failures. For example, the undeformed model of the bulkhead at BS 188 is shown in Figure 7 (a). This bulkhead is located behind the rear of the cabin and beneath the overhead rotor transmission. The model of the bulkhead consists of a ring of shell elements representing the outer skin, with one row of horizontal shell elements representing the floor. The two support beams that form the “V” in the center of the bulkhead were modeled with beam elements and are illustrated with dark lines. The failure of one of the support beams, as predicted by the model at time 0.125 seconds, is shown in Figure 7(b). The buckling of the lower portion of the subfloor is also apparent in the figure. An example of a support beam failure from a post-test photograph is shown in Figure 7 (c).
One of the major goals of this modeling effort was an accurate prediction of the subfloor crushing response. Unfortunately, the displacement transducers that were used to measure the dynamic subfloor deformation failed to perform during the test. Consequently, only the post-test floor deformation was available for comparisons with predictions. Since no dynamic crush was measured, only a qualitative comparison can be given. The predicted crush of the lower Kevlar portion of the subfloor is shown in Figures 8 (a) and (b). For clarity, the upper non-crushing graphite portion of the subfloor is not shown. The initiation of crushing at BS 188 at time 0.105 seconds, which is shortly after contact of the bottom of the helicopter with the impact surface, is shown in Figure 8 (a). At time 0.125 seconds, all of the inner keel beams exhibit buckling and crushing, as indicated in Figure 8 (b). An examination of the post-test ACAP fuselage showed that the Kevlar subfloor had permanent crush of about 0.25-inch at BS 188. The crush increased forward to a maximum of 1.5 inches at BS 143. The permanent crush at BS 122 ranged from 0.5 to 1.0 inches. A photograph of the subfloor from BS 188 through BS 143 is shown in Figure 8 (c) that highlights the actual post-test deformation.

Rigid Body Model Results

A comparison of predicted and measured vertical velocities for three locations in the helicopter including the top of the right gear, the right engine location, and the copilot seat floor location is shown in Figure 9. The predicted responses were obtained from the rigid structural model, and the experimental responses were obtained from integration of the accelerometer data. The rigid model required approximately 1.8 CPU hours to simulate 0.2 seconds “real time” on a Sun Ultra Enterprise 450 computer. The rigid fuselage model impacts the contact surface at approximately 0.1 seconds. Thus, the rigid results are only applicable for the first 0.1 seconds of simulation. Since the ACAP impacted with a pitch angular velocity, the vertical velocity at a given location is the sum of the center-of-gravity (cg) velocity plus or minus the rotational component. Since the helicopter had a negative pitch rate at impact, the vertical velocity at the copilot cabin floor is actually less than the engine and gear vertical velocity, which is near the cg-location. As can be seen in Figure 9, the model follows the measured motion of the ACAP helicopter quite well.

The gear forces, predicted by the rigid body simulation for the main and nose gear, are shown in Figure 10. The left gear tire impacts at time zero, the right gear at time 0.16 seconds, and the nose gear at time 0.68 seconds. These
Figure 7. Model of bulkhead at BS 188 and post-test photo showing failure of support beam.
(a). Model of the lower portion of the subfloor near the fuselage impact point at 0.105 s.

(b). Model of the lower portion of the subfloor region at 0.125 seconds after impact.

(c). Photograph of the ACAP helicopter subfloor between BS 122 and 188.

Figure 8. Deformed plots of the fuselage Kevlar subfloor beams and a post-test photograph showing actual damage and deformation.
times correlate well with the measured values shown in Table 2, which are 0.00, 0.12, and 0.69, respectively.

Flexible Model Results

The flexible model was initiated at 0.045 seconds using the grid point position and velocity data from the rigid model. To run the flexible model for 0.1 seconds real time (from 0.045 to 0.145 seconds) required approximately 12 CPU hours on a Sun Ultra Enterprise 450. The flexible model is computationally intense compared with the rigid model. To conserve resources, a flexible simulation such as this one with a time step of approximately one microsecond would normally be started a fraction of a millisecond before fuselage impact. However, to include the tail failure in the model, which initiates at about 0.025 seconds before fuselage impact with the surface, the flexible model had to be started approximately 0.05 seconds before fuselage impact.

Figure 9. Comparison of predicted rigid model velocities with experimental data.

Figure 10. Predicted right and left main gear and nose gear axial force time histories.

The flexible model included the effects of gravity, however friction of the impact surface was not included. Friction is difficult to measure experimentally, and would only be significant after fuselage contact. Since friction was not modeled, no results are shown for the longitudinal or lateral directions. The correlation between the flexible model z-velocity and the measured vertical velocity for the three locations that were tracked by the rigid model (the top of the right main gear, the
Figure 11. Comparison of predicted flexible model velocities with experimental data.

right engine, and the copilot seat floor location) are shown in Figure 11. As would be expected, the flexible model shows vibratory oscillations, even more than measured experimentally. The magnitude of the oscillations was enhanced due to the impulse of the landing gear loads being applied at full load at start-up of the flexible simulation. The transition from the rigid to the flexible model is not trivial and can cause considerable problems if not properly administered.

The comparisons of the experimental and predicted vertical velocity responses of the right engine, right main landing gear, and copilot floor for both the rigid and flexible models, shown in Figures 10 and 11, respectively, were performed to validate the two-stage modeling approach. This comparison demonstrated that equivalent results were obtained for both the rigid and flexible models for the period before fuselage impact, up to 0.1-seconds after left gear contact. The fact that the analytical results obtained from both the rigid and flexible models demonstrated the same high level of agreement with the experimental data indicated that the two-stage modeling approach was successful.

Comparison of Predicted Acceleration Time Histories with Data

To remove the high-frequency ringing from the low-frequency crash pulse, both the experimental data and the predicted accelerations were filtered using a zero-phase digital 60-Hz low-pass, 3rd order Butterworth filter generated by the mathematical code Matlab. To produce zero-phase distortion in time, the algorithm used both forward and backward filtering.

A major goal of this modeling effort was an accurate prediction of the floor-level accelerations since this information is needed for the proper design of crashworthy seats. The predicted and measured vertical acceleration responses for the pilot and copilot seat floor locations are shown in Figures 12 (a) and (b), respectively. The experimental data is shown from time zero, corresponding to the time of left gear contact. The predicted values were obtained from the flexible model, which initiated at 0.045 seconds. The initiation and duration of the two acceleration pulses compare quite favorably. The predicted acceleration peak for the pilot was 115-g, while the experimental value is 90-g. For the copilot, the predicted peak acceleration also exceeded the experimental, with about 115-g predicted and with 83-g measured peak accelerations. The over
prediction of the peak acceleration for the pilot and copilot suggests that the model is more elastic than the actual structure. For simplicity and robustness of the model, the Kevlar subfloor material properties were represented as elastic-plastic without failure. To obtain more accurate results, the Kevlar subfloor model may require more refined meshing and a more accurate material model. In addition, since energy is proportional to velocity squared, the rotational slap-down velocity of the nose portion of the helicopter must be modeled accurately to predict the actual accelerations more closely.

In contrast to the predicted pilot and copilot accelerations, which exceeded the measured values, the predicted peak acceleration for the right troop seat, shown in Figure 13 (a), was slightly low at 64-g as compared with the experimental value of about 75-g. As shown in Figure 13 (b), the predicted peak acceleration of the left
troop seat floor, 51-g, was about 40% lower than the measured acceleration, which was 85-g. The explanation for this discrepancy was not readily apparent.

Predicted and experimental acceleration responses for the floor at BS 182 are plotted in Figure 14. The predicted acceleration pulse is quite similar in shape to the experimental pulse with a peak acceleration of 54-g compared with the measured value of 61-g. The predicted peak lags the experimental peak in time by about 0.01 seconds. The acceleration time histories for the right and left engine masses are given in Figure 15 (a) and (b), respectively. The predicted peak acceleration for the right engine matched the experimental value of 41-g, although the response was slightly out of phase in time. The predicted peak for the left engine is matched with the experiment in time, but is slightly lower in magnitude.

![Figure 14. Predicted and measured acceleration responses of the floor BS182 location.](image)

![Figure 15. Predicted and measured acceleration responses for the right and left engines.](image)

The predicted and measured vertical acceleration responses of the top of the right main landing gear, the bulkhead at BS 255 which is just in front of the main gear, and the top of the bulkhead at BS 188, are shown in Figures 16, 17, and 18, respectively. The comparisons are favorable, except for the bulkhead acceleration at BS 188. The predicted acceleration response at BS 188 is of shorter duration and higher magnitude than the measured acceleration. These results indicate the model was too stiff in this location. A large number of vertical rod elements were used to model the fuselage wall and bulkhead at BS 188 in the original modal-vibration model. Similar rod elements were used in the subfloor as well. For the crash model, the rod elements in the subfloor were either removed or replaced with beam elements, since rod elements are too stiff to represent the actual structural crush behavior. It is important to note that the rod elements at BS 188 were not removed or modified, which resulted in the high acceleration shown in Figure 18.
Concluding Remarks

A finite element model of the Sikorsky Advanced Composite Airframe Program (ACAP) helicopter was developed using the nonlinear, explicit transient dynamic code, MSC.Dytran. Analytical predictions were correlated with experimental data obtained from a full-scale crash test of the Sikorsky ACAP helicopter flight test article, that was conducted at the Impact Dynamics Research Facility of NASA Langley Research Center in 1999. The helicopter was impacted at 38-ft/s vertical and 32.5-ft/s forward velocity with an attitude of 6.25° pitch (nose up) and 3.5°-left roll at impact.

The objective of the crash simulation was to evaluate the capabilities of a commercially available transient dynamic code in predicting the response of a composite airframe subjected to impact loading through extensive analytical/experimental correlation. The crash model of the ACAP helicopter was developed from an existing MSC.Nastran modal-vibration model of the helicopter. Considerable modifications were required to convert the original modal-vibration input deck to an MSC.Dytran model for crash simulation.

Following the model conversion and modification, a two-stage rigid-to-flexible modeling approach was used to generate analytical predictions. Due to the relatively long pulse duration, a rigid fuselage model was executed in which an external subroutine was used to represent the response of the landing gear, including both the oleo-pneumatic and crushable honeycomb...
stages. Approximately 0.05 seconds before fuselage contact, the nodal displacements and velocities from the rigid fuselage model were output. Then, a flexible structural model was executed with the nodal displacements and velocities used as initial conditions. This two-stage modeling approach worked well except for a slight discontinuity that occurred when the flexible model was initiated due to the sudden onset of the landing gear forces. Comparisons of the experimental and predicted vertical velocity responses of the right engine, right main landing gear, and copilot floor were performed for both the rigid and flexible fuselage models to validate the two-stage modeling approach. This comparison demonstrated that equivalent results were obtained for both the rigid and flexible models for the period before fuselage impact; i.e., up to 0.1-seconds after left gear impact.

The initial correlation between the finite element simulation and the experimental results was a comparison of the time sequence of events. In general, the simulation accurately predicted the time of right main gear contact, nose gear contact, tail cone failure, and the timing of the peak accelerations of the left and right troop seat floor, copilot floor, and the right engine locations.

A comparison of the predicted and experimental acceleration responses for the pilot and co-pilot floor, the right and left troop seat floor, the right and left engine, right main gear, and several other locations on the fuselage were made. For this comparison, both the experimental and predicted acceleration data were filtered using a three-pole Butterworth 60Hz low-pass filter. In general, the simulation accurately predicted the overall shape, magnitude, and pulse duration for each of the experimental acceleration responses mentioned previously. The predicted peak acceleration at the upper bulkhead location was substantially higher than the experimental data at that location. The discrepancy was attributed to the large number of rod elements in the bulkhead that were unmodified from the original Sikorsky modal-vibration model. Typically, rod elements are extremely stiff and do not accurately reflect the response of the actual bulkhead beams.

In addition, the crash simulation correctly predicted the subfloor crushing response and structural failures of the engine support beams. Overall, the high level of agreement obtained between model and test should build confidence in the future use of nonlinear, explicit transient dynamic finite element codes as a crashworthy design and evaluation tool for aircraft.

**Acknowledgements**

The authors would like to recognize several organizations and companies for their support of this research program. The ACAP helicopter was provided by the US Army Aviation Applied Technology Directorate, located at Ft. Eustis, Virginia. Financial support for the full-scale crash test and simulation activity was provided by the Aviation Safety Program of NASA Langley Research Center. The MSC.Nastran modal-vibration model of the ACAP helicopter was provided by Sikorsky Aircraft Company. The authors would like to acknowledge Charles Clarke and Joseph Shen of Sikorsky Aircraft for their many helpful discussions. Finally, the authors wish to acknowledge the excellent technical support provided by the MSC.Software Company.

**References**

5. PAM-CRASH, Engineering Systems International SA, F-94588 Rungis, France


