

Simulation of Ballistic Impact on Composite Panels

Matti Loikkanen

matti.j.loikkanen@boeing.com

*Propulsion Engineering Technology & Research,
Boeing Commercial Airplanes, Seattle, WA 98124*

Gramma Praveen

Global Research Center,

General Electric Company, Niskayuna, NY 12309

David Powell

*Mechanical Engineering Department,
University of California, Berkeley, CA 94720*

Abstract

Ballistic impact on composite panels is studied in this work both experimentally and computationally. The purpose was to develop computational methods to analyze a high speed jet engine fragment impacting on composite targets. 12 inch by 12 inch laminated panels with 8, 16, and 32 plies and the standard +-45, 0 /90 degree stacking sequences were considered. With the nominal ply thickness of 0.0075 inches, the corresponding panel thicknesses were 0.06, 0.125 and 0.25 inches. The panels were mounted on a heavy steel frame and spherical and cylindrical projectiles were shot against composite plates. Several shots with varying impact speeds were fired against each panel thickness. The impact damage was observed, and the initial and exit speeds were measured. The ballistic tests indicated that the amount of energy absorbed during impact by a target is nearly constant showing only a slight increase with increasing initial energy. The amount of energy absorbed per ply increases only slightly for the thicker samples. In addition, the tests showed that the cylindrical projectiles required a larger amount of energy to penetrate the composite panels than did the spherical projectiles

LS-DYNA® was used to simulate the tests. The panels were modeled with 8-node solid elements. *MAT_COMPOSITE_DMG_MSC (162) was used to model the orthotropic ply material. This model can be used to model progressive failure of composites with unidirectional and woven fabric fibers. One layer of solid elements was used through the thickness of each ply and several mesh densities were studied. A new Cohesive Contact formulation *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK (and ONE WAY TIEBREAK) made available as a "DYCOSS" option was defined between each ply to model delaminations. The cohesive contact - DYCOSS option 9 has been developed during this work. This delamination modeling feature is based on fracture mechanics and requires fracture toughness inputs for the composite material. Option 9 has both the power law and BK-law to account for Mode I and Mode II interaction and allows for a better definition of the constitutive laws. The main advantages of cohesive contact are that it allows the user to toggle very easily between ordinary tie-break based delamination models and cohesive contact models and there is no need for separate cohesive elements in the model. Good overall agreement was found between computations and testing.

Keywords: finite element analysis, ballistic impact, composite structures, ply failure, penetration, delamination, absorbed energy, exit velocity, cohesive contact

Introduction

Over the past several years The Boeing Company in cooperation with the Federal Aviation Administration (FAA) and University California, Berkeley (UCB) and other research facilities has conducted material testing and numerical computations on ballistic impact on various airplane structures. FAA has funded and administered the work, with UCB mainly performing the testing and Boeing the computations. The purpose has been to develop computational methods to analyze a high speed fragment impacting on the chosen targets. Material testing had two aspects: small coupon tests to determine the material properties needed as input to the computer program and ballistic testing where a projectile was shot against a target specimen, where penetration, impact and exit velocities were measured. LS-DYNA® (from hereon referred as DYNA) finite element (FE) models were created and analyzed and the computational results compared with the test results. Once a reliable computational procedure has been developed, then it is much cheaper and faster to use computer simulations to replace further and extensive laboratory testing. In addition, computer analysis yields much more information than can be measured in a test and this makes it possible to understand the underlying physics better, which in turn makes the design process more accurate and faster. Many factors affect the accuracy of a DYNA ballistic simulation but it is absolutely necessary to rely on some testing when the simulation methods are developed [Ref. 1]. The accuracy of the simulation depends on the chosen material model, the material data, especially the failure characteristics and in the case of composites structures this includes the delamination data, the element types (4-node shell vs. 8-node brick), the mesh density, the contact definition input (such as the frequency of the bucket sorting), and finally on the DYNA control input.

Ballistic testing

Boeing composite material designated as BMS8-212 was used in the study [Ref. 4, 5]. The technical data of this material is in public domain and therefore could be used in a publicly funded FAA project. Flat panels with three different thicknesses, 0.06 in, 0.12 in and 0.24 in were tested. The nominal ply thickness is about 0.0075 in and so the panels had 8, 16, and 32 plies, respectively. The stacking sequence, for the 16 ply panel, starting from the bottom is: 45, 90, -45, 0 / 45, 90, -45, 0, /center/ 0, -45, 90, 45 / 0, -45, 90, 45 degrees, i.e., these are the ply rotations from the reference axis. The stacking is symmetric about the center plane. In addition, the 4-ply sequence 0, -45, 90, 45 - repeats itself around the center plane. The 32-ply panel has four 4-ply stacks both above and below the center plane in the same pattern. The overall panel with all three thicknesses with this stacking sequence has almost isotropic bending and in-plane stiffness properties (therefore called “quasi-isotropic”). This is actually a common design. Due to the symmetric lay-up, the bending and in-plane stiffness are also uncoupled. The total size of each panel in the tests was 12x12 inches and when mounted onto a steel test frame, the clear test area was 10x10 inches. The mounting steel frame was stiff enough so that the plate boundary was considered clamped in the 10 by 10 in² FE models. With the given target size and projectile speeds, the reflecting waves from the boundary reach the impactor before complete penetration

A nitrogen gas gun with half-inch diameter spherical steel projectiles was used in all the tests shots. The gun used industrial grade compressed nitrogen. The gun was able to propel the

projectiles up to 1000 ft/s speed, which was high enough for complete penetration with the 32-ply panels. The advantage gas gun over powder gun is that the pressure and the projectile velocity can be controlled more precisely.

Two different projectiles were used in the study: a ½ inch diameter steel ball and a ½ inch diameter by a 1 inch long steel cylinder. The impact speeds ranged from 100 ft/sec up to 900 ft/sec. All the shots were perpendicular and at the center of the plate. The impact and exit (residual, after penetration) velocities were measured. Each shot was videotaped with a high speed camera and the damage and hole to the panels were photographed.

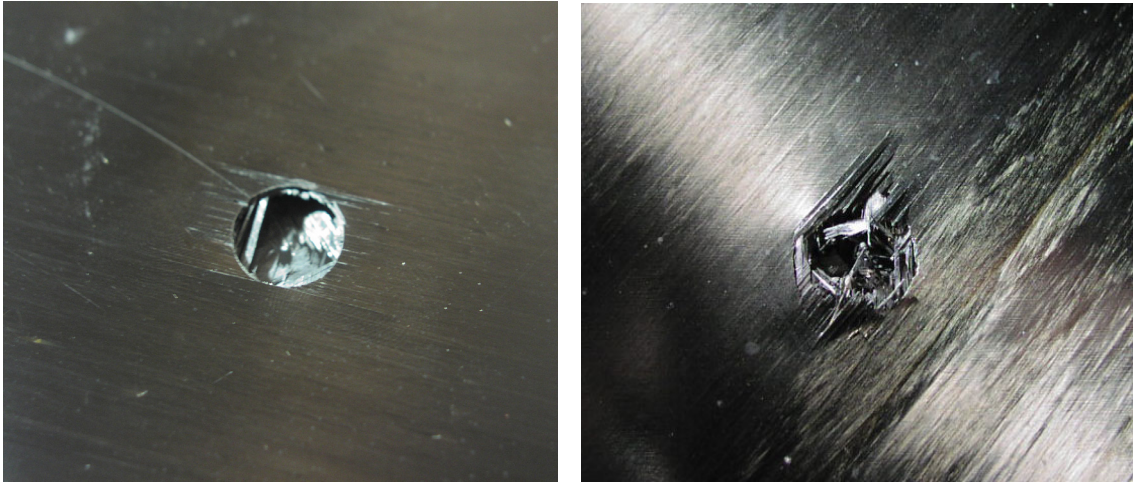


Figure 1: Impact and exit side damage from spherical impactor

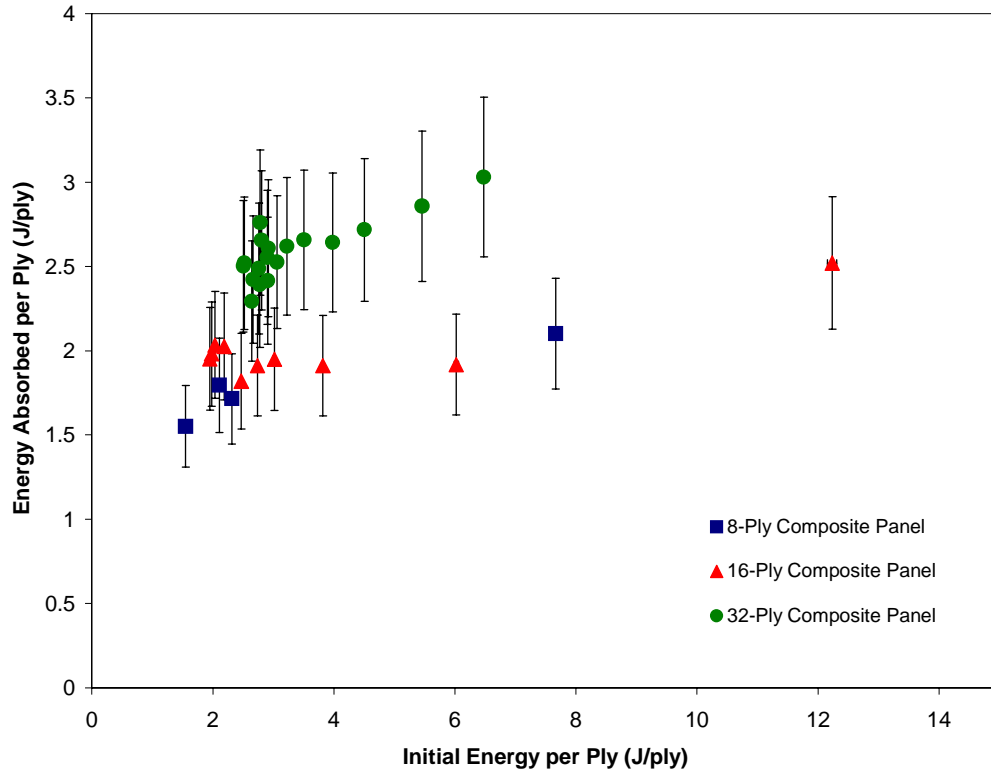


Figure 2: Energy absorbed per ply for spherical projectile

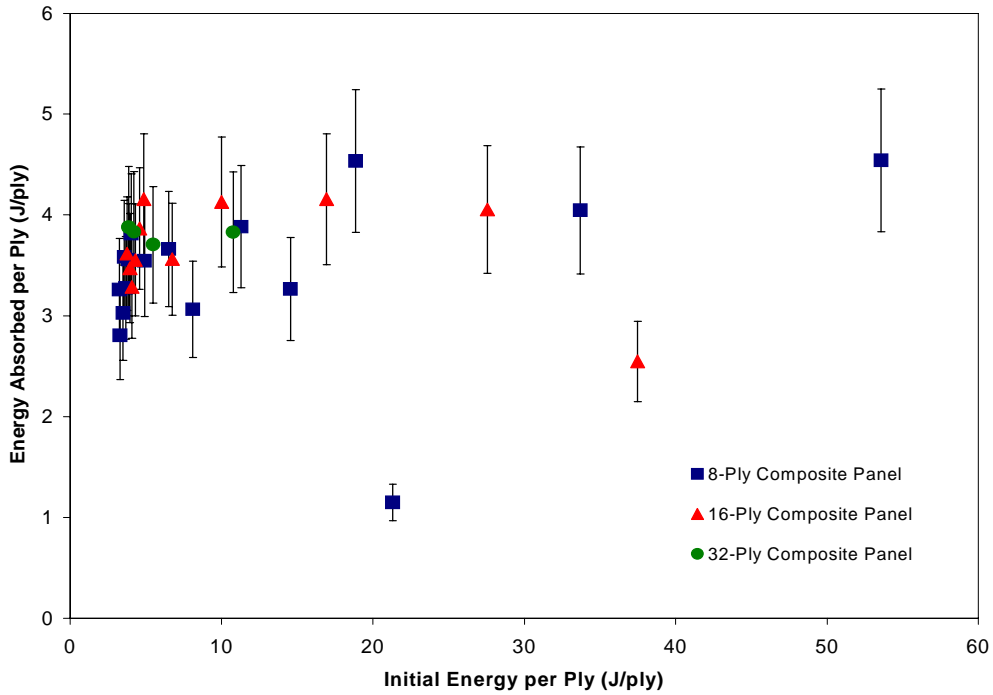


Figure 3: Energy absorbed per ply for cylindrical projectile

Figure 1 shows typical impact and exit holes for the cylindrical projectile. The measured impact and exit velocities were plotted and shown in Figures 9 and 10 (along with the computed exit velocities). The energy absorbed per ply for spherical projectile is shown in Figure 2 and for cylindrical projectile in Figure 3. These figures show that the energy per ply dissipation is almost constant regardless number of plies in the laminate. Furthermore, comparison shows that the cylinder takes much more energy to penetrate than the sphere. In other words the projectile shape is a big factor in ballistic penetration at the speed range considered here [Ref. 4, 5]. It was not possible to measure the amount of delamination in the test specimens with the available equipment.

In addition to ballistic testing, program consisted limited amount of static testing for lamina strength in all directions. Some testing was also made to get the dynamic fracture toughness of the bonding between plies. (Presently ASMT does not have specification for dynamic fracture toughness testing.)

Computer Simulations

Finite element models of both the target and projectile were made and the impact and penetration events were simulated with DYNA [Ref. 2, 3 and 5]. One half of the 10 by 10 in² target plate was modeled with clamped boundaries around the plate and symmetry boundary in the middle. Similarly one half of the projectile was modeled. Composite laminate is not necessarily symmetric around the middle line, but the experience has shown that for impact loading perpendicular to the plate is it almost irrelevant how the plies are rotated. Therefore it is safe to assume that half symmetric model is accurate to use.

As the projectile diameter is only 0.5 inches and the laminate thicknesses is from 0.006 to 0.25 inches, the deformation under penetration very localized. Therefore, each ply was modeled explicitly with 8-node solid elements; one element through the thickness (0.0075 in) of the each ply. It would have been desirable to have several elements through each ply, but given the ply thickness, this was not practical. Each ply is a separate part in the model and there is no nodal connectivity between the plies. The plies are connected together with tiebreak contact to allow delamination between the plies (this is explained later). The best element size was found by trial and error to be about 0.05 by 0.05 in². Smaller elements tend to fail too early and dissipated energy is less than measured in the test. Larger elements fail too late and absorb too much energy. The optimal element size do not depend only on the laminate thickness, plate material characteristics, but also on the projectile size, shape and the speed. Therefore, it may be necessary to get new test data when any of these conditions change.

The ply failure is fully three dimensional. All the stress/strain components interact in the failure and most elements under the projectile have significant hydrostatic pressure at the failure. We found that for high speed impact problems, *mat_composite_dmg_msc (type 162) works well - if not the best - of all the available composite material models in DYNA. The material data consists of the stiffness values and tensile and compressive strength values in all three directions. The fiber model can be either unidirectional or woven fabric. The progressive failure is based on the method developed by Hashin [Ref. 6]. The model allows also delamination by allowing the material to fail in the through thickness tension (c-direction). Since there is now a more accurate

way to simulate delamination (explained in next chapter), the through thickness tensile strength (STC) was set to artificially high value to prevent the through thickness ply tensile failure.

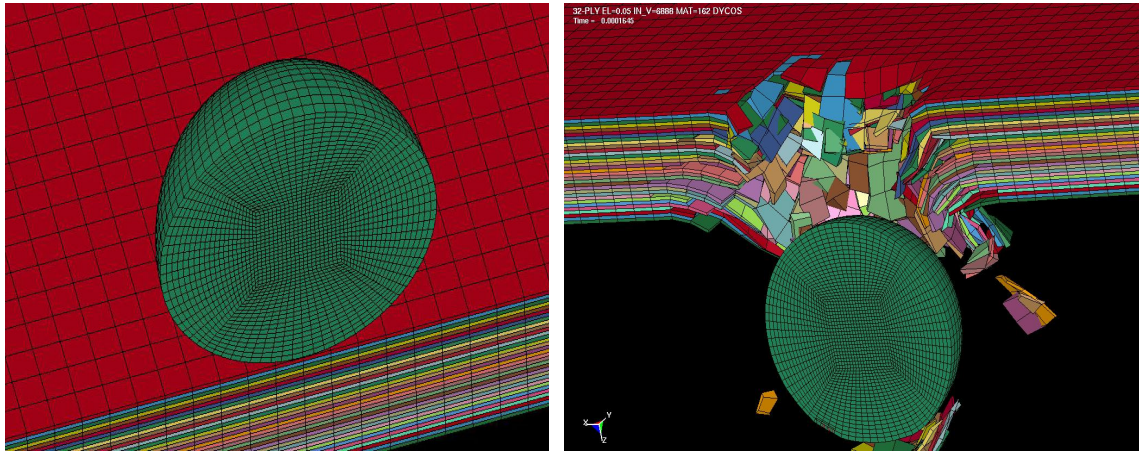


Figure 4: 1/2 inch sphere impacting on 32-ply composite panel

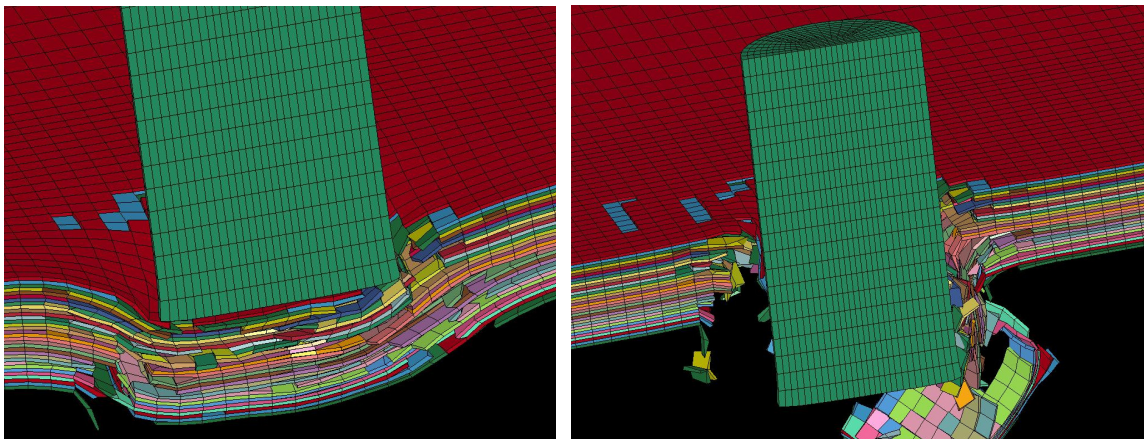


Figure 5: 1/2 inch cylinder impacting on 32-ply composite panel

The limits of volume strain for element erosion were set to “reasonable” values and the viscoelastic analysis was ignored (IVISCO=0). Softening due to the progressive failure follows an exponential curve where the user can define the exponent AM1 ... AM4. We chose 32 for all these. Hourglass control type 2 with default coefficient 0.1 was used. To keep the runs stable the time step scale factor (TSSFAC) had to be reduced to 0.1.

Cohesive Contact Delamination Modeling

The original tie-break models in DYNA are very primitive in that they do not account in any manner for the fracture toughness of the interface. Recently cohesive element models have been developed to properly account for crack initiation, energy dissipation, crack growth and mode mixity based on empirical criteria. Cohesive elements to model delaminations are typically zero thickness “solid” elements with the deformation being governed by the cohesive traction-displacement constitutive law. The simplest form of the traction-displacement constitutive relationship has a linear elastic response till the crack initiation criterion is reached and then followed by a linear softening to zero traction when the damage is complete, i.e. the shape is triangular [Ref. 7]. The area under this bilinear traction-displacement curve is the fracture toughness in single (pure) mode. The penalty stiffness is the slope of the loading part of the traction-displacement relationship. Typically, the user specifies the stiffness in the normal and shear modes independently. Mixed mode (I and II) crack growth is accounted by quadratic stress based interaction law (strain based laws could also be used). The crack growth is based on an empirical relationship involving the energy release rates and fracture toughness of the interface.

A damage parameter (α) is also defined such that it evolves irreversibly from 0 (zero) at crack initiation to 1 (unity) at full interface failure. The tractions at any intermediate damaged state are a simple linear function of the damage parameter. The irreversibility condition, $d(\alpha) \geq 0$, ensures that the state of damage is preserved under unloading and evolves from that state under re-loading. Thus the cohesive element constitutive and evolution equations properly account for the strength and energetics associated with crack growth. However, one of the drawbacks about the cohesive elements is that existing models will have to be changed by adding layer of 0 thickness cohesive elements at each tie-break interface. This leads to larger models and a cumbersome reconstruction of existing models. In order to obviate the need for explicitly defining zero-thickness cohesive elements while still retaining all the constitutive constructs, the authors proposed the enhancement of existing tie-break contacts within DYNA to include all the aspects of standard cohesive elements. The authors along with LSTC developed new cohesive contact formulations to model delamination (tie-break, option 9). The use of cohesive contact formulation is straightforward and similar to the classical tie-break contact specification. It only requires a few additional input parameters such as fracture toughness in pure mode I, and mode II, the interfacial stiffness for normal and shear modes, and a parameter for the power-law or the BK-law that describes crack propagation. The interfacial strengths are specified in the same manner as regular tie-break. Consequently existing tie-break based FE models need only minimal changes to take advantage of cohesive contact models with full fracture mechanics based capabilities.

Simple Mode II simulations (ENF) were performed to evaluate the performance of cohesive contact models [Fig. 6]. The simulations were used to compare the predictions of existing tie-break contact, cohesive elements in ABAQUS, cohesive elements in DYNA and cohesive contact in DYNA. The specimen was modeled with 24 layers 0 deg plies (along the specimen length) with the crack between the 11 th and 12 th layers. Mid-plane cracks were also simulated. The early DYNA simulations showed that as soon as the crack initiation criterion was satisfied, the crack grew in an unstable manner along the entire length of the specimen. It was realized that the transverse impact on the specimen induced spurious oscillation in the contact penalty springs and caused premature interface failure. The remedy was to artificially dampen out the

spurious oscillation between the contact interfaces. Finding the correct way to accomplish this was a trial and error process.

Damping elements were introduced within the tie-break layer to eliminate the effects of the spurious oscillations. The damping constant was varied across several orders of magnitude and it was found that at a value of 0.01, excellent agreement was achieved between typical test results, ABAQUS cohesive element model, and DYNA cohesive contact model results. ABAQUS simulations successfully captured the post peak behavior seen in the tests. The DYNA simulations were able to match ABAQUS predictions when the interface damping feature was added. Several other damping constructs were also tried such as varying the contact damping factor “vdc” within the contact card (fraction of critical damping) or by adding a damping force proportional to the velocity of the slave nodes. The results from these simulations were reasonable. The best agreement was found when the explicit one-dof viscous dampers were added between the opposing nodes of the contacting surfaces. However, more work is needed to fully understand the role of the contact damping. On the one hand the crack growth has to be stable, but on the other hand, the damping cannot be artificially high lest it distort the dynamics.

The mode II specimen is typically 6 inches long, with a 1.65 in initial crack (delamination) along the length of the specimen. The supports are 4.7 in apart. This results in an initial crack length of 1 inch in the supported region. An impactor drops vertically on the mid-span location, at a drop velocity of 5 fps and shear force due to bending causes the crack to grow in pure mode II. The experiments are modeled both in DYNA and ABAQUS and comparisons are made between the models [Fig. 7]. Based on the success with explicit 1-dof damper elements, the authors along with LSTC implemented two other damping approaches – the first, using the standard “vdc” (fraction of critical damping) that is available in all contact cards, and second, by applying a damping force proportional to the slave node velocity in each direction. The results from each of these cases showed improvement compared to the initial baseline run, but were not as accurate as the model with the explicit dampers. The authors recognize that the role of this explicit damper needs to be more clearly understood. The results have been presented to show that damping plays a critical role in modeling delamination growth in impact related problems using DYNA. The results from the mode II simulation are presented with the explicit damper element, and also with the “vdc” damping input and internal damping proportional to the slave node velocity.[Fig. 8]. These preliminary results suggest that modeling delaminations with cohesive contact shows great promise [Ref. 7]. The main advantages of cohesive contact are that it allows the user to very easily toggle between ordinary tie-break based delamination models and cohesive contact models and there is no need for separate cohesive elements in the model. Based on computational results, the authors recommend the use of the one-way tie break contact .

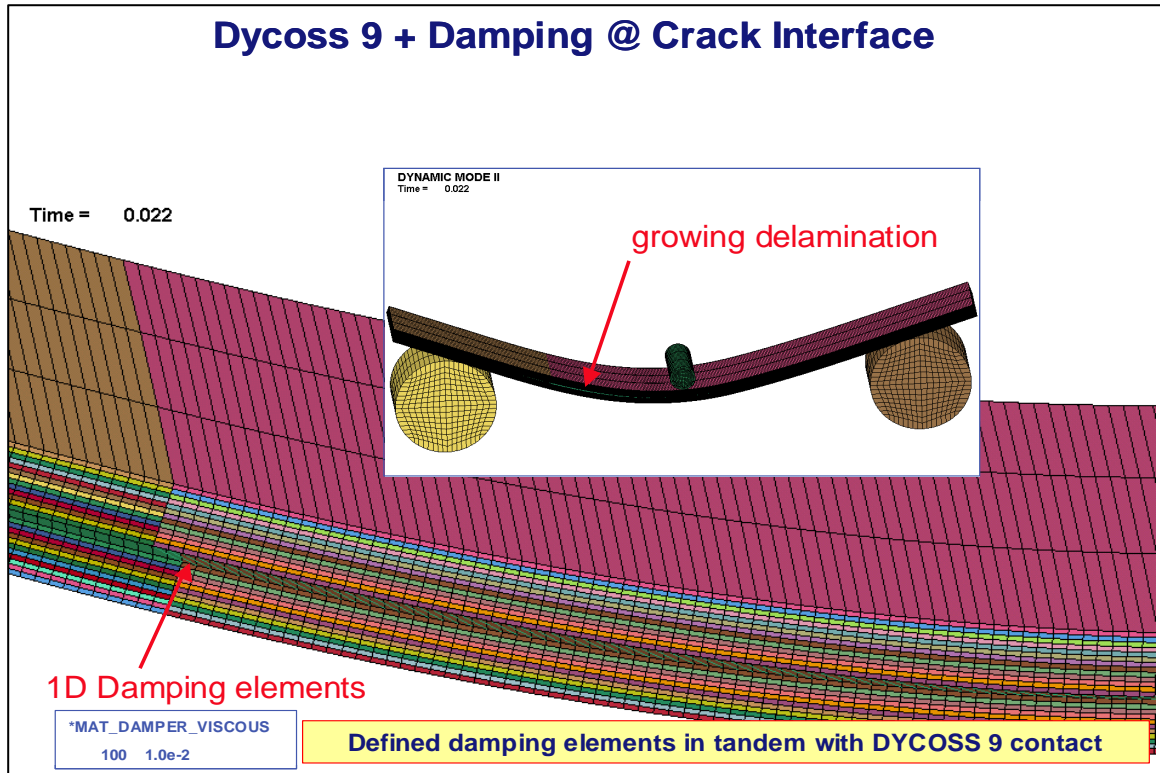


Figure 6: Simulation of Mode II fracture toughness

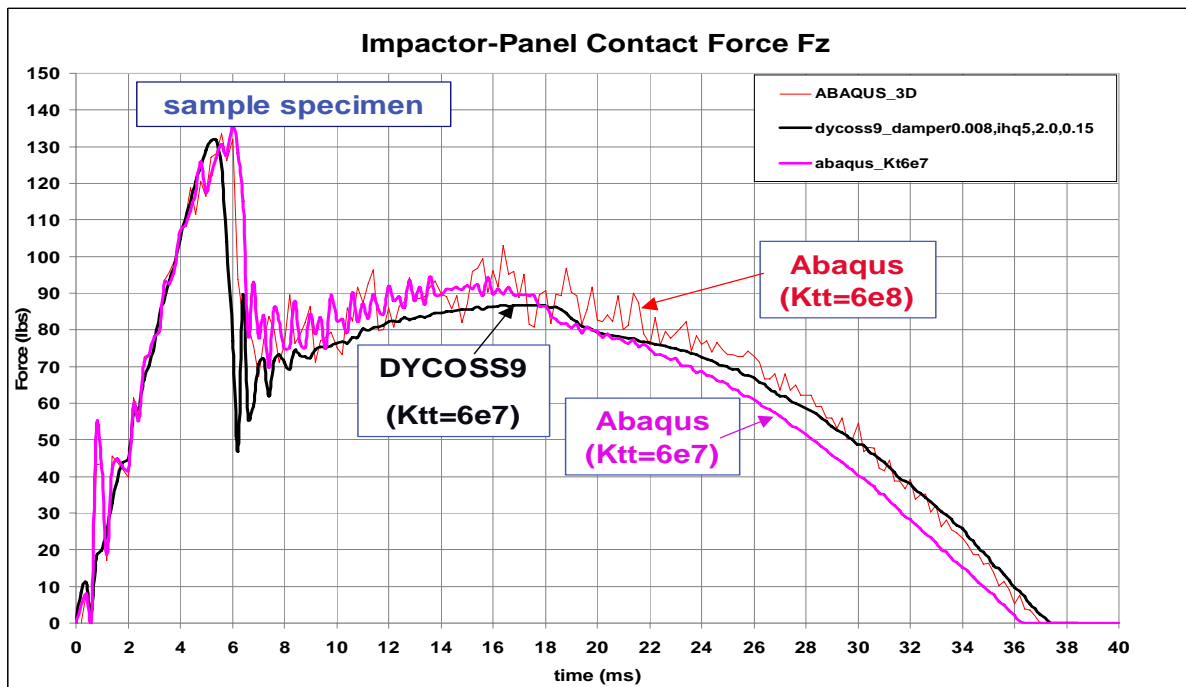


Figure 7: Mode II Test simulation results using LS-DYNA and ABAQUS

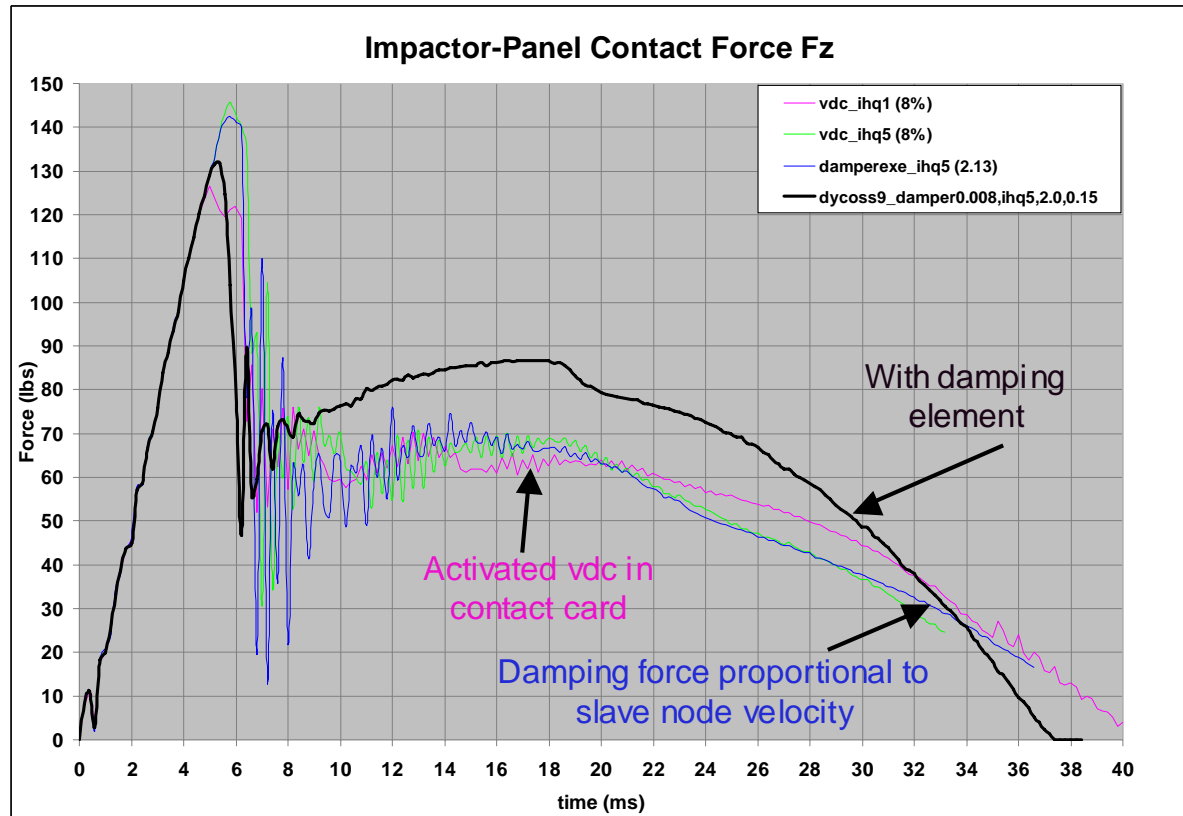


Figure 8: Mode II delamination results using explicit damping element in LS-DYNA

The computational results along with the experimental results for the ballistic plate impact are summarized below in Figures 9 and 10. Impact speed is on the horizontal scale and the exit speed [Ref. 1]. Triangles indicate the shots on the 32-ply planes, squares represent the 16-ply panel, and spheres represent for the 8-ply panel. Black markers represent the DYNA simulations and light gray markers represent the test results. The dotted line is the 45-degree reference line when the impact and exit speeds are the same and the vertical distance between the line and the data point represent the dissipated energy.

Typical failure patterns are shown in Figures 4 and 5. Plies fail under the impactor due to the combination of all the stress/strain components including relatively large hydrostatic pressure. Some delamination seems to develop around the hole, but at the present it was not possible to quantify this. (LSTC is enhancing PREPOST so that TIEBREAK option 9 delamination in the future can be visualized.)

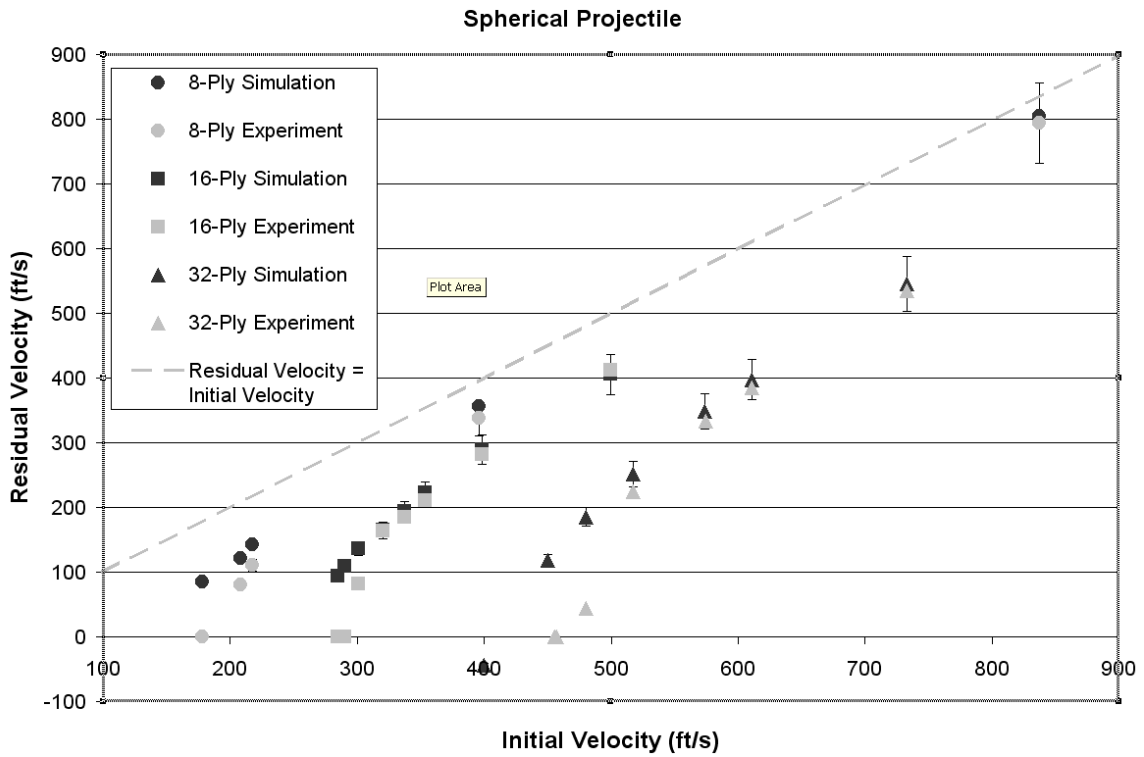


Figure 9: Computed vs. tested exit velocities for 8-ply, 16-ply and 32-ply composite panels impacted with 1/2 in diameter steel sphere

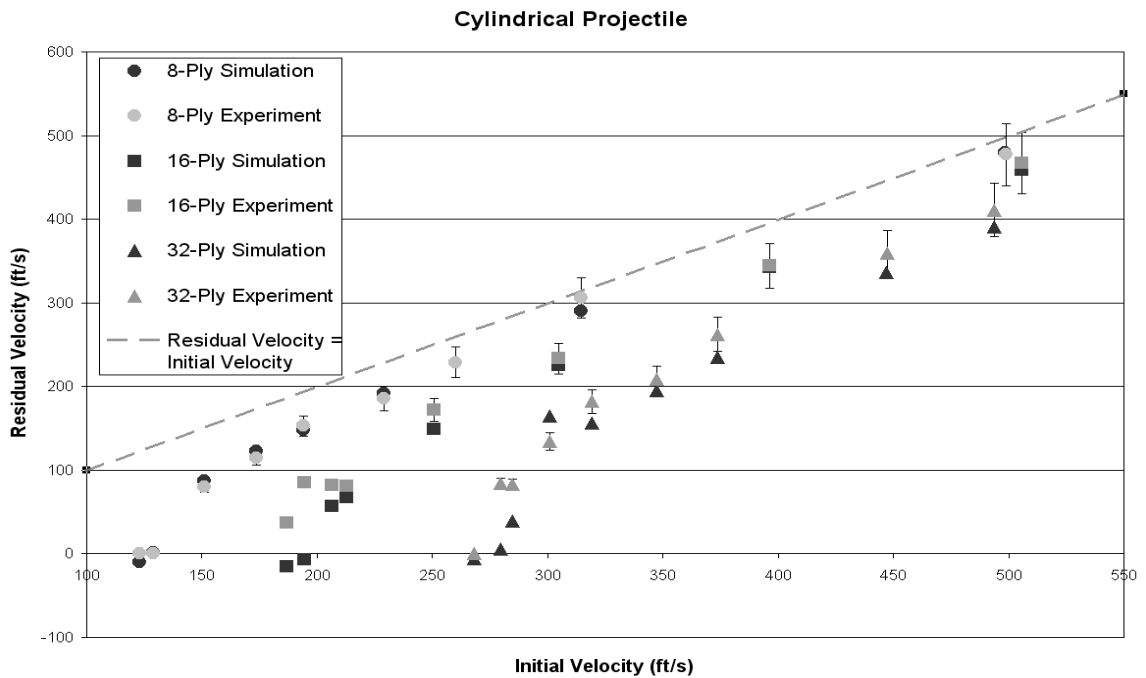


Figure 10: Computed vs. tested exit velocities for 8-ply, 16-ply and 32-ply composite panels impacted with 1/2 in diameter steel cylinder

Discussion

Spherical and cylindrical projectile impact on composite panels has been studied both experimentally and computationally. The computer hardware and software have reached maturity so that reliable ballistic simulations are now possible, provided that accurate material data is available and the analyst has some experience with the software. The available composite material models all treat the ply material as orthotropic homogenous solid and so do not account for the very different and complex failure characteristics of the resin and fiber. Simulation of the ply failure and delamination require very specialized material data and usually is not available without testing. A typical simulation run in this work takes overnight in the SMP mode. Nevertheless, the computer simulations are substantially less expensive and yield more information than testing.

Finite element analysis of material failure does not converge to anything with the mesh refinements. Too coarse a mesh overestimates the energy required to fail the material. When mesh is refined the required failure energy decreases without bounds. There does not seem to be any absolutely reliable way to determine the optimum mesh density without tuning to known test results.

References

1. Goldsmith, Werner, "Review of Non-Ideal projectile impact on targets", *International Journal of Impact Engineering*, Vol. 22, 1999.
2. Hallquist, John, O. "LS-DYNA[®] Theoretical Manual", Livermore Software Technology Corporation, Livermore, CA, March 2006.
3. Hallquist, John, O, et al, "LS-DYNA[®] Keyword User's Manual", Livermore Software Technology Corporation, Livermore, CA, May 2007.
4. Powell, David, Zohdi T, and Johnson, George, "Failure Characterization of Composite Aircraft Materials Under Ballistic Impact", University of California, Berkeley, Report to FAA to be published in 2008.
- 5 Powell, David, Loikkanen, Matti, "Explicit Finite Element Modeling of Composite Plates for Containment of Critical Aircraft Components from Jet Engine Debris", University of California, Berkeley, Report to FAA to be published in 2008.
6. Hashin, Z "Failure Criteria for Unidirectional Composites," *Journal of Applied Mechanics*, Trans. ASME, Vol. 47, June 1980, 329-335
7. Praveen, Grama N., Jiang, Xiaoai, Loikkanen, Matti J., Erhart, Tobias, Benson, David J., "Cohesive Contact Based Delamination Modeling With LS-DYNA." Manuscript in Preparation, 2008.