NASA/CR-1998 206825

FINAL
IN-05-CR

Old Dominion University Research Foundation

IMPACT AND PENETRATION SIMULATIONS FOR COMPOSITE WING-LIKE STRUCTURES

OCIT
2 REF

– Final Report for NAG-1-1858 –

Submitted to:

Structural Mechanics Branch
Structures Division
NASA Langley Research Center
Hampton, Virginia 23681-0001
Dr. Damodar R. Ambur, Technical Monitor

by the

Department of Aerospace Engineering
College of Engineering and Technology
Old Dominion University

Principal Investigator:

Norman F. Knight, Jr., D. Sc.
Professor of Aerospace Engineering
Old Dominion University, Norfolk, Virginia 23529-0247
Telephone No. (757) 683-4265
FAX No. (757) 683-3200
Email: knight@aero.odu.edu

January 30, 1998

IMPACT AND PENETRATION SIMULATIONS

FOR COMPOSITE WING-LIKE STRUCTURES

– Final Report for NAG-1-1858 –

Submitted to:

Structural Mechanics Branch
Structures Division
NASA Langley Research Center
Hampton, Virginia 23681-0001
Dr. Damodar R. Ambur, Technical Monitor

by the

Department of Aerospace Engineering
College of Engineering and Technology
Old Dominion University

Principal Investigator:

Norman F. Knight, Jr., D. Sc.
Professor of Aerospace Engineering
Old Dominion University, Norfolk, Virginia 23529-0247
Telephone No. (757) 683-4265
FAX No. (757) 683-3200
Email: knight@aero.odu.edu

January 30, 1998

IMPACT AND PENETRATION SIMULATIONS

FOR COMPOSITE WING-LIKE STRUCTURES

– Final Report for NAG-1-1858 –

by

Norman F. Knight, Jr.
Department of Aerospace Engineering
College of Engineering and Technology
Old Dominion University

SUMMARY

The goal of this research project was to develop methodologies for the analysis of wing-like structures subjected to impact loadings. Low-speed impact causing either no damage or only minimal damage and high-speed impact causing severe laminate damage and possible penetration of the structure were to be considered during this research effort. To address this goal, an assessment of current analytical tools for impact analysis was performed. Assessment of the analytical tools for impact and penetration simulations with regard to accuracy, modeling, and damage modeling was considered as well as robustness, efficient, and usage in a wing design environment. Following a qualitative assessment, selected quantitative evaluations will be performed using the leading simulation tools. Based on this assessment, future research thrusts for impact and penetration simulation of composite wing-like structures were identified.

BACKGROUND

The design of aerospace structures generally results in lightweight structural designs which exploit advanced structural materials and fabrication concepts. The goals of aerospace structural design are to meet design requirements based on the operating conditions and flight envelope of the vehicle, service life and damage tolerance, and manufacturing cost. Over their intended life cycles, aerospace structures may be subject to internal pressure loads, thermal cycling, bending, axial, and shear loads, impact, and fatigue. Aerospace structures frequently involve flat and curved, stiffened and unstiffened panels, with and without cutouts, that are interconnected by frames, stringers and bulkheads. In addition to satisfying the requirements of the normal operating environment, the design should also be analyzed to assess the ability of the structure to contain damage due to impact and penetration, thus establishing, in part, the residual strength and crashworthiness of the vehicle. Improved crashworthiness will increase the probability of survival for the passengers and crew.

Many researchers have experimentally examined the susceptibility of selected composite laminates to low-speed impact damage using relatively small-scale specimens. Others are investigating the dynamic crushing response of composite structures for improved crashworthiness. In addition, researchers have been taking an analytical approach to studying the impact response of composite structures using a Rayleigh-Ritz approach, a finite element approach, or an integral equations approach. Representative papers in these areas are available in Reference 1.

The solution approach commonly used for this class of problems is the solution to the nonlinear transient dynamic analysis problem. This approach requires the solution of the equations of motion as a function of time during the impact/penetration and/or crash event. Various spatial approximations or discretizations are used giving rise to a semi-discrete system of equations (i.e., ordinary differential equations in the time domain). The conventional solution methods for direct time integration are the implicit time integration methods and the explicit time integration methods. Each solution method is described in the subsequent sections. Evaluation of high-performance equation solvers and a comparison of implicit and explicit procedures are given in References 2 and 3, respectively.

Conventional Implicit Solution Methods

The semi-discrete finite element equations of motion for time increment $n+1$ may be written as

$$\mathbf{M}\ddot{\mathbf{D}}^{n+1} + \mathbf{C}\dot{\mathbf{D}}^{n+1} + \mathbf{F}(\mathbf{D}^{n+1}) = \mathbf{P}^{n+1} \tag{1}$$

where $\mathbf{M}$ is the mass matrix, $\mathbf{C}$ is the damping matrix, $\mathbf{F}$ is the internal force vector, and $\mathbf{P}$ is a vector of external loads. The parameters $\ddot{\mathbf{D}}, \dot{\mathbf{D}}$, and $\mathbf{D}$ represent acceleration, velocity and displacement vectors, respectively. It should be noted that internal force vector $\mathbf{F}$ is a function of the displacements as

$$\mathbf{F}(\mathbf{D}^{n+1}) = \mathbf{K}_o\mathbf{D}^{n+1} + \mathbf{Q}(\mathbf{D}^{n+1}) \tag{2}$$

where $\mathbf{K}_o$ represents the linear stiffness matrix and $\mathbf{Q}$ is a vector of nonlinear terms which may arise from geometric/material nonlinearities, contact/boundary conditions, and follower forces. The solution of equation (1) may be obtained by employing any one of many direct time integration algorithms. A popular choice is the implicit Newmark method which provides the following approximations for the velocity $\dot{\mathbf{D}}^{n+1}$ and acceleration $\ddot{\mathbf{D}}^{n+1}$ vectors at time step $n+1$:

$$\dot{\mathbf{D}}^{n+1} = \frac{\alpha}{\beta h}(\mathbf{D}^{n+1} - \mathbf{D}^n) - \left(\frac{\alpha}{\beta} - 1\right)\dot{\mathbf{D}}^n - \left(\frac{\alpha}{2\beta} - 1\right)h\ddot{\mathbf{D}}^n \tag{3a}$$

$$\ddot{\mathbf{D}}^{n+1} = \frac{1}{\beta h^2}(\mathbf{D}^{n+1} - \mathbf{D}^n) - \frac{1}{\beta h}\dot{\mathbf{D}}^n - \left(\frac{1}{2\beta} - 1\right)\ddot{\mathbf{D}}^n \tag{3b}$$

where h represents the time step (i.e., $h = t_{n+1} - t_n$), and the parameters α and β govern properties of the algorithm. Unconditional stability is achieved when $\alpha = \frac{1}{2}$ and $\beta = \frac{1}{4}$ which results in the constant average-acceleration version of the Newmark method. Using these definitions for α and β and substituting equations (3) into equation (1), the equations of motion may now be expressed as

$$\left(\frac{4}{h^2}\mathbf{M} + \frac{2}{h}\mathbf{C}\right)\mathbf{D}^{n+1} + \mathbf{F}(\mathbf{D}^{n+1}) = \mathbf{P}^{n+1} + \mathbf{F_m}^n + \mathbf{F_c}^n \tag{4}$$

The accompanying equations for $\mathbf{F_m}^n$ and $\mathbf{F_c}^n$ which represent historical inertial and viscous force vectors from the previous time step, respectively, are

$$\mathbf{F_m}^n = \mathbf{M}\left(\frac{4}{h^2}\mathbf{D}^n + \frac{4}{h}\dot{\mathbf{D}}^n + \ddot{\mathbf{D}}^n\right) \tag{5a}$$

$$\mathbf{F_c}^n = \mathbf{C}\left(\frac{2}{h}\mathbf{D}^n + \dot{\mathbf{D}}^n\right) \tag{5b}$$

The right-hand side of equations (4) is now independent of displacements at time $n+1$. For most complex applications such as an impact/penetration or crash analysis, these equations are nonlinear owing to equation (2) for the internal forces. As such, the Newton-Raphson iterative technique is generally applied at each time step, and the desired displacement solution $\mathbf{D}^{n+1}$ is obtained through incremental updates for the kth iteration as

$$\mathbf{D}_{k+1}^{n+1} = \mathbf{D}_k^{n+1} + \Delta\mathbf{D}_k^{n+1} \tag{6}$$

where $\mathbf{D}_k^{n+1}$ and $\mathbf{D}_{k+1}^{n+1}$ denote the current (kth) and updated ($k+1st$) iterative displacement estimates at time $n+1$ for the Newton-Raphson iteration, respectively. The displacement increment $\Delta\mathbf{D}_k^{n+1}$ is obtained by solving a system of linearized equations which must be evaluated, assembled, factored or decomposed, and solved for each iteration. That is,

$$\bar{\mathbf{K}}_k^{n+1}\Delta\mathbf{D}_k^{n+1} = \mathbf{R}_k^{n+1} \tag{7}$$

where the "effective" tangent stiffness matrix $\bar{\mathbf{K}}$ and the residual or out-of-balance force vector $\mathbf{R}$ are given by

$$\bar{\mathbf{K}}_k^{n+1} = \left(\frac{\partial\mathbf{F}}{\partial\mathbf{D}}\right)_k^{n+1} + \frac{2}{h^2}\mathbf{M} + \frac{2}{h}\mathbf{C} \tag{8}$$

$$\mathbf{R}_k^{n+1} = \mathbf{P}^{n+1} + \mathbf{F_m}^n + \mathbf{F_c}^n - \mathbf{F}(\mathbf{D}_k^{n+1}) \tag{9}$$

The computational intensity of nonlinear transient dynamic analyses using an implicit time integration method derives from the fact that each iteration requires the assembly and factorization of $\bar{\mathbf{K}}$, and that this procedure must be repeated for each time step. Use of a modified Newton-Raphson scheme may reduce the number of factorizations; however, for highly nonlinear applications such as an impact/penetration or crash analysis, the associated gains are generally outweighed by a substantial increase in the number of iterations. It should be noted that the development just outlined is based on the more general case of transient dynamic analysis. For static analysis, the time dependence would be removed, and the inertia and damping terms in equation (1) are neglected. Solutions for the static analysis also require the Newton-Raphson iterative technique and are also computationally intensive.

The implicit scheme considered herein is unconditionally stable, and the time step size is limited only by accuracy considerations. This generally permits the use of a large time step. However, several Newton-Raphson increments are often required for each time step, and this number can increase substantially for problems which are highly nonlinear or exhibit structural instabilities. Each Newton-Raphson iteration requires the solution of a linearized system of equations of the form $\mathbf{Ax} = \mathbf{b}$, where $\mathbf{A}$, the effective tangent stiffness matrix, is sparse, symmetric, and may or may not be positive definite. The solution is obtained using either direct or iterative methods for solving systems of linear algebraic equations.

Direct solvers are based on elimination or factorization techniques and are variants of Gaussian elimination. The most widely used and heavily researched technique is the Cholesky method. These techniques are computationally intensive (i.e., $O(n^3)$ floating-point operations for the factorization step alone) but reliable - a solution is guaranteed after execution of a fixed number of operations. However, they are difficult to parallelize efficiently due to the sparsity of the system of equations, and the communication needs. Furthermore, they are very memory intensive since the full system of equations is required.

Iterative solvers for systems of linear algebraic equations are successive approximation techniques based on an initial guess for the solution. One such method which is being heavily researched is the Preconditioned Conjugate Gradient or PCG (e.g., see Reference 2). This method may be implemented on an element-by-element basis resulting in low memory and communication requirements, and is therefore well-suited for parallel processing. However, its reliability is sensitive to the conditioning of $\mathbf{A}$. Ill-conditioning due to geometric and material disparities can lead to extremely slow convergence, no convergence, or even convergence to the wrong results.

In summary, the kernel of the implicit approach is the repeated evaluation, assembly, and solution of a linearized system of equations. Conventional direct solvers are fast and reliable on sequential computers; however, their communication and memory requirements lead to poor scalability in the context of parallel processing. Conventional iterative solvers are very amenable to parallel processing; however, they suffer from convergence problems due to ill-conditioning. Thus, the need exists for a reliable alternative approach which is inherently suited for parallel processing. This alternative approach should accommodate the computational requirements associated with very difficult and complex structures such as composite flight-vehicle structures. It should also exploit the capabilities of a wide range of high-performance parallel computer architectures.

Conventional Explicit Solution Methods

The use of an explicit, rather than implicit, time integration technique to solve the semi-discrete system of equations given by equation (1), is attractive for parallel computations. Central-difference approximations are typically employed for the temporal derivatives. That is,

$$\dot{\mathbf{D}}^{n+1} = \frac{1}{2h}(\mathbf{D}^{n+1} - \mathbf{D}^{n-1}) \tag{10a}$$

$$\ddot{\mathbf{D}}^{n+1} = \frac{1}{h^2}(\mathbf{D}^{n+1} - 2\mathbf{D}^{n} + \mathbf{D}^{n-1}) \tag{10b}$$

Substituting these approximations into equation (1), the following equation for displacements at time $n+1$ may be obtained

$$\left(\frac{1}{h^2}\mathbf{M} + \frac{1}{2h}\mathbf{C}\right)\mathbf{D}^{n+1} = \mathbf{P}^n - \mathbf{F}(\mathbf{D}^n) + \frac{2}{h^2}\mathbf{M}\mathbf{D}^n - \frac{1}{h^2}\mathbf{M}\mathbf{D}^{n-1} + \frac{1}{2h}\mathbf{C}\mathbf{D}^{n-1} \tag{11}$$

where h is the time step. These equations are linear (even for nonlinear problems), and the left-hand-side matrix is constant unless the time step h changes during the solution process. Also, if diagonal mass and damping matrices are used, these equations represent an uncoupled system of algebraic equations in which each solution component may be computed independently. For transient dynamic analysis, a time history of displacements (system response) is sought. Mass and damping vectors which best model the physical properties of the system are used. Techniques for estimating the maximum allowable time step are available, such that the time step size may change during the transient dynamic analysis. As such, explicit time integration techniques are attractive candidates for implementation on parallel computers. These techniques generally have low memory and communication requirements but are also only conditionally stable numerically. Effective solution to the static problem on parallel computer systems still remains the same; however, recent work on adaptive dynamic relaxation procedures is very promising (e.g., see Reference 4).

OVERALL RESEARCH GOALS

The goal of this research activity was to develop and assess methodologies for the simulation of impact and penetration of composite wing-like structures. This effort focussed on low-speed, single-site impact damage with the overall goal of being able to simulate general impact damage at multiple sites for large built-up composite structures. Four primary objectives were originally included in this overall research activity:

1. Identification of fundamental mechanics issues associated with impact and penetration simulations for composite structures.
2. Identification of analytical tools for these simulations and assessment of their capabilities.
3. Develop an intelligent computational system for modeling and analysis of wing-like structures subjected to impact and penetration.
4. Application of these methodologies to the analysis of composite wing-like structures.

Since this research activity was originally proposed for multiple years of work and due to program re-direction only lasted for one year, only the first two objectives were addressed.

RESEARCH OBJECTIVES AND FINDINGS

It was anticipated that this research project would involve several years of research that would result in an accurate simulation tool for composite structure design for impact loading. The goal of this work was to establish the state-of-the-art in simulation tools for impact analysis of composite structures. Benchmark cases were used to establish comparative assessments and limited comparisons with existing test data were performed. Specific research objectives for this grant were as followed:

- Identify key modeling and analysis issues needed to simulate the physics of impact and penetration of composite wing-like structures.
- Assess selected leading large deformation impact and penetration simulation tools for applicability to composite wing structures.
- Perform parametric studies using the leading simulation tools to identify key modeling aspects and appropriate analysis options.
- Identify key research thrusts for developing advanced techniques for impact and penetration simulations.

The principle investigator and two graduate students were involved in this research. One graduate student completed his Master's thesis (see Reference 5) using DYNA3D as part of this effort, and a second student initiated some follow-on activities using LS-DYNA3D that contributed to identifying research thrusts for future work. A copy of the Master's thesis has been sent to the grant monitor under a separate cover letter. This report summarizes those findings and those research thrusts identified in the follow-on effort.

Identify Key Modeling and Analysis Issues

To simulate the impact or penetration event accurately, the simulation tool must incorporate several key features. First, the analysis should account for large deformation effects and possibly tearing of the structure. Second, the simulation tool should include capabilities for modeling both 2-D and 3-D geometries, laminated structures with and without damage, contact, frictional interfaces, and mesh refinement strategies. The composite damage modeling is dependent on the failure modes and mechanisms implemented in the analysis tool. New failure modes and damage models may need to be incorporated as they are developed. The contact algorithm should incorporate sliding frictional interfaces as well as possible surface separation or complete penetration. Third, the analysis most likely will involve an explicit direct-time integration procedure; however, an optional implicit direct-time integration procedure should be available for long duration response predictions beyond the initial impact event. Only a preliminary exploratory study of the penetration mechanics problem was done. The following issues were identified:

- Large deformation effects must be included in the simulation. The results for composites may still involve small strains but the kinematics involved are large deflections and rotations.
- Impact or crash events of metal structures typically result in "folding" of thin sheets or large strain effects leading to material failures. In composites, such behavior results

in delaminations, cracks and brittle failures. Tearing of material requires a modeling and analysis capability much different than a standard nonlinear transient analysis simulation.

- Erosion of elements or tearing of connected elements must be a feature of the analysis tool in order to simulate the progression of through-the-thickness damage. Two of the modeling approaches are indicated in Figure 1. Some methods conserve mass (e.g., node release) but require *a priori* knowledge and modeling of the damage growth directions. Other methods (e.g., element erosion and elimination) use material failure modes to determine damage growth directions but require localized refinement in order to minimize the mass loss due to element erosion.
- Modeling features need to include both a 2-D and 3-D capability depending on the fidelity of the response prediction desired. In the 2-D plate/shell simulations, the through-the-thickness damage is difficult and computationally intensive to simulate. Since these formulations generally assume inextensibility in the thickness direction, interlaminar stresses are not predicted even though they may be drivers in the failure mode. In the 3-D solid simulations, the through-the-thickness damage can be modeled but the spatial discretization in the thickness direction has a direct impact on the planar spatial discretization (i.e., element aspect ratio). The interlaminar normal stresses play a dominate role in damage initiation and growth for laminated composite structures and sandwich structures with soft core materials.
- Existing failure models in most of the simulation tools include extensive elasto-plastic material models for metal structures and only point-stress failure analyses with ply discounting for laminated composite structures. A capability to add user-defined material models is needed in order to implement and assess evolving failure models for laminated composites and sandwich structures. One such study of progressive failure analysis methods for laminate composite structures under static loading conditions is given in Reference 6. The need for reliable material data, a failure model and material degradation models is clearly established.
- Simulation tools should provide both explicit and implicit direct time integration methods. The initial impact event and the response immediately after impact, including possible penetration, occur over a very short time interval and is best analyzed using an explicit solution method. However, for the long time response prediction, an implicit solution method is desirable in order to move forward in time.

Assess Impact and Penetration Simulation Tools

A review of several simulation tools was performed. Several of the leading commercially available finite element codes were included in this assessment including DYNA3D, LS-DYNA3D, MSC/DYTRAN, and STAGS. ODU has the LLNL version of DYNA3D and NIKE3D and is a member of the LLNL Collaborator Program. LS-DYNA3D is available on one Unix workstation at NASA Langley, and ODU obtained an academic license of LS-DYNA3D to support this work. ODU negotiated with MSC to extend our existing license for NASTRAN and PATRAN to include MSC/DYTRAN, but it was not possible to obtain and install the software during the grant period. MSC/DYTRAN is available on a limited basis from a single computer at NASA Langley. STAGS is available on the NASA computers, and ODU did not obtain a copy from COSMIC for our on-site computational systems primarily due to the limited time period for the grant. Most of these codes, if not all, already have several of the analysis capabilities needed for impact/penetration or crash analysis of metal structures. For composites, the damage models and treatment of historical data associated with nonlinear material response is the most critical. Current damage models are based on the observed failure modes which are not documented for large-scale composite built-up structures. Developing and/or incorporating new material models will most likely be required in any of the simulation tools.

Currently, several analysis codes are available from commercial companies or government laboratories for simulating impact, crash, and penetration events. For the most part, these analysis codes focus on the impact event for metal structures in order to assess and improve the crashworthiness of automobiles and aircraft, and on ballistic impacts for military applications. Over the past 15 years, several comparisons have been made to document their features, capabilities, and performance (e.g., see References 7-11). The leading analysis tools for such simulations are DYNA3D, LS-DYNA3D, NIKE3D, ANSYS, ADINA, WHAMS, DYCAST, MSC/DYTRAN, ABAQUS/Explicit, and STAGS. A brief description of each code is given herein.

- DYNA3D - This code was developed by Lawrence Livermore National Laboratory (LLNL) and represents a fully-explicit, nonlinear, transient dynamic, finite-element code that models large deformations of nonlinear materials and contact. Sliding contact with friction and voids (i.e., separation after contact) is permitted. The integration is based on an explicit central-difference time integrator with automatic time step adjustment for efficiency and numerical stability. Very limited support for this code is available from LLNL.

- LS-DYNA3D - This code is available from Livermore Software Technology Corporation and represents the commercial version of DYNA3D from LLNL. Enhanced analysis features and user interfaces are provided. Impact, penetration, crash, and airbag deployment are all possible analysis options in this code.

- NIKE3D - This code was developed by Lawrence Livermore National Laboratory and represents a fully-implicit, nonlinear, transient dynamic, finite element code that models large deformations of nonlinear materials and contact. Sliding contact with friction and voids (i.e., separation after contact) is permitted. The integration is based on an implicit time integrator with automatic time step adjustment for efficiency and accuracy.

- MSC/DYTRAN - This code is available from the MacNeal Schwendler Corporation as a general-purpose, nonlinear, transient dynamic, finite-element code which includes both a Lagrangian and an Eulerian formulation. This code shares its origins with the DYNA3D code. MSC/DYTRAN also offers a fluid and fluid/structure interaction analysis feature based on PISCES. Impact, penetration, crash, and airbag deployment are all possible analysis options in this code.

- ABAQUS/Explicit - This code is available from Hibbitt, Karlsson, and Sorenson, Inc. (HKS) and represents a fully-explicit, nonlinear, transient dynamic, finite-element code that models nonlinear materials and contact.

- ANSYS - This code is available from ANSYS, Inc. (formerly Swanson Analysis Systems, Inc.). It is a general-purpose, finite-element code for linear and nonlinear, 2-D and 3-D structures.
- ADINA - This code is available from ADINA R&D, Inc. It is a general-purpose, finite-element code for linear and nonlinear 2-D and 3-D structures. It claims that the overall reliability, efficiency, and accuracy of ADINA for state-of-the-art practical analysis distinguishes it from other finite-element codes. The code and company were developed by Professor K. J. Bathe of MIT and his associates.
- DYCAST - This code was developed by Northrop-Grumman Corporation under NASA sponsorship. It contains both explicit and implicit direct time integration methods and is focused on crashworthiness predictions for aircraft, helicopter and automotive applications. This code has been developed over a number of years with Dr. Allan Pifko as the main technical developer. The capabilities are heavily focused on crash simulation. However, due to limited development manpower, the code lacks many features needed for general crash and impact simulations of composite structures, and it lacks a user interface.
- WHAMS - This code was developed by Professor Ted Belytschko of Northwestern University and represents a crashworthiness analysis program. The program features an h-adaptive procedure which automatically refines or coarsens the finite-element discretization in order to maintain solution accuracy. It uses subcycling and a contact-impact pinball penalty algorithm with an explicit time integration algorithm.
- STAGS - This code was developed by Lockheed-Martin Palo Alto Research Laboratory under NASA, Navy, and Air Force sponsorship. It represents a general-purpose, nonlinear shell, finite-element analysis code with static, transient dynamic, and eigenvalue analysis options. The transient response is predicted using an implicit time integration procedure and only a static contact algorithm is available. If the impact force versus time curve is known, STAGS can be used to predict general trends and overall structural response. This code is available on NASA computers and through COSMIC.

Perform Parametric Studies

Parametric studies were performed in order to define the modeling and analysis needs for such simulations. These studies focused on simulating existing test data for low-speed impact problems using NASA experimental results and other results available in the literature. Initially these studies were aimed at predicting the impact (or interface) force as a function of time assuming no damage to the laminate. The results presented here primarily represent a summary of the studies performed in Reference 5. All simulations were performed on a SGI Indigo II workstation with the R4400 processor. As a result, very long run times were experienced.

The starting point for the impact simulations was a DYNA3D sample problem (Sample Problem 3) which is the impact of a cylindrical rod at the center of a square plate and the supporting structure. The plate is isotropic with an elasto-plastic material model. The finite element model is shown in Figure 2 which assumes a doubly symmetric response from this quarter model. This model is referred to as the original model (or OM). A slightly modified form was considered wherein the physical supports were replaced by simple-support boundary conditions (or OM/SS). The baseline model (or BL) refers to the OM/SS model with a very high yield stress so that the response remains elastic. In this BL model, only one integration point through the thickness is used. Then a series of models denoted by LCi were examined where i increased from 2 to 7 indicating an increase in the number of integration points through the thickness. The structure is still isotropic so the results from BL and all LCi models should give the same response with the difference being the computational effort to obtain that response.

The transient response of the transverse center deflection for each model is shown in Figure 3, The response obtained by changing support conditions is obvious and consistent with what was expected. The response for the other models indicate a vibration about the undeflected state after the impactor loses contact with the plate (i.e., rebound event). During the early part of the transient, all models give essentially the same solution. However, later on, differences are readily noted for models with few integration points through the thickness of the plate. As the number of integration points increases to 4, the transient response becomes more consistent.

The computational effort for each of these simulations is given in Table 1. The column labeled Run Times is the wall clock time needed to perform the simulation in a non-dedicated mode (i.e., other jobs running at the same time possibly) and is not a clear measure of the increase in execution time as a result of increased model fidelity. For the LCi models, the same number of time steps is used. From this table, the increase in the number of integration points from 1 to 7 nearly doubled the required storage space on disk for the computational database and also nearly doubled the execution CPU time. As a consequence for laminated plates, the modeling of through-the-thickness effects using 2-D plate/shell elements for a typical 8-ply laminate will result in a significant increase in computational effort unless ply clustering is used.

The next study considered the same basic geometry for the plate and examined the effects of mesh refinement in the anticipated contact region. In addition, the impactor shape is changed to spherical rather than a flat end cylindrical rod. A schematic of this problem (Problem 3) is shown in Figure 4. Eight different finite element meshes were considered as indicated in Figure 5. Meshes 1 and 2 have uniform element spacings, while the other meshes have a graded mesh with more elements along the symmetry planes and in the contact region. Table 2 lists some general characteristics of each mesh. As the mesh is refined and element size decreases, the critical time step for the explicit time-integration method also decreases, and hence the total number of time steps needed to reach a specific simulation time (say 1000 micro-seconds) increases. This consequence is clearly indicated by the data given in Table 3.

The transient response of the transverse deflection at the center of the plate and also at the point $x = y = 2.5$ inches from the plate center are shown in Figures 6 and 7, respectively. The point directly under the impactor begins to move immediately (see

Figure 6), while the other point requires about 75 micro-seconds for motion to begin (see Figure 7). For the most part, all of the results from the different meshes exhibit the same overall trends, and results obtained using Mesh 7 are considered to be the more accurate results.

The transient response of the surface strains on the lower surface of the plate at the point $x = y = 2.5$ inches from the plate center are shown in Figures 8 and 9. The strain in the $x-$direction is shown in Figure 8, and the strain in the $y-$direction is given in Figure 9. Again the overall trends are similar, and the finite element mesh in the neighborhood of this point away from the center has approximately the same spatial discretization for all the graded mesh cases.

The next simulation is for a laminated panel using the modeling strategies that have been developed from the earlier simulations of related geometries. A schematic diagram of this problem is shown in Figure 10, and this problem is referred to as Problem 4. The panel is a flat rectangular panel, and the impactor is a hemispherical dropped-weight assembly. The panel is a 48-ply quasi-isotropic graphite-epoxy laminate, and the impactor is steel. Material data are from Reference 8. Two different finite element models were used for this simulation as shown in Figure 11. The uniform mesh is considered to be the coarse mesh to reflect the modeling in the neighborhood of the impact, while the graded mesh is referred to as the refined mesh because of the high refinement in that same region. The smallest element size in the coarse mesh is ten times the smallest element size in the refined mesh. However, the total number of elements in the coarse mesh exceeds the number of elements in the refined mesh, while the number of elements in the contact region for the refined mesh is 25 times that in the coarse mesh. This has a direct consequence on the computational effort for the simulations (i.e., the simulation using the refined mesh will take approximately ten times as many time steps to reach the same amount of simulation time).

The transient response for the predicted impact force versus time is shown in Figure 12. The results obtained from two DYNA3D simulations with different spatial discretizations are compared with the analytical results presented in Reference 8. The results obtained using the coarse mesh with DYNA3D are in reasonably good agreement with those of Reference 8 as indicated in Figure 12. The results obtained using the refined mesh exhibit a similar trend with increased amplitudes and a shorter impact/contact time. These differences may be attributed to the simulation fidelity of Reference 8, laminate modeling, contact modeling, or a combination. Further studies will be needed to resolve these differences.

The final simulation was performed using the LS-DYNA3D code. This simulation involved an oblique impact of a rod and a plate with both modeled only with solid elements. This problem formed the basis of a study to examine alternative ways to represent the contact surface and the associated events (e.g., rigid impactor). The results shown in Figure 13 are only qualitative and indicate the complexities of an impact/penetration simulation that will be needed for a composite structure. From Figure 13, the impactor is observed to deform and fragment as well as slide along the surface of the plate. In addition, the plate is punctured with through-the-thickness damage, and a hole is created by the impactor, thereby requiring elements to collapse and/or erode. This type of capability is needed for simulation events such as the rotor burst problem.

Identify Research Thrusts

As a results of these studies, additional new research thrusts are defined. Simulation of the impact and penetration problem for composite structures involves many aspects of computational mechanics and composite mechanics that need to interact in a synergistic manner leading to an intelligent computational system for this class of problems. Issues associate with damage growth, large deformations coupled with fragmentation, and residual strength predictions for composite wing-like structures include the need to:

- Establish modeling criteria to develop crack simulation and growth models based on material response or fracture criteria. One such approach to developing a validated modeling strategy is included in Figure 14.
- Identify failure modes and develop failure models for laminated composite structures including different failure mechanisms, fracture models, and material degradation models to be used in combination with point stress failure models.
- Provide through-the-thickness damage modeling capability for laminated and sandwich structures.
- Provide an adaptive contact modeling algorithm that evolves as the impact and/or penetration progresses and contact surfaces change.
- Provide robust and efficient procedure to establish the pre-stress state from which to initiate failure rather than simulate the penetration event. Adaptive dynamic relaxation is one procedure to evaluate.
- Assess the structural response characteristics for assumed quasi-static stable crack growth simulations and those incorporating inertia effects from the impact and/or penetration event itself. Experimental tests on dynamic crack growth are needed in addition to numerical simulations.

CONCLUDING REMARKS

A preliminary investigation of simulating impact and penetration of composite structures was performed under this grant. Existing finite element analysis codes were reviewed and their capabilities generally defined as related to the impact event. Literature review on the impact response of composite plates and composite curved panels was performed, and a detailed summary is reported in Reference 5. Limited detailed analysis simulations are available, and only limited experimental results are available. The analysis simulations of impact nearly always assume a contact force profile (or "footprint" of the impactor) as well as an impact force magnitude versus time distribution. In a design setting, neither of these attributes is generally available unless testing is performed. For flight hardware, such data are rarely available.

Simulations performed to-date are limited to low-speed impact simulations which do not cause any material damage. These simulations indicate the modeling fidelity needed to capture the transient dynamic response accurately, and these studies indicate the computational complexities to be anticipated for impact and penetration studies for laminated composites experiencing damage.

REFERENCES

1. Poursartip, A. and Street, K. (editors), *Proceedings of the Tenth International Conference on Composite Materials - Volume V: Structures*, ICCM-10, Whistler, British Columbia, Canada, August 14-18, 1995, WoodHead Publishing Limited, 1995.
2. Poole, E. L., Knight, N. F., Jr., and Davis, D. D., Jr., "High-Performance Equation Solvers and Thir Impact on Finite Element Analysis," *International Journal for Numerical Methods in Engineering*, Vol. 33,No. 2, February 1992, pp. 855-868.
3. Bathe, K. J., Guillermin, O., Walczak, J., and Chen, H. Y., "Advances in Nonlinear Finite Element Analysis of Automobiles," *Computers and Structures,* Vol. 64, No. 5/6, 1997, pp. 881-891.
4. Oakley, D. R. and Knight, Jr., N. F., "Nonlinear Structural Response Using Adaptive Dynamic Relaxation on a Massively Parallel Processing System," *International Journal for Numerical Methods in Engineering*, Vol. 39, 1996, pp. 235-259.
5. Barlow, Kenneth William, *DYNA3D Simulations of Low-Speed Impact Response of Laminated Composite Panels,* Master's Thesis, Aerospace Engineering Department, Old Dominion University, Norfolk, VA, August 1997.
6. Sleight, David W., *Progressive Failure Analysis Methodology for Laminated Composite Structures,* Master's Thesis, Aerospace Engineering Department, Old Dominion University, Norfolk, VA, May 1997.
7. Zukas, J. A., Jonas, G. H., Kimsey, K. D., Misey, J. J., and Sherrick, T. M., "Three-Dimensional Impact Simulations: Resources and Results," *Computer Analysis of Large-Scale Structures*, ASME AMD-Vol. 49, K. C. Park and R. F. Jones, Jr. (editors), New York, 1981, pp. 35-68.
8. Belytschko, T., "On Computational Methods for Crashworthiness," *Computers and Structures*, Vol. 42, No. 2, 1992, pp. 271-279.
9. Noor, A. K. and Carden, H. D., *Computational Methods for Crashworthiness*, NASA CP-3223, 1993.
10. Jackson, Karen E., "Analytical Crash Simulation of Three Composite Fuselage Concepts and Experimental Correlation," *Journal of the American Helicopter Society,* April 1997, pp. 116-125.
11. Matemilola, S. A. and Stronge, W. J., "Impact Response of Composite Cylinders," *International Journal for Solids and Structures,* Vol. 14, No. 21, 1997, pp. 2669-2684.
12. Ambur, D. R., Starnes, Jr., J. H., and Prasad, C. B., *Influence of Transverse-Shear and Large-Deformation Effects on the Low-Speed Impact Response of Laminated Composite Plates,* NASA TM-107753, April 1993.

Table 1. Comparison of computational effort for Problem 2.

Model Designation	Run Time [5] (Hrs)	Number of Time Steps	Disk Space (in Bytes)	Execution Time, CPU (sec)	IO	Model Description[3]
OM[1]	2.2	12,191	499,052	7,556	26.62	Original Model used as a BaseLine for comparison
OM/SS[1]	1.68	12,163	424,680	6,155	16.71	Original Model with Simply Supported Boundary Conditions
BL[2]	3.45	12,062	424,680	6,048	44.84	Original Model with No Supports and modified material properties
LC2[2]	4.38	13,350	502,535	7,747	36.78	Laminated Composite with 2 integration points
LC3[2]	5.02	13,350	583,662	8,867	44.69	Laminated Composite with 3 integration points
LC4[2]	5.78	13,350	663,989	10,165	59.83	Laminated Composite with 4 integration points
LC5[2]	7.88	13,350	744,716	11,735	59.13	Laminated Composite with 5 integration points
LC6[2]	7.32	13,350	825,443	12,844	68.83	Laminated Composite with 6 integration points
LC7[2]	3.82	13,350	906,170	13,607	38.32	Laminated Composite with 7 integration points

(1) Material properties as listed in Table 4.1 $\left.\begin{matrix}\\ \\ \end{matrix}\right\}$ Ref. 5
(2) Material properties as listed in Table 4.3
(3) All simulation were run on the SGI in double precision
(4) Data Dump Time interval was 1.05e+02 with 95 d3plot files
(5) Wall-clock time needed to run in a non-dedicated environment

Table 2. Comparison of finite element mesh information for Problem 3.

Mesh Name	Element Length Smallest (in.)	Element Length Lgest. (in.)	Max Aspect Ratio	Total Panel Elements	Total Panel Nodes	Length of Contact Reg. (in.)	Contact Region Elements	Contact Region Nodes
Mesh 1	0.2	0.2	1	625	676	0.4	4	9
Mesh 2	0.1	0.1	1	2500	2601	0.3	9	16
Mesh 3	0.02	0.1175	5.875	2500	2601	0.1	25	36
Mesh 4	0.01	0.13	13	2500	2601	0.05	25	36
Mesh 5	0.005	0.15	30	2916	3025	0.04	64	81
Mesh 6	0.00375	0.2	53.333	3969	4096	0.04125	121	144
Mesh 7	0.0025	0.15	60	4225	4356	0.04	256	289
Mesh 8	0.002	0.15	75	5329	5476	0.03	225	256

Table 3. Comparison of computational effort for Problem 3.

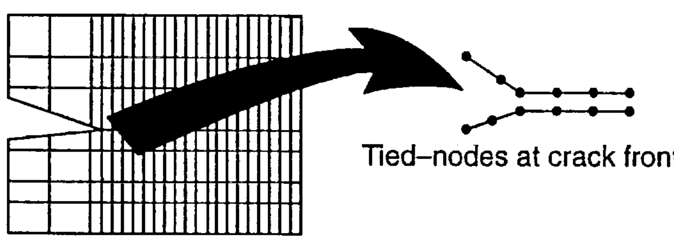
Mesh Name	Smallest Ele. Length (in.)	Run Time (in hours)*	Number of Time Step	CPU	IO
Mesh 1	0.2	0.183	2,301	543	8.05
Mesh 2	0.1	3.5	4,602	3,184	25.03
Mesh 3	0.02	10.7	23,008	16,016	83.06
Mesh 4	0.01	12.9	46,016	30,632	107.18
Mesh 5	0.005	20.5	92,032	69,894	212.04
Mesh 6 **	0.00375	65.8	122,710	114,022	520.48
Mesh 7	0.0025	62.5	184,063	198,430	756.68
Mesh 8 ***	0.002	99.1	230,066	308,820	1305.00

* All simulations were run in a non-dedicated environment with the run time equal to the wall clock time.

** This simulation was terminated at 5.0e-4 sec, therefore the computational results have been multiplied by 1.0e-3/5.0e-4 = 2.0 for scaling effects.

*** This simulation was only run out to 9.0e-5 sec, therefore the computational results have been multiplied by 1.0e-3/9.0e05 = 11.11 for scaling effects.

- Break tied nodes or coincident nodes concept
  - similar to approach used for quasi–static analyses
  - no mass loss
  - assumed crack growth path
  - break tie based on plastic strain



- Eroded element concept
  - requires small elements in the crack front
  - needs transition modeling
  - mass loss; function of element size
  - “failed” elements removed or eroded from mesh

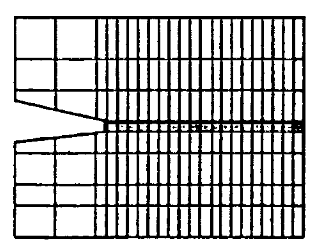
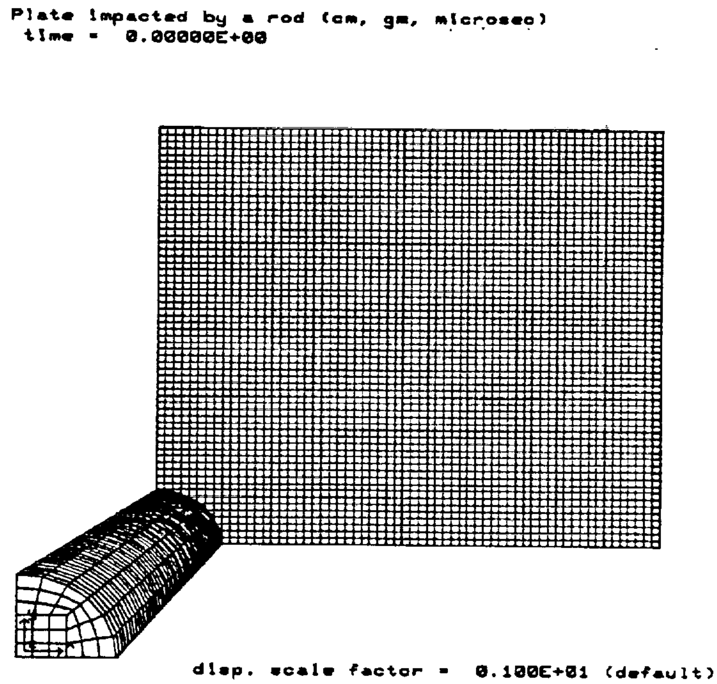
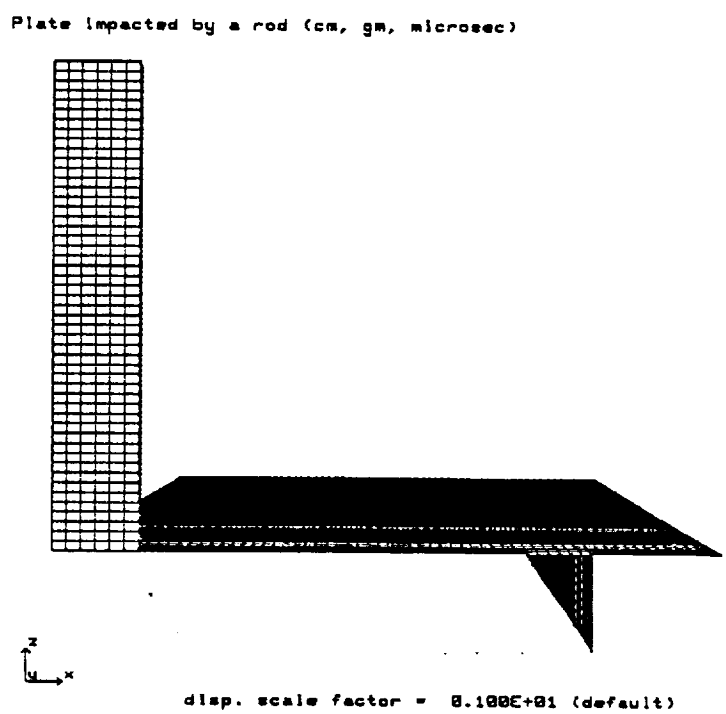


Figure 1. Crack modeling concepts.



(a) Top View



(b) Side View

Figure iiquad Finite element mesh for DYNA3D sample problem for a cylindrical rod impacting a plate - quarter model.

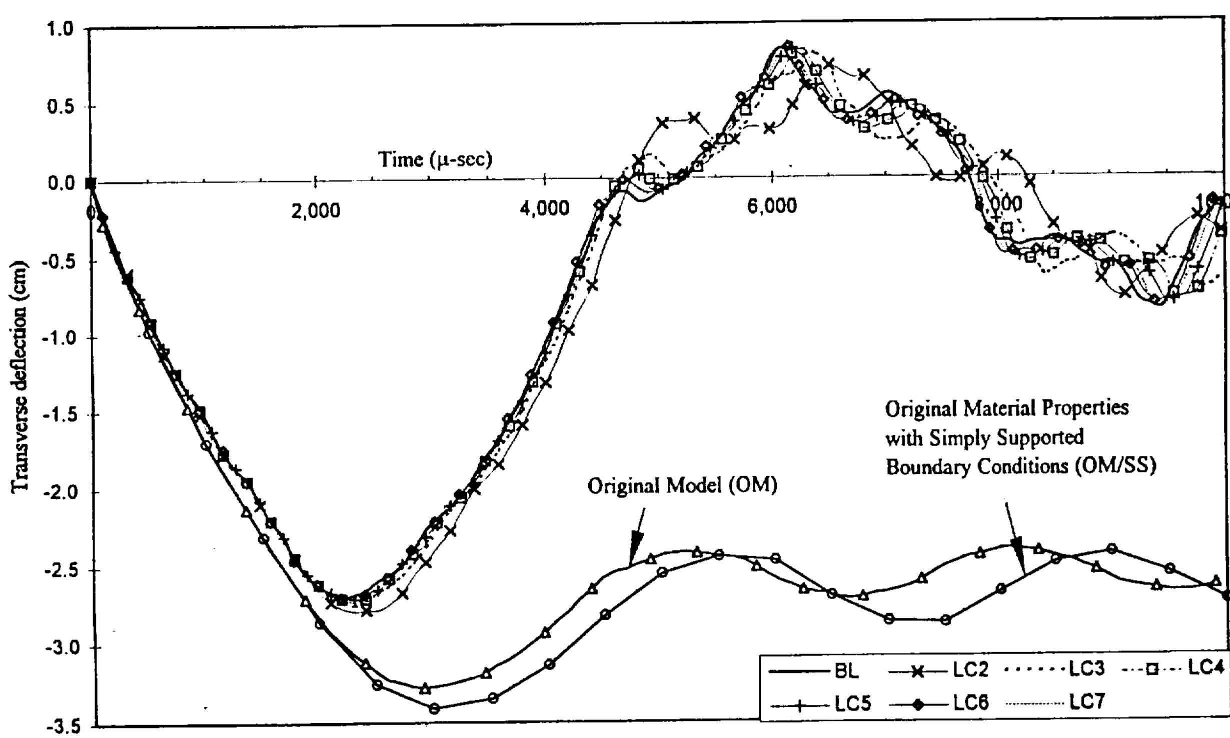


Figure 3. Transient response for the transverse deflection of the center node for Problem 2.

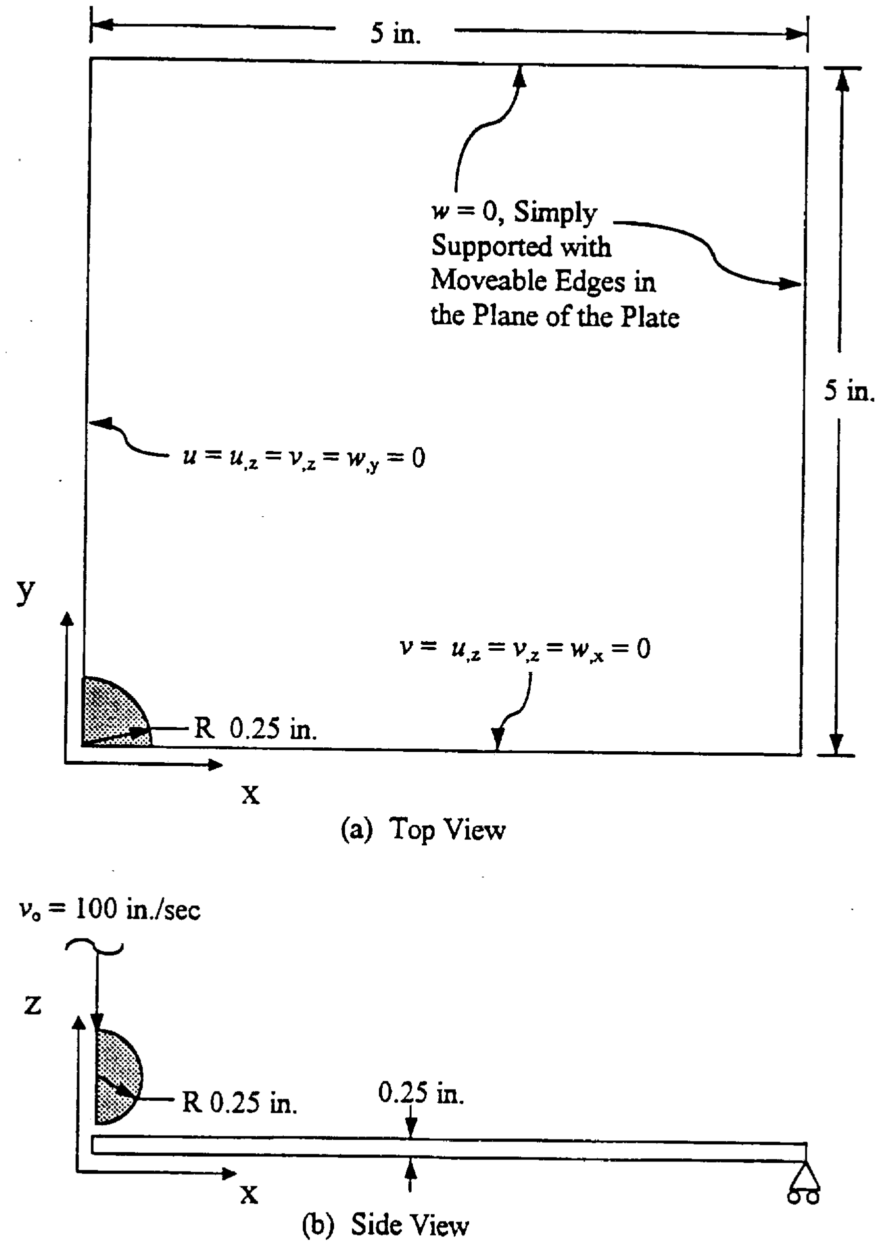


Figure 4. Panel dimensions for the quarter model of Problem 3.

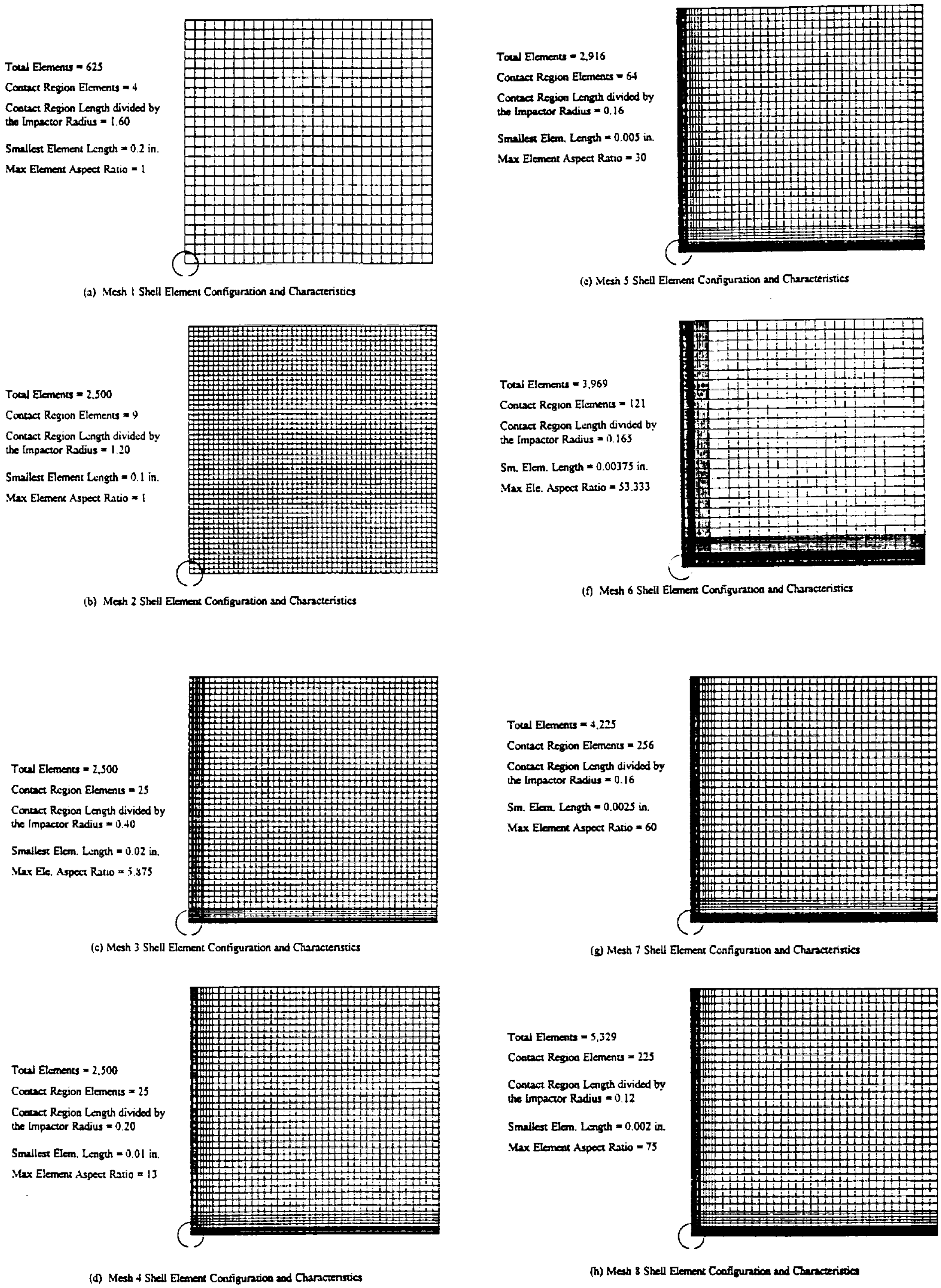


Figure 5. Mesh configurations used to simulate and study an impact event.

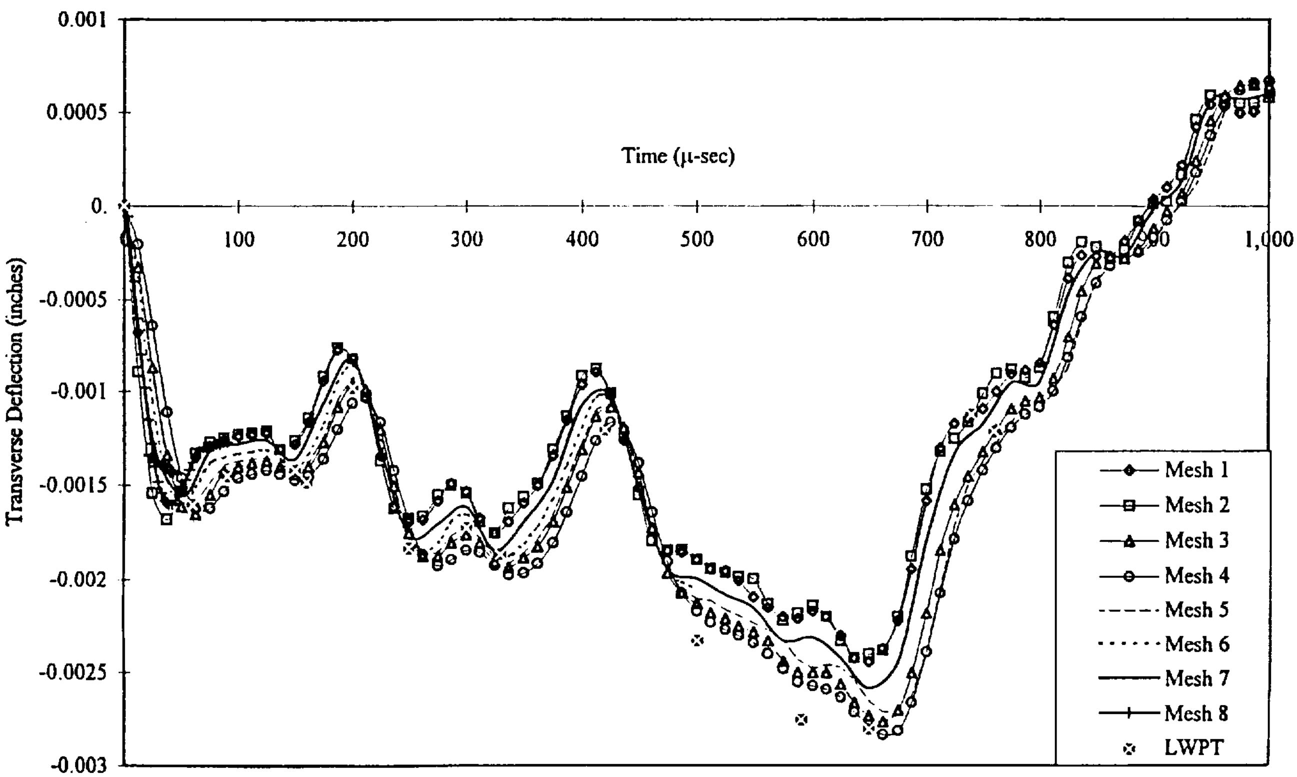


Figure 6. Transient response for the transverse deflection of the center node for Problem 3.

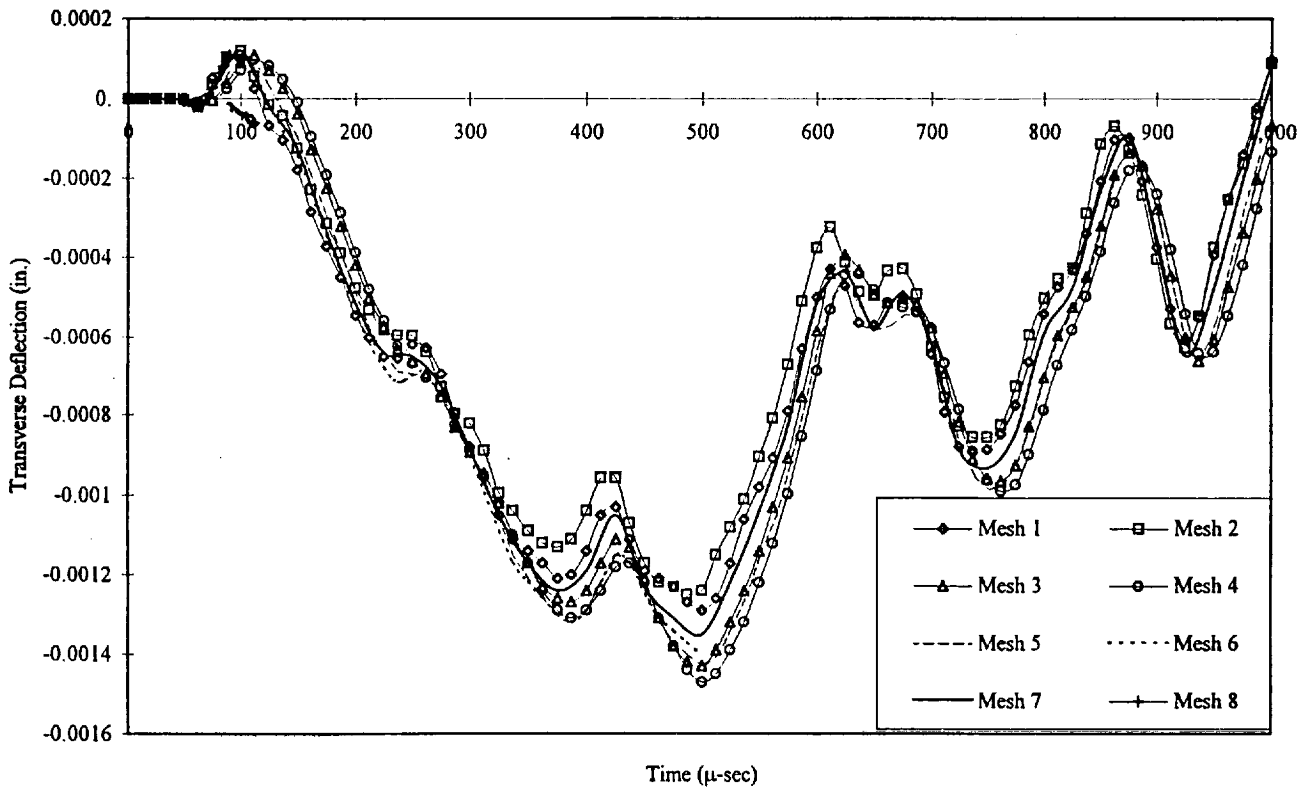


Figure 7. Transient response for the transverse deflection of the point located at $x = y = 2.5$ inches from panel center for Problem 3.

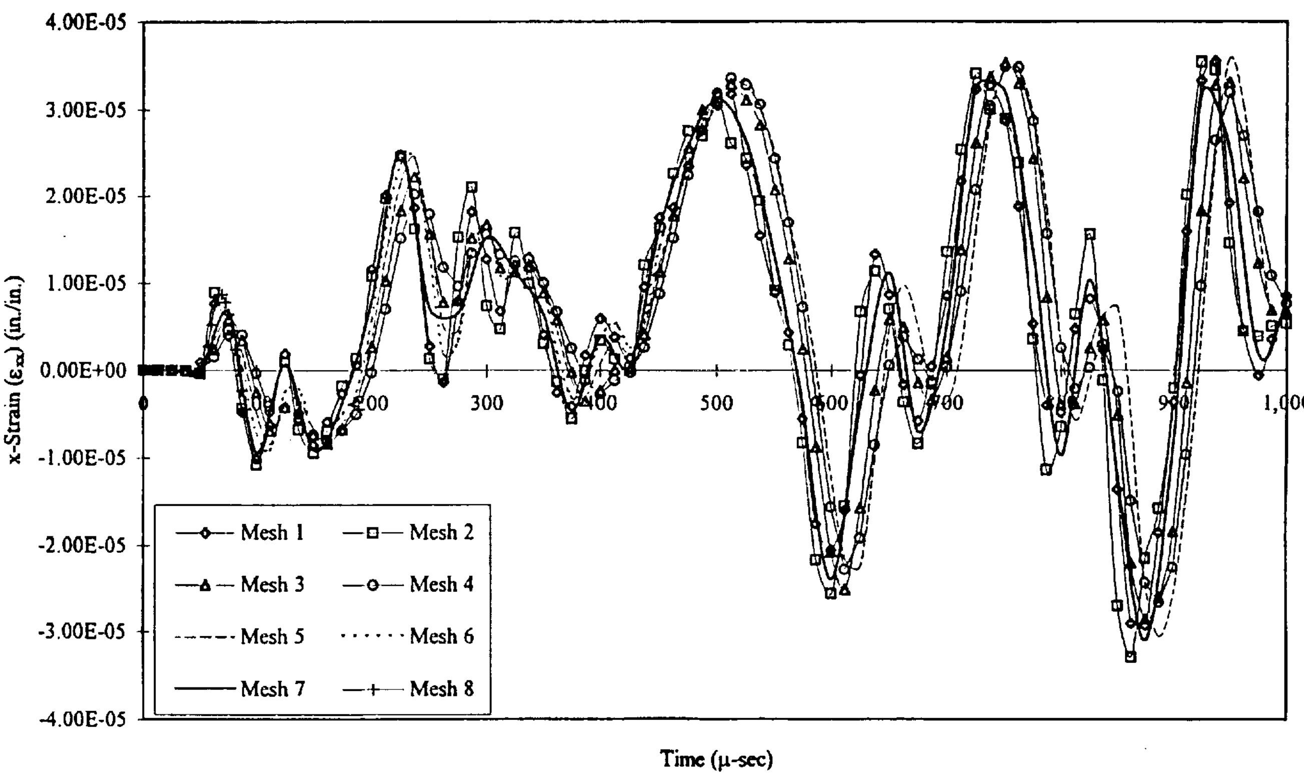


Figure 8. Transient response for the lower surface strain in the x−direction at the point located at $x = y = 2.5$ inches from panel center for Problem 3.

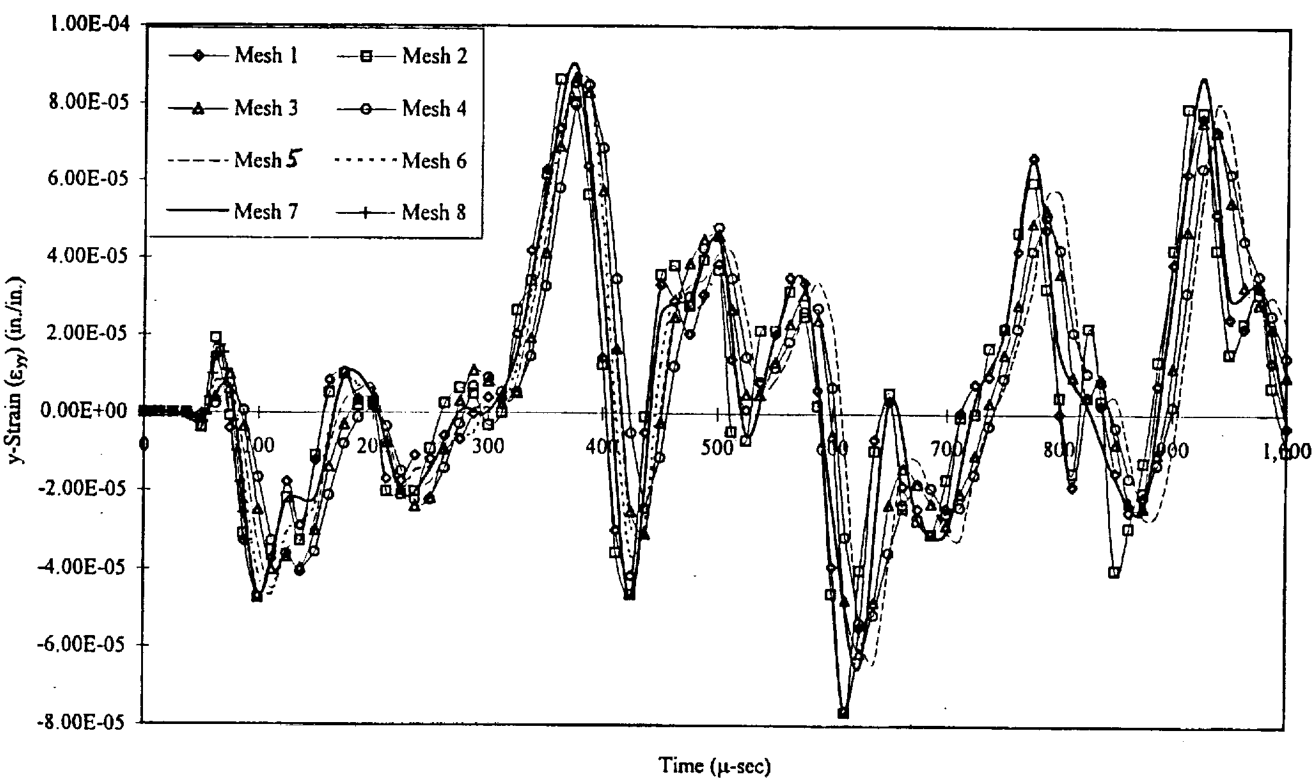
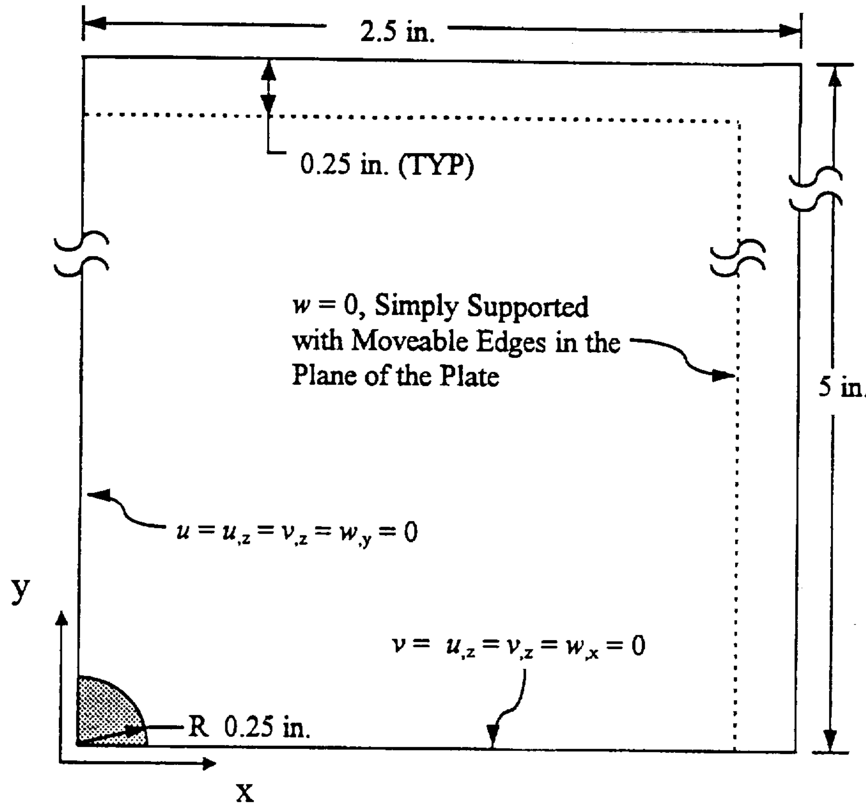
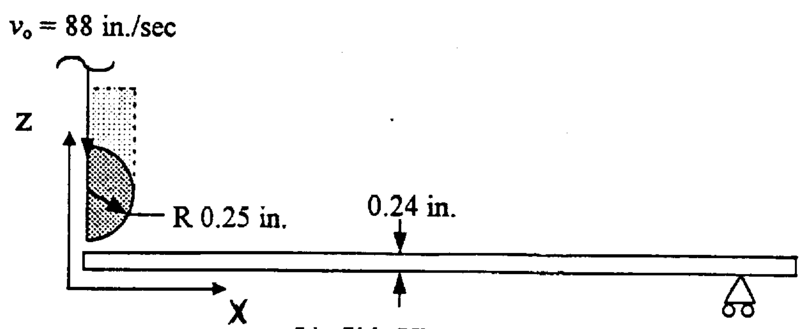


Figure 9. Transient response for the lower surface strain in the y−direction at the point located at $x = y = 2.5$ inches from panel center for Problem 3.



(a) Top View



(b) Side View

Figure 10. Panel dimensions for the quarter model of panel impacted by a 2.6 lb_m dropped weight assembly - Problem 4.

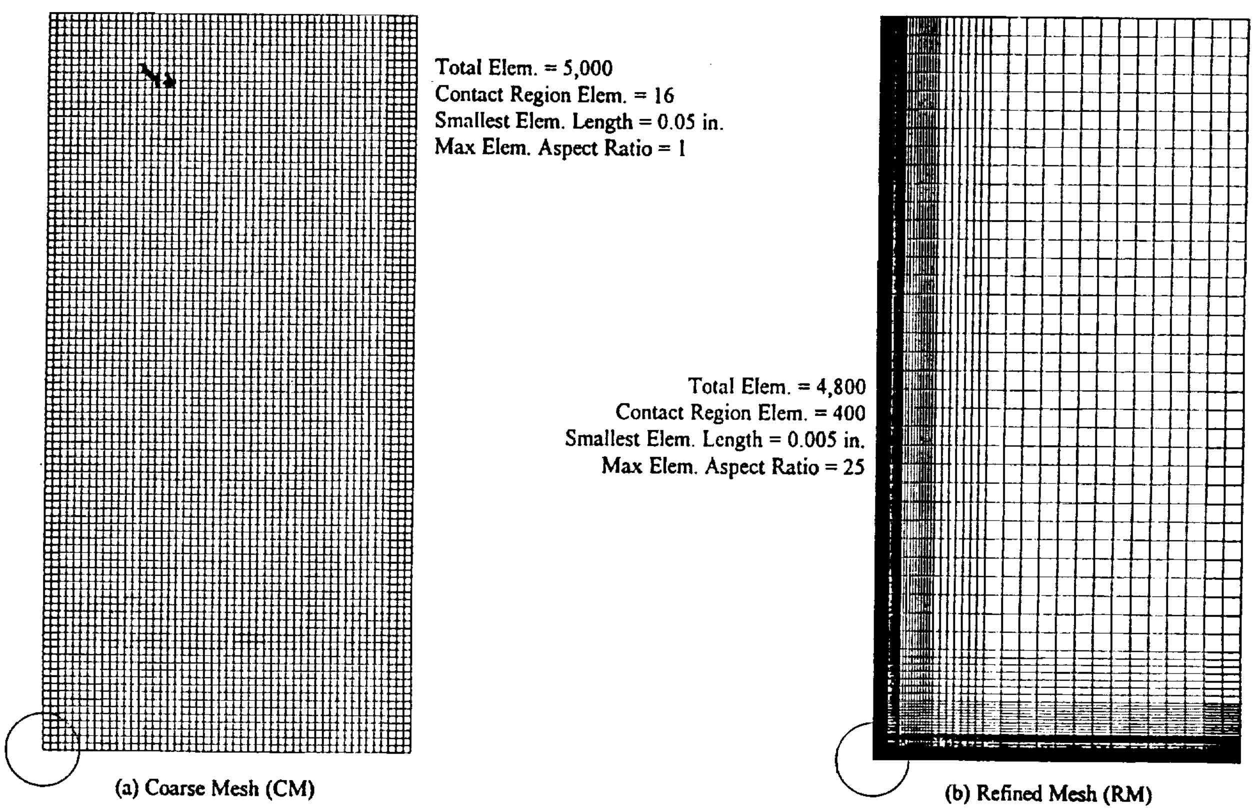


Figure 11. Mesh configurations used for Problem 4.

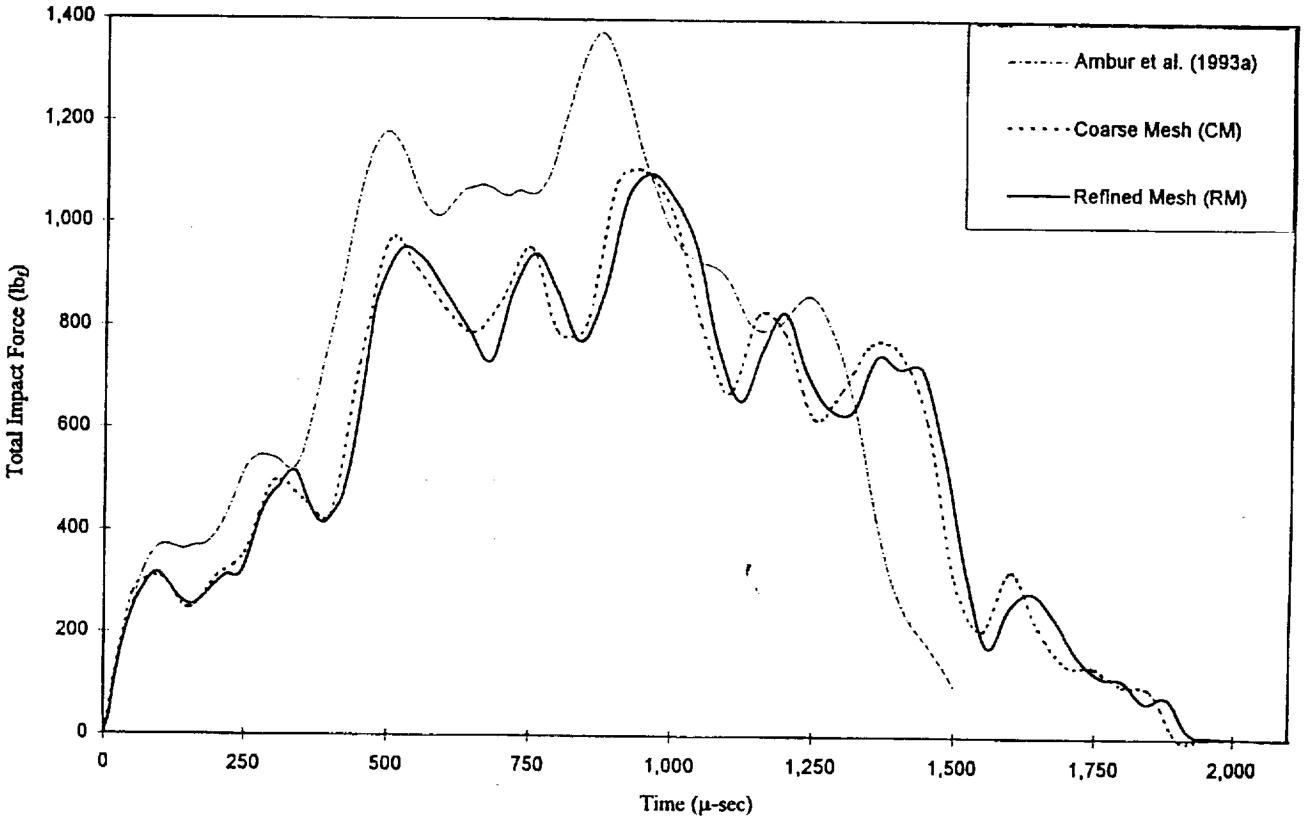


Figure 12. Impact force profile for Problem 4.

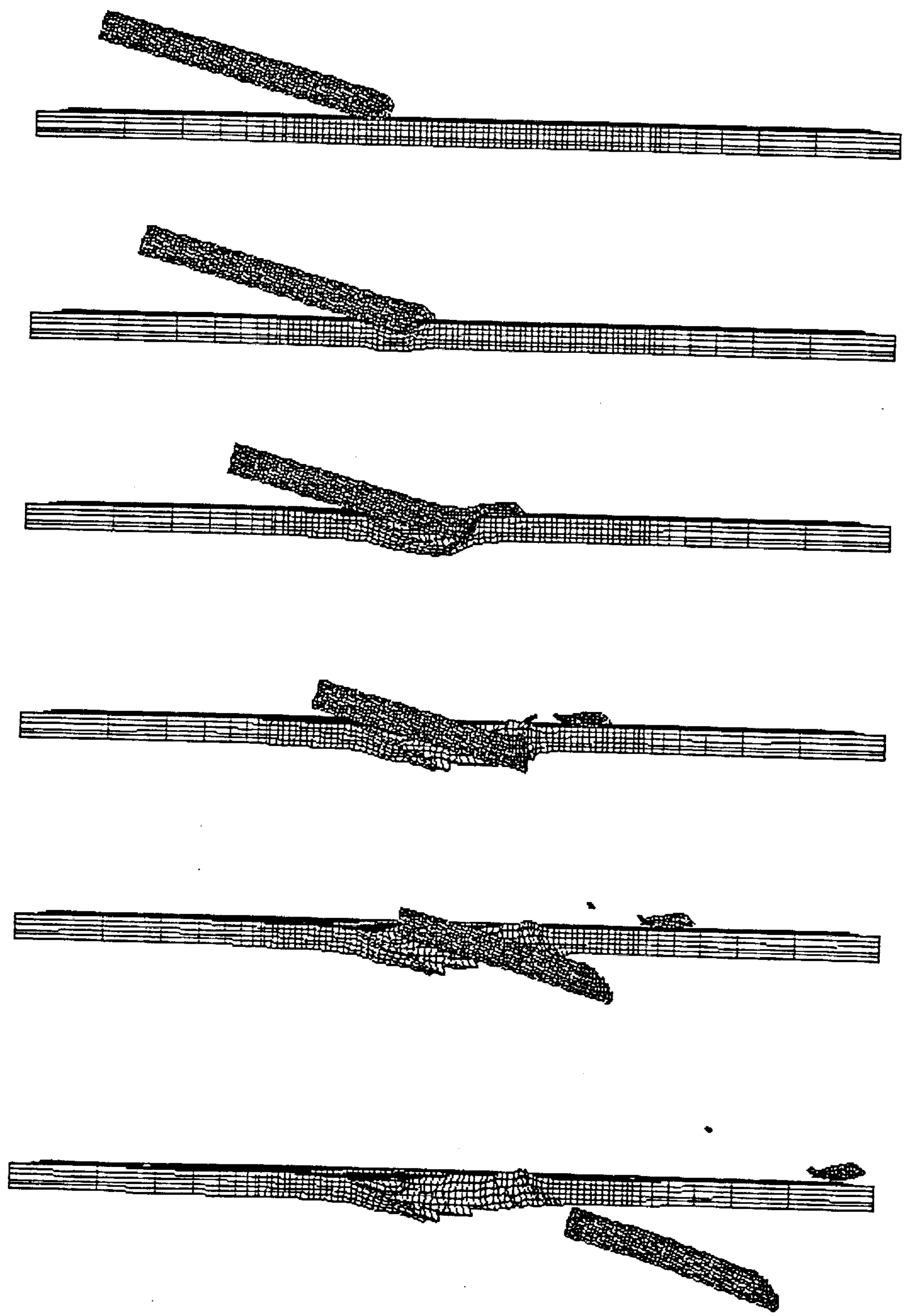


Figure 13. Impact and penetration sample simulation from LS-DYNA3D.

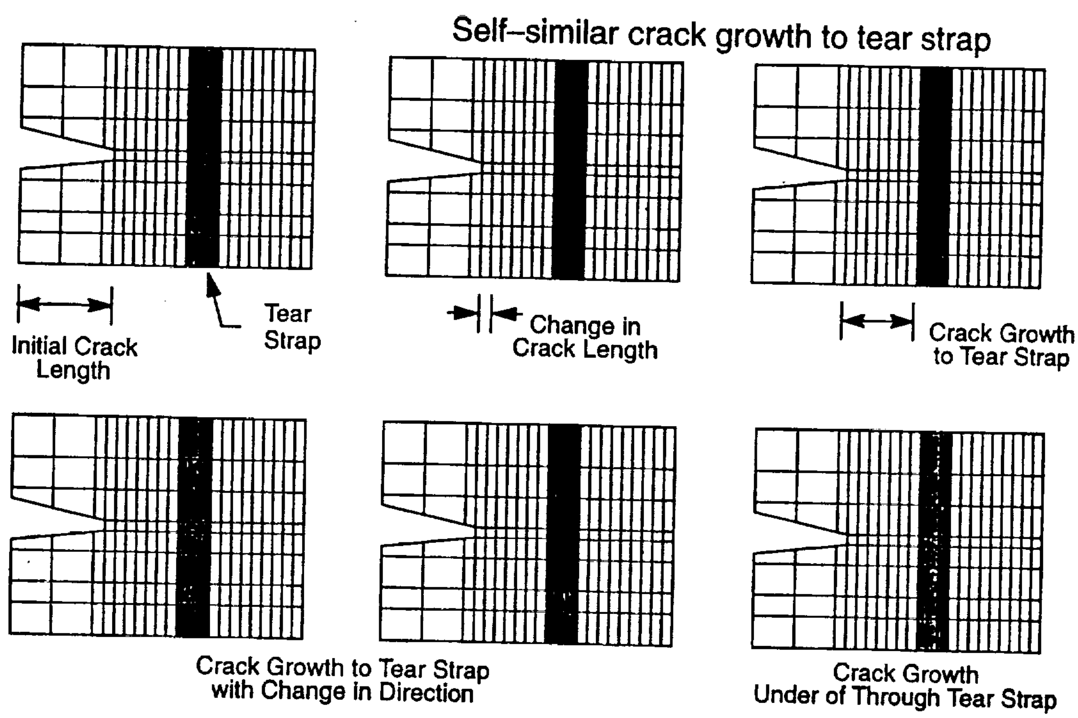


Figure 14. Crack growth modeling concept strategy.