Multiscale Failure Analysis of Laminated Composite Panels Subjected to Blast Loading Using FEAMAC/Explicit

Evan J. Pineda and Anthony M. Waas
University of Michigan, Ann Arbor, Michigan

Brett A. Bednarcyk and Steven M. Arnold
Glenn Research Center, Cleveland, Ohio

Craig S. Collier
Collier Research Corporation, Newport News, Virginia

NASA STI Program . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) program plays a key part in helping NASA maintain this important role.

The NASA STI Program operates under the auspices of the Agency Chief Information Officer. It collects, organizes, provides for archiving, and disseminates NASA's STI. The NASA STI program provides access to the NASA Aeronautics and Space Database and its public interface, the NASA Technical Reports Server, thus providing one of the largest collections of aeronautical and space science STI in the world. Results are published in both non-NASA channels and by NASA in the NASA STI Report Series, which includes the following report types:

- TECHNICAL PUBLICATION. Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.

- TECHNICAL MEMORANDUM. Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.

- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.

- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.

- SPECIAL PUBLICATION. Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.

- TECHNICAL TRANSLATION. English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services also include creating custom thesauri, building customized databases, organizing and publishing research results.

For more information about the NASA STI program, see the following:

- Access the NASA STI program home page at *http://www.sti.nasa.gov*

- E-mail your question via the Internet to *help@sti.nasa.gov*

- Fax your question to the NASA STI Help Desk at 443–757–5803

- Telephone the NASA STI Help Desk at 443–757–5802

- Write to:
  NASA Center for AeroSpace Information (CASI)
  7115 Standard Drive
  Hanover, MD 21076–1320

Multiscale Failure Analysis of Laminated Composite Panels Subjected to Blast Loading Using FEAMAC/Explicit

Evan J. Pineda and Anthony M. Waas
University of Michigan, Ann Arbor, Michigan

Brett A. Bednarcyk and Steven M. Arnold
Glenn Research Center, Cleveland, Ohio

Craig S. Collier
Collier Research Corporation, Newport News, Virginia

National Aeronautics and
Space Administration

Glenn Research Center
Cleveland, Ohio 44135

October 2009

Acknowledgments

Level of Review: This material has been technically reviewed by technical management.

Available from

Multiscale Failure Analysis of Laminated Composite Panels Subjected to Blast Loading Using FEAMAC/Explicit

Evan J. Pineda and Anthony M. Waas
University of Michigan
Ann Arbor, Michigan 48109

Brett A. Bednarcyk and Steven M. Arnold
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

Craig S. Collier
Collier Research Corporation
Newport News, Virginia 23607

ABSTRACT

This preliminary report demonstrates the capabilities of the recently developed software implementation that links the Generalized Method of Cells to explicit finite element analysis by extending a previous development which tied the generalized method of cells to implicit finite elements. The multiscale framework, which uses explicit finite elements at the global-scale and the generalized method of cells at the micro-scale is detailed. This implementation is suitable for both dynamic mechanics problems and static problems exhibiting drastic and sudden changes in material properties, which often encounter convergence issues with commercial implicit solvers. Progressive failure analysis of stiffened and un-stiffened fiber-reinforced laminates subjected to normal blast pressure loads was performed and is used to demonstrate the capabilities of this framework. The focus of this report is to document the development of the software implementation; thus, no comparison between the results of the models and experimental data is drawn. However, the validity of the results are assessed qualitatively through the observation of failure paths, stress contours, and the distribution of system energies.

1 INTRODUCTION

Multiscale methods are effective tools for analyzing damage and failure in fiber-reinforced composite structures. To accurately capture the complicated damage and failure mechanisms in a fiber reinforced laminate (FRL), the contributions to the failure mechanism from the fiber and matrix constituents within a lamina should be considered separately. Even though homogenized methods can be used to model the globally observed damage mechanisms, the synthesis of the mechanisms and the contributions of the constituents (fiber, matrix and fiber/matrix interface) to the global damage mechansims must be understood; in doing so, assumptions must be made as to how local damage and failure in the fiber and matrix are interacting to yield the global damage modes. Micromechanical theories allow the composite damage mechanisms to manifest naturally from the damage and failure within the constituents. Unfortunately, large scale, structural analyses are difficult to perform using micromechanics alone. Therefore, multiscale methods can be used to link the micromechanics to scales at the lamina and laminate (structural) level. Although the power of modern computers is significant, the micromechanical technique that is linked with higher length scales must be chosen wisely so that models remain computational feasible.

The Generalized Method of Cells (GMC), developed by Paley and Aboudi [1], divides a representative volume element (RVE) into a number of subcells each occupied by a constituent of the composite. The dimensions of the subcells are calculated based on the volume fractions of the constituents in the composite. Fig. 1 shows an RVE containing 26 x 26 subcells (312 fiber subcells and 364 matrix subcells) representing a square-packing architecture. Due to the complete generality of the subcell structure, any micro-architecture can be represented with various levels of refinement. This micromechanical model provides semi-closed form solutions for the local subcell stresses and strains in terms of the applied strains and constitutive properties of the subcells, and as a result, is extremely efficient and ideal for use in multiscale analyses. Additionally, composite lamina properties, such as stiffness, can be computed from using the material properties and volume fractions of the constituents. Furthermore, any constitutive law can be used to govern the behavior of the materials occupying the subcells.

Wilt [2] demonstrated the capability to link GMC to the finite element method. Bednarcyk and Arnold [3] used GMC to predict failure in metal-matrix composites with the MAC/GMC Suite of Micromechanics Codes (MAC/GMC) [4, 5] developed by NASA Glenn Research Center. Subsequently, Bednarcyk and Arnold [6] performed multiscale progressive failure analysis (PFA) of composites by coupling MAC/GMC with the commercial available ABAQUS/Standard finite element (FE) software [7] using the Finite Element Analysis - Micromechanics Analysis Code (FEAMAC). FEAMAC uses the ABAQUS/Standard user material subroutine, UMAT, to coordinate between ABAQUS/Standard and MAC/GMC. Pineda et al. [8, 9] used FEAMAC and ABAQUS/Standard to model the behavior of center-notched panels under uniaxial tension exhibiting lamina-level damage and micro-level failure with a square-packed RVE containing 2 x 2 constituent subcells. The RVE used was later refined to contain 7 x 7 subcells [10].

During progressive damage and failure finite element analyses (FEAs), convergence issues can arise when using implicit solution techniques. In most instances, this occurs because there are sizable regions within a structural configuration that have undergone damage resulting in relatively large changes in the local stiffness properties. Because of
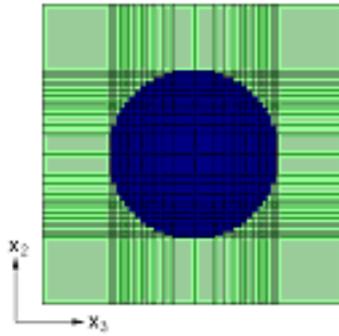
Fig. 1: Demonstrative 26 x 26 GMC RVE.

this large mismatch in stiffness (usually associated with softening), an incremental implicit scheme, unless properly conditioned, may never yield a converged solution in an iterative setting for some increment during the solution phase. Thus, there has been a recent surge towards using explicit solvers, even for static problems, understanding that explicit solvers are not unconditionally stable. This report documents the extension of FEAMAC to FEAMAC/Explicit, developed by NASA Glenn and the University of Michigan, which links MAC/GMC to ABAQUS/Explicit. FEAMAC/Explicit is suitable for dynamic and static, multiscale, progressive failure analysis (PFA) where regions of a damaging structure undergo large and sudden changes in material properties.

The capabilities of FEAMAC/Explicit are demonstrated using two examples: un-stiffened and stiffened composite panels subjected to dynamic, blast loading. At each material point, the fiber and matrix constituents are modeled using GMC with an appropriately chosen RVE. Matrix failure is accounted for with the Tsai-Hill stress failure criterion, and fiber failure is determined using a maximum strain criterion. The global failure mechanisms for the two panels are reported. Additionally, stress fields and system energies are analyzed. These example demonstrate the ability to run multiscale PFA using MAC/GMC an ABAQUS/Explicit.

2 Linking GMC to ABAQUS

2.1 FEAMAC

Multiscale analysis of fiber-reinforced laminates (FRLs) has been achieved by linking the micro-scale, modeled using GMC, to the lamina/laminate scale, modeled with FEs. Communication between MAC/GMC and ABAQUS/Standard implicit FE software was achieved with the FEAMAC software implementation [6]. FEAMAC consists of four ABAQUS/Standard user defined subroutines, as well as six subroutines exclusive to the FEAMAC package (see Fig. 2). Mechanical analysis is achieved through the ABAQUS/Standard subroutine `UMAT`. For every material point in an FE mesh, `UMAT` is called by ABAQUS/Standard, and it provides the strains, strain increments, and current values of state variables to MAC/GMC through the front end subroutine `FEAMAC`. MAC/GMC then returns a new stiffness and stress state to the `UMAT` via the `FEAMAC` subroutine. The ABAQUS/Standard user subroutine `UEXPAN` is used for thermal analysis by providing the integration point temperature, temperature increment, and current state to MAC/GMC and then obtains new thermal strains and thermal strain rates. Problem set-up task, initialization, and writing MAC/GMC level output data to files is achieved through the ABAQUS/Standard user subroutine `UEXTERNALDB`, which communicates between ABAQUS/Standard, the `FEAMAC_PRE`, and `FEAMAC_PLOTS` subroutines. The reader is referred to Bednarcyk and Arnold [6] for further details on the FEAMAC software implemntation.

To set up an FEAMAC problem, a standard ABAQUS input file is used that includes a user material with a name ending in either ".`mac` or "`_mac`. These extensions indicate to FEAMAC that the material is a MAC/GMC composite material whose constituent properties and architecture (e.g., fiber volume fraction and fiber arrangement) are defined in a MAC/GMC input file of the same name. The applicable MAC/GMC input file(s) must be located in the same directory as the ABAQUS input file. Materials that are not associated with MAC/GMC are also permitted in FEAMAC problems. The ABAQUS input file will also typically include an orientation definition (as composite materials
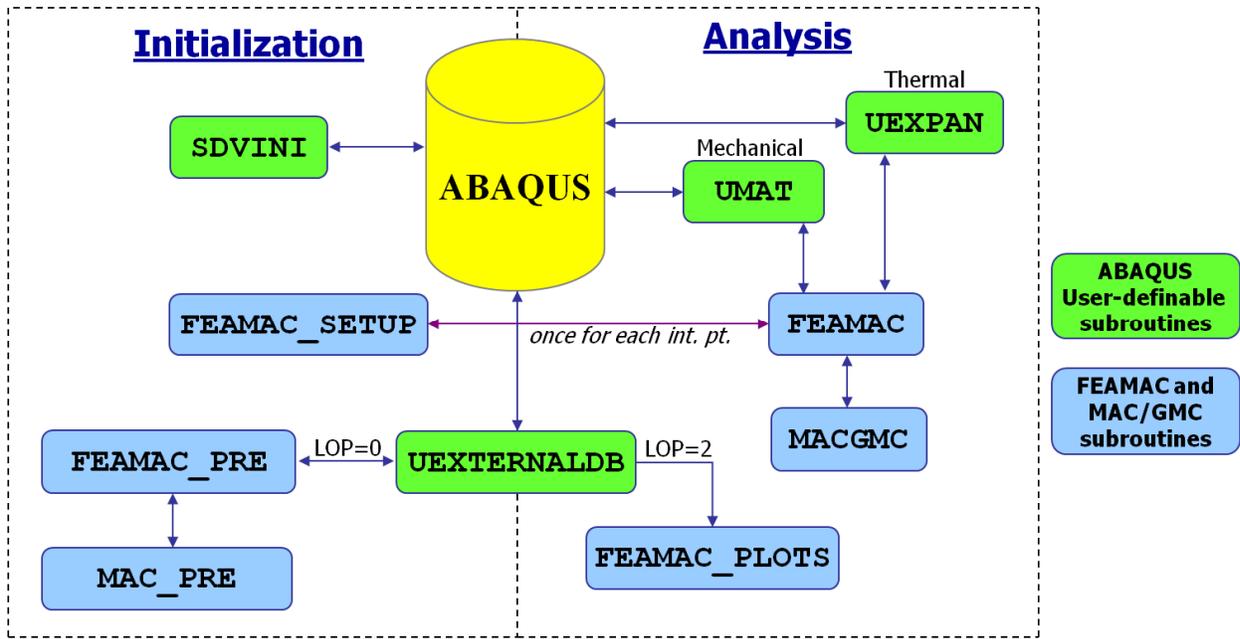
Fig. 2: Hierarchy of FEAMAC software implementation [6].

are usually anisotropic), while the necessary cards usually associated with a user material must be specified as well. Only one additional card, not typically associated with a user material problem must be specified in order to trigger certain initialization tasks: `*INITIAL CONDITIONS, TYPE=SOLUTION, USER`. These initialization tasks are executed within the ABAQUS/Standard user subroutine `SDVINI`. FEAMAC problem execution is accomplished identically to any problem that utilizes a user material, wherein the FORTRAN source file containing the appropriate user subroutines is specified. The FEAMAC subroutines are compiled in a static `.lib` library file which is linked to ABAQUS/Standard when the FEA job is run. The location of the `.lib` file is indicated in the `abaqus_v6.env` file. Finally, FEAMAC problem post processing is accomplished identically to any ABAQUS problem, as all typical ABAQUS output, including the `.odb` file, is available. Constituent level field variables are stored internally within the ABAQUS state variable space and are also available for post-processing.

2.2 FEAMAC/Explicit

The goal of the current development is to achieve the same compatibility between MAC/GMC and ABAQUS/ Explicit that was previously developed between MAC/GMC and ABAQUS/Standard without altering the original FEAMAC `.lib` library, while at the same time utilizing the pre-existing ABAQUS/Standard `UMAT` user subroutine used in the FEAMAC software implementation. Furthermore, it was desired that the segments of the ABAQUS input file pertaining to the usage of MAC/GMC remain unchanged whether using ABAQUS/Standard or ABAQUS/Explicit. For mechanical analysis, ABAQUS/Explicit utilizes the user defined subroutine `VUMAT` to define user material behavior. Since the `UMAT` was used previously to communicate between ABAQUS/Standard and MAC/GMC for mechanical analyses, the `VUMAT` developed is a front end subroutine that coordinates between the `UMAT` subroutine and ABAQUS/Explicit. The computational framework of ABAQUS/Explicit differs significantly from that of ABAQUS/Standard; therefore, several steps, which are described in the following paragraphs, are included in the `VUMAT` to resolve the communication between ABAQUS/Explicit and the original ABAQUS/Standard `UMAT` used in FEAMAC. Unfortunately, there currently exists no ABAQUS/Explicit analog for the ABAQUS/Standard `UEXPAN` subroutine; therefore, thermal analysis is excluded from MAC/GMC when linked to ABAQUS/Explicit. Additionally, state variable initialization is handled directly in the `VUMAT`; so, there is no need for the ABAQUS/Standard user subroutine `SDVINI`, and the card `*INITIAL CONDITIONS, TYPE=SOLUTION, USER` is excluded from the ABAQUS input file.
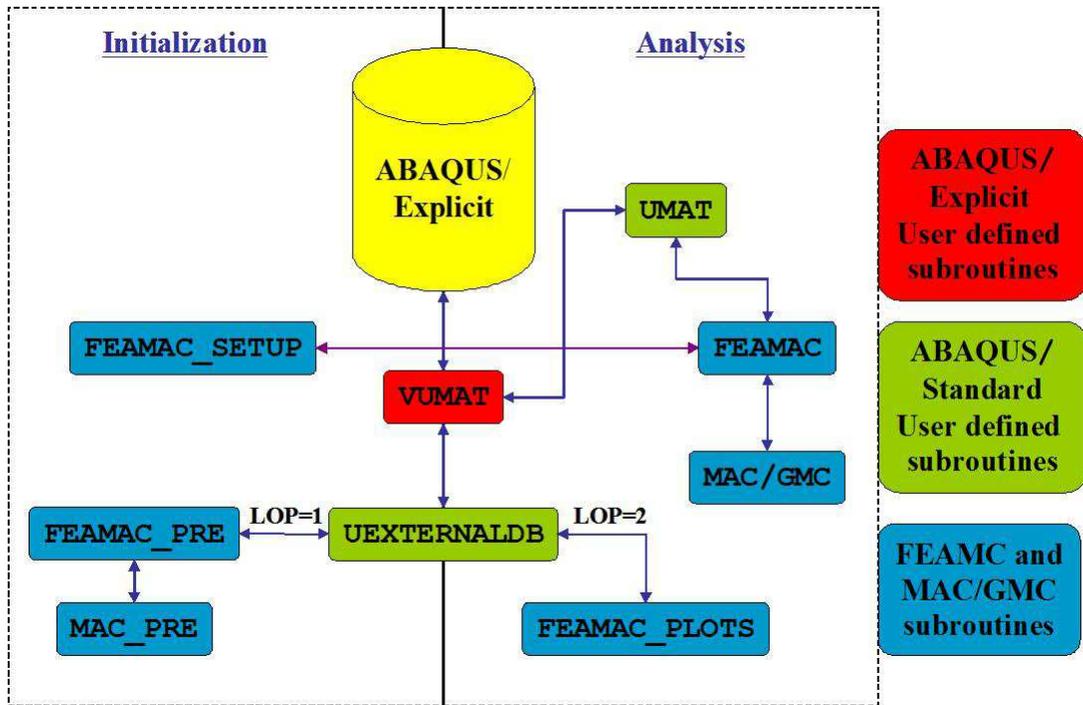
Fig. 3: Hierarchy of FEAMAC/Explicit software implementation.

The FEAMAC/Explicit software implementation includes the ABAQUS/Explicit `VUMAT`, in addition to all of the the FEAMAC subroutines described in the previous section, excluding `UEXPAN` and `SDVINI`. The hierarchy of the FEAMAC/Explicit software implementation is shown in Fig. 3.

The function of the `VUMAT` subroutine in FEAMAC/Explicit is to facilitate the communication between ABAQUS/ Explicit and the `UMAT` subroutine, originally used in ABAQUS/Standard (refer to Fig. 4 for schematic detailing the `VUMAT` subroutine). The ABAQUS/Standard `UMAT` uses different arguments than the ABAQUS/Explicit `VUMAT`. Furthermore, the arguments that are common to both `UMAT` and `VUMAT` differ in structure. ABAUQS/Explicit calls the `VUMAT` once for a given block of material points, and the `VUMAT` loops through all material points in that block, updating the stress state for each material point in the block (the strain increment in the x_3-direction must be calculated to maintain zero stress in that direction for plane stress problems). Thus, each argument is indexed by the material point number within that block. Moreover, an ABAQUS/Explicit model may contain multiple material blocks. Whereas, ABAQUS/Standard calls the `UMAT` for each material point in the model, rather than for a block of material points; therefore, none of the arguments in the `UMAT` are indexed by a material point number. This indexing of arguments in the `VUMAT` by material point numbers results in arguments that are vectors in the `UMAT` being arrays that are indexed by material point numbers in the `VUMAT` and arguments that were arrays in the `UMAT` remaining arrays in the `VUMAT`, but containing material point numbers in one column and all of the analogous `UMAT` array data in vector from in the other column. For instance the stress state σ_{ij} is represented in the `UMAT` by the vector "`STRESS(NDI+NSHR)`" where "`NDI`" is the number of direct stress components and "`NSHR`" is the number of shear stress components, but in the `VUMAT` the stress state is an array "`STRESSNEW(NBLOCK,NDIR+NSHR)`" where "`NBLOCK`" is the material point number within a block. Additionally in the `UMAT`, the deformation gradient at the beginning of a time increment F_{ij} is stored in the array "`DFGRD0(NDI+NSHR,NDI+NSHR)`"; whereas in the `VUMAT`, the array "`DEFGRADOLD(NBLOCK,NDI+2*NSHR)`" contains the deformation gradient information at the beginning of the time step. For this reason, the `VUMAT` must convert all of the arguments used when calling the `UMAT` into the appropriate from. Once, these variables are updated by the `UMAT` the `VUMAT` must convert them back to the form recognized by the `VUMAT`.
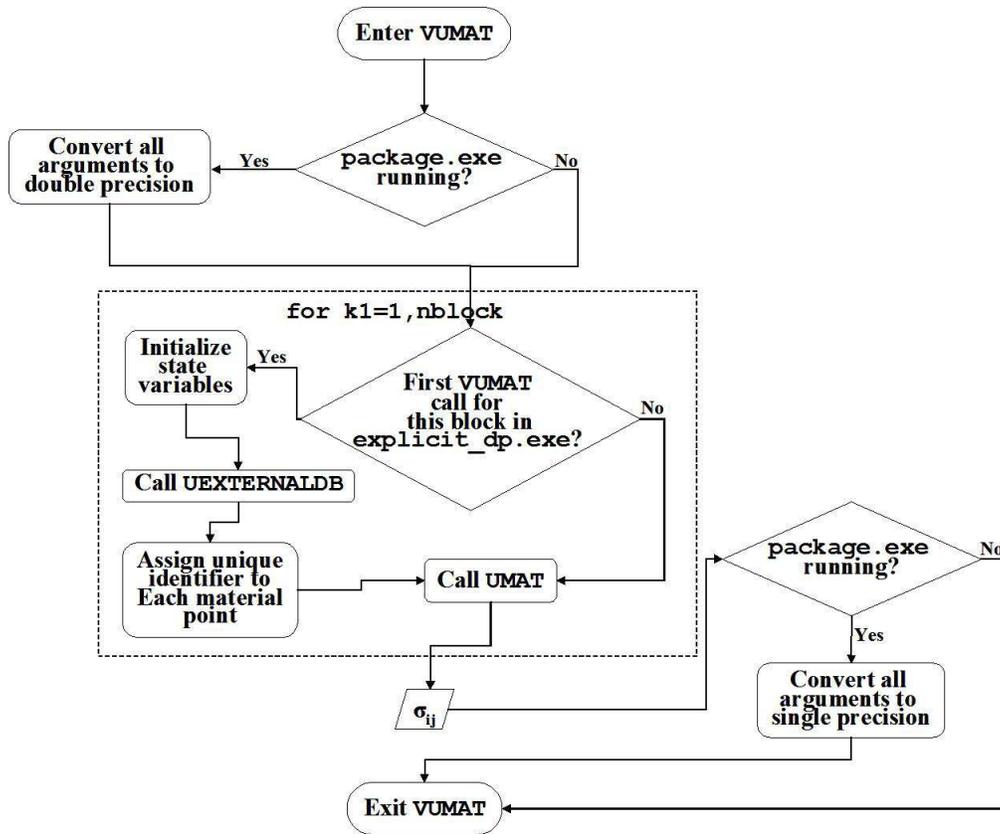
Fig. 4: FEAMAC/Explicit Algorithm.

Additionally, MAC/GMC must identify each material point in the model, to store data associated with that material point not stored in the ABAQUS/Explicit state variable space. Unfortunately, element number, integration point number, and section point number (for shells) are not VUMAT arguments. Therefore, the first time the VUMAT is called for a given block of material points within the double precision ABAQUS/Explicit executable explicit_dp.exe, the VUMAT assigns a unique identifier to each material point in that block. Furthermore, UEXTERNALDB is called and all state variables are initialized (previously implemented by calling SDVINI in FEAMAC) for each material point upon the first call of VUMAT, within explicit_dp.exe, for each unique block of material points.

Finally, ABAQUS/Standard runs all jobs in double precision, unless specified by the user; therefore, the variables in all of the subroutines called by the UMAT, which are contained in the FEAMAC .lib library, are double precision. ABAQUS/Explicit runs a preprocessing executable package.exe which passes in fictitious strains to the model, determines the stiffness in the model, and calculates the critical time step Δt using

$$\Delta t = \frac{L^e}{c_d} \tag{1}$$

where L^e is the characteristic element length, and c_d is the dilatational wave speed. The wave speed is calculated with

$$c_d = \sqrt{\frac{E}{\rho}} \tag{2}$$

where E is the material stiffness, and ρ is the material density. ABAQUS/Explicit uses the smallest calculated critical time step to govern the initial time incrementation in the problem [7]. This procedure is always executed using single precision. Since all of the variables in contained in the FEAMAC library are declared in double precision, the VUMAT

(a) Un-stiffened panel.   (b) Stiffened panel.
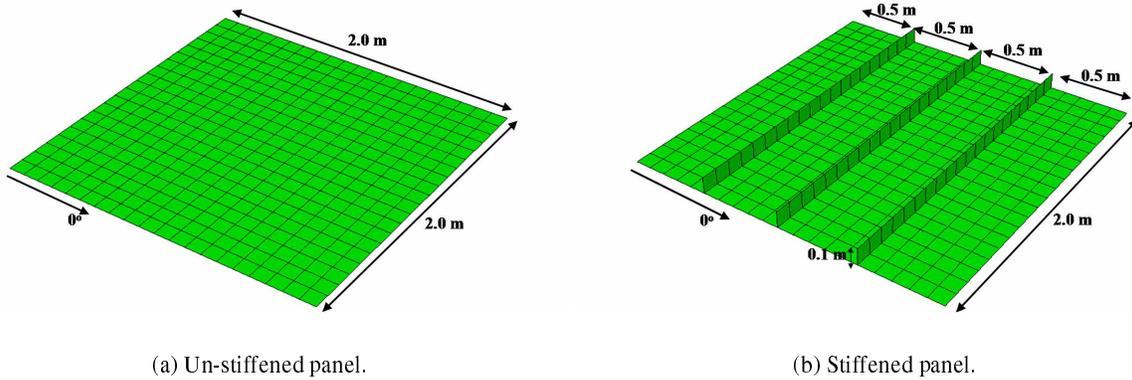
Fig. 5: Meshes used in blast loading of composite panels examples.

must convert the variables provided by ABAQUS/Explicit that are passed to the `UMAT` (which calls the other FEAMAC subroutines) from single precision to double precision. When the `UMAT` returns the updated variables, the `VUMAT` must convert these variables back to single precision from double precision.

3 FE MODEL - BLAST LOADING OF STIFFENED AND UN-STIFFENED COMPOSITE PANELS

To demonstrate the functionality of FEAMAC/Explicit two example problems of panels subjected to dynamic, blast loading scenarios are examined: a stiffened composite panel, and the same un-stiffened composite panel. The dimensions of the panels are 2.0 m x 2.0 m. The depth of the three stiffeners on the stiffened panel is 0.1 m and they are spaced 0.5 m apart. Both panels are meshed using plane stress S4R shell elements and shown in Fig. 5. The un-stiffened panel contains 440 elements and the stiffened panel contains 460 elements. The FEA is performed using ABAQUS/Explicit 6.7.

Both panels are composed of T800/3900-2 carbon fiber/epoxy composite. The elastic properties and density of a T800/3900-2 lamina are given in Table 1. The panel and stiffener lay-ups are given in Table 2 (lay-up angles are with respect to the 0° direction displayed in Fig. 5), along with the total thickness of the laminates. The lay-ups in the panels and stiffeners are almost identical; the stiffeners contain four less 0° layers.

A 2 subcell x 2 subcell RVE is used in MAC/GMC containing one fiber subcell (blue) and 3 matrix subcells (green). The x_2-x_3 plane of this architecture is displayed in Fig. 6, whjere the x_1-direction is the local fiber direction.

Property	Value
E_{11} (GPa)	160.0
E_{22} (GPa)	9.0
G_{12} (GPa)	6.2
ν_{12}	0.28
ρ (kg/m^3)	3540

Table 1: Elastic properties of T800/3900-2 lamina.

	Panels	Stiffeners
Lay-up	[45/0/0/0/-45/90]$_S$	[45/0/-45/90]$_S$
Thickness (mm)	23.8	15.8

Table 2: Lay-up of composite panels and stiffeners.
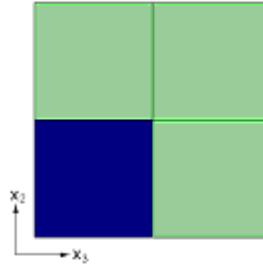
Fig. 6: 2 x 2 RVE used in MAC/GMC.

Fiber Properties	Values	Matrix Properties	Values
E_{11}^{f} (GPa)	293.0	E_{11}^{m} (GPa)	2.4
E_{22}^{f} (GPa)	91.0	E_{22}^{m} (GPa)	2.4
ν_{12}^{f}	0.23	ν_{12}^{m}	0.35
ν_{21}^{f}	0.45	ν_{21}^{m}	0.35
G_{12}^{f} (GPa)	55.2	G_{12}^{m} (GPa)	2.3

Table 3: Elastic properties of fiber and matrix constituents used in MAC/GMC.

The elastic properties of the fiber and matrix (3) are such that they produce lamina level properties consistent with those in Table 1. Note that the Young's modulus E_{ii}^{m}, Poisson's ratio ν_{ij}^{m}, and shear modulus G_{ij}^{m} are not direction dependent; however, the isotropic relationship between E_{ij}^{m}, ν_{ij}^{m}, and G_{ij}^{m} is not maintained in order to achieve consistent composite properties.

MAC/GMC is used to model failure in the matrix and fiber constituents. At each material point, MAC/GMC is used to resolve the applied strains into local subcell stresses and strains. The solution is semi-closed form; thus the subcells shown in Fig. 6 do not contain any integration points. Failure criteria are then evaluated at the micro-constituent level. A maximum strain criterion is utilized for the fiber in tension; compressive fiber failure is ignored. The quadratic Tsai-Hill criterion [11] is employed to model failure in the matrix. The axial strengths for this criterion are different in tension, represented with a superscript 'T', and in compression, marked with a superscript 'C'. All strengths used in the micromechanical model are presented in Table 4; a subscript 'f' indicates a fiber strength and an 'm' designates a matrix strength. Since the behavior of the matrix is independent of direction, all of the axial strengths (Y) are the same, and all of the shear strengths (S) are identical. The calculated subcell stresses are uniform within each subcell; so, any of the failure criteria are met in the subcells, the subcell properties are degraded to 0.01% of the virgin values. The global, composite properties are then recalculated by MAC/GMC, and the global stress state is updated. For this initial demonstration, all material properties are time independent; although, time dependent properties may by included in the analysis [4, 5].

Both the stiffened and un-stiffened panels are clamped on all four boundaries and subjected to a normal pressure blast load. The pressure histories applied to the two panels are displayed in Fig. 7. Since this study focuses on failure, the amplitude of the blast load applied to the stiffened panel is much higher than that applied to the un-stiffened panel to induce failure in both panels. The total simulation time is 50 msec, and the stable time increment calculated by ABAQUS/Explicit is 0.019 msec. Automatic time incrementation is used throughout the simulations.

| Strength	Value
ϵ_{f}^{U}	0.0125
Y_{m}^{T} (MPa)	147.4
S_{m}^{T} (MPa)	150.0
Y_{m}^{C}	120.7

Table 4: Constituent strengths used in MAC/GMC.
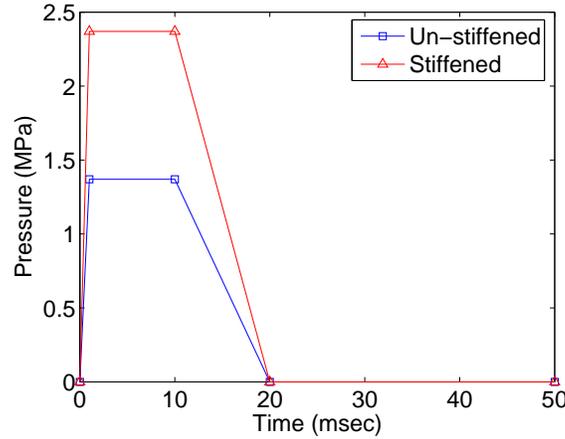
Fig. 7: Pressure histories (blast loads) applied to stiffened and un-stiffened panels.

4 RESULTS AND DISCUSSION

Since the focus of this report is to exemplify the ability of the FEAMAC/Explicit software implementation to link MAC/GMC to ABAQUS/Explicit, no comparison between experimental data and the results of the dynamic, multiscale model is made. Rather, a qualitative overview of the multiscale model is made to asses the functionality of FEAMAC/Explicit. The failure patterns for the un-stiffened and stiffened panels, subjected to blast loads, are shown at six different times in Figs. 8 and 9, respectively. A turquoise element indicates a single failed matrix subcell, green represents two failed matrix subcells, and three failed matrix subcells is visualized with yellow elements. Once all four subcells have failed, including the fiber subcell, the elements are removed. The corresponding Von Mises stress contours are plotted in Figs. 10 and 11.

In the un-stiffened panel, matrix failure develops at the clamped edges initially. This leads to complete failure (one failed fiber and three failed matrix subcells) in these regions. Failure then begins to propagate parallel to the top and bottom edges of the panel. At this point, only the boundaries of the panel remain intact at two of the corners. Stress concentrations develop around the failed regions; these failed elements essentially act like large voids in the material. Matrix failure also initiates at the edges of the stiffened panel; however, it is additionally distributed among the stiffeners. This leads to complete failure on the edges and at various points along the stiffeners, and eventually leads to complete failure developing in the center of the panel propagating in a direction perpendicular to the length of the stiffeners. The addition of the stiffeners effectively changed the failure mode of the panel and distributed the internal stresses produced by the pressure loading throughout the panel.

The elastic strain energy density of each material point is calculated using

$$W = \frac{1}{2}\sigma_{ij}\epsilon_{ij} \tag{3}$$

This energy density is integrated over the volume of the panel and used to obtain the total strain energy of the system throughout the simulation. This quantity is plotted against time along with the kinetic energy, applied work, and dissipated energy for the un-stiffened and stiffened panels in Fig. 12. The dissipated energy is calculated as

$$E_{dissipated} = E_{applied} - E_{strain} - E_{kinetic} \tag{4}$$

The dissipated energy is the energy dissipated due to the failure in the elements. Comparing Fig. 12(a) to Fig. 12(b), reveals that the addition of the stiffeners allows more applied energy to be transferred to kinetic energy, rather than dissipated due to failure or used to deform the panel (strain energy). So, the addition of stiffeners not only changes the failure mode of the panel and the distribution of stresses, but also alters how the applied energy is distributed among strain energy, kinetic energy, and energy dissipated due to failure. The dissipated energy is plotted, normalized by the

applied work, for both panels in Fig. 13. Observing Fig. 13 reveals that the un-stiffened panel is losing more energy due to failure than the stiffened panel. Unfortunately due to strain localization and mesh dependent effects resulting from failure, the element size used is too large to have confidence in the quantitative predictions of these simulations. However, it can be concluded that the stiffened panel can sustain higher pressure loads, but an un-stiffened panel will be more effective at dissipating energy caused by blast loads.


(a) t = 4.007 ms.

(b) t = 5.006 ms.

(c) t = 6.503 ms.

(d) t = 7.507 ms.

(e) t = 10.07 ms.

(f) t = 12.01 ms.
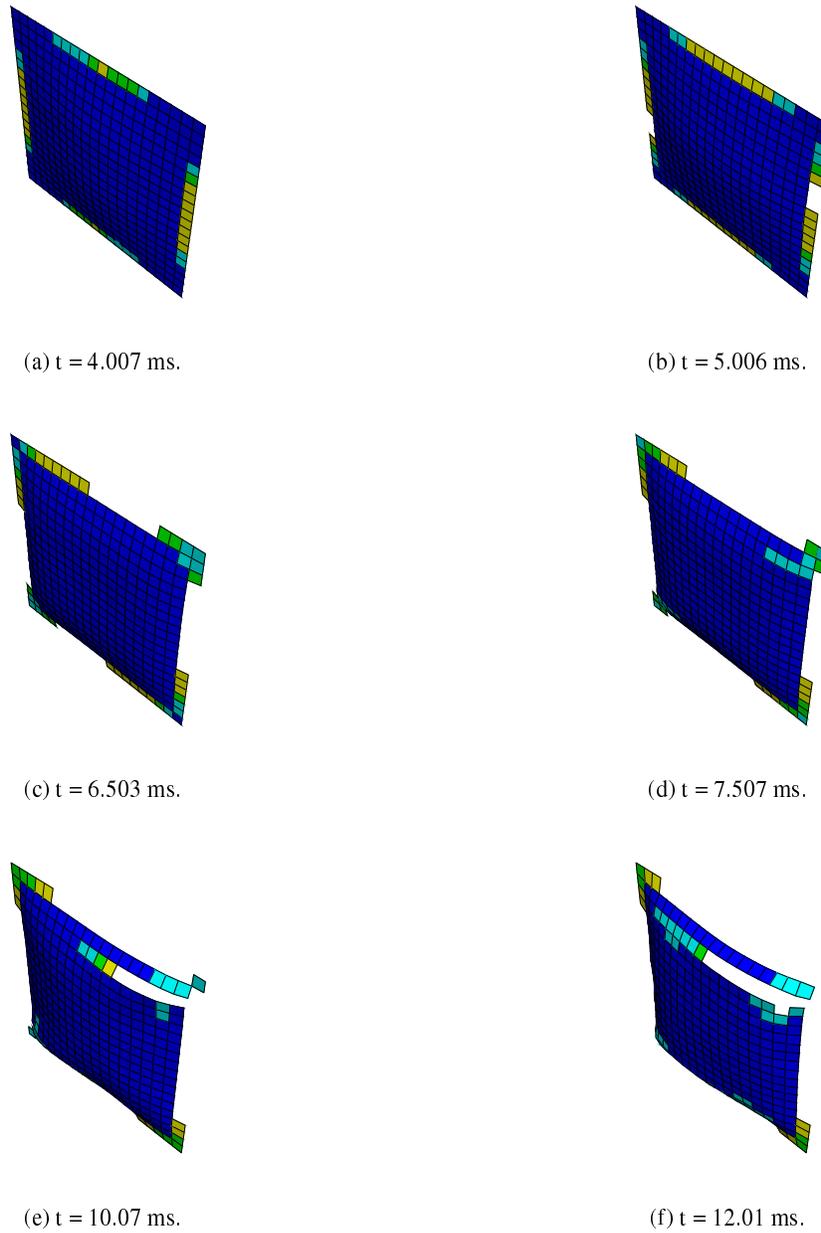
Fig. 8: Failure evolution of un-stiffened composite panel subjected to a blast load.
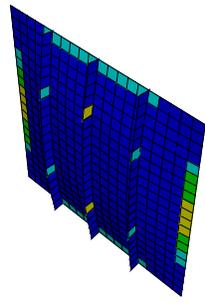
(a) t = 3.002 ms.
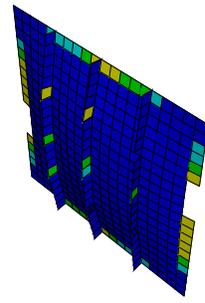
(b) t = 3.500 ms.
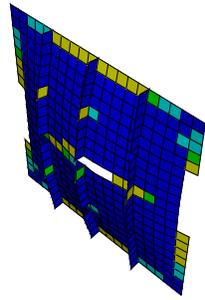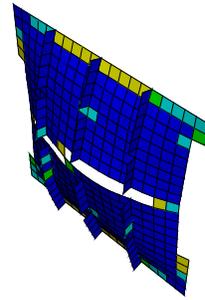
(c) t = 4.006 ms.

(d) t = 5.012 ms.
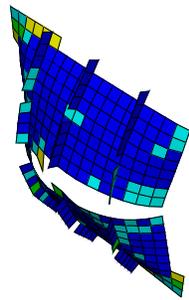
(e) t = 7.006 ms.
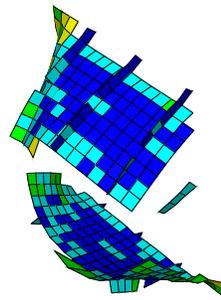
(f) t = 11.00 ms.

Fig. 9: Failure evolution of stiffened composite panel subjected to a blast load.
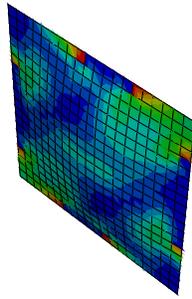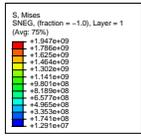
(a) t = 4.007 ms.
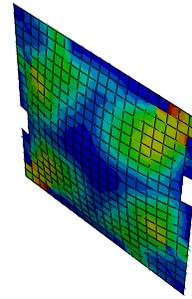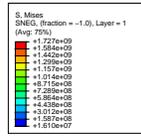
(b) t = 5.006 ms.
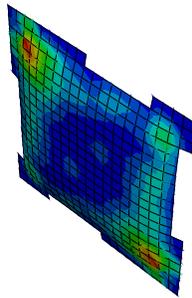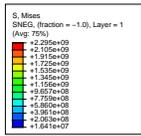
(c) t = 6.503 ms.
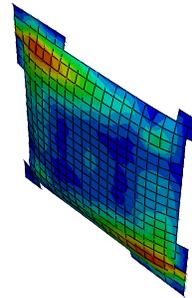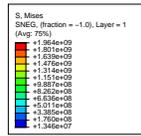
(d) t = 7.507 ms.
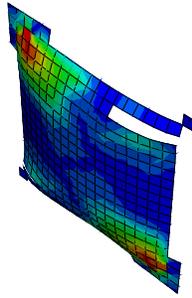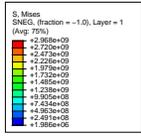
(e) t = 10.07 ms.
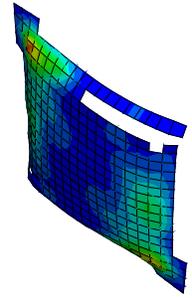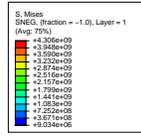
(f) t = 12.01 ms.

Fig. 10: Von Mises stress evolution of un-stiffened composite panel subjected to a blast load.
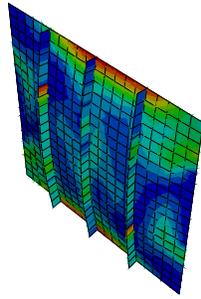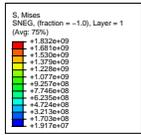
(a) t = 3.002 ms.
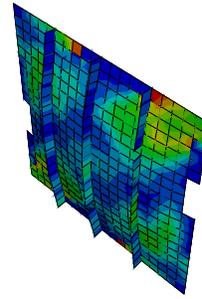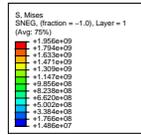
(b) t = 3.500 ms.
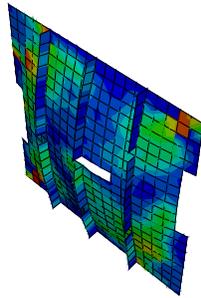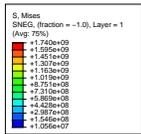
(c) t = 4.006 ms.
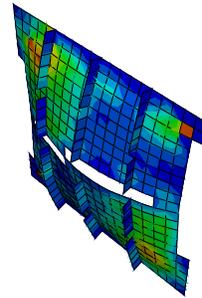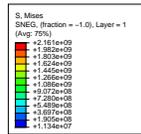
(d) t = 5.012 ms.
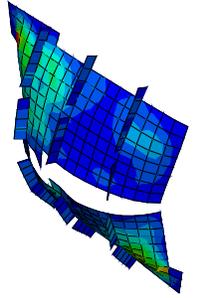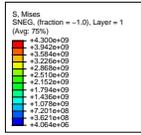
(e) t = 7.006 ms.
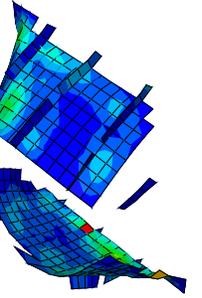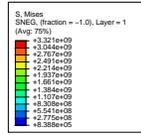
(f) t = 11.00 ms.

Fig. 11: Von Mises stress evolution of stiffened composite panel subjected to a blast load.

(a) Un-stiffened Panel.
(b) Stiffened Panel.
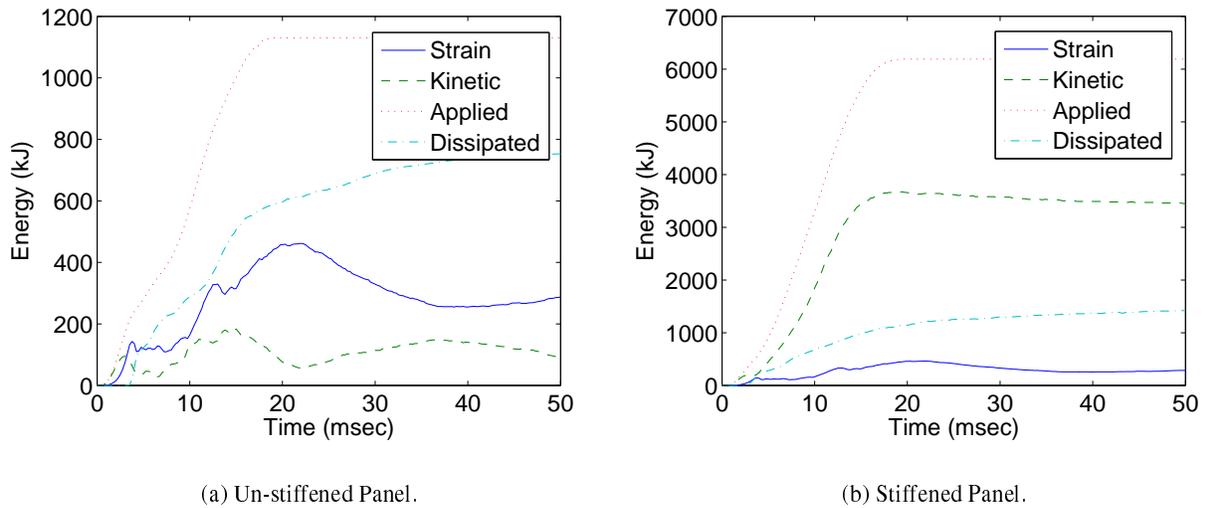
Fig. 12: Energy versus time for stiffened and un-stiffened panels.
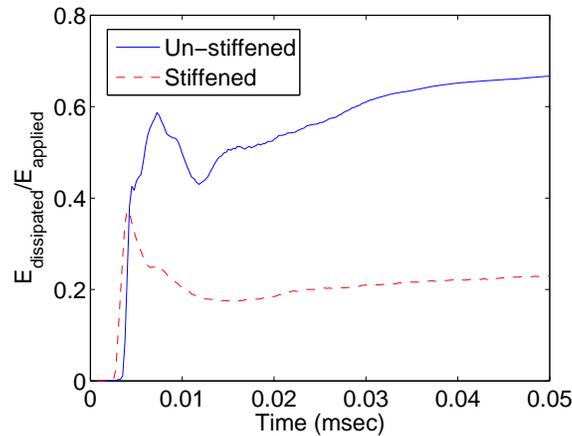

Fig. 13: Energy dissipated due to failure in elements normalized by applied work as a function of time for stiffened and un-stiffened panels.

5 CONCLUSIONS

This brief report documents the capabilities of the FEAMAC/Explicit software implementation, developed by NASA Glenn Research Center and the University of Michigan, which links the MAC/GMC suite of micromechanics codes to the ABAQUS/Explicit FE software. This development allows for multiscale methods involving an explicit FE solver. Dynamic problems maybe simulated with multiscale methods. Furthermore, static problems involving large changes in material properties and/or material softening can be analyzed using multiscale methods without convergence issues.

The failure capabilities of multiscale analyses using FEAMAC/Explicit are demonstrated in this report with the simulation of un-stiffened and stiffened composite panels subjected to time-dependent pressure blast loads. Since the primary focus of this report is to document the functionality of FEAMAC/Explicit, no comparisons with experimental data are made. The element size used in these rudimentary analyses is too large to make any concrete quantitative predictions; however, the general qualitative behavior of the panels is observed, as well as the benefits and disadvantages of adding stiffeners. The addition of stiffeners changes the failure mode of the panel, distributing the failure and

stresses throughout the panel rather than leaving them concentrated at the boundaries. Furthermore, including stiffeners increases the pressure necessary to induce catastrophic failure of the panel. More applied energy is transmitted to kinetic energy in the stiffened panel rather than to energy used to deform and fail the material, as in the un-stiffened panel. However if the objective if the panel is to absorb large portions of the applied energy, the un-stiffened panel is more desirable.

The examples presented successfully demonstrate the base capabilities of FEAMAC/Explicit. Due to the computational feasibility of MAC/GMC, this development opens avenues to numerous novel multiscale methods. With more refined material models and a suitable framework for progressive damage and failure[8], dynamic and static damage and failure in FRLs can be modeled more robustly by capturing the damage and failure evolution directly in the constituents of the composite.

References

1. Paley, M., and Aboudi J. (1992). Micromechanical analysis of composites by the generalized cells model. *Mechanics of Materials*, 14(2), 127-139.
2. Wilt, T.E. (1995). On the finite element implementation of the generalized method of cells micromechanics constitutive model. *NASA-CR-195451*.
3. Bednarcyk, B.A., Arnold, S.M. (2002). Full couple micro/macro deformation, damage, and failure prediction for SiC/Ti-15-3 laminates. *J. Aerosp. Eng.* 15(3), 74-83.
4. Bednarcyk, B.A., and Arnold, S.M. (2002). *MAC/GMC 4.0 User's Manual - Keywords Manual*, NASA/TM 2002-212077/VOL2.
5. Bednarcyk, B.A., and Arnold, S.M. (2002). *MAC/GMC 4.0 User's Manual - Example Problems Manual*, NASA/TM 2002-212077/VOL3.
6. Bednarcyk, B.A., Arnold, S.M. (2006). A framework for performing multiscale stochastic progressive failure analysis of composite structures. *Proceedings of the 2006 ABAQUS User's Conference, Boston, MA, May 23-25*.
7. *ABAQUS User's Manual*. (2003). Vol 1-3, Version 6.7, Hibbit, Karlsson, and Sorenson, Pawtucket, RI.
8. Pineda, E.J., Waas, A.M., Bednarcyk, B.A., Collier, C.S., Yarrington, P.W. (2008). A novel multiscale physics based progressive failure methodology for laminated composite structures. *49th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference*, 7 - 10 April, Schaumburg, Illinois.
9. Pineda, E.J., Waas, A.M., Bednarcyk, B.A., Collier, C.S., Yarrington, P.W. (2008). A novel multiscale physics based progrssive failure methodology for laminated composite structures. *NASA-TM-2008-215448*.
10. Pineda, E.J., Waas, A.M., Bednarcyk, B.A., Collier, C.S. (2008). A novel, multiscale high fidelity progressive damage and failure modeling approach for laminated fiber reinforced composites. *Proceedings of the American Society for Composites Twenty-third Technical Conference*, 9 - 11 September, Memphis, Tennessee.
11. Jones, R.M. (1999). *Mechanics of Composite Materials, 2nd Ed.*. Taylor and Francis, Inc., Philadelphia.

1. REPORT DATE (DD-MM-YYYY) 01-10-2009	2. REPORT TYPE Technical Memorandum	3. DATES COVERED (From - To)

4. TITLE AND SUBTITLE Multiscale Failure Analysis of Laminated Composite Panels Subjected to Blast Loading Using FEAMAC/Explicit	5a. CONTRACT NUMBER NNL07AA29C
	5b. GRANT NUMBER
	5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S) Pineda, Evan, J.; Waas, Anthony, M.; Bednarcyk, Brett, A.; Arnold, Steven, M.; Collier, Craig, S.	5d. PROJECT NUMBER
	5e. TASK NUMBER
	5f. WORK UNIT NUMBER WBS 984754.02.07.03.16.05

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration John H. Glenn Research Center at Lewis Field Cleveland, Ohio 44135-3191	8. PERFORMING ORGANIZATION REPORT NUMBER E-17075

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001	10. SPONSORING/MONITOR'S ACRONYM(S) NASA
	11. SPONSORING/MONITORING REPORT NUMBER NASA/TM-2009-215813

12. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified-Unlimited
Subject Category: 39
Available electronically at http://gltrs.grc.nasa.gov
This publication is available from the NASA Center for AeroSpace Information, 443-757-5802

13. SUPPLEMENTARY NOTES

14. ABSTRACT
This preliminary report demonstrates the capabilities of the recently developed software implementation that links the Generalized Method of Cells to explicit finite element analysis by extending a previous development which tied the generalized method of cells to implicit finite elements. The multiscale framework, which uses explicit finite elements at the global-scale and the generalized method of cells at the micro-scale is detailed. This implementation is suitable for both dynamic mechanics problems and static problems exhibiting drastic and sudden changes in material properties, which often encounter convergence issues with commercial implicit solvers. Progressive failure analysis of stiffened and un-stiffened fiber-reinforced laminates subjected to normal blast pressure loads was performed and is used to demonstrate the capabilities of this framework. The focus of this report is to document the development of the software implementation; thus, no comparison between the results of the models and experimental data is drawn. However, the validity of the results are assessed qualitatively through the observation of failure paths, stress contours, and the distribution of system energies.

15. SUBJECT TERMS
Micromechanics; Failure; Finite element method; Damage; Predictions; Laminates; Fiber composites

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON STI Help Desk (email:help@sti.nasa.gov)
a. REPORT U	b. ABSTRACT U	c. THIS PAGE U	UU	21	19b. TELEPHONE NUMBER (include area code) 443-757-5802

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std. Z39-18