AF-Shell 1.0 User Guide

Mark W. McElroy
Langley Research Center, Hampton, 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 Report Server, thus providing one of the largest collection 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 having 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.

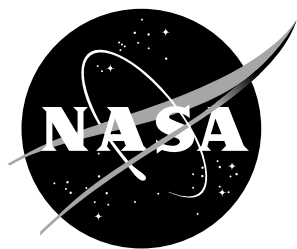- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.

- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or co-sponsored 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 organizing and publishing research results, distributing specialized research announcements and feeds, providing information desk and personal search support, and enabling data exchange services.

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 to `help@sti.nasa.gov`

- Phone the NASA STI Information Desk at 757-864-9658

- Write to:
  NASA STI Information Desk
  Mail Stop 148
  NASA Langley Research Center
  Hampton, VA 23681-2199

AF-Shell 1.0 User Guide

Mark W. McElroy
Langley Research Center, Hampton, Virginia

September 2017

Abstract

This document serves as a user guide for the AF-Shell 1.0 software, an efficient tool for progressive damage simulation in composite laminates. This guide contains minimal technical material and is meant solely as a guide for a new user to apply AF-Shell 1.0 to laminate damage simulation problems.

Contents

1 Introduction

AF-Shell 1.0 is a tool for simulating progressive damage in laminates fabricated from uni-directional tape that is intended to offer a level of computational efficiency and ease of use not currently found in existing high mesh fidelity modeling techniques for progressive damage in composites. The increased efficiency is achieved by using finite element models comprised of a single layer of enriched shell elements. AF-Shell 1.0 may be used for static or dynamic type analyses within the Abaqus/Standard (Implicit) solver.

The enrichment, based on use of the Floating Node Method (FNM) [1] and a damage algorithm based on the Virtual Crack Closure Technique [2], is such that, given a user-defined initial delamination, a damage path need not be predefined and can evolve in a model solution procedure in three dimensions throughout a laminate of varying ply orientations. The name, AF-Shell, stands for Adaptive Fidelity Shell, which refers to the adaptive mesh splitting approach that the FNM uses to split elements on demand as a damage pattern forms throughout an analysis solution. AF-Shell 1.0 is implemented as a user-defined element and external database subroutines in Abaqus/Standard. Thorough technical documentation of the model formulation can be found in several sources [3–7]. In addition to a description of the model formulation, these publications contain verification and validation cases of the model, along with some investigation into its current limitations.

2 Scope of Applicability

AF-Shell 1.0 should be used to evaluate damage growth starting from an initial user-defined delamination in a flat laminate plate. The nature of loading can be general, however, large and non-linear deformation type analyses (such as stability) are not yet included in the model capabilities. The following is a list of problems that AF-Shell 1.0 may be suitable for:

1. Test coupon design: The model can be used to quickly predict damage growth behavior in composite laminate test coupons. AF-Shell 1.0 may be useful in the design of test specimens and/or the planning of test programs.

2. Damage tolerance: The model can be used to evaluate damage growth potential from an existing characterized flaw in a plate/panel type structure.

3. Skin-stringer flange pull off: Given a small user-defined "starter delamination" at a skin-stringer flange interface, the model can be used to predict damage tolerance.

4. Large scale structures: The shell formulation and high efficiency of AF-Shell 1.0 potentially allows its use in quick and cost effective simulations of large structures. While an initial delamination is still required, damage growth scenarios can be quickly evaluated at multiple locations throughout the structure without extensive model modifications or the use of a local region consisting of a high fidelity mesh and damage modeling technique.

5. Compression after impact: AF-Shell 1.0 may be useful in predicting damage initiation and growth up until a point of instability for compression after impact analyses.

6. Rapid design tool: Any of the listed uses above may utilize AF-Shell 1.0 in a rapid design type function. AF-Shell 1.0 may help early on in a design process, to guide and inform design of structures that ultimately may be subjected to testing or more expensive detailed simulations.

3 Limitations

The following are limitations and considerations for the user to keep in mind while using AF-Shell 1.0.

1. Applicable only to laminates containing uni-directional fiber-reinforced-matrix composite plies. The plies may be oriented in varying directions throughout the laminate.

2. An initial delamination must be defined by the user (i.e., no damage initiation capability).

3. The initial delamination should contain a path of at least three successive "interior nodes" emanating in the crack wake from any delamination front node. For example, to satisfy this requirement, an initial delamination, square in geometry, that is not located on a free edge must have a minimum size of 4×4 elements. Examples of initial delaminations that are and are not acceptable are shown in Figure 1.
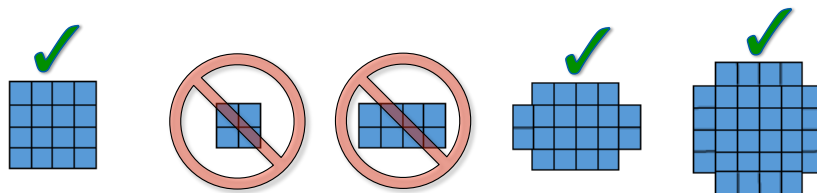
Figure 1. Examples of initial delaminations that are and are not acceptable.

4. A maximum of one delamination (i.e., initial or otherwise) can be included at a specific location in the mesh (i.e., one delamination per element).

5. The only damage modes included are delamination and transverse matrix cracking. Furthermore, transverse matrix cracking can only occur in the model if it is associated with a delamination-changing ply interface. Additional damage modes such as fiber breaking, fiber kinking, matrix crushing, etc. are not included in the AF-Shell 1.0 simulation capabilities.

6. Matrix cracking associated with delamination-migration that traverses multiple ply blocks of differing fiber orientation (i.e., "staggered migration" [5])

is represented in the mesh such that the model may be too flexible in these regions.

7. Care should be taken in sizing the mesh so that element edge dimensions are not less than 0.2 times the total laminate thickness.

8. Only square and rectangular elements are supported. The mesh should consist entirely of uniform square elements if possible.

9. The mesh should be aligned with the global model coordinate system.

10. Use caution if a bi-material crack tip condition exists. The model was verified to be accurate in one bi-material delamination problem [3], however, testing and investigation in this area is incomplete.

11. Progressive damage processes occurring in general layups consisting of transverse cracking and delamination-migration should be Mode II dominated for accurate transverse cracking predictions.

12. The initial delamination should be defined in a mesh located in the x-y plane (analysis of curved structures may be possible but this has not yet been verified or validated).

13. Maximum model size of 10,000 enriched shell elements and 50,000 nodes (including floating nodes).

4   Quick Start

AF-Shell 1.0 must be used in conjunction with Abaqus/Standard 6.14 as a user-defined element subroutine [8]. To model damage growth in a composite structure, an Abaqus input file consisting of a shell mesh should be created with the following modifications:

1. Enrich mesh with floating nodes: The mesh can be created initially as a single layer of native S4 shell elements. The nodes in the initial layer of shells are referred to as "real nodes". Each element is enriched by including a "floating node" in the input file at the same coordinates of each real node (i.e., the total number of nodes is doubled). The shell element definitions in the input file should be modified to include a total of 8 nodes in their definitions (i.e., append the original element definition with four floating nodes whose numbering convention follows the real nodes [8]).

2. Define the Abaqus parameters for user elements in the model using the following command:
```
*USER ELEMENT,TYPE=U6,NODES=8,COORDINATES=3,PROPERTIES=1,VARIABLES=6
1,2,3,4,5,6
```

3. Assign property to element set of user elements using the following command line (in this example the element set is named "userel"):

```
*UEL PROPERTY, ELSET=userel
1
```

4. Populate user-defined input file, "*afshelluser.txt*", with desired model parameters (see Section 5).

5. Create boundary condition at *all nodes in the model* to restrain degree of freedom 6 (i.e., θ_z).

A working directory should be created containing the following three files. The working directory should be on the same drive on which Abaqus is installed (i.e., in a Windows environment, if the working directory is on drive D:, the analysis cannot be run using Abaqus that is installed on drive C: or installed on a remote server).

1. job_filename.*inp* – Abaqus model input file

   (contents vary, see Section 8 for an example)

2. *afshelluser.txt* – text file containing user input model parameters

   (see Section 5 for contents)

3. *nabaqus.bat* – batch file (contents shown below)

```
@echo off
if NOT exist "afshelluser.txt" (ECHO "file not found (see nabaqus.bat
file)")
If NOT exist "afshelluser.txt" goto :exit
SET CWD=%cd%
"abaqus" %* :exit
```

A model is run by opening an Abaqus command prompt, navigating to the working directory, and using the modified .bat file "nabaqus" to execute a job by typing the command:

```
>nabaqus job=job_filename user=subroutine_filename interactive
```

By default, several variables, defined in Appendix B, central to the AF-Shell 1.0 code functionality are printed in the .msg file after each converged solution increment. The user may obtain detailed information from these variables which can be used to debug, plot model results (such as energy release rate along a delamination front), or gain an understanding of the model results that would otherwise be unobtainable.

The user may choose to print additional information beyond the default variables. This may include information related to nodal forces and displacements, element nodal ties, or the element mass and stiffness matrices. Printing this information may be useful in certain instances, but doing so can increase both runtime and .msg file size significantly. See Section 5 for further explanation.

5   User-Defined Parameters File

A text file should be included in the working directory named "*afshelluser.txt*". The file includes user-defined model parameters related to material properties, damage growth parameters, layup configuration, and some model parameters unique to AF-Shell 1.0. Appendix A contains a description of how to obtain selected user-defined parameters. The syntax of the user parameter file is similar to an Abaqus input file in that keywords preceded by an "*" identify the user parameter that is included in the next line. An example user input file is shown in Appendix C.1 and is related to an example model described in Section 8. A description of each user input parameter is shown below. The units should be consistent with those of the Abaqus input file.

*ELASTIC – Elastic material properties of cured lamina.
    Column 1 = E_{11} (Young's Modulus)
    Column 2 = E_{22} (Young's Modulus)
    Column 3 = ν_{12} (Poisson's ratio)
    Column 4 = G_{12} (Shear Modulus)
    Column 5 = G_{13} (Shear Modulus)
    Column 6 = G_{23} (Shear Modulus)

*DENSITY – Density of cured laminate.

*ZK – z-coordinates of ply interfaces from the bottom up (maximum of 101 entries).

*BETAK – Angle of fibers in each ply from the bottom up (maximum of 100 entries).

*CONTACT – Crack surface contact modeling technique. 0 = Abaqus surface-to-surface, 1 = z-tie, 2 = conditional z-tie (see Section 7).

*NPLYTOT – Total number of plies in layup.

*GIC – Mode I critical energy release rate (i.e., fracture toughness).

*GIIC – Mode II critical energy release rate (i.e., fracture toughness).

*GETA – Exponent parameter η used in the BK mixed-mode critical energy release rate equation [9].

*ZFRACLOC – User-defined initial delamination. Multiple rows can be used for multiple initial delaminations. For each row the columns correspond to:
    Column 1 = ply interface
    Column 2 = geometry type (1=rectangle or 2=circle, see below)
        1=rectangle
            Columns 3 and 4 = lower and upper x-limits of rectangle
            Columns 5 and 6 = lower and upper y-limits of rectangle

2=circle
  Columns 3 and 4 = x and y coordinates of circle center
  Column 5 = circle radius
  Column 6 = 0 (N/A, set to zero)

*DELADEF – Average area of elements (ideally element size is uniform).

*ALPHACRIT – Critical angle between shearing direction and bounding fibers such that delmaination-migration is allowed [6] (units are degrees). Based on preliminary investigation [7], this parameter should be defined in the range of 40°-55°.

*INITMUL – Multiplier for G_{IIc} at locations on a thin insert boundary (i.e., at the boundary of a PTFE insert used to simulate a pre-existing delamination).

*SPLITRAD – Number of elements ahead of crack tip that are split but tied at 4 nodes (default value is 8, minimum of 4 is recommended).

*SMOOTHING – Parameters to define application of Gaussian smoothing along a delamination front for energy release rate values (GT) or for in-plane shear tie force vector angle (FSA). The values included in the examples in this guide are the default and were found to be the most accurate in previous model verification and validation work [7].
  Column 1 = FSA number of smoothing passes (i.e., smoothing can be applied repeatedly to smooth already smoothed values. Set this to zero if FSA smoothing is not desired).
  Column 2 = FSA minimum number of nodes on a delamination front to apply smoothing (i.e., for example if this parameter is set to 16, FSA smoothing will not be performed on delamination fronts with 16 or fewer nodes).
  Column 3 = FSA number of nodes included in the Gaussian population on one side of a smoothed node (for example if this parameter is set equal to 1, a node that is being smoothed will include 1 node on either side along the delamination front in the Gaussian population when a smoothing factor is determined).
  Column 4 = GT number of smoothing passes (similar to Column 1).
  Column 5 = GT minimum number of nodes on a delamination front to apply smoothing (similar to column 2).
  Column 6 = GT number of adjacent nodes to include in the Gaussian population (similar to column 3).
  Column 7 = Apply GT smoothing to VCCT calculations? 1 = yes or 2 = no (recommend 2 in general use).

*DEBUGPRINT – Request printing extra information in .msg file to aid with debugging. 0 = no, >0 = element number of interest (can only request information on one element at a time).

*DEBUGLOC – Location in code where extra information should be printed. 0

= before analysis, 1 = start of each increment, 2 = end of each increment, 3 = element stiffness formulation. Include all desired locations, separated by a space, on one line. If line is blank, printing is suppressed.

Notes or comments can be included in the user parameter file following a comma after a keyword. An example of a comment included for the keyword *GIC is shown below:

```
*GIC, VALUE OBTAINED FROM TESTING MATERIAL SYSTEM C12K/R6376
0.33D0
```

6   Ghost Elements

User-defined elements are unable to be plotted in the Abaqus post-processor. Post processing AF-Shell 1.0 simulation results can be done by printing information to the .msg file or, if visualization of the deformed model results is required, by using "ghost elements" superimposed over the AF-Shell 1.0 enriched elements.

Ghost elements are defined using reduced integration membrane elements (Abaqus element type M3D4R) with stiffness approaching zero that are superimposed on the AF-Shell 1.0 element nodes. Different unconnected sets of ghost elements should be defined for the real nodes and the floating nodes. AF-Shell 1.0 is set up such that a ghost mesh is defined using membrane elements that have a user-defined material. The purpose of the user-defined material is so that a state variable (STV(1)) can be assigned to each ghost element, indicating the number of plies. As delaminations grow during a solution procedure and new elements are split, the state variable can be plotted in the Abaqus post-processor to indicate the delamination interface in each element. This visualization technique also illustrates the location of transverse matrix cracks. *Note that if ghost elements are used, computational burden is added to the solution and the efficiency benefits offered by the Floating Node Method are not fully realized.* The Abaqus input file structure for defining ghost elements is:

```
*Element, type=M3D4R, elset=ghostel


    (ghost element definitions)


*Orientation, name=Ori-1
1.,0.,0.,0.,1.,0.
3, 0.
*Membrane Section, elset=ghostel, material=UMAT
1.,
*Hourglass stiffness
1
```

The ghost elements are assigned the user material named "UMAT". This material

is defined as:

```
*Material, name=UMAT
*Depvar
8,
*User Material, constants=9
1E-9,1E-9,1E-9,0.32,.32,.45,1E-9,1E-9,
1E-9
```

7 Contact

Contact between delamination crack surfaces can be captured in AF-Shell 1.0 using three different techniques. The user must designate the desired approach in the user parameter file using keyword *CONTACT as described in Section 5. The three contact techniques and some comments on their use are listed below:

1. *Abaqus surface-to-surface contact*: The user-defines surfaces in the Abaqus input file between real and floating nodes according to typical contact modeling procedures in Abaqus. Ghost elements may also be used to define an element-based surface for contact. This option is best for problems where (i) mode-mixity throughout the model either varies or is unknown and (ii) frictional effects of the sliding crack surfaces need to be accounted for. Typical convergence difficulties related to contact modeling in Abaqus may arise when using this approach [8]. If no contact is desired, this option should be selected in the user parameter file but with the contact definition omitted from the Abaqus input file.

2. *z-tie contact*: The crack surfaces are tied together in the z-direction only (i.e., they can slide freely, but cannot separate). This option should only be used in cases where the (i) delaminations are Mode II dominated and (ii) friction does not need to be included. The z-tie contact option does not suffer from convergence difficulties seen in the surface-to-surface option.

3. *Conditional z-tie contact*: Same as z-tie contact, but the tie is removed if a real node and a floating node are trying to separate. This option should be used in cases where (i) mode-mixity throughout the model either varies or is unknown and (ii) friction does not need to be included. Contact "chattering", where a nodal location repeatedly comes in and out of contact, is possible when using conditional z-tie contact. This behavior can lead to convergence difficulties and/or inaccurate progressive damage predictions and as a result this option is poorly suited for dynamic analyses.

8 Example Problem: Double Cantilever Beam

An example AF-Shell 1.0 model is described in this section (run using Abaqus/Standard 6.14). A rudimentary model of a Double Cantilever Beam (DCB) is selected. The

example model is not based on an experiment. A summary of the specimen geometry and finite element model is shown in Figure 2. The test consists of a fiber reinforced composite laminate specimen containing a pre-existing delamination at the midplane through the width at one end. Opposing displacement controlled loads are applied normal to the specimen along the edge where the pre-existing delamination is located. Note that, in this example, the crack wake surfaces are not in contact with one another so, while the contact method is assigned to the z-tie technique, this functionality is not activated in this analysis. Complete user parameter and Abaqus input files for this example are included in Appendices C.1 and C.2, respectively.
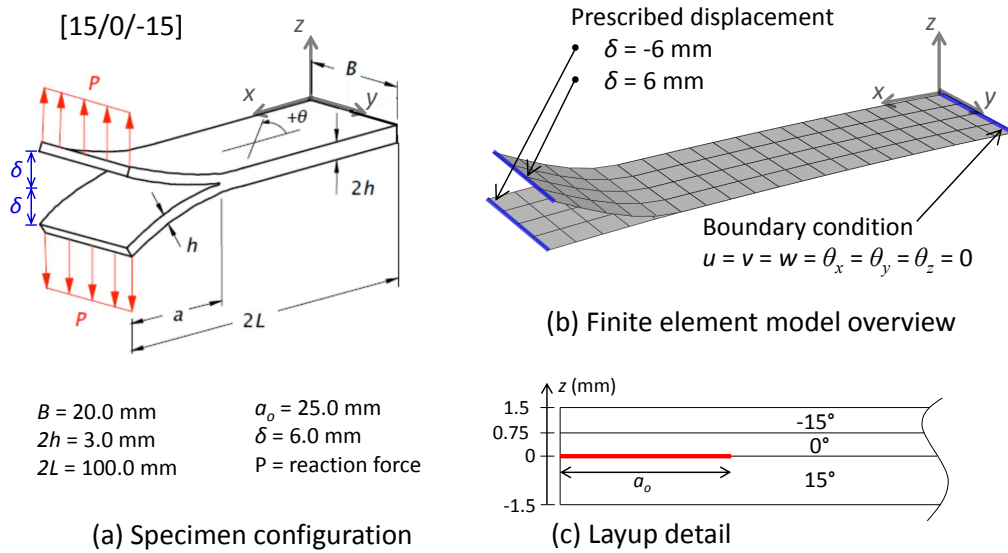

Figure 2. Summary of DCB specimen geometry.

After an initial linear elastic response, a peak force is reached and delamination growth initiates. The force-displacement (P-δ) response of the upper loading arm on the DCB is shown in Figure 3. Each peak in the sawtooth shape of the force displacement curve represents one row of nodal ties being released. If desired, the best way to minimize the sawtooth response in AF-Shell 1.0 is to refine the mesh. The deformed shape of the model at the beginning and end of the analysis is shown in Figure 4, where the contours correspond to state variable 1 (SDV1). AF-Shell 1.0 assigns SDV1 to each ghost element to represent the number of plies in that region of material.
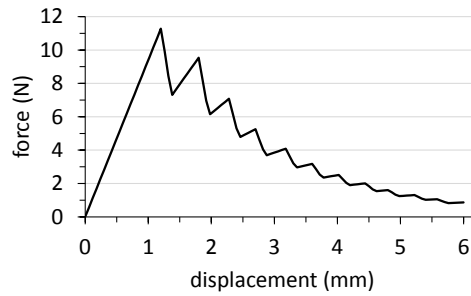
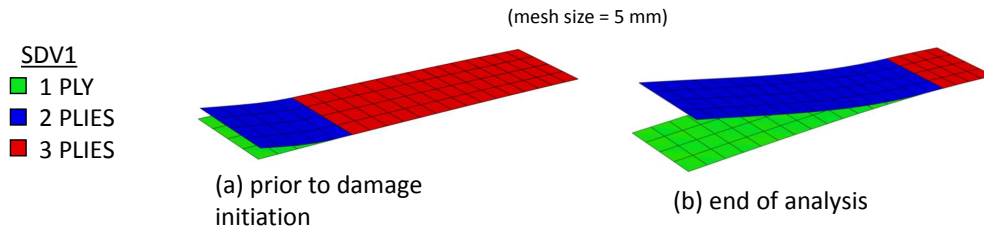Figure 3. Force-displacement response of upper loading arm in DCB.

(a) prior to damage initiation

(b) end of analysis

Figure 4. Deformed shape of DCB model at the beginning and end of the analysis.

9 Licensing and Distribution

The AFS software is licensed and distributed via the NASA Langley Research Center Technology Transfer Office. Interested parties should contact the Technology Transfer Office at:

NASA Langley Research Center
Mail Stop 151
Hampton, VA 23681-2199
Phone: (757) 864-1178
Email: LARC-DL-technologygateway@mail.nasa.gov

Alternatively, the technical point of contact can be reached at:

Mark McElroy
mark.w.mcelroy@nasa.gov

References

1. B Chen, S Pinho, N De Carvalho, P Baiz, and T Tay. A floating node method for the modelling of discontinuities in composites. *Engineering Fracture Mechanics*, 127:104–134, 2014.

2. R Krueger. Virtual crack closure technique: history, approach, and applications. *Applied Mechanics Reviews*, 57(2):109–143, 2004.

3. M McElroy. *An enriched shell finite element for progressive damage simulation in composite laminates, NASA/TM 2016-219211.* NASA, 2016.

4. M McElroy. Use of an enriched shell finite element to simulate delamination-migration in a composite laminate. *Composite Structures*, 167:88–95, 2017.

5. M McElroy, W Jackson, R Olsson, P Hellstrom, S Tsampas, and M Pankow. Interaction of delaminations and matrix cracks in a CFRP plate, Part I: A test method for model validation (submitted). *Composites Part A: Applied Science and Manufacturing*, 2017.

6. M McElroy, R Gutkin, and M Pankow. Interaction of delaminations and matrix cracks in a CFRP plate, Part II: Simulation using an enriched shell finite element model (submitted). *Composites Part A: Applied Science and Manufacturing*, 2017.

7. M McElroy. *An Enriched Shell Finite Element for Progressive Damage Simulation in Composite Laminates.* PhD thesis, North Carolina State University, Raleigh, NC, 2016.

8. *Dassault Systems, Abaqus 6.13 Analysis User Manual.* 2013.

9. M Benzeggagh and M Kenane. Measurement of mixed-mode delamination fracture toughness of unidirectional glass/epoxy composites with mixed-mode bending apparatus. *Composites Science and Technology*, 56(4):439–449, 1996.

10. ASTM D3039-01 Standard test method for tensile properties of polymer matrix composite materials. In *2004 Annual Book of Standards, Volume 15.03*. ASTM International, West Conshohoken, PA (USA), 2004.

11. ASTM D5379-01 Standard test method for shear properties of composite materials by the V-notched beam method. In *2004 Annual Book of Standards, Volume 15.03*. ASTM International, West Conshohoken, PA (USA), 2004.

12. ASTM D5528-01 Standard test method for mode I interlaminar fracture toughness of unidirectional fiber-reinforced polymer matrix composites. In *2004 Annual Book of Standards, Volume 15.03*. ASTM International, West Conshohoken, PA (USA), 2004.

13. ASTM D7905 Standard test method for determination of the mode II interlaminar fracture toughness of unidirectional fiber-reinforced polymer matrix composites. In *2014 Annual Book of ASTM Standards, Volume 15.03*. ASTM International, West Conshohoken, PA (USA), 2014.

14. ASTM D6671m-04 Standard test method for mixed mode I-mode II interlaminar fracture toughness of unidirectional fiber-reinforced polymer matrix composites. In *2004 Annual Book of Standards, Volume 15.03*. ASTM International, West Conshohoken, PA (USA), 2004.

15. T O'Brien, W Johnston, and G Toland. *Mode II interlaminar fracture toughness and fatigue characterization of a graphite epoxy composite material, NASA/TM 2010-216838*. NASA, 2010.

Appendices

A   Obtaining Selected User Input Parameters

*ELASTIC

E_{11}, E_{22}, and ν_{12}: ASTM D3039 Standard Test Method for Tensile Properties of Polymer Matrix Composite Materials: Flat material specimen placed into mechanical frame, monotonic tensile load applied; force, displacement, and strain recorded [10].

G_{12}, G_{23}, G_{13}: ASTM D5379 Standard Test Method for Shear Properties of Composite Materials by the V-Notched Beam Method: Rectangular beam specimen with two symmetric V-notches placed in a special shear load fixture; strain gauges at +/- 45 degree in the center of the beam measure shear response [11].

*GIC

G_{Ic}: ASTM D5528 Standard Test Method for Mode I Interlaminar Fracture Toughness of Unidirectional Fiber-Reinforced Polymer Matrix Composites: Flat unidirectional laminate specimen containing a non-adhesive insert to act as a crack starter, loaded out of plane by opposing tensile forces to open and grow crack (DCB test); load and delamination length are recorded. Mode I interlaminar fracture toughness is calculated using a modified beam theory or compliance calibration method [12].

*GIIC

G_{IIc}: ASTM D7905 Standard Test Method for Determination of the Mode II Interlaminar Fracture Toughness of Unidirectional Fiber-Reinforced Polymer Matrix Composites: Rectangular unidirectional laminate specimen containing a non-adhesive insert to act as a crack starter, supported at two ends by rollers and loaded out of plane in between the supports (End Notch Flexure test). Mode II Interlaminar Fracture Toughness is obtained using the compliance calibration method [13].

*GETA

η_{BK}: Curve fit the the Benzeggah-Kenane equation [9]

$$G_c = G_{Ic} + (G_{IIc} - G_{Ic})(G_{II}/G_T)^{\eta_{BK}} \tag{1}$$

to three G_c vs. G_{II}/G_T data points. Two of these data points are known, i.e., pure Mode I and pure Mode II. A third must be determined experimentally (could use a Mixed-mode Bending test to obtain the experimental value [14]).

*ALPHACRIT

α_c - Perform a biaxial-bending test as described by McElroy et al. [5]. If performing this test is not feasible, perform a numerical parametric study using AF-Shell where α_c varies between 40° and 55° and choose a value for which transverse cracking is logical in terms of fiber direction and crack continuity (i.e., a predicted transverse crack should not be such that it breaks fibers).

*INITMUL

f_{IIc} - Use data from O'Brien et al. [15] or perform a similar study. The average value determined by O'Brien et al. was 1.59 for an IM7/8552 carbon-epoxy material system containing a thin Polytetrafluoroethylene insert used to simulate an initial delamination.

B   Selected Variable Definitions

The following variables are printed to the message file (*.msg*) during an analysis. The data contained in these variables may be used to debug, generate plots, or gain insight to the model results.

DELAMAREA – Total area of delamination at each ply interface. Each row corresponds to a ply interface. The rows are ordered by interface number from the bottom up in the laminate.
    Column 1 = delamination area

FORMOD – Contains information on smoothed in-plane shear tie force vector angle (FSA). Each row corresponds to one node along the delamination front. The rows are sorted in nodal order along the front.
    Column 1 = node number
    Column 2 = FSA before Gaussian smoothing (degrees)
    Column 3 = FSA after Gaussian smoothing (degrees)
    Column 4 = F_x tie force component after Gaussian smoothing
    Column 5 = F_y tie force component after Gaussian smoothing
    Column 6 = Magnitude of shear tie force vector (i.e., resultant) after Gaussian smoothing

FRONTFOR – A three dimensional matrix containing information on tie forces and adjacent node numbers for each delamination front node. Each row corresponds to a different tied delamination front node. The rows are sorted in nodal order along the delamination front. The 1st index in the third dimension corresponds to forces before Gaussian smoothing. The 2nd index in the third dimension corresponds to values after Gaussian smoothing.
    1st index in third dimension:
        Column 1 = node number
        Column 2 = F_x tie force component before Gaussian smoothing
        Column 3 = F_y tie force component before Gaussian smoothing
        Column 4 = F_z tie force component before Gaussian smoothing
        Column 5 = M_x tie moment component before Gaussian smoothing
        Column 6 = M_y tie moment component before Gaussian smoothing
        Column 7 = adjacent node in -x direction
        Column 8 = adjacent node in +x direction
        Column 9 = adjacent node in -y direction
        Column 10 = adjacent node in +y direction

2nd index in third dimension:
    Column 1 = node number
    Column 2 = F_x tie force component after Gaussian smoothing
    Column 3 = F_y tie force component after Gaussian smoothing
    Column 4 = F_z tie force component before Gaussian smoothing
    Column 5 = M_x tie moment component before Gaussian smoothing
    Column 6 = M_y tie moment component before Gaussian smoothing
    Column 7 = adjacent node in -x direction
    Column 8 = adjacent node in +x direction
    Column 9 = adjacent node in -y direction
    Column 10 = adjacent node in +y direction

FRONTINFO – Matrix containing energy release rate and delamination growth information at tied nodes along a delamination front. Each row corresponds to a different front node. The rows are sorted in nodal order along the delamination front.
    Column 1 = node number
    Column 2 = total energy release rate, G_T
    Column 3 = critical energy release rate, G_c
    Column 4 = Mode I energy release rate, G_I
    Column 5 = Mode II energy release rate, G_{II}
    Column 6 = Mode II ratio, G_{II}/G_T
    Column 7 = number of adjacent tied nodes
    Column 8 = length of delamination advance, Δa
    Column 9 = x-coordinate of node
    Column 10 = y-coordinate of node
    Column 11 = direction of delamination propagation
        (1=+x, -1=-x, 2=+y, -2=-y)
    Column 12 = tie shear force in propagation direction
    Column 13 = F_x tie force component after Gaussian smoothing
    Column 14 = F_y tie force component after Gaussian smoothing

MIGRATION – Energy release rate information at tied nodes along a delamination front. Each row corresponds to a different node. The rows are sorted in nodal order along the delamination front.
    Column 1 = node number
    Column 2 = Mode II energy release rate calculated for delamination growth in the x direction, GSX
    Column 3 = Mode II energy release rate calculated for delamination growth in the y direction, GSY
    Column 4 = $\sqrt{GSX^2 + GSY^2}$
    Column 5 = $tan^{-1}(GSY/GSX)$

MIGRATION_EL – Contains information for each element in the direction of delamination growth that contains a nodal tie that was released in the previous solution increment. Information in this matrix is related to evaluation of whether or not

migration occurs when this element is split as part of delamination growth. Each row corresponds to a different element.

Column 1 = element number

Column 2 = previous delamination ply interface (i.e., in the wake of delamination growth)

Column 3 = new delamination interface

Column 4 = released node number associated with delamination growth in this element

Column 5 = previous angle of bounding fibers (degrees)

Column 6 = new angle of bounding fibers (degrees)

Column 7 = direction of delamination growth (1=+x, -1=-x, 2=+y, -2=-y)

Column 8 = tie shear force angle, FSA (after Gaussian smoothing)

Column 9 = difference in angle between FSA and bounding fibers, α

Column 10 = did delamination-migration occur? 1 = yes, 2 = no

Column 11 = energy release rate associated with transverse crack growth, G_{mig}

Column 12 = transverse growth tendency of delamination
(1 = ply above, 2 = ply below)

Column 13 = index RATIOMIG determining if migration is forced in order to maintain transverse matrix crack continuity (if RATIOMIG<=1.49, a transverse matrix crack exists in an adjacent element and therefore migration may be forced to maintain crack continuity)

NODEMASTER – Contains information at each node in the model concerning tie forces and relative displacements used in the VCCT calculations. Each row corresponds to a different node number.

Column 1 = node number

Column 2 = F_x tie force component

Column 3 = F_y tie force component

Column 4 = F_z tie force component

Column 5 = M_x tie moment component

Column 6 = M_y tie moment component

Column 7 = adjacent node number located in the crack wake for delamination growth in the $+x$ direction

Column 8 = relative Mode II crack wake opening for delamination growth in the $+x$ direction

Column 9 = relative Mode I crack wake opening for delamination growth in the $+x$ direction

Column 10 = relative crack wake opening rotation (θ_y) for delamination growth in the $+x$ direction

Column 11 = adjacent node number located in the crack wake for delamination growth in the -x direction

Column 12 = relative Mode II crack wake opening for delamination growth in the -x direction

Column 13 = relative Mode I crack wake opening for delamination growth in the -x direction

Column 14 = relative crack wake opening rotation (θ_y) for delamination growth

in the -x direction

Column 15 = adjacent node number located in the crack wake for delamination growth in the +y direction

Column 16 = relative Mode II crack wake opening for delamination growth in the +y direction

Column 17 = relative Mode I crack wake opening for delamination growth in the +y direction

Column 18 = relative crack wake opening rotation (θ_x) for delamination growth in the +y direction

Column 19 = adjacent node number located in the crack wake for delamination growth in the -y direction

Column 20 = relative Mode II crack wake opening for delamination growth in the -y direction

Column 21 = relative Mode I crack wake opening for delamination growth in the -y direction

Column 22 = relative crack wake opening rotation (θ_x) for delamination growth in the -y direction

Column 23 = is the node tied to its corresponding floating node?
$\quad$ 0 = no, 1 = yes, 2 = release tie in this step

STATEMASTER – Designates, for all elements, split location and whether or not each real node is tied to its corresponding floating node. Each row corresponds to a different element.

Column 1 = element number

Column 2 = is element node 1 tied? 0 = no, 1 = yes

Column 3 = is element node 2 tied? 0 = no, 1 = yes

Column 4 = is element node 3 tied? 0 = no, 1 = yes

Column 5 = is element node 4 tied? 0 = no, 1 = yes

Column 6 = delamination ply interface (0 = no delamination)

C Example Problem Input Files

C.1 Example Problem User Parameter File (filename: *"afshelluser.txt"*)

```
dcb2.inp
*ELASTIC
139400 10160 0.3 4600 4600 3540
*DENSITY
.00000000159
*ZK
-1.5 0 0.75 1.5
*BETAK
15 0 -15
*NPLYTOT
3
*CONTACT
1
*GIC
0.33D0
*GIIC
0.8D0
*GETA
2.284D0
*ZFRACLOC
2 1 75 200 -1 100
*DELADEF
6.25
*ALPHACRIT
55
*INITMUL
1.0
*SPLITRAD
8
*SMOOTHING
1 16 1 1 16 1 2
*DEBUGPRINT
1
*DEBUGLOC
0 1
```

C.2 Example Problem Abaqus Input File (filename: *"dcb2.inp"*)

```
**Example user input file for AF-Shell 1.0 model
**This input file is published in the AF-Shell 1.0 User Manual (2017)
**Nodes 1-105 are "real nodes"
**Nodes 106-210 are "floating nodes"
**Elements 1-80 are user elements (i.e., enriched shell elements)
**Elements 50001-50160 are ghost elements
**
**Input file begins here:
**
*Part, name=Part-1
*End Part
*Assembly, name=Assembly
*Instance, name=Part-1-1, part=Part-1
0, 0, 0.
*Node
1,0,0,0
2,5,0,0
3,10,0,0
4,15,0,0
5,20,0,0
6,25,0,0
7,30,0,0
8,35,0,0
9,40,0,0
10,45,0,0
11,50,0,0
12,55,0,0
13,60,0,0
14,65,0,0
15,70,0,0
16,75,0,0
17,80,0,0
18,85,0,0
19,90,0,0
20,95,0,0
21,100,0,0
22,0,5,0
23,5,5,0
24,10,5,0
25,15,5,0
26,20,5,0
27,25,5,0
28,30,5,0
29,35,5,0
30,40,5,0
31,45,5,0
32,50,5,0
33,55,5,0
34,60,5,0
35,65,5,0
```

```
36,70,5,0
37,75,5,0
38,80,5,0
39,85,5,0
40,90,5,0
41,95,5,0
42,100,5,0
43,0,10,0
44,5,10,0
45,10,10,0
46,15,10,0
47,20,10,0
48,25,10,0
49,30,10,0
50,35,10,0
51,40,10,0
52,45,10,0
53,50,10,0
54,55,10,0
55,60,10,0
56,65,10,0
57,70,10,0
58,75,10,0
59,80,10,0
60,85,10,0
61,90,10,0
62,95,10,0
63,100,10,0
64,0,15,0
65,5,15,0
66,10,15,0
67,15,15,0
68,20,15,0
69,25,15,0
70,30,15,0
71,35,15,0
72,40,15,0
73,45,15,0
74,50,15,0
75,55,15,0
76,60,15,0
77,65,15,0
78,70,15,0
79,75,15,0
80,80,15,0
81,85,15,0
82,90,15,0
83,95,15,0
84,100,15,0
85,0,20,0
86,5,20,0
87,10,20,0
```

```
88,15,20,0
89,20,20,0
90,25,20,0
91,30,20,0
92,35,20,0
93,40,20,0
94,45,20,0
95,50,20,0
96,55,20,0
97,60,20,0
98,65,20,0
99,70,20,0
100,75,20,0
101,80,20,0
102,85,20,0
103,90,20,0
104,95,20,0
105,100,20,0
106,0,0,0
107,5,0,0
108,10,0,0
109,15,0,0
110,20,0,0
111,25,0,0
112,30,0,0
113,35,0,0
114,40,0,0
115,45,0,0
116,50,0,0
117,55,0,0
118,60,0,0
119,65,0,0
120,70,0,0
121,75,0,0
122,80,0,0
123,85,0,0
124,90,0,0
125,95,0,0
126,100,0,0
127,0,5,0
128,5,5,0
129,10,5,0
130,15,5,0
131,20,5,0
132,25,5,0
133,30,5,0
134,35,5,0
135,40,5,0
136,45,5,0
137,50,5,0
138,55,5,0
139,60,5,0
```

```
140,65,5,0
141,70,5,0
142,75,5,0
143,80,5,0
144,85,5,0
145,90,5,0
146,95,5,0
147,100,5,0
148,0,10,0
149,5,10,0
150,10,10,0
151,15,10,0
152,20,10,0
153,25,10,0
154,30,10,0
155,35,10,0
156,40,10,0
157,45,10,0
158,50,10,0
159,55,10,0
160,60,10,0
161,65,10,0
162,70,10,0
163,75,10,0
164,80,10,0
165,85,10,0
166,90,10,0
167,95,10,0
168,100,10,0
169,0,15,0
170,5,15,0
171,10,15,0
172,15,15,0
173,20,15,0
174,25,15,0
175,30,15,0
176,35,15,0
177,40,15,0
178,45,15,0
179,50,15,0
180,55,15,0
181,60,15,0
182,65,15,0
183,70,15,0
184,75,15,0
185,80,15,0
186,85,15,0
187,90,15,0
188,95,15,0
189,100,15,0
190,0,20,0
191,5,20,0
```

```
192,10,20,0
193,15,20,0
194,20,20,0
195,25,20,0
196,30,20,0
197,35,20,0
198,40,20,0
199,45,20,0
200,50,20,0
201,55,20,0
202,60,20,0
203,65,20,0
204,70,20,0
205,75,20,0
206,80,20,0
207,85,20,0
208,90,20,0
209,95,20,0
210,100,20,0
*USER ELEMENT,TYPE=U6,NODES=8,COORDINATES=3,PROPERTIES=1,VARIABLES=6
1,2,3,4,5,6
*Element, type=U6, elset=userel
1,1,2,23,22,106,107,128,127
2,2,3,24,23,107,108,129,128
3,3,4,25,24,108,109,130,129
4,4,5,26,25,109,110,131,130
5,5,6,27,26,110,111,132,131
6,6,7,28,27,111,112,133,132
7,7,8,29,28,112,113,134,133
8,8,9,30,29,113,114,135,134
9,9,10,31,30,114,115,136,135
10,10,11,32,31,115,116,137,136
11,11,12,33,32,116,117,138,137
12,12,13,34,33,117,118,139,138
13,13,14,35,34,118,119,140,139
14,14,15,36,35,119,120,141,140
15,15,16,37,36,120,121,142,141
16,16,17,38,37,121,122,143,142
17,17,18,39,38,122,123,144,143
18,18,19,40,39,123,124,145,144
19,19,20,41,40,124,125,146,145
20,20,21,42,41,125,126,147,146
21,22,23,44,43,127,128,149,148
22,23,24,45,44,128,129,150,149
23,24,25,46,45,129,130,151,150
24,25,26,47,46,130,131,152,151
25,26,27,48,47,131,132,153,152
26,27,28,49,48,132,133,154,153
27,28,29,50,49,133,134,155,154
28,29,30,51,50,134,135,156,155
29,30,31,52,51,135,136,157,156
30,31,32,53,52,136,137,158,157
```

```
31,32,33,54,53,137,138,159,158
32,33,34,55,54,138,139,160,159
33,34,35,56,55,139,140,161,160
34,35,36,57,56,140,141,162,161
35,36,37,58,57,141,142,163,162
36,37,38,59,58,142,143,164,163
37,38,39,60,59,143,144,165,164
38,39,40,61,60,144,145,166,165
39,40,41,62,61,145,146,167,166
40,41,42,63,62,146,147,168,167
41,43,44,65,64,148,149,170,169
42,44,45,66,65,149,150,171,170
43,45,46,67,66,150,151,172,171
44,46,47,68,67,151,152,173,172
45,47,48,69,68,152,153,174,173
46,48,49,70,69,153,154,175,174
47,49,50,71,70,154,155,176,175
48,50,51,72,71,155,156,177,176
49,51,52,73,72,156,157,178,177
50,52,53,74,73,157,158,179,178
51,53,54,75,74,158,159,180,179
52,54,55,76,75,159,160,181,180
53,55,56,77,76,160,161,182,181
54,56,57,78,77,161,162,183,182
55,57,58,79,78,162,163,184,183
56,58,59,80,79,163,164,185,184
57,59,60,81,80,164,165,186,185
58,60,61,82,81,165,166,187,186
59,61,62,83,82,166,167,188,187
60,62,63,84,83,167,168,189,188
61,64,65,86,85,169,170,191,190
62,65,66,87,86,170,171,192,191
63,66,67,88,87,171,172,193,192
64,67,68,89,88,172,173,194,193
65,68,69,90,89,173,174,195,194
66,69,70,91,90,174,175,196,195
67,70,71,92,91,175,176,197,196
68,71,72,93,92,176,177,198,197
69,72,73,94,93,177,178,199,198
70,73,74,95,94,178,179,200,199
71,74,75,96,95,179,180,201,200
72,75,76,97,96,180,181,202,201
73,76,77,98,97,181,182,203,202
74,77,78,99,98,182,183,204,203
75,78,79,100,99,183,184,205,204
76,79,80,101,100,184,185,206,205
77,80,81,102,101,185,186,207,206
78,81,82,103,102,186,187,208,207
79,82,83,104,103,187,188,209,208
80,83,84,105,104,188,189,210,209
*Element, type=M3D4R, elset=ghostel
50001,1,2,23,22
```

```
50002,2,3,24,23
50003,3,4,25,24
50004,4,5,26,25
50005,5,6,27,26
50006,6,7,28,27
50007,7,8,29,28
50008,8,9,30,29
50009,9,10,31,30
50010,10,11,32,31
50011,11,12,33,32
50012,12,13,34,33
50013,13,14,35,34
50014,14,15,36,35
50015,15,16,37,36
50016,16,17,38,37
50017,17,18,39,38
50018,18,19,40,39
50019,19,20,41,40
50020,20,21,42,41
50021,22,23,44,43
50022,23,24,45,44
50023,24,25,46,45
50024,25,26,47,46
50025,26,27,48,47
50026,27,28,49,48
50027,28,29,50,49
50028,29,30,51,50
50029,30,31,52,51
50030,31,32,53,52
50031,32,33,54,53
50032,33,34,55,54
50033,34,35,56,55
50034,35,36,57,56
50035,36,37,58,57
50036,37,38,59,58
50037,38,39,60,59
50038,39,40,61,60
50039,40,41,62,61
50040,41,42,63,62
50041,43,44,65,64
50042,44,45,66,65
50043,45,46,67,66
50044,46,47,68,67
50045,47,48,69,68
50046,48,49,70,69
50047,49,50,71,70
50048,50,51,72,71
50049,51,52,73,72
50050,52,53,74,73
50051,53,54,75,74
50052,54,55,76,75
50053,55,56,77,76
```

```
50054,56,57,78,77
50055,57,58,79,78
50056,58,59,80,79
50057,59,60,81,80
50058,60,61,82,81
50059,61,62,83,82
50060,62,63,84,83
50061,64,65,86,85
50062,65,66,87,86
50063,66,67,88,87
50064,67,68,89,88
50065,68,69,90,89
50066,69,70,91,90
50067,70,71,92,91
50068,71,72,93,92
50069,72,73,94,93
50070,73,74,95,94
50071,74,75,96,95
50072,75,76,97,96
50073,76,77,98,97
50074,77,78,99,98
50075,78,79,100,99
50076,79,80,101,100
50077,80,81,102,101
50078,81,82,103,102
50079,82,83,104,103
50080,83,84,105,104
50081,106,107,128,127
50082,107,108,129,128
50083,108,109,130,129
50084,109,110,131,130
50085,110,111,132,131
50086,111,112,133,132
50087,112,113,134,133
50088,113,114,135,134
50089,114,115,136,135
50090,115,116,137,136
50091,116,117,138,137
50092,117,118,139,138
50093,118,119,140,139
50094,119,120,141,140
50095,120,121,142,141
50096,121,122,143,142
50097,122,123,144,143
50098,123,124,145,144
50099,124,125,146,145
50100,125,126,147,146
50101,127,128,149,148
50102,128,129,150,149
50103,129,130,151,150
50104,130,131,152,151
50105,131,132,153,152
```

```
50106,132,133,154,153
50107,133,134,155,154
50108,134,135,156,155
50109,135,136,157,156
50110,136,137,158,157
50111,137,138,159,158
50112,138,139,160,159
50113,139,140,161,160
50114,140,141,162,161
50115,141,142,163,162
50116,142,143,164,163
50117,143,144,165,164
50118,144,145,166,165
50119,145,146,167,166
50120,146,147,168,167
50121,148,149,170,169
50122,149,150,171,170
50123,150,151,172,171
50124,151,152,173,172
50125,152,153,174,173
50126,153,154,175,174
50127,154,155,176,175
50128,155,156,177,176
50129,156,157,178,177
50130,157,158,179,178
50131,158,159,180,179
50132,159,160,181,180
50133,160,161,182,181
50134,161,162,183,182
50135,162,163,184,183
50136,163,164,185,184
50137,164,165,186,185
50138,165,166,187,186
50139,166,167,188,187
50140,167,168,189,188
50141,169,170,191,190
50142,170,171,192,191
50143,171,172,193,192
50144,172,173,194,193
50145,173,174,195,194
50146,174,175,196,195
50147,175,176,197,196
50148,176,177,198,197
50149,177,178,199,198
50150,178,179,200,199
50151,179,180,201,200
50152,180,181,202,201
50153,181,182,203,202
50154,182,183,204,203
50155,183,184,205,204
50156,184,185,206,205
50157,185,186,207,206
```

```
50158,186,187,208,207
50159,187,188,209,208
50160,188,189,210,209
*Orientation, name=Ori-1
1.,0.,0.,0.,1.,0.
3, 0.
*Membrane Section, elset=ghostel, material=UMAT
1.,
*Hourglass stiffness
1
*UEL PROPERTY, ELSET=userel
1
*End Instance
*Nset, nset=fix,instance=Part-1-1
1,22,43,64,85,106,127,148,
169,190
*Nset, nset=loadup,instance=Part-1-1
126,147,168,189,210
*Nset, nset=loaddn,instance=Part-1-1
21,42,63,84,105
*Nset, nset=allnodes,instance=Part-1-1,generate
1,210,1
*Elset, elset=all,instance=Part-1-1,generate
1,80,1
*End Assembly
*Material, name=UMAT
*Depvar
8,
*User Material, constants=9
.000000001,.000000001,.000000001,0.32,.32,.45,.000000001,.000000001,
.000000001
*Boundary
fix, 1, 6
allnodes,6,6
*Step, name=Step-1, inc=20000
*Static
.01, 1., 1e-05, .01
*Boundary
loadup, 3, 3, 6
loaddn, 3, 3,-6
*Restart, write, frequency=0
*Output, field
*node output
CF,RF,RM,RT,TF,U
*element output,directions=YES
LE,SDV,E
*Output, history
*Node Output, nset=loadup
RF3, U3
*Node Output, nset=loaddn
RF3, U3
*End Step
```

REPORT DOCUMENTATION PAGE

The public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number.
PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.

1. REPORT DATE *(DD-MM-YYYY)*	2. REPORT TYPE	3. DATES COVERED *(From - To)*
01-09-2017	Technical Memorandum	

4. TITLE AND SUBTITLE
AF-Shell 1.0 User Guide

5a. CONTRACT NUMBER

5b. GRANT NUMBER

5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S)
McElroy, Mark W.

5d. PROJECT NUMBER

5e. TASK NUMBER

5f. WORK UNIT NUMBER
432938.11.01.07.43.40.09

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
NASA Langley Research Center
Hampton, Virginia 23681-2199

8. PERFORMING ORGANIZATION REPORT NUMBER
L–20798

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Washington, DC 20546-0001

10. SPONSOR/MONITOR'S ACRONYM(S)
NASA

11. SPONSOR/MONITOR'S REPORT NUMBER(S)
NASA/TM–2017–219604

12. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified-Unlimited
Subject Category 64
Availability: NASA STI Program (757) 864-9658

13. SUPPLEMENTARY NOTES
An electronic version can be found at http://ntrs.nasa.gov.

14. ABSTRACT

This document serves as a user guide for the AF-Shell 1.0 software, an efficient tool for progressive damage simulation in composite laminates. This guide contains minimal technical material and is meant solely as a guide for a new user to apply AF-Shell 1.0 to laminate damage simulation problems.

15. SUBJECT TERMS
composites, delamination, shell element, progressive damage

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT	b. ABSTRACT	c. THIS PAGE			STI Information Desk (help@sti.nasa.gov)
U	U	U	UU	37	19b. TELEPHONE NUMBER *(Include area code)* (757) 864-9658