A USER DEFINED MATERIAL MODEL FOR SIMULATION OF VARIABLE MIXED MODE DELAMINATION IN AN EXPLICIT FEM CODE

By

Zaryan Ijaz

NUST-2013-62003-MSMME-62013F

Supervised By

Dr. Mushtaq Khan

A thesis submitted in partial fulfillment of the requirements for the degree of Masters of Science in Design and Manufacturing Engineering

Design and Manufacturing Group,
School of Mechanical and Manufacturing Engineering,
National University of Sciences and Technology (NUST),
Islamabad, Pakistan

(November, 2016)

National University of Sciences & Technology

MASTERS THESIS WORK

We here by recommend that the dissertation prepared under our supervision by: Zaryan Ijaz (NUST-2013-62003-MSMME-62013F) Titled: "A user defined material model for simulation of variable mixed mode delamination in an explicit FEM code" be accepted in partial fulfillment of the requirements for the award of Masters of Science in Design and Manufacturing Engineering degree with (_____ grade)

Examination Committee Members

1. Name: Dr. Liaqat Ali Signature:__________________

2. Name: Dr. Aamir Mubashar Signature:__________________

3. Name: Dr. Hussain Imran Signature:__________________

Supervisor's Name: Dr. Mushtaq Khan Signature:__________________

Date:__________________

______________________ ______________________

Head of Department Date

COUNTERSIGNED

Date:________________ ______________________

Dean/Principal

Declaration

I certify that this research work titled "A user defined material model for simulation of variable mixed mode delamination in an explicit FEM code" is my own work. The work has not been presented elsewhere for assessment. The material that has been used from other sources it has been properly acknowledged / referred.

Signature of Student
Zaryan Ijaz
NUST-2013-62003-MSMME- 62013F

Copyright Statement

- Copyright in text of this thesis rests with the student author. Copies (by any process) either in full, or of extracts, may be only in accordance with the instructions given by author and lodged in the Library of SMME, NUST. Details may be obtained by the librarian. This page must be part of any such copies made. Further copies (by any process) of copies made in accordance with such instructions may not be made without the permission (in writing) of the author.
- The ownership of any intellectual property rights which may be described in this thesis is vested in SMME, NUST, subject to any prior agreement to the contrary, and may not be made available for use of third parties without the written permission of SMME, NUST which will describe the terms and conditions of any such agreement.
- Further information on the conditions under which disclosure and exploitation may take place is available from the library of SMME, NUST Islamabad.

Acknowledgments

I would like to thank Allah for blessing me with this honor. I would also like to thank my parents for their support and prayers. I would also like to thank Dr. Amir for his thought provoking and eye opening directions during my research work.

Last but not the least, I would like to thank Dr. Mushtaq for his support, guidance and his patience. Without his mentorship and guidance my research work could not have been completed.

Abstract

Delamination, a common failure mechanism for composite materials, more than often grows in a variable mode ratio, especially in impact problem. The majority of numerical models present in literature and cohesive elements available in commercial FEM packages don’t account for the correct dissipation of energy when crack propagation takes place in a variable mixed mode ratio, and permit the restoration of the cohesive state which violates the principles of thermodynamics. Thus a material model, developed for an explicit FEM code, is implemented using ABAQUS subroutine VUMAT. The implemented material model is then validated against numerous published result and applied in a number of experimental and aerospace test coupons. The implemented shows excellent coherence with published results and is ideally suited for dealing with problems involving variable mixed mode delamination.

Table of Contents

List of Figures and Tables

Symbols

Symbols	**Definition**
FEM	Finite Element Method
UEL	User Element
VUMAT	Virtual User Material
COH3D8	3-D Cohesive Element with 8 nodes
C3D8R	3-D Reduced Integration Continuum Element with 8 nodes
VCC	Virtual Crack Closure
CDM	Continuum Damage Mechanics
CZM	Cohesive Zone Modeling
DCB	Double Cantilever Beam
ENF	End Notch Flexure
MMB	Mixed Mode Bending
FML	Fiber Metal Laminate
VUEL	Virtual User Element
Δf	Displacement at final failure
G	Energy Release Rate
ΔO	Displacement at damage onset
K	Penalty Stiffness
τ	Traction
Δ	Displacement Jump
G_c	Critical Energy Release Rate
G_I & G_{II}	Mode I & Mode II Energy Release Rate
M	Mode Mix
η	Mode Mix Curve Fitting Parameter
x_i	Global Coordinate
X_i	Natural Coordinates
U_{top}	Displacement Jump of the upper Boundary

U_{bottom}	Displacement Jump of the lower Boundary
Δnormal	Normal component of the displacement jump
Δshear	Shear Component of the displacement jump
δons	Displacement norm at damage onset
δfinal	Displacement norm at final damage
R	Damage threshold
R_{max}	Maximum Value of threshold during iteration
L_{fpz}	Length of fracture process zone
T_o	Interfacial Strength

Chapter 1

Introduction

1.1 Research Motivation

The use of composite materials in major industries is severely hampered by the difficulty in predicting the mechanical response of composite parts. Despite their superior specific properties composite parts face a problem of over designing as the failure mechanisms that are at play, are difficult to predict. Delamination is one of the major failure mechanism of composite materials that can appear anytime during a life time of composite part/structure from transport, installation to service life. Much research is still needed to grasp over these ever complex failure mechanisms and further utilize the potential of composite materials.

One of the most effective and relatively simple apparatus to capture delamination in a finite element method environment is the method of cohesive zone modeling. Cohesive elements are now readily available in commercial FEM packages like ABAQUS and are a very effective tool in capturing the interfacial behavior of composite laminates and other de-bonding phenomenon. Majority of the numerical material models presented in the literature, and the corresponding finite elements being built on these material model have an inherent flaw of modelling delamination with a cohesive state restoration, when crack propagation takes place in a variable mixed mode ratio. These models work fine when crack grows in a constant mode mix, as the damage variable grows in a monotonic manner, but as soon as the mode mix change they don’t account for the proper dissipation of energy. The reason for this inherent flaw is the dependence of the final displacement at delamination propagation, on the mode mix ratio. Turon[1], identifying this flaw developed a thermodynamically consistent damage model for variable mix mode delamination and Gonzalez[2] modified the code for an explicit FEM code

The following plot shows the above mentioned phenomenon when a single cohesive element of the built-in ABAQUS library is loaded in a variable mode ratio. The traction force clearly shows a restoration (rise in the RF after damage initiation) which is clearly in violation of the principles of thermodynamics, namely the Clausius Duhem inequality. This restoration can bring about erroneous results when delamination of composite test coupons is being studied, especially in an impact environment, ballistic impact being an example, where the external forces are very volatile. The input file for the test is provided in the appendix.

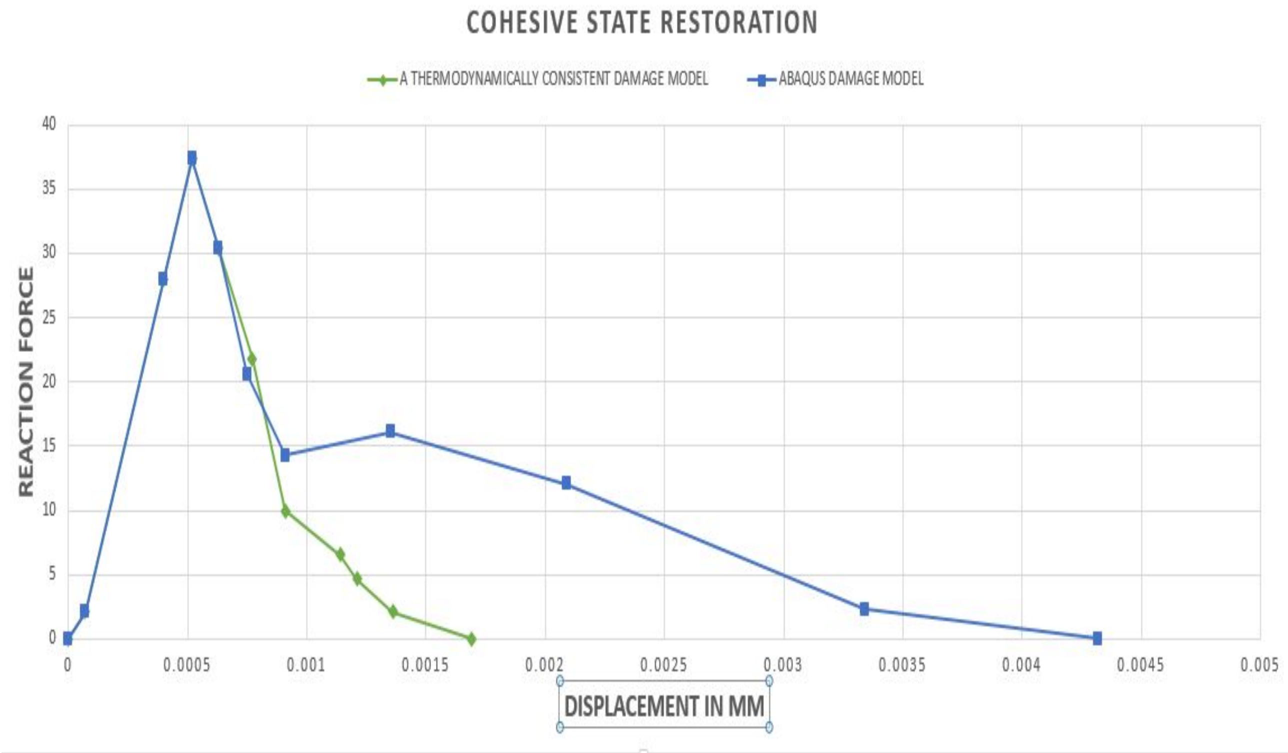

Figure 1.1: Variable mixed mode loading-A Comparison

1.2 Research Objectives

Identifying the inherent flaw of built in cohesive element, following research objectives are established:

- To implement Turon’s [1] ABAQUS subroutine (UEL) and configure the system for implementing ABAQUS subroutines.
- To implement Gonzalez’s [2] algorithm for a thermodynamically consistent simulation of variable mixed mode delamination in an explicit FEM environment using the ABAQUS subroutine VUMAT.
- To develop subroutines for delamination modelling using both the zero thickness elements (COH3D8) and continuum based elements (C3D8R).
- To implement the subroutine in virtual simulations involving experimental and aerospace test coupons, and analyze the viability of the subroutine

The test resulting in Fig.1-1 is repeated again using the developed subroutine and the result is shown below. As is obvious from the comparison of the two test results, the subroutine damage models does a far better result in limiting the restoration of the cohesive state and hence is more thermodynamically realistic.

1.3 Thesis Layout

Chapter 1 provides with the research motivation and the consequent research aims developed for this thesis.

Chapter 2 provides a background of; composite materials and their manufacturing, failure mechanisms, virtual simulation and the phenomenon of delamination. Chapter 2 also provides a literature review of the concepts of cohesive zone modelling and its salient features.

Chapter 3 provides explicit details about the user element developed by Turon[1] and also provides recommendations about the various aspects of cohesive zone modelling.

Chapter 4 discussed about the implementation of Gonzalez[2] material model developed for the explicit finite element code using the ABAQUS subroutine VUMAT.

Chapter 5 provides details about the various virtual simulations carried out to explore the viability and performance of the implemented subroutines. Details about model generation are discussed. The final research conclusion and some domains of corresponding future research are also provided.

Appendix provides with the subroutine and the various ABAQUS input files to develop the models discussed throughout the thesis.

Chapter 2

Literature Review

2.1 Composite Materials- Constituents, Manufacturing and Aerospace Industry

Composite materials are defined as materials consisting of at least two distinct ingredients existing in different phases and hence are easily distinguishable at a macroscopic levels. These two distinct materials are brought together either for improving structural performance or for reducing the manufacturing cost. Composites consist of a reinforcing material, which is responsible for negotiating the main loading applied to the composite, and a binder also known as the matrix that acts as a “glue” that holds the whole structure in place. Wood occurs as a natural composite with cellulose fibers and lignin as a binder or matrix. Generally in the engineering sector industrial composite are a set of well-defined materials consisting of a strong, stiff reinforcement either in the form of long, thin fibers or in the form of particles coupled with a binder. Among this set of artificially produced materials, Carbon Fiber Reinforced Plastics have become the industry leaders in use and in versatility especially when one is talking about the aerospace industry.

Carbon fibers are generally in the form of long, thin fibers available in market in the form of “tows”. The fibers are either unidirectional or in the some form of a textile. The matrix is generally epoxy that is either a thermo set or a thermo plastic that correspondingly require a chemical reaction or a high temperature for the binders to cure.

Instead of a separate treatment of the reinforcement and the binders composites are now generally available in the form of pre-pegs. Pre-pegs consist of “plies” that consist of fixed thickness of fibers pre-impregnated with the matrix. Pre-pegs must be kept in a refrigerated atmosphere such that the matrix does not undergo premature curing. The plies can be oriented at any desired direction and the part/structure is built up till the required thickness using a mold having the desired shape proving composite with one of their fundamental advantage of tailor ability. Pre-pegs also bring about a significant reduction in matrix usage. From a manufacturing point of view the composites may prove superior over the conventional materials as they may provide better aerodynamic and economic solutions as composite materials allow production of large parts avoiding the use of joints.

Fiber Glass was one of the earliest available composite material to be used on boats. Most aircraft structures are mostly in compression. Compression is very hard to figure out. The blades in the helicopter can be approximated as beams with large deflection and rotation. In order to keep its dynamic performance within a certain range you need a certain dynamic response. Blades will have a huge centrifugal load as you try to make them bigger in length or in chord, it becomes impossible to do it with a metal structure as it becomes too heavy and hence gives an undesired frequency response. Composite material could solve the problem and give laminar flow as well, as composites can be manufactured in bulk.

Thus when building huge aerospace structures composite manufacturing methods can greatly reduce cost and improve aerodynamic performance.

2.2 Failure Mechanisms

Composite materials fails in a combination of inter and intra-laminar failures. Intra laminar failures are categorized as fiber, matrix or inter-fiber matrix failures. Composite material being orthotropic materials have two planes of symmetry. A ply consists of fibers and matrix. An ideal approach would be to characterize the fibers and matrix properties and predict their collective properties. The stiffness is predicted quite well but the strength properties fail miserably. The mechanisms of failure are not easily understood. The failures work at a different scale working from the micron level fiber scratches to the matrix cracking at the millimeter level. So there are different failures or different combination of failures at different scales and it is important as to decide what level of modelling one has decided to model these. The molecular level represents the most advanced form of modelling and hence requires the greatest amount of computer and knowledge resource. Normally the failures are predicted at the ply level, although there are many factors at play, and those ply predictions are used in the full scale level.

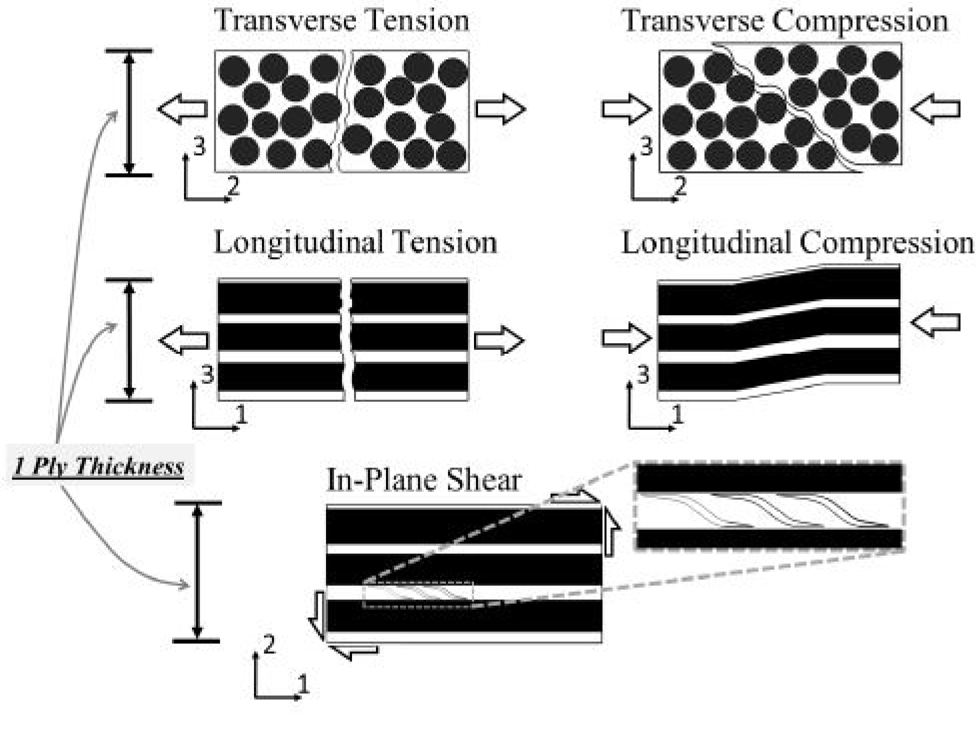

Figure 2.1: Intra-laminar damage mechanisms[3]

2.3 Delamination

Delamination is the primary reason for restricting the ever increasing potential of the composite material industry. Due to the weak inter-laminar strength of the adjacent plies, the interface offers a very low resistance path to the crack. As a result matrix cracks while approaching the free edge convert into delamination cracks because of low crack resistance. Delamination can appear during phase of life of a composite material and are very difficult to inspect. It also results in an increase of maintenance hours and hence is a source of increasing maintenance cost.

The causes of delamination are wide ranging. They can occur do the presence of curved sections. A rapid change in cross-section such as in ply-drop offs can also result in an increase of inter-laminar forces and hence are a source of potential delamination. Adverse service conditions can also bring about delamination, particular case of interest being the problem of impact as the resulting delamination severely hamper the bending stiffness of the laminate.

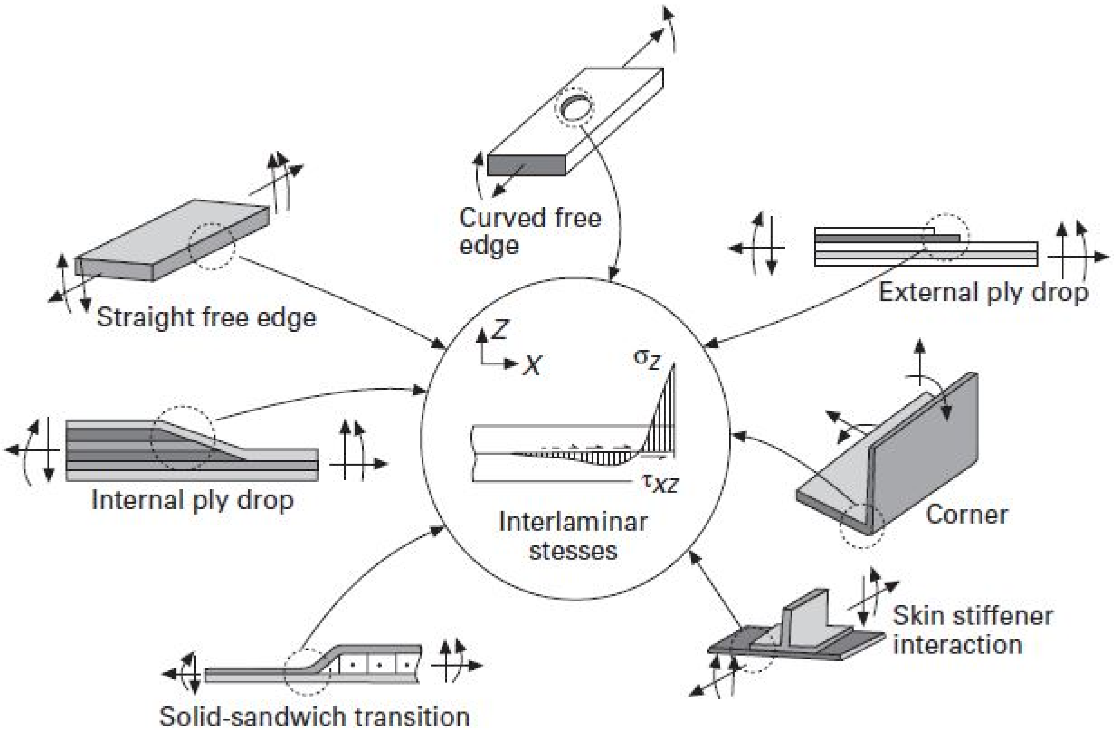

Figure 2.2: Sources of delamination[4]

2.4 Fracture Mechanics and Continuum Damage Mechanics

Linear Elastic fracture mechanics is very effective in dealing with problems involving crack propagation. The said concept is based on the concept of Griffith's critical energy [5]. Crack propagation takes place when the energy release rate is greater than a critical value. Methods incorporating the Linear Elastic Fracture Mechanics include the Virtual Crack Closure (VCC) and the J-Integral techniques that have been implemented in a FEM environment. VCCT implies that the energy released during crack propagation is equal to the work done to close the crack back to the original position. Although these method are effective in capturing the delamination phenomenon, they face severe draw backs; as they require an initial crack path line for the propagation of path, require information from front and back of the crack to perform calculations, and require certain mesh changing algorithm for crack advancement. As per the ideology of Fracture Mechanics delamination can be categorized as: A normal opening mode (Mode-I) and two tangential (shear) modes (Mode-II, Mode-III)(Fig2.3). The crack can grow in a pure mode or in some ratio of the two different types of displacement, the ratio known as the mixed mode ratio.

The concept of continuum damage mechanics is often used to address the phenomenon of delamination. As per the concept of CDM the behavior of material is controlled by damage variables that are a measure of the reduction of the material stiffness in a particular direction of loading. CDM is a simple yet powerful tool to capture the various damage mechanism that are at play prior to complete failure

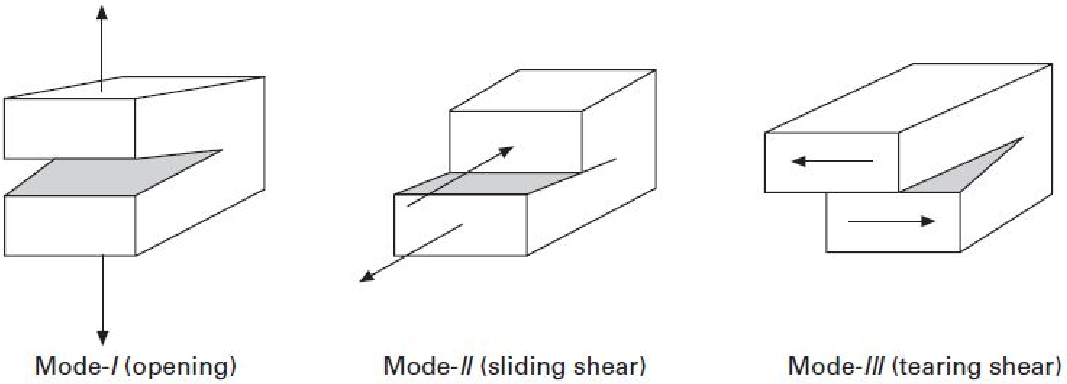

Figure 2.3: Fracture modes

2.5 Cohesive Zone Modelling

Cohesive zone modelling represents the most advanced and effective method of capturing delamination. They have the inherent intricacy of modelling the damage initiation and crack propagation with the same general frame work and are also easily implemented in the FEM environment. They also easily overcome the problems posed by the Linear Elastic Fracture Mechanics as they don’t require re-meshing and easily model dynamically growing cracks. The cohesive elements have become a regular feature of the commercial FEM packages and have become indispensable for modelling the process of interfacial fracture and de-bonding. They combine the ideologies of Fracture Mechanics and Damage Mechanics under a single platform.

The concept of cohesive forces preceding a crack was first presented by Dugdale[6]. He presented for the first time the concept of an area of cohesive interactions surrounding the advancing crack front that consisted of closing forces having a magnitude equal to the yield strength of the material. Since then the theory has been used to model various types of failure like brittle fracture, and interfacial fracture and so on.

Cohesive material models are built upon the Traction Displacement relation rather than the conventional stress strain relation where the displacement refers to the opening jump at the interface corresponding to the three fracture modes.

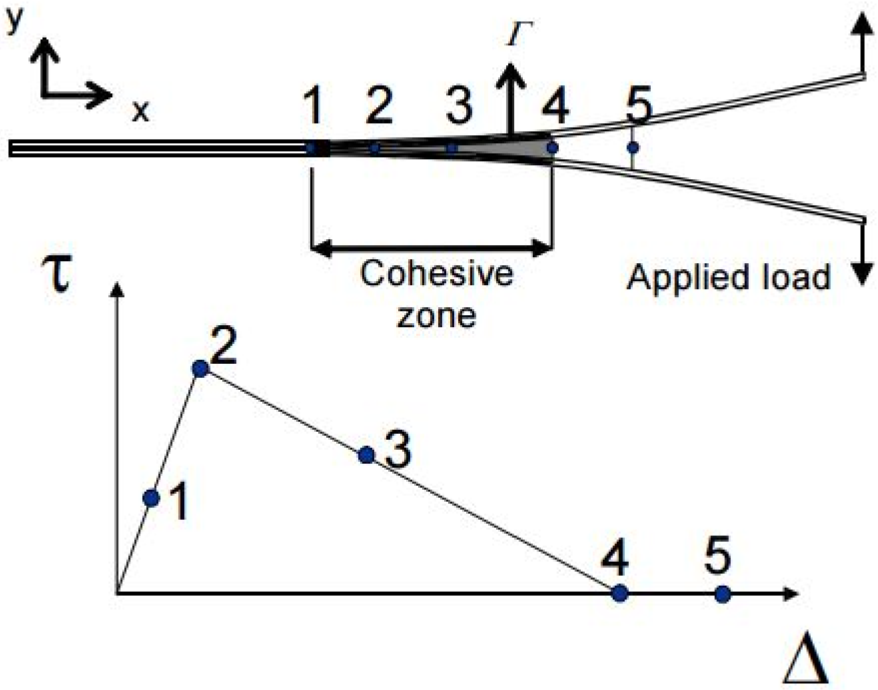

Figure 2.4: Traction Displacement Relation

Fig 2.4 defines some characteristic features of the cohesive zone models:

$$G = \frac{1}{2}\Delta f \tau^{o} \quad \text{(Area under the curve)}$$

$$\Delta o = \frac{\tau}{K}$$

Where Δ is the displacement jump with Δf and Δo being the final and onset displacement at damge initiation. The above equtions define the pure state of loading which could be either a normal or one of the two shearing modes. When the crack grows in a combined way the loading is known as the mixed mode loading and mixed mode is defined as

$$m = G_{II} / G_{I} + G_{II}$$

When mixed mode loading occurs the onset and propagation criterion have to be modified to account for the interaction of pure mode components. Various propagation and initiation have been suggested in the literature, a review provided in [7]. A common criteria for damage propagation is known as the power law, which has shown by Gonzalez[2] to be used as an initiation criteria as well. More on this in the following chapters.

$$G_c = G_{Ic} + (G_{IIc} - G_{Ic})\left(\frac{GII}{GI + GII}\right)\eta$$

Where η is an experimental curve fitting parameter.

2.6 Experimental Procedures

A number of experimental procedures have also been developed over the years to study delamination. In this regard the Double Cantilever Beam, End Notch Flexure and mixed mode bending are of particular importance and are used to establish certain parameters that are required for use in cohesive zone modelling. Figure 2.5 and 2.6 are relevant in this regard.

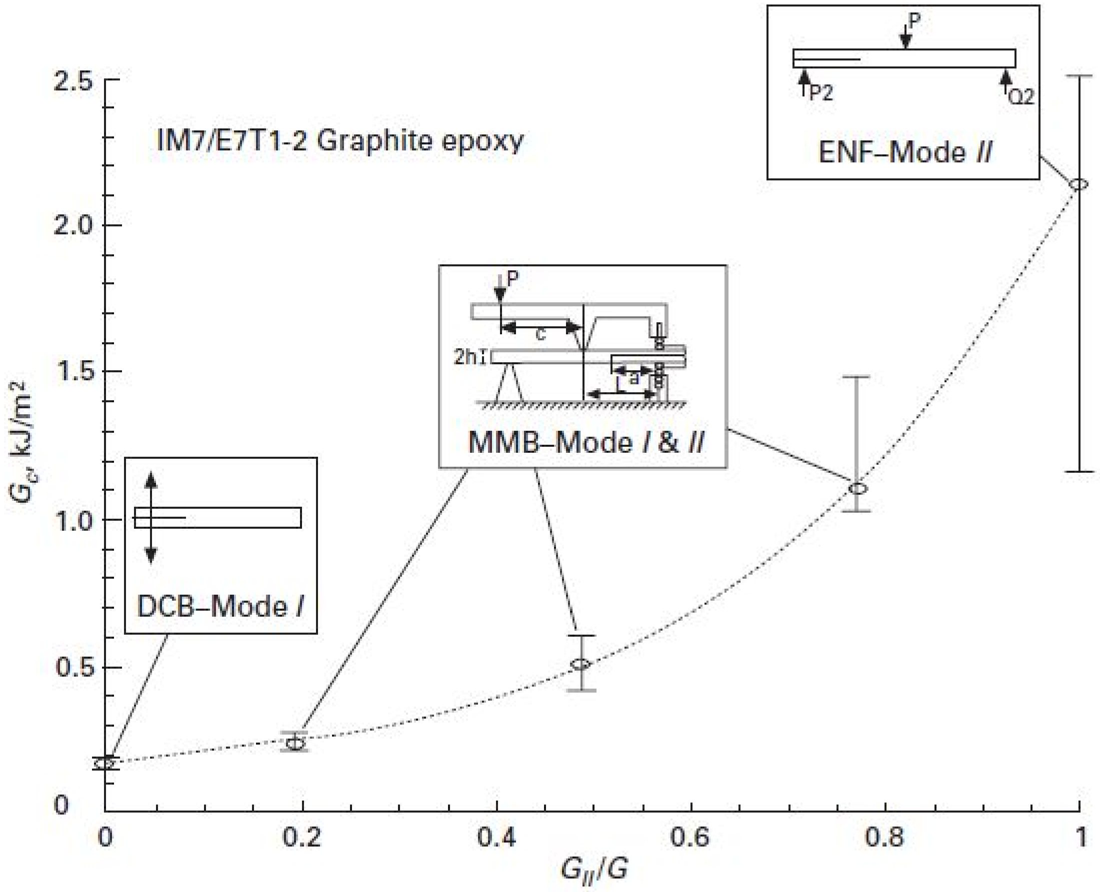

Figure 2.5: Experimental Delamination[4]

Following material properties are required for building cohesive zone models in an FEM environment[8]:

- Mode I fracture toughness, obtained through DCB
- Mode II fracture toughness, obtaibed through ENF
- Mode III fracture toughness, assumed to be equal to that of Mode II value.
- η, obtained through experimental curve fitting.
- Interfacial Strengths approximated as being equal to yield strengths.
- Interface Stiffness – Discussed in Chapter 3

Chapter 3

Turon’s Model

3.1 An Irreversible damage model

Cohesive zone models have been implemented in FEM by a number of authors. [9-11]. Turon[1] pointed that majority of these models have been formulated with the final displacement jump calculated as the propagation criteria. The final displacement jump itself is a function of the mode mix and hence its use as the propagation criteria may lead to the restoration of the cohesive state provided the mode mix changes. Hence he established the need for developing a robust cohesive damage model that can account for mode changes after the damage has been initiated.

3.2 UEL

Turon implemented his damage model using the ABAQUS subroutine UEL. UEL is the subroutine that is used to define finite elements for their use in the ABAQUS solver. UEL generally requires complete information like the Tangent/Secant Stiffness Matrix, right hand Side Matrix, Jacobian, and recommendations for the time increment. Turon’s UEL was available in the open source literature and provided with his PHD thesis [12]. The UEL provided with a great opportunity to configure the computer to be running subroutines which consisted of the following steps:

- ABAQUS installation
- Installation of Intel Studio XE 2013 that contained the compiler for compiling the code written in fortran language.
- Installation of Visual Studio 2012 for the installation of relevant libraries and linking between the compiler and ABAQUS.

The subroutine consisted of following relevant coding required for developing the Cohesive Finite Element

3.2.1 Kinematics

The kinematics of the elements were traced using the mid surface concept proposed by Ortiz[13]. The boundaries of the interface are related to the displacement jump and the natural coordinates of the element as:

$x_i = X_i + 1/2(u_{top} - u_{bottom})$

3.2.2 Numerical Integration

The suitable integration scheme for cohesive elements is well researched with the superiority of the Newton Cotes quadrature over the Gaussian Quadrature, which results in spurious oscillation in the calculation of the load vector [13]. The selection of Newton Cotes as the integration quadrature implies the same coordinates for the nodes and integration points of the element.

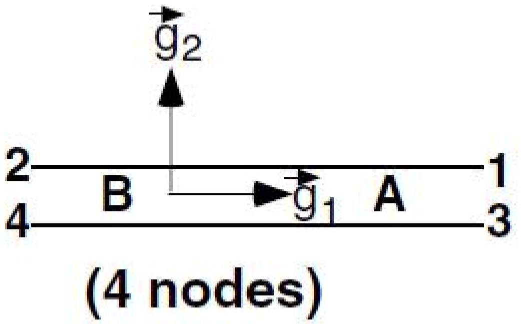

Figure 3.1: A four node interface element

3.2.3 Shape Functions

Standard lagrangian linear shape functions for 4-node elements are used and the shape function are used to formulate the interpolation function for determining the geometry and the displacement function.

3.2.4 Damage Model

In order to track the damage variable for the reduction of the stiffness of the cohesive element the model provided by Simo [14]. The said model is based on Continuum Damage Mechanics and uses a strain based formulation that ensure thermodynamic irreversibility. The model requires the calculation of a displacement norm as:

$\Delta = (\delta_{\text{normal}} + \delta_{\text{shear}})$ exp ½

The displacement norm is used for the calculation of the damage onset and damage criterion developed from a unified power law criterion. Then finally a monotonic scalar function is developed that tracks the development of damage and also determines whether the interface is being loaded or reloaded.

$$R = (\delta_{ons} * \delta_{final}) /(\delta_{final} – R_{max}* (\delta_{final} – \delta_{ons})$$

R_{max} requires the storage of R as a solution dependent variable.

3.2.5 Shape of the Constitutive Law

The shape of the constitutive law best suited for dealing with a numerical treatment of delamination has been the subject of much debate and various forms and shapes have been suggested over the years. Each shape has its pros and cons while being implemented in an FEM environment with respect to the accuracy of the result as compared to the actual physical result as well the computational efficiency it requires. Discontinuities are generally troublesome especially when solving with an implicit FEM code rendering the continuous polynomial degradation law more feasible. However, even with these function discontinuities are generally non-avoidable when reloading or unloading occurs. Overall the bi-linear law provides the best balance between computational efficiency and physical accuracy [24].

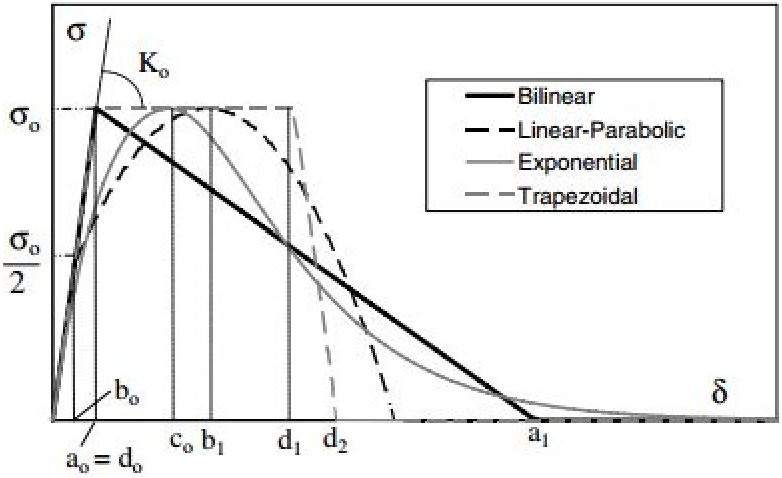

Figure 3.2: Shape of the Cohesive Law [24]

3.3 Engineering Solutions

Turon also provided with some analytical formulas and a methodology to artificially increase the length of fracture process zone to be able to simulate larger structural problems by reducing the number of elements in the direction of crack propagation. [16]

3.3.1 Interface Stiffness

One of the critical decisions in cohesive zone modelling is the selection of value of the interface stiffness. The value should be large enough to not affect the global compliance and small enough to cause numerical problems. Over the years different values have been predicted by different authors typically being of the order of 10^5 N/mm. Turon developed an analytical expression for determining the stiffness value relating it to the interfacial strength and thickness of the laminate. [16]

$K = \alpha E_3 / t$ (where $\alpha = 50$)

3.3.2 Fracture Process Zone

The local phenomenon of delamination can be treated by ensuring a minimum number of elements in fracture process. Normally 3-4 elements are required in the direction of crack propagation to account for the fracture process zone. Turon developed an analytical expression for the calculation of the length of the fracture process zone from material properties.[16]

$L_{fpz} = MEG_c / (T_o)^2$

The above expression can result in a very small fracture process zone and hence a very large number of elements required in the potential location of delamination rendering the possibility of simulation of large scale composite structures as impossible. There is a way around this problem as suggested by turon[16]. The method is to artificially reduce the interfacial strength of the interface whilst keeping the critical energy at crack propagation as same. This helps in increasing the number of element in the fracture process zone while keeping the accuracy of the energy dissipation.[16]

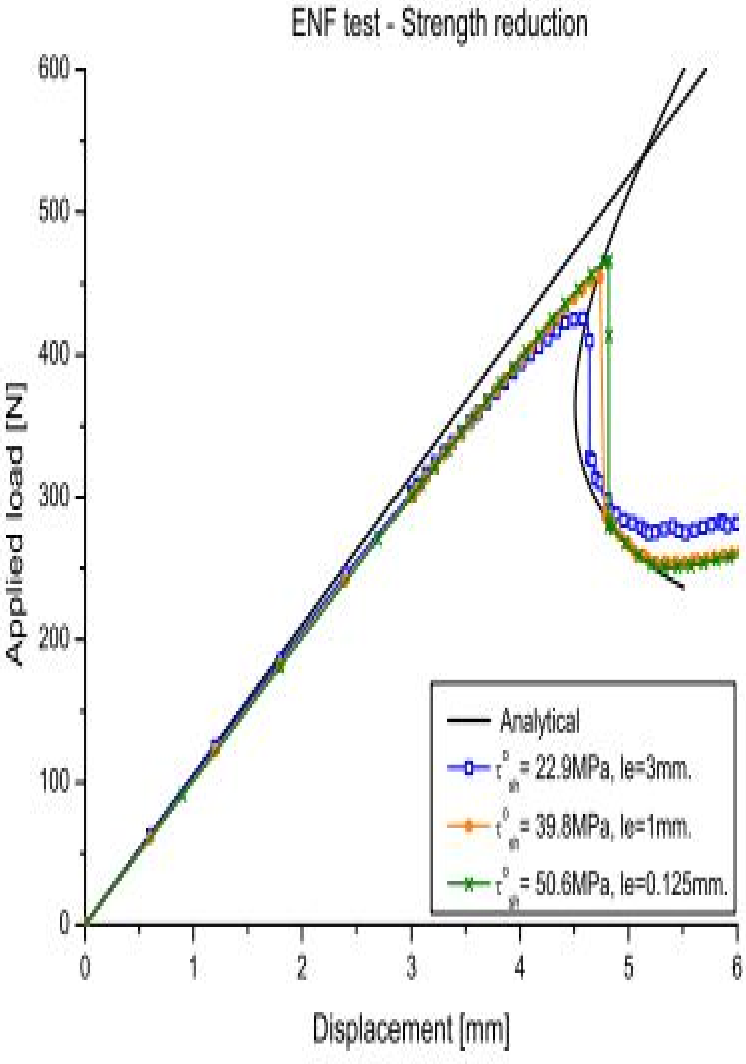

Figure 3.2: Artificial Strength Reduction[16]

Chapter 4

Gonzalez Models

4.1 Explicit Finite Element Method

Explicit finite element methods are going to play an ever increasing role in the simulation, especially related to aerospace problems. Instead of forming a global stiffness matrix explicit FEM depends on kinematic advancement of the solution from one node to another.

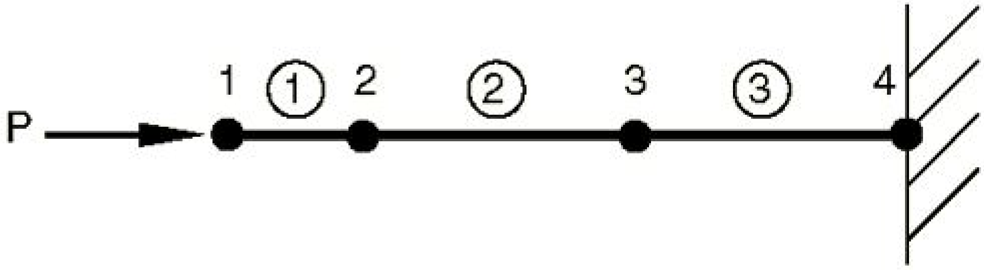

Figure 4.1: Kinematic advancement of solution [17]

The implicit method suffer from convergence issues when dealing with impact problems, material softening as a result of damage mechanisms, and thus require special and complex control algorithms like the arc length method to deal with problems like bi-furcation and snap back.

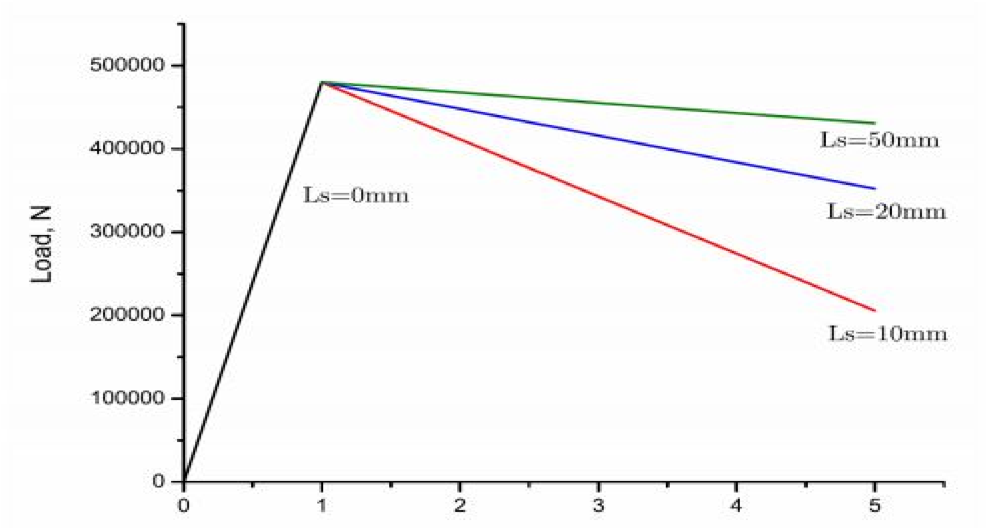

Figure 4.2: Bifurcation [18]

4.2 Gonzalez Algorithm

Gonzalez [2] adopted Turon’s model for implementation in an explicit FEM code. The algorithm is as follow

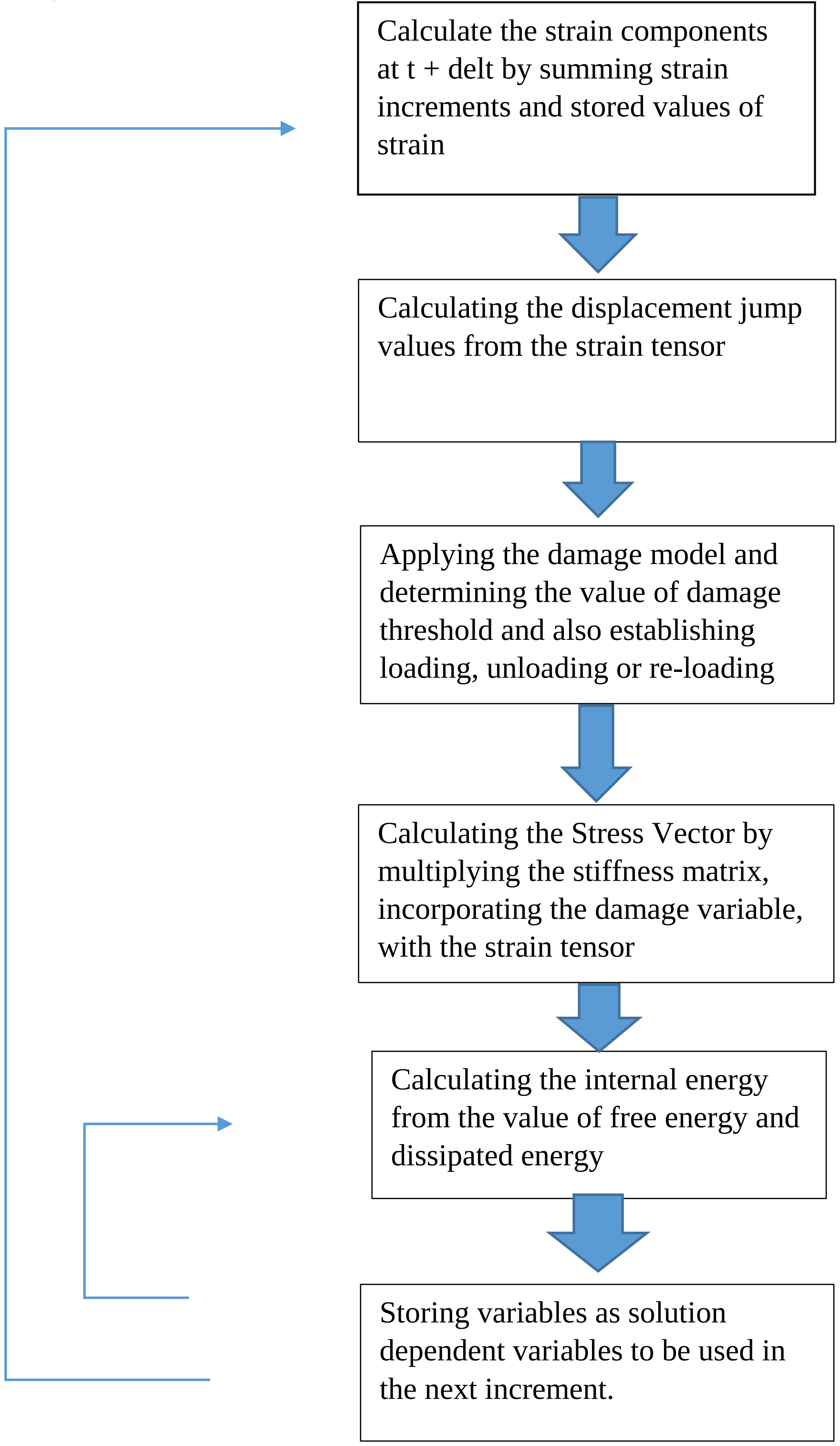

Figure 4.3: Gonzalez’s Algorithm [2]

4.3 VUMAT

Gonzalez’s model is implemented in ABAQUS using the subroutine VUMAT. The subroutines are provided in the appendix. In ABAQUS/Explicit the subroutine VUMAT is used to develop material models when one feels that none of the available material and damage models in the default ABAQUS library is ideal for the problem in hand. Thus VUMAT allows the user to develop and implement proprietary constitutive models. The constitutive equation has to be fed either in an explicit form or in a rate form. The constitutive equation is then integrated using forward or backward euler method. All these procedures may also include/involve internal, user defined variables that evolve with the solution. The integration procedures have a stability limit and thus require a very small stable time increment. These small increments can result in a great number of increments but the increments are relatively in expensive as compared to the iterations involved in the implicit procedures. Further procedure like mass scaling further increase the time increment of the problem by artificially increasing the mass of the element, but there is a limit to this procedure. The kinetic energy of the problem has to be kept well below the internal energy of the system if the option of mass scaling is applied.

Quantities like the strain tensor increment, deformation gradients, internal and dissipated energies are available using the subroutine VUMAT and can be used for coding the constitutive response of the element. VUMAT is called for a number of material points stored in the array (NBLOCK). The user has to update the stress tensor and the internal energy array

4.4 Implementation

The VUMAT is implemented for two different type of structural elements

4.4.1 COH3D8

COH3D8 is the cohesive element that uses the in-built material model for delamination/debonding problems. It is very important to consider the correct stack direction for the cohesive elements. The 2-D and 3-D cohesive elements have to be stacked in the thickness direction of the adhesive. The cohesive elements can be considered as elements having a top and a bottom surface where the current configuration of the element is calculate by averaging the position of the top and the bottom surface [17]. COH3D8 can either have a continuum based response or response based on the traction separation response. In the later the stress tensor consists of the normal six components where as in the former case the stress tensor that has to be updated consists of three terms; the two shearing terms and one normal term. The strains are returned directly as displacement jumps if a thickness of one is used with the latter response. The latter can also be zero thickness elements.

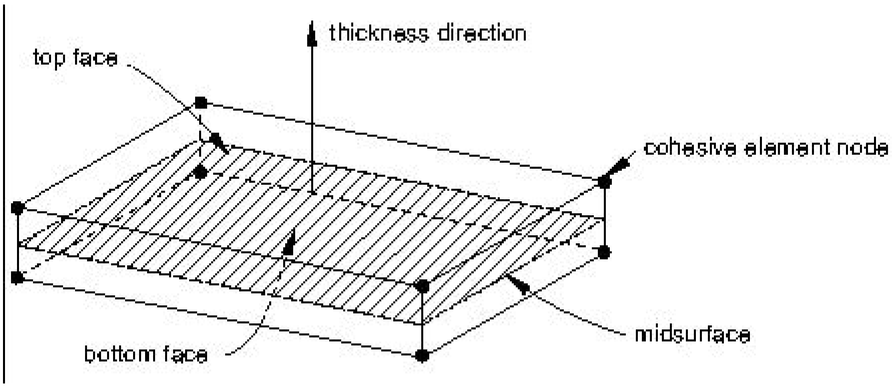

Figure 4.4: COH3D8 ELEMENT [14]

4.4.2 C3D8R

C3D8R is the standard three dimensional, reduced integration element for modelling structural response. When the subroutine is built using these elements, it has certain advantages. First the interface properties can be introduced in the numerical model as interface stiffness and thickness. Secondly, models are far easily built using ABAQUS/CAE as the zero thickness elements have to be implemented through the ABAQUS input file method.

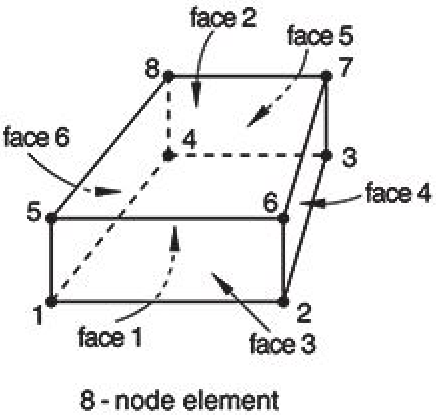

Figure 4.5: C3D8R ELEMENT [14]

4.5 One Element Simulation

Before the subroutines are tested on bigger models, it is wise to test the constitutive models on smaller models. In this regard the single element test provided in the ABAQUS Benchmark Manual [14] were applied and the subroutine was tested and debugged before proceeding to the bigger models. The input files for the one element test are provided in the appendix.

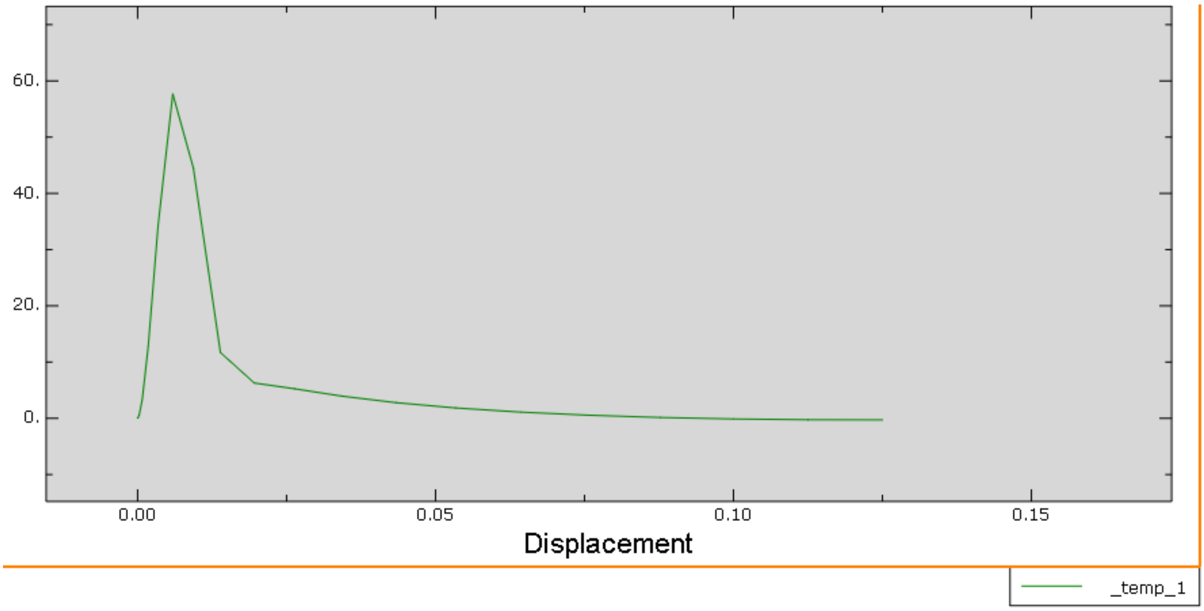

Figure 4.6: One Element Simulation

Chapter 5

5.1 Virtual Simulations

Designing new parts for the aerospace industry is a very complex process as every new structure/sub component that has to go in service conditions has to go through some extensive certifying tests. This require extensive testing of the new components as the mechanical properties of the constituent properties is still not known well as is the case for other conventional material. This may result in some extensive mechanical testing of the developed articles which can only really be affordable for the aerospace manufacturing giants. The coupons, sub-components, components and full scale structures are experimentally tested in the corresponding number of test, and the respective order also represents the increasing complexity of the testing methodologies involved.

Virtual simulation represents the use of state of art computing technology to replace some of these really expensive mechanical tests and provide an alternative for aiding in the certification process for these structures and new materials. The lack of accuracy and confidence in these methods is still not up to the mark for composite materials and the computation power and efficiency required to deal with such problems are still a big hurdle in the use of these computational tool in the design phase of composite materials. Even then, though still not being able to completely replace the expensive mechanical testing, the apparatus of virtual mechanical testing has enabled the designers to design these experiments more effectively using the computational methods and have been proved as an important design tool.

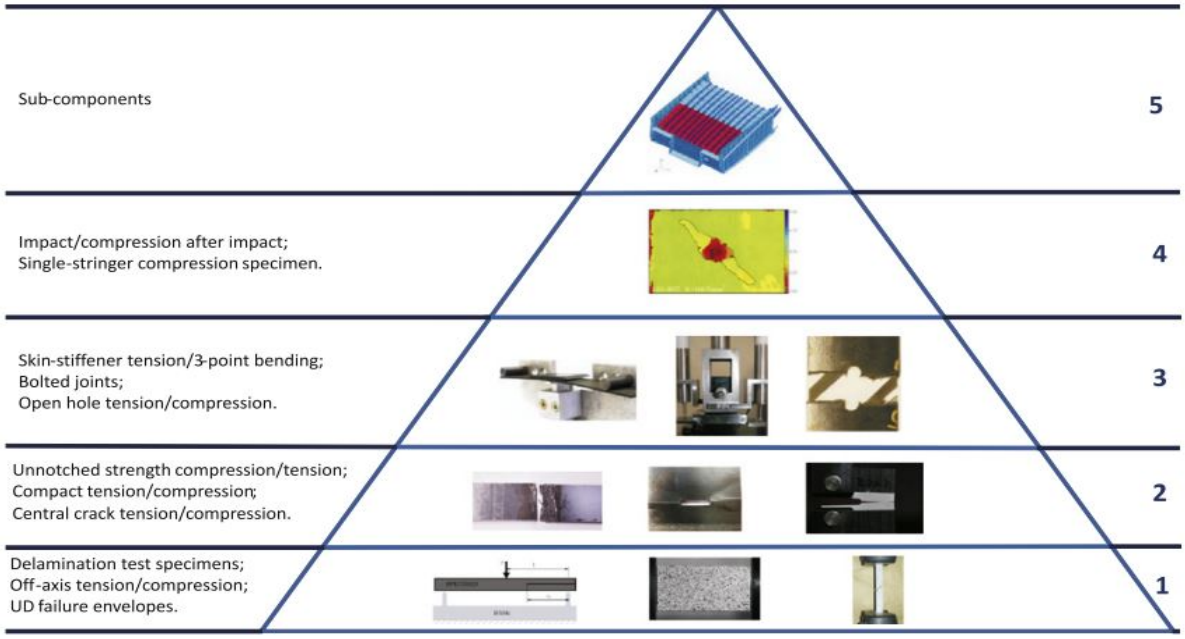

Figure 5.1: The Building Block Approach (19)

The building block approach [19], which applies equally to the experimental and numerical procedures, represents the increasing complexity of tests required to get through the process of certification.

Delamination is involved in two important concepts in the design of aerospace structures; damage resistance and damage tolerance. Damage resistance represents the ability of the composite structure to resist the initiation of damage under service conditions whereas damage tolerance is a measure of the resistance of the member when damage is initiated. Delamination is a major concept involved in both these design procedures, particularly in the damage tolerance design, where the modern aircraft structures require material with a greater fracture toughness or otherwise will result in economic losses corresponding to the frequent maintenance checks resulting in increased labor and reduced capital.

5.2 Numerical Validations

A number of numerical simulations were carried out in ABAQUS to evaluate the performance of the implemented material model.

5.2.1 Mode-I

The first numerical simulation performed was that of a Mode-I delamination as performed experimentally by [20]. The Mode-I test represents the normal opening mode behavior of the laminates. The specimens had a dimension of: 102mm long, 25.4mm wide and each laminate is 1.56mm thick. The laminates were meshed using the C3D8R elements, the conventional three dimensional element having 8 nodes with reduced integration. Along length the, direction of crack propagation, an element dimension of 0.3mm was chosen as was suggested by [2]. The thickness direction had a number of four elements to capture the laminate rotations. Elements in the width direction were not of importance and hence only two elements were used. The interface was meshed using the zero thickness cohesive elements (COH3D8). The placement of these elements in the mesh is not straight forward and the ABAQUS input method is used to build the mesh generation. The reader is referred to the Input Files presented in the Appendix to wrap their mind around the process which also contain the material properties used in the simulation. One of the bottom edge was fixed while the other bottom edge had a roller support. Since ABAQUS Explicit is used to solve the problem the problem has a stability criteria known as the Courant condition. The magnitude of the velocity should be as per the following [2].

$V_l < 0.01(E_m/p_m)^{0.5}$

A loading velocity of 0.5mm is used which is smaller than the above mentioned criteria. The loading velocity is easily applied in a smooth manner using the AMPLITUDE keyword in

ABAQUS. Since ABAQUS Explicit is used to simulate a process that is basically Quasi-Static in Nature, the process has to be accelerated using an artificial “Mass Scaling”. A mass scaling value of 1000 was chosen to bring the time increment to a reasonable value. The simulation took 10 hours on a Core I3, 4-GB standalone PC.

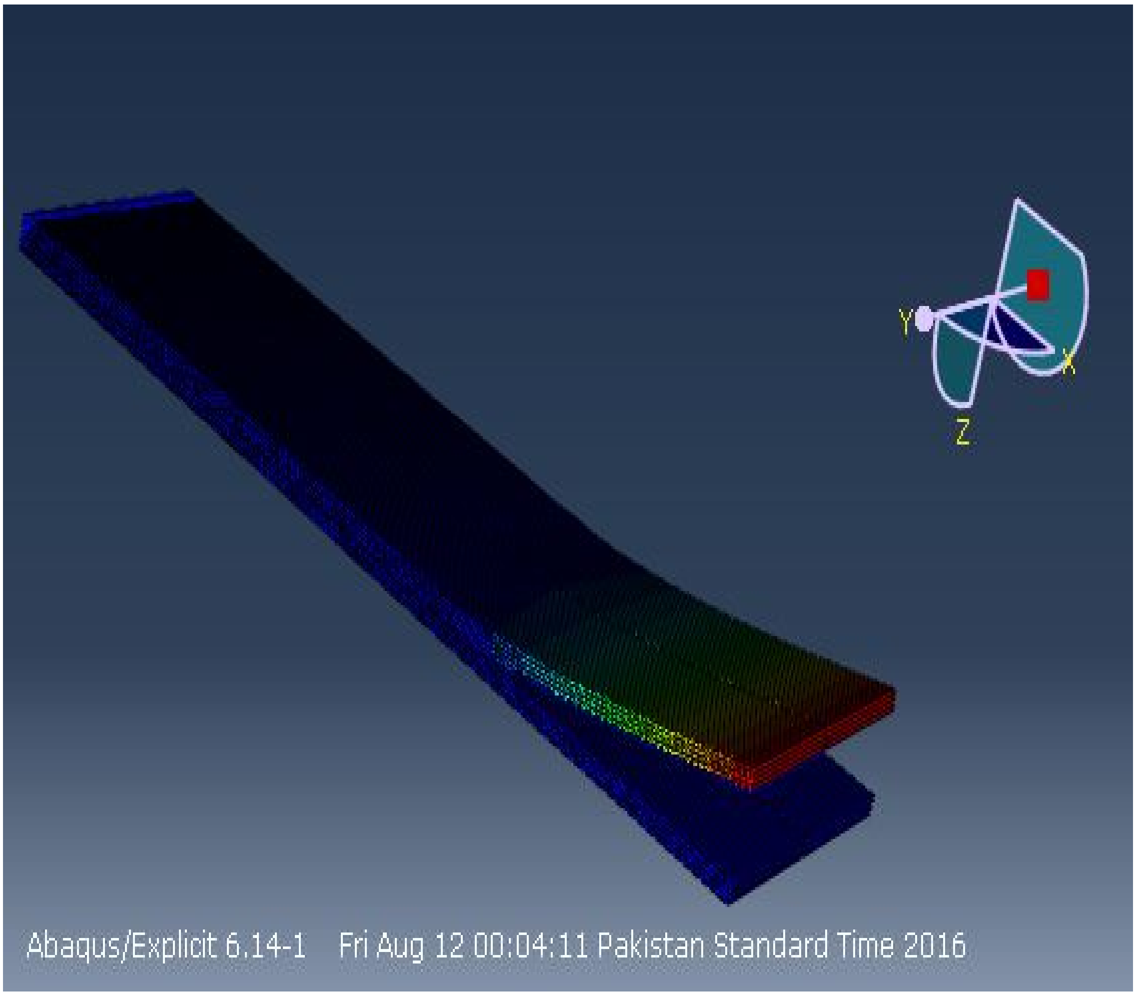

Figure 5.2: Mode-I Simulation

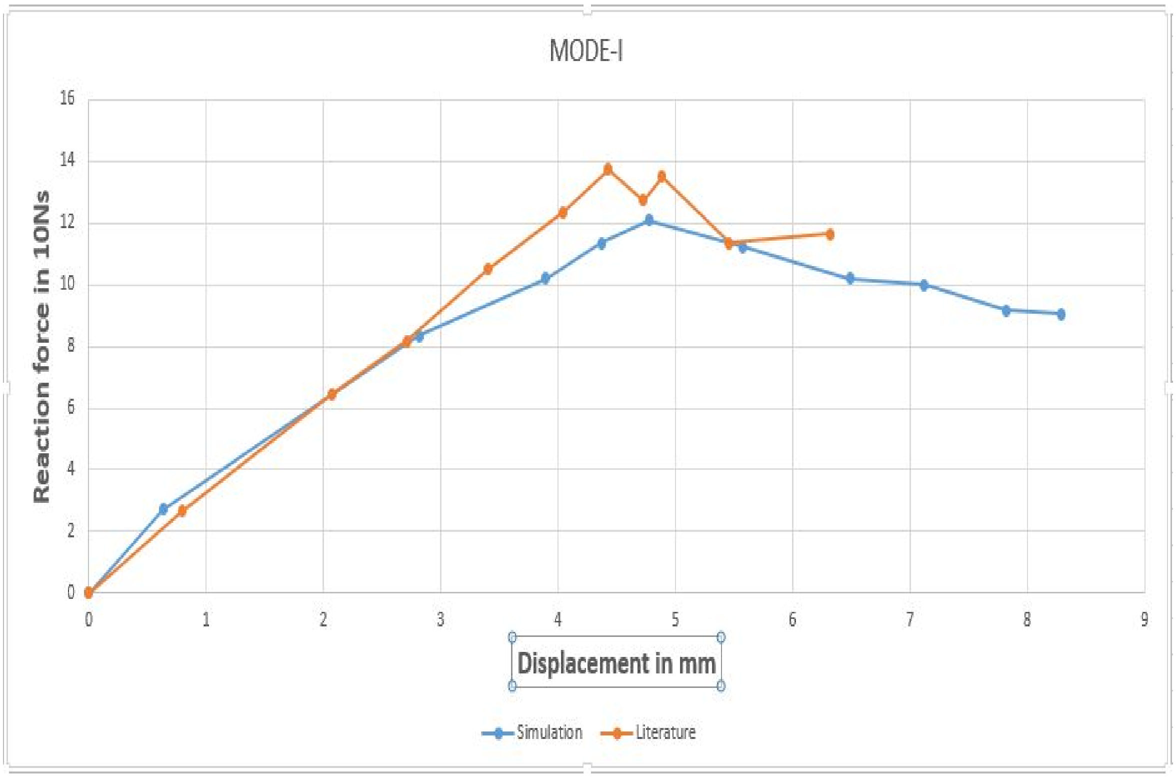

Figure 5.3: Mode-I Results

5.2.2 ENF – MODEII

The next simulation carried out was a pure Mode II delamination procedure [20]. The meshing and boundary were carried out from the Mode-I test. Only the loading conditions were changed. The top middle node were given a velocity of -0.5mm/s. The loading and the boundary condition cause one laminate to bend more than the other resulting in a pure Mode –II loading.

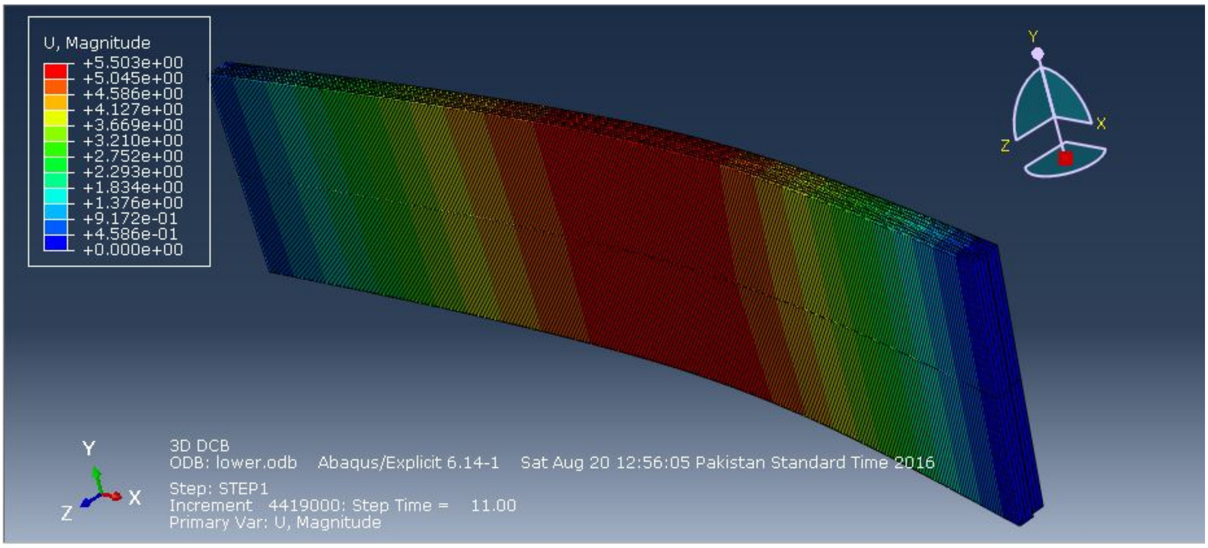

Figure 5.4: Mode-II Simulation

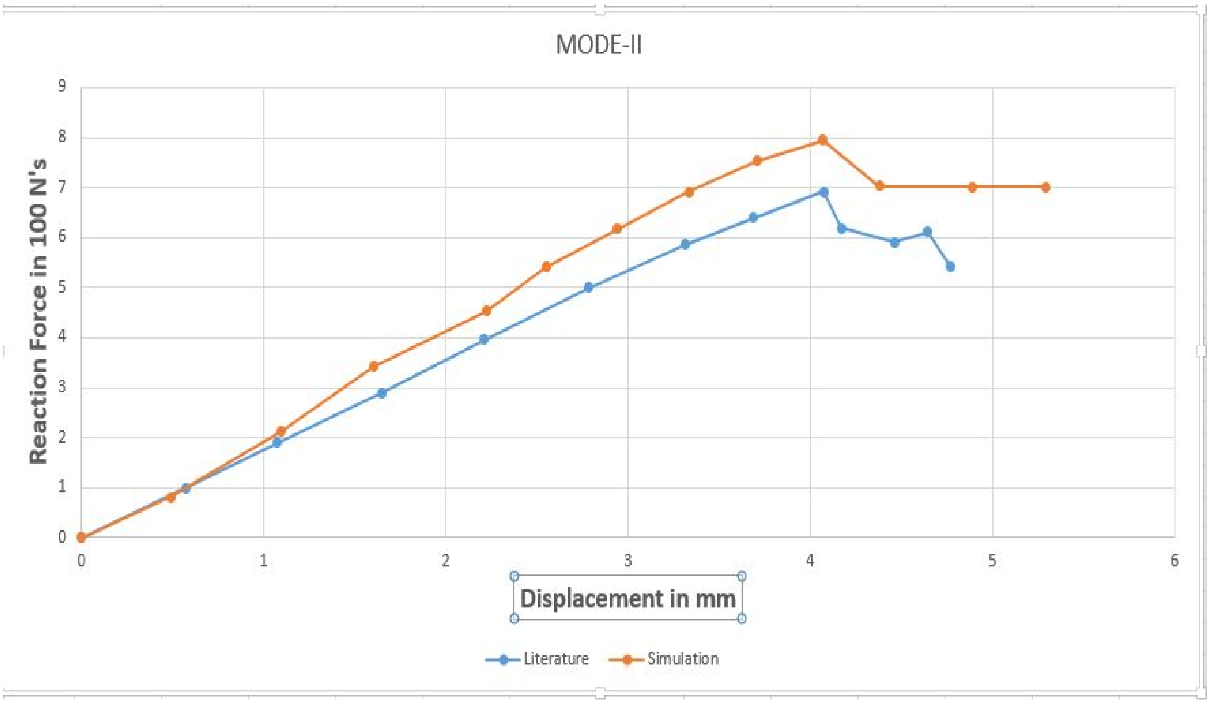

Figure 5.5: Mode-II Results

5.2.3 Blunt Notched Fiber Metal Laminates

The next example is taken from [21]. ABAQUS explicit is used to simulate inter-laminar and intra laminar damage in fiber metal laminates. A FML is subjected is uni-axial loading conditions and both damage mechanisms are allowed to operate and interact. Cohesive elements with VUMAT as the material model is used to simulate inter laminar damage whereas the built in ABAQUS damage model is used for the intra laminar damage model.

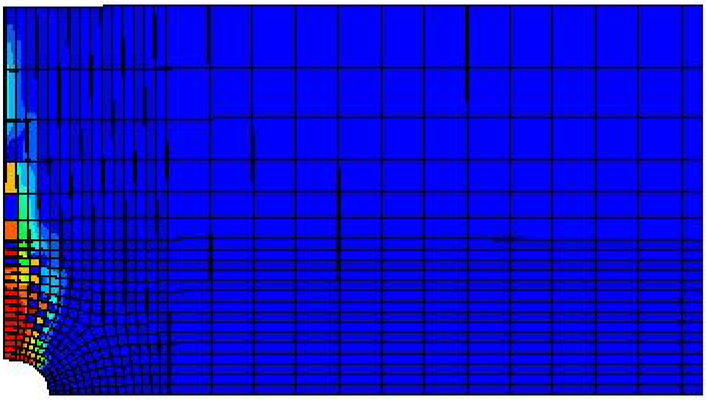

Figure 5.6: FML Inter & Intra Laminar Damage

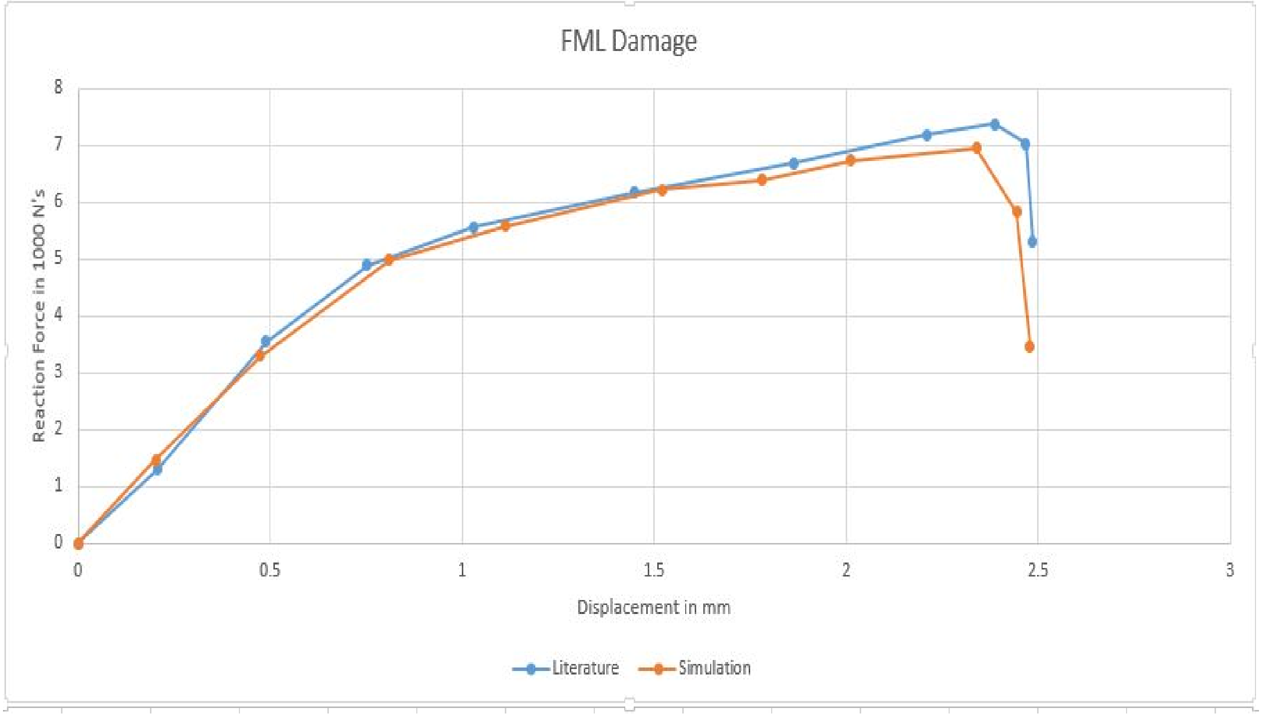

Figure 5.7: FML Inter & Intra Laminar Damage Results

5.2.4 Skin Stiffener Debonding

The next simulation was taken from [22]. The debonding between a skin and a stiffener is simulated using interface cohesive elements. The simulation consists of two steps, in first the specimen is exposed to a curing temperature of 159 deg. C. The difference in the orthotropic thermal expansion coefficients because of different layups of the skin and the stiffener result in residual stresses being developed in the laminate. In the second step, with one fixed the skin is pulled at the other end. The tapered region of the interfaces is meshed with a different density so that the delamination is visible.

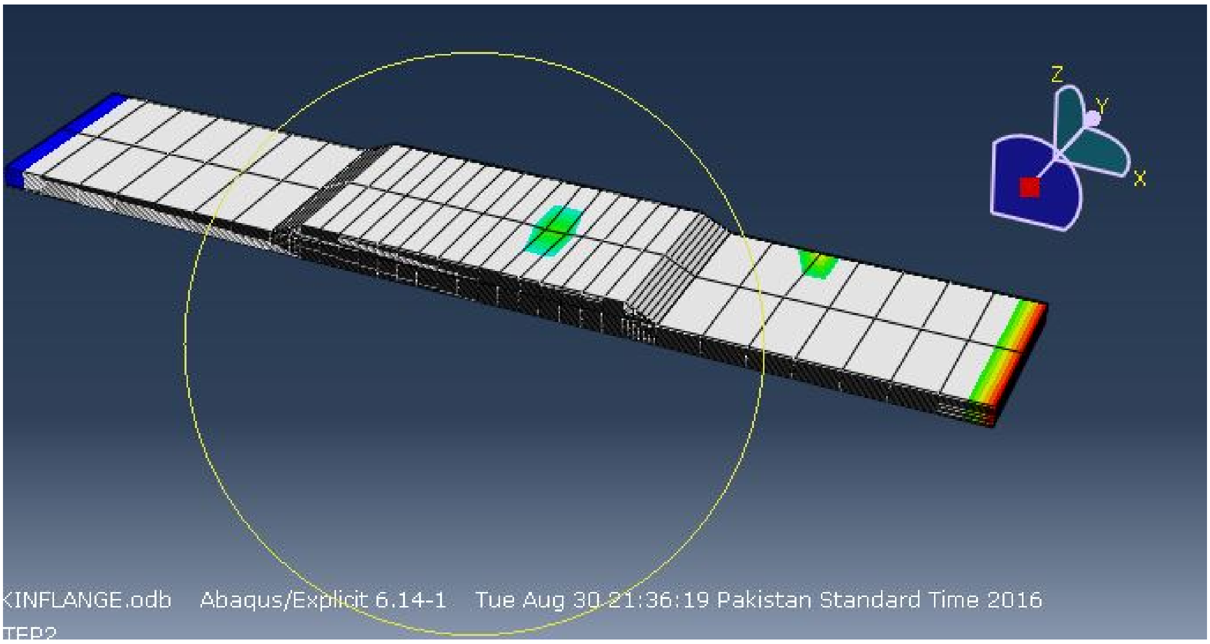

Figure 5.8: Skin Stiffener Debonding

\

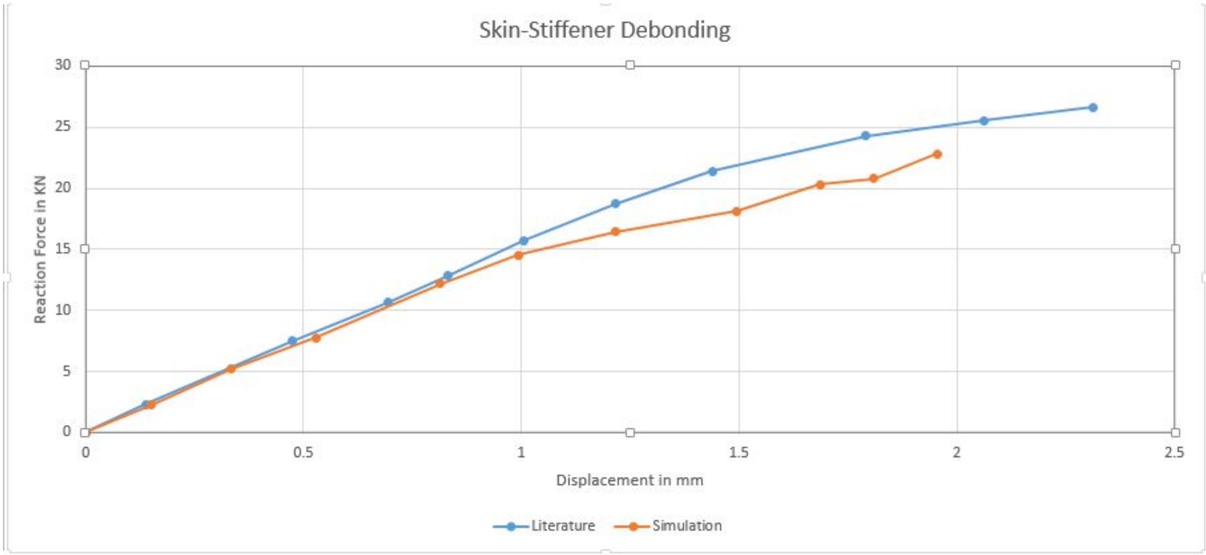

Figure 5.9: Skin Stiffener Debonding Results

5.2.5 Tee Joint

The final simulation was built from [23]. The T-Joint represents a common structural application for adhesive bonding and is a serious concern from delamination point of view because of the curved sections. The T-Joint is a representative of a number of aerospace structural joints. The noodle region also known as the fillet and the deltoid region is a cause of serious vulnerability with respect to delamination. The model was built and meshed as per the provided configuration. Delamination interfaces in the form of zero thickness were introduced in various spot such as the interface between the two laminates, the interface between the laminate and the deltoid. The result showed good correlation with the published result not only concerning the numerical values of concern but also matching the potential locations of delamination.

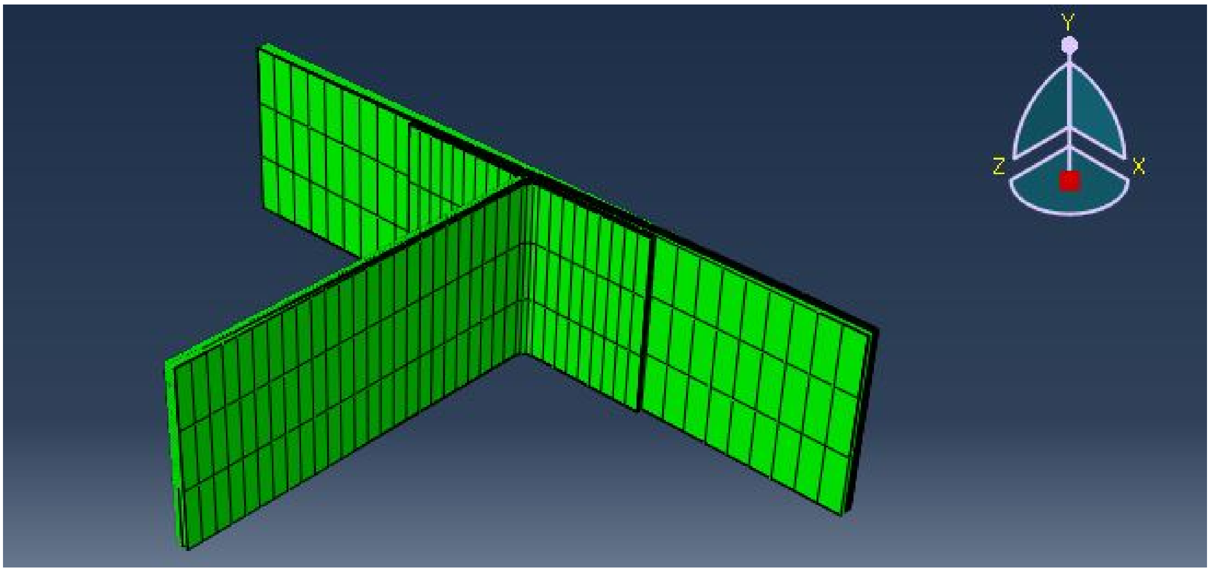

Figure 5.10: T-Joint

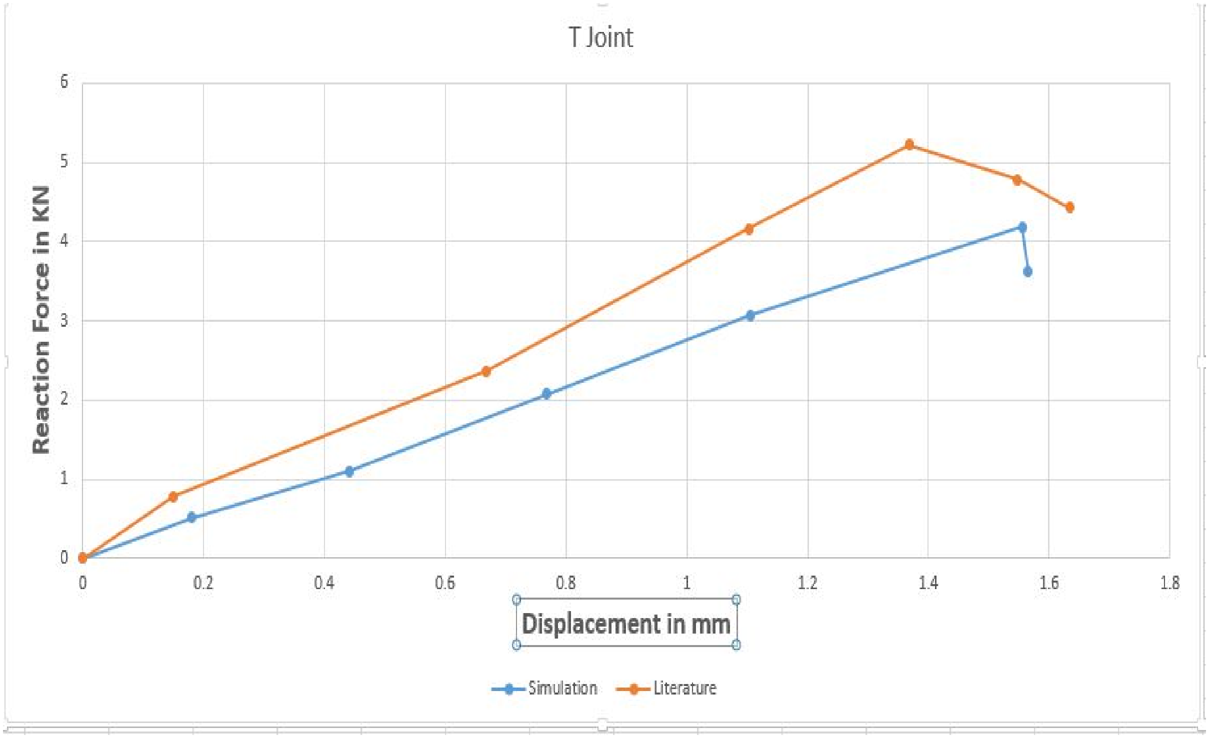

Figure 5.11: T-Joint Results

5.3 Conclusions

- A thermodynamically consistent damage model for the simulation of delamination in composite, for an explicit FEM code was implemented, to overcome some inherent flaws in the in-built ABAQUS cohesive elements.
- The material model developed through a user subroutine was implemented in a number of aerospace structural applications and experimental coupons and overall the model predicted excellent coherence with the published results.
- The continuum based model provided with far greater advantages as compared with the traction separation response model w.r.t the ease of modeling and introducing the interface properties into the numerical model

5.4 Future Research

- Testing of the implemented sub-routine in dynamic situation like impact where the model will clearly show its superiority over the in-built models
- Development of an intra-laminar damage on the basis of the most effective failure criterion till date leading to
- A third level simulation (as proposed by the building block approach) like: bolted joint, compression after impact and so on.
- Development of a VUEL based on this material model proving better kinematics and minimum time increments.

REFERENCES

[1]. A. Turon, P.P. Camanho, J. Costa, C.G. Dávila, A damage model for the simulation of delamination in advanced composites under variable-mode loading, *Mechanics of Materials*, Volume 38, Pages 1072-1089, November 2006.

[2]. E.V. Gonzalez, P. Maimi, A. Turon, P Camanho, J. Renart, Simulation of delamination by means of cohesive elements using an explicit finite element code, *Computers, Materials, & Continua*, Volume 9, Pages 51-92, 2009.

[3]. B. Egan, Finite Element Modelling and high speed testing of countersunk composite aircraft joints, PHD Thesis, University of Limerick, Ireland, 2014.

[4]. S. Sridharan, Delamination behavior of composites, Chapter-1, Pages 3-25, 2008

[5]. A.A. Griffth, The phenomena of rupture and flow in solids, *Philosophical transactions of the Royal Society,* Series A, Pages 163-198, 1921.

[6]. D.S. Dugdale, Yielding of steel sheets containing slits. *Journal of the Mechanics and Physics of Solids*, Volume-8, Pages 100-104, 1960.

[7]. A.C. Orifici, I. Herszberg, R.S. Thomson, Review of methodologies for composite material modelling incorporating failure, *Composite Structures*, Volume 86, Issues 1–3, Pages 194-210, November 2008.

[8]. P. Camanho A. Turon, C Lopez, D. Trias, Delamination Assessment Tool, Best Practice Analysis Methods, A European Space Agency Contract Report, June 2010.

[9]. G. Alfano, M.A.Crisfield, Finite element interface models for the delamination analysis of laminated composites: mechanical and computational issues, *International Journal for Numerical Methods in Engineering,* Volume-50, Pages 1701-1736, 2001.

[10]. O. Allix, P. Ladevéze, A. Corigliano, Damage analysis of interlaminar fracture specimens, *Composite Structures,* Volume 31, Issue 1, Pages 61-74, 1995.

[11]. Y. Mi, M. A. Crisfield, G. A. O. Davies and H. B. Hellweg, Progressive Delamination Using Interface Elements, *Journal of Composite Materials,* Volume-32, Issue 14, Pages 1246-1272, 1998.

[12]. A. Turon, Simulation of delamination in composites under quasi-static and fatigue loading using cohesive zone models, PHD Thesis, University of Girona, 2006.

[13]. M. Ortiz, A. Pandolfi, Finite-deformation irreversible cohesive elements for three-dimensional crack-propagation analysis, *International Journal for Numerical Methods in Engineering,* Volume 44, Issue 9, Pages 1267-1282, 1999.

[14]. J. Schellekens, R Borst, On the numerical integration of interface elements, *International Journal for Numerical Methods in Engineering,* Volume 36, Pages 43-66, 1993.

[15]. J. Simo, J. Ju, Strain and stress based damage model-I Formulation, *International Journal of Solid Structures,* Volume 23, Issue 7, Pages 821-840, 1987.

[16] A. Turon, C.G. Dávila, P.P. Camanho, J. Costa, An engineering solution for mesh size effects in the simulation of delamination using cohesive zone models, *Engineering Fracture Mechanics*, Volume 74, Issue 10, Pages 1665-1682, July 2007.

[17]. ABAQUS (2011) `ABAQUS Documentation', Dassault Systèmes, Providence, RI, USA.

[18]. Advances in the simulation of damage and fracture of composite structures, X Reunión de Usuarios de ABAQUS.

[19]. P.P. Camanho, M.A. Bessa, G. Catalanotti, M. Vogler, R. Rolfes, Modeling the inelastic deformation and fracture of polymer composites – Part II: Smeared crack model, *Mechanics of Materials*, Volume 59, Pages 36-49, April 2013.

[20]. Crews, J.H.; Reeders, J.R, Mixed-mode bending method apparatus for delamination testing. In: NASA TM 100662.

[21]. Hagenbeek, M., C. Van Hengel, O. J. Bosker, and C. A. J. R. Vermeeren, "Static Properties of Fibre Metal Laminates," *Applied Composite Materials*, Vol. 10, Pages 207–222, 2003.

[22]. Davila, C. G., and P. P. Camanho, "Analysis of the Effects of Residual Strains and Defects on Skin/Stiffener Debonding using Decohesion Elements," SDM Conference, Norfolk, VA, April 7–10, 2003.

[23] Li Jun, Xiao Yu Liu, Yuan Ya Nan & Yao Xuefeng, Numerical and experimental analysis of delamination in the T-stiffener integrated composite structure, *Mechanics of Advanced Materials and Structures*, Volume 23, Issue-10, Pages 1188-1196, 2016.

[24] Giulio Alfano, On the influence of the shape of the interface law on the application of cohesive-zone models, *Composites Science and Technology*, Volume 66, Issue 6, Pages 723-730, May 2006.

APPENDIX

• INPUT FILE FOR THE RESEARCH JUSTIFICATION PROBLEM

```
*Node
1,          0.,      0.,        0.,
2,          1.,      0.,        0.,
102,        1.,      1.,        0.,
101,        0.,      1.,        0.,
1001,       0.,      0.,        0.0,
1002,       1.,      0.,        0.0,
1102,       1.,      1.,        0.0,
1101,       0.,      1.,        0.0,
*element, type=coh3d8
1, 2,102,101,1,1002,1102,1101,1001
*nset, nset=top
1001,1002,1102,1101
*nset, nset=bottom
1,  2,  102, 101
*elset, elset=all
1
*ORIENTATION, NAME=ORI-1
1., 0., 0., 0., 1., 0.,
*COHESIVE SECTION, ELSET=all, MATERIAL=USEREPOXY,
RESPONSE=TRACTION SEPARATION, THICKNESS=SPECIFIED
1.,
*MATERIAL,NAME=adhesive
*Density
1.6e-09,
```

```
*ELASTIC,type=traction
**<Emod>,<Emod>,<Emod>
1.0E6,1.0E6,1.0E6
*DAMAGE INITIATION,CRITERION=QUADS
**<ultI>, <ultII>,<ultII>
150,150,150
*DAMAGE EVOLUTION,TYPE=ENERGY,MIXED MODE BEHAVIOR=BK,POWER=1.45
**<GIc>,<GIIc>,<GIIc>
0.075,  0.547,0.547
**DEPVAR, DELETE = 5
**9,
**USER MATERIAL, CONSTANTS=9
**10100.,     0.3,       0.01,     80,     0.075,    0.547
**   2.284,
**INITIAL CONDITIONS, TYPE=SOLUTION
**all, 0.0, 0.0, 0.0, 0.0, 1.0, 0.0,
**0.0,0.0,0.0
*MATERIAL, NAME=USEREPOXY
*DAMPING, ALPHA=0.015
*DENSITY
 1.6E-09,
*DEPVAR, DELETE = 5
6,
*USER MATERIAL, CONSTANTS=9
1000000.,     -0.5,       1.,      150.,     150., 1E+06,    0.075,    0.547
    1.45,
*INITIAL CONDITIONS, TYPE=SOLUTION
all, 0.0, 0.0, 0.0, 0.0, 1.0, 0.0
*boundary
bottom, encastre
```

```
top,1,1
*AMPLITUDE, NAME = AMP1, DEFINITION = SMOOTH STEP
0., 0., 1., 1.
*Step, name=Step-1009, nlgeom=YES
*Dynamic, Explicit, ELEMENT BY ELEMENT
,0.15
*Bulk Viscosity
0.06, 1.2
*Fixed Mass Scaling, Factor=1000, Elset=ALL
*Boundary, AMPLITUDE = AMP1, type=VELOCITY
top, 3, 3, 0.75
top, 2, 2, 0.25
*Output, field
*Node Output
RF, U, V
*ELEMENT Output
SDV, NE
*End Step
*Step, name=Step-6767, nlgeom=YES
*Dynamic, Explicit, ELEMENT BY ELEMENT
,0.35
*Bulk Viscosity
0.06, 1.2
*Fixed Mass Scaling, Factor=1000, Elset=ALL
*Boundary, AMPLITUDE = AMP1, type=VELOCITY
top, 3, 3, 0.25
top, 2, 2, 0.75
*Output, field
*Node Output
RF, U, V
```

```
*ELEMENT Output
SDV, NE
*End Step
```

- **INPUT FILE FOR THE MODE-II SIMULATION**

```
** PARTS
**
*Part, name=CFRP
*Node, NSET = MASTERNODES
1071,          0.,       0.,       0.,
1172,          0.,     12.7,       0.,
1273,          0.,     25.4,       0.,
103071,        102,      0.,       0.,
103172,        102,    12.7,       0.,
103273,        102,    25.4,       0.,
*NGEN, NSET = A1
1071, 103071, 300
*NGEN, NSET = A2
1172, 103172, 300
*NGEN, NSET =A3
1273, 103273, 300
*NCOPY, CHANGE NUMBER = 1, OLD SET = A1, NEW SET = B1, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = A2, NEW SET = B2, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = A3, NEW SET = B3, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = B1, NEW SET = C1, SHIFT
0., 0., 0.39
```

```
*NCOPY, CHANGE NUMBER = 1, OLD SET = B2, NEW SET = C2, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = B3, NEW SET = C3, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = C1, NEW SET = D1, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = C2, NEW SET = D2, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = C3, NEW SET = D3, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = D1, NEW SET = E1, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = D2, NEW SET = E2, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = D3, NEW SET = E3, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = E1, NEW SET = F1, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = E2, NEW SET = F2, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = E3, NEW SET = F3, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = F1, NEW SET = G1, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = F2, NEW SET = G2, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = F3, NEW SET = G3, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = G1, NEW SET = H1, SHIFT
0., 0., 0.39
```

```
*NCOPY, CHANGE NUMBER = 1, OLD SET = G2, NEW SET = H2, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = G3, NEW SET = H3, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = H1, NEW SET = I1, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = H2, NEW SET = I2, SHIFT
0., 0., 0.39
*NCOPY, CHANGE NUMBER = 1, OLD SET = H3, NEW SET = I3, SHIFT
0., 0., 0.39
*ELEMENT, TYPE=C3D8R
1, 1371, 1472, 1172, 1071, 1372, 1473, 1173, 1072
*ELGEN, ELSET = SOLID
1, 340, 300, 1, 2, 101, 340, 8, 1, 680
*Node, NSET = COHESIVEMASTER
201075,           0.,        0.,       1.55999994
201176,           0.,      12.7,       1.55999994
201277,           0.,      25.4,       1.55999994
303075,          102,        0.,       1.55999994
303176,          102,      12.7,       1.55999994
303277,          102,      25.4,       1.55999994
*NGEN, NSET = X1
201075, 303075, 300
*NGEN, NSET = X2
201176, 303176, 300
*NGEN, NSET =X3
201277, 303277, 300
*ELEMENT, TYPE=COH3D8
5441, 201375, 201476, 201176, 201075, 1375, 1476, 1176, 1075
*ELGEN, ELSET = COHESIVE
```

```
5441, 250, 300,1, 2, 101, 250
*ELSET, ELSET = COHESIVE
COHESIVED, COHESIVEUND
*SURFACE, NAME = LAMINAFRICTION, TYPE = ELEMENT
COHESIVE, S1
*ORIENTATION, NAME=ORI-1
1., 0., 0., 0., 1., 0.,
*SOLID SECTION, ELSET=LAMINATE, ORIENTATION=ORI-1, CONTROLS=CTRLS_LAMINAE, MATERIAL=CFRP
*COHESIVE SECTION, ELSET=COHESIVE, MATERIAL=USEREPOXY, RESPONSE=TRACTION SEPARATION, THICKNESS=SPECIFIED
1.,
*SECTION CONTROLS, NAME=CTRLS_LAMINAE, HOURGLASS=RELAX STIFFNESS
 1.0, 1.0, 1.0, 1.0, 1.0
*AMPLITUDE, NAME = ZEE, DEFINITION = SMOOTH STEP
0., 0., 1., 1.
*ELSET, ELSET=MSCALING
COHESIVE, LAMINATE
*MATERIAL, NAME=CFRP
*DAMPING, ALPHA=0.015
*DENSITY
 1.6E-09,
*ELASTIC, TYPE=ENGINEERING CONSTANTS
122700.,10100.,10100.,  0.25,  0.25,  0.45, 5500., 5500.
 3700.,
*MATERIAL, NAME=USEREPOXY
*DAMPING, ALPHA=0.015
*DENSITY
 1.6E-09,
*DEPVAR, DELETE = 5
```

```
6,
*USER MATERIAL, CONSTANTS=9
1000000.,     -0.5,       1.,      80.,     106., 1E+06,    0.969,    1.719
   2.284,
*INITIAL CONDITIONS, TYPE=SOLUTION
COHESIVEUND, 0.0, 0.0, 0.0, 0.0, 1.0, 0.0
*INITIAL CONDITIONS, TYPE=SOLUTION
COHESIVED, 0.0, 0.0, 0.0, 1.0, 1.0, 0.0
*Surface Interaction, name=Laminae_Friction
*Friction
 0.5,
*Surface Behavior, pressure-overclosure=HARD
*BOUNDARY
pinn, 3,3
clamp, 1,3
*STEP, NAME=STEP1, NLGEOM=YES
*DYNAMIC, EXPLICIT, ELEMENT BY ELEMENT
, 11.
*BULK VISCOSITY
0.06, 1.2
*FIXED MASS SCALING, ELSET=MSCALING, FACTOR=10000.
*Contact, op=NEW
*Contact Inclusions, ALL EXTERIOR
*Contact Property Assignment
 ,  , LAMINAE_FRICTION
*BOUNDARY, AMPLITUDE=ZEE, TYPE=VELOCITY
topl, 3, 3, -0.5
*RESTART, WRITE, NUMBER INTERVAL=1, TIME MARKS=NO
*OUTPUT, FIELD
*NODE OUTPUT
```

```
RF, U, V
*END STEP
```

VUMAT – ZERO THICKNESS COH3D8 ELEMENT

```
      SUBROUTINE VUMAT(
C READ ONLY (UNMODIFIABLE)VARIABLES -
     1  NBLOCK, NDIR, NSHR, NSTATEV, NFIELDV, NPROPS, LANNEAL,
     2  STEPTIME, TOTALTIME, DT, CMNAME, COORDMP, CHARLENGTH,
     3  PROPS, DENSITY, STRAININC, RELSPININC,
     4  TEMPOLD, STRETCHOLD, DEFGRADOLD, FIELDOLD,
     5  STRESSOLD, STATEOLD, ENERINTERNOLD, ENERINELASOLD,
     6  TEMPNEW, STRETCHNEW, DEFGRADNEW, FIELDNEW,
C WRITE ONLY (MODIFIABLE) VARIABLES -
     7  STRESSNEW, STATENEW, ENERINTERNNEW, ENERINELASNEW )
C
      INCLUDE 'VABA_PARAM.INC'
C
      DIMENSION PROPS(NPROPS), DENSITY(NBLOCK), COORDMP(NBLOCK,*),
     1  CHARLENGTH(NBLOCK), STRAININC(NBLOCK,NDIR+NSHR),
     2  RELSPININC(NBLOCK,NSHR), TEMPOLD(NBLOCK),
     3  STRETCHOLD(NBLOCK,NDIR+NSHR),
     4  DEFGRADOLD(NBLOCK,NDIR+NSHR+NSHR),
     5  FIELDOLD(NBLOCK,NFIELDV), STRESSOLD(NBLOCK,NDIR+NSHR),
     6  STATEOLD(NBLOCK,NSTATEV), ENERINTERNOLD(NBLOCK),
     7  ENERINELASOLD(NBLOCK), TEMPNEW(NBLOCK),
     8  STRETCHNEW(NBLOCK,NDIR+NSHR),
     8  DEFGRADNEW(NBLOCK,NDIR+NSHR+NSHR),
     9  FIELDNEW(NBLOCK,NFIELDV),
```

```
     1  STRESSNEW(NBLOCK,NDIR+NSHR), STATENEW(NBLOCK,NSTATEV),
     2  ENERINTERNNEW(NBLOCK), ENERINELASNEW(NBLOCK)
C
      CHARACTER*80 CMNAME
C
      PARAMETER(ZERO = 0.0D0, ONE = 1.0D0, TWO = 2.0D0, HALF = .5D0)
C
C
      EM = PROPS(1)
      VM = PROPS(2)
      HE = PROPS(3)
      T3 = PROPS(4)
      T2 = PROPS(5)
      PEN = PROPS(6)
      GIC = PROPS(7)
      GIIC = PROPS(8)
      ETA = PROPS(9)
C
      GM = EM / (TWO * (ONE + VM))
      D30 = T3 / PEN
      D20 = T2 / PEN
      D3F = (TWO * GIC)/T3
      D2F = (TWO * GIIC)/T2
      DO K=1,NBLOCK
            S1 = STATEOLD(K,1)
            S2 = STATEOLD(K,2)
            S3 = STATEOLD(K,3)
            S4 = STATEOLD(K,4)
            S5 = STATEOLD(K,5)
            S11 = STATEOLD(K,6)
```

```
      D3 = S1 + STRAININC(K,1)
      D1 = S2 + STRAININC(K,2)
      D2 = S3 + STRAININC(K,3)
C
C
      D4 =  HALF * ( D3 + ABS(D3) )
      DS =  (D1*D1 + D2*D2) ** HALF
      ALAM = (D4*D4 + DS*DS) ** HALF
      BETA = (DS*DS) / (ALAM * ALAM)
      B = (PEN * BETA) / ( (PEN * BETA) + PEN * ( ONE - BETA) )
         BN = (B)**ETA
      D30B = ( ((PEN * ( ONE - BN ))*(D30*D30) + ((PEN*BN)*(D20*D20))) / ( PEN +
((BETA/ONE-BETA) * PEN )) ) ** HALF
      D3FB = ( ( PEN * ( ONE - BN ) * D30 * D3F ) + ( PEN * BN * D20 * D2F ) )/  ( ( PEN
+ ((BETA/ONE-BETA) * PEN ) ) * D30B )
      D20B = ( ( BETA / ONE - BETA ) ** HALF ) * D30B
      D2FB = ( ( BETA / ONE - BETA ) ** HALF ) * D3FB
      DONS = (( D30B * D30B ) + ( D20B * D20B ) )** HALF
      DF =   (( D3FB * D3FB ) + ( D2FB * D2FB ) )** HALF
      G = ( DF * ( ALAM - DONS ) ) / ( ALAM * ( DF - DONS ) )
         IF (G.GT.ONE) THEN
         G = ONE
         END IF
      IF (G.GT.S4) THEN
          S6 = G
      ELSE
          S6 = S4
         END IF
         IF (S6.EQ.ONE) THEN
         S12 = ZERO
         ELSE
```

```
          S12 = ONE
          END IF
          IF (D3.LT.ZERO) THEN
          S13 = ZERO
          ELSE
          S13 = S6
          END IF
          IF (STEPTIME.EQ.ZERO) THEN
      STRESSNEW(K,1) =   EM * D3
      STRESSNEW(K,2) =   GM * D1
      STRESSNEW(K,3) =   GM * D2
      ELSE
      STRESSNEW(K,1) =  ( ONE - S13 ) * EM * D3
      STRESSNEW(K,2) =  ( ONE - S6 ) * GM * D1
      STRESSNEW(K,3) =  ( ONE - S6 ) * GM * D2
          END IF
C
C
      D5 = HALF * ( (-D3) + ABS(D3) )
      E1 = HALF * ( (S6 - S4) / DT ) * ( (PEN * ( (D3*D3) +
     1 ( D3 * D5 ))) + ( PEN * ( (D1*D1)+(D2*D2) )))
      S7 = S11 + E1
      FREEE = HALF * (  (( ONE - S6 ) * ( (PEN * ( ( D1*D1) +
     1 (D2*D2) )) + (PEN * ( D3 * D3 ) ))) -
     2 ( HALF * S6 * PEN * D3 * D5 ))
      ENERINTERNNEW(K) = S7 + FREEE
          STATENEW(K,1) = D3
          STATENEW(K,2) = D1
          STATENEW(K,3) = D2
          STATENEW(K,4) = S6
```

```
            STATENEW(K,5) = S12
            STATENEW(K,6) = S7
      END DO
           RETURN
           END
```

- **VUMAT FOR FINITE THICKNESS C3D8R RLEMENT**

```
      SUBROUTINE VUMAT(
C READ ONLY (UNMODIFIABLE)VARIABLES -
     1  NBLOCK, NDIR, NSHR, NSTATEV, NFIELDV, NPROPS, LANNEAL,
     2  STEPTIME, TOTALTIME, DT, CMNAME, COORDMP, CHARLENGTH,
     3  PROPS, DENSITY, STRAININC, RELSPININC,
     4  TEMPOLD, STRETCHOLD, DEFGRADOLD, FIELDOLD,
     5  STRESSOLD, STATEOLD, ENERINTERNOLD, ENERINELASOLD,
     6  TEMPNEW, STRETCHNEW, DEFGRADNEW, FIELDNEW,
C WRITE ONLY (MODIFIABLE) VARIABLES -
     7  STRESSNEW, STATENEW, ENERINTERNNEW, ENERINELASNEW )
C
      INCLUDE 'VABA_PARAM.INC'
C
      DIMENSION PROPS(NPROPS), DENSITY(NBLOCK), COORDMP(NBLOCK,*),
     1  CHARLENGTH(NBLOCK), STRAININC(NBLOCK,NDIR+NSHR),
     2  RELSPININC(NBLOCK,NSHR), TEMPOLD(NBLOCK),
     3  STRETCHOLD(NBLOCK,NDIR+NSHR),
     4  DEFGRADOLD(NBLOCK,NDIR+NSHR+NSHR),
     5  FIELDOLD(NBLOCK,NFIELDV), STRESSOLD(NBLOCK,NDIR+NSHR),
     6  STATEOLD(NBLOCK,NSTATEV), ENERINTERNOLD(NBLOCK),
     7  ENERINELASOLD(NBLOCK), TEMPNEW(NBLOCK),
     8  STRETCHNEW(NBLOCK,NDIR+NSHR),
     8  DEFGRADNEW(NBLOCK,NDIR+NSHR+NSHR),
     9  FIELDNEW(NBLOCK,NFIELDV),
```

```
     1  STRESSNEW(NBLOCK,NDIR+NSHR), STATENEW(NBLOCK,NSTATEV),
     2  ENERINTERNNEW(NBLOCK), ENERINELASNEW(NBLOCK)
C
      CHARACTER*80 CMNAME
C
      PARAMETER(ZERO = 0.0D0, ONE = 1.0D0, TWO = 2.0D0, HALF = .5D0, THREE =
3.0D0, MINUS = -1.0D0)
C
C
      EM = PROPS(1)
      VM = PROPS(2)
      HE = PROPS(3)
      T3 = PROPS(4)
      GIC = PROPS(5)
      GIIC = PROPS(6)
      ETA = PROPS(7)
          T2 = T3 * ((GIIC/GIC)**HALF)
C
      GM = EM / (TWO * (ONE + VM))
         PEN1 = EM / HE
         PEN2 = GM / HE
      D30 = T3 / PEN1
      D20 = T2 / PEN2
      D3F = (TWO * GIC)/T3
      D2F = (TWO * GIIC)/T2
      DO K=1,NBLOCK
          S1 = STATEOLD(K,1)
          S2 = STATEOLD(K,2)
          S3 = STATEOLD(K,3)
          S4 = STATEOLD(K,4)
          S5 = STATEOLD(K,5)
```

```
         S11 = STATEOLD(K,6)
         S15 = STATEOLD(K,7)
         S16 = STATEOLD(K,8)
         S17 = STATEOLD(K,9)
C
C
      D33 = S1 + STRAININC(K,3)
      D11 = S2 + STRAININC(K,5)
      D22 = S3 + STRAININC(K,6)
         D15 = S15 + STRAININC(K,1)
         D16 = S16 + STRAININC(K,2)
         D17 = S17 + STRAININC(K,4)
         D1 = (TWO * D11) * HE
         D2 = (TWO * D22) * HE
         D3 = HE * D33
C
C
      D4 =  HALF * ( D3 + ABS(D3) )
      DS =  (D1*D1 + D2*D2) ** HALF
      ALAM = (D4*D4 + DS*DS) ** HALF
      BETA = (DS*DS) / (ALAM * ALAM)
      B = (PEN2 * BETA) / ( (PEN2 * BETA) + PEN1 * ( ONE - BETA) )
         BN = (B)**ETA
      D30B = ( ((PEN1 * ( ONE - BN ))*(D30*D30) + ((PEN2*BN)*(D20*D20))) / ( PEN1 + ((BETA/ONE-BETA) * PEN2 )) ) ** HALF
      D3FB = ( ( PEN1 * ( ONE - BN ) * D30 * D3F ) + ( PEN2 * BN * D20 * D2F ) )/  ( ( PEN1 + ((BETA/ONE-BETA) * PEN2 ) ) * D30B )
      D20B = ( ( BETA / ONE - BETA ) ** HALF ) * D30B
      D2FB = ( ( BETA / ONE - BETA ) ** HALF ) * D3FB
      DONS = (( D30B * D30B ) + ( D20B * D20B ) )** HALF
      DF =   (( D3FB * D3FB ) + ( D2FB * D2FB ) )** HALF
```

```
      G = ( DF * ( ALAM - DONS ) ) / ( ALAM * ( DF - DONS ) )
            IF (G.GT.ONE) THEN
            G = ONE
            END IF
      IF (G.GT.S4) THEN
             S6 = G
      ELSE
             S6 = S4
            END IF
            IF (S6.EQ.ONE) THEN
            S12 = ZERO
            ELSE
            S12 = ONE
            END IF
            IF (D3.LT.ZERO) THEN
            S13 = ZERO
            ELSE
            S13 = S6
            END IF
            APHI = ONE - (TWO*(VM**TWO)*(ONE-S13)) - VM
            AGAMMA = ONE - (TWO*(VM**THREE)*(ONE-S13)) -
 ((VM**TWO)*(THREE-(TWO*S13)))
            COMPLIANCE1 = EM * (ONE - ((VM**TWO)*(ONE-S13)))
            COMPLIANCE2 = VM * EM * ( MINUS - ((VM)*(ONE-S13)))
            COMPLIANCE3 = VM * EM * (ONE-S13)
            COMPLIANCE4 = (EM * (ONE-S13) * (ONE-VM))
            IF (STEPTIME.EQ.ZERO) THEN
      STRESSNEW(K,1) =   EM * D15
      STRESSNEW(K,2) =   EM * D16
      STRESSNEW(K,3) =   EM * D33
            STRESSNEW(K,4) =   GM * D17
```

```
      STRESSNEW(K,5) =   GM * D11
      STRESSNEW(K,6) =   GM * D22
      ELSE
      STRESSNEW(K,1) =   (((COMPLIANCE1/AGAMMA)*D15)-
((COMPLIANCE2/AGAMMA)*D16)+((COMPLIANCE3/APHI)*D33))
      STRESSNEW(K,2) =   (-
((COMPLIANCE2/AGAMMA)*D15)+((COMPLIANCE1/AGAMMA)*D16)+((COMPLIA
NCE3/APHI)*D33))
      STRESSNEW(K,3) =
(((COMPLIANCE3/APHI)*D15)+((COMPLIANCE3/APHI)*D16)-
((COMPLIANCE4/APHI)*D33))
           STRESSNEW(K,4) =   GM * D17
      STRESSNEW(K,5) =   GM * D11 * (ONE-S6)
      STRESSNEW(K,6) =   GM * D22 * (ONE-S6)
           END IF
C
C
      D5 = HALF * ( (-D3) + ABS(D3) )
      E1 = HALF * ( (S6 - S4) / DT ) * ( (PEN1 * ( (D3*D3) +
     1 ( D3 * D5 ))) + ( PEN2 * ( (D1*D1)+(D2*D2) )))
      S7 = S11 + E1
      FREEE = HALF * (  (( ONE - S6 ) * ( (PEN2 * ( ( D1*D1) +
     1 (D2*D2) )) + (PEN1 * ( D3 * D3 ) ))) -
     2 ( HALF * S6 * PEN1 * D3 * D5 ))
      ENERINTERNNEW(K) = S7 + FREEE
           STATENEW(K,1) = D33
           STATENEW(K,2) = D11
           STATENEW(K,3) = D22
           STATENEW(K,4) = S6
           STATENEW(K,5) = S12
           STATENEW(K,6) = S7
           STATENEW(K,7) = S15
```

```
          STATENEW(K,8) = S16
          STATENEW(K,9) = S17
END DO
         RETURN
         END
```