ADVANCED MESOMECHANICAL MODELING OF TRIAXIALLY BRAIDED COMPOSITES FOR DYNAMIC IMPACT ANALYSIS WITH FAILURE

A Dissertation

Presented to

The Graduate Faculty of The University of Akron

In Partial Fulfillment

of the Requirements of the Degree

Doctor of Philosophy

Zifeng Nie

August, 2014
ADVANCED MESOMECHANICAL MODELING OF TRIAXIALLY BRAIDED
COMPOSITES FOR DYNAMIC IMPACT ANALYSIS WITH FAILURE

Zifeng Nie

Dissertation

Approved: ____________________________    ______ ____________________
Advisor      Department Chair
Dr. Wieslaw K. Binienda    Dr. Wieslaw K. Binienda

Accepted: ____________________________                              __________________________
Committee Member     Dean of the College
Dr. Ernian Pan                  Dr. George K. Haritos

Committee Member     Dean of the Graduate School
Dr. Guo-Xiang Wang                 Dr. George R. Newkome

Committee Member        Date
Dr. Robert K. Goldberg

Committee Member
Dr. Qindan Huang

Committee Member
Dr. Kevin Kreider
ABSTRACT

Numerical simulation plays an irreplaceable role in reducing time and cost for the development of aerospace and automotive structures, such as composite fan cases, car roof and body panels etc. However, a practical and computationally-efficient methodology for predicting the performance of large braided composite structures with the response and failure details of constituent level under both static and impact loading has yet to be developed.

This study focused on the development of efficient and sophisticated numerical analysis modeling techniques suitable for two-dimensional triaxially braided composite (TDTBC) materials and structures under high speed impact. A new finite element analysis (FEA) based mesomechanical modeling approach for TDTBC was developed independently and demonstrated both stand alone and in the combined multi-scale hybrid FEA as well. This new mesoscale modeling approach is capable of considering the detailed braiding geometry and architecture as well as the mechanical behavior of fiber tows, matrix, and the fiber tow interface, making it feasible to study the details of localized behavior and global response that happen in the complex constituents. Furthermore, it also accounts for the strain-rate effects on both elastic and inelastic behavior and the failure/damage mechanism in the matrix material, which had been long observed in experiments but were neglected for simplicity by researchers. It is capable of
simulating inter-laminar and intra-laminar damage and delamination of braided composites subjected to dynamic loading. With high fidelity in both TDTBC architecture and mechanical properties, it is well suited to analyze high speed impact events with improved simulation capability in both accuracy and efficiency. Special attention was paid to the applicability of the method to relatively large scale components or structures.

In addition, a novel hybrid multi-scale finite element analysis method, entitled Combined Multiscale Modeling (CMM) approach, has been developed in this comprehensive study in conjunction with dynamic submodeling technique. It was based on the newly developed mesoscale and existing macroscale approaches for modeling the braided composite materials. The CMM hybrid FEA approach enables the full use of the advantages of both the macroscale and the mesoscale approaches, with the mesoscale model or a more detailed macro-scale model to describe the details of local deformation and the macro-scale model or a coarser mesoscale model to capture the global overall response feature of the entire structure. The approach was verified with simple testing specimens and coupon plates, and may be extended to large systems like jet engine containment or automotive body panels. Without directly connecting different portions of the structure modeled with disparate approaches in the same analysis model, the submodeling technique maps the solution of a global model analysis performed for the full structure with less details onto the connecting interface on the portion of the same structure, the submodel, modeled with high fidelity and details. The CMM approach presented here captures the response feature of a triaxially braided composite structure under impact accurately with a much lower computational expense, making it feasible to analyze this type of analysis for exceedingly large structures.
ACKNOWLEDGEMENTS

I would like to gratefully thank my advisor Dr. Wiesław Binienda for his prompt support, valuable guidance, and continuous encouragement throughout the long journey of my graduate studies. Sincere acknowledgement also goes to Dr. Robert Goldberg of NASA Glenn Research Center, who read preliminary versions of my papers and publications with constructive directions.

Acknowledgements are extended to my IDC committee members, Dr. Ernian Pan and Dr. Qindan Huang from Civil Engineering Department, Dr. Guo-Xiang Wang from Mechanical Engineering Department and Dr. Kevin Kreider from Applied Mathematics, for their continuous advice and support.

I would also like to thank my colleagues, Dr. Juan Hurtado, Dr. Bill Grimes, Dr. Victor Oancea and Mr. Mark Bohm at DS Simulia Corp, for their beneficial discussions, Dr. Gary Roberts and Dr. Lee Kohlman of NASA Glenn Research Center, Dr. Xuetao Li of General Electric for their ideas, test results, and other support to conduct this research. Special thanks are given to my fellow graduate students Dr. Chris Zhang and Zhuopei Hu, for their consistent help and discussion throughout my graduate study.

Finally, I owe my family much for leaving all the housework and parents responsibility to my dear wife, Julie Li. And thanks to my sons Yunbo and Bryan for their independence, understanding and love during my long-lasting PhD work.
TABLE OF CONTENTS

LIST OF TABLES ............................................................................................................... x
LIST OF FIGURES........................................................................................................... xi
CHAPTER ....................................................................................................................... 1

I. INTRODUCTION ...................................................................................................... 1
   1.1 Background ...................................................................................................... 1
   1.2 Motivation ..................................................................................................... 2

II. LITERATURE REVIEW .......................................................................................... 7
   2.1 Numerical and analytical modeling of textile composites ......................... 7
      2.1.1 Overview and classification of Multi-scale modeling ...................... 8
      2.1.2 Micro-scale modeling approaches ............................................... 17
      2.1.3 Meso-scale modeling approaches ............................................. 22
      2.1.4 Macro-scale modeling approaches ......................................... 25
      2.1.5 Multiscale modeling ............................................................... 27
   2.2 Modeling damage and failure of composites ....................................... 29
      2.2.1 Failure criteria approach ......................................................... 29
      2.2.2 Damage mechanics approach ............................................... 31
      2.2.3 Fracture mechanics approach ............................................... 34
      2.2.4 The CZone approach for crush .......................................... 40
      2.2.5 Plasticity approach ............................................................... 41
   2.3 Review of Abaqus/Explicit capability for dynamics and composites modeling.42
2.3.1 Abaqus/Explicit solver review in modeling impact analysis ..........42
2.3.2 Modeling composites with Abaqus...............................................43
2.4 Current Research .........................................................................44

III. INVESTIGATIONS AND CHARACTERIZATION OF TRIAXIAL BRAIDED
COMPOSITE MATERIAL PROPERTIES AND RESPONSE..........................50
3.1 Material Properties of composite components .................................51
3.2 Strain-rate effect of the resin ..........................................................52
  3.2.1 Strain-rate dependent elastic properties ........................................52
  3.2.2 Strain-rate dependent plastic properties ........................................53
3.3 Failure / Damage Modeling of the matrix........................................53
  3.3.1 Strain-rate dependent damage initiation strain ................................56
  3.3.2 Strain-rate dependent damage evolution displacement and energy ....56
3.4 Bonding interface property ...............................................................56

IV. DEVELOPMENT OF A MESO MECHANICAL MODEL FOR DYNAMIC
TRIAXIAL BRAIDED COMPOSITE ANALYSIS WITH DAMAGE AND STRAIN
RATE EFFECT..........................................................................................61
4.1 Component Material Background .....................................................64
4.2 Object composites geometry and unit constituent cell .......................65
4.3 Determination of constituent dimensions and fiber volume ratio ........69
4.4 Method of modeling each constituent components in TDTBC .............71
  4.4.1 Pure resin matrix modeling ..............................................................73
  4.4.2 Fiber tows modeling ....................................................................77
  4.4.3 Bonding interface consideration ......................................................85
4.5 The stable time limit ......................................................................86
4.6 Verification example – 2 by 3 unit cells impacted by a rigid projectile ....88
V. DEVELOPMENT OF A COMBINED MULTISCALE MODELING APPROACH FOR DYNAMIC ANALYSIS OF TRIAXIAL BRAIDED COMPOSITE

5.1 Methods to Associate Regions Modeled with Different Approaches

5.1.1 Direct connecting method

5.1.2 Submodeling Method

5.2 Effectiveness of the submodeling method

5.3 Assumptions and approximations

5.4 Measures and cautions for the effectiveness of submodeling method

5.5 Submodel analysis of a circular plate under impact

5.6 Submodel analysis of a square plate under impact

5.7 Basics for choosing location of the submodel boundary

5.8 Special cares for material damage or failure

5.9 Cause and preventative measures for mesh sensitivity on material failure

VI. APPLICATION EXAMPLES

6.1 Simulation of the axial tensile test of a simple specimen

6.1.1 Simulation of the full specimen model

6.1.2 Simulation of a quarter specimen model

6.2 Ballistic impact of a circular plate

6.2.1 The analysis model and general information

6.2.2 Simulation result

6.2.3 Validation of the CMM approach with submodeling

VII. SUMMARY AND FUTURE WORK

7.1 Summary

7.2 Future work
REFERENCES .......................................................................................................................... 151
APPENDIX .................................................................................................................................. 164
LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1 Composite Fiber Properties</td>
<td>51</td>
</tr>
<tr>
<td>3.2 Composite Resin Properties</td>
<td>52</td>
</tr>
<tr>
<td>3.3 Bonding Interface fracture toughness (J/mm²)</td>
<td>60</td>
</tr>
<tr>
<td>4.1 Unit cell and fiber bundle architecture dimensions</td>
<td>70</td>
</tr>
<tr>
<td>4.2 Finite element model geometric dimensions</td>
<td>71</td>
</tr>
<tr>
<td>4.3 Unit cell feature comparison with existing mesoscale models</td>
<td>73</td>
</tr>
<tr>
<td>4.4 Rate dependent elastic-plastic material properties of the E862 matrix</td>
<td>75</td>
</tr>
<tr>
<td>4.5 Rate Dependent Damage properties of the E862 matrix</td>
<td>76</td>
</tr>
<tr>
<td>4.6 Elastic constants of fiber bundles predicted by micromechanical RVE model</td>
<td>78</td>
</tr>
<tr>
<td>4.7 Strengths of fiber bundles predicted using micromechanical RVE model</td>
<td>78</td>
</tr>
<tr>
<td>6.1 Full meso model and submodel comparison of circular plate impact analysis</td>
<td>129</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1 Typical material response showing progressive damage</td>
<td>54</td>
</tr>
<tr>
<td>3.2 Schematic representation of elastic-plastic matrix with progressive damage</td>
<td>54</td>
</tr>
<tr>
<td>3.3 Energy-based damaged evolution</td>
<td>56</td>
</tr>
<tr>
<td>3.4 Traction-separation behavior of the bonding surface</td>
<td>58</td>
</tr>
<tr>
<td>3.5 Damaged traction-separation response for cohesive bonding surface</td>
<td>59</td>
</tr>
<tr>
<td>4.1 TDTBC representative geometry and dimension of the unit cell</td>
<td>66</td>
</tr>
<tr>
<td>4.2 Braided model representations – four unit cells</td>
<td>67</td>
</tr>
<tr>
<td>4.3 Assembled unit constituent cell</td>
<td>68</td>
</tr>
<tr>
<td>4.4 Microscopic axial cross-sectional image of a six-layer 0°/±60° carbon/epoxy</td>
<td>70</td>
</tr>
<tr>
<td>4.5 Four assembled conventional unit cells</td>
<td>72</td>
</tr>
<tr>
<td>4.6 Nominal stress-strain curves of E862 matrix at different strain rates</td>
<td>74</td>
</tr>
<tr>
<td>4.7 True stress - log strain curves of E862 at different strain rates</td>
<td>75</td>
</tr>
<tr>
<td>4.8 Unit cell geometry and mesh of the matrix</td>
<td>76</td>
</tr>
<tr>
<td>4.9 The geometry and mesh of the axial fiber-tows (two unit cell long)</td>
<td>83</td>
</tr>
<tr>
<td>4.10 The geometry and mesh of the bias fiber-tows</td>
<td>84</td>
</tr>
<tr>
<td>4.11 Cohesive contact interaction properties for the bonding material</td>
<td>86</td>
</tr>
<tr>
<td>4.12 The verification analysis model</td>
<td>88</td>
</tr>
<tr>
<td>4.13 Delaminating and damage of the fiber tows</td>
<td>89</td>
</tr>
<tr>
<td>4.14 Delamination and damage of the matrix</td>
<td>89</td>
</tr>
</tbody>
</table>
4.15 Failure of the fiber tows ......................................................................................90
4.16 Original and damaged bonding interface ............................................................90
5.1 Illustration of the direct connecting method ............................................................93
5.2 Illustration of the submodeling technique ...............................................................94
5.3 Different damage shapes from test .........................................................................97
5.4 Different damage shapes from simulation ...............................................................97
5.5 Submodeling illustration .......................................................................................98
5.6 Circularly clamped TDTBC plate impacted at 100m/s ...........................................101
5.7 Vertical Displacement of a circularly clamped TDTBC plate impacted at 100m/s .102
5.8 Comparison of critical displacement of the circular plate - impact point ..........102
5.9 Comparison of critical displacement of the circular plate - projectile ..............103
5.10 Impact of a clamped square TDTBC plate and the submodels .........................104
5.11 Vertical displacement at 0.805 ms .....................................................................105
5.12 Vertical displacement at 0.965 ms .....................................................................105
5.13 Vertical displacement at 1.0 ms .........................................................................105
5.14 Comparison of impact point displacement of the square plate .......................106
5.15 Comparison of projectile displacement of the square plate .........................106
5.16 Vertical Displacement of a circular clamped TDTBC plate impacted .............110
5.17 Damage initiation and progress mechanism for composite material ............111
6.1 Specimen simulation using CMM approach with submodeling ....................116
6.2 Test and numerical gauges used in the simulation .............................................117
6.3 Stress-strain curves from full and submodel simulation ....................................118
6.4. damaged pattern of the submodel ...................................................................119

xii
6.5 Quarter global model ........................................................................................................ 120
6.6 Stress locations used for comparison in the quarter global model ......................... 121
6.7 Principal stresses in the axial fiber tow ........................................................................ 121
6.8 Principal stress in positive 60° bias tows ..................................................................... 122
6.9 Principal stresses in negative 60° bias tows ............................................................. 122
6.10 Principal strain in positive 60° bias tows ............................................................... 123
6.11 Principal strain in negative 60° bias tows ............................................................ 123
6.12 Stress-strain curves from quarter model, submodel and the 3 unit cell model........ 124
6.13 Mesomechanical full model and submodel of the circular coupon plate.............. 127
6.14 Full model and submodel of the circular coupon plate - matrix removed ............. 128
6.15 Energy history: initial 50 m/s. full model - Circular plate of T700/E862 ............. 132
6.16 Energy history: initial 80 m/s. Circular plate of T700/E862 ............................... 132
6.17 Energy history: initial 150 m/s. Circular plate of T700/E862 .............................. 133
6.18 Damage pattern, at impact speed 150 m/s, circular plate of T700/E862 ............... 134
6.19 Damage pattern, macroscale model and test result, ............................................. 134
6.20 Front view of damage propagation, impact speed 80 m/s, circular plate of T700/E862 ................................................................................................................ 137
6.21 Back view of damage propagation, impact speed 80 m/s, circular plate of T700/E862 ................................................................................................................ 138
6.22 Damage propagation, impact speed 150 m/s, ......................................................... 139
6.23 Final damage pattern, impact speed 80 m/s, ......................................................... 141
6.24 Final damage pattern, impact speed 150 m/s, circular plate of T700/E862 ......... 142
6.25 Out-of-plane displacement, velocity and acceleration history of the projectile, impact speed 50 m/s, circular plate of T700/E862 ................................................................. 144
6.26 Out-of-plane displacement and velocity history of the projectile, impact speed 80 m/s, circular plate of T700/E862 ...........................................................................................................................144

6.27 Out-of-plane displacement, velocity and acceleration history of the projectile, impact speed 150 m/s, circular plate of T700/E862 ...........................................................................................................................145

6.28 Out-of-plane displacement history along the radial line, gauge point 1-4: impact speed 50 m/s, circular plate of T700/E862 ...........................................................................................................................145

6.29 Out-of-plane displacement history along the radial line, gauge point 1-4: impact speed 80 m/s, circular plate of T700/E862 ...........................................................................................................................146

6.30 Out-of-plane displacement history along the radial line, gauge point 1-4: impact speed 150 m/s, circular plate of T700/E862 ...........................................................................................................................146

6.31 Out-of-plane displacement history of the impact front and back point, .............147

6.32 Out-of-plane displacement history of the impact front and back point, .............147

6.33 Out-of-plane displacement history of the impact front and back point, .............147
CHAPTER I
INTRODUCTION

1.1 Background

Composite materials are tuned to exhibit outstanding performance in their mechanical, thermal, and/or electrical properties depending on the material mixture and properties of the microstructure. The overall material behavior is vastly influenced by the geometric architectures and material properties of the constituents, and the interfaces between them, for example, the braiding angles and constituent mechanical properties of both the fiber tows and the matrix resin for a braided composite. This tweaks the composite properties and thus it is essential to be included in the performance description of the final design.

Furthermore, even in situations in which the different polymers may behave similarly when tested as constituents, they, as fabricated composite materials having different fiber and polymer combinations, can still behave differently when under identical static or impact loading conditions. Therefore, the interaction between the fibers and polymer matrix, also known as the bonding and debonding, is also of great significance and interest, and must be examined. Compared with unidirectional composites, tri-axially braided composites in the three systems of yarns are intertwined diagonally. This fiber architecture offers an improved resistance to interlaminar cracking
and delamination due to impact. Moreover, the geometrical parameters and material behavior of the composite such as the braiding angle, fiber undulation, resin properties and the fiber-matrix bonding interface all contribute to affect the deformation behavior and failure mechanisms.

1.2 Motivation

With promises of great performance, textile polymer composite materials and structures still present many challenges to researchers. Numerical simulation plays a critical role in reducing time and cost for the development of structures and systems made with newly-developed material like Two-dimensional Triaxially Braided Composites (TDTBC) in aerospace and automotive industry. However, a practical, computationally-efficient methodology for fully predicting the performance of braided composite structures in relatively large scale under impact loading has yet to be developed. The main difficulty of the task lies in that, with the high complexity of the material architecture and the engineering phenomena involved for the exceedingly large structures, the real applications still require both accuracy and efficiency to capture the essential geometric architecture with a practical computational cost.

Predictive simulations are key to the development of braided composite materials, as well as their application in structural design. Mirroring the full process chain in the simulation approach of designing new parts provides the means to shorten development time and to reduce cost for experimental testing. Numerical simulation is essential for evolving the braided composites simulation methodology to deliver more accurate results to meet regulatory and competitive demands. Especially, in both automotive and aerospace industry, appropriate advanced simulation tools can improve the design,
increase the value of testing, and significantly reduce the amount and scale of physical testing from what is currently required. Therefore, it is critical to develop effective FEA methods capable of performing not only static, but also dynamic simulations such as impact, collision, BVID, and bird strikes.

Modeling and accurately simulating the impact characteristics of braided composite structures is challenging. A dilemma exists here: on one hand a sophisticated, detailed model with high geometric and material fidelity is needed to predict the failure and damage and other nonlinearity involved in the response, such as delamination and matrix cracking and fiber tow breakage etc.; on the other hand, the large scale of the actual structures requires that the model be sufficiently simplified to make the computation practical. This contradiction leads to two distinctive representative approaches that have co-existed historically: the so-called macro-scale approach and meso-scale approach.

Typically, a macro-scale modeling approach simplifies the braided composite as a homogeneous layer(s) or textile layer(s) and uses the homogenized material properties calculated from the classical theory or other analytical or experimental methods based on the constituent details of the composite to represent the material in a “macro” way and ignore the detailed geometry on constituent level. On the contrary, a meso-scale modeling approach describes the fiber tow architecture and geometry in details and often addresses the bonding interface between the constituents with detailed material properties including failure/damage initiation and propagation. In macro-scale or meso-scale approach, oftentimes, analysis on the fiber filament and matrix level is needed to obtain the material properties on the constituent level, which is termed as the micro-scale analysis and
usually employed as a supplementary tool to determine the material properties. It is very rare to analyze a real structure with the micro-scale approach because of its huge computational cost. Generally, the macro-scale approach has the computationally efficient feature, while the meso-scale approach has more accurate predictive capability. It is obvious that using sophisticated meso-scale model over the entire structure can be not only too expensive to be practical, but also not necessary as the severe deformation or damage and high nonlinearity only happen at limited locations. Most parts of the structure actually behave in the linear scope, far from reaching the state of failure. A simplified approach to modeling these zones such as a less refined meso-mechanical model or a macro-scale model is economical and sufficient. Hence, the problem left to be resolved is how these two regions of different modeling details or even different approaches can be combined to coexist harmonically and effectively in the same analysis model, with each playing full of its advantage. And consequently, developing a hybrid method to combine the two regions and/or approaches appropriately becomes one of the primary interests and objectives of the current study.

Generally speaking, to perform a large-scale structure made of triaxially braided composite material, the challenges are:

1) Judge the regions for which the particular approach most suitable such that the critical response feature of the structure is captured efficiently and economically;

2) Apply appropriate techniques to transition and associate from one region to the other where different techniques or details are applied to ensure the analysis result is smooth and reliable.
The primary objective of the current study is to address these challenges and develop a novel method which fully utilizes the advantage of both the macro-level and the meso-level approaches. Knowing the background as such, the goal of this research is twofold: to develop sophisticated approaches and reliable combining techniques.

A critical issue was encountered when the goal was set: there were no existing reported mesomechanical TDTBC models that were suitable for impact dynamic analysis of real scale structures, not even one for the full simulation of simple specimen or coupon test. Consequently, the development of a new meso-mechanical model suitable for real structure dynamic analysis became the utmost important and certainly the foundation task to perform first. To meet this need, a new “unit constituent cell with damage and strain-rate sensitivity” meso-scale model was developed, which is robust and suitable for high speed impact analysis using explicit finite element program. Compared with the currently existing and well used macro and mesoscopic approaches, the new approach developed in this study accounts for the strain rate dependency and material failure/damage of the matrix, the former of which was observed for resin matrix for TDTBC material but no numerical implementation had been reported and the latter of which must be treated for analysis in which failure is involved. Meanwhile, as part of the development work of the new meso-mechanical model, the plane-stress based Hashin failure criteria model was modified and extended to 3D analysis model and coded as a user material subroutine and implemented via the user interface provided by Abaqus/Explicit solver. Also, a progressive damage initiation and failure mechanism combined with elastic-plastic properties has first been employed in the matrix material for predicting the behavior of braided composite structures under impact correctly.
The combined meso-macro scale analysis approach was first verified with simple specimen testing and then applied to simulate a circular coupon plate. Nonlinear explicit dynamic analysis code Abaqus/Explicit was used throughout the research work with several of its user subroutines to facilitate special need for material modeling.

In addition, the discontinuity and non-physical numerical reflection along the interface where meso-mechanical model and macro-scale (shell) model meet were studied and an approach to effectively reducing this numerical shortcoming was recommended. Two different methods may be used to connect the macro-level modeled region and the meso-level modeled region: one is the natural geometric connection, or mathematical interpolation when meshing difficulties is hard to overcome; the other is the use of Dynamic Submodeling technique. A comparison was performed between the two methods in their efficiency and accuracy.
CHAPTER II

LITERATURE REVIEW

This chapter presents a literary review of the relevant research done for the topics proposed in this study. The overview is intended to give a background regarding some of the methods currently in use, and present comparisons and extensions for the applicability of this research. Primarily in this review, the numerical methods and techniques of finite element analysis modeling and some applicable results for the textile composites will be presented comprehensively with an emphasis on the two dimensional triaxially braided composites (TDTBC) and their dynamic behavior and failure/damage modeling. First, the previous research into multi-scale modeling techniques for textile composites will be reviewed in the format of the three typical classifications: the micro-, meso- and macro-scale approaches. Afterwards, an overview of the capability of the Abaqus/Explicit code for modeling and simulating dynamic response of structures and for composites with failure/damage modeling will be explored briefly. Finally, a brief discussion and summary on the current research target objects and goals, and the implementation methods will be given.

2.1 Numerical and analytical modeling of textile composites

This section gives a comprehensive review on the existing FEA methods for modeling composites. The micro-, macro- and meso-mechanical modeling approaches is
discussed one by one in detail. Particularly, the multiscale hybrid modeling and the predictability for composite damage and failure are emphasized.

2.1.1 Overview and classification of Multi-scale modeling

A significant amount of research has been done for textile polymer composites, both woven and braided. Based on the need for different applications, numerical analysis models may adopt different modeling scales and analysis methods. Methods commonly used in the modeling and analysis of textile composites can be generally classified into three major groups: micro-, meso-, and macro-scale modeling, plus a newly emerged hybrid modeling method. Although some of the three fundamental methods were already discussed briefly in section 1.2, a more general description on the primary features and application of these different methods is summarized here as below:

1. Microscale modeling: A microscale model defines the arrangement of fiber in the fiber bundle. Most of the microscale modeling approaches are usually used as first level homogenization in single laminar property prediction to obtain material parameters for fiber tows. Great success has been achieved to predict stiffness of laminar. Strength prediction is less successful due to the complexity of composite failure in local level. Meanwhile, great efforts have also been made to develop straightforward micro-scale approaches for textile fabric composites.

2. Mesoscale modeling: A mesoscale model relates to the fabric structure and fiber bundle undulation. The main characteristic of mesoscale modeling approach lies in that the textile fabric geometry and architecture are explicitly modeled. Volumes of fabric reinforcement and matrix are distinguished and specific material properties will be assigned accordingly. Thanks to this feature of detailed composite architecture, some
sophisticated behavior such as debonding or delamination between differently orientated fiber tows, matrix failures and other types of damage mechanisms can be implemented.

This dissertation developed a new mesoscale modeling approach that is suitable for the impact analysis of large TDTBC structures as one of the main objectives due to the absence of such a method in published reports.

3. Macroscale modeling: A macroscale model connects the local representative unit cells to form the overall structure. This approach is phenomenological, aiming at establishing overall constitutive models for the proposed composite material. Anisotropic constitutive equations are obtained to depict the composite behavior, which requires knowledge of a set of effective material parameters, such as material moduli, Poisson’s ratios, and strengths in different material directions. Usually empirical material models are adopted. Simple models could be used and are usually applied in the impact simulations due to computational efficiency. However, Macroscale models are generally not able to predict component level behavior such as fiber tows slippage, breakage, resin damage and rate effect or interface debonding explicitly.

4. Hybrid modeling: This is one of the focuses of the current research. It is a multiscale method that combines the mesoscale and macroscale modeling approach appropriately and efficiently for the same analysis to make full use of the advantages of each approach. This approach ideally balances the accuracy requirements and the details of modeling, thus very useful for analyzing exceedingly large systems where the computing resources become an issue. However, issues often rise on finding an effective and correct way to connect and associate the global models built with a macro-scale approach and the local models built with a mesoscale model.
In general, there are two methods commonly employed to obtain material parameters in each scale level, i.e. the “bottom up” approach and the “top down” approach (Littell 2008). The “bottom up” approach refers to using some of the homogenization theories and implementing constituents’ properties directly into model development; whereas the “top down” approach refers to developing different methods for material properties incorporation by using material parameters obtained from higher scale material testing.

Although the majority of the early efforts for the analysis and modeling of textile composites concentrated on developing means to determine the effective mechanical properties of woven materials. As the main object of this research, the braiding architecture competes with other composites by its low production cost and outstanding bending and crack resistance. The most recent state of art review of application of two-dimensional braided composite can be found in the work of Ayranci and Carey (2008).

The earliest attempts were primarily using the “bottom up” approach to model textile materials. And when used in the finite element analysis of a structure composed of woven or braided composite materials, homogenized elements were used for most cases, in which the architecture and geometry of the textile material were not directly considered in the numerical models. Among these early approaches for textile modeling was the work done by Chou and Ishikawa (1982, 1983). Originally, they used a mosaic model, in which the woven composite was approximated as a one-dimensional series of laminated cross-ply composites, and classical laminate theory in combination with iso-stress or iso-strain assumptions were applied to obtain the effective stiffness properties of the material. Later, the model was extended to account for the fiber undulations that were
commonly found in actual woven materials (1989). Naik and Shembekar extended this approach to two-dimensions (1992) by applying a mixture of parallel and series assumptions to obtain the effective material properties.

When it is needed to analyze more complicated fiber tow architectures like the braided composites, researchers attempted to model the fibers as a series of rods connected at different angles and utilized simple iso-strain assumptions to obtain the overall effective properties of the composite. Representative work in this regards includes those done by Pastore and Gowayed (1994) and Byun (2000). More sophisticated analysis methods, such as those developed by Tanov and Tabiei (2001) and Bednarcyk and Arnold (2003), used an approach based on the concept of the so-called unit cell (Mishnaevsky and Schmauder 2001) for the woven composite with micromechanics-based approaches applied to compute the overall effective properties of the woven material. Again, when applying these methods within a finite element frame, a homogenized set of material properties for the constituents were used to the discretized model with appropriate analysis procedure imposed to the structure to generate the effective properties of the composite.

This method seemed to work with the regularly oriented woven materials; however, it is not effective for modeling the triaxially-braided composites as it is evident that the behavior of this class of materials strongly depend on architecture details such as the braiding angles, interface cohesive bonding etc. It is quite confirmative that the damage would propagate along the fiber directions when subject to impact, as reported by Roberts (2002). Therefore, the detailed braid architecture should be effectively included within the finite element model to be able to capture the realistic damage patterns. In an
attempt to account for architecture of the fiber tow, Cheng (2006) proposed a “Braided Through the Thickness” approach in which the braided composite was modeled as a series of layered shell elements of laminated composite with the appropriate fiber layup included and the braiding angle change in thickness were accounted for by changing the material orientations over the integration point in thickness direction. To make the method computationally efficient and practical, no actual geometry on individual fiber tows was described in the model. As a matter of fact, it accounted for the architecture of the fiber tow in the material definition input by using the effective stiffness and strength as material properties of the equivalent unidirectional composite, but not geometrically. Still, it was a homogenized model by using the “bottom up” method; the details of the geometry were neglected.

Littell et al. (2008) made two major extensions to the approach developed by Cheng: 1) including the shifting of fiber tows in thickness direction in defining the discretization of the braid as in the real world the shifting actually exists commonly and randomly; 2) getting the equivalent unidirectional properties directly from coupon tests of the braided composite. The second feature is significant in that it assists to reduce the tests needed by Cheng’s approach otherwise and avoid lots of computation by using micromechanics techniques. Recall that, in the Cheng (2006) method, the effective unidirectional composite properties of the materials utilized in a braided composite either had to be measured experimentally or computed by using micromechanics techniques, resulting in an extensive additional experimental program, with tests on either the matrix constituent or the equivalent unidirectional composites.
In the approach developed by Littell, equivalent unidirectional properties were obtained from coupon tests of the braided composite. In this manner, the in-situ properties were directly incorporated into the input for the material model, which could reduce the amount of testing required. In addition, an approach of this type can be useful as a tool to verify more sophisticated analysis approaches. The stiffness and strength values obtained through simulations of quasi-static coupon tests correlated reasonably well to experimentally obtained values. However, when flat panel impact tests were simulated, while the predicted penetration velocity correlated reasonably well with experimentally obtained values, the predicted damage patterns did not correlate well to what was observed experimentally. This can be mainly attributed to the numerical analysis model not reflecting the braiding architecture, thus a more detailed model would be needed to capture more details on both the geometry and material properties.

In both Cheng (2006) and Littell (2008)’s work, a layered shell approach within each shell element to approximate the composite architecture was utilized. As found by Goldberg and Blinzler (2010), the fiber shifting between different layers had very minimal effects in an impact event. In this sense, the significant contribution made by Littell was the possibility of acquiring required material properties including those of the constituent-oriented ones needed in meso-scale modeling. So he really opened the era of using “top-down” approach in modeling braided composites.

Goldberg and Blinzler (2010) further simplified the macro approach by applying a semi-analytical approach to determine the equivalent properties of each element based on the composite layup. In the method developed by Goldberg and Blinzler (2010), the element itself was modeled as a smeared continuum shell element. The philosophy
developed by Littell, et al. (2008) of utilizing a “top-down” approach to determine the required strength properties for the material model was applied once again. However, the methodology was adjusted to reflect the different geometry of the analysis model and to account for the varying geometry within the composite unit cell in a more theoretically consistent manner. Simulations of quasi-static coupon tests and the correlation of the simulations to experimental results, for several representative composites were described. Simulations of a flat panel impact test for a representative braided composite, and the comparison of the simulated penetration velocity and damage patterns to experimentally-obtained results was also reported. As the smeared homogenized method by Goldberg and Blinzler was further simplified in such a way that even fewer through-the-thickness integration points were allowed, making the model more efficient for large structures. Later, Cater et al. (2013) further extended the approach by developing a semi-analytical absorbed matrix model and predicted the moduli and out-of-plane tension-twist displacement in a good manner. Again, despite their efficiency for solving large problems, these modeling approaches shared the same shortcomings: they were not able to simulate the delamination and interlaminar failure that are common in an impact event like engine fan blade-out; nor were they able to predict the damage initiation and progress in individual constituents that would help the designer understand the damage physics.

Whitcomb (1994) developed techniques for performing three dimensional finite element analyses of plain weave composites and used it to study effects of variation of microstructure and properties on composite effective moduli. As TDTBC structure is concerned, to capture the detailed braided architecture and failure mechanism, Binienda and Li (2010) and Li (2010) developed a sophisticated mesoscale three-dimensional
finite-element method to model the $0^\circ/+60^\circ/-60^\circ$ TDTBC, in which a unit cell scheme is used to take into account braiding architecture as well as mechanical behavior of fiber tows, matrix, and fiber tow interface individually. In their work, a progressive damage evolution model with failure criterion for fiber tows and tow interface was applied to predict interlaminar and intralaminar failure. The model was sophisticated enough for addressing small scale static or quasi-static analysis. However, the strain rate effect described by Cheng (2006) and observed by Littell (2008) for the pure matrix resin was not considered; nor was the matrix failure. To simulate the impact events, both the matrix failure and the strain rate dependence need to be addressed. Hence, this became another focus of the current study.

Zhang (2013) further improved the mesoscale FEA analysis model for the $0^\circ/+60^\circ/-60^\circ$ TDTBC by carefully modeling the fiber tows and matrix with eight node elements throughout the material architecture. The mesh generated by this type of elements should possess better quality than that by the six node prism elements employed in Li’s approach (2010), which often presents an overly-stiff model. However, in Zhang’s model, modeling of the bonding interface was avoided by assuming it as “perfect” between the fiber tows and the matrix, and between the fiber tows themselves. This hypothesis may not be always acceptable, especially, when the weak interface were the case when Epoxy 5208 material was used as the matrix. In addition, Zhang’s model needed to use very refined mesh to maintain the shape of the fiber tows with the brick solids. This limited its use for large structures.

From the point view of pure mechanics, both Zhang and Li’s approach were quite sophisticated and detail-oriented. Actually, their models almost presented every detail for
a damage/failure analysis except for that involved in the pure resin matrix. Unfortunately, the use of the 3D continuum shells (Abaqus, 2013) for modeling the fiber tows unexpectedly limited its use only to the scenarios of small deformation where thickness change needed to be small, say, less than 10 \%, to maintain the effectiveness of the plane-stress (i.e. nearly pure bending) assumption used in the shell formulation. To be able to apply the model in the impact simulations where large deformation including the thickness change is involved, the model needs to be modified not only to facilitate the debonding but include the 3D solid formulation as well.

From the above review, there have been two different motivations driving the research. On one hand, researchers were trying to find the simplified approaches to model the textile composites to satisfy the increasing need to solve large scale structures such as fan blade containments and automotive body panels. On the other hand, the complex dynamic response requires more and more detailed solutions from high fidelity material and geometry models to guide the designers in the production. These two parallel needs have made the hybrid method come into being, which makes use the benefit of both the macro and mesoscale modeling approaches by balancing accuracy and the details of modeling with computational efficiency. Barauskas and Abrautiece (2007) developed a combined meso-and macro- mechanical model where thin shells were used to model the individual yarns around the impact zone and roughly meshed membrane elements were used in the surrounding regions. Rao et al. developed two multiscale models in which the shape of the constituent, or the yarn level architecture, varied between the shape of a rectangle and a cross (2007). Gaurav et al. (2008) presented an approach to the multi scale modeling of the impact of woven fabrics using LS-DYNA. This hybrid element
Analysis (HEA) method incorporated the use of different finite elements at a single and multilevel modeling. A yarn level resolution was maintained in the impact zone or local region, while a homogenized resolution used for the far-field or global region. This HEA approach maintained the accuracy of using a fabric model comprised entirely with yarn level resolution utilizing solid elements, but at a fraction of the computational expense. Apparently, the hybrid approaches are fairly new, yet very promising in solving large scale problems; however, when utilized to a complicated composite like the TDTBC, nonphysical premature failure often happens and make the analysis incorrect or fail prematurely. There is much left to be desired when handling the connecting interfaces between the regions modeled in disparate level of details.

With the research review conducted above as the mainframe, now some more details can be added to address each of the approaches in a more focused way as a supplement in the sections that follows.

2.1.2 Micro-scale modeling approaches

Micro-scale modeling approaches are commonly applied to predict the material properties of transversely isotropic unidirectional laminate. In the simulation of textile composites, it can be used to predict the material properties of the individual fiber tows with different fiber volume ratio in different orientated directions. Both the elastic properties and strength properties of individual fiber tows of the composites may be predicted with appropriately applied modeling techniques.

Many of successful methods have been developed for calculating the effective stiffness of a composite unit cell within a laminate. Principally, the micromechanical models are based on a representative volume element (RVE) of the composite. In some
literatures, the RVE was replaced by a repeated volume cell (RVC) or unit cell with the same fundamental concept. A variety of micromechanical models exist for the prediction of the mechanical properties of a unidirectional fiber-reinforced composite, among them was Chamis (1984), Aboudi (1988) and Huang (2001). These models correlated well with experiments in the prediction of the effective longitudinal modulus and strength; however, the predictions for the transverse and shear stiffness and strength properties showed significant discrepancy and deviation. Chamis (1984) specially made the effort in developing an independent program to improve the prediction in transverse properties.

However, the composite strength plays an important role in a composite design and the numerical modeling and simulation as a critical mechanical property. Unfortunately, the prediction of strength using microscale model has been more challenging due to the complexity of composite failure at constituent level. Hinton, Kaddour and Soden (2002 and 2004) and Kaddour, Hinton and Soden (2004) surveyed the predictive capability of a number of the most important strength theories in the applications for polymer composite laminates. Most of these theories such as those by Tsai and Wu (1972), Echaabi et al (1996), Zinoviev et al (1998) etc. were developed phenomenological, in which the composites were treated as homogeneous, anisotropic materials. Limitation of these phenomenological theories lies in that they are not able to predict which constituent initiates the failure and the stress level at failure in the constituent. This is especially true for cases of textile composites, either woven or braided. A possible way to improve is to incorporate some appropriate failure/damage mechanism for the individual filaments and the impregnated pure matrix, and the interface debonding between them to “precisely” obtain the final strength properties of
the constituent. This would usually need the specialized FEA analysis tools involved in the modeling with intensive support from test.

Mechanics of material approach is simple yet the most intuitive method (Daniel and Ishai 2006) for micro-scale modeling, in which most common ones like rule of mixture, full plane stress method were often utilized. For instance, Sun and Chen (1991) used square cell method and full plane stress assumptions to deduct a close form and explicit relationship between laminar strain and stress, which is suitable for implementation in strain increment-based finite element, and the strain rate dependent plastic behavior of the matrix can be included as well. Besides, other mechanics approaches such as the method of cell and the multi-continuum method also found application in the micro-scale modeling. Aboudi (1991) applied the former to set up a direct compliance matrix of composite based on constituents’ properties while Garnich and Hansen (1997) determined the strains and stresses in the individual constituents using the latter.

In addition to the aforementioned analytical methods, finite element methods (Barbero 2007) were used to explicitly model the composite representative volume element (RVE). By applying appropriate periodic boundary conditions, effective stiffness properties can be determined. Additionally, strength and failure phenomena can be also studied by using the point failure criterion. Tabiei (1999 and 2001) and coworkers developed a micromechanical model which is suitable to be implemented into finite element code as nonlinear constitutive model for woven composites. The incremental constitutive model considered nonlinearity of both matrix and yarns. In the model, average strain and perfect interface bonding conditions were assumed. Construction of
subcells with different combination of warp/fill yarns and matrix was used to represent woven fabric architecture. Karkkainen and Sankar (2006) used a laminated model to model plain weave textile composites, with an FEA model employed to replace physical experiment to produce input data for the laminated model.

Ayranci and Carey (2008) made a general literature review on applications of braided composites as well as relevant analytical stiffness predictive models. Ishikawa and Chou (Ishikawa and Chou 1982, Ishikawa and Chou 1983) modified the classical laminated plate theory to model the thermo and stiffness properties of woven fabric composite, as mentioned briefly earlier. Later, this method was further applied by Yang, Ma and Chou for two dimensional braided (1984) and three dimensional braided composites (1985), in which the proposed model was called fiber inclination model or diagonal brick model that treated the RVC as an assemblage of inclined unidirectional laminar. Naik and Shembekar (1992) and Raju and Wang (1994) developed models for woven composites which accounted for yarn undulations. Overall stiffness of woven composite was computed based on iso-strain or iso-stress or a combination of these two assumptions, and then analytically integrating through the volume of RVC. Naik (1994) developed analytical models to predict thermal and mechanical properties of textile composite, including predefined geometrical models for woven and braided composites. Classic laminate theory was applied to average orientated yarn properties. Moreover, computer based program, TEXCAD (1994), was implemented to provide a user friendly design interface. Its capability later expanded into failure analysis of textile composites (Naik 1994). Bednarcyk et al. (2003) discretized weave woven composites into a number of subcells; two step of generalized method of cells (GMC) was utilized then to
determine the in-plane effective properties of the composites. Elastic prediction models have also been found for 3D textile composites by Wang and Wang (1995), Pandy and Hahn (1996), and Cox and Dadkhah (1995).

Based on unit cell, Byun (2000) developed an analytical model to evaluate the geometric influence and predict the engineering constants of TDTBC, in which the unit cell was decomposed into four parts: two braider yarns, one axial yarn and matrix. The modeling concept for the geometric aspect is similar to that used by Li (2010) and Zhang (2013) and this study for a meso level modeling. The difference is that Byun’s approach was used for extracting the TDTBC material properties rather than predicting the response of a structure made of TDTBC. In addition, with the constant strain assumption, Byun’s model utilized the volume averaging technique to obtain effective stiffness of braided composites.

Based on a RVC, Yan and Hoa (2002) obtained a closed form expression for effective stiffness of TDTBC by elastic energy method. The braiding path and profile shape of the braider yarns were both precisely taken into account when integrating the effective stiffness matrix.

Besides, Zebdi et al. (2009) proposed an inverse approach based on laminate theory to back calculate the virtual ply properties, which would be used further to predict elastic properties of any braiding orientation. As the basic assumption in these two approaches was that a woven or braided composite is equivalent to a stack of angle plied laminates, the undulation and strand shear effect cannot be included when applied to textile composites.
As seen from the above review, microscale modeling approaches are quite diversifying and have some limitations in their application for textile composites in accounting for detailed fabric geometry and predicting the strength and transverse and shear stiffness. The performance would be much improved if restricting its application for the prediction in the individual constituent instead of extracting the overall material macro properties of the composite. Despite the existing shortcomings, they provide a fast and convenient way to simulate complicated textile composite as the first level homogenization in the multi-scale modeling frame. In this research, the fiber bundles were modeled as a transversely isotropic unidirectional composite using finite element method, and the microscale modeling approach was employed to evaluate the effective moduli and strengths using a representative volume element (RVE) for the individual fiber tows in conjunction with the plastic approach for damage/failure prediction.

2.1.3 Meso-scale modeling approaches

Because of the large element number usually involved in the mesoscale modeling, literature reported that only a small portion of the real structure like a specimen for a tensile test (Li 2012, Zhang 2013), were extracted to be included in the simulation model with carefully defined boundary conditions. As far as only a few unit cells were included in the model like the tensile specimen simulation conducted by Zhang (2013), periodical boundary conditions or other forms of prescribed displacement (Li 2010) needed to be carefully imposed on the model to approach the reality.

This kind of analysis for mesoscale models usually result in an un-uniform stress distribution over the Repeating Volume Cell (RVC), or unit cell, which is expected and different from the analysis results obtained by most of micro scale approaches.
Furthermore, by implementing specific failure behavior into distinguished constituents, damage initiation and development are able to be investigated. To achieve this, more elaborate numerical solutions are preferable due to the limited size of the model that can be included in the simulation.

Lomov and coworkers in Composite Material Group developed software, WiseTex (Lomov et al. 2001), for geometry modeling of internal structure of textile reinforcement, such as 2D/3D woven, two- and three- axial braided and knitted etc., which can transfer data into the format of general FE codes. With this tool, Fiber yarn cross-sections and undulations could be explicitly modeled and controlled as required. The material properties used in the mesoscale finite element model were calculated using microscale homogenization discussed in last section. Damage modeling was included by adopting damage mechanics approached to be discussed soon in Section 2.2. The model was verified to be able to successfully predict local stress and strain field by comparison with full field measurement (Lomov 2005). A similar tool, TexGen, was developed by (Sherburn 2007). It is a Python based software that combines the geometry building and volume meshing algorithms together with an efficient user interface. Both these tools could handle orthogonal textile structures, like woven, but they cannot handle intersections like the flat yarns in non-orthogonal structures such as the braided ones, especially for the case of high global fiber volume fraction.

Miravete (2001) carried out meso-mechanical FE analysis using solid elements to mesh fiber yarns and matrix, failure was taken into account by using Hashin failure criteria. Three axial braided composites with two bias angles, 20° and 30°, were studied in tension and shear tests and compared with proposed analytical model which is based
on principle of superposition. Zeng et al. (2004) used a brick element model to study the local stress distribution of 3D braided composites and damaged mechanical properties. Song and Wass et al. (2008) predicted compression strength of 2D braided textile composites based on meso-mechanical finite element unit cell (single and multi) model. Micromechanics homogenization model was used for fiber tows, boundary conditions and imperfection imposing was discussed in eigenmode and response analysis. Fang (2009) analyzed damage development of 3D four directional braided composites based on mesoscale finite element model by implementing anisotropic damage model. Other attempts of developing geometry modeling technique have been made such as Sun (2001).

The finite element mesh with correct geometry is essential for applying a meso-mechanical model, while implementing damage mechanics is as critical to capture the damage involved. The reported mesoscale models could predict overall response to certain extent of success; these models did not usually study damage mechanisms such as the debonding and delamination, however. Li and Binienda (2010) developed a sophisticated mesoscale model that was able to address the damage commonly observed between various oriented fiber tows. A cohesive layer of zero thickness between the fiber tows was modeled with complex changed geometry and detailed damage mechanism. The model was capable for predicting damage involved in quasi-static response of small scale structures. However, the thin cohesive layer modeled between the matrix and fiber tows, which was easy to be distorted under finite deformation, often led to premature failure of the analysis. In addition, the large number of elements needed in a unit cell always resulted in an impractical large model even for some lab specimen or test
coupons, let alone real structures like an automotive hood panel or engine containment. The current work strives to improve these areas with developing efficient mesoscale model for dynamic simulation and co-existing multiscale modeling approaches.

2.1.4 Macro-scale modeling approaches

The macro-scale modeling approach has obvious limitations in application. It is not able to provide the detailed fabric geometry; nor is it able to give detailed stress and strain solution among stacking plies and components with the iso-stress assumption. Besides, the approach is not effective in prediction of localized damage or failure.

Despite of the aforementioned defects, the approach has great practical value in that it is able to deliver an efficient solution in an approximate way for the simulation of complicated textile composites. This is especially true when the dynamic response of an exceedingly large structure is to be analyzed. The determination of the material parameters for elastic properties and failures is essential and needs experiment support, which could be tedious and expensive. For this reason, majority of the composite macro scale models were applied on unidirectional composite lamina with relatively less material parameters. Investigations on the macro scale modeling of textile composites were relative rare, due to the difficulty to include complex architecture such as the undulation and intercrossing of fiber tows in the model.

As early as 2005, Zeng used the composite material damage model available in LS-DYNA, i.e., the MAT59, as macroscopic model to simulate the response of 3D braided composite tube under compression. It is necessary to mention that the material parameters used for MAT59 were obtained by the meso scale model already reviewed in last section (Zeng 2004). Iannucci (2006) predicted structure failure under impact loading
by applying the damage modeling methodology on woven carbon composite with consideration of the strain rate dependence in the constitutive equation. The use of regular shell elements enabled each ply in the laminate represented through the thickness integration points. Later, Cheng (2006) extended this concept to the TDTBC structures by establishing and utilizing the well-known “Braiding through the thickness” technique as discussed already in an earlier section. In Iannucci model, three damage variables were considered: fiber fracture in the local warp bundles, fiber fracture in the local weft bundles and fiber-matrix deterioration due to in-plane shear. Good agreement was reached between predicted result of the model and experiment measurement for the impact test. Xiao et al. (2007) developed a macro material model using LS-DYNA (2007) to simulate the onset and evolution of damage in woven composite in 3D stress fields. This model was capable of modeling failure modes including tensile, compression, shear as well as delamination without the need of modeling the interface. Accordingly, a methodology of coupling Quasi-static Punch Shear Test and mode-based simulations was developed in the research to determine four damage parameters used in the model.

Xie and coworkers (2006) treated triaxially braided composite \((0^\circ/\pm45^\circ)\) as elastic plastic orthotropic homogenized material. The in-plane effective mechanical properties were measured by ASTM tests, while the plastic behavior was characterized by static off-axis compression tests. In the same work, cohesive zone model was used to simulate mode I and mix mode fracture. As touched earlier, Littell (2008) also made use of the conventional shell elements with through thickness integration point scheme to model the triaxially braided composites. The key contribution from his work was to employ the
“top-down” approach to characterize material parameters as input to the damage material model.

In summary, regardless of its limitation, the macroscale modeling approach can be an effective tool with its homogenization feature to be used to seek for the solution of large scale structure as the first level overall solution in multi-scale modeling. Detailed local behavior can then be predicted with the efficient use of a local or partial mesomechanical model. More discussion on this combined application will be addressed in next section.

2.1.5 Multiscale modeling

Certain investigations were also conducted in attempt to build up systematic multiscale analysis framework for textile composites. A hybrid element analysis (HEA) approach proposed by Nilakantan (Nilakantan 2008) was a typical effort in this regard, which enables the numerical simulation of multi layered woven fabric with large domains that would be otherwise very difficult to solve with detailed modeling for the entire structure due to the impractical computational requirement. However, the approach requires a strict smooth transition with matched acoustic impedance at the numerical interface connecting the portion of the structure modeled with costly element type and refined mesh and that modeled with coarse mesh of a different (usually simpler) type of element. This requirement may be achieved without much effort for traditional laminated composites and woven composites. However, for braided composite being dealt with in the current study, it is very hard to be satisfied and thus making the method less practical because of the complexity of the geometry and architecture of the constituents. Usually, a special design with great care of the numerical connection is needed and often hard to be
successful. Oftentimes, the interface connecting the regions modeled with disparate types of elements would lead to nonphysical failure of the material, making the simulation results unreliable or fail pre-maturely.

Considering the extremely complex architecture and material behavior in braided composites, Nie and Binienda (2012) proposed a combined Meso-Macroscale (CMM) finite element multiscale analysis approach that unites the currently existing macroscale and mesoscale approaches for modeling the braided composite materials to make full use of the advantage of each approach, with the mesoscale model using the embedded unit cell with damage and delamination to describe the details of local deformation and possible material failure, and the macro-scale model capturing the global response feature of the entire structure. Later on, the method was further improved and verified by using the dynamic submodeling technique (Nie et al. 2012) for better efficiency and flexibility in implementation. In the current research, the CMM approach was systematically enhanced by extending the scope and meaning of the multiscale from the mere meso-macroscopic to meso-mesoscopic or macro-macroscopic combination with different modeling details as well as will be detailed in chapter 5. The method is herewith still called CMM for consistency, but termed as the combined multiscale modeling approach (still known as CMM in abbreviation).

In a parallel research direction, Fish and Yu (2001) reported a theory and computational frame work on multi-scale (i.e. micro-, meso- and macro-scale) damage modeling for textile composites to extend a two scale (micro-macro) non-local damage theory proposed earlier (Fish and Yu 2001) in attempt to account for evolution of damage in heterogeneous microphases. A multiscale non-local damage theory for brittle
composite materials was developed based on the triple-scale asymptotic expansions of
damage and displacement fields. The closed-form expressions relating microscale,
mesoscale and overall strains and damage were derived in the work. In addition, the
damage evolution was stated on the smallest scale of interest and the non-locality was
taken into account to alleviate the spurious mesh dependence by introducing the weighted
phase average fields over the micro- and mesophases.

2.2 Modeling damage and failure of composites

As one of the principal aspects in the current research, a comprehensive review on
the approaches currently available for predicting damage in composites is given in this
section. Incorporation of failure and damage modeling into composite constitutive
models is one of the advanced issues and most challenging areas in nowadays’ research.
Remarkable advances have been reported in recent years toward a better understanding of
the failure mechanism in composite mechanics and dynamics. Contrary to the traditional
metallic materials, failure mechanism of composite material is considered to be
composed of a series of cracking or fracture at different scales and characteristics.
Roughly, the damage modeling approaches are divided into following four areas: 1)
failure criteria approach, 2) damage mechanics approach, 3) fracture mechanics based
approach, and 4) plasticity approach. The details follow for each of these areas.

2.2.1 Failure criteria approach

Failure theory of anisotropic composite material was primarily used as an
indication of failure initiation for each observed failure mode in the progressive failure
methodology. Failure criteria theories for composites were principally established by
extending the existing failure theories for isotropic material to account for the anisotropic
characteristics of composites in the strength. The damage of composite can be classified into several basic types, such as longitudinal fiber tensile breakage, compressive buckling or kinking, transverse matrix cracking and delamination, etc., based on either equivalent stress or strain. Failure surface is defined by a set of polynomial equations in terms of two or three dimensional stress and strain quantities with adjustable parameters to be determined from test or numerical experiment.

There is huge number of reports on theoretical forms for these failure criteria. The most recent literature review was conducted by Christensen (2001), Paris (2001) and Hinton et al (2004). The most widely applied theories on lamina level are herewith listed: Tsai-Hill theory (1965), Tsai-Wu theory (1971), Hashin and Rotem failure criteria (1973), Hashin 3D failure criteria (1980), Chang-Chang criteria (1987), Christensen failure criteria (1997), Puck’s criteria (1998), while a comprehensive evaluation in this regards is beyond the scope of this literature review.

Well-developed failure criteria on lamina level fall short in predicting failure of textile composites, due to the severe anisotropy in individual ply of this type of composites. Only textile composites with minimal fiber undulations and low 3D-reinforcement density may be approximated by failure criteria built on lamina level such as Puck and Tsai-Wu theory. Appropriate failure criteria suitable for textile composites line weaves and braids are yet far from maturely developed. Juhasz et al. (2001) developed a failure criterion for orthogonal 3D fiber reinforced plastics e.g. non-crimp fabrics. Xiao, et al. (2007) generalized Hashin failure criteria of a lamina to characterize the damage for a plain contexture layer, in which the fill and warp fiber interaction were
considered. Six failure modes were considered with 15 strength parameters, including delamination mode. In addition, the strain rate effect was considered as well.

As aforementioned, the failure criteria just mark the onset of damage in composite. According to progressive damage theory, no material damage or removal will happen if the post-failure algorithm is not defined in the constitutive model. Post failure behavior can be assumed to be either ideally brittle manner, that is, the stiffness is reduced to zero abruptly, or treated numerically as progressive stiffness degradation over certain time steps. The degrading the stiffness over a range of increased strain, representing softening and fracture energy dissipation, is more physically reasonable and numerically stable and thus suggested. As strain hardening has been successfully modeled by plasticity theory, strain softening behavior was more rigorously dealt with through models based on the concept in continuum damage mechanics or fracture mechanics. The relevant discussion is followed next continuously.

2.2.2 Damage mechanics approach

Continuum damage mechanics has found more and more application in depicting post failure behavior of anisotropic composite material after failure initiation within the frame of the progressive failure methodology. The central concept of this approach is to replace a micro crack by a damage zone over a finite volume. Usually damage variables in scalar or tensor form are introduced, which determine full range of degeneration in a composite material, from damage initiation to the complete damage. Then, mathematical kinematic constitutive equations for the damaged composite, leading to a strain softening behavior, are proposed in terms of the damage variables.
A literature review was made by Krajcinovic (2000) to highlight the accomplishments achieved on damage mechanics and research trends and needs for further development. The earliest works included that done by Kachanov (1958). It was hypothesized that the loss of stiffness attributed to micro cracks can be measured by a macroscopic damage parameter to theoretically bridge the fracture mechanics and the damage mechanics. In the work done by Krajcinovic (1984), inhomogeneous micro cracking behavior was successfully averaged over a volume of material and described by a macroscopic plastic strain tensor. From now, the review that follows will be focused on the usage aspect of the approach in the composites.

One of the earliest applications of damage mechanics to composite was made by Talreja (1985), in which two damage variables were proposed to a lamina model with each representing a major material orientation. The model was used to predict angle ply laminate stiffness reduction, and showed good agreement with experimental results. Another early application was done by Frantziskonis (1988), who proposed a single damage scalar model in which the damage threshold and growth was taken the form of polynomial of the strains. With tensile and compressive tests to characterize the model parameters, the numerical prediction based on the proposed model showed good agreement with laminate tests, too. Engblom and Yang (1995) developed a simple model based on damage mechanics to predict the effect of intralaminar damage with only the matrix failure considered. In addition, Randles and Nemes (1992) developed a composite model with a delamination damage variable incorporated to particularly address the delamination, which was found to be another critical failure mode.
Matzenmiller et al. (1995) developed a rigorous anisotropic damage constitutive model to describe elastic-brittle behavior of fiber reinforced composite, in which four damage variables corresponding to the four failure modes were introduced with regard to the 2D Hashin failure criteria, and with the plane stress assumption, a simple relationship between the effective stress and the nominal stress was built by a rank-four damage operator. The model was a strain-controlled model; therefore, it is well suited for implementation in the standard finite element codes, such as the anisotropic damage model for fiber reinforcement composite implemented in ABAQUS.

Williams and Vaziri (2001) developed a progressive damage model to improve prediction of impact damage on composite structures, in which the Chang-Chang failure model was combined with Matzenmiller’s damage model and programmed into LS-DYNA as a user material. Additionally, efforts have been exerted on creating damage model on composite laminate, such as the work done by Maimi, Camanho, Mayugo and Davila (2007).

Hufner and Accorsi (2009) adopted the progressive failure approach and developed a phenomenal constitutive model suitable for woven polymer-based composites, which was formulated to characterize the mechanical response for fully non-linear, rate dependent, anisotropic behavior under dynamic loading. It is necessary to mention that a mechanistic fiber failure theory was implemented, with a degradation model used to describe the successively reduced stiffness after the onset of failure.

Damage mechanics was widely used to predict post failure behavior in different composite failure modes, such as matrix cracking, fiber fracture, and delamination in conjunction with failure criteria theories. The difference between all these models mainly
relies on the damage modes and numerical degradation scheme employed. For further information, The World Wide Failure Exercise (Hinton et al. 2002 and 2004) provided an extensive comparison between various approaches.

In this study, the damage mechanics approach was employed to model all the damage mechanisms but the interface debonding involved in the braided composites such as the matrix progressive damage, fiber tows damage, while the debonding of the interface that bonds all constituents together was modeled by a Fracture Mechanics approach that follows next as a surface interaction behavior introduced to general contact.

2.2.3 Fracture mechanics approach

Fracture mechanics approach is oftentimes used to evaluate the growth and evolution of existing cracks for the cases in which the cracks and their locations and initial dimensions are identified and defined. Considering the feature that damage in brittle composite is composed of a series of cracks, the scale effect associated with the length of cracks subject to the same stress field needs to be dealt with. Since the failure criteria theories cannot model these cases correctly, the use of fracture mechanics becomes essential even though it cannot be used to predict the initiation of fractures.

The basics of fracture mechanics methodologies include the linear elastic fracture mechanics (LEFM) originally proposed by Griffith (1921), the elastoplastic fracture mechanics further expanded by Irwin (1956), and path-independent integration of J-integrals developed later by Rice (1968). A recent comprehensive literature reviews on the application of fracture mechanics approach was performed by Cotterell (2002).

Later on, lots of notable research work fell into the class of application of the fracture mechanics in modeling the fracture process zone. Direct applications of elastic
and plastic fracture mechanics has been successful, and well incorporated into many commercial finite element codes like Abaqus. However, only small crack growth can be modeled because the continuous predictions of crack propagation are not only expensive but usually mesh refinement and re-analysis are needed as well.

Conventionally, Fracture mechanics merely evaluates the propagation possibility of the existing cracks by giving the index factor on whether the crack would progress, such as energy release rate or stress intensity factors, instead of physically modeling the crack advancing. To be able to actually predict crack propagation, new modeling techniques have been developed based on the fracture mechanics concept, such as virtual crack closure technique (VCCT) and cohesive processing zone model (CPZ) or cohesive zone model (CZM). Compared with CPZ or CZM approach, the only drawback of VCCT is the requirement of assuming existing crack(s) in the material; that is, it cannot be used to model crack initiation. Because of the requirement of intensive numerical calculation, the application of VCCT for prediction in crack propagation has been usually incorporated into commercial finite element packages, say, Abaqus (DS SIMULIA Corp, 2013). Cornell Fracture Group first implemented VCCT in their specialized codes in conjunction with general finite element codes (Singh et al. 1998).

VCCT was traditionally used just for computing energy release rate or stress intensity factor as the traditional Fracture Mechanics does, based on the results of continuum finite element analysis for cases in which mixed mode fracture criterion is considered (Rybichi and Kaminen 1977), (Raju 1987), (Buchholz et al. 1988) and (Krueger 2002). After embedded into the commercial FEA code Abaqus from version 6.5, VCCT technology was evolved as part of the Composite Affordability Initiative
(CAI) in 2004 at Boeing Commercial Aircraft Group, being used for predicting crack propagation in large integrated structures, and evaluating interlaminar damage tolerance requirement. This new capability for fracture analysis is available for general public use by all users without the need of a special add-on license (DS Simulia, 2014). The embedded VCCT assumes that the crack interface and intermediate crack-tip location with no need of re-meshing. Additionally, the mesh-sensitivity was minimized by careful formulation combined with cutting-edge FEA technology.

To deal with the crack initiation and progress in the simulation of fracture, CPZ is more useful. Dugdale (1960) and Barenblatt (1962) were the first to apply the concept of a cohesive stress zone to fracture modeling. Many extensions have been developed since then by Hillerborg (1976) and others. To address the strain softening issue, the fracture energy to open a unit area of crack was assumed a material property by Hillerborg. The softening response after damage initiation was characterized by a stress-displacement response rather than a stress-strain response. This required the introduction of the characteristic length associated with a material point (Lawn 1993), which was later discussed in detail by Taylor (2006).

Needleman (1987) made another remarkable extension afterwards, which revealed that cohesive elements were particularly attractive when interface strengths were relatively weak compared with the adjoining materials, for example, composite laminates and parts bonded with adhesives. This is also the case for the composites constituents in textile materials, or the stacked plies in a laminate as the ply-ply interface is relatively weak compared with neighboring fiber bundle materials. Being able to automatically allocate the potential crack propagation path without pre-knowledge of crack initiation
location, cohesive zone model is suitable and thus widely used to simulate debonding or delamination in the laminates, textile (woven or braided) composites (Li 2010, Nie and Binienda 2012).

Reeder (1992) characterized a mixed mode damage model in consideration of that it would be more comprehensive and accurate to incorporate the mixed modes actually involved in the delamination phenomena, such as opening, tearing and shearing mode. The interaction between the modes could be defined by use of non-dimensional displacement (Tvergaard and Hutchinson 1993), traction components (Xu and Needleman 1994), damage surface defined by relative displacement (Benzegagh and Kenane 1996) etc. Many researchers have proposed various other cohesive constitutive laws; among them were: simple bilinear by Reedy (1997) and Mi (1998); mathematically continuous exponential by Needleman (1987), and Xu and Needleman (1994); trapezoidal by Tvergaard and Hutchinson (1993); perfectly plastic by Cui and Wisnom (1993). Li and coworkers (2006) proposed a different cohesive law for Mode I fracture by introducing cohesive strength and characteristic strength simultaneously. Alfano (2006) gave an evaluation on how the cohesive laws affect application of the cohesive zone model.

Additionally, various types of element technology were developed to apply cohesive zone model (Moura et al. 1997 and 2000; Reedy et al. 1997; Chen et al. 1999, Mi 1998 and Petrossian 1998). Specially, the 3D eight nodes zero thickness cohesive element with mixed mode damage law was used by Camanho and Davila (2001) to simulate the stiffener-flange debonding. Johnson and Holzapfel (2006) predicted the impact damage using shell elements with cohesive element as interface, and obtained
good agreement with test for delamination and penetration. Davila, Camanho and Turon (2007) also reported their work on applying cohesive element between stacked, non-coincident layers of shell elements, and the result showed that shell element model can retain many of the predictive attributes that 3D solid models would possess.

Some researchers (Bazant and Oh 1983, Bazant and Jirasek 2002) found that the computed energy dissipation decreased with the reduction in element size employed in the analysis model when strain softening was involved in the cohesive law, which was against the fact that a unique solution should be independent of the mesh refinement. Characteristic length of finite element was introduced then to eliminate this non-objective effect, like the work done by Camanho et al. (2007), in which simulation on failure of laminate plates with holes was modeled using cohesive zone model. Alfano and Crisfield (2001) incorporated the damage mechanics into cohesive constitutive law to reflect the material softening, and compared the predicted results of CPZ and VCCT. In the same work, a parametric sensitivity study was conducted on the cohesive strength and stiffness with the mesh density change. It was concluded that the result was strongly dependent on the cohesive strength. Since then, many investigations have been done on the mesh sensitivity. For instance, Turon et al. (2007) discussed mesh size effects, and developed a methodology to determine the constitutive parameters for CZM using a close form expression for penalty stiffness of the cohesive; constitutive law built with this proposed methodology allowed for a coarser finite element mesh, making the large scale analysis feasible. Zhou and Molinari (2004) also discussed the mesh sensitivity for the case of dynamic crack propagation. Besides, Chandra et al. (2002) and Borst (2003) addressed some numerical issues in implementation of cohesive element models.
Modeling the delamination of laminated composite as well as textile composites has been widely used. Zou et al. (2003) discussed about the parameter determination of cohesive model for the simulation of delamination growth of laminated beams. Allix and Blanchard (2006) included cohesive zone in a meso scale laminate model to predict the delamination of laminated plates containing circular holes. A similar method was also employed in the current study for modeling the interface between the stacked plies of the TBTDC and that between the constituents; instead of the cohesive elements, a surface based constraint with the contact surface property was used though. Similarly, Guedes et al. (2008) proposed a 3D finite element of sixteen layer laminates including the cohesive element as interface between plies. The result correlated with the test result on high strain rate compression response of laminates. In addition, mesh density was studied in their research and no significant mesh sensitivity effect was found. Okabe and coworkers (2008) used the so-called embedded progress zone (EPZ) for the transverse crack and delamination in cross ply laminates, and studied interaction between these two types of failure modes. In the application for braided composites, Li 2010 used the cohesive element with zero thickness to model the delamination between TDTBC constituents modeled with mesoscale approach. And Nie and Binienda (2012) further improved the bonding interface modeling by using the surface based cohesive constraint as a replacement in conjunction with general contact capability with the mix-modes cohesive constitutive law proposed by Benzegagh and Kenane (1996).

The concept of cohesive zone presented a new method to explicitly model interface behavior. In the current work, it was again adopted in conjunction with the surface based contact constraints (DS Simulia, 2013) to model the bonding interface
response between the fiber tows. This was distinguished from the other published
literature in that the thin cohesive elements was avoided which would make the meshing
and modeling work arduous when the structure is complicated. Therefore, effectively
dealing with this issue was one of the goals in the current work.

2.2.4 The CZone approach for crush

Crushable structures that absorb energy during impact are used in transportation
tools such as automobiles, aircraft, and trains to help protect occupants and cargo from
shock and injury during a crash. Composite materials hold great potential for providing
increased energy absorbing in lower-weight crushable structures as compared to heavier
metallic designs. The CZone technology developed by Engenuity (2010) provided an
alternative and complementary approach to academic efforts to develop a constitutive
approach to composite crush. Calibrated by coupon tests of the crush response a material,
CZone for Abaqus provides direct implementation of crush-based element failure in
defined “crush zones” at the leading edges of the structure. Usually, the crush strength,
or the stress at which crush happens, is determined from the component’s area and impact
force via component coupon impact testing.

Accurately simulating the crush characteristics of composite structures is
challenging. Generally, the crushing response cannot be described by conventional failure
theories because it is difficult to accurately represent the failure mechanisms involved,
such as localized buckling, delamination, matrix cracking and fiber breakage. However,
CZone is only appropriate for materials that crush. Such materials provide an ongoing
resistive force at the crush front, known as the crush zone. Therefore, not all composites
can be accurately modeled using CZone. Until the micro-mechanics of the crushing
failure mechanisms are fully understood, it is necessary to test the crush properties of candidate materials (DS Simulia, 2008).

2.2.5 Plasticity approach

The plasticity approach adopts a phenomenological constitutive equation developed on the basis of the concept of plasticity such as yield function, flow rules and hardening parameters. This approach was applied specifically to thermoplastic composite, like Boron/AL or G/PEEK, which possess notable nonlinear and rate dependence behavior. Due to this, very limited literature has been found in this area.

One representative works found was that done by Sun (1989 and 2001), in which the strain rate dependency in conjunction with other nonlinear behavior were characterized for both laminate and woven composites.

Other notable works was a continuum elastic-plastic model developed by Odegard et al. (2001) for woven-fabric and polymer-matrix composite materials. The model was validated under bi-axial stress loading conditions. Later, Ryon et al. (2007) analyzed woven composite made of glass fabrics and epoxy resin with the utilization of the elasto-viscoplastic theory, focusing on the nonlinear and rate dependent deformation responses of the material. In their work, the hardening parameters were determined by uni-axial tension and compression tests. The model was further used to verify three-point bending test for proposed woven composite materials.

Despite the limited literature, this approach was in fact employed in the current research to determine the damage properties for the fiber tows when the numerical techniques like finite element methods (Barbero 2008, Xia 2003) was used to evaluate the effective composite moduli and strengths using a representative volume element (RVE),
as will be discussed in Chapter 4. Because of the plasticity existing in the matrix, the stress-strain curves of shear and transverse loading conditions can be nonlinear. Therefore, both fiber and matrix in the tows are modeled as an elastic perfect plastic material for simplification. The initial linear portion was used to determine the elastic modulus for the particular direction, and the computed yield stress was set to be the corresponding strength considering the linear elastic nature of unidirectional composites (Zhang 2013).

2.3 Review of Abaqus/Explicit capability for dynamics and composites modeling

As a general purposed finite element analysis code, Abaqus (DS Simulia, 2013) has been widely used in all industries and academics for solving all types engineering problems, including automotive, aerospace, defense and energy and life sciences. An all-aspects review would not be possible or necessary. Here only the principle functionality and features of two fields of the code closely concerning to this research is given as Abaqus has been used heavily in this study as the primary FEA tool.

2.3.1 Abaqus/Explicit solver review in modeling impact analysis

Abaqus (DS Simulia, 2014) provides two relatively independent complementary solvers: Abaqus/Standard and Abaqus/Explicit. Abaqus/Explicit is suitable for the impact dynamics analysis of TDTBC as it is facilitated to analyze large-scale nonlinear, high-speed dynamics or quasi-static analyses with highly discontinuous post buckling, material failure and structure collapse due to extreme deformations.

The deciding factor of choosing Abaqus/Explicit over Abaqus/Standard is that Abaqus/Explicit uses a finite-strain, large-displacement, large-rotation formulation by default whereas it may not be possible to obtain an efficient solution with
Abaqus/Standard as there are significant discontinuities in the solution of the TDTBC structure when subjected to high speed impact. The sources of discontinuity in such a solution come from 1) the impact, 2) buckling or local wrinkling of the fiber tows, 3) material degradation or failure, such as matrix cracking, ply delamination etc.

2.3.2 Modeling composites with Abaqus

Both Abaqus/Explicit as well as Abaqus/standard solvers have variety of functionalities and leading technology for basic and advanced modeling and analyzing composite materials and structures for many applications. CAE engineers and analysts in Aerospace, Automotive, Shipbuilding, Defense, and Energy are using Abaqus to model and study the linear and nonlinear behavior of composites to improve their understanding of composite behavior and their designs. Additionally, Abaqus provides a comprehensive pre and post processing tool called Abaqus/CAE, to offer a Complete Abaqus Environment, which is especially efficient and powerful for modeling and visualizing the analysis result for composites, say, defining and viewing the composite layup and the varying orientations.

Depending on the purpose of the analysis, different modeling techniques equipped in Abaqus for composites can be used as outlined below:

1) Microscopic modeling: The matrix and reinforcement material are both modeled separately as deformable continua.

2) Macroscopic modeling: The composite is modeled as a single orthotropic material or a single fully anisotropic material.

3) Mixed modeling: The composite is modeled by a number of discrete, macroscopically modeled reinforced layers.
4) Discrete reinforcement modeling: Reinforcement modeled with discrete elements or other modeling tools (e.g., rebar).

5) Submodeling: Useful for studying stress concentrations around the tips of reinforcing fibers for both static and dynamic analysis. This technique will be intensively used in this research and more detailed review will be provided in section 5.1.2.

In addition to the modeling of matrix and reinforcement, progressive damage and failure of the materials and their interfaces can be modeled as well. The capability for modeling progressive damage and failure, that is, the prediction of failure modes for both fiber and matrix materials includes: 1) Hashin Failure Criteria limited to plane-stress formula, so it is not available for 3D solid elements; 2) user material subroutine UMAT (Abaqus/Standard) and VUMAT (Abaqus/Explicit) to give the users opportunity to incorporate material models into Abaqus that are not available as standard embedded function; 3) Delamination—Separation of adhesively bonded sections of laminated composites; 4) Virtual Crack Closure Technique (VCCT); 5) Cohesive Elements and Cohesive Contact to define the bonding and debonding between lamina. 6) CZone, an Add-on capability to Abaqus/Explicit that allows a user to include material crush behavior in FEA simulations of composite structures subjected to impact (DS Simulia, 2008).

2.4 Current Research

The primary objective of this research was to develop a practical methodology for modeling and analyzing dynamic response of large-scale TDTBC systems and structures. The reliability of impact simulations for aircraft components made with triaxial braided
carbon fiber composites is currently limited by inadequate material models to appropriately utilize the material property data obtained from experiments and lack of validated efficient computer models accounting for the detailed local dynamic behaviors in large structures under impact. Improvements to existing methods are needed to 1) account for dynamic features and local deformation, material damage and delamination; 2) make the simulation reasonably fast thus suitable for large structures.

Two dimensionally triaxially braided composites (TDTBC) are composed of three sets of fiber tows with different orientations intertwined to form a single $0^\circ/\pm\theta^\circ$ layer. Bias tows alternatively undulate over and under each other while $0^\circ$ tows are straight and define the axial direction of the composite (so also referred to as axial tows). In particular, the fiber architecture of braided composites with a bias tow orientation angle of $\pm 60^\circ$ is often utilized because of its quasi-isotropic nature and excellent capacity of resisting crack initiation and propagation as well as formation of delamination when subjected to impact (Roberts et al. 2003). Therefore, through this research we will consistently use this particular fiber tow architecture formed by $0^\circ/\pm 60^\circ$ braiding orientation angles. Particularly two typical TDTBC material systems were primarily dealt with: the T700S/E862 and T700S/PR520.

Previous researches had not identified a solution for accounting for the strain rate effect and failure of the composite constituent, the pure polymer resin, in FEA modeling. In current research, a new 3D meso-scale finite element model was developed and employed to capture the deformation and damage / failure as well as strain-rate sensitivity of the resin matrix that had been observed long in experiment. A unit constituent geometry based approach, as a replacement of the so-called existing unit cell
scheme, was used to define the details of the braiding architecture as well as the material properties of the fiber tows, the matrix and the tow-tow and tow-matrix interface. This new approach allows a much better improved quality of mesh and also the flexibility in defining the interaction and bonding interfaces between the tows themselves and also between the fiber-tows and the matrix. Damage initiation and progress mechanism was considered in conjunction with elastic-plastic behavior of the matrix under impact. The interface and interaction between the fiber-tows and between the fiber tows and the matrix is improved by using a cohesive surface approach with zero thickness. A mixed mode cohesive law was adopted based on the fracture mechanics principals to evaluate the crack initiation and predict crack propagation. The commercial transient dynamic finite element code Abaqus/Explicit was used to implement the methodology and conduct the simulations and an embedded continuum damage mechanics model was used as the material constitutive model for the fiber-tows and coded as a user material model. The constitutive model requires stiffness and strength properties of an equivalent unidirectional composite.

Also, very little about effective approach to utilizing hybrid multi-scale modeling in the same analysis had been reported in literature, especially for the TDTBC structures. Hence, this research was focused on effective combination of the two major classes of modeling approaches, mesoscale and macroscale modeling, to form a novel hybrid method, named Combined Meso-macroscale Modeling (CMM) approach, (or Combined Multiscale Modeling approach in the cases only one type of approach is used with different modeling detail), for simulating triaxially braided composite structures and systems. With the flexibility to use either the meso-mechanical or the macro-mechanical,
or a combination of the modeling approaches, the CMM approach is particularly efficient for the situations where the prediction in the locations of critical local behaviors such as damages is difficult. The Submodeling method was employed as the core technique to establish the CMM method for efficiently solving the large-scale problem with details of the localized deformation and behavior such as damages and delamination by extending the Saint-Venant's principle to the scope of dynamics with nonlinearity and severe discontinuity like changing contact and damage caused by perforation.

The widely used unit cell scheme was found too restrictive in this research and needed revising to be useful for the new mesoscale modeling approach. The large number of irregular, tiny artificial geometric divisions caused by the use of the unit cells not only makes the meshing work arduous but deteriorates the element quality as well. To overcome all these issues and defects generated by the unit cell divisions, the new concept of the unit constituent cell was proposed to eliminate the artificial borders otherwise generated by the unit cell, thus greatly reduced the modeling work needed. Another unique advantage of the unit constituent cell lies in that it is not necessary to adopt conforming meshes between the fiber tows and matrix any more. Instead, independent meshes were allowable between the fiber tows and the matrix. Based on the newly developed unit constituent geometry scheme, a new three dimensional finite element meso-mechanical modeling methodology was developed and verified suitable for dynamic analysis under high speed impact. The unit cell scheme was replaced by the unit constituent geometry scheme to take into account of internal braiding architectures as well as mechanical properties of the three primary phases: the fiber-tows, the matrix and the tow interfaces. Micromechanics approaches combined with RVE were applied to
produce material parameters for the 3-D material mesoscale model. Failure initiation and
progressive degradation model had been implemented into matrix by using of Hashin
failure criterion and a damage evolution law. This mesoscale modeling technique was
used to examine and predict the failure and delamination responses observed in impact or
other severely deformed local regions, while the macro model was used to model the
global or far-field region from the critical regions.

The key contribution of the current research has two primary fields as stated below:

1) A brand new meso-mechanical analysis approach was set forth first time and
established based on unit constituent geometry rather than the so-far widely
used unit cell approach, which was characterized by the small number of
elements within the unit cell and the considerably increased stable time limit
for the explicit finite element analysis code. As a result, the size of the same
analysis models were reduced to less than one-tenth of that would be required
by a regular currently-existing meso-mechanical model, and furthermore the
stable time limit increased by over 2~3 times. These two main gains made the
approach attractive for the simulation of large-scale dynamic events. It makes
the large models small and solvable in short time.

2) A combined meso-macro, or combined multiscale, modeling analysis
approach, entitled CMM approach, was developed with the full use of the
advantage of each of the mesomechanical and macromechanical modeling
approach, making the use of the approach to large scale dynamic problems
more efficient and flexible.
In summary, modeling and analysis approach developed in this research made the
dynamic meso-level analysis of TDTBC material and components much more efficient
and the novel CMM analysis approach made the numerical analysis of large systems
practical.
CHAPTER III

INVESTIGATIONS AND CHARACTERIZATION OF TRIAXIAL BRAIDED COMPOSITE MATERIAL PROPERTIES AND RESPONSE

This chapter gives the main concepts of various material models, and the properties of all constituents in the target triaxially braided composites in this study. The material models and the damage/failure mechanism required in a dynamic impact analysis were developed and described in full length. Triaxially braided carbon fiber preforms, as opposed to traditionally laminated composite layups, have enabled composite structures with complex shapes to be designed and fabricated with components reinforced in a fiber architecture optimized locally for overall performance. One potential mechanism which may account for the difference in performance between braided composites and traditional composites is the load transfer between fiber bundles within the braid in triaxially braided composites. Hence, a more detailed investigation of the deformation processes and failure mechanism in triaxial braid composites is presented in this section. A careful analysis of existing composite test results not only examines the overall material response by presenting a series of stress vs. strain curves, but also provides insight into the nature of progressive composite failure by developing techniques to directly account for the local failure mechanisms.
3.1 Material Properties of composite components

Two fiber/resin combination material systems were examined in this research. The two material systems all had the same fiber, but were infused with a different resin. Toray’s T700 carbon fibers were infused with two different resin systems, Epon’s 862 and Cytec's CYCOM® PR520, which were used to fabricate the finished composite panels. The Toray fiber was a high strength, standard modulus fiber used in applications where tensile strength was needed. The fibers are known to behave as linear elastic material exhibiting an abrupt or brittle failure when the maximum failure strain and stress is reached. Table 3.1 shows the properties of the T700 fiber, as reported by Toray.

Table 3.1 Composite Fiber Properties

<table>
<thead>
<tr>
<th>Tensile Strength (MPa)</th>
<th>Young’s Modulus (GPa)</th>
<th>Failure Strain (%)</th>
<th>Density (g/cm³)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Toray T700 Fiber</td>
<td>4900</td>
<td>230</td>
<td>2.1</td>
</tr>
</tbody>
</table>

Epon’s 862 resin was a low viscosity, high flow thermoset resin ideal for resin transfer molding. Cytec's CYCOM® PR520 resin was a partially toughened thermoset resin with a cure temperature of 179 °C specifically designed for resin transfer molding (RTM) processes. These resins were chosen because they could withstand elevated temperature applications, and were usable up to 104 °C environments. Table 3.2 lists the mechanical properties of each of the above-mentioned resin, as reported by each manufacturer, unless noted.
Table 3.2 Composite Resin Properties

<table>
<thead>
<tr>
<th></th>
<th>Tensile Strength (MPa)</th>
<th>Young's Modulus (GPa)</th>
<th>Density (g/cm³)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cytec CYCOM ® PR520</td>
<td>82</td>
<td>4.0</td>
<td>1.256</td>
</tr>
<tr>
<td>Epon 862(3)</td>
<td>61</td>
<td>2.7</td>
<td>1.200 (2)</td>
</tr>
</tbody>
</table>

**Notes:** 2 – Measured, 3 – Reported at 2.5 hour cure time

3.2 Strain-rate effect of the resin

Based on the experimental data obtained by Littell (2008), Li (2010) had performed a detailed analysis on the strain rate-effect of matrix on the overall mechanical properties of the T700/E862 TDTBC composites and concluded that strain rate dependent resin properties affects strongly the global behavior of the composites by contributing to the properties of fiber tows as fiber tows are taking more than 90% of the load; and the transverse properties are affected strongly compared with the axial ones based on the straight sided specimen. Furthermore, it is apparent that the effects of the strain–rate on the material strength are stronger than on the elastic modulus, or yield stress. As the test data available are only to the 10-1 order per second, whereas in aerospace application, the composite would be loaded at strain rate up to hundreds per second. Accordingly, much more influence of strain rate on the real structure response would be expected and it is very meaningful to reflect it in the material constitutive model. More details and the related test data will be provided in section 4.4.1 of chapter 4.

3.2.1 Strain-rate dependent elastic properties

The rate dependent elastic properties, Young’s modulus $E$, and Poisson’s ratio $\nu$, observed for the resin can be defined by a user Abaqus subroutine **VUSDFLD**, which allows the redefinition of field variables at a material point as the functions of strain rates.
The strain rates at material points can be updated at each time increment. This gives the great convenience of making the strain rate dependence input in tabular form. The same methodology applies to the orthotropic materials as well. Therefore, if the strain rate dependent data are available for the fiber tows, or for the homogenized layers, they can be implemented similarly. For the current study, no rate-dependencies for the fiber-tows due to lack of experimental data was considered while it was considered for the pure resin matrix.

3.2.2 Strain-rate dependent plastic properties

Abaqus allows the yield stress hardening and rate dependency input as direct entries of test data in tabular format. As observed from test data, the yield of the Epon 862 resin is actually strain hardening. Especially, curve fitting would give the chance to adopt possible analytical format if desired, say, the Cowper-Symonds overstress power law or Johnson-Cook for the strain rate effect can be used (DS SIMULIA 2013). A tabulated form with properties value and rate dependency shown in table 4.4 of chapter 4 was used in this study.

3.3 Failure / Damage Modeling of the matrix

The elasto-pastic model combined with the progressive damage model was used to model the matrix material. The resin matrix material model is basically composed of four representative components of material definition (Abaqus, 2013), as sketched in figure 3.1: 1) Undamaged constitutive behavior (e.g., elastic-perfectly plastic or elastic-plastic with hardening); 2) Damage initiation at point A; 3) Damage evolution along the path A–B; 4) Choice of element removal at point B.
As illustrated in figure 3.2, for an elastic-plastic material, the damage manifests in two forms: 1) Softening of the yield stress (equation 3.1); 2) Degradation of the elasticity (equation 3.2) and can be expressed as follows:

\[ \sigma = (1 - d)\bar{\sigma} \]  

(3.1)  

\[ E = (1 - d)E \]  

(3.2)
The strain softening part of the curve cannot represent a material property based on the fracture mechanics and mesh sensitivity considerations. To address the strain-softening feature post damage initiation properly, Hillerborg’s (1976) proposal was adopted. That is, the fracture energy to open a unit area of crack, $G_f$, was assumed to be a material property to describe the progressive damage. Accordingly, the softening response after damage initiation is characterized by a stress-displacement response (rather than a stress-strain response). This requires the introduction of a characteristic length $L$ associated with a material point. The fracture energy is written as

$$G_f = \int \frac{\tilde{\varepsilon}^p}{\varepsilon_0^p} L \sigma_y \tilde{\varepsilon}^p = \int_0^{\tilde{\varepsilon}^p} \sigma_y \tilde{U}^p$$

(3.3)

Where $\tilde{U}^p$ is the equivalent plastic displacement.

The characteristic length $L$ is computed automatically by Abaqus based on element geometry. Elements with large aspect ratios should be avoided to minimize mesh sensitivity. The damage evolution law can be specified either in terms of fracture energy (per unit area) or in terms of the equivalent plastic displacement. Both approaches take into account the characteristic length of the element. The formulation ensures that mesh-sensitivity is minimized.

The energy-based damage evolution may take the linear or exponential format, as shown in figure 3.3. However, the response is linear or exponential only if the undamaged response is perfectly plastic. In this study, the failure strains were obtained at different strain rates from the tests done by Littell (2008) as shown in figure 4.6 (as will be discussed more in chapter 4). The values were converted log strain as listed in table 4.5.
3.3.1 Strain-rate dependent damage initiation strain

The damage initiation strain, or the equivalent plastic strain at the onset of fracture, was tabulated as the function of strain rate in the FEA analysis model. The data were obtained from converting the test data into the logarithmic strain as listed in table 4.5.

3.3.2 Strain-rate dependent damage evolution displacement and energy

The damage evolution displacement, or the equivalent plastic displacement measured from the onset of fracture to the complete failure, was tabulated as the function of strain rate in the FEA analysis model. Similarly, the data was obtained from multiplying the plastic strains by a characteristic length of the typical element in the model or the test coupon. The damage evolution properties used in this study can be found in Table 4.5, as will be discussed more in chapter 4. Damage evolution energy could have also been used otherwise as a substitute if it is available.

3.4 Bonding interface property

Interfacial debonding between fiber tows in different orientations and between fiber tows and the pure resin matrix has been identified as a potential key damage mode in braided composites (Littell, 2008). The high stress concentration and change occurring...
near geometric discontinuities promote initiation of the debonding, which may further cause significant loss of structure integrity. Surface-based cohesive constraints are proposed in this research to simulate the fiber-tow interface for the reasons to be elaborated in section 4.4.3. In addition, the same method is also used to model the bonding (and debonding) between the TDTBC layers as the targeted structure has six stacked layers bonded together via the matrix material. It is convenient to consider cohesive constraints as top and bottom surfaces that are initially well bonded.

Surface-based cohesive behavior provides a simplified way to model cohesive connections with negligibly small interface thicknesses using the traction-separation constitutive model (DS SIMULIA, 2014). The cohesive surface behavior is defined as a surface interaction property for general contact in Abaqus/Explicit. The formulae and laws that govern surface-based cohesive behavior are characterized by following three elements:

1) Linear elastic traction-separation,

2) Damage initiation criteria, and

3) Damage evolution laws.

However, it is important to recognize that damage in surface-based cohesive behavior is an interaction property, not a material property. Traction and separation are interpreted as the contact separation and the contact stress.
The constitutive equation of the cohesive surface, also known as the cohesive law, is illustrated in Figure 3.4. The cohesive law is formed in terms of three components of tractions and separations between two surfaces, one through thickness component, and two transverse shear components. Each component represents a corresponding fracture mode: opening, sliding and shearing modes. In this sense, tractions are considered to be interfacial stresses. It has been shown that a cohesive law can be related to a theory of fracture if the area under the traction-separation curve is equal to the corresponding fracture toughness (Rice, 1968). For behavior representing a pure fracture mode, a bilinear traction-displacement law represented in Figure 3.10 (ABAQUS, 2014) is used. A very high initial stiffness \( (K_n, K_s, K_t) \): normal and tangential stiffness components as shown in figure 3.4), the penalty stiffness, is used to hold the top and bottom surface together. It is necessary to mention that this stiffness is not needed to be defined as the default means the rigid-damage law is used. No deformation occurred before the separation strength was reached. After the traction reaches a peak value, that is the interface strength, the traction decreases with increasing separation, as shown in figure 3.5. At the moment traction is equal to zero, debonding between the top surface and the bottom surface occurs.

Figure 3.4 Traction-separation behavior of the bonding surface
Figure 3.5 Damaged traction-separation response for cohesive bonding surface

Two critical separations $\delta_{\text{max}}$ and $\delta^f$ in the cohesive behavior could be obtained once the interface parameters are known. For the triaxially braided composite being investigated in this study, since the three interfacial strengths are all considered as matrix dominated properties, they are assumed to be equal to respective fiber tow transverse strengths (Li, 2012) and their value can be approximated by averaging those listed in Table 4.5 for the axial and the bias fiber tows, which were predicted using the RVE model as will be described in section 4.3 of chapter 4.

Quadratic stress was used as the damage initiation criteria, which is defined as:

$$\left(\frac{t_n}{t_{n,\text{max}}}\right)^2 + \left(\frac{t_s}{t_{s,\text{max}}}\right)^2 + \left(\frac{t_t}{t_{t,\text{max}}}\right)^2 = 1$$

(3.4)

Where $t_n$ is the normal contact stress in the pure normal mode, $t_s$ the shear contact stress along the first shear direction and $t_t$ the shear contact stress along the second shear direction.
For surface-based cohesive behavior, damage evolution describes the degradation of the cohesive stiffness, which can be based on energy or separation. In this study, an energy based damage evolution was used. Therefore, it is required to specify the total fracture energy as a property of the cohesive interaction. The fracture energy can be defined as tabular or analytical forms. Here, the widely used BK formulation (Benzeggagh and Kenane, 1996) was used that describes the mix mode dependency by

\[ G_{IC} + (G_{IIIC} - G_{IC}) \left( \frac{G_{\text{shear}}}{G_{T}} \right)^{n_{\eta}} = G_{TC} \]  

(3.5)

Where

\[ G_{\text{shear}} = G_{II} + G_{III} \]
\[ G_{T} = G_{I} + G_{\text{shear}} \]

And \( G_{I} \), \( G_{II} \) and \( G_{III} \) are the work done by tractions and their conjugate relative displacements corresponding to opening, shearing and tearing modes respectively. Due to lack of the relevant experimental data for this bonding material, as suggested by (Li 2010), the similar values obtained by Alfano (Alfano 2006) as listed in Table 3.3, for composite of carbon fiber impregnated by epoxy resin, were used for this study. The power, \( \eta \), is a material parameter, here calibrated to 1.45 for carbon fiber composite per Benzeggagh and Kenane (1996).

### Table 3.3 Bonding Interface fracture toughness (J/mm²)

<table>
<thead>
<tr>
<th>Model I ( G_I )</th>
<th>Mode II ( G_{II} )</th>
<th>Mode III ( G_{III} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.268</td>
<td>1.45</td>
<td>1.45</td>
</tr>
</tbody>
</table>
CHAPTER IV

DEVELOPMENT OF A MESO MECHANICAL MODEL FOR DYNAMIC TRIAXIAL BRAIDED COMPOSITE ANALYSIS WITH DAMAGE AND STRAIN RATE EFFECT

Binienda and Li (2010) developed a sophisticated mesoscale analysis model for two-dimensional triaxially braided composites (TDTBC) and used it for studies and simulation of a simple specimen test (Binienda and Li 2010, and Li, Binienda and Goldberg 2010). Material failure and progressive damage of the fiber tows and tow interfaces were taken into account in the model to effectively predict interlaminar and intralaminar failure modes. The model was proven effective for static or quasi-static analysis using an implicit method of Abaqus/Standard (DS SIMULIA, 2014). However, when used for impact analysis in which the explicit finite element analysis method (such as Abaqus/Explicit solver) is needed, the mesoscale model needs to be modified to include the failure and element deletion mechanism for the matrix material. Also, the model proposed by Binienda and Li (2010) used prism element type, termed as C3D6 (six node solid element) and SC6R (six node continuum shell) in Abaqus code, for all three constituents in the composites: the matrix, the axial and bias fiber tows. Specifically, prism solids (C3D6) were used for the pure matrix, prism continuum shells (SC6R) for all the fiber tows and cohesive elements (COH6) for the bonding interface between the
constituents. However, it is known that the prism type of elements can be overly stiff, thus leading to the need of overly sized model due to the need of very densely discretized mesh division. Later, Zhang (2013) proposed a modified mesoscale model to be able to use brick elements unanimously in the model with the prism elements replaced with the eight node solid brick elements, termed as C3D8 in Abaqus, for the matrix and the 3D continuum shell elements for the fiber-tows, termed SC8R in Abaqus. The advantage of Zhang’s model was indeed the thorough use of the eight node brick elements, either C3D8 or SC8R, which have much better mesh quality. However, to maintain the correct edge shape of the fiber-tows in bias directions, an enormous number of elements were still needed in a unit cell. Actually, in both Li and Zhang’s model, the modeling was carefully done to form conformable meshes between different constituents, making a very refined mesh unavoidable. It is necessary to note that the refined mesh in the above-mentioned two mesomechanical models were not for the reason of obtaining better numerical accuracy, but to large extent, due to the need of the geometry build-up, either to maintain the architecture or to match the fiber volume ratio in the real composites. To further simplify the model, the bonding interface between different constituents was idealized as “perfect” so that no interface or debonding needed to be modeled (Zhang 2013) and the bonding was actually realized by sharing common nodes between the constituents.

Obviously, these two approaches are not suitable for the multi-layer braided structures with large domain because of the impractical requirement of computational resources in both space and memory need. Therefore, a brand new mesomechanical analysis methodology was developed in this research and actually employed to build the
FEA model with Abaqus (DS SIMULIA Corp, 2013) to meet the need for high speed impact analysis of large-scale structures. The new model is characterized with a relatively smaller number of elements and fewer degrees of freedom needed in each unit constituent cell than existing mesoscale model for triaxially braided composites, non-conforming mesh between fiber tows and matrix and between different fiber-tows intertwiningly braided together. Due to the need of much fewer elements in the model, the use of a relatively coarse mesh became possible for such a complex material structure with complicated material properties, making the new methodology capable of directly simulating relatively large complex structures like a tensile test specimen and impact coupon panel without the assistance of a simplified approach like a macromechanical model, or a repeated periodic boundary condition when other mesoscale model is used. With all its superb efficiency, the new model is suitable for simulating the behavior of the impact zone or other areas where severe deformation and/or material failure occur locally in an exceedingly large structure such as an automobile roof panel, a radome, fuselage, engine fan containment etc.

On the other hand, in a ballistic impact event, the deformation speed is so high that the rate sensitivity becomes a concern. Polymers have long been known to have a rate-dependent constitutive response, and this was experimentally observed by Littell (2008). Littell (2008) and Littell et al (2008) conducted an extensive experimental and analytical study of the mechanical and impact response of 0°/60°/-60° triaxially braided polymer matrix composites. One important experiment result dealt with the strong strain-rate sensitivity of the resin – the matrix material. Traditionally, for very small strain responses, linear viscoelastic techniques have been used to model the rate dependent
behavior on a phenomenological level (Cheng 2006). However, Cheng’s method, implemented by *MAT_RATESENSITIVE_COMPOSITE_FABRIC (or MAT_158) of LS-DYNA (LSTC, 2007) is generally macroscopic, which is not suitable for the high speed impact where large strain and rotations happen and numerical discontinuities are involved due to material failure and interlaminar delamination. This is especially true when the damage details such as failure initiation and progress amongst the constituents are desired. Hence, it is important for the mesoscale model to be facilitated to reflect the strain-rate effect and material failure in the matrix. Furthermore, upon failure the material or element removal from the model being analyzed needs to be dealt with for all constituents as well.

This chapter gives the details on the new meso-mechanical approach developed to solve for the dynamic response and behavior of structure made with TDTBC materials, in which material failure and strain-rate sensitivity (in the matrix due to the lack of test data for fiber tows) are to be considered.

4.1 Component Material Background

Two material systems, T700S/PR520 resin and T700S/E862, from the tested composite coupons as described in Chapter 3 were considered and the related simulations were conducted in this research. The T700s fiber / PR520 resin and T700s fiber / E862 resin material systems were chosen because they have the highest strength as shown by the test data and are therefore of the greatest interest for use in composite structures. As described in the previous chapters, the fiber was Toray’s high strength, standard modulus T700s Carbon Fiber, and the resins were Cytec’s CYCOM PR520 (Cytec Industries, Inc.) and Epon 862. Cytec’s PR520 was a one part toughened resin
with a cure temperature of $179^\circ$C, specifically designed for resin transfer molding (RTM) processes. Epon’s 862 resins was a low viscosity, high flow resin. Both PR520 and E862 resin are toughened compared with other existing resin systems such as Cytec's 5208 Epoxy and Hexcel’s 3502.

4.2 Object composites geometry and unit constituent cell

It is vital to clarify the fundamental features of object composite TDTBC in question such as the geometry, architecture, and the assistant analytical concept used. As mentioned in section 2.4 of chapter two, the focus of this research was on a particular kind of textile composite, that is, a $[0^\circ/\pm 60^\circ]$ two dimensional triaxially braided composite (TDTBC). Figure 4.1 shows the geometry and architecture of the representative piece of the material build-up with only the fiber tows displayed, including the fiber-tows in both axial (or $0^\circ$) tow and the $\pm 60^\circ$ bias tow directions. The preform architecture of $[0^\circ/60^\circ/–60^\circ]$ braided fiber tows was composed with 24k tows in the axial (or $0^\circ$) direction and 12k tows in the bias ($60^\circ$ and $–60^\circ$) directions. The number of 24k tows in the axial direction was halved to get the 12k tows in the bias directions, so the total fiber volume in each direction was the same. Here, it is necessary to note that “24k” is a designation of the number of fiber filaments in a fiber tow. As a result, the composite may be reasonably assumed quasi-isotropic in each direction.

The modeling process often starts with the concept of a representative volume cell, known as unit cell (Mishnaevsky and Schmauder 2001), which is the smallest portion of a composite whose behavior is representative of the overall behavior of the entire composite (see figure 4.1 and 4.2). For braided composites, the unit cell size depends on the dimensions of the fiber tows and the magnitude of the bias angle (Chou
and Ko, 1989). When the macroscale modeling approach is used either in the “braided over the through thickness integration points” approach (Cheng 2006) or in some kind of smearing or homogenization approach (Littell 2008, Blinzler 2012), the utilization of the unit cell is essential and critical for the determination of both representative geometry and representative material properties. This is because, in these macroscopic approaches, the complex architecture is simplified and represented by geometrically uniform shells without the actual presence of all the details of the braids. In addition, the unit cell is needed to represent the material unit to be used for homogenizing the properties of all the constituents encompassed in the composites architecture as a “single” macro material. As an example, Littell employed the shell elements with the “through thickness integration points” scheme to model triaxially braided composite, and used the “top-down” approach for the unit cells to characterize material parameters for input to damage material model (2008).

![Figure 4.1 TDTBC representative geometry and dimension of the unit cell](image)

**2W=18.5 mm**

**L=5.3 mm**
Figure 4.2 Braided model representations – four unit cells

However, as discussed earlier, the new mesomechanical modeling approach proposed in this research did not actually use the unit cell as a building block for the whole structure, though it was widely used by many researchers either for macroscale modeling (Cheng 2006, Littell 2008 and Blinzler 2012) or mesoscale modeling (Li 2010, Zhang 2013). Instead, the concept of unit constituent cell was used in a replacement for actually building the analysis models for more flexibility, better mesh quality and computational efficiency. There are two major reasons for this decision: 1) the unit cell is not a must any more in the mesomechanical modeling approach because of the detailed presence of the braided architecture; 2) the modeling becomes more flexible and effective to abandon the unit cell structure as it would adversely restrict the meshing work. That
being said, the term unit cell will still be frequently used in this dissertation purely, for the purpose of consistency and the convenience of discussion.

As its name suggests, the basic concept of unit constituent cell is to take the smallest continuous representative volume of each individual constituent, i.e. the matrix, the fiber tows in axial, bias 60° and −60° directions, without binding them together to form a conforming mesh, as shown in Figure 4.3 for the assembled composite architecture. The bonding between constituents was modeled with the surface-based constraints (DS Simulia, 2013) without using any elements. More details on dimensions, modeling and material property determination of each individual constituent will be given in the following sections.

Figure 4.3 Assembled unit constituent cells
4.3 Determination of constituent dimensions and fiber volume ratio

As mentioned earlier, the unit cell was not used as building block for model generation in this research in order to avoid the small and often irregular local geometric entities that would otherwise be induced by the nonphysical, virtual borders introduced between unit cells. However, we still needed a way to describe the details of the representative geometry of the composite and its constituents accurately. To this end, the unit cell concept was used to describe the geometry and dimensions but not for actual mesh generation.

Zhang (2013) introduced a very detailed method to “accurately” determine the cross section of the fiber-tows and the overall dimension of the target TDTBC composite with the unit cell concept. In this method, the unit cell size depends on the dimension of the fiber bundles, the distribution of fiber bundles, and the braid angle. The length $L$ of the unit cell, as shown in Figure 4.1, was selected to be the axial distance between the center lines of two neighboring bias yarns, while the width of the unit cell is determined by twice the braiding space, $W$. The braiding space is the transverse distance between center lines of two neighboring axial yarns. Meanwhile, from Chou and Ko (1989)’s discussion, the unit cell length and braiding space follow the relationship,

$$L = \frac{W}{\tan(\theta)}$$

where $\theta$ denotes the braid angle.

The real geometry was determined by image analysis for the cross sectional area of the fiber bundles, as well as other geometric parameters needed for constructing the finite element model, such as the height and width of the fiber bundles. Figure 4.4 shows
a microscopic image of a cross section of a six layered specimen in the axial direction (Littell 2008). The black areas indicate the axial fiber bundle cross sections, the white areas represent the bias fiber bundles, and the gray areas are the pure matrix. From figure 4.4, it is clear that the layer shift exist in the thickness direction, which was not considered in this study due to its randomness and complexity.

![Microscopic axial cross-sectional image of a six-layer 0°/±60° carbon/epoxy braided composite (Littell 2008)](image)

Following the same procedure proposed by Zhang (2013), we got the composite geometry of unit cells as summarized in Table 4.1, which was slightly different from those used in Zhang’s report for better consistency and accuracy in geometry dimensions. The geometric dimensions used in the finite element model in this research are summarized in Table 4.2, which match those in Table 4.1 closely.

<table>
<thead>
<tr>
<th>Unit cell architecture properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>2W(mm)</td>
</tr>
<tr>
<td>18.1</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Fiber bundle architecture properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>x_a(mm^2)</td>
</tr>
<tr>
<td>1.2</td>
</tr>
</tbody>
</table>
Unlike Zhang’s model, the same invariable shape and area of the cross section as the measured ones can be used for the fiber volume ratio calculation in current model. In the other existing mesoscale models, the volume ratio was calculated in terms of the actual calculated volumes of the fiber tows and the matrix as the cross section changed and sometime may become quite irregular over the geometric scope of interest such as in the unit cell. On contrary, a constant cross section shape and area were utilized for all fiber tows in the current research, thanks to the non-conforming meshes. However, since the layer shift was neglected, the resultant finite element mesh may have different fiber bundle volumes as compared to the actual composite. With the fiber volumes for different fiber-tows, the material properties were computed, as will be discussed in later sections.

4.4 Method of modeling each constituent components in TDTBC

This section deals with the modeling details used for each constituent in the TDTBC composite being discussed in this study. Different FEA modeling techniques and material models were used for the fiber tows and the matrix, as they possess disparate features in both material properties and geometry. The adoption of the new unit constituent cell concept, as discussed earlier and shown in figure 4.3, made the modeling of each constituent considerably flexible and independent. No conforming mesh between
constituents or matching bonding interface was required. Modeling of delamination between layers became much easier. Furthermore, as the unit cell border does not exist anymore in the model, the unit cell becomes merely a concept for the convenience of comparison and discussion.

![Figure 4.5 Four assembled conventional unit cells](image)

The disappearance of the unit cell border made it possible to avoid the irregular geometric entities that would be otherwise created due to the use of unit cells, thus many low quality elements disappear accordingly. Hence, the model size, or the number of elements needed in a conventional unit cell, was reduced over ten-fold and the stable time limit, which will be addressed in section 4.5, was increased by nearly 2.5 times, leading to an efficient FEA model suitable for large scale structure analysis. A complete comparison of the important features between the current model and some traditional unit cell based mesoscale models (as shown in figure 4.5) is presented in table 4.3. For the consistence of comparison, the numbers counted in a unit cell were adopted. From Table 4.3, it was evident that the meso-mechanical model proposed in this study was much more efficient than similar pre-existing approaches.
Table 4.3  Unit cell feature comparison with existing mesoscale models

<table>
<thead>
<tr>
<th>Per Unit Cell</th>
<th>Total element number</th>
<th>Total number of DOFs</th>
<th>Stable time limit (second)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Li (2010)</td>
<td>13,562</td>
<td>41,864</td>
<td>7.84e-9</td>
</tr>
<tr>
<td>Zhang (2013)</td>
<td>13,520</td>
<td>46,780</td>
<td>1.07e-8</td>
</tr>
<tr>
<td>Current</td>
<td>1,250</td>
<td>7,080</td>
<td>2.5e-8</td>
</tr>
</tbody>
</table>

4.4.1 Pure resin matrix modeling

1) Material consideration

The in-situ material properties of pure resin phase in mesomechanical model were assumed to be the same as those of the bulk material. In the previous mesomechanical modeling of the TDTBC, the dynamic feature of rate sensitivity was completely ignored by researchers for the matrix, and for simplicity, its material property usually took the form of elastic-plasticity expediently, corresponding to the lowest strain rate (Li 2012, Zhang 2013). This is acceptable for static or quasi-static scenarios. However, based on results for the stress-strain response of the epoxy resins obtained by Littell (2008), the strain-rate effects on both the elastic and plastic properties were very evident. For example, as shown in Figure 4.6, it is clear that the E862 resin has strain rate sensitivity in Young’s modulus, yield stress and fracture strain, although the test data for very high strain rate were lacking. Hence, it would be worthy of including the strain rate effect to improve the accuracy in the simulation of high speed dynamic events like impact. For the rate range within which the data lack, interpolation and extrapolation were utilized.
The nominal stress versus strain curves obtained by Littell (2008) were converted into Cauchy (“true”) stress and logarithmic strain (Shown in figure 4.4.) using equations 4.1 and 4.2 in this study as the FEA model needed the input in this format.

\[
\sigma = (1 + \epsilon_{nom})\sigma_{nom} \quad (4.1)
\]

\[
\epsilon = \ln (1 + \epsilon_{nom}) \quad (4.2)
\]

The elastic and plastic material properties used in the FEA model were obtained by performing material calibration with the engineering values from test shown in figure 4.7. Abaqus preprocessor, Abaqus/CAE (DS SIMULIA, 2014), has a convenient tool for material property calibration and was used for determining the Young’s modulus and identifying the yield stress and hardening parameter. With the strain rate dependency directly input from test data entry in tabular form, the basic elastic material properties utilized for the matrix material in this study are tabulated in Table 4.4. The damage initiation strains corresponding to the onset of damage were conservatively taken as those
close to the yield point, and the damage evolution plastic displacement corresponding to the complete breakage of the material was measured at the complete failure of specimen from the data shown in figure 4.6 and 4.7 and calculated with the consideration of the characteristic length (Lawn 1993; Camanho et al 2007) of the typical element in the model. The two damage properties needed to model the damage initiation and progressive damage evolution are summarized in Table 4.5.

More discussion on the rate dependency of the elastic and plastic properties and the damage/failure properties utilized in the FEA model input data can be found in section 3.2 and 3.3, respectively.

![Figure 4.7 True stress - log strain curves of E862 at different strain rates](image)

**Table 4.4 Rate dependent elastic-plastic material properties of the E862 matrix**

<table>
<thead>
<tr>
<th>Rate (s⁻¹)</th>
<th>Young’s modulus (GPa)</th>
<th>Yield stress (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1e⁻⁵</td>
<td>2.17</td>
<td>31</td>
</tr>
<tr>
<td>1e⁻³</td>
<td>2.41</td>
<td>39</td>
</tr>
<tr>
<td>0.1</td>
<td>2.71</td>
<td>66</td>
</tr>
<tr>
<td>1e⁻⁵</td>
<td>31</td>
<td>39</td>
</tr>
<tr>
<td>1e⁻³</td>
<td>39</td>
<td>66</td>
</tr>
</tbody>
</table>
Table 4.5 Rate Dependent Damage properties of the E862 matrix

<table>
<thead>
<tr>
<th>Rate (s⁻¹)</th>
<th>Damage initiation strain (%)</th>
<th>Damage evolution displacement</th>
</tr>
</thead>
<tbody>
<tr>
<td>1e⁻⁵</td>
<td>1e⁻³</td>
<td>0.1</td>
</tr>
<tr>
<td>1.10</td>
<td>1.68</td>
<td>2.16</td>
</tr>
<tr>
<td>1e⁻⁵</td>
<td>1e⁻³</td>
<td>0.25</td>
</tr>
<tr>
<td>1e⁻³</td>
<td>0.28</td>
<td>0.22</td>
</tr>
</tbody>
</table>

2. Meshing

The geometry of the matrix was precisely created by using the merge/cut geometry functionality to form new part from existing parts in Abaqus/CAE to cut the geometry of the encompassed fiber tows from a cuboid. Solid elements C3D8R was primarily used except for limited area where the transition of mesh was needed. In such transition area the C3D6 elements were used. The size and area of the conventional unit cell was used for and only for the matrix as it happened to be the same as those of the unit constituent cell currently used in the context.

The meshed unit cell of the matrix is shown in figure 4.8. It has only 650 elements in total. In a real structure model, this meshed part can be repeatedly used as many times as needed to match the real shape of the structure to be modeled.

![Figure 4.8 Unit cell geometry and mesh of the matrix](image-url)
4.4.2 Fiber tows modeling

Fiber tows are classified into three groups: the axial fiber-tows aligned in axial (or 0°) direction, the positive and negative bias fiber-tows aligned in ±60° directions, respectively. The modeling of each group of fiber tows will be addressed individually in subsequent sections.

4.4.2.1 Material modeling

Before the onset of damage, the fiber tows are assumed to be linearly elastic. The initial or undamaged elastic material properties of the fiber tows used in the meso-mechanical model proposed in this study were predicted using a representative volume element (RVE) model of a unidirectional composite established by Zhang (2013). This RVE model was based on Huang's micromechanical model for a transversely isotropic composite (Huang 2001), which was used to develop elastic constants for the fiber bundles in the triaxial braided composite. Compared to other existing micromechanical models (Aboudi 1989, Teply and Reddy 1991, for example), Huang’s model provides direct equations to calculate the elastic constants. In addition, Huang’s model can take into account the nonlinear deformation of each constituent.

Each group of the fiber tows can be considered as a transversely isotropic unidirectional composite, whose engineering elastic constants were calculated in terms of matrix properties, fiber properties, and the fiber volume fraction \( v_f \). The \( x \) coordinate is defined as the direction aligned with the fibers in the bundle, and the \( y-z \) plane is perpendicular to this direction. All the properties obtained from the RVE model are listed in table 4.6 and 4.7 and were used to model the axial \((v_f=81\%)\) and bias fiber-tows \((v_f=78\%)\), respectively.
Table 4.6 Elastic constants of fiber bundles predicted by micromechanical RVE model

<table>
<thead>
<tr>
<th>$v_f$</th>
<th>$E_x$ (GPa)</th>
<th>$E_y$ (GPa)</th>
<th>$E_z$ (GPa)</th>
<th>$v_{xy}$</th>
<th>$v_{xz}$</th>
<th>$v_{yz}$</th>
<th>$G_{xy}$ (GPa)</th>
<th>$G_{yz}$ (GPa)</th>
<th>$G_{xz}$ (GPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>81%</td>
<td>184.54</td>
<td>10.22</td>
<td>10.22</td>
<td>0.294</td>
<td>0.295</td>
<td>0.521</td>
<td>6.54</td>
<td>3.30</td>
<td>6.54</td>
</tr>
<tr>
<td>78%</td>
<td>180.00</td>
<td>9.56</td>
<td>9.56</td>
<td>0.296</td>
<td>0.296</td>
<td>0.518</td>
<td>5.86</td>
<td>3.16</td>
<td>5.86</td>
</tr>
</tbody>
</table>

Table 4.7 Strengths of fiber bundles predicted using micromechanical RVE model

<table>
<thead>
<tr>
<th>$v_f$</th>
<th>$F_{1T}$ (MPa)</th>
<th>$F_{IC}$ (MPa)</th>
<th>$F_{2T}$ (MPa)</th>
<th>$F_{2C}$ (MPa)</th>
<th>$F_{1S}$ (MPa)</th>
<th>$F_{2S}$ (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>81%</td>
<td>3980</td>
<td>3582</td>
<td>110</td>
<td>212</td>
<td>79</td>
<td>78</td>
</tr>
<tr>
<td>78%</td>
<td>3834</td>
<td>3451.5</td>
<td>108.65</td>
<td>208.63</td>
<td>77</td>
<td>76</td>
</tr>
</tbody>
</table>

As discussed later, the best element type suitable for dynamic analysis of fiber tows with damage is the solid element (C3D8R). It is assumed that the elastic stress-strain relations are given by orthotropic damaged elasticity. However, only the plane-stress based two-dimensional Hashin’s failure criterion is available in Abaqus as a standard facility to model the damage initiation and progress of fiber-reinforced composite-material. Hence, the Hashin failure criterion was extended to three dimension material in this study and coded as a user material via VUMAT (Abaqus, 2013). The source code of the material model was in FORTRAN and is attached in the Appendix. Both the elastic and damage mechanics based behavior were included in this user material model as required.

The initial or undamaged elastic constants were calculated from the values of Young’s modulus and Poisson’s ratio obtained from the RVE model by the following expressions:
The constitutive model of failure for the fiber tows was based on Hashin’s failure criteria for unidirectional fiber composites (Hashin, 1980). For the matrix failure modes, a constitutive model based on Puck’s action plane theory (Puck, 1998) was used. The 3D version of the Hashin model was then used to predict the damage and failure of the fiber tows in conjunction with solid elements C3D8R.

In this 3D Hashin damage criterion, four distinct failure modes were considered: tensile fiber failure, compressive fiber failure, tensile matrix failure, and compressive matrix failure, expressed mathematically as below.

Tensile fiber mode \((\sigma_{11} > 0)\):
\[
d_{ft} = \left( \frac{\sigma_{11}}{F_{1T}} \right)^2 + \alpha \left( \frac{\sigma_{12}}{F_{1S}} \right)^2 + \beta \left( \frac{\sigma_{13}}{F_{2S}} \right)^2 = 1
\] (4.4)

Compressive fiber mode \((\sigma_{11} < 0)\):
\[
d_{fc} = \left( \frac{\sigma_{11}}{F_{1C}} \right)^2 = 1
\] (4.5)

Tensile and compressive matrix modes:
\[
d_m = \left[ \frac{\sigma_{11}}{2F_{1T}} \right]^2 + \left( \frac{\sigma_{22}}{F_{2T}F_{2C}} \right)^2 + \left( \frac{\sigma_{12}}{F_{1S}} \right)^2 + \sigma_{22} \left[ \frac{1}{F_{2T}} + \frac{1}{F_{2C}} \right] = 1
\] (4.6)

\[
d_m = \begin{cases} 
  d_{mt}, & \text{when } \sigma_{22} + \sigma_{33} > 0 \\
  d_{mc}, & \text{when } \sigma_{22} + \sigma_{33} < 0
\end{cases}
\]
$d_{mt}, d_{mc}$ are current damage variables with the range of $[0,1]$ and $F_{1T}, F_{1C}, F_{2T}, F_{2C}, F_{1S}$ and $F_{2S}$ are material strengths corresponding to different pure failure modes. The behavior of the material is linear elastic before satisfying one of the damage criteria which compete with each other. For the tensile fiber failure calculation, two coefficients, $\alpha$ and $\beta$, ranging from 0 and 1, were introduced to weigh the contribution of shear stresses to the initiation of fiber tensile failure as expressed in Equation 4.4. Because of the complex fiber architecture, even under axial or transverse tensile loading conditions, the braided composites would be subject to relatively large in-plane shear stresses. Hence, properly considering the effects of shear stress on the fiber tensile failure of the fiber tows is necessary for accurately modeling the global composite behavior. Numerical correlation studies were conducted for a simple tensile specimen test. And the appropriate values of both $\alpha$ and $\beta$ were both determined as 0.4 based on these numerical tests as they correlated with the tensile test result well.

Once a damage criterion was satisfied, a mechanism was included in the model to remove failed elements. An anisotropic damage initiation law similar to the plane–stress formula provided by Abaqus as standard functionality was utilized, which accounts for the four failure modes. Progressive damage evolution was not considered in the model in consideration of the brittle failure feature of the fiber tows. A damping factor was facilitated into the failure model to improve the stability of the numerical analysis by the finite element solver as needed; however, even without using it for many cases, stable solutions were still able to be obtained. This was due to the fact that the plasticity and progressive failure mechanism already included in the pure resin matrix material.
In terms of the initial elastic constants $C_{ij}^0$ obtained via equation 4.3, the damaged elastic constants in the material stiffness matrix were computed from:

\[
C_{ij} = (1 - d_f)(1 - d_m)C_{ij}^0
\]

Where

\[
\begin{align*}
C_{11} &= (1 - d_f)(1 - d_m)C_{11}^0 \\
C_{22} &= (1 - d_f)(1 - d_m)C_{22}^0 \\
C_{33} &= (1 - d_f)(1 - d_m)C_{33}^0 \\
C_{12} &= (1 - d_f)(1 - d_m)C_{12}^0 \\
C_{23} &= (1 - d_f)(1 - d_m)C_{23}^0 \\
C_{31} &= (1 - d_f)(1 - d_m)C_{31}^0 \\
G_{12} &= (1 - d_f)(1 - s_{mt}d_{mt})(1 - s_{rc}d_{rc})C_{12}^0 \\
G_{23} &= (1 - d_f)(1 - s_{mt}d_{mt})(1 - s_{mc}d_{mc})C_{23}^0 \\
G_{31} &= (1 - d_f)(1 - s_{mt}d_{mt})(1 - s_{mc}d_{mc})C_{31}^0
\end{align*}
\] (4.7)

are the global fiber and matrix damage variables, computed from fiber tension and compression failure modes ($d_f$ and $d_m$) and matrix tension and compression failure modes ($d_{mt}$ and $d_{mc}$), respectively, corresponding to equations 4.4–4.6. And the factors $s_{mt}$ and $s_{mc}$ in the definitions of the shear moduli were introduced to control the loss of shear stiffness caused by matrix tensile and compressive failure respectively. The following values were assumed empirically: $s_{mt} = 0.8$ and $s_{mc} = 0.5$.

Finally the damaged elastic stress strain relations took the form shown below in equation 4.9:

\[
\begin{bmatrix}
\sigma_{11} \\
\sigma_{22} \\
\sigma_{33} \\
\sigma_{12} \\
\sigma_{23} \\
\sigma_{31}
\end{bmatrix} = \begin{bmatrix}
C_{11} & C_{12} & C_{13} & 0 & 0 & 0 \\
C_{12} & C_{22} & C_{23} & 0 & 0 & 0 \\
C_{13} & C_{23} & C_{33} & 0 & 0 & 0 \\
0 & 0 & 0 & 2G_{12} & 0 & 0 \\
0 & 0 & 0 & 0 & 2G_{23} & 0 \\
0 & 0 & 0 & 0 & 0 & 2G_{31}
\end{bmatrix} \begin{bmatrix}
\varepsilon_{11} \\
\varepsilon_{22} \\
\varepsilon_{33} \\
\varepsilon_{12} \\
\varepsilon_{23} \\
\varepsilon_{31}
\end{bmatrix}
\] (4.9)
referred to a local coordinate system (O123) with the axis O1 aligned with the fiber direction, the axis O2 corresponding to the in-plane direction perpendicular to the fiber direction and the O3 normal to the plane of the braided composite.

4.3.2.1 Meshing

Although continuum shell elements discretize a three-dimensional body, care should be taken to verify whether the overall deformation sustained by these elements is consistent with their layer-wise plane stress assumption; that is, the response is bending-dominated and no significant thickness change is observed (say, approximately less than 10% thickness change). Otherwise, regular three-dimensional solid elements should be used (Abaqus, 2013). Even though the so-far reported mesoscale methods by Abaqus use continuum shell (SC8R) to mesh the fiber tows unexceptionally, the current study suggested using the solid element (C3D8R) to model all fiber tows. This is because, when simulating dynamic event the deformation may go very severe like these examples shown in chapter 6, the thickness may change very significantly, causing the analysis to fail prematurely if the continuum shell were used. Even in a simple tensile test, the specimen may undergo thickness change of greater than 10% of its original thickness. A robust algorithm should be able to consider this. An inherent limitation and disadvantage exist with continuum shell is that the thickness change on the SC8R elements should not be significantly larger than 10%. Even though the SC8R elements do provide a larger stable time increment than the same dimensional C3D8R favorably enough for an explicit simulation, the solid elements are essentially needed for modeling the impact response of the materials like braided composites. Utilizing the solid elements even results in further extra cost – the Hashin’s failure criteria for unidirectional fiber composites had to be
extended to three dimensional problems and coded with a user material model as the program only provides the capability for plane stress formula such as conventional and continuum shells (DS Simulia, 2013). And this would, in turn, cause extra memory consumption as a result of user subroutine calling and data transfer.

The mesh used in the meso-mechanical model for the axial and bias fiber-tows is described in detail below, respectively.

1) Axial fiber tows

As shown in figure 4.9, the total number of solid elements is only 36 in an individual unit cell of the axial (zero) fiber tows with good quality. Despite a great reduction in model size compared with existing mesoscale models (Li 2010, Zhang 2013) the geometry of the tows is accurately captured. It is necessary to note that a sharp end was used for tow geometry, similar to that observed in the real material as shown in figure 4.

Figure 4.9 The geometry and mesh of the axial fiber-tows (two unit cell long)
2. Bias fiber tows

The real geometry is depicted accurately with only 344 elements per individual unit constituent cell. The geometry and mesh of the unit constituent cell of bias fiber-tows are shown in Figure 4.10, respectively, for $+60^0$ and $-60^0$ direction. As discussed before, a complete, continuous representative constituent length range was modeled for each tow so as to avoid possible small geometric entities caused by the artificial borders between the unit cells. Despite the undulation in the representative length due to the braiding over each other, constant cross section profile is maintained, providing the accuracy and convenience in meshing and volume ratio calculation.

![Figure 4.10 The geometry and mesh of the bias fiber-tows](image)

Figure 4.10 The geometry and mesh of the bias fiber-tows
4.4.3 Bonding interface consideration

One important experiment result from (Littell 2008) and analysis done by Binienda and Li (2010) is that for different material systems where the constituent properties of the fiber and matrix were nominally the same, the resulting static and impact strengths of the resulting braided composites varied significantly. One possible cause of this variation is hypothesized to be related to the fiber tow-matrix interface in various composites. Therefore, accurate modeling of the bonding surface is critically important to get the correct simulation result. On the other hand, because the bonding between the fiber tows are very thin, modeling the bonding material with regular cohesive elements would be very hard in the meshing work if not impossible and the elements are vulnerable to excessive distortion that may lead to prematurely failure of analysis. To resolve this issue, an improvement was done in this study by replacing the cohesive elements with surface-based cohesive connection. By doing so, no elements were needed to define the bonding surface. Instead, the bonding (and possible debonding!) was modeled by defining cohesive contact interaction properties between the two bonding surfaces in conjunction with the general contact capability of Abaqus/Explicit. The typical bonding properties and damage mechanism used in this research are illustrated in figure 4.11. As shown, it is typically characterized by peak strength \( (N) \) and mode-dependent fracture energy \( (G_{TC}) \). In addition, the failure mode interaction may be included. The damage properties for the bonding surfaces were already discussed with depth in Chapter 3 and the values used in this study can be found in Table 3.3.
4.5 The stable time limit

One of the main modeling tasks when using explicit FEA solver like Abaqus/Explicit would be to increase the stable time limit by all possible means to reduce the total time to run the analysis. This is because the stable time increment is one of the key deciding factors to control the total time required to complete the analysis. As the explicit dynamics procedure solves every problem as a wave propagation problem, out-of-balance forces are propagated as stress waves between neighboring elements. A bounded solution is obtained only when the time increment ($\delta t$) is less than the stable time increment. The stable time increment is the minimum time that a dilatational (i.e., pressure) wave takes to move across any element in the model. Therefore, the element shapes and sizes directly influence the total run time. As a quick reference, the stable time increment is calculated automatically in an explicit code by

$$\Delta t \leq \min \left( L_c \sqrt{\frac{\rho}{\hat{\lambda} + 2\hat{\mu}}} \right); \quad \hat{\lambda} = \frac{E_v}{(1+v)(1-2v)}; \quad \hat{\mu} = \frac{E}{2(1+v)}$$

(4.10)
For the reason mentioned above, the mesh density in a constituent has to be carefully decided not only in the sense to reduce the model size, but also to effectively reduce the total run time to complete the analysis as well. When modeling TDTBC composites at the fiber tow level, very small element dimensions can be encountered due to a more complex geometry. This can lead to a very small stable time increment, resulting in requiring a large number of increments to complete an analysis. For a typical fiber tow modeled where $L_e=0.002$ mm as in the model proposed by Li (2010), $E_x=185$ GPa, $E_y=11$ GPa, Density=$1.6e-9$ (ton/mm$^3$), the stable time increment is on the order of $5.5e-10$ seconds. And this can be quickly reduced to less than $1.e-10$ sec due to the element distortion after the impact starts. This means that to simulate 1 millisecond will require 10 million increments! In the current modeling technique, the mesh size was carefully chosen such that no extremly small elements would be generated. Actually, as an example, the original stable time limit $5.5e-10$ was increased to about $2.2.e-7$ by simply modifying the smallest thickness of the axial fiber tow used in Li’s model from the 0.002 mm to 0.8 mm. It would increase the computation efficiency by about 400 times without any result degradation. Actually, the result change would be trivial due to this minimal geometry change.

Mass scaling available in Abaqus/Explicit would have been used to increase the stable time; unfortunately, for the high speed dynamics, small amount of the mass scaling would affect the analysis result significantly. Sometime it makes the analysis fail prematurely due to excessive non-physical deformation speed change. Therefore, care should be taken that only a very small amount of mass scaling was applied so that the mass change in the system was maintained as 1~2% throughout the analysis.
4.6 Verification example – 2 by 3 unit cells impacted by a rigid projectile

To verify the newly developed dynamic mesomechanical model, a single layer composite plate made up of T700/E862 material, impacted by a rigid projectile at 160 m/s (shown in Figure 4.12), was analyzed. The plate was made with merely 6 (=2 by 3) unit cells. Symmetric condition was assumed; therefore, only a quarter of the structure was modeled. The delamination and damage of the fiber tows, the resin and the bonding interface were effectively simulated as shown in figure 4.13~4.16. Although there is no experimental result available to compare with, the successful simulation of all the possible behavior and damage phenomena of the structure and material that may occur in this simple structure when subjected to impact verifies that the methodology developed in this research is effective in predicting the material failure and the interlaminar delamination or debonding etc.

Simulations for more complex cases and the relevant comparisons with appropriate experimental results will be presented in chapter 6.

Figure 4.12 The verification analysis model
Figure 4.13  Delaminating and damage of the fiber tows

Figure 4.14  Delamination and damage of the matrix
Figure 4.15 Failure of the fiber tows

Figure 4.16 Original and damaged bonding interface
CHAPTER V

DEVELOPMENT OF A COMBINED MULTISCALE MODELING APPROACH FOR DYNAMIC ANALYSIS OF TRIAXIAL BRAIDED COMPOSITE

The previous chapters describe both the meso and the macro-mechanical methods developed and implemented to simulate the material response of the composite constituents and coupons for TDTBC. In order for the two approaches to be utilized in real applications for large analysis structure with many degrees of freedoms (DOFs) efficiently, methods and techniques are required to combine them into the same analysis model effectively and appropriately. Hence, in this chapter, a comprehensive and systematic discussion on the corresponding approaches will be given regarding the association and connection for the regions or portions of a structure or object being analyzed. A highly efficient and practical method for analyzing large-scale problems was developed based on these cutting-edge studies.

5.1 Methods to Associate Regions Modeled with Different Approaches

There are two methods that can be used to relate the regions modeled with disparate detail levels or different modeling approaches: one is called the Direct Connecting Method; the other is the so-called Submodeling Method. As will be discussed soon, each method has its advantages and shortcomings, with the latter providing more flexibility and robust, and better accuracy and efficiency in modeling complex impact
events for the scenarios in which the critical region(s) needing special care cannot be predicted. Therefore, the focus will be put on the Submodeling Method in this research.

For static analyses, the effectiveness of the Submodeling Method can be easily addressed by the Saint-Venant's Principle (Saint Venant 1855); however, when a dynamic problem is analyzed, the reflection and propagation of stress wave across the connecting interface could become a concern: will it really be appropriate to use a submodel with higher modeling fidelity to represent the local region(s) of interest, sliced from the entire model (known as global model)? Are there any measures available to be taken to alleviate the adverse effect if any? These questions will be answered in this chapter.

5.1.1 Direct connecting method

The direct connecting method divides the structure into two distinct parts with different levels of modeling detail generated with the meso and / or macroscale approach. It uses either sharing nodes or mathematical interpolation imposed on the geometric connections that couples the two disparate meshes, different type of elements, and material properties across the interface of the mesh transitioning zone. It is necessary to mention that these differences in modeling detail do not necessarily appear in the same analysis model. This method is direct, intuitive, and the modeling methods with different levels of fidelity such as mesoscale and macroscale approach coexist in the same finite element analysis model. However, the nonphysical response may happen in the connecting/coupling zone, for instance, extremely high stress/strain concentrations often leading to unrealistic pre-mature failure of the material. Furthermore, the modeling and meshing work is usually arduous and challenging in the transitioning zone across the connecting interface. The requirement of kinematic and dynamic continuity conditions in
the interface makes modeling work overwhelming for the braided composites with complex architectures. Therefore, except for relatively simple cases with smooth geometry across the connecting border, this method is not recommended and will not be further discussed in this research.

Figure 5.1 Illustration of the direct connecting method

5.1.2 Submodeling Method

Submodeling is a technique used to study a local part of a model with a refined mesh, detailed geometry or a material model of higher fidelity, based on interpolation of the solution from an initial, global model onto appropriate parts of the boundary of a submodel. The method is most useful when it is necessary to obtain an accurate, detailed solution in local region(s) whereas the use of the overall detailed modeling for the entire object or structure to be analyzed is computationally too expensive or not worthwhile. This method requires the detailed modeling of the local region(s) to have a negligible effect on the overall solution to guarantee the accuracy. The response at the boundary of the local region(s) is defined by the solution from the global model and this, together with any loads applied to the local region, determines the solution of the submodel. The basic concept of the submodeling is illustrated in figure 5.2. The technique relies on the global model defining the submodel boundary response with sufficient accuracy. More details
regarding the method can be found in the Abaqus Users Analysis Manual (DS SIMULIA, 2013). The region(s) in the global model used to deduce or interpolate the response of the submodel boundary is known as driving region while the submodel boundary defined by the global as driven boundary. For dynamic analyses, the explicit solver of finite element analysis like Abaqus/Explicit deals with any problems as wave propagation; therefore, wave reflection across the connecting region would cause some concerns when a direct connecting method is used. It is evident that one of the great advantages of using submodeling for dynamic problem is that the connecting driven boundary of the submodel is directly defined from the global model through appropriate interpolation scheme. It is not obtained by solving the equations of motion, and as a result, the boundary sensitivity is avoided tactically.

In addition, another critical topic in performing submodeling analysis is the selection of the submodel boundary locations, that is, how close or far they should be away from the area of interest. Luckily, two classical theories support the method: the uniqueness of solutions to elastodynamic and plastodynamic boundary-value problems and the dynamic version of Saint-Venant's Principle, as will be discussed in more details later in section 5.1.3.

![Figure 5.2 Illustration of the submodeling technique](image)

Figure 5.2 Illustration of the submodeling technique
With such features of this technique, it is very attractive and natural to consider building a global model using a relatively simple method with less complexity like the macro modeling approach, and then combine its solution with a submodel built for local critical regions with more details. With much smaller analysis region, the submodel includes regions where more detailed models are necessary to correctly capture the real physics such as material failure or delamination when performing the detailed analysis for the whole structure is too expensive or impractical.

Generally, there are two forms of the submodeling techniques implemented in FEA codes like Abaqus (DS Simulia, 2013). The first technique, that is, the more general node-based submodeling technique, transfers node-located solution variables, most commonly displacements from global model nodes to submodel nodes. The second available submodeling technique is a surface-based technique, which transfers material point stress results from the global model to surface load integration points in the submodel. As node-based submodeling is generally preferable when the displacements in the global model correspond closely with the expected displacements in the submodel, the *Node-based submodeling* will only be considered in this context.

The submodel analysis is run as a separate analysis from the global analysis with the sole link between the two models being the transfer of the time-dependent values of variables saved in the global analysis to the relevant boundary nodes (known as the driven nodes) of the submodel or to the relevant boundary surfaces. In FEA code like Abaqus, this transfer is accomplished by saving the results from the global model either in the results file or in the output database, then reading these results into the submodel analysis.
In summary, an initial global analysis of a structure usually identifies areas where the response to the loading is deemed crucial. Submodeling then provides easy model enhancements of these areas without having to resort to remeshing and reanalyzing the entire model. While the local model (the submodel) built with mesoscale or other a more detailed approach can include all possible advanced features of the material and response such as nonlinearity, damage/failure, delamination and erosion, the global model built with a less complex method- like a coarse mesoscale model with simplified material description, or a macro model, or other simplified approaches- may be present at various approximate levels.

Compared with the direct connecting method as described, the submodeling method offers the following advantages:

1) It provides the flexibility to observe, determine the size, shape and location of the critical zone as the submodel after studying and observing the global model result – thus making it possible to make post-event decision.

2) It can be used to any number of levels; a submodel can be used as the global model for a subsequent submodel. This makes it possible for multi-level use of the technique.

3) It makes meshing/modeling work easier, as there is no need to consider the geometric continuity, mesh transition smoothness etc.

For many applications, the damage shape and area are sometimes hard to be estimated before the analysis is actually done. Even for the same structure, different damage patterns may happen due to different loading and boundary conditions. As shown in figure 5.3, the plate used in an impact test and analysis may have various damage patterns
Analysts are required to give the detailed local response information after analyzing the general response of the structure. For these cases, the submodeling provides the unique advantage as the determination of the location, size, shape and the material models of the submodel to be used can be done after the global analysis is performed.

![Figure 5.3](image1.png) Different damage shapes from test

![Figure 5.4](image2.png) Different damage shapes from simulation

5.2 Effectiveness of the submodeling method

The solution uniqueness of the elasto- and plastodynamic boundary-value problems can be utilized here to give a brief illustration of the effectiveness of the submodeling method with the strict proof being omitted. The detailed proof of the uniqueness of the solution of elastodynamic boundary-value problem based on energy
consideration can be found in the book titled *Wave Propagation in Elastic Solids* by Achenbach (1984) and *Elastodynamics* by Zhang (1988). And the proof of the uniqueness of the solution of plastodynamic boundary-value problem is available in *Plastodynamics* by Yang and Xiong (1984). Using the elasto-dynammic case as an example, the elastodynamic problem can be given as followed:

The wave equation of elastodynamics in domain $V$ (see figure 5.5) is:

$$
\sigma_{ji,j} + F_i = \rho \ddot{u}_i = \rho \partial_t u_i, \quad \text{in } V
$$

(5.1)

with the boundary conditions on the boundary $S$ of $V$ given by

$$
u_i = U_i(x_j, t) \quad \text{on } S_1
$$

(5.2)

$$
t_{ji}n_j = t_i(x_j, t) \quad \text{on } S - S_1
$$

(5.3)

Under condition (5.2) and (5.3), the solution of equation (5.1) is unique with the displacement solution denoted by

$$
u_i = U_i^*(x_j, t) \quad \text{in } V
$$

(5.4)

![Figure 5.5 Submodeling illustration](image.png)
Particularly on an arbitrarily given surface $S_{\text{inter}}$ in $V$, the displacement solution is known and from (5.4) denoted by

$$u_i = U_{i\text{ inter}}(x_j,t) \quad \text{on} \quad S_{\text{inter}}$$ \hfill (5.5)

The solution of (5.1) in $V_{\text{inter}}$ formed by $S_{\text{inter}}$ must also be the solution of

$$
\begin{align*}
\sigma_{ji,j} + F_i &= \rho \ddot{u}_i = \rho \partial_t u_i, \quad \text{in} \quad V_{\text{inter}} \\
u_i &= U_{\text{inter}i}(x_j,t), \quad \text{on} \quad S_{\text{inter}}
\end{align*}
$$ \hfill (5.6)

(5.7)

This is because expression (5.5) is both the solution of equation (5.1) and boundary value of equation (5.6). Up to now, the effectiveness of submodel method is shown assuming the $V_{\text{inter}}$ is the submodel boundary with regard to the global model in $V$. It is obvious that, in the domain $V_{\text{inter}}$, both equation (5.1) and (5.6) has the identical solution. This explains that, if the submodel takes exact the same part of global model it will have the identical solution as the global model with the same boundary and loading conditions.

Now the trick to play is, with the same boundary and loading condition, to change the portion of the submodel to a different mesh (usually more refined), geometry (usually more detailed) and possibly with different material properties (usually more sophisticated). The circular plate impact problem shown soon below with a sliced strip passing through the impact zone as the submodel demonstrates the validity of the submodeling method.

5.3 Assumptions and approximations

From the effectiveness proof of the submodeling method in section 5.2, it is seen that the only assumption or approximation made by the method is actually that the detailed modeling of the local region has negligible effect on the overall solution. From
the discussion below, this effect can be minimized by paying careful attention to modeling.

5.4 Measures and cautions for the effectiveness of submodeling method

The global model for a submodeling analysis must define the submodel boundary response with sufficient accuracy. It is the analyst’s responsibility to ensure that any particular use of the submodeling technique provides physically meaningful results. In general, the solution at the boundary of the submodel must not be altered significantly by the different local modeling. Unfortunately, it is not possible to embed a standard check of this criterion that works unanimously in an FEA code like Abaqus; it is a matter of judgment on the user’s part.

5.5 Submodel analysis of a circular plate under impact

To demonstrate how to evaluate a submodel analysis, the analysis of a circular plate as shown in figure 5.6 was conducted. The plate was made up of six layers’ T700/E862 TDTBC impacted at its center by a rigid projectile at various speeds, say, 100 m/s. Actually, this was an FEA model that can be used to simulate the impact tests conducted in the NASA Glenn Research Center Ballistic Impact Laboratory, which are described in detail in by Pereira et al. (2010). The simulation was originally done by Goldberg and Blinzler (2010) using LS-DYNA. Here, for the sake of demonstrating the submodeling method, the model was rebuilt using Abaqus with macroscale approach and the braiding through thickness integration point scheme (Cheng 2006, Li 2010). The plate used in the impact tests were modeled using a finite element mesh 305 mm wide (L), 305 mm tall and 3.175 mm thick with 2700 shell elements, as the global model. The finite element model of the panel, including the circular boundary conditions of diameter
D=254 mm that were applied to simulate the experimental conditions, is shown in Figure 5.6. The submodel, as shown in figure 5.6, was the central strip of one third of the total width L, sliced from the global model as described above. The boundary response on submodel was driven from the results of the full plate analysis. For this example, same material model and mesh density were used as a special case to demonstrate the method.

Later in chapter 6, this model will be revisited when impact speed is high enough to cause material failure. There, a comprehensive study for the same problem will be conducted with a more sophisticated model built with the mesomechanical modeling approach newly developed in this study.

Figure 5.6 Circularly clamped TDTBC plate impacted at 100m/s

In general, the accuracy of a submodel analysis can be evaluated by comparing contour plots of important variables around the boundaries of the submodel region with that of the global model. Figure 5.7 shows the final displacement contour obtained from the global and submodel analysis. Excellent agreement was reached showing the submodel analysis result was correct.
The displacement history of some critical locations, such as the impact point and the projectile, could also be used to check the accuracy of a submodel analysis. As shown in figure 5.8 and 5.9, identical results are obtained between the submodel and the global model, showing the effectiveness of the method.

Figure 5.7 Vertical Displacement of a circularly clamped TDTBC plate impacted at 100m/s

Figure 5.8 Comparison of critical displacement of the circular plate - impact point
5.6 Submodel analysis of a square plate under impact

As another example to demonstrate the dynamic submodeling method, a square plate made up of 6 layers’ T700/PR520 TDTBC, impacted at its center by a rigid projectile at 100 m/s, was analyzed. This time the analyses were conducted for several different submodels. The global model and submodels used are shown in figure 5.10. Various submodels were chosen, either as sliced patches of different shapes and sizes as shown in figure 5.10, or adopting different material or modeling approaches.

Figure 5.11, 5.12 and 5.13 show the displacement contours obtained from the global and submodel analysis at instant of 0.805, 0.965 and 0.1 millisecond after impact, respectively. For clarity, it is necessary to have a brief note for each of the different modeling approaches used in the model and termed in the pictures. “Aligned Braided” refers to the macroscale approach of “braiding through the integration point” originally developed by (Cheng 2006); “Randomly Braided” refers to that extended by Littell
in consideration of the possible random position shifting in real stacking of fiber tows; the “Smeared” refers to the macroscale modeling approach proposed by Blinzler (2012); “Refined mesh” refers to the case in which same modeling approach was used but with a refined mesh; “Meso” refers to the new mesoscale approach developed in this research. Good agreement was reached when the same mesh and modeling approach were used showing the submodel analysis result was reliable and correct. It is interesting to notice that some disparate local responses were found close to impact center for different submodels, as shown in figure 5.12 and 5.13. This is reasonable and just shows the need of detailed modeling in the critical regions of the analysis object to correctly capture the physics. Generally, the mesomechanical model and those with finer mesh would lead to solution of higher accuracy. The matching contour between the global model and submodel near the submodeling boundary as shown in these pictures indicates the reliability of the solution, despite their disparity in the critical regions.

The displacement history at the impact point and the projectile center are shown in figure 5.14 and 5.15, respectively. Very good consistent results were obtained between the submodels and the global model, showing the effectiveness of the submodeling method.

Figure 5.10 Impact of a clamped square TDTBC plate and the submodels
Figure 5.11  Vertical displacement at 0.805 ms

Figure 5.12  Vertical displacement at 0.965 ms

Figure 5.13  Vertical displacement at 1.0 ms
Figure 5.14  Comparison of impact point displacement of the square plate

Figure 5.15  Comparison of projectile displacement of the square plate
When submodeling is used in dynamic impact problems, a sufficiently large number of output frames needs to be written to the output database or results file for the global model. A high frequency of output is necessary in particular for problems dealing with elastic materials that involve elastic impact to avoid possible aliasing (under sampling), which can cause solution distortion in the submodel. In order to prevent excessive vibration exhibited in these problems, the displacement results for the nodes that are used to drive the submodel should be saved for each increment. Problems with significant plastic deformations or material failure/damage require less frequent global output since oscillations in the displacements are greatly reduced. We can see this later from the application examples in chapter 6. Even for the cases in which the frequency at which the displacement responses on the driving boundary was saved was low by using large time intervals, the solution was still satisfactory, thanks to the material plasticity, damage and failure involved. Practically, this is extremely advantageous since these models are usually huge; limiting the output would save the computer resources and time for post processing.

5.7 Basics for choosing location of the submodel boundary

Deciding the actual location of the local boundary requires some engineering judgment. The submodel boundary should be far enough away from the area of the submodel where the response is changed by the different modeling. That is, the solution at the boundary should not be altered significantly by the different local modeling (Saint Venant 1855). The Saint-Venant's principle allows elasticians to replace complicated stress distributions or weak boundary conditions into ones that are easier to solve, as long as that boundary is geometrically short (von Mises, 1945). Now we need to extend this
principle to the scope of dynamics with nonlinearity, which has been found still valid in both practical and theoretical aspects. Experimental support and numerical simulations were regarded by several researchers as evidence (Karp, 2005; Karp and Durban, 2011). Obviously, in order to get stable solutions, the locations where damage or material removal might happen should be carefully avoided being chosen as the driven boundary of a submodel. And more rigorously, the submodel boundary should be chosen as far as possible away from these sensitive regions.

5.8 Special cares for material damage or failure

When using the submodeling method for the cases in which material damage or failure is involved, special care and concerns are needed to ensure correct solutions if progressive damage mechanism is employed.

For the same analysis shown in Figure 5.6, when the impact velocity increases such that the penetration and damage happen in the center of the plate, the boundary effects become observable. Figure 5.16 showed the case when impact speed was 500 m/s. Some localized change in damage patterns was observed when the mesh size is not sufficiently small. The applicability of the submodeling method for damage would be more empirical and expedient as the stringent proof of the method becomes very challenging if not impossible. To reduce the computational cost and minimize the resource need, it is still attractive as an effective method and thus has the potential to be widely used because of its practical value in engineering applications when solving extraordinary large impact problems (even involving fracture and damage) is mandatory. Special treatment is required to minimize the noises in submodel to increase the accuracy and applicability.
One of those factors needing special attention is the mesh sensitivity. The compatibility of the mesh with the composite material defined becomes important and dominant for obtaining reliable solutions. More specifically, the mesh needs to be refined sufficiently for the given material damage property. A too coarse mesh would cause the stresses to drop to zero instantaneously as soon as the damage initiation criterion is reached. This introduces a significant amount of dynamic noise in the solution and makes the problem very sensitive because the behavior is extremely brittle – an elastic Hashin damage model is adopted. Furthermore, the model will dissipate more energy than the damage energy specified in the model. In order to ensure that the stresses are degraded in a progressive manner and that the model dissipates the correct amount of energy, the element length must be smaller than a critical length which is approximately given as:

\[ L_c = \min \left( \frac{2 E_1 G_{ft}}{(X^t)^2}, \frac{2 E_1 G_{fc}}{(X^c)^2}, \frac{2 E_2 G_{mt}}{(Y^t)^2}, \frac{2 E_2 G_{mc}}{(Y^c)^2} \right) \]

The element length of the mesh should be less than \( L_c \). If this condition is violated, then the elastic (stored) energy of the material at the onset of damage is already greater than the specified damage energy and the stresses will be degraded instantaneously causing the instability in solution.

Using one of the material definitions in the model shown in figure 5.6, the critical length was computed to be \( L_c \approx 0.024 \), whereas the mesh is of the model has a length of about 0.2, which is much larger than the critical value.

An alternative way to show this is simply modifying all the damage evolution definitions by multiplying the values of damage energy dissipation. This would also resolve the problem reasonable and allowable. For instance, increasing the damage
energy dissipation by a factor of 10 would also increase the value of $L_c$ by a factor of 10, which should now be compatible with the element length of the original mesh. It is verified that the new results of the global model and submodel are much more consistent after this modification, though not identical.

Figure 5.16  Vertical Displacement of a circular clamped TDTBC plate impacted at 500 m/s

5.9 Cause and preventative measures for mesh sensitivity on material failure

Figure 5.17 illustrates the damage initiation and progress mechanism for the composite materials modeled with macroscale approach used in this study. This is not a problem for the mesomechanical modeling approach proposed in this research. For the cases in which mesomechanical approach was used, brittle failure was employed and the numerical instability was controlled by the damping stress factors (for details refer to section 4.4.2). From figure 5.17, it should be expected that equivalent displacement $\delta_{eq}^f$ at which the material is completely damaged is greater than the displacement at the onset of damage initiation $\delta_{eq}^0: \delta_{eq}^f \geq \delta_{eq}^0$. 

110
Figure 5.17 Damage initiation and progress mechanism for composite material

Note that the fracture energy $G_f$ is the area under the triangle:

$$G_f = \frac{1}{2} \sigma_{eq}^0 \delta_{eq}^f \Rightarrow \delta_{eq}^f = 2 \frac{G_f}{\sigma_{eq}^0}$$

If the composite material is loaded uniaxially along the fiber tension direction, then

$\sigma_{eq}^0 = X^T$, $G_f = G_{ft}$, and $\delta_{eq}^f = 2 \frac{G_{ft}}{X^T}$. Similarly $\delta_{eq}^e = L^e X^T / E_1$, where $L^e$ is the characteristic element length and $E_1$ is the corresponding elastic modulus. The condition $\delta_{eq}^f \geq \delta_{eq}^0$ becomes:

$$2 \frac{G_{ft}}{X^T} \geq L^e X^T / E_1$$

Or

$$L^e \leq \frac{2 E_1 G_{ft}}{(X^T)^2}$$

Therefore the element length should be smaller than this critical value. We can make similar arguments for the case in which the material is loaded uniaxially along the fiber
compression or matrix tension/compression directions, leading to the condition $L_e \leq L_C^e$, where the critical element length $L_C^e$ is defined as

$$L_C^e = \min \left( \frac{2E_1G_{ft}}{(X^r)^2}, \frac{2E_1G_{fc}}{(X^c)^2}, \frac{2E_2G_{mt}}{(Y^r)^2}, \frac{2E_2G_{mc}}{(Y^c)^2} \right)$$
CHAPTER VI

APPLICATION EXAMPLES

To demonstrate and verify the meso-mechanical modeling methodology developed for dynamic impact analysis of TDTBC in this research, two application example problems were analyzed with the newly developed technique. In addition, the CMM approach developed with submodeling technique was also employed to show the efficiency and savings in solving large-scale structures made up of TDTBC. Comparisons were performed with experimental results where appropriate, and between the full model analysis and the submodel analysis as well.

6.1 Simulation of the axial tensile test of a simple specimen

Zhang (2013) performed the analysis of a single layer specimen tensile test with a mesoscale model, and good agreement with test was obtained. To reduce the model size, only two unit cells were included in the model with periodic boundary conditions applied to mimic the long size specimen. Meanwhile, Li (2010) used a three cell mesoscale model to simulate the tensile test of the entire straight side coupon made of six TDTBC plies and obtained satisfactory results. Symmetric boundary conditions and prescribed displacement were applied to approximate the actual full length specimen that has 720 unit cells with an FEA model of very limited size.
The highly efficient meso-mechanical model newly developed in this research made it possible to model the full six-layered specimen with the mesoscale FEA model. Alternatively, with the availability of the combined multiscale modeling (CMM) approach implemented using the submodeling technique; more accurate simulation for the same tests could be done faster and with higher reliability and fidelity. A global analysis can be done first with well-developed macroscale modeling approaches such as the “braided through the thickness integration points” approach (Cheng, 2006; Littell, 2008) or the homogenized method (Goldberg and Blinzler 2010). Within the scope of the submodeling, the global solution was used to drive the boundary of the submodels modeled by mesoscale approach to reach the boundary condition closer to the real test situations, instead of having to use prescribed boundary conditions (Li 2010, Zhang 2013). Certainly, the global model may take any simpler model with fewer details to achieve the computational savings compared with the local submodel. It may have the geometry of the same complexity as the submodel but using a simpler material model (or vice versa) or both geometry and material model are much less complicated as in the case of a macroscopic modeling. Optionally, the TDTBC model could be a 3D solid model without considering damage or debonding, and is not necessarily a macro-scale model. The best combination of the modeling approaches employed for the global model and submodel would be a trade-off between the accuracy need and the computational cost.

6.1.1 Simulation of the full specimen model

As a verification example of the unit constituent cell based mesomechanical analysis model developed in the current study, the simulation of axial tensile quasi-static test for a straight sided specimen with six layers of T700S/E862 0°/60°/-60° TDTBC
material was conducted. The FEA models of the specimen are illustrated in figure 6.1 with an in-plane size of 37 mm width, 305 mm length, and 6 layers in thickness totaling 3.2 mm. The full and the partial mesoscale model, and the full macroscale models are also shown in the same picture. The full size specimen model was used as the baseline for comparison. The submodel was located in the middle of the full specimen, modeled with 48 unit cells of six layers, 2 cell wide by 4 cell high by 6 cell thick, corresponding to 37 mm by 20.7 mm by 3.2 mm in dimensions. The full model was modeled with 720 unit cells. The macro-scale model had 1927 elements and was modeled with homogenized method proposed by Goldberg and Blinzler (2010) and converted to an Abaqus equivalent. The location of the measured strain is shown in figure 6.2. Comparison of the nominal stress and strain curves between the simulation result of a submodel analysis with that of the full model made with both meso-scale and macro-scale model were performed and shown in figure 6.3. Very good agreement between the simulated stress-strain curve and that from experiment was reached. Particularly, it is interesting to notice that the submodel result with the full mesoscale model as the global model is very close to the result of the global mesoscale model – indicating that it is possible to approach highly accurate result with the CMM method given the driving boundary is sufficiently “precise”. Furthermore, it is notable that the submodel result from a macro global model agreed with the experimental curve better than that from a mesoscale global model, implying that a precise sophisticated submodel analysis can get very satisfactory result even though the global model is relatively less accurate or less sophisticated. This is just the attractive aspect of the CMM approach combined with the dynamic submodeling. Furthermore, it is interesting and worthwhile to notice that the mesomechanical model
presented some nonlinearity in the nominal stress and strain relations while the macromechanical model tended to give a nearly linear response. One of the reasons for this difference is likely attributed to the plasticity presented in the matrix model by the mesomechanical modeling approach, which was not captured by the macromechanical modeling approach.

Figure 6.1 Specimen simulation using CMM approach with submodeling
Figure 6.2 Test and numerical gauges used in the simulation
Finally, the damaged pattern of the submodel towards the end of the pulling is shown in Figure 6.4, in which the left picture shows the whole composite structure while the right picture shows that with the pure resin matrix removed. From figure 6.4, it is understood that the damage initiated from the surface matrix along the $60^\circ$ direction. The damage propagation in that direction resulted in the complete failure of the matrix and the enlarged damage in the fiber tows. The delaminations of the matrix and between the plies were clearly predicted with the bulged edge effects on both surface of the specimen.
6.1.2 Simulation of a quarter specimen model

The same simulation was also performed for a quarter full model for simplicity and efficiency. The quarter FEA model is shown in 6.5. Strictly speaking, it is known that the real specimen is not symmetrical in any directions. Therefore, the quarter model analysis should be just regarded as an expedient, approximate analysis. Surprisingly, it gave a very satisfactory result. This can be largely attributed to the macro-isotropic feature of the $0^\circ/60^\circ/-60^\circ$ TDTBC system. Figure 6.6 shows the locations where the quarter model stress results were compared with that of a submodel. Also, the result from the 3 unit cell model of the tensile specimen employed by Li et al (2010) was presented in the comparison. Figure 6.7 shows the comparison of the in-plane principal stress history in the axial fiber tow of the three models. The submodel result agreed with that of the quarter global model very well while the 3 unite cell model showed significant
discrepancy, even though all three models showed consistently good agreement with the test in the nominal stress and nominal strain, as shown in figure 6.12, in which the static model and the new explicit model stand for the 3 unit cell model solved using the static and explicit solver of Abaqus, respectively. This indicates that the submodeling based CMM method is capable of accurately predicting the local response details effectively. This is rational as the local displacement is no longer a prescribed displacement from assumption. The conclusion is further confirmed by comparing the in-plane principal stress (and strain) history in the bias fiber tows, as shown in figure 6.8 (and 6.10) and 6.9 (and 6.11) for the positive and negative 60° bias fiber tows, respectively. From 6.8 (and 6.10) and 6.9 (and 6.11), consistent in-plane principal stress (and strain) history were obtained for the global model and submodel while the three unit cell model continuously gave discrepant result. The result reveals that, with the correct submodeling boundary response obtained from the global model, the submodel analysis combined with the CMM approach is capable of providing not only the correct overall stress-strain relations on an averaged sense but the reliable local response details as well.

Figure 6.5 Quarter global model
Figure 6.6 Stress locations used for comparison in the quarter global model

Figure 6.7 Principal stresses in the axial fiber tow
Figure 6.8 Principal stress in positive $60^0$ bias tows

Figure 6.9 Principal stresses in negative $60^0$ bias tows
Figure 6.10  Principal strain in positive $60^\circ$ bias tows

Figure 6.11  Principal strain in negative $60^\circ$ bias tow
6.2 Ballistic impact of a circular plate

To verify the method established in this study, that is, the unit constituent cell based meso-mechanical model for TDTBC analysis modeling approach and the submodeling based CMM approach, systematic simulations were performed to seek for the dynamic response of a clamped circular panel plate made of the T700s/E862 material system under impact. A series of simulations were conducted for this plate modeled with different approaches, including a full meso-scale model, and a partial meso-mechanical model – the submodel. The full meso-scale model with a coarse mesh also served as the global model to drive the boundary response for the submodel analysis.

6.2.1 The analysis model and general information

In the numerical analysis model described previously, the flat panels were modeled using a 3.2 mm thick circular shaped finite element mesh of diameter 254 mm.
The clamped portion of plate in real test coupon was purposely excluded from the model in order to reduce the model size to make analysis as efficient as possible. Fully clamped boundary was imposed to the circular edge circumferentially. This means that the circumference of the circular plate was fully fixed in all the three translational directions to approach the real clamping effect. By doing so, the model size was substantially reduced without degrading the analysis result. There were no concerns of the model size when the simulation was using a macroscale model. However, when a meso-scale model was adopted like in the current situation, any means to reduce the total number of elements in the analysis model should be valued. A circular central part of the plate with a diameter of 111 mm was chosen as the submodel, which had 6 unit cells in horizontal direction.

This structure was chosen because it is representative, and the corresponding experimental results are available (Boeing, 2007). The test was conducted in the NASA Glenn Research Center Ballistic Impact Laboratory and reported in detail by Pereira et al. (2010). In the test, a single stage compressed-gas gun was used to propel an aluminum 2024 projectile into a 305 mm by 305 mm by 3.2 mm composite plate. The composite plate was held in a circular fixture with an aperture of 254 mm. The projectile was a thin-walled hollow AL 2024 cylinder with a nominal mass of 50 gm and a front face with a compound radius. The overall length of the projectile was 49.5 mm, and the nominal diameter was 50.67 mm. In the FEA model the projectile was idealized as a rigid body considering that it is relatively stiff compared to the composite plate. Another reason for this simplification was the need for the accurate and convenient tracking of the projectile.
in the simulation result. It is known that the motion of a rigid body was fully controlled by its reference point that has merely six degrees of freedom.

Tests were performed over a range of impact velocities to determine the velocity for the onset of damage, the growth of damage with increasing velocities, the penetration threshold, and the damage pattern induced by penetration. Twelve panel tests were conducted using impact velocities ranging from 157 m/s to 175 m/s. The threshold velocity for penetration was between 161 m/s and 168 m/s.

The simulated final configuration was compared with the analysis results from the macroscale model reported by Goldberg and Blinzler (2010) and that from the test reported by Pereira et al. (2010). In the submodel analysis, the global model whose result was used to drive the submodel boundary was also built with a mesomechanical model approach. The reason for this is not only because such a detailed model was available anyhow as the full model analysis, but more importantly, the similar meshes made of the same type of elements would help reduce the noise on the submodel boundary and deviation in the solution due to the reduced wave reflection from shape change. The global model used a coarse mesh without consideration of the delamination in the material compared with the submodel though. The damage details and patterns were also compared with the real impact tests conducted in the NASA Glenn Research Center Ballistic Impact Laboratory and reported in detail by Pereira et al. (2010).

The full meso-scale model and the submodel used in the simulation are shown in figure 6.13. The full (global) model was modeled with 3.94 million solid elements with reduced integration. The model had 20.94 million variables, which is the sum of the total
DOF number plus the maximum number of all the Lagrange multiplier involved in the analysis system.

Figure 6.13 Mesomechanical full model and submodel of the circular coupon plate
The center part of the circular plate of diameter of 111mm, which has 6 (horizontal) cells by 20 (axial) cells by 6 (thickness) cells in conjunction with the projectile, was chosen as the submodel (as shown in figure 6.13 and 6.14). The partial mesoscale model, i.e. the submodel, had 717 k same type of elements and 3.815 million variables to solve for. So the size of the submodel is only about 18% of that of the global
model while they produced the identical simulation result. As a result, the typical CPU time elapsed by the submodel analysis was reduced to 6 hours with 64 CPUs, a substantial saving compared with the 43 hours needed by the corresponding global analysis run with the same number of CPUs. The advantage of using submodeling analysis is evident and enormous when a detailed local response and behavior is desired for a very large structure. An overall comparison in the model size and computer resource needed between the full mesoscale model analysis and the submodel analysis is summarized in table 6.1.

Table 6.1 Full meso model and submodel comparison of circular plate impact analysis

<table>
<thead>
<tr>
<th></th>
<th>Element number</th>
<th>Variable number</th>
<th>Unit cell number</th>
<th>Response time (sec.)</th>
<th>Stable time limit (sec.)</th>
<th>CPU time (hours)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full model</td>
<td>3,940 k</td>
<td>20,940 k</td>
<td>3160</td>
<td>8.5e-4</td>
<td>1.4e-8</td>
<td>43</td>
</tr>
<tr>
<td>Submodel</td>
<td>717 k</td>
<td>3,815 k</td>
<td>720</td>
<td>8.5e-4</td>
<td>1.4e-8</td>
<td>6</td>
</tr>
</tbody>
</table>

The full mesoscale panel was modeled with 48 unit cells in axial or the zero-tow direction, 14 unit cells horizontally and 6 cells in thickness direction, totaling 3160 cells as shown in figure 6.13 in full and 6.14 with matrix moved. Detailed architecture and geometry of the composite braid geometry (i.e. the fiber-tows in three directions, the matrix and the bonding interface) were modeled using the approach as described in Chapter 4. For simplicity, as mentioned earlier, the projectile was modeled as a rigid body because it is much stiffer than the composite plate, although it would take little effort to turn it into a deformable body to account for the elastic deformation of the projectile. Another reason for modeling it as rigid was that a deformable projectile would decrease the stable time limit drastically, making the running time inappropriately and
unnecessarily long. The density of the projectile in the simulation was taken directly from the used by Goldberg and Blinzler (2010). The general contact algorithm of Abaqus was used to simulate the arbitrary contact in the system (DS Simulia, 2014). Specially, the internal surfaces inside the plate were defined and included in the contact scheme to account for the possible perforation and penetration of the projectile into the plate. Surface based cohesive constraint was used to model the bonding and possible debonding between the fiber-tows and the matrix, between the fiber tows themselves, and between all the 6 layers of the composite braids through the thickness.

6.2.2 Simulation result

Simulations were run for various impact velocities above and below the penetration threshold in an attempt to capture the accurate threshold speed: 50 m/s, 80m/s, 130m/s, 150m/s, 160m/s, 200m/s etc. Goldberg and Blinzler (2010) defined the penetration threshold in the simulations as the velocity at which the projectile fully penetrated the plate with a residual projectile velocity close to zero. The current mesoscale model predicted the penetration taking place at an impact velocity of 150 m/s, which was only 6% less than that predicted by Goldberg and Blinzler (2010) and the experimental threshold velocity of 161 to 168 m/s (Pereira et al, 2010). The reason for the discrepancy could be related to the fact that quasi-static properties were utilized to compute the equivalent strength parameters of the tows applied in the analysis (Goldberg and Blinzler 2010). Although the strain rate sensitivity of the matrix were included in the current analysis model, the properties of the fiber-tows were also likely a function of strain rate because of the resin impregnated inside, with the various failure stresses and strains being higher at higher strain rates.
The first thing to check after the completion of an explicit dynamic analysis is the energy variations over time in the system. An energy balance equation can be used to help evaluate whether a simulation yielded an appropriate response. In an explicit FEA code, the energy balance equation is written as:

\[ E_I + E_{VD} + E_{FD} + E_{KE} - E_W = E_{TOT} = \text{constant}, \]

Where \( E_I \) is the internal energy (the sum of all elastic and inelastic strain energy, including the energy consumed by the plasticity and damage); \( E_{VD} \) is the energy absorbed by viscous dissipation; \( E_{FD} \) is the frictional dissipation energy; \( E_{KE} \) is the kinetic energy; \( E_W \) is the work done by the external forces, and \( E_{TOT} \) is the total energy in the system.

Figure 6.15-6.17 show the energy history in the dynamic system for different impact speeds: 50, 80 and 150 m/s. It is shown that, for all the 3 analysis cases, the total energy (ETOTAL) in the system basically remained constant as expected. And the kinetic energy (ALLKE) was gradually converting into internal energy (ALLIE) as the impact went on. It is interesting to note that a symmetric line (marked by a dashed line in the figures) exists between the internal energy curve and the kinetic energy curve for each analysis case, indicating all the analysis results were basically reasonable and reliable.
Figure 6.15 Energy history: initial 50 m/s. full model - Circular plate of T700s/E862

Figure 6.16 Energy history: initial 80 m/s. Circular plate of T700/E862
Figure 6.17 Energy history: initial 150 m/s. Circular plate of T700/E862

A diagram of the simulated impact damage patterns from both the full model and the submodel at the impact speed of 150 m/s is shown in figure 6.18. Figure 6.19 shows a photograph of the damage patterns obtained in a panel test impacted near the threshold velocity (Pereira et al, 2010). Goldberg and Blinzler (2010) performed the same simulation using a macroscale model. The damage pattern from their simulation is also shown in figure 6.19 for comparison. It is evident that the result from the current mesoscale model captured more details of the damage pattern and damage progress process than the macroscale model did. This was expected because with the current model, the detailed damage initiation and progress mechanism and the delamination in the composite architecture become predictable, and the interaction between the components and the constituent-wise damage were modeled in details. More detailed discussion on the simulated damage initiation and propagation in the composite at different impact speed will be done in later part of this section.
Figure 6.18  Damage pattern, at impact speed 150 m/s, circular plate of T700/E862

Figure 6.19  Damage pattern, macroscale model and test result, Circular plate of T700/E862
It was reported by Goldberg and Blinzler (2010) that an unsymmetrical damage pattern was observed in the simulation results due to the fact that the projectile rotated during the simulations and did not impact exactly head on. The same projectile rotation had also been observed in the experiments (Pereira et al, 2010). To clarify this and investigate the profound underlying causes, in the simulations of this study, the projectile was controlled such that no rotation had happened during the impact. This was realized by only giving the projectile the initial velocity in impact direction with all the rest degrees of freedom of the projectile (both translational and rotational) fully constrained. Thanks to the rigid body idealization of the projectile, the implementation of this control became very straightforward. It is interesting to note that, even with this strictly controlled motion of the projectile, the final damage patterns were consistently non-symmetric for all cases, revealing that the unsymmetrical pattern was primarily due to the physics involved, instead of the tilted projectile during the impact. Actually, these asymmetrical damage patterns were expected when the local dynamic instability like damage was involved, even if the structure was ideally symmetric, let alone the TDTBC is NOT symmetric. The material fracture and damage can easily lead to a bifurcation or local instability of other form. That being said, the oblique of the projectile certainly would increase the asymmetry of the response and damage pattern.

In the experimental impact tests, failure initiated along the axial fiber tows in the center of the specimen and then propagated outward along both the axial and the bias fiber tows. The simulated results by both the full and submodel agreed well with this phenomenon. In figure 6.20–6.21 and 6.22, a series of consecutive patterns of the predicted damage progress shown sequentially for the impact speed at 80 m/s and 150
m/s, respectively. It is clear that the failure initiated along both the axial and the bias fiber tows around the borders of the damaged area, then propagated in both the positive and negative $60^\circ$ directions and vertically near the center of the damage area until finally the penetration happened. However, in the simulation, a more rounded opening than that seen in the experimental tests was created. It is possible this difference was due again to the use of quasi-static properties. Or the damage evolution energy defined for the matrix was not large enough. Further research needs to be done to investigate both the effects of strain rate and the damage evolution energy as influential material properties to the damage patterns.

For the case of impact speed at 80 m/s, the crack and fracture was produced but the impact energy was not sufficiently large to make the projectile penetrate the plate. From figure 6.20 and 6.21, it is interesting to notice that the fracture was first generated in the front surface of the plate while there were no any signs of crack on the back up to 0.5 millisecond of the response after impact. However, as the fracture propagated and enlarged, even more severe fracture occurred on the back surface in both axial and bias tows directions. The complex stress wave propagation and reflection contributed to this interesting observation.
Crack initiated along the +bias direction
at $t = 0.02$ ms

Crack propagated in 3 tow directions
at $t = 0.05$ ms

Crack developed in –bias direction
at $t=0.33$ ms

Final damaged pattern
at $t=0.84$ ms

Figure 6.20  Front view of damage propagation, impact speed 80 m/s
No crack initiation signs on the back
at $t = 0.02$ ms

More crack developed in back
at $t = 0.33$ ms

Final damaged pattern
at $t = 0.84$ ms

Figure 6.21 Back view of damage propagation, impact speed 80 m/s, circular plate of T700/E862
Figure 6.22  Damage propagation, impact speed 150 m/s, circular plate of T700/E862
As mentioned earlier, a series of simulations for various impact velocities were performed in an attempt to capture the penetration threshold: 50 m/s, 80 m/s, 150 m/s, 160 m/s, 200 m/s. From these simulation results, it was concluded that 150 m/s was the threshold for this particular design. 6% discrepancy was observed compared with the experiment. A few factors contribute to this discrepancy, among them are the material rate sensitivity, accuracy of the properties of the individual constituents and the interlaminar and intralaminar bonding strength.

6.2.3 Validation of the CMM approach with submodeling

The current impact simulation and analysis for the circular panel also serve the purpose of validating the proposed CMM approach in this study based on the dynamic submodeling technique. A comprehensive discussion in this regards is listed below:

1) The final shape of the deformed configuration and damage pattern at different impact speed of 80 and 150 m/s are shown, respectively, in figure 6.23 and 6.24 for both the full scale model and the partial submodel. Good agreement between the configurations from the full model and the submodel shows the effectiveness of the method employed. Very slight discrepancy in the final damage pattern can be attributed to the mesh sensitivity and the numerical round-off.

2) The velocity, displacement and acceleration history of the projectile at different impact speeds (50, 80 and 150 m/s) are shown in figure 6.25, 6.26 and 6.27, respectively, for both the full scale model and the partial submodel. Excellent agreement shows again the method is effective and reliable.
Figure 6.23  Final damage pattern, impact speed 80 m/s, Circular plate of T700/E862
Figure 6.24 Final damage pattern, impact speed 150 m/s, circular plate of T700/E862
3) The velocity and displacement history of the locations on the back surface along the radial direction from the impact center to the clamped edge of the plate at different impact speeds (50 m/s, 80 m/s and 150 m/s) are shown in figure 6.28, 6.29 and 6.30, respectively, for both the full scale model and the partial submodel. Except for the locations where damage happened, generally very good agreement was obtained. At locations where failed elements were removed due to the material damage, the instability made the response very sensitive and unstable; therefore, the results from the full model and a submodel were expected to be somewhat different while the overall response between the two consistently correlated very well.

4) The velocity and displacement history on the front and the back surface of the plate corresponding to the impact center are shown in figure 6.31, 6.32 and 6.33 for both the full scale model and the partial submodel, respectively. As touched earlier, once material failure was involved like in the cases of impact speed equal to 80 m/s and 150 m/s, the discrepancy were gross as expected. When no damage or limited local damage happened, excellent agreement was reached.

The above listed results also evidently show that the identical responses were obtained at important locations such as the projectile, impact area etc. from the global full model analysis and the partial submodel model analysis, although slight difference in the final damage pattern were observed. This is because the submodel and global model adopted different mesh density and the removal of failed material causing slight local difference. This was expected as a result of the discretization and the numerical round-off
which may have magnified the discrepancy; therefore, a double precision was recommended and used to minimize the error.

Figure 6.25 Out-of-plane displacement, velocity and acceleration history of the projectile, impact speed 50 m/s, circular plate of T700/E862

Figure 6.26 Out-of-plane displacement and velocity history of the projectile, impact speed 80 m/s, circular plate of T700/E862
Figure 6.27 Out-of-plane displacement, velocity and acceleration history of the projectile, impact speed 150 m/s, circular plate of T700/E862

Figure 6.28 Out-of-plane displacement history along the radial line, gauge point 1-4: impact speed 50 m/s, circular plate of T700/E862
Figure 6.29 Out-of-plane displacement history along the radial line, gauge point 1-4: impact speed 80 m/s, circular plate of T700/E862

Figure 6.30 Out-of-plane displacement history along the radial line, gauge point 1-4: impact speed 150 m/s, circular plate of T700/E862
Figure 6.31 Out-of-plane displacement history of the impact front and back point, 
Impact speed 50 m/s, circular plate of T700/E862

Figure 6.32 Out-of-plane displacement history of the impact front and back point, 
Impact speed 80 m/s, circular plate of T700/E862

Figure 6.33 Out-of-plane displacement history of the impact front and back point, 
Impact speed 150 m/s, circular plate of T700/E862
CHAPTER VII

SUMMARY AND FUTURE WORK

The purpose of this chapter is to conclude the work presented in the dissertation as a whole, to highlight the significant findings and novelty technologies, to present ongoing work, and to address the difficulties of the present research. Based on the methods and data presented in this work, recommendations for future research in this field are presented to suggest the right way for fully understanding the complex performance of the triaxially braided composites, and its related test and simulation technologies.

7.1 Summary

The work presented in this research outlines investigations in the numerical analysis techniques and modeling methodology of triaxial-braided composite material response and composite constituent response with finite element method. Below is a highlighted summary of the important points shown throughout the research:

1) A new meso-mechanical model was established which takes into account the strain-rate dependence on elastic properties (Young’s modulus and Poisson’s ratio), plastic properties (yield stress and strain hardening) and damage properties of the matrix material as observed from experiments. The fiber tows were also modeled with solid elements. Both the damaged and undamaged elastic material
behavior, in conjunction with the Hashin’s failure criteria, was extended to three dimension formula and implemented with a user material model via Abaqus/Explicit VUMAT. The failure model was incorporated into the material with a combined elastic-damage material model. The new model improved both efficiency and accuracy for explicit dynamic analysis.

2) The newly developed the meso-mechanical model for TDTBC needs less than 10% degrees of freedom of the currently existing mesoscale model and can be used for both static and dynamic problems for accurate and fast solutions.

3) A novel hybrid FEA methodology, termed as Combined Multiscale Modeling (CMM) approach for simulating static and impact analysis of triaxially braided composites was proposed, which combined the benefit of macromechanical and mesomechanical finite element models with the mesoscale modeled portion of the structure detailing the architecture of the braided composite with damage/delamination included explicitly. Actually, the methodology applies to any problems with disparate level of details existing for the same model.

4) The proposed CMM method in conjunction with the FEA submodeling technique captures the response feature of triaxially braided composite structure under impact accurately and flexibly with a much lower computational expense.

5) The approach enables the multiscale modeling and analysis of exceedingly large scale braided composite structures subjected to high speed impact.

6) The method was shown efficient for a circular plate made up of T700/E862 composites subjected to ballistic impact and shown good correlation with experimental results.
7) Two methods for associating the mesoscale modeled region and macroscale modeled regions were discussed.

8) The submodeling method overperforms the direct connecting method in its flexibility in deciding both its detailed modeling regions and its convenience for multi-level practical applications.

9) Some application examples presented showed the efficiency of the proposed methodology.

7.2 Future work

Future efforts will focus on the investigation of the effects of strain rate-dependent material properties on the response of TDTBC material. In addition, we will continue to work on the sensitivities of the location of submodeling boundaries when applying the CMM approach.

Investigative efforts should also focus on developing more accurate and efficient algorithms for interactions amongst the multi-level modeled regions. In fact, this was the original interest of this research. However, because of the lack of a robust meso-mechanical model for the dynamic analysis of large TDTBC structures with failure, lots of effort had to be devoted to fixing this dilemma. A thorough investigation will be definitely beneficial.
REFERENCES


[27] Cater C, Xiao X, Goldberg RK, Kohlman LW. Improved subcell model for the prediction of braided composite response. NASA TM 2178752013.


APPENDIX

ABAQUS VUMAT 3D HASHIN FAILURE CRITERIA

SUBROUTINE VUMAT(
  C READ ONLY -
  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  TEMPNW, STRETCHNW, DEFGRADNW, FIELDNW,
  C WRITE ONLY -
  7  STRESSNW, STATENW, ENERINTERNNW, ENERINELASNEW )
C
  INCLUDE 'VABA_PARAM.INC'
C
C 3D ORTHOTROPIC ELASTICITY WITH HASHIN 3D FAILURE CRITERION
C
C THE STATE VARIABLES ARE STORED AS:
C  STATE(*,1) = MATERIAL POINT STATUS
C  STATE(*,2:7) = DAMPING STRESSES
C
C USER DEFINED MATERIAL PROPERTIES ARE STORED AS
C  * FIRST LINE:
C    PROPS(1) --> YOUNG'S MODULUS IN 1-DIRECTION, E1
C    PROPS(2) --> YOUNG'S MODULUS IN 2-DIRECTION, E2
C    PROPS(3) --> YOUNG'S MODULUS IN 3-DIRECTION, E3
C    PROPS(4) --> POISSON'S RATIO, NU12
C    PROPS(5) --> POISSON'S RATIO, NU13
C    PROPS(6) --> POISSON'S RATIO, NU23
C    PROPS(7) --> SHEAR MODULUS, G12
C    PROPS(8) --> SHEAR MODULUS, G13
C
C  * SECOND LINE:
C    PROPS(9) --> SHEAR MODULUS, G23
C    PROPS(10) --> BETA DAMPING PARAMETER
C    PROPS(11) --> "NOT USED"
C    PROPS(12) --> "NOT USED"
C    PROPS(13) --> "NOT USED"
C    PROPS(14) --> "NOT USED"
C    PROPS(15) --> "NOT USED"
C    PROPS(16) --> "NOT USED"
C * THIRD LINE:
C PROPS(17) --> ULTIMATE TENS STRESS IN 1-DIRECTION, SIGU1T
C PROPS(18) --> ULTIMATE COMP STRESS IN 1-DIRECTION, SIGU1C
C PROPS(19) --> ULTIMATE TENS STRESS IN 2-DIRECTION, SIGU2T
C PROPS(20) --> ULTIMATE COMP STRESS IN 2-DIRECTION, SIGU2C
C PROPS(21) --> ULTIMATE TENS STRESS IN 3-DIRECTION, SIGU3T
C PROPS(22) --> ULTIMATE COMP STRESS IN 3-DIRECTION, SIGU3C
C PROPS(23) --> "NOT USED"
C PROPS(24) --> "NOT USED"

C * FOURTH LINE:
C PROPS(25) --> ULTIMATE SHEAR STRESS, SIGU12
C PROPS(26) --> ULTIMATE SHEAR STRESS, SIGU13
C PROPS(27) --> ULTIMATE SHEAR STRESS, SIGU23
C PROPS(28) --> "NOT USED"
C PROPS(29) --> "NOT USED"
C PROPS(30) --> "NOT USED"
C PROPS(31) --> "NOT USED"
C PROPS(32) --> "NOT USED"

DIMENSION PROPS(NPROPS), DENSITY(NBLOCK),
1 COORDMP(NBLOCK,*),
2 CHARLENGTH(*), STRAININC(NBLOCK,NDIR+NSHR),
3 RELSPININC(NBLOCK,NSHR), TEMPOLD(NBLOCK),
4 STRETCHOLD(NBLOCK,NDIR+NSHR),
DEFGRADOLD(NBLOCK,NDIR+NSHR+NSHR),
5 FIELDOLD(NBLOCK,NFIELDV), STRESSOLD(NBLOCK,NDIR+NSHR),
6 STATEOLD(NBLOCK,NSTATEV), ENERINTERNOLD(NBLOCK),
7 ENERINELASOLD(NBLOCK), TEMPNEW(*),
8 STRETCHNEW(NBLOCK,NDIR+NSHR),
DEFGRADNEW(NBLOCK,NDIR+NSHR+NSHR),
9 FIELDNEW(NBLOCK,NFIELDV), STRESSNEW(NBLOCK,NDIR+NSHR),
1 STATENEW(NBLOCK,NSTATEV),
2 ENERINTERNNEW(NBLOCK), ENERINELASNEW(NBLOCK)
*
CHARACTER*80 CMNAME
*
PARAMETER( ZERO = 0.D0, ONE = 1.D0, TWO = 2.D0, HALF = .5D0 )
*
PARAMETER(
  * I_SVD_DMGFIBERT = 1,
  * I_SVD_DMGFIBERC = 2,
  * I_SVD_DGMATRIXT = 3,
  * I_SVD_DGMATRIXC = 4,
  * I_SVD_STATUSMP = 5,
C  * I_SVD_DAMPSTRESS = 6,
C  * I_SVD_DAMPSTRESSXX = 6,
C  * I_SVD_DAMPSTRESSYY = 7,
C  * I_SVD_DAMPSTRESSZZ = 8,
C  * I_SVD_DAMPSTRESSXY = 9,
C  * I_SVD_DAMPSTRESSYZ = 10,
C  * I_SVD_DAMPSTRESSZX = 11,
C  * I_SVD_DAMPSTRESSXX = 11,
C  * I_SVD_DAMPSTRESSZZ = 12,
* I_SVD_STRAIN  = 6,
C * I_SVD_STRAINXX = 12,
C * I_SVD_STRAINYY = 13,
C * I_SVD_STRAINZZ = 14,
C * I_SVD_STRAINXY = 15,
C * I_SVD_STRAINYZ = 16,
C * I_SVD_STRAINZX = 17,
CNIE * N_SVD_REQUIRED = 17 )
* N_SVD_REQUIRED = 6 )
*
PARAMETER(
  * I_S33_XX = 1,
  * I_S33_YY = 2,
  * I_S33_ZZ = 3,
  * I_S33_XY = 4,
  * I_S33_YZ = 5,
  * I_S33_ZX = 6 )
*
* STRUCTURE OF PROPERTY ARRAY
PARAMETER ( 
  * I_PRO_E1    = 1,
  * I_PRO_E2    = 2,
  * I_PRO_E3    = 3,
  * I_PRO_NU12  = 4,
  * I_PRO_NU13  = 5,
  * I_PRO_NU23  = 6,
  * I_PRO_G12   = 7,
  * I_PRO_G13   = 8,
  * I_PRO_G23   = 9,
  * I_PRO_BETA  = 10,
  * I_PRO_SIGU1T = 17,
  * I_PRO_SIGU1C = 18,
  * I_PRO_SIGU2T = 19,
  * I_PRO_SIGU2C = 20,
  * I_PRO_SIGU3T = 21,
  * I_PRO_SIGU3C = 22,
  * I_PRO_SIGU12 = 25,
  * I_PRO_SIGU13 = 26,
  * I_PRO_SIGU23 = 27 )
* TEMPORARY ARRAYS
  DIMENSION EIGEN(MAXBLK*3)
* READ MATERIAL PROPERTIES
* 
  E1 = PROPS(I_PRO_E1)
  E2 = PROPS(I_PRO_E2)
  E3 = PROPS(I_PRO_E3)
  XNU12 = PROPS(I_PRO_NU12)
  XNU13 = PROPS(I_PRO_NU13)
  XNU23 = PROPS(I_PRO_NU23)
  G12 = PROPS(I_PRO_G12)
  G13 = PROPS(I_PRO_G13)
G23 = PROPS(I_PRO_G23)

* 
XNU21 = XNU12 * E2 / E1
XNU31 = XNU13 * E3 / E1
XNU32 = XNU23 * E3 / E2

* 
* COMPUTE TERMS OF STIFFNESS MATRIX
GG = ONE / ( ONE - XNU12*XNU21 - XNU23*XNU32 - XNU31*XNU13
     - TWO*XNU21*XNU32*XNU13 )
C11  = E1 * ( ONE - XNU23*XNU32 ) * GG
C22  = E2 * ( ONE - XNU13*XNU31 ) * GG
C33  = E3 * ( ONE - XNU12*XNU21 ) * GG
C12  = E1 * ( XNU21 + XNU31*XNU23 ) * GG
C13  = E1 * ( XNU31 + XNU21*XNU32 ) * GG
C23  = E2 * ( XNU32 + XNU12*XNU31 ) * GG

* 
F1T = PROPS(I_PRO_SIGU1T)
F1C = PROPS(I_PRO_SIGU1C)
F2T = PROPS(I_PRO_SIGU2T)
F2C = PROPS(I_PRO_SIGU2C)
F3T = PROPS(I_PRO_SIGU3T)
F3C = PROPS(I_PRO_SIGU3C)
F12 = PROPS(I_PRO_SIGU12)
F13 = PROPS(I_PRO_SIGU13)
F23 = PROPS(I_PRO_SIGU23)

* 
BETA = PROPS(I_PRO_BETA)

* 
* ASSUME PURELY ELASTIC MATERIAL AT THE BEGINNING OF THE ANALYSIS
*
* IF ( TOTALTIME .EQ. ZERO ) THEN
IF ( NSTATEV .LT. N_SVD_REQUIRED ) THEN
   CALL XPLB_ABQERR(-2,'SUBROUTINE VUMAT REQUIRES THE '//' 
   'SPECIFICATION OF %I STATE VARIABLES. CHECK THE '//' 
   'DEFINITION OF *DEPVAR IN THE INPUT FILE.',
   'N_SVD_REQUIRED,ZERO,' )
   CALL XPLB_EXIT
END IF
CALL ORTHOE A3DEXP ( NBLOCK,
   STATEOLD(1,I_SVD_DMGFIBERT),
   STATEOLD(1,I_SVD_DMGFIBERC),
   STATEOLD(1,I_SVD_DMGMATRIXT),
   STATEOLD(1,I_SVD_DMGMATRIXC),
   C11, C22, C33, C12, C23, C13, G12, G23, G13,
   STRAININC,
   STRESSNEW )
RETURN
END IF
* 
* UPDATE TOTAL ELASTIC STRAIN
CALL STRAINUPDATE ( NBLOCK, STRAININC,
   STATEOLD(1,I_SVD_STRAIN), STATENEW(1,I_SVD_STRAIN) )
*
* STRESS UPDATE
  CALL ORTHOELA3DEXP ( NBLOCK,
    * STATEOLD(1,I_SVD_DMGFIBERT),
    * STATEOLD(1,I_SVD_DMGFIBERC),
    * STATEOLD(1,I_SVD_DMGMATRIXT),
    * STATEOLD(1,I_SVD_DMGMATRIXC),
    * C11, C22, C33, C12, C23, C13, G12, G23, G13,
    * STATENEW(1,I_SVD_STRAIN),
    * STRESSNEW )
*
* FAILURE EVALUATION
*
  CALL COPYR ( NBLOCK,
    * STATEOLD(1,I_SVD_DMGFIBERT), STATENEW(1,I_SVD_DMGFIBERT) )

  CALL COPYR ( NBLOCK,
    * STATEOLD(1,I_SVD_DMGFIBERC), STATENEW(1,I_SVD_DMGFIBERC) )

  CALL COPYR ( NBLOCK,
    * STATEOLD(1,I_SVD_DMGMATRIXT), STATENEW(1,I_SVD_DMGMATRIXT) )

  CALL COPYR ( NBLOCK,
    * STATEOLD(1,I_SVD_DMGMATRIXC), STATENEW(1,I_SVD_DMGMATRIXC) )

  NDMG = 0
  CALL EIG33ANAL ( NBLOCK, STRETCHNEW, EIGEN )
  CALL HASHIN3D  ( NBLOCK, NDMG,
    * F1T, F2T, F3T, F1C, F2C, F3C, F12, F23, F13,
    * STATENEW(1,I_SVD_DMGFIBERT),
    * STATENEW(1,I_SVD_DMGFIBERC),
    * STATENEW(1,I_SVD_DMGMATRIXT),
    * STATENEW(1,I_SVD_DMGMATRIXC),
    * STATENEW(1,I_SVD_STATUSMP),
    * STRESSNEW, EIGEN )
*
  -- RECOMPUTE STRESSES IF NEW DAMAGE IS OCCURRING
  IF ( NDMG .GT. 0 ) THEN
    CALL ORTHOELA3DEXP ( NBLOCK,
      * STATENEW(1,I_SVD_DMGFIBERT),
      * STATENEW(1,I_SVD_DMGFIBERC),
      * STATENEW(1,I_SVD_DMGMATRIXT),
      * STATENEW(1,I_SVD_DMGMATRIXC),
      * C11, C22, C33, C12, C23, C13, G12, G23, G13,
      * STATENEW(1,I_SVD_STRAIN),
      * STRESSNEW )
  END IF
*
* BETA DAMPING
  IF ( BETA .GT. ZERO ) THEN
    CALL BETADAMPING3D ( NBLOCK,
      * BETA, DT, STRAININC,
      * STRESSOLD, STRESSNEW,
      * STATENEW(1,I_SVD_STATUSMP),
      * STATEOLD(1,I_SVD_DAMPSTRESS),
      * STATENEW(1,I_S_V_DAMPSTRESS) )
  END IF
* INTEGRATE THE INTERNAL SPECIFIC ENERGY (PER UNIT MASS)
* CALL ENERGYINTERNAL3D ( NBLOCK, STRESSOLD, STRESSNEW,
  * STRAININC, DENSITY, ENERINTERNOLD, ENERINTERNNEW )
* RETURN
END

************************************************************
* ORTHOELA3DEXP: ORTHOTROPIC ELASTICITY - 3D             *
************************************************************
SUBROUTINE ORTHOELA3DEXP ( NBLOCK,
  * DMGFIBERT, DMGFIBERC, DMGMATRIXT, DMGMATRIXC,
  * C11, C22, C33, C12, C23, C13, G12, G23, G13,
  * STRAIN, STRESS )
* INCLUDE 'VABA_PARAM.INC'
*
** ORTHOTROPIC ELASTICITY, 3D CASE - **
** PARAMETER( ZERO = 0.D0, ONE = 1.D0, TWO = 2.D0 )
** PARAMETER( *
  * I_S33_XX = 1,
  * I_S33_YY = 2,
  * I_S33_ZZ = 3,
  * I_S33_XY = 4,
  * I_S33_YZ = 5,
  * I_S33_ZX = 6,
  * N_S33_CAR = 6 )
** DIMENSION  STRAIN(NBLOCK,N_S33_CAR),
  * DMGFIBERT(NBLOCK), DMGFIBERC(NBLOCK),
  * DMGMATRIXT(NBLOCK), DMGMATRIXC(NBLOCK),
  * STRESS(NBLOCK,N_S33_CAR)
* -- SHEAR FRACTION IN MATRIX TENSION AND COMPRESSION MODE
PARAMETER ( SMT = 0.8D0, SMC = 0.5D0 )
*
DO K = 1, NBLOCK
* -- COMPUTE DAMAGED STIFFNESS
  DFT = DMGFIBERT(K)
  DFC = DMGFIBERC(K)
  DMT = DMGMATRIXT(K)
  DMC = DMGMATRIXC(K)
  DF = ONE - ( ONE - DFT ) * ( ONE - DFC )
*  DC11 = ( ONE - DF ) * C11
  DC22 = ( ONE - DF ) * ( ONE - DMT ) * ( ONE - DMC ) * C22
  DC33 = ( ONE - DF ) * ( ONE - DMT ) * ( ONE - DMC ) * C33
  DC12 = ( ONE - DF ) * ( ONE - DMT ) * ( ONE - DMC ) * C12
  DC23 = ( ONE - DF ) * ( ONE - DMT ) * ( ONE - DMC ) * C23
  DC13 = ( ONE - DF ) * ( ONE - DMT ) * ( ONE - DMC ) * C13

169
DG12 = ( ONE - DF ) * ( ONE - SMT*DMT ) * ( ONE - SM*DM ) * G12
DG23 = ( ONE - DF ) * ( ONE - SMT*DMT ) * ( ONE - SM*DM ) * G23
DG13 = ( ONE - DF ) * ( ONE - SMT*DMT ) * ( ONE - SM*DM ) * G13

-- STRESS UPDATE
STRESS(K,I_S33_XX) = DC11 * STRAIN(K,I_S33_XX) + DC12 * STRAIN(K,I_S33_YY) + DC13 * STRAIN(K,I_S33_ZZ)
STRESS(K,I_S33_YY) = DC12 * STRAIN(K,I_S33_XX) + DC22 * STRAIN(K,I_S33_YY) + DC23 * STRAIN(K,I_S33_ZZ)
STRESS(K,I_S33_ZZ) = DC13 * STRAIN(K,I_S33_XX) + DC23 * STRAIN(K,I_S33_YY) + DC33 * STRAIN(K,I_S33_ZZ)
STRESS(K,I_S33_XY) = TWO * DG12 * STRAIN(K,I_S33_XY)
STRESS(K,I_S33_YZ) = TWO * DG23 * STRAIN(K,I_S33_YZ)
STRESS(K,I_S33_ZX) = TWO * DG13 * STRAIN(K,I_S33_ZX)

END DO
*
RETURN
END

*******************************************************************************
* STRAINUPDATE: UPDATE TOTAL STRAIN                                      *
*******************************************************************************
SUBROUTINE STRAINUPDATE ( NBLOCK,
                           * STRAININC, STRAINOLD, STRAINNEW )
*
INCLUDE 'VABA_PARAM.INC'
*
PARAMETER(
     * I_S33_XX = 1,
     * I_S33_YY = 2,
     * I_S33_ZZ = 3,
     * I_S33_XY = 4,
     * I_S33_YZ = 5,
     * I_S33_ZX = 6,
     * N_S33_CAR = 6 )
*
DIMENSION STRAININC(NBLOCK,N_S33_CAR),
            STRAINOLD(NBLOCK,N_S33_CAR),
            STRAINNEW(NBLOCK,N_S33_CAR)
*
DO K = 1, NBLOCK
   STRAINNEW(K,I_S33_XX) = STRAINOLD(K,I_S33_XX) + STRAININC(K,I_S33_XX)
   STRAINNEW(K,I_S33_YY) = STRAINOLD(K,I_S33_YY) + STRAININC(K,I_S33_YY)
   STRAINNEW(K,I_S33_ZZ) = STRAINOLD(K,I_S33_ZZ) + STRAININC(K,I_S33_ZZ)
   STRAINNEW(K,I_S33_XY) = STRAINOLD(K,I_S33_XY) + STRAININC(K,I_S33_XY)
   STRAINNEW(K,I_S33_YZ) = STRAINOLD(K,I_S33_YZ) + STRAININC(K,I_S33_YZ)
   STRAINNEW(K,I_S33_ZX) = STRAINOLD(K,I_S33_ZX) + STRAININC(K,I_S33_ZX)
END DO
*
STRAINNEW(K,I_S33_YZ) = STRAINOLD(K,I_S33_YZ)  
*                        + STRAININC(K,I_S33_YZ)  
STRAINNEW(K,I_S33_ZX) = STRAINOLD(K,I_S33_ZX)  
*                        + STRAININC(K,I_S33_ZX)  
END DO  
*  
RETURN  
END  

************************************************************  
*   HASHIN3D W/ MODIFIED PUCK: EVALUATE HASHIN 3D FAILURE  *  
*   CRITERION FOR FIBER, PUCK FOR MATRIX                   *  
************************************************************  
SUBROUTINE HASHIN3D ( NBLOCK, NDMG,  
*     F1T, F2T, F3T, F1C, F2C, F3C, F12, F23, F13,  
*     DMGFIBERT, DMGFIBERC, DMGMATRIXT, DMGMATRIXC,  
*     STATUSMP, STRESS, EIGEN )  
*  
INCLUDE 'VABA_PARAM.INC'  

PARAMETER( ZERO = 0.D0, ONE = 1.D0, HALF = 0.5D0, THREE = 3.D0  
)
PARAMETER(  
*     I_S33_XX = 1,  
*     I_S33_YY = 2,  
*     I_S33_ZZ = 3,  
*     I_S33_XY = 4,  
*     I_S33_YZ = 5,  
*     I_S33_ZX = 6,  
*     N_S33_CAR = 6 )  
*  
PARAMETER(I_V3D_X=1,I_V3D_Y=2,I_V3D_Z=3 )  
PARAMETER(N_V3D_CAR=3 )  
*  
PARAMETER ( EMAX = 1.00D0, EMIN = -0.8D0 )  
*  
DIMENSION DMGFIBERT(NBLOCK), DMGFIBERC(NBLOCK),  
*     DMGMATRIXT(NBLOCK), DMGMATRIXC(NBLOCK),  
*     STRESS(NBLOCK,N_S33_CAR),  
*     EIGEN(NBLOCK,N_V3D_CAR),  
*     STATUSMP(NBLOCK)  
*  
F1TINV = ZERO  
F2TINV = ZERO  
F3TINV = ZERO  
F1CINV = ZERO  
F2CINV = ZERO  
F3CINV = ZERO  
F12INV = ZERO  
F23INV = ZERO  
F13INV = ZERO  
*  
IF ( F1T .GT. ZERO ) F1TINV = ONE / F1T
IF ( F2T .GT. ZERO ) F2TINV = ONE / F2T
IF ( F3T .GT. ZERO ) F3TINV = ONE / F3T
IF ( F1C .GT. ZERO ) F1CINV = ONE / F1C
IF ( F2C .GT. ZERO ) F2CINV = ONE / F2C
IF ( F3C .GT. ZERO ) F3CINV = ONE / F3C
IF ( F12 .GT. ZERO ) F12INV = ONE / F12
IF ( F23 .GT. ZERO ) F23INV = ONE / F23
IF ( F13 .GT. ZERO ) F13INV = ONE / F13

DO K = 1, NBLOCK
  IF ( STATUSMP(K) .EQ. ONE ) THEN
    LFAIL = 0
    S11 = STRESS(K,I_S33_XX)
    S22 = STRESS(K,I_S33_YY)
    S33 = STRESS(K,I_S33_ZZ)
    S12 = STRESS(K,I_S33_XY)
    S23 = STRESS(K,I_S33_YZ)
    S13 = STRESS(K,I_S33_ZX)

    EVALUATE FIBER MODES
    IF ( S11 .GT. ZERO ) THEN
      -- TENSILE FIBER MODE
      RFT = (S11*F1TINV)**2 + (S12*F12INV)**2 + (S13*F13INV)**2
      IF ( RFT .GE. ONE ) THEN
        LDMG = 1
        DMGFIBERT(K) = ONE
      END IF
    ELSE IF ( S11 .LT. ZERO ) THEN
      -- COMPRESSIVE FIBER MODE
      RFC = ABS(S11) * F1CINV
      IF ( RFC .GE. ONE ) THEN
        LDMG = 1
        DMGFIBERC(K) = ONE
      END IF
    END IF

    EVALUATE MATRIX MODES
    IF ( ( S22 + S33 ) .GT. ZERO ) THEN
      -- TENSILE MATRIX MODE
      RMT = ( S11 * HALF * F1TINV)**2
      + ( S22**2 * ABS(F2TINV * F2CINV) )
      + ( S12 * F12INV )**2
      + ( S22 * (F2TINV + F2CINV) )
      IF ( RMT .GE. ONE ) THEN
        LDMG = 1
        DMGMATRIX(K) = ONE
      END IF
    ELSE IF ( ( S22 + S33 ) .LT. ZERO ) THEN
      -- COMPRESSIVE MATRIX MODE
      RMC = ( S11 * HALF * F1TINV)**2
      + ( S22**2 * ABS(F2TINV * F2CINV) )
* + ( S12 * F12INV )**2
* + ( S22 * (F2TINV + F2CINV) )
IF ( RMC .GE. ONE ) THEN
  LDMG = 1
  DMGMATRIXC(K) = ONE
END IF
END IF
*
EIGMAX = MAX(EIGEN(K, I_V3D_X), EIGEN(K, I_V3D_Y), EIGEN(K, I_V3D_Z))
EIGMIN = MIN(EIGEN(K, I_V3D_X), EIGEN(K, I_V3D_Y), EIGEN(K, I_V3D_Z))
  ENOMMAX = EIGMAX - ONE
  ENOMMIN = EIGMIN - ONE
*
  IF ( ENOMMAX .GT. EMAX .OR.
    *       ENOMMIN .LT. EMIN .OR.
    *       DMGFIBERT(K) .EQ. ONE ) THEN
    STATUSMP(K) = ZERO
  END IF
*
  NDMG = NDMK + LDMG
*
END IF
*
END DO
*
RETURN
END

*******************************************************************************
* BETADAMPING: ADD BETA DAMPING                                              *
*******************************************************************************
SUBROUTINE BETADAMPING3D ( NBLOCK,
  * BETA, DT, STRAININC, SIGOLD, SIGNEW,
  * STATUSMP, SIGDAMPOLD, SIGDAMPNEW )
  *
  INCLUDE 'VABA_PARAM.INC'
  *
  PARAMETER(
  *  I_S33_XX = 1,
  *  I_S33_YY = 2,
  *  I_S33_ZZ = 3,
  *  I_S33_XY = 4,
  *  I_S33_YZ = 5,
  *  I_S33_ZX = 6,
  *  N_S33_CAR = 6 )
  *
  DIMENSION  SIGOLD(NBLOCK,N_S33_CAR),
  *  SIGNEW(NBLOCK,N_S33_CAR),
  *  STRAININC(NBLOCK,N_S33_CAR),
  *  STATUSMP (NBLOCK),
  *  SIGDAMPOLD(NBLOCK,N_S33_CAR),
  *  SIGDAMPNEW(NBLOCK,N_S33_CAR),
  *
173
* SIGDAMPNEW(NBLOCK,N_S33_CAR)
*
* PARAMETER ( ZERO = 0.D0, ONE = 1.D0, TWO = 2.0D0,
* HALF = 0.5D0, THIRD = 1.D0/3.D0 )
* PARAMETER ( ASMALL = 1.D-16 )
*
BETADDT = BETA / DT
*
DO K = 1 , NBLOCK
SIGDAMPNEW(K,I_S33_XX) = BETADDT * STATUSMP(K) *
( SIGNEW(K,I_S33_XX)
 - ( SIGOLD(K,I_S33_XX) - SIGDAMPOLD(K,I_S33_XX) ) )
SIGDAMPNEW(K,I_S33_YY) = BETADDT * STATUSMP(K) *
( SIGNEW(K,I_S33_YY)
 - ( SIGOLD(K,I_S33_YY) - SIGDAMPOLD(K,I_S33_YY) ) )
SIGDAMPNEW(K,I_S33_ZZ) = BETADDT * STATUSMP(K) *
( SIGNEW(K,I_S33_ZZ)
 - ( SIGOLD(K,I_S33_ZZ) - SIGDAMPOLD(K,I_S33_ZZ) ) )
SIGDAMPNEW(K,I_S33_XY) = BETADDT * STATUSMP(K) *
( SIGNEW(K,I_S33_XY)
 - ( SIGOLD(K,I_S33_XY) - SIGDAMPOLD(K,I_S33_XY) ) )
SIGDAMPNEW(K,I_S33_YZ) = BETADDT * STATUSMP(K) *
( SIGNEW(K,I_S33_YZ)
 - ( SIGOLD(K,I_S33_YZ) - SIGDAMPOLD(K,I_S33_YZ) ) )
SIGDAMPNEW(K,I_S33_ZX) = BETADDT * STATUSMP(K) *
( SIGNEW(K,I_S33_ZX)
 - ( SIGOLD(K,I_S33_ZX) - SIGDAMPOLD(K,I_S33_ZX) ) )
*
SIGNEW(K,I_S33_XX) = SIGNEW(K,I_S33_XX) + SIGDAMPNEW(K,I_S33_XX)
SIGNEW(K,I_S33_YY) = SIGNEW(K,I_S33_YY) + SIGDAMPNEW(K,I_S33_YY)
SIGNEW(K,I_S33_ZZ) = SIGNEW(K,I_S33_ZZ) + SIGDAMPNEW(K,I_S33_ZZ)
SIGNEW(K,I_S33_XY) = SIGNEW(K,I_S33_XY) + SIGDAMPNEW(K,I_S33_XY)
SIGNEW(K,I_S33_YZ) = SIGNEW(K,I_S33_YZ) + SIGDAMPNEW(K,I_S33_YZ)
SIGNEW(K,I_S33_ZX) = SIGNEW(K,I_S33_ZX) + SIGDAMPNEW(K,I_S33_ZX)
*
END DO
*
RETURN
END

***********************************************************************
ENERGYINTERNAL3D: COMPUTE INTERNAL ENERGY FOR 3D CASE
***********************************************************************
SUBROUTINE ENERGYINTERNAL3D(NBLOCK, SIGOLD, SIGNEW ,
STRAININC, CURDENSITY, ENERINTERNOLD, ENERINTERNNEW)
*
INCLUDE 'VABA_PARAM.INC'

174
PARAMETER(
    I_S33_XX = 1,
    I_S33_YY = 2,
    I_S33_ZZ = 3,
    I_S33_XY = 4,
    I_S33_YZ = 5,
    I_S33_ZX = 6,
    N_S33_CAR = 6 )

PARAMETER( TWO = 2.D0, HALF = .5D0 )

DIMENSION SIGOLD (NBLOCK,N_S33_CAR), SIGNEW (NBLOCK,N_S33_CAR),
    STRAININC (NBLOCK,N_S33_CAR), CURDENSITY (NBLOCK),
    ENERINTERNOLD(NBLOCK), ENERINTERNNEW(NBLOCK)

DO K = 1, NBLOCK
    STRESSPOWER  = HALF * ( ( SIGOLD(K,I_S33_XX) + SIGNEW(K,I_S33_XX) )
        * ( STRAININC(K,I_S33_XX) )
        + ( SIGOLD(K,I_S33_YY) + SIGNEW(K,I_S33_YY) )
        * ( STRAININC(K,I_S33_YY))
        + ( SIGOLD(K,I_S33_ZZ) + SIGNEW(K,I_S33_ZZ) )
        * ( STRAININC(K,I_S33_ZZ))
        + TWO * ( SIGOLD(K,I_S33_XY) + SIGNEW(K,I_S33_XY) )
        * STRAININC(K,I_S33_XY)
        + TWO * ( SIGOLD(K,I_S33_YZ) + SIGNEW(K,I_S33_YZ) )
        * STRAININC(K,I_S33_YZ)
        + TWO * ( SIGOLD(K,I_S33_ZX) + SIGNEW(K,I_S33_ZX) )
        * STRAININC(K,I_S33_ZX) )

    ENERINTERNNEW(K) = ENERINTERNOLD(K) +
        STRESSPOWER/CURDENSITY(K)
END DO

RETURN
END

************************************************************************************
* COPYR: COPY FROM ONE ARRAY TO ANOTHER                                             *
************************************************************************************
SUBROUTINE COPYR(NCOPY, FROM, TO )

INCLUDE 'VABA_PARAM.INC'

DIMENSION FROM(NCOPY), TO(NCOPY)

DO K = 1, NCOPY
    TO(K) = FROM(K)
END DO

RETURN
END
**EIG33ANAL: COMPUTE EIGENVALUES OF A 3X3 SYMMETRIC MATRIX ANALYTICALLY**

SUBROUTINE EIG33ANAL( NBLOCK, SMAT, EIGVAL )
* 
INCLUDE 'VABA_PARAM.INC'
* 
PARAMETER(I_S33_XX=1,I_S33_YY=2,I_S33_ZZ=3 )
PARAMETER(I_S33_XY=4,I_S33_YZ=5,I_S33_ZX=6 )
PARAMETER(I_S33_YX=I_S33_XY )
PARAMETER(I_S33_ZY=I_S33_YZ )
PARAMETER(I_S33_XZ=I_S33_ZX,N_S33_CAR=6 )
* 
PARAMETER(I_V3D_X=1,I_V3D_Y=2,I_V3D_Z=3 )
PARAMETER(N_V3D_CAR=3 )
* 
PARAMETER ( ZERO = 0.D0, ONE = 1.D0, TWO = 2.D0,
    *     THREE = 3.D0, HALF = 0.5D0, THIRD = ONE / THREE,
    *     PI23 = 2.094395102393195D0,
    *     FUZZ = 1.D-8,
    *     PRECIZ = FUZZ * 1.D4 )
* 
DIMENSION EIGVAL(NBLOCK,N_V3D_CAR), SMAT(NBLOCK,N_S33_CAR)
* 
DO K = 1, NBLOCK
SH =
THIRD*(SMAT(K,I_S33_XX)+SMAT(K,I_S33_YY)+SMAT(K,I_S33_ZZ))
S11 = SMAT(K,I_S33_XX) - SH
S22 = SMAT(K,I_S33_YY) - SH
S33 = SMAT(K,I_S33_ZZ) - SH
S12 = SMAT(K,I_S33_XY)
S13 = SMAT(K,I_S33_XZ)
S23 = SMAT(K,I_S33_YZ)
* 
FAC = MAX(ABS(S11), ABS(S22), ABS(S33))
FACS = MAX(ABS(S12), ABS(S13), ABS(S23))
IF( FACS .LT. (PRECIZ*FAC) ) THEN
    EIGVAL(K,I_V3D_X) = SMAT(K,I_S33_XX)
    EIGVAL(K,I_V3D_Y) = SMAT(K,I_S33_YY)
    EIGVAL(K,I_V3D_Z) = SMAT(K,I_S33_ZZ)
ELSE
    Q =
    THIRD*((S12**2+S13**2+S23**2)+HALF*(S11**2+S22**2+S33**2))
    FAC = TWO * SQRT(Q)
    IF( FAC .GT. FUZZ ) THEN
        OFAC = TWO/FAC
    END IF
    OFAC = ZERO
    END IF
    S11 = OFAC*S11
S22 = OFAC*S22
S33 = OFAC*S33
S12 = OFAC*S12
S13 = OFAC*S13
S23 = OFAC*S23
R = S12*S13*S23
* + HALF*(S11*S22*S33-S11*S23**2-S22*S13**2-S33*S12**2)
IF( R .GE. ONE-FUZZ ) THEN
   COS1 = -HALF
   COS2 = -HALF
   COS3 = ONE
ELSE IF( R .LE. FUZZ-ONE ) THEN
   COS1 = -ONE
   COS2 = HALF
   COS3 = HALF
ELSE
   ANG = THIRD * ACOS(R)
   COS1 = COS(ANG)
   COS2 = COS(ANG+PI23)
   COS3 = -COS1-COS2
END IF
EIGVAL(K, I_V3D_X) = SH + FAC*COS1
EIGVAL(K, I_V3D_Y) = SH + FAC*COS2
EIGVAL(K, I_V3D_Z) = SH + FAC*COS3
END IF
END DO
*
RETURN
END