GENERALIZED METHOD OF COMPOSITE MATERIAL ANALYSIS USING FEM

Sagar Suryavanshi[1], Addanki Sambasiva Rao[2], Sandeep Desai[3]

Abstract
This paper mainly focuses on steps to analyze composite material part using FEA. It makes use of concept of "Representative Volume Element" which is the smallest volume over which a measurement can be made that will yield a value representative of the whole. Following paper has illustrated generalized steps to be carried out for composite material analysis using FEM which gives more accurate results and is computationally efficient. It aims at understanding interaction between matrix and fibre and failure analysis of composite materials making using of concept of homogenization.
Keywords: Finite element method (FEM), Representative volume element (RVE).

I. Introduction

Composite materials are often considered a better choice over traditional materials, for example, metals, owing to their superior characteristics such as high specific strength/stiffness, low coefficient of thermal expansion, corrosion resistance, etc. In addition, the most attractive feature of composite materials is that the properties can be tailored. By modifying the constituents and their volume fractions, composite materials with a wide range of properties can be possibly manufactured. However, the resulting composite materials have to be tested for the collective stiffness or homogenized properties of the constituents for use in structural design.

Physical testing of composite materials is often time consuming and expensive. Furthermore, owing to the different possible combinations of fiber materials, matrix materials, micro-structural arrangements, and volume fractions, physical testing may be impractical which can limit the choice of constituents available for design. An alternative approach for testing composite materials is using numerical methods such as the FEM. Closed form solutions do exist for some class of composite materials, for example, Rule-of-Mixtures formulae for unidirectional plies where the constituent properties and volume fractions are assumed known [1]. Although these closed-form solutions provide reasonable estimates for some composite ply properties, they do not account for the micro-structural interaction of the constituents. A finite element micromechanics model that incorporates the composite microstructure can provide higher fidelity results than most closed form solutions.

Micromechanics modeling and analysis of composite structures using the FEM requires identifying the volume of the structure to be included in the simulation. Based on the concepts of RVE, a micromechanics model for a composite must be large enough so it captures the structural variations of the constituents. However, the RVE must be small enough for practical use in FEM simulations. To minimize the size of the RVE, periodicity is often assumed. In periodicity, the randomness in structural arrangement of the constituents is removed and the arrangement is then uniform. With assumed periodicity, RVE geometry can be reduced to an idealized unit cell with fewer constituents. Here, periodicity in unit cells extends to loads, boundary conditions, and field variables (displacements, strains, and stresses).

In finite element method, a structure is represented using nodes and elements. To enforce periodicity in the FEM model of a unit cell, the degrees of freedom of the boundary nodes, which are continuum points, are connected through multi-point constraints. A common practice is to use eight-node hexahedral elements for finite element discretization of three-dimensional unit cell models. Furthermore, a one-to-one correspondence is enforced on the boundary nodes. That is, for every node on a face/edge, a corresponding unique node must exist at a similar location on a connecting face/edge. This is referred to as the **traditional method**.

The traditional method is not always feasible owing to the complex architecture of some unit cell models. Four-node tetrahedral elements can be used to mesh complex unit cell models. However, the regularity of the generated mesh cannot be predicted. Enforcing periodicity on a unit cell model meshed with tetrahedral elements requires a method that does not necessitate a one-to-one nodal correspondence on boundary nodes.

A. GENERALISED WORK FLOW.

Step 1:- Identification of RVE.

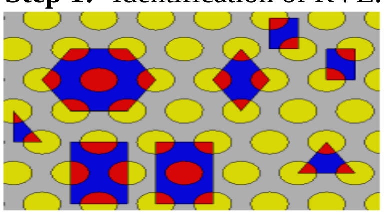

Figure 1

Step2:- Meshing

Figure 2

Step 3:- Application of Periodic boundary conditions & Load cases.

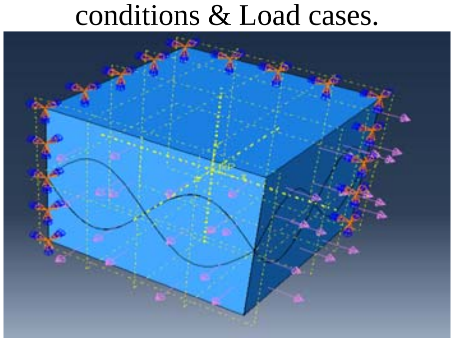

Figure 3

Step 4:- Getting overall properties of Composite material (**Homogenization**).

Step 5:- Applying homogenized properties to part model.

Figure 4

Step 6:- Identifying Critical points.

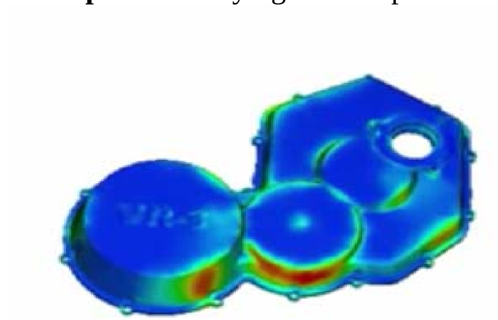

Figure 5

Step 7:- Applying back, properties of critical points to RVE to check Matrix-Fibre Interaction at that point.

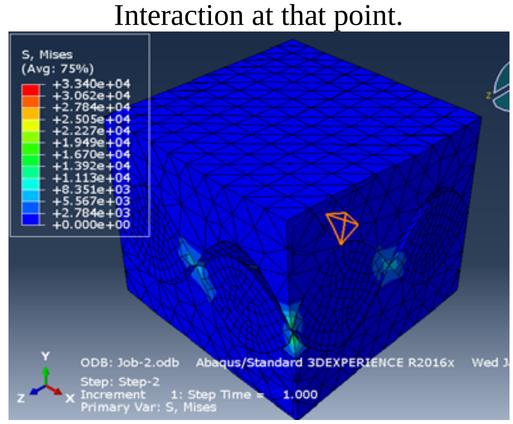

Figure 6

B. Results and Discussion.

This approach mainly reduces computational time and gets results closer to actual value. This approach is based FEM. The inputs are RVE type, RVE parameters, Material properties of matrix and fiber, Material orientation etc. The intermediate output is homogenized material properties which are used by actual part model for analysis.

The material property when applied to part model gives deformation as shown in figure 5 below. Then any deformed point is picked and its properties like stress, strain, etc are applied back to RVE to understand actual interaction at micro-level between matrix and fiber as shown below in figure 6.

II. Conclusion:

Two import concerns for accurate analysis using FEM method are identification of RVE and modeling geometry correctly and applying appropriate boundary conditions to RVE to get material properties.

Out of all methods currently available the FEM approach is most preferred one due to best computational efficiency and high cost savings as the material failure simulation is done virtually and not in physical world saving time and money. Apart the simulation results are much closer to actual experimental values thereby validating the FEM approach.

III. Future Scope:

Current thesis is only focused on one RVE i.e., “Woven RVE” with only matrix and fiber .One can go for RVE’s having matrix, fibre and other reinforcements which would involve understanding geometries of RVE’s at micro-level and finding suitable equations to construct them which is a challenging task.

Second important task is to apply FEA method’s Boundary Conditions to get the material properties. Apart as we move ahead new composites would be developed which needs to modeled. Current industry is mainly focusing on composites for strength, reliability and weight reduction and hence the composite material analysis is a new domain to explore.

Third important task would be to model all existing composites with use of mathematical equations. In this method we do consider some amount of approximation while modeling RVE which can be further improvised by devising some new methods which is a big challenge for industry as perfect modeling of RVE would increase computational time.

References

[1] Jones, Robert. “Mechanics of Composite Materials”, Taylor & Francis; 2nd Edition, 1999.

[2] Nemat Nasser S. & Hori, “M. Micromechanics: Overall Properties of Heterogeneous Materials”, Elsevier (Amsterdam); 1993.

[3] Whitcomb, J. D., Chapman, C. D., & Tang, X., “Derivation of Boundary Conditions for Micromechanics Analysis of Plain and Satin Weave Composites”, Journal of Composite Materials 34 (724-747);2000

[4] Tang, X. & Whitcomb, J. D., “General Techniques for Exploiting Periodicity and Symmetries in Micromechanics Analysis of Textile Composites”, Journal of Composite Materials 37 (1167-1189);2003

[5] Karami G., & Garnich, M. R., “Effective Moduli and Failure Considerations for Composite with Periodic Fiber Waviness” , Composite Structures 67(461-475);2005

[6] Eric Li , Zhongpu Zhang , CC Chang , GR Liu & Q Li, “Homogenization for composite material properties using smoothed finite element method”, School of Aerospace, Mechanical and Mechatronic Engineering, The University of Sydney, Sydney, NSW 2006, Australia

[7] https://en.wikipedia.org/wiki/Composite_material

[8] https://en.wikipedia.org/wiki/Micromechanics