International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES)
Impact Factor: 5.22 (SJIF-2017), e-ISSN: 2455-2585
Volume 4, Issue 10, October-2018

SOFTWARE DEVELOPMENT FOR THE STATIC ANALYSIS OF SHORT FIBER COMPOSITES WITH SPHERICAL INCLUSIONS

Babu Reddy[1], Aditya Hosur[2], Mahesh Sanje[3], Ambadas[4]

[1]*Dept, of Machine Design, VTU Centre for PG Studies Kalaburagi,*
[2]*Dept, of Machine Design, VTU Centre for PG Studies Kalabuaragi,*
[3]*Dept, of Machine Design, VTU Centre for PG Studies Kalaburagi,*
[4]*Dept, of Machine Design, VTU Centre for PG Studies Kalaburagi,*

Abstract: Analysis of composite materials, especially the estimation of elastic properties is a tedious job considering the number of parameters that govern the behaviour of materials. Hence automation of variation of this variable is suggested. In the present work, a FE based interface tool named VTUMAT is developed. VTUMAT at present interfaces with Abaqus/Standard for generating the RVE of the composite material and for visualising the stress and strains induced in the RVE*. The accuracy of VTUMAT is verified by comparing the results obtained from it with the result of Rule of Mixture. It is found that VTUMAT results match closely with the theoretical results.

Keywords—VTUMAT, Abaqus/CAE, Composite Material Analysis, Rule of Mixture, Short Fiber Composites

I. INTRODUCTION

We know that unidirectional composites offer a high degree of stiffness in fiber direction and possess very low stiffness in the transverse direction. This can be eliminated by manufacturing lamina which is strong in all directions But even in such a design the top and bottom layers will be weak in the transverse direction. This problem can be overcome by manufacturing lamina with inplane isotropy. To achieve this isotropy, short fibers are used. The length of the short fibers is between 1 cm to 8 cm. [1]

Elastic Properties of Composite materials can be estimated using Analytical [2][4], Experimental and Finite Element Method [3][5]. Experimental method is a most accurate method for estimation of composite material properties. However experimental technique is costly owing to material cost, testing cost and labour involved. A better cost effective option would be to use numerical technique followed by an experimental technique. FE based Softwares or Automation tools are developed to accelerate the development process for Composite Materials. Automation tools also help in reducing the time and money required for composite material analysis. These automation tools generate RVE of the composite, predict the material properties, and also give the stress and strain values in the RVE when loads are applied. VTUMAT is one of the Automation tools developed to ease the composite material analysis.

II INTRODUCTION TO VTUMAT.

VTUMAT is a homogenization Software used to model Representative volume Elements (RVE) of a large variety of material Microstructures. VTUMAT can be interfaced with any commercially available FE Tools such as Abaqus/CAE, ANSYS Workbench, LS-Dyna, and Marc. Current Version of VTUMAT (v.1.0.0) is interfaced with Abaqus/CAE. Based on the user inputs VTUMAT generates python script or simply script file (extension .py) and geometry data file. The script file contains the information related to material properties, phase type, element type, seed size and loading parameters. The geometric data file contains information related to inclusion, inclusion orientation and volume fractions of each phase. Abaqus/CAE reads the script and geometric data file and generates the RVE. Inclusion shape, inclusion orientation can be visualized in Abaqus/CAE GUI.

A. *Tools used to Develop VTUMAT*

1) *Microsoft Visual Studio (v.2010)*: Microsoft visual studio is an integrated development environment. It is used to develop windows based computer applications, web apps, websites and web services. Visual studio c# development environment used for developing VTUMAT.
2) *Microsoft Access (v.2007)*: Microsoft Access is database program used to store user data. In VTUMAT Microsoft access database used to store data related to material properties.

B. *Getting Familiar with VTUMAT*

VTUMAT general window can be divided into the Main menu area, Left menu area, and Tab area. These areas are numbered as 1, 2 & 3 respectively in Fig.1

*In the present work, Analysis of SFC with spherical inclusion is carried out.

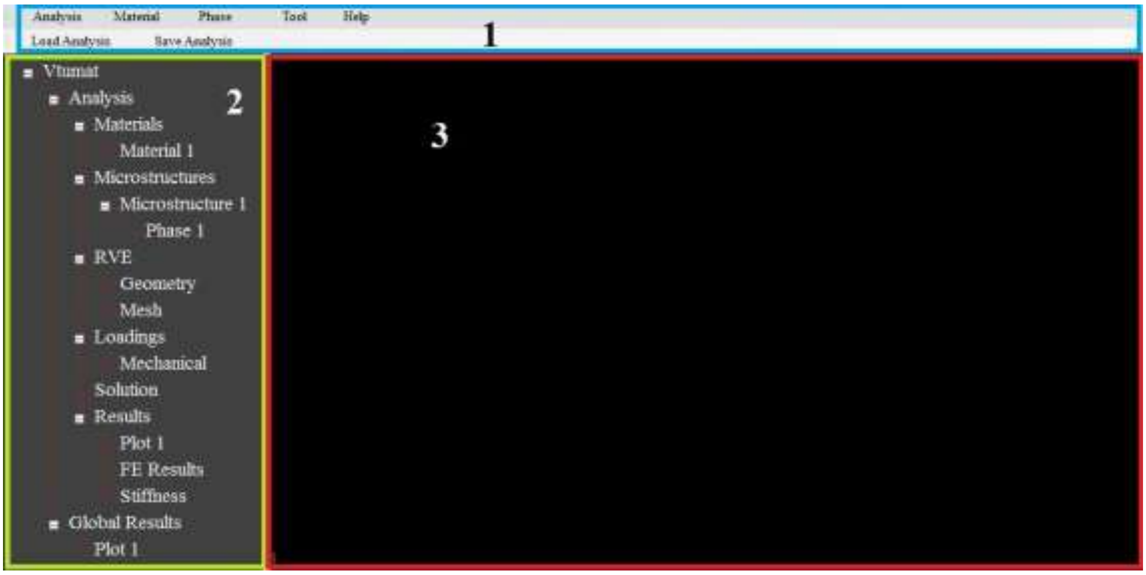

Fig.1 VTUMAT GUI

1) *Main menu strip area*: Main menu or primary menu is used to interact with system directories, for example, operations such as saving the file to the system directory, importing files. The main menu strip consists of Analysis, Material, Phase, Tools and Help items.
2) *Treestructure or Left menu area*: The Dynamic Tree structure is anchored at the left side in the VTUMAT Software. Tree structure comprises a number of nodes, each node is assigned to a do particular function**.** When the user clicks the left mouse button, Tabs will appear at the right side of the tree(adjacent to Treeview). Context Menu will pop up when the user hits the right mouse button. Items in Context menu are almost the same as the items in the main menu strip.
3) *TabControl Area or Tab area*: Different tabs are Created and assigned to each node in the Treestructure. Tabs Consist of Textboxes with designated names. These textboxes store the user given data. When the user hits the Create button, data from the textboxes will be Sent to Database (MS Access 2007). The following Tabs are available in the VTUMAT. (v.1.0.0).

C. *Tabs Used in VTUMAT*

1) *General tab*: General tab used to set the working Directory.
2) *General parameter tab*: Following data are defined in this tab: Name of the Analysis, Type of FE Code (Abaqus or Ansys), Analysis Type (Mechanical or Thermal), and Representative Volume Element Type (3D or 2D).Current Version of VTUMAT Supports only Mechanical Analysis and Abaqus FE Code.
3) *Material tab*: In the Material tab properties of all the materials present in analysis are defined. The material tab consists of two pages, Model page and the Parameters page. The material laws are defined in Model page and Properties such as Young's Modulus, Poisson's Ratio, and Density are defined in the parameters page. The materials can be loaded from system storage using "Load material" item in material Context menu or from the material menu strip at the top.
4) *Microstructure tab*: The Microstructure item in the Treestructure allows the user to create multiple phases. To create Microstructure one matrix phase and at least one inclusion phase must be defined. Microstructures phases can be loaded from system storage. Phase (single) can also be saved, loaded and deleted. At present VTUMAT is constrained for single layer analysis and a number of phases in the microstructure is fixed to 4. "Phase 1" is dedicated for matrix phase. Materials for the Creating phases are selected from the combo box available in microstructure tab. Validate button in the Phase tab used switch from type page and parameters page. To define inclusion phase user need to provide volume fraction, a number of inclusions, inclusion shape, interface behavior and angles theta and phi for orientation of inclusions.
5) *RVE tab*: Representative volume element (RVE) is the elementary volume of composite that is being analysed in VTUMAT. RVE can either comprise one or several layers of the microstructure. The information related to RVE is stored in a geometric file (extension .zip).
6) *MESH tab:* Mesh tab pops up when user click the "Mesh" item in Treestructure. Mesh Tab requires the information related to Element Type, Seed Size, and Number of Refinement Steps. In VTUMAT Default seed size is 0.05. Available Element types are C3D4, C3D10, C3D8, C3D8R, C3D20, and C3D20R. C3D10M

element offers good contact results and these are ideally used in when there is an edge to surface or point to surface contact.

7) *Loading tab*: To complete the analysis appropriate loading conditions on the RVE should be defined. The quantity of loadings to be characterized depends on the kind of analysis that is being performed. Appropriate boundary conditions need to be applied. VTUMAT (v.1.0.0) supports only periodic boundary condition.
   This option will impose periodic boundary conditions on all the faces of the RVE. Periodic boundary conditions guarantee that the motion of the field variable (displacement) is not intermittent regarding the characteristics (i.e. faces) of the volume component.
8) *Solution tab*: The Solution tab used to export the FE Code to Abaqus/CAE. The Data Related to Final Time, Maximum Number of Increments, minimum time increment, initial time increment & maximum Time Increment read by FE Code Solver to Generate RVE. When the User Clicks the Export button in VTUMAT solution tab, Open -file dialog box will appear, the user needs to define the directory to export python script and geometric data. Export button also opens Abaqus/CAE and run the generated script.

D. *Types of Files used in VTUMAT or VTUMAT file system*

File System helps in storing and retrieving data from storage medium. VTUMAT handles different type of files during analysis. Files are identified by its extensions. The extension indicates characteristic of the contents of file. Types of files used during analysis are as follows:

1) *Analysis File*: Every input data given by the user during Analysis is Stored in Analysis File (.daf).It contains information related to Materials, Phases, Microstructures, RVE, Loadings, and Solution.
2) *Material file*: Stores material name, elasticity type, young's modulus, Poisson's ratio and density. It also Stores axial young, InPlane_Young, InPlane_Possion, Transverse_possion, transverse shear in the case of the transversely isotropic material model (File extension .dmf).
3) *Phase File*: Phase file contains phase name, type (inclusion and matrix),material selected to create phase, angles phi and theta, interface behaviour, volume fraction number of inclusions, aspect ratio and more (File extension .dpf).
4) *Script File*: Script File is used generate RVE in the Abaqus/CAE. It contains each and every data given by the user during analysis, VTUMAT takes user data and combines them to Create FE Code Suitable for Abaqus/CAE (File extension .py)
5) *Geometric data File*: This is a compressed file contains Parasolid files with extension *.xmt_text (X_T).

III WORKING WITH VTUMAT

Short Carbon Fiber and Epoxy are considered for the estimating the elastic properties of composite using VTUMAT.

A. *Material Properties*[1]:

Table. I Carbon Properties

Property Name	Value
Density	1800 Kg/m^3
Young's Modulus	275GPA
Poisson Ratio	0.22

Table.II Epoxy Properties.

Property Name	Value
Density	1400Kg/m^3
Young's Modulus	3.5GPA
Poisson Ratio	0.35

In addition to the material properties following data are entered in the VTUMAT tabs before exporting a script and generating the RVE of Composite.

Table.III Addition data for composite material analysis

Name	Value
Volume Fraction of Carbon (V_f=V_i)	0.10.
Volume Fraction of Epoxy (V_e=V_m)	0.90.
Loading Type	UNIAXIAL
Maximum Strain	0.03
Element Shape	Spherical.
Seed Size	0.05
Element Type	C3D10M
Boundary Conditions	Periodic

After entering data into VTUMAT, Analysis (i.e. Script file) is exported to Abaqus/CAE where it generates RVE of composite depending upon the input data.

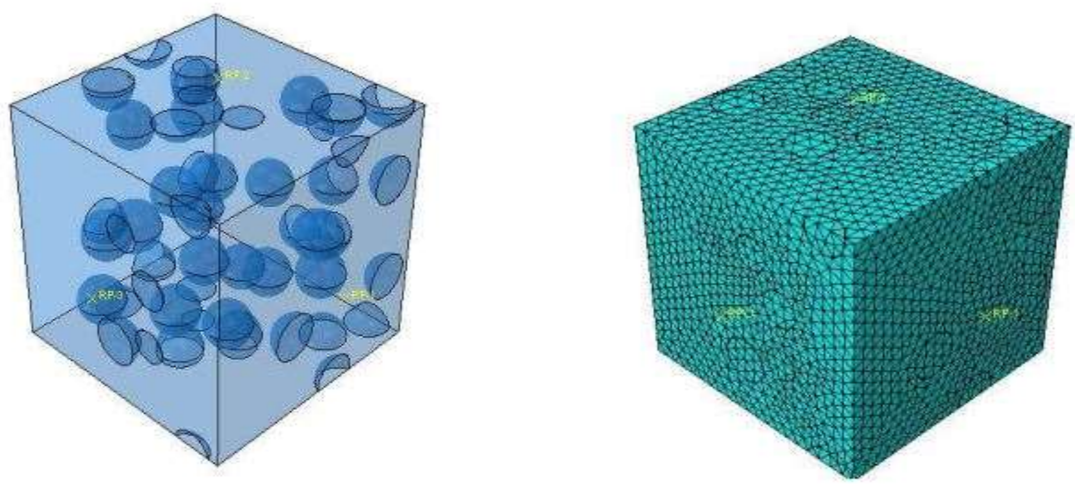

Fig.2 RVE of Carbon-epoxy Composite.

IV RESULTS AND DISCUSSION

In this section results obtained from VTUMAT and Abaqus/CAE are compared with Theoretical results obtained from Rule of Mixture.

A. Theoretical Results

From Rule of Mixture:

$$E_1=V_fE_f+V_mE_m$$

Where:-
E_1= Young's Modulus or Elastic modulus along Longitudinal direction
E_2=Young's or Elastic Modulus along Transverse direction
E_f=Young's Modulus of Fiber
E_m=Young's Modulus of Epoxy.
V_f =Fiber Volume Fraction.
V_m=Matrix Volume Fraction.
From Table I, II, III
Young's Modulus=E_1= (0.10*275) + (0.90*3.75)

E_1=30.83GPA.

B. Abaqus Results

1) Elastic Constants

Engineering constants

	Value
Young's modulus E1	4604.64
Mismatch on E1	0.00295724
Young's modulus E2	4603.49
Mismatch on E2	0.00429894
Young's modulus E3	4568.43
Mismatch on E3	0.00280271
Poisson's ratio 12	0.33605
Poisson's ratio 21	0.335867
Poisson's ratio 13	0.340785
Poisson's ratio 31	0.338075
Poisson's ratio 23	0.342282
Poisson's ratio 32	0.339711
Global density	1.8E-009

Fig.3 Elastic constants for Carbon-Epoxy RVE is found using Abaqus Solver

2) Stress and Strains in RVE

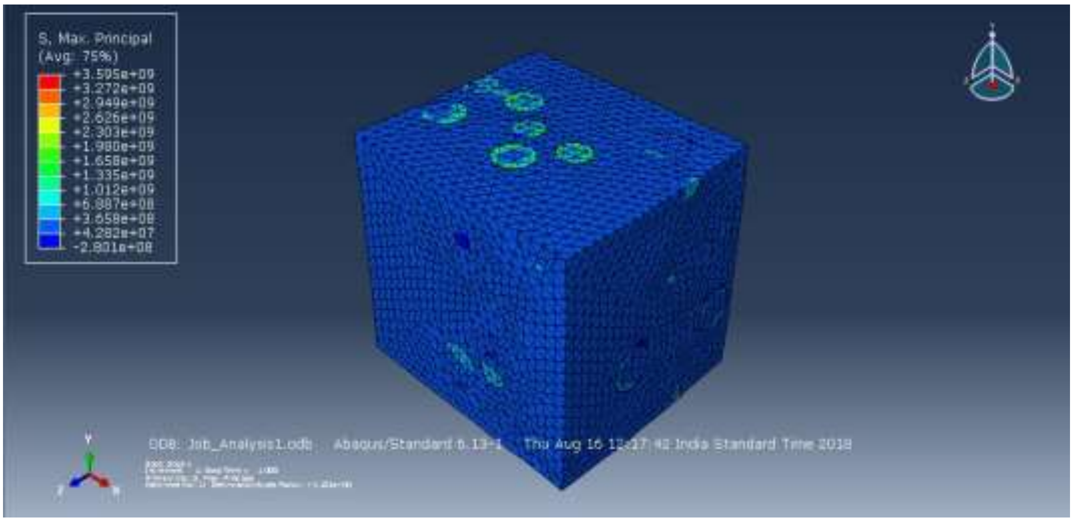

Fig 4 Stress in RVE

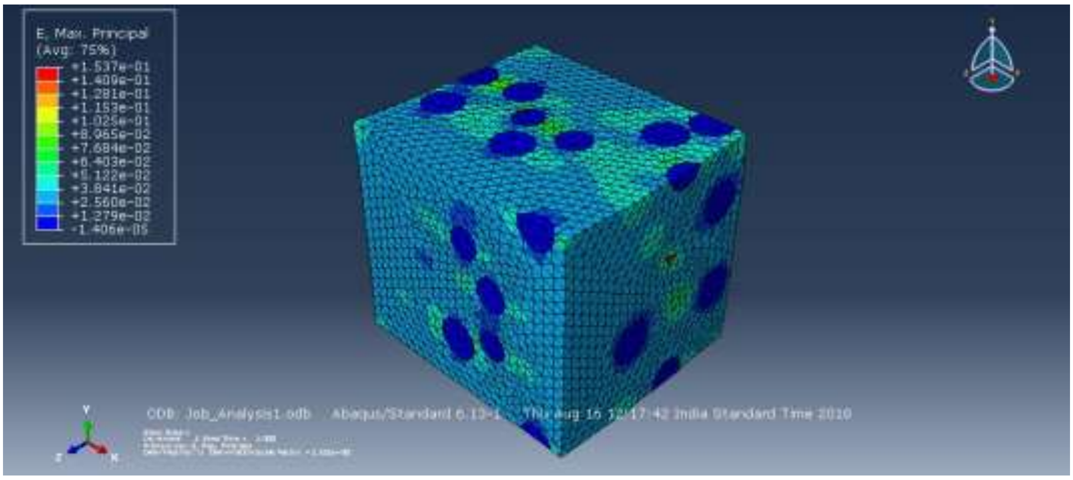

Fig.5 Strains in RVE

It's found that the elastic constants obtained from the Abaqus solver lies below the theoretical elastic constants calculated using Rule of Mixture. It's also known that Rule of mixture always gives the maximum value of Elastic constants.

V CONCLUSION

Important Conclusions can be drawn from this paper is that VTUMAT is capable of solving composite material problems by generating the RVE of composite material. Different type of Loading Conditions and Boundary conditions can be imposed on the composite material using VTUMAT. RVE of the composite can also be visualised using Abaqus Therefore Time and Money required for the composite material analysis can thus be reduced by using VTUMAT.

REFERENCES

[1] Kaw, "Micromechanical Analysis of a Lamina", Mechanical Engineering Series, 2005. Following Authors used Analytical, Experimental, FE technique to estimate elastic properties of Composite materials.
[2] Hyoung Seop Kim, Sun Ig Hong, Sun Jae Kim. "On the rule of mixtures for predicting the mechanical properties of composites with homogeneously distributed soft and hard particles" Journal of Materials Processing Technology, 2001.
[3] G. R. Liu "A step-by-step method of rule-of-mixture of fiber- and particle-reinforced composite materials "Composite Structures Vol. 40, Nos 3-4, pp. 313-322, 1998.
[4] Meisam Omidi,Hossein Rokni D.T., Abbas S. Milani Rudolf J. Seethaler ,R. Arasteh "Prediction of the mechanical characteristics of multi-walled carbon nanotube/epoxy composites using a new form of the rule of mixtures "CARBON 48 (2010) 3218 –3228 Elsevier Ltd.
[5] S. S. RANGARAJ, S. B. BHADURI "A modified rule-of-mixtures for prediction of tensile strengths of unidirectional fiber-reinforced composite materials" Journal of materials science 29 (1994) 2795-2800.