Proc. of the Conference "Applications of Physics in Mechanical and Material Engineering" (APMME 2020)

Proposal of Physical Model for Damage Simulation of Composite Structures Produced by 3D Printing

M. Sága*, J. Majko, M. Handrik, M. Vaško and A. Sapietová

Department of Applied Mechanics, Faculty of Mechanical Engineering, University of Žilina Univerzitná 8215/1, 010 26 Žilina, Slovak Republic

Doi: 10.12693/APhysPolA.138.245 *e-mail: milan.saga@fstroj.uniza.sk

The article focuses on proposal and analysis of the physical models suitable to stress simulation and limit state prediction of components made of composite structure on Onyx basis. The observed parameters are physical properties of matrix and reinforcement. The additive manufacturing technologies also allow the production of complex shape constructions with cavities. The cavities are regions characteristic of damage initiations, but a realization of experimental measurements on inner surfaces of them to predict failure is problematic. Therefore, the exploitation of appropriate physical models and subsequently mathematical models is a method for observing of material behaviour using selected physical quantities beyond surface layers. The modelling of composite structures is viable using physical models, which working principle is based on homogenization of structure, for instance, representative volume elements (RVE). The primary disadvantage of RVE models is a complicated analysis of physical properties at inhomogeneity places of reinforcement. The physical model proposed in this paper describes each fibre separately regarding authentic physical properties and precise determination of fibre deposition in the matrix. Implementation of FEM programs with a primary objective to perform formation and analysis of the physical models, provides solution of various multiphysical problems, such as nonlinear behaviour of material induced by contact, temperature variation or interaction with other environments (temperature, fluid, electromagnetic fields), and so on. The main aspect of the presented concept is the generation of computational models with a precise definition of fibre deposition in the composite structure. The numerical analysis of loading and limit state prediction was executed on models of the composite structure. The structure includes onyx as a thermoplastic matrix and carbon fibres. The fundamental physical criterions for damage initiation assessment were stress state and strain at critical locations of matrix and reinforcement.

topics: 3D printing, finite element analysis (FEA), Matlab, carbon fiber, onyx

1. Introduction

The development of additive manufacturing technologies represents a significant breakthrough in possibilities of formation of objects in various sectors of industrial production. Even 3D printing is included within this group of technologies. Due to its development in the last decades, it provides us not only plastic production but also production from various materials, e.g., metals or continuous fiber reinforced thermoplastics (CFRTP) [1]. A presence of the continuous fiber in the structure results in better mechanical properties of printed composites by maintaining their light weight in comparison with plastics [2–4]. In mechanical tests, specimens reinforced by carbon fiber reach tensile strength up to 600 MPa [5]. An advantage of 3D printing is a possibility to set production parameters, even an accurate way of the fiber position in the structure [6]. Conventional technologies do not provide that. This parameter affects significantly physical properties of the printed composite. To enable a design of the optimum mode of fiber placement for each specific case, it is convenient to design such a physical and consequently a mathematical model that enables a partially accurate and precise description of properties of the composite for purposes of simulation to determine the stress-strain state in the matrix, fibers and in the inner structure of the composite.

2. Homogenisation

When modelling composites, models based in structure homogenisation are frequently applied. It means that non-homogeneous material composed of two and more constituents from microstructural standpoint, even randomly arranged, is substituted by the equivalently homogenous one [7]. The most frequently applied way of homogenisation is considering an ideal periodical arrangement of the fiber via the application of so-termed Representative Volume Elements (RVE) [8]. Possible shapes of RVE composites reinforced with uniformly deposited fiber ($2a_2 = 2a_3$) in the structure are defined in Fig. 1 [9].

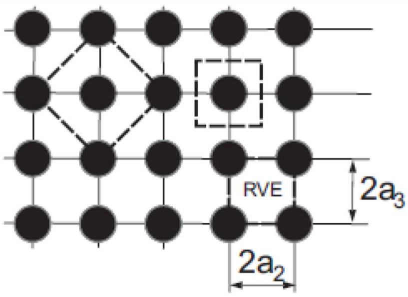

Fig. 1. Shapes of RVE [9].

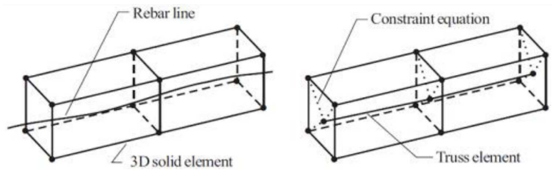

Fig. 2. The process of the generation of REBAR elements and constraint conditions [11].

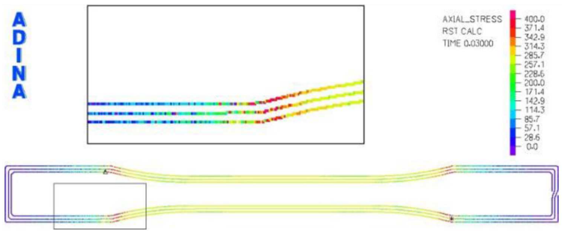

Fig. 3. Stress distribution in fibers in modelling of the composite via REBAR elements.

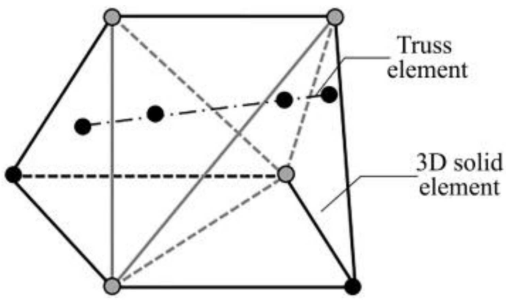

Fig. 4. Various length of truss elements.

A condition for the application of RVE is the structural conformity with the average of the composite sample [10]. The aim of homogenisation is to define non-experimentally mechanical properties of RVE by means of FEM software or various computational models, e.g., Voigt and Reuss models, etc. That can be consequently applied in composite modelling. This principle is complex from the standpoint of accuracy and time consumption. As a result of potential variability of fiber arrangement in one layer of the composite structure defined by the user of the 3D printing machine, homogenisation seems to be an inefficient tool for a definition of physical properties. The reason is a need to define properties of a large amount of RVE which provide characteristics of each change of the fiber placement.

Besides homogenisation, we may apply other physical models of CFRTP composite based on the definition of the exact arrangement and position of the fiber in the matrix. They will be presented and described in the following lines.

3. Modelling via the application of REBAR elements

By means of so-termed discrete REBAR elements, it is possible to define each fiber individually in the physical model. In case of an application of the finite element software ADINA it is possible to apply the procedure described further. The reinforcement in the structure representing the rebar line intersects the faces of the generated 3D elements of the mesh (Fig. 2a). In these intersections nodes, there occur nodes that are consequently interconnected by means of truss elements. The connection between REBAR truss elements and the generated mesh of matrix is defined by constraint equations among nodes in intersections and three closest nodes of 3D elements in the generated mesh (Fig. 2b) [11].

Constraint equations are implemented in the equation system of the solved task, and thus the calculation becomes more demanding in terms of time and computational technique. Another disadvantage is the level of accuracy of the solved task as the applied method is approximating and its accuracy depends on the quality of the generated mesh. Inaccuracies in the calculation result in non-uniformity of stress distribution in fibers (details in Fig. 3).

A significant discontinuity in values of stress distribution may be a result of insufficiently strict criteria of convergence in the solution of a non-linear problem. However, a change of convergence criteria did not result in a required effect. By analysing the model in detail, monitoring elements and stress errors, we identified a problem in an element dimension and its relationship with accuracy of the calculation we are able to reach. In the model, there are formed truss elements of very small lengths when the line for REBAR elements crosses close to a node element of the four-sided element (Fig. 4). For this reason, it is not possible to generally define parameters which will secure the calculation accuracy depending on an increase of the number of elements in the generated finite element mesh.

The method of modelling proposed here by authors exceeds the limits of the previous models. Based on experience, we found out that a modification of the object geometry is necessary. The geometry is algorithmised by specially designed programs

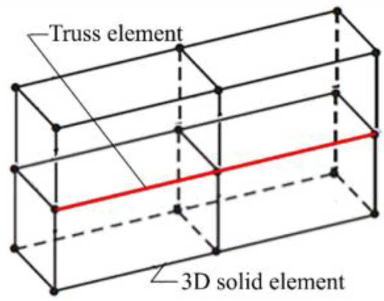

Fig. 5. A generation of truss elements by the split geometry method.

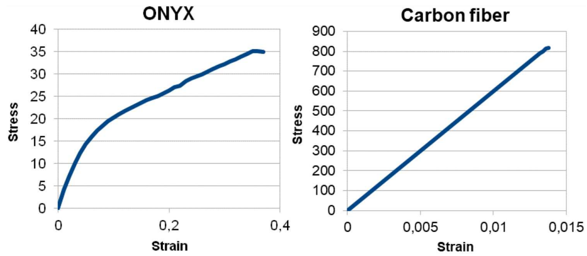

Fig. 6. Material model of matrix (a), and (b) the fiber.

in Matlab. In the first step, a geometry division is performed in layers in which reinforced fibers are placed in printing. Consequently, there are generated lines on the faces of geometric models of geometry distribution. The result is that in the edge of the element, there is a truss element representing the composite reinforcement (Fig. 5). Truss elements are thus interconnected with solid elements without an application of constraint conditions.

In strength evaluation of the surface layer of the composite matrix we may assume that the obtained stress does not exceed the tensile strength of the matrix material which means that in case of the stated loading, the matrix damage will not occur. Regarding the fact that the fiber transfers most of the load, one may assume that a total breakage of the specimen will occur in the point of the fiber damage.

4. Modelling via the split geometry method

Multilinear material models for matrix (Fig. 6a) and the fiber (Fig. 6b) mathematically processed from the series of experimental measurements were applied for the purpose of a definition of individual composite parts.

In the model formation for the simulation of stress-strain state boundary conditions were specified so that all degrees of freedom were taken away on one end and on the other end, we defined the time-dependent displacement.

Stress distribution in long reinforcing fibers for the model designed by the split geometry method is shown in Fig. 7. We may observe a continuous

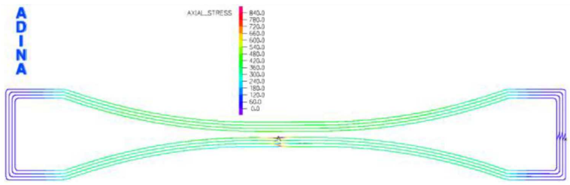

Fig. 7. A stress distribution in fibers by the application of the split geometry method, identification of fiber damage at maximum stress zone.

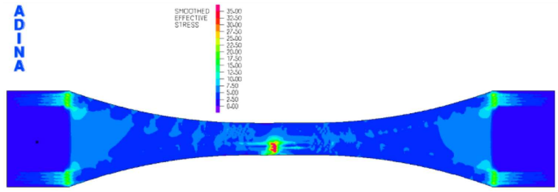

Fig. 8. Von Mises stress distribution in the matrix by the application of the split geometry method.

change of values of stress in fibers. In contrast to REBAR elements, in case of the proposed methodology of modelling, it is possible to obtain strikingly more accurate solutions by the application of the suitable element mesh. In terms of the strength criterion, the highest stress values in the reinforced layers in the geometric centre of the specimen exceed tensile strength of carbon fibers which is a condition for the damage of the reinforcement fiber.

The physical model assumed three layers of carbon fibers. The aim was to simulate the limit state. Therefore, we proposed a modeled formation of fiber failure in the middle layer of the lower half of the model by applying very little fiber failure by breaking them. This causes an increased state of stress at the critical point, as shown in Fig. 7. Subsequently, the state of stress in the matrix also increases sharply at the point of fiber failure (see Fig. 8). This mathematical-physical approach was implemented to verify the suitability of the proposed physical and computational model, its numerical stability and appropriate convergence.

The mentioned process of damaging is accompanied by delamination that is a frequent case of composite constructions. An interface between laminas is a suitable place for crack growth, as a connection between laminas depends only on matrix properties. Delamination may be described by means of cohesive damage models or breakage mechanics. In case of breakage mechanics that enables us to foresee the development of an already-existing crack, the existence and delamination are analysed by comparing the amount of energy release rate with the fracture toughness of the interface. For instance, following the Griffith theory and the energetic approach, we compare the energy release rate with the energy required for the crack process that is defined for a given material [9].

5. Conclusion

A significant property of the application of the finite element method is the impact of elements sizing of the generated mesh on the solution accuracy. Normally, we may assume that the higher mesh density, the more precise and accurate results are obtained. However, it is only within the accuracy of the physical model and the applied method of calculation. In case of the application of REBAR elements for the purpose of modelling of the fiber in the composite structure, we cannot increase the accuracy of calculation by a modification of dimensions of mesh elements. Due to this reason, it is not possible to obtain more accurate and precise results of a given task. As opposed to REBAR elements, the proposed split geometry method enables us to obtain more accurate solution of the task by an increase of the mesh density, which represents the basic assumption for a possibility of an application of the gained results.

Detailed knowledge of the stress state may lead in the future to the development of special methods for evaluation of, e.g., fatigue damage based on simulation analyses with or without consideration of the effect of delamination of the composite structure. Therefore, even further research of authors will lead towards the formation of a suitable approach towards the prediction of delamination in CFRTP composites produced by the additive manufacturing technology.

Acknowledgments

This work has been supported by grant agency VEGA project No. 1/0073/19.

References

[1] J.C. André, *From Additive Manufacturing to 3D/4D Printing 1*, ISTE Ltd., London 2017.

[2] G.W. Melenka, B.K.O. Cheung, J.S. Schofield, M.R. Dawson, J.P. Carey, *Compos. Struct.* **153**, 866 (2016).

[3] G.D. Goh, V. Dikshit, A.P. Nagalingam, G.L. Goh, S. Agarwala, S.L. Sing, J. Wei, W.Y. Yeong, *Mater. Des.* **137**, 79 (2018).

[4] M. A. Caminero, J.M. Chacón, I. García-Moreno, G.P. Rodríguez, *Compos. Part B* **148**, 93 (2018).

[5] G.D. Goh, Y.L. Yap, S. Agarwala, W.Y. Yeong, *Adv. Mater. Technol.* **4**, 1 (2018).

[6] CH.K. Chua, K.F. Leong, *3D Printing and Additive Manufacturing 5th Edition* World Scientific Publishing Company, Singapore 2017.

[7] V.V. Silberschmidt, in: *Multiscale Modelling and Simulation of Composite Materials and Structures* Eds. Y.W. Kwon, D.H. Allen, R. Talreja, New York 2008, p. 630.

[8] Q.H. Qin, Q.S. Yang, *Macro-Micro Theory on Multifield Coupling Behavior of Heterogeneous Materials* Higher Education Press & Springer, Beijing 2008.

[9] J.E. Barbero, *Finite Element Analysis of Composite Materials Using ANSYS*, CRC Press, Boca Raton 2014.

[10] J. Aboudi, *Mechanics of Composite Materials: A Unified Micromechanical Approach*, Elsevier Science Publishers, Amsterdam 1991.

[11] ADINA, *Theory and Modelling Guide*, 2017.