



Model simulation of reconstructed fiber yarns

Josef Vosáhlo¹ 

¹Technical University of Liberec, Studentska 2, 461 17 Liberec 1, Czech Republic
josef.vosahlo@tul.cz

Abstract. These abstract deals with fiber yarn model simulation, a technique used to optimize realistic simulations associated with the fiber manufacturing process. Filament yarn is produced using a complex process that involves spinning the fibers into a single thick and continuous strip called yarn. The objective of this study is to develop a mathematical model that simulates the behavior of fiber yarn during normalized stress in the vertical fiber axis. The model is based on the finite element principle, which allows to simulate the interaction between the individual fibers, and based on the resulting deformation and stress, it allows to select an appropriate optimization to be applied to the equipment in the textile machine. To achieve accurate results, material properties of the fibers such as elasticity, tensile strength and Poisson's number were implemented in the model. The simulation results provide important information about the behavior of the fiber yarn during the loading process in the transverse direction. Using this information, the manufacturing process can be optimized and better quality and performance of the fiber yarn can be achieved. Based on the model simulation, the production process in the textile industry can be improved and contribute to more efficient use of resources and improve the overall quality of the fiber yarn.

Keywords: 3D geometry, fibres, material model, deformation

1 Introduction

Fiber structures are a fascinating and important element in many fields such as engineering and materials science. In recent years, 3D modeling techniques have been increasingly used to create and analyze these structures. In this way, it is possible to accurately represent the geometry of fibers and their interconnections. One of the key steps in the process is to create the geometry, using scanned CT images which are then reconstructed into a 3D model. The process of creating the geometry requires precise modeling and a combination of different techniques to capture the complex structure of the fibers. Another key aspect is the material model, which describes the behavior of the fibers and their interaction with the environment. A properly chosen material model is a key factor for accurate simulation and prediction of the behavior of the structure in real conditions. After the creation of the geometry, an important stage is followed by finite element model simulation [1].

This simulation allows to analyse the mechanical properties of the fibrous structure and to perform a detailed assessment of its behavior under different loads. FEM simulation provides valuable information on the strength, stiffness and durability of the structure. Finally, optimization becomes an essential part of the process. The goal of optimization is to find the most efficient fiber arrangement and properties that maximize the desired properties of the structure. Through a combination of 3D modeling, FEM simulations, material models, and optimization, improved design of fiber structures with high efficiency and strength can be achieved [2-4]. This area of research and applications offers many opportunities and potential for innovation and advancement in various industries. Exploring and improving techniques for 3D modeling of fiber structures, geometry generation, FEM simulations, material models and optimization brings new perspectives and opportunities for the development of advanced materials and technologies [5].

2 Materials and methods

2.1 Creating geometry

From these images, center points were selected in Matlab and imported into the 3D modeling program Solidworks student from DSS (DassaultSystemes). In the CAD program, the center points were imported and were loaded as a point cloud. Using a written macro, these points were used to create 3D curves into which the average values of the 0.035 mm fibers were inserted. The curve-drawing function was used to create a 3D model of the reconstructed fibers for each 3D curve. This allows detailed information about the yarn structure to be obtained. Based on the images, the centre points of the yarns were identified in Matlab. These points were then imported into the 3D modeling program Solidworks student. In the CAD program, the center points were loaded as a point cloud. Using a specially written macro, 3D curves were created from these points. Cross sections with a diameter of 0.035 mm were then inserted into these curves. see Fig. 1 (a, b). The final 3D model of the reconstructed fibers was created using the "curve dragging" function applied to each 3D curve. Using this method, it was possible to accurately model the complex structure of the fiber. see Fig. 1 (c, d) [3- 6].

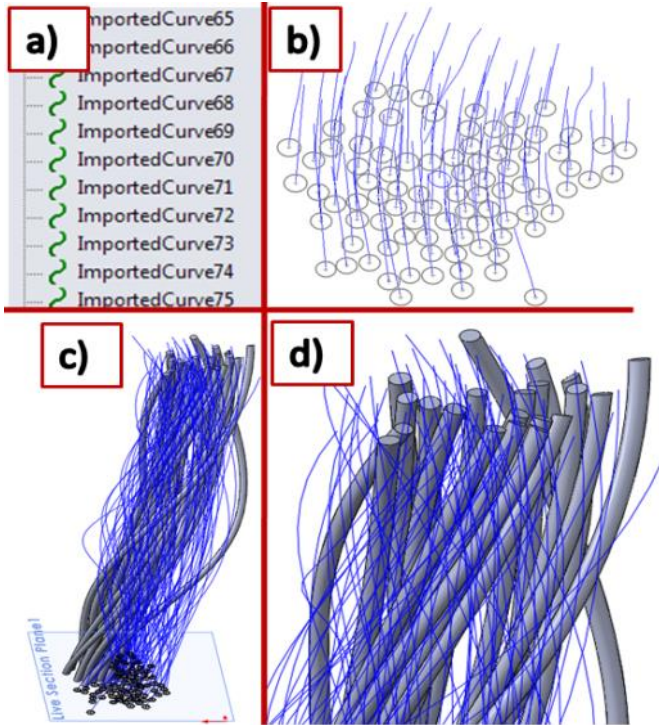


Fig. 1a) Imported individual curves into the 3D surface, **b)** Creation of 3D model by the dragging method, **c)** Creation of 3D model by the curve dragging method, **d)** detailed view of the curve dragging.

2.2 Theory of mechanical behavior of fibers

The fiber itself can be generally characterized as having linear properties as stated e.g. in [7]. Therefore, the basic linear relationship between stress and strain can be used for the mathematical description, so that the linear elasticity of the fiber is the ability to elastically deform. The deformation of linear elastic materials is instantaneous (time independent) and perfectly reversible. This can be simply characterized by the fact that if a set of forces F_1, F_2, F_3 is applied to an infinitesimal volume element of a fiber, the strain of the element is determined by 9 stress components: 6 shear and 3 normal.

$$|\sigma_{12}| = |\sigma_{21}| \quad |\sigma_{23}| = |\sigma_{32}| \quad |\sigma_{13}| = |\sigma_{31}| \quad (1)$$

Keeping the condition of equilibrium with respect to the centre of gravity of the element, the relation is valid and therefore 6 stress components out of 9 are sufficient to determine the stress of the system. The linear relationship between stress and strain is expressed by Hooke's law according to (1), or in extended matrix form according to (2) [7].

$$\sigma = k\varepsilon, \sigma \neq f(t) \tag{2}$$

$$\begin{pmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{13} \\ \sigma_{23} \\ \sigma_{12} \end{pmatrix} = \begin{pmatrix} C_{1111} & C_{1122} & C_{1133} & C_{1113} & C_{1123} & C_{1112} \\ & C_{2211} & & & & \\ & \dots & & & & \\ & \dots & & & & \\ & \dots & & & & \\ & C_{1211} & & & & C_{1212} \end{pmatrix} \cdot \begin{pmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \varepsilon_{13} \\ \varepsilon_{23} \\ \varepsilon_{12} \end{pmatrix} \tag{3}$$

The fibers investigated in this work are of synthetic polymeric material (PES) origin, where the stress state can be described through tensor dependence. The calculation can be based on the characteristic description of the stress and strain invariant, which are coordinate system independent quantities as reported in [7]. Subsequently, we can express the relative strains according to relations (1).

$$\lambda_1 = 1 + \varepsilon_{11} \quad \lambda_2 = 1 + \varepsilon_{22} \quad \lambda_3 = 1 + \varepsilon_{33} \tag{4}$$

The following relations that explain the behavior of a linear elastic material are given in reference [7].

2.3 KP model for the study of mechanical pro fiber

In order to determine the mechanical properties between the individual fibers, the yarn model was subjected to simulated testing in Ansys R.20. The first step was to create a new material model. For this purpose, the literature [...] with material properties of plastic and composite materials was used. The obtained material properties of Polyether (PES) were successively inserted into the material property boxes. The newly created material model was stored under the name PES_01 and used for the simulation process under this name. See Table 1.

Table 1. Table captions should be placed above the tables.

Material	Density [kg*m-3]	Youngl module [MPa]	Poisson's number [-]	Volume module [MPa]	Shear module [MPa]	Yield strength [MPa]	Tangent module [MPa]
PES_01	1339	2898	0,388	4339	1043	90,22	95,45

One way to insert geometry into a simulation program is by direct import. Ansys Workbench supports the import of many different CAD file formats. The most commonly supported formats are STEP, IGES, Parasolid (XT), ACIS (SAT), and many native CAD formats. If further editing is required in the model before starting the

simulation, the loaded geometry can be opened and edited in the supported internal ANSYS SpaceClaim or Design Modeler programs see Fig. 2.



Fig. 2 Imported fiber yarn model into Ansys.

2.4 Contacts

The choice of the appropriate contact model is crucial and must be appropriate to the nature of the problem we are solving. In this particular case, I am concerned with modeling surface friction between fibers, and this problem limits the choice of appropriate contact types. Most of the standard contacts provided are unsuitable for this type of model. In some cases, the modeling can be avoided by using nonlinear contacts. To achieve better results, it is necessary to use these nonlinear contacts and, however, we must take into account that nonlinearity will increase the computational effort and this may lead to difficulties in convergence of results see Fig. 3 (a, b) [5, 6].

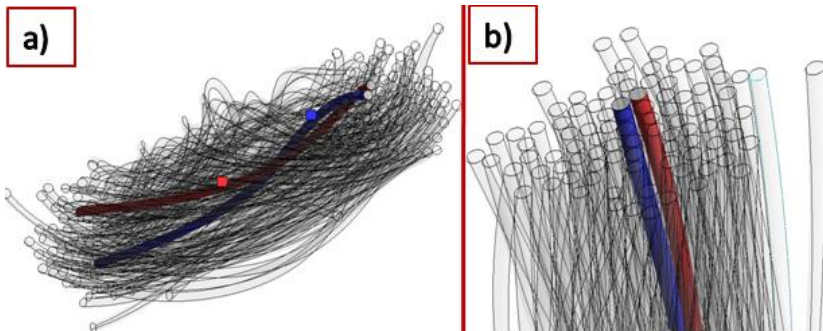


Fig. 3 a) Created contact impressions, b) Detailed view.

2.5 Element settings

Shape functions for 3D solid 285 volume elements. These features are available in a number of configurations, including certain combinations of the following features: Feature shapes can be tetrahedrons, needles, wedges, or bricks (hexahedra).

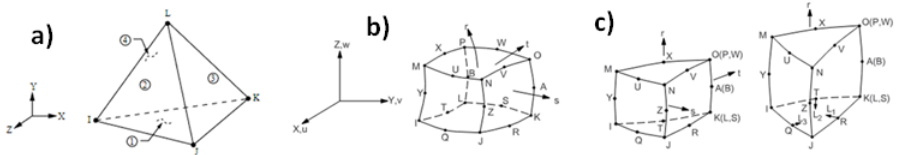


Fig. 4 a) 3D solid element 285, b)20 - Nodal brick element,c) 20 - Nodal brick element.

If the elements are wedge or brick, with or without special form features (ESF) With or without rotational degrees of freedom (RDOF) With or without central nodes Wedge elements with central nodes (15-node wedges) are either a condensation of a 20-node brick element or are based on wedge-shaped features. These shape functions are either straight 4-node tetrahedral, such as SOLID285, or a condensed 8-node brick element, such as SOLID5, FLUID30, or SOLID98 see Fig. 4 (a-c) [6].

2.6 Introduction of the FEM mesh

The optimal choice of meshing method in ANSYS depends on the specific geometry of the model and the desired accuracy of the calculation. The meshing environment provides users with a variety of tools and options to efficiently generate and modify the mesh for their specific finite element calculation needs. A hexagonal volume element was chosen for this type of fiber yarn model. In the Body sizing, the element growth parameters were set to 0.01mm in detail and the heavy transformation value was changed in the advanced settings. Along the length of the yarns, Face Meshing was introduced to the faces to transform the individual elements into a mapped mesh. For the overall fiber model, 504483 elements and 669041 nodes were meshed. See Fig. 5 (a, b).

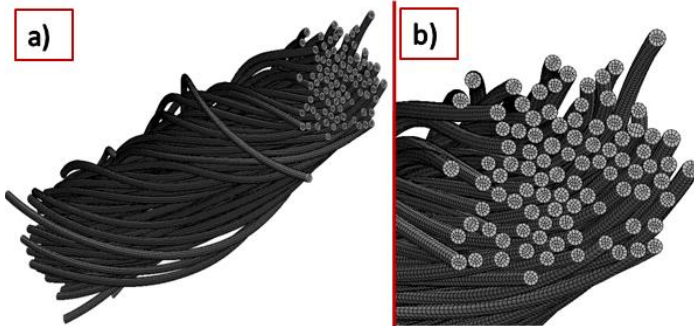


Fig. 5 a) Generated FEM mesh, b) detailed view.

2.7 Boundary conditions

Two types of boundary conditions were selected for this type of simulated loading. The first boundary condition (Fixed) was placed at the bottom of the model to prevent rotations and displacements in all directions see Fig. 6 a). The following boundary condition, displacement, was applied to the surface of the controlled displacement. See Fig. 6b), the definition allowed to specify displacement values in each direction

depending on the rotation of the Cartesian coordinate system. From this option, one direction of displacement was selected into which, a linear displacement value of 0.25 mm was introduced. The remaining values were set to zero, thus depriving the model of unnecessary degrees of freedom.

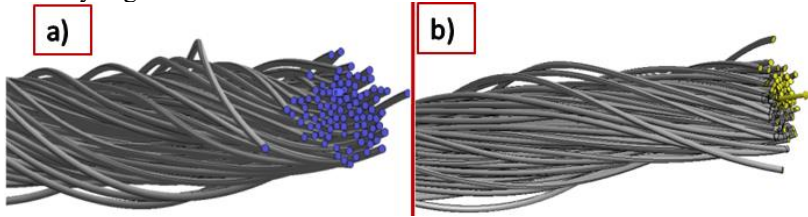


Fig. 6 a) Fixation boundary condition, b) sliding boundary condition.

3 Results and Discussion

The output of the simulated loading of the fiber yarn produced a series of results in standing extension lengths with initial strain lengths ranging from 2% to 10% of the actual yarn length. At 2% deformation of the total length of the fiber yarn, a maximum elongation length of 0.05 mm was deducted. The maximum resultant stress of 225 MPa (von-Misses) was read in the sliding region of the boundary condition. Average stress values of 100 MPa were observed throughout the model of the filament yarn as the stretch developed along the length and diameter of the yarn, see Fig. 7(a, b) [6].

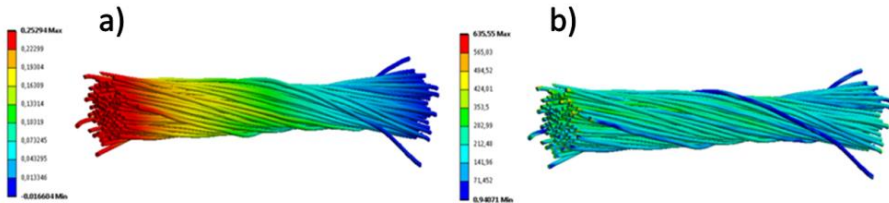


Fig. 7 a) Record of the deformation, b) the resulting stress (von-Mises) in stretching time steps.

Table 2. Readings in individual length sections.

Section	Deformation [mm]	Average value von-Misses stress [MPa]
2%	0,05	100
4%	0,1	119
6%	0,15	180
8%	0,2	251
10%	0,25	280

4 Conclusions

Based on the midpoints obtained by image analysis from CT images of the fiber yarn, the midpoints of the reconstructed fiber twist were created in a three-dimensional coordinate system. The obtained center points were imported into a CAD program where 3D curves were generated using the curve fitting function. The average fiber cross-section values were entered into the newly created trajectory and the individual fibers were pulled along the curve using the volume pull command. The 3D model of the reconstructed fiber structure was imported into the simulation software ANSYS R20. In the basic structure of the program, a new linear elastic material model named PES_01 was inserted. Subsequently, contact pairs, KP network and boundary conditions were added to the model. The deformations and stresses due to the stretching of the fiber yarn were observed, evaluated and then entered into a spreadsheet. Throughout the modeling process, an inherent stage is to find the most efficient fiber configuration to achieve maximum performance characteristics. Through an innovative combination of 3D modeling, FEM simulation, material models and optimization techniques, significant improvements in the properties of fiber structures can be achieved.

References

1. Kovalova, Natalia, PetrKulhavý, Josef Vosáhloa Antonin Havelka. Numerical modelanalysisofmultiaxialstressonthe sewing seam. In: 16th World Tex. Conf. AUTEX 2016 Proc. [disk]. Ljubljana: University of Ljubljana, 2016. ISBN 978-961-6900-17-1.
2. Zak J. Modelling of textile structures, Fibres and Textiles, Slovak University of Technology in Bratislava, Vol. 22, 2015, ISSN 1335-0617.
3. Kulhavý P., Kovalova N. and VosáhloJ.. Methods of creating a numerical model of a real seam based on experimental data. EAN 2015-53rd Conf. on Experimental Stress Analysis. CeskyKrumlov; Czech Republic, 2015: 195-202.
4. Žára, J.,Beneš, B. Modern computer graphics. Brno: Computer Press, 2004, 609 s. ISBN 80-251-0454-0.
5. Barbulov-Popov, Daniela, Nenad CIRKOVIC a Jovan STEPANOVIĆ. The Influence of Stitch Density and of the Type of Sewing Thread on Seam Strength. TEM Journal. 2012 (2), 104-110.
6. Madenci, E., Guven, I.: The Finite Element Method and Applications in Engineering Using ANSYS, Springer 2006, ISBN-13: 978-0387282893
7. Fyzikapolymerů: Lineárníelasticita - studijní text. ÚstavFyziky a Materiálovéhoinženýrství [online]. Zlín: UniverzitaTomášeBativeZlíně, 2017 [cit. 2018-03-24].

Open Access This chapter is licensed under the terms of the Creative Commons Attribution-NonCommercial 4.0 International License (<http://creativecommons.org/licenses/by-nc/4.0/>), which permits any noncommercial use, sharing, adaptation, distribution and reproduction in any medium or format, as long as you give appropriate credit to the original author(s) and the source, provide a link to the Creative Commons license and indicate if changes were made.

The images or other third party material in this chapter are included in the chapter's Creative Commons license, unless indicated otherwise in a credit line to the material. If material is not included in the chapter's Creative Commons license and your intended use is not permitted by statutory regulation or exceeds the permitted use, you will need to obtain permission directly from the copyright holder.

