

The Effect of Different Nozzle Configurations on Airflow Behaviour and Yarn Quality

D. Yilmaz, and M.R. Usal

Abstract—Nozzle is the main part of various spinning systems such as air-jet and Murata air vortex systems. Recently, many researchers worked on the usage of the nozzle on different spinning systems such as conventional ring and compact spinning systems. In these applications, primary purpose is to improve the yarn quality. In present study, it was produced the yarns with two different nozzle types and determined the changes in yarn properties. In order to explain the effect of the nozzle, airflow structure in the nozzle was modelled and airflow variables were determined. In numerical simulation, ANSYS 12.1 package program and Fluid Flow (CFX) analysis method was used. As distinct from the literature, Shear Stress Turbulent (SST) model is preferred. And also air pressure at the nozzle inlet was measured by electronic mass flow meter and these values were used for the simulation of the airflow. At last, the yarn was modelled and the area from where the yarn is passing was included to the numerical analysis.

Keywords—Nozzle, compressed air, swirling airflow, yarn properties.

I. INTRODUCTION

AIR has great usage for processing of textile fibres and yarns into end-products in textile field. Fibre extrusion, spinning preparation, spinning, texturing, and weaving, cleaning and cooling of the machines are well-known examples for the air usage. These efforts offer certain advantages, especially high production speeds. In yarn spinning process, air, in particular high speed air has been used for the twisting of staple fibres. Air-jet and Murata air vortex spinning systems are one of the commercial applications of this technology. In these spinning systems, a staple fibre bundle is twisted by means of the air jet nozzles.

Nowadays, there are the trials based on the usage of the nozzle component of air-jet spinning system in the spinning or winding processes. In spinning, the nozzle has been applied on to the conventional ring and compact spinning systems. The main objective in these applications is to improve the yarn quality. In literature, many researchers studied the effect of the nozzle on yarn properties. They determined that the system is capable to produce the yarns with improved yarn properties [1-7]. These systems mainly consist of three components: nozzle, pressurized air and the yarn. Therefore, the researchers mainly analysed the effect of air pressure level and the

structural parameters of the nozzle on airflow character and also the yarn quality either experimentally or computationally. In the studies, they widely centre upon the effect of air pressure level [2-3, 8-12] and main hole diameter on yarn quality [8-9, 13-14].

In numerical analyses, Zeng and Yu (2004) used PHOENICS while Patnaik et al. (2005; 2006) and Rengasamy et al. (2006; 2008) realized the simulations by Fluent package programs. On the other hand, Guo et al. (2007; 2009a; 2009b) worked on the simulation of the nozzle used in air-jet spinning system. Standard k- ϵ turbulent model is widely preferred for the turbulent model in the engineering application and hence all these numerical analyses were realized by standard or modified k- ϵ model. Nevertheless, the model does not perform very well for the flows with boundary layer separation, sudden changes in swirling and rotating flows and flows over the curved surfaces [12, 18-19]. However, Shear Stress Transport (SST) model accounts for the transport of the turbulent shear stress and gives highly accurate predictions of the onset and the amount of the flow separation under adverse pressure gradients [20]. On the other hand, in literature, it was assumed that the yarn passing from the nozzle can not affect the flow inside the nozzle and therefore the yarn was not modelled during the simulation. In addition, there is not any information about the values used in numerical analysis. Considering these facts, in present study, it was aimed to simulate and analyse the airflow in the nozzle and also on to the yarn using the Shear Stress Transport (SST) turbulent model.

II. METHOD

A. Nozzle Configuration

The nozzle used in present work has a cylindrical cross section and consists of main hole (1), injectors (2), twisting chamber (3) and nozzle outlet (4) (Fig. 1). Main hole lies starting from the nozzle inlet to nozzle outlet and the nozzle has a diameter of 3.0 mm. Injectors are positioned at certain angles with respect to the nozzle axis and so they lie tangentially. The vertical angles of the injectors (θ) are 15° (Fig. 2). Air enters with this mentioned angle to the main hole. The nozzle has four injectors and their diameters are 0.5 mm.

D. Yılmaz is with the Department of Textile Engineering, Engineering Faculty, Suleyman Demirel University, Batı Kampusu, Cunur, 32260 Isparta, Turkey (phone: +90 246 2111186; fax: +90 246 2370859; e-mail: demetyilmaz@sdu.edu.tr).

M. R. Usal is with the Department of Manufacturing Engineering, Technology Faculty, Suleyman Demirel University, Batı Kampusu, Cunur, 32260 Isparta, Turkey (phone: +90 246 2111409; fax: +90 246 2370859; e-mail: resitusual@sdu.edu.tr).

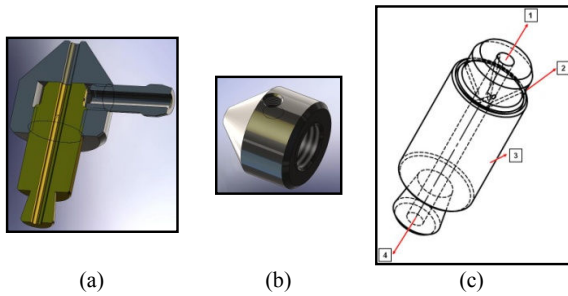


Fig. 1 Nozzle (a), nozzle head (b) and the nozzle body (c) (1: Main hole, 2: Injectors, 3: Twisting chamber, 4: Nozzle outlet)

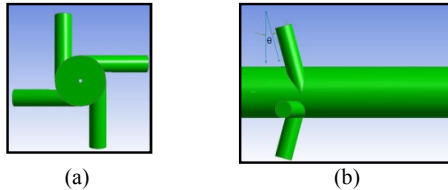


Fig. 2 Profile of injectors (a) and injector angle (b)

Compressed air is supplied from the compressor to the nozzles through the pipe (Fig. 3). Nozzle head transfers the pressurized air coming from the compressor into the nozzle body by means of the injectors (2). The pressurized air goes out of the nozzle inlet and outlet. In the study, the air pressure (gauge pressure) was kept at 0.5 bar.

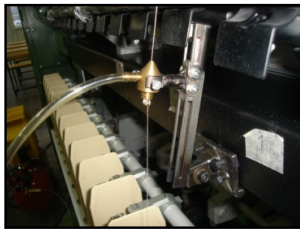


Fig. 3 Transferring the compressed air into the nozzle

In order to determine the airflow structure, the compressed air in the nozzle was simulated with ANSYS 12.1 package program and Fluid Flow (CFX) analysis method [21]. In numerical analysis, at first, each nozzle configuration was created in ANSYS Workbench as a solid. In present study, two types of nozzle were studied to analyse the airflow behaviour and changes in yarn properties. In one of the nozzle, nozzle injectors were placed close to the nozzle inlet and this nozzle was called as normal type of nozzle. In the other type, injectors were taken to the nozzle centre and it was named as centre positioned nozzle.

In literature, it was assumed that the cross-sectional area of the yarn is about 1/100th to that of the main hole of the nozzle and the yarn occupies a small area in the nozzle. Hence, the flow inside the nozzle affects the yarn, but not vice versa [16-17]. Therefore, the yarn was not modelled in the numerical analyses concerning with spinning systems with nozzle component. In this work, it was included the area from where the yarn was passing and studied the flow field around the yarn even if the effect of the yarn was accepted as insignificant. Additionally, it was given a speed which is equal

to production speed of spinning machine to the yarn (approximately 0.2 m/s).

B. Grid Generation and Meshing Procedure

Following to the geometry, the nozzle was divided into many small elements called as the cells or grids. Each cell can be thought as a tiny control volume in which discretized versions of the conservation equations are solved. Therefore, they define the cells on which flow variables (velocity, pressure, etc.) are calculated throughout the computational domain [22].

The quality of a CFD solution is highly dependent on the quality of the grid. During the grid generation, sharp-pointed angles were occurred due to the intersections of the injectors and twisting chamber. Hence, it was difficult to obtain a regular mesh in the entire geometry. On the other hand, injectors were one of the most important nozzle structural parameters concerning with the fluid flow and therefore it was given importance to generate a regular grid structure in this section. We followed a strategy applying the automatic mesh option to the section between the injectors and yarn entering zone while using a structured mesh to the section of nozzle exit (Fig. 4).

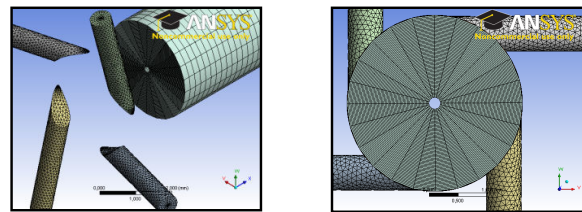


Fig. 4 Grid structure

The mesh was built with the finite volumes varying between 200.000 and 9.000.000. Considering the computational effort and time consuming, it was better to employ the mesh cases having 400.000 elements.

C. Solution Algorithm: Turbulence Models

Therefore, many models have been developed to calculate the effect of turbulent flow. In present work, it was used "Shear Stress Transport (SST)" turbulent model, which is one of the Reynolds stress models, as a solution algorithm. Shear Stress Transport (SST) model accounts for the transport of the turbulent shear stress and gives highly accurate predictions of the onset and the amount of the flow separation under adverse pressure gradients [20]. Particularly, standard and also modified k-ε turbulent models were widely used for the numerical analyses of the nozzle in which pressurized air was fed [12, 16-17]. Although k-ε turbulent model produces the reasonable results, the model does not perform very well for the flows at the boundary layer. Additionally, k-ε model assumes the turbulent viscosity as isotropic. This assumption is being a substantial mistake for the strong flows in which turbulent eddy viscosity is anisotropic. Actually, the equations used for the turbulent generation and dissipation are similar to the standard k-ε model. However, the constants in both models are different.

D. Boundary Conditions

Appropriate boundary conditions are required in order to obtain an accurate CFD solution. At the inflow or outflow cases velocity or pressure conditions are specified at the boundaries which fluid enters the computational domain or leaves the domain. The present work used the pressure condition as a boundary condition because we did not have any measured data about the air velocity in any nozzle geometry. Therefore, air pressure at the nozzle inlet was measured in terms of the electronic mass flow meter (Fig. 5). In the measurements, a mass flow meter of Alicat Scientific firm was used and placed between the compressor and nozzle opening. Therefore, we specified the total pressure at the nozzle inlet in which the flow was coming into the computational domain from the compressor.



Fig. 5 Measurements by electronic mass flow meter

The nozzle used in present work consists of the yarn, nozzle inlet or main hole, injectors and nozzle outlet. Therefore, there were a few pressure inlets and outlets in the computational domain. The rest of the geometry was called as “wall or plane”.

- **Internal boundary conditions :** The direction of the airflow at the nozzle inlet varies depending on the nozzle configuration and so it was not possible to specify accurately at the beginning of the analysis. Therefore, nozzle inlet or main hole was defined as “opening” and the cases of air coming in or leaving was being the optional depending on the main flow conditions in the nozzle. At the nozzle inlet or main hole, the air pressure was the atmospheric pressure conditions.
- **External boundary conditions :** At the nozzle outlet, atmospheric pressure was assumed.
- **Wall boundary conditions:** Non-slip boundary condition was applied to the nozzle walls. Therefore, the flow between the air and nozzle wall was assumed non-slip and having any velocity.
- **Yarn boundary condition:** Boundary condition for the yarn was taken as non-slip. However, it was given the equal speed to the production speed of conventional ring spinning machine (about 0.2 m/s).

E. Fluid Type and Properties

The fluid type in this work was the compressible airflow and fluid properties were specified as ideal gas.

F. Initial Conditions

In a CFD solution process, the conservative equations were solved iteratively for each cell. Sometimes hundreds or even thousands of iterations were required to converge on a final solution and thus the residuals may decrease by several orders of magnitude. In the work, iteration number was taken five as a minimum value and 300 as a maximum. Depending on the converging condition of the solution, iteration was performed over 300. Sensitivity of the solution was set at 10^{-5} .

III. RESULTS AND DISCUSSION

A. Numerical Analysis

As mentioned in the literature, the analyses of ANSYS Fluid Flow (CFX) indicate that pressurized air swirls along the nozzle and an air vortex is occurred in the nozzle (Fig. 6).

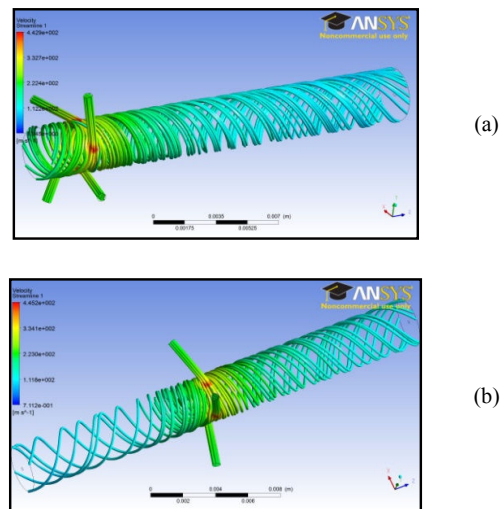


Fig. 6 Airflow behaviour coming from injectors of normal (a) and centre positioned (b) nozzle

On the other hand, it is not possible to explain the airflow characters in the nozzle with a single structure. Actually, different types of airflow arise with different air velocities and characters. These forms can be divided into three different parts of the nozzle. One of them is the airflow sent from the injectors and moving on the nozzle walls (Fig. 6). Another one is coming from the nozzle inlet and under the effect of airflow getting from the injectors. The last one is the airflow around the yarn. These mentioned fluid behaviours lead to the air displaying different flow attitudes at the nozzle injectors, main hole and nozzle outlet associated with different airflow properties. In addition, flow field variables of the velocity and pressure vary at the position where the injectors are opened into the main hole, nozzle inlet and also nozzle outlet. Therefore, all airflow structures and variables should be analysed for each nozzle type (Table I).

TABLE I
AIRFLOW VARIABLES FOR DIFFERENT NOZZLE TYPE

Nozzle type	Air velocity (m/s)			Air pressure (Pa)	Mass flow ($\times 10^{-5}$ kg/s)		
	Injectors	Inlet	Outlet		Injectors	Inlet	Outlet
Normal	226.3	80.4	55.9	131262	36.96	10.6	-44.6
Centre positioned	212.2	40.3	76.4	125452	35.1	-7.7	-37.2

When we studied the fluid variables, it was determined that there is not considerably significant differences between velocity, pressure and mass flow values of normal and centre positioned nozzles. However, air velocity values of centre positioned nozzle are lower than that of the normal nozzle at the nozzle inlet while centre positioned nozzle has higher velocity values at the nozzle outlet. This case is agreed with the findings of Guo, Chen and Yu (2009a). Airflow velocity values at the nozzle injectors are higher than that of the other parts of the nozzle. And its velocity decreases along the nozzle. In centre positioned nozzle, airflow goes through the nozzle outlet with lower distance and hence airflow velocity is being higher. There is similar case at the nozzle inlet and airflow in the normal nozzle reaches to nozzle injectors with lower distance and hence its velocity is higher in comparison to that of the centre positioned nozzle.

At the main hole or nozzle inlet, some air is sucked from the nozzle outside. Then the air is started to act into the nozzle without any outstanding swirling motion. At the injector region, some part of the air returns and goes towards the nozzle inlet with a swirling motion. When the mass flow values of the nozzle are analysed, these cases can be observed. In the normal nozzle, suction case is dominant contrary to the centre positioned nozzle and mass flow value is positive (Table I). It is believed that this case is resulted from the interactions between the airflows sent from the injectors and sucked from the nozzle inlet or main hole (Fig. 7).

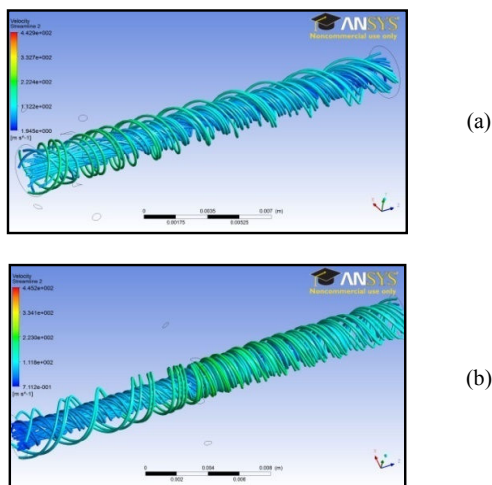


Fig. 7 Airflow behaviour inside the nozzle for normal (a) and centre positioned (b) nozzle

At the injector outlet, two types of rotating airflow are created (Fig. 8). One of the airflow is moving towards the nozzle centre and the other is rotating around this airflow. Actually, an airflow structure arises moving in form of the one

within the other and having different air velocities. The airflow form of the one within the other leads to the sandwich and bulky, but regular and intensive airflow structure. This airflow form is considerably intensive in centre positioned nozzle type in comparison to that of the normal nozzle type.

The velocity of the airflow decreases towards the nozzle outlet. Sometimes, the airflow leaves from the nozzle slowly and thus it is accumulated and heaped at the nozzle outlet. In addition, the airflow returns interior of the nozzle and then moves with swirling motion to the nozzle outlet (Fig. 8). This case was observed in both nozzles. However, the airflow is more intensive in centre positioned nozzle.

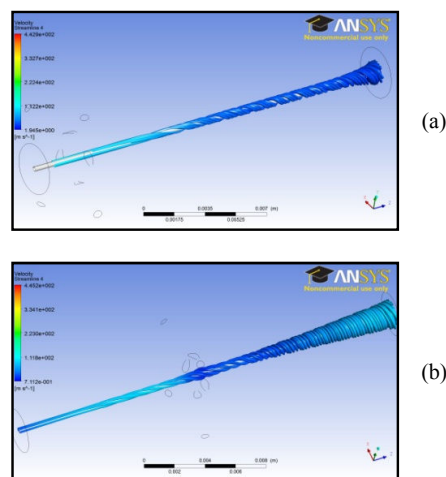


Fig. 8 Airflow behaviour at the nozzle outlet for normal (a) and centre positioned (b) nozzle

B. The Analysis of Yarn Properties

In this part, 100% combed cotton yarns were produced with normal and centre positioned nozzles. For each nozzle, three yarn cops were obtained and yarn hairiness and yarn tenacity were determined. All the yarns were spun with the same material and spinning variables.

Yarn hairiness is one of the most important yarn properties due to its effect to the post spinning processes and fabric appearance. Therefore, it is important to be able to control the yarn hairiness. Yarn hairiness is characterized by s3 value in Zweigle hairiness tester. Zweigle s3 parameter is the total number of protruding hairs having 3 mm and upper lengths in 100 m yarn.

In addition to yarn hairiness, tenacity of the yarns was also measured. Uster Tensorapid tester was used for the measuring of the tenacity. The results were given in Table II.

TABLE II
YARN PROPERTIES

Nozzle type	S3	Yarn tenacity (cN/tex)
Normal	554	14.56
Centre positioned	332	14.77

As seen in Table II, the yarns produced with centre positioned nozzle have lower hairiness and higher yarn tenacity values. The differences in hairiness values are statistically significant. Therefore, centre positioned nozzles

decreases hairiness significantly (Table III). On the other hand, there is not any significant difference in tenacity values and hence both yarns have similar tenacity. As a result, centre positioned nozzle improve the yarn properties, in particular yarn hairiness.

TABLE III
ANOVA TEST RESULTS

	Hairiness	Tenacity
Sig.	0.022*	0.690

*: The mean difference is significant at the 0.05 level.

It is believed that intensive airflow cases interior of the nozzle and also at the nozzle inlet leads to wrap the protruding hairs onto the yarn body and hence hairiness of the yarns decreases in centre positioned nozzle.

IV. CONCLUSION

In present study, two types of the nozzle were analysed numerically. Airflow behaviour was simulated and airflow variables were determined. The changes in yarn properties were indicated and explained with the airflow form inside the nozzle.

ACKNOWLEDGMENT

This work is supported by the grants from Unit of Scientific Research Projects of Isparta in Turkey (Project No: 1795-D-09) and The Scientific and Technological Research Council of Turkey (Project No: 109M372). The authors also wish to express their thanks Ekrem Gülsevinler for his collaboration to the firms of Isparta Mensucat San. Tic. A.Ş. and Yumak Tekstil San. Tic. A.Ş. for offering testing services and contributions to the study and also Technician Ufuk Korkmaz and other staff in the laboratory for his generous help.

REFERENCES

- [1] Ramachandralu, K., Dasaradan, B.S., "Design and fabrication of air jet nozzles for air vortex ring spinning system to reduce the hairiness of yarn", *IE (I) Journal-TX*, 84, August, 2003, pp. 6-9.
- [2] Zeng, Y.C., Yu, C.W., "Numerical and experimental study on reducing yarn hairiness with the JetRing and JetWind", *Textile Research Journal*, 74 (3), 2004, pp. 222-226.
- [3] Rengasamy, R.S., Kothari, V.K., Patnaik, A., Puneekar, H., "Airflow simulation in nozzle for hairiness reduction of ring spun yarns. Part I: Influence of airflow direction, nozzle distance, and air pressure", *Journal Of Textile Institute*, 97 (1), 2006, pp. 89-96.
- [4] Subramanian, S.N., Venkatachalam, A., Subramaniam, V., "Effect of some nozzle parameters on the characteristics of jet-ring spun yarns", *Indian Journal Of Fibre & Textile Research*, 32, 2007, pp. 47-52.
- [5] Yilmaz, D., Usal, M.R., "Characterization of Jetring Yarn Structure and Properties", *Science and Engineering of Composite Materials*, 18 (3), 2011a, pp. 127-137.
- [6] Yilmaz, D., Usal, M.R., "A Comparison of Compact-Jet, Compact, and Conventional Ring-Spun Yarns", *Textile Research Journal*, 81(5), 2011b, pp. 459-470.
- [7] Yilmaz, D., Usal, M.R., "Effect of Nozzle Structural Parameters on Hairiness of Compact-Jet Yarns", *Journal of Engineered Fibres and Fabrics*, Volume 7, Issue 4, 2012.
- [8] Sawhney, A.P.S., Kimmel, L.B., "Air and Ring Combination in Tandem Spinning", *Textile Research Journal*, 67 (3), 1997, pp. 217-223.
- [9] Jeon, B.S., "Effect of an air-suction nozzle on yarn hairiness and quality", *Textile Research Journal*, 70 (11), 2000, pp. 1019-1024.
- [10] Cheng, K.P.S., Li, C.H.L., "JetRing Spinning and Its Influence on Yarn Hairiness", *Textile Research Journal*, 72 (12), 2002, pp. 1079-1087.
- [11] Zeng, Y.C., Yu, C.W., "Numerical Simulation of Air Flow in the Nozzle of an Air-Jet Spinning Machine", *Textile Research Journal*, 73 (4), 2003, pp. 350-356.
- [12] Guo, H.F., Chen, Z.Y., Yu, C.W., "Numerical Study Of An Air-Jet Spinning Nozzle With A Slotting-Tube", *Journal Of Physics: Conference Series*, 96, 2007, pp.1-5.
- [13] Rengasamy, R.S., Kothari, V.K., Patnaik, A., Ghosh, A., Puneekar, H., "Reducing Yarn Hairiness in Winding by Means of Jets: Optimisation of Jet Parameters, Yarn Linear Density and Winding Speed", *AUTEX Research Journal*, 5 (3), 2005, pp. 127-132.
- [14] Patnaik, A., Rengasamy, R.S., Kothari, V.K., Bhatia, S.K., "Some Studies on Hairiness Reduction of Polyester Ring Spun Yarns by Using Air-Nozzles During Winding", *TJTI*, 99 (1), 2008, pp. 17-27.
- [15] Patnaik, A., Rengasamy, R.S., Kohari, V.K., Ghosh, A., "Hairiness reduction of yarns by nozzles at ring spinning airflow simulation approach", *JTATM*, 4 (4), 2005, pp. 1-11.
- [16] Patnaik, A., Rengasamy, R.S., Kothari, V.K., Puneekar, H., "Airflow Simulation In Nozzle For Hairiness Reduction Of Ring Spun Yarns. Part II: Influence Of Nozzle Parameters", *Journal Of Textile Institute*, 97 (1), 2006, pp. 97-101.
- [17] Rengasamy, R.S., Patnaik, A., Anandjiwala, R.D., "Simulation Of Airflow In Nozzle-Ring Spinning Using Computational Fluid Dynamics: Study On Reduction In Yarn Hairiness And The Role Of Air Drag Forces And Angle Of Impact Of Air Current", *Textile Research Journal*, 78 (5), 2008, pp. 412-420.
- [18] Guo, H.F., Chen, Z.Y., Yu, C.W., "Simulation Of The Effect Of Geometric Parameters On Tangentially Injected Swirling Pipe Airflow", *Computers & Fluids*, 38, 2009a, pp. 1917-1924.
- [19] Guo, H.F., Chen, Z.Y., Yu, C.W., "Numerical Study Of The Function Of The Slotting-Tube In Air-Jet Spinning", *The Journal Of Textile Institute*, 2009b, pp. 1-7.
- [20] ANSYS, *ANSYS CFX-Solver Theory Guide*, ANSYS Inc., 121p., 2006, USA.
- [21] Yilmaz, D., "Development and Numerical Modelling of Plied Yarn Production Process Based On the Usage of High Velocity Air", Suleyman Demirel University, PhD Thesis, 2011, 332 p., Isparta, Turkey.
- [22] Çengel, Y., Cimbala, J.M., *Fluid Mechanics: Fundamentals and Applications*, McGraw Hill Higher Education (ISBN 978-0071115667), 2005, 864 p., USA.