Numerical Study of an Air-Jet Spinning Nozzle with a Slotting-tube

Hui-Fen Guo1, Zhi-Yong Chen2, Chong-Wen Yu1*

1 College of Textiles, Donghua University, Shanghai 201620, P. R. China
2 Shanghai Institute of Applied Mathematics And Mechanics, Shanghai University, Shanghai 200072, P. R. China

E-mail: yucw@dhu.edu.cn

Abstract. Slotting-tube is a critical component of a nozzle of an air-jet spinning machine. It affects flow characteristics in the nozzle, and the fibres motion and yarn properties as well. A realizable $k$-$\varepsilon$ turbulence model is adopted to simulate the airflow field inside the nozzle with the slotting-tube.

1. Introduction

It is well known that air-jet spinning can offer advantages in respect of processing speed and cost. The heart of this spinning technology is the design of the nozzle. Several researchers have used experimental and numerical methods to study the influence of the parameters such as nozzle pressure [1-4], injector angle [2, 3], and the diameter of the twisting chamber [2, 3] on the spun yarn properties. However, researchers have paid little attention to the influence of the slotting-tube. Only Yu [5] and Gao et al. [6] used spinning experimental methods to study the influence of different-shape slotting-tube and geometric parameters of rectangular grooves on the yarn tenacity, respectively. From the standpoint of the fluid mechanics, the slotting-tube can affect flow characteristics in the nozzle, and therefore affect the fibres motion and yarn properties. Hence, in this work, we focus our attention on the flow field in the nozzle with the slotting-tube.

A swirling recirculating flow can be formed in the nozzle with the slotting-tube. The standard $k$-$\varepsilon$ two- equation turbulence model has been widely applied to engineering practice, but has been criticized as being only qualitatively correct in the simulation of confined swirling flows [7,8]. This is because of the neglect of anisotropic viscosity and additional turbulence generation arising from the effects of streamline curvature in the standard $k$-$\varepsilon$ model [7]. Hence, some corrections [8-10] concerning the standard $k$-$\varepsilon$ model have to be made in order to simulate confined swirling flow. However, none of such modification was found to work well under all kinds of geometries and for a wide range of swirl number [8-10].

The realizable $k$-$\varepsilon$ turbulence model is an eddy viscosity model, which consists of a new model dissipation rate equation and a new realizable eddy viscosity formulation proposed by Shih et al [11]. It has shown substantial improvements on the standard $k$-$\varepsilon$ model where the flow features include strong streamline curvature, vortices and recirculation. Hence, the realizable $k$-$\varepsilon$ model is adopted to simulate flow characteristics in the nozzle with the slotting-tube. Through these works, the functions

* To whom any correspondence should be addressed.
of the slotting-tube are demonstrated. And the effect of nozzle pressures on the flow characteristics and yarn properties are also discussed.

2. Theoretical model

2.1. Nozzle Structure

Normally the nozzle is made cylindrical shape and the slotting-tube has four-rectangular grooves in spinning process (Figure 1). In this research, the computational domain is the twisting chamber of the nozzle from the inlet to the outlet, including the injectors and the grooves. And a Cartesian coordinate system as illustrated in Figure 1 is used. The origin of the coordinates system is located at the center of the nozzle inlet. The z-axis is taken as the streamwise direction and the x-y plane is perpendicular to the z-axis (i.e. the nozzle inlet). According to spinning experiments [2, 5], the twisting chamber diameter $D$ is 2mm, the nozzle length $L$ is 33mm, the diameter of the injector $d$ is 0.45mm, the injection angle $\theta$ is 45˚, and the position of the injector $l_1$, which is the distance from the injector to the inlet, is 11mm. The groove’s length $l_2$, depth $h$ and width $w$ are 8mm, 0.8mm and 0.3mm, respectively.

![Figure 1. Geometrical profiles of 3D model and projections of the nozzle with the slotting-tube.](image)

2.2. Numerical Model

As compressed air is forced into the twisting chamber through the injectors from the air reservoirs, its Mach number is large (on the range of 0.6 to 0.9) and the twisting process occurs in a very short time. Therefore, three-dimensional, steady, viscous turbulent flow of a perfect gas (air) in the absence of body forces is considered [12].

A finite volume technique is used to discretize the governing equations inside the computational domain. Due to compressible effects, the coupled implicit approach is adopted. It performs a simultaneous solution of the conservation of mass, momentum, energy, turbulent kinetic energy and its dissipation rate equations within the physical domain. The conservation equations are solved using the second-order upwind scheme, and the other ($k$ and $\varepsilon$) equations use the QUICK scheme of which provides high accuracy for swirling flows. In order to accelerate the convergence of the solver, a block Gauss-Seidel algorithm is used in conjunction with an algebraic multigrid (AMG) method to solve the discretized equations.

2.3. Boundary Conditions

The inlet boundary: Because the pressure of the air reservoir is known, pressure inlet condition is used at the injector inlets. However, at the nozzle inlet, while the fibers or strands output from the front
roller and go into the nozzle, the outer air is supplied into the nozzle, so velocity inlet boundary can be set.

The outlet boundary: At the nozzle outlet, the pressure is supposed to be the external pressure.
The wall boundary: At wall, non-slip boundary condition is applied.

3. Results and Discussion
The velocity vector distributions of the different sections are shown in Figure 2. A weak swirling balloon is formed in upstream of the injectors due to reverse flow near the wall caused by reverse jet, and its direction of twist is contrary to that of the downstream (Figure 2 (a, b)). At a distance about 5mm (viz., \(z = 16\text{mm}\)) downstream of the injectors, the velocity vectors in the core region show flow reversals, i.e., internal recirculation zones (IRZ) are generated as a result of the asymmetrical vortex breakdown, see also Figure 2 (a). And near the wall a small corner recirculation zone (CRZ) is found behind the sudden expansion of the slotting-tube. The internal recirculation is the result of the adverse pressure gradient at the axis created by the swirl, while, the corner recirculation is caused by the flow separation from the sidewall of the groove (i.e. backward-facing step) as the fluid stream enters. These flow phenomena, such as IRZ and CRZ, have been observed [10]. The velocity vectors at the cross section show that the velocity increases gradually with radius in the central region and reaches the largest near the wall of the twisting chamber, thereafter, it decreases rapidly in the grooves (Figure 2 (c)). Hence, large velocity gradient exists in the shear layer (see also Figure 4). And as shown in Figure 2 (c), there are air currents in mutually opposite direction in the grooves and the twisting chamber.

![Figure 2](image)

*Figure 2.* Velocity vector plots of the nozzle with slotting-tube: (a) the \(y-z\) plane at \(x=0\text{mm}\); (b) the \(y-z\) plane at \(x=0.6\text{mm}\) (c) the \(x-y\) plane at \(z=26\text{mm}\)

In accordance with flow characteristics, when the fibre strands pass through the nozzle with slotting-tube, the weak opposite swirling balloon in the injectors upstream can delay the edge fibre end to be picked up by the core strand. It’s helpful to form a larger twist difference between the edge fibres and the core ones. Besides the buffeting of the vortex breakdown, the compound force acting on the edge fibres, including airflow counterforce in the grooves and impact force between fibre and the vertexes of the grooves, make the edge fibres to loose wrapping and produce more wrapping fibres. Hence, the function of the slotting-tube can be demonstrated and yarn tenacity can be improved. The results are supported by the experimental study of Gao et al. [6]. According to these results, it can be
inferred that the number of the free edge fibres increases with the number of the grooves as the other parameters keep constant, because the intensity and extent of the opposite direction airflow increase, yarn tenacity increases consequently. But, it is difficult to machine too more grooves, and too more edge fibres increase random wrapping, as a result, yarn strength decrease [5].

To further demonstrate the functions of the slotting-tube, at the same condition, the flow characteristics in the nozzle without slotting-tube (i.e., the whole nozzle is a cylindrical tube) are also studied. Comparison is done by plotting the streamline plots in a longitudinal section along the z-axis in Figure 3 for the nozzles with and without the slotting-tube. For the nozzle with the slotting-tube, the size of the recirculation near upstream wall increases and its location shifts towards the nozzle inlet, and the occurrence of the IRZ is much earlier and larger than the counterpart of the nozzle without the slotting-tube. Thus, both larger twist difference between the edge fibres and the main fibre strand and more wrapping fibres are formed, as a result, yarn tenacity is improved significantly for the nozzle with the slotting-tube [6].

![Figure 3. Predicted streamline plots in a longitudinal section along the z-axis: (a) without the slotting-tube; (b) with the slotting-tube.](image)

Several spinning experimental studies showed that the nozzle pressure is the most important parameter affecting the mechanical properties of air-jet yarns [1-3]. As shown in Figure 4, for all cases, it is observed that there is a larger velocity gradient in the shear layer, and the tangential velocity in the grooves is opposite to that of the twisting chamber. In addition, both velocity gradient and velocity decay quickly along streamwise direction. With increasing the nozzle pressure, both the velocity and the strength of vortex breakdown increase while the area of vortex breakdown reduces. This is because as the nozzle pressure increases, both the fluid flow becomes much more unstable and vortex breakdown in the core zone occurs at increasingly earlier stage. Therefore, it can be inferred that yarn tensile should be increased with the increase of the nozzle pressure. However, note that the increase in velocity tends to decline when the nozzle pressure is higher than $2.5 \times 10^5$ Pa. And if nozzle pressure is too high, strong reverse-jet of the injector upstream will not generate sufficient suction to draw the fibers into the nozzle. And strong buffeting of the vortex breakdown will produce too more wrapper fibers at high pressure and may lead to random wrapping of the wrapper fibres [1,2]. From perspective of energy conservation and the results of the simulation, it is not sensible to increase nozzle pressure to a very high level. And we suggest that the rational pressure of the nozzle is from $2.0 \times 10^5$ Pa to $3.0 \times 10^5$ Pa.
4. Conclusions

In this paper, a realizable $k$-$\varepsilon$ model is adopted to simulate airflow in the nozzle with the slotting-tube. There are air currents in mutually opposite direction in the grooves and the twisting chamber, it is helpful to disperse fibre bundle. As compared with the non-slotting-tube case, the strength and size of both the recirculation near the upstream wall of the injectors and vortex breakdown in the downstream core of the injectors increase for the nozzle with the slotting-tube. Therefore, yarns with better tensile properties can be produced. High nozzle pressures can generate high velocities and strong vortex breakdown. Subsequently, yarns properties can be improved. However, if pressure is increased beyond a certain optimum value, too more wrapper fibres may lead to random wrapping and yarn strength instead decreases.

Acknowledgements

This work was supported by the Specialized Research Fund for the Doctoral Program of Higher Education (SRFDP) in China under grant No. 20040255009.

References

[1] Lawrence C A, Baqui M A 1991 Textile Res. J. 61 123
[2] Oxenham W, Basu 1993 Textile Res. J. 63 674
[3] Zeng Y C, Yu C W 2003 Textile Res. J. 73 350
[4] Liu Y, Xu L 2006 Int. J. Nonlinear Sci. Num. Simu. 7 389
[5] Yu C W, 1997 J. Textile Res. 18 75
[6] Gao F, Gu X F, Yao M 1993 J. Textile Res. 14 199
[7] Jones W P, Pascua A 1989 J. Fluids Eng. Trans. ASME 111 248
[8] Nallasamy M 1987 Comput. Fluids 3 151
[9] Khodadadi J M and Vlachos N S 1990 AIAA J. 28 750
[10] Chang K C and Chen C S 1993 Int. J. Num. Meth. Fluids 16 421
[11] Shih T H, Liou W W, Shabbir A, Zhu J 1995 Computers Fluids 24 227
[12] Versteeg H K, Malalasekera W 1995 An introduction to computational fluid dynamics: the finite volume method. (New York: Wiley)