Rotor spinning transfer channel design optimization via CFD

Abstract

The conventional rotor spinning unit generates flow vortices in the transfer channel upstream region which affect the fiber configuration and consequently yarn properties. Geometry and spinning parameters such as transfer channel length, inlet width, rotor outlet pressure, opening roller speed and diameter were found to be key parameters influencing airflow characteristics. To reduce the flow vortices in the upper stream region, modifications of the transfer channel were proposed, and their airflow fields were analyzed using Computational Fluid Dynamics (CFD). Three designs were studied; round transfer channel inlet, a bypass channel for extra air supply and one with both the bypass and the round inlet. Analysis of airflow revealed that the design with both round transfer channel inlet and a bypass proved to be very effective in properly directing the flow and minimizing vortices. The design was also characterized by smoother velocity streamlines and maximum mass flow across the transfer channel. A conventional rotor spinning unit was modified in which a round transfer channel inlet corner and a bypass channel were utilized to conduct the experimental tests. Three sets of yarn samples were produced using the conventional and modified rotor spinning units under different rotor speed conditions. Yarn properties were tested. Properties such as tenacity, CVm%, thin and thick places of the spun yarns produced by the new design improved compared to that of the conventional yarn.

Keywords

Air flow, rotor spinning, transfer channel, yarn, CFD simulation

Introduction

Of all the new spinning technologies, rotor spinning is the most extensively used. Rotor spinning has been developed into a high yield technology, with an expansion in the variety of rotor-spun yarns during the last few decades [1-3]. However, it is undeniable that rotor spun yarn tenacity is lower than that of ring spun yarn of comparable cross section, which is because almost 50 percent of fibers in the rotor spun yarn are curly, bending, intertwined and folded. Increasing fiber straightness in a rotor spun yarn is an effective way to increase the yarn tenacity. In production, enhancing the drawing and carding in the fore-spinning process to improve raw materials (fiber slivers) opening and straightening is commonly adopted by the manufacturers. The adequate separation of fibers by the opening roller is also necessary. However, fiber breakage can occur during the carding process, leading to the insufficient use of single fiber strength and eventually a reduction of yarn quality [4, 5]. If we consider the rotor spinning process, it mainly contains four stages. Firstly,

the sliver is pulled through a condenser by the feed roller and then combed by the opening roller. The individual fibers are transported through the transfer channel by the airflow. Thirdly, the fibers are collected in the rotor groove from the transfer channel outlet, and finally, the fibers are twisted into yarn. Individual fibers are separated, transported and then collected again during these stages, leading to a loss of fiber orientation and straightness. During fibre transportation in the transfer channel, the risk of fiber bending may be increased due to the possible adverse airflow which is closely related to the rotor structure and spinning parameters.

To obtain better yarn properties, research on the design of the rotor spinning unit has been conducted experimentally over the past few decades [6-9]. Lawrence and Chen [9, 10] investigated fiber removal from the opening roller and fiber trajectories in the transfer channel using high-speed cinematography. Their analysis drew a conclusion that a transfer channel with narrow rectangular cross-section was more conducive to generate better fiber configuration compared to that with a circular one. They also stated that the relative velocity between airflow and opening roller surface in the fiber separation area was a crucial factor for fiber configuration. The air speed should be larger than the opening roller surface speed to provide fiber with a continual acceleration and prevent fiber bending. In Eskandarnejad's thesis [11], the transfer channel was modified that its top and side walls were extended 16 mm above the trash ejection slot, which could reduce fiber flicking over the outer wall of the channel and restrict the displacement of the fibers prior to the channel inlet. Zhang et al. [12] studied the transfer channel experimentally and led to an optimized design for the transfer channel by cluster analysis. They recommended that, either increasing transfer channel inlet air velocity or reducing channel inlet dimension could be advantageous to fiber-straightening. A transfer channel with a narrow slot perpendicular to the rotor axis at the channel outlet was proposed. Seyed et al. [13] discussed the effect of geometry of the slot on yarn properties and found that a reduction of the slot width could increase yarn tenacity.

With the aid of computational fluid dynamics, Kong and Platfoot [14] developed a two-dimensional computational model of the transfer channel. They analyzed the effect of transfer channel inlet dimensions, the ratio of opening roller circumferential velocity to mean flow velocity and Reynolds number on transfer channel airflow patterns. Findings revealed a recirculation zone generated in the transfer channel inlet, which was affected by any variation of those factors. When evaluating fiber movement, their simulation results showed that the recirculation zone could not only cause fiber bending but also increase fiber transportation time within the channel, thus increasing the risk of fiber collision with the wall [15]. Smith and Roberts [16] modeled the fiber motion in a converging transport duct. They demonstrated that a larger air velocity difference along the transport channel would achieve more straightened fibers. Lin et al. [17] developed a three-dimensional numerical model to simulate the airflow characteristics inside the rotor. They declared that a small rotor speed (60000 rpm) generated vortices around the rotor groove, which were responsible for fiber buckling. When rotor speed was 100000 rpm or more, the velocity around the rotor groove was increased and maintained stable. Increasing rotor

speed and rotor diameter or decreasing the rotor slide wall angle could reduce the vortices formation in the rotor, and therefore contribute to yarn quality improvement.

The current paper contains two parts. The first is to study the influence of some geometric and spinning parameters on the transfer channel airflow patterns based on the conventional rotor spinning unit. The second is to propose transfer channel modifications generating no air vortices inside the transfer channel. The aim of this work is to evaluate the rotor spinning unit numerically and experimentally to improve fiber configuration and hence induce improved rotor spun yarn properties.

Theoretical model

Main characteristics of the rotor spinning unit

Based on JWF1612-2157A rotor spinning unit, the rotor spinning unit model along with its main dimensions is presented in Figure 1. In the present study, the computational domain includes the rotor, the yarn guiding mouth, the transfer channel, the opening roller central part and the trash extraction chute. The rotor has a total height of 13 mm with an estimated outlet dimension of 1 mm. The rotor slide wall angle is 66° and the lower rotor diameter is 36 mm. The internal and external diameters of the opening roller are 65 mm and 69 mm, respectively. The trash extraction chute is directly connected to the atmosphere with a channel length of 30 mm and a width of 6 mm. The transfer channel connects the rotor and the opening roller. The transfer channel upper and lower sections are respectively $18.5 \times 6 \text{ mm}^2$ and 7.3 mm^2 . The transfer channel geometry is shown in Figure 1c). The transfer channel inlet width W is the length of the two intersections between the transfer tube and the opening roller housing. The inlet height H is defined as the perpendicular distance from the intersection between the channel short side and the opening roller housing to the channel long side. The transfer channel length L is the length of the center line from its inlet to its outlet, and its original value is 40 mm.

Due to the pressure difference generated between the trash extraction chute and the rotor outlet, air enters the trash extraction chute and flows along the opening roller surrounding housing. Then it follows the path established by the transfer channel and enters the rotor where the yarn will be generated. Finally, the air leaves across the circular rotor outlet. It is noticeable that fibers are fed into the rotor spinning unit through a channel located on the left of the trash extraction chute. Although the fibers are not considered in the present study, the fiber inlet is constantly packed with slivers when the machine is working, which will block the entrance of the atmospheric air. Therefore, the fiber inlet is not depicted in Figure 1.

According to previous research [9, 13, 14], many design parameters of the rotor spinning unit were believed to affect the airflow characteristics and hence yarn properties. One of the biggest problems associated with the transfer channel actual design was that it generated a negative vortex at the transfer channel inlet [14]. Such vortex reduced the effective channel section, and among other problems, created neps due to the trapped fibers. To evaluate the effect of the geometric and spinning parameters of the rotor spinning unit on the vortices in the transfer channel, the

transfer channel length, inlet width, opening roller internal diameter, opening roller speed and rotor outlet relative pressure were studied via CFD simulation. Also, to reduce such vortex, three possible modifications were discussed. The first one was to create a bypass channel, which was placed on the right side of the transfer channel with its flow direction tangent to the transfer channel. The air was injected either automatically due to the pressure difference between the bypass inlet and the rotor, or by an air blower which can adjust the supply air mass flow rate. The bypass inlet boundary condition was set either as atmospheric pressure inlet or velocity inlet, with the velocity ranging from 2 to 8 m/s. The second method consisted in rounding the transfer channel inlet corner in order to facilitate airflow, and hence three connecting radii of 0, 1.5 and 3 mm were evaluated. The third design was to combine both the bypass channel and the round transfer channel inlet. Figure 2 specifies what it is understood as transfer channel inlet left-hand side rounded corner radius, and defines as well the bypass channel position and its dimensions.

The different sets of parameters evaluated are presented in Tables 1 and 2. Table 1 specifies the four different transfer channel lengths, opening roller diameters, opening roller speeds and rotor outlet pressures studied. The second table clarifies the transfer channel modifications performed to minimize the undesired transfer channel inlet vortex. The rest of parameters were kept constant as specified in Case 2.

The different initial boundary conditions employed for the simulations were: opening roller speed 6000 rpm, clockwise; rotor speed 100000 rpm, clockwise; and rotor outlet relative pressure -7000 Pa. The trash extraction chute and yarn guiding mouth absolute pressure were set to 101000 Pa. Initially, the bypass and the round inlet were not considered. Non-slip boundary condition was taken into account in all walls. All simulations were undertaken using ANSYS-FLUENT version 14.0.

Figure 1. a) Geometry and dimensions of the rotor spinning unit; b) upper view; c) side view.

Table 1. Initial set of parameters evaluated

Case	Transfer channel length L (mm)	Roller speed ω (rpm)	Roller internal diameter $\varnothing$ (mm)	Rotor outlet relative pressure P (Pa)
1	28	6000	65	-7000
2	40	6000	65	-7000
3	52	6000	65	-7000
4	64	6000	65	-7000
5	40	4800	65	-7000
6	40	7200	65	-7000
7	40	8400	65	-7000
8	40	6000	60	-7000
9	40	6000	70	-7000
10	40	6000	80	-7000
11	40	6000	65	-6000
12	40	6000	65	-8000
13	40	6000	65	-9000

Figure 2. Modifications of the rotor spinning unit; a) with a bypass channel and b) with the left corner rounded and a bypass.

Table 2. The second set of parameters evaluated

Case	Transfer channel inlet width	Corner Radius	Bypass	Bypass inlet boundary conditions
------	------------------------------	---------------	--------	----------------------------------

	W (mm)	r (mm)		
A	18.5	1.5	-	-
B	18.5	3.0	-	-
C	18.5	-	Yes	inlet atmospheric pressure
D	18.5	-	Yes	inlet velocity 2 m/s
E	18.5	-	Yes	inlet velocity 5 m/s
F	18.5	-	Yes	inlet velocity 8 m/s
G	20.5	-	-	-
H	20.5	1.5	-	-
I	20.5	3.0	-	-
J	15.5	-	-	-
K	15.5	1.5	-	-
L	15.5	3.0	-	-
M	18.5	1.5	Yes	inlet atmospheric pressure
N	18.5	1.5	Yes	inlet velocity 2 m/s
O	18.5	1.5	Yes	inlet velocity 5 m/s
P	18.5	1.5	Yes	inlet velocity 8 m/s

Governing equations and turbulence model

For simplification, the simulations were conducted without introducing fibers. Because the rotor spinning unit was of relatively small size, and it took a very short time, less than 0.1 seconds, to finish a complete spinning process, heat-transfer was ignored. Since the maximum Reynolds number in the rotor spinning unit was very high, $Re = 70163$ for the current case, and it was $Re = 64000$ in Ref. [15], the airflow had to be regarded as turbulent. Therefore, in this paper, the airflow in the rotor spinning unit was assumed to be turbulent, viscous, incompressible and isothermal.

According to the previous statements, the turbulent flow governing equations will be reduced to continuity and momentum equations. The system should be closed via turbulent model characteristic equations. For the present study, the κ - ε turbulence model was chosen due to its proved reliability in evaluating internal flows. In Cartesian coordinate system, the general expression of the governing equations can be written as:

$$\frac{\partial(\rho\psi)}{\partial t} + \text{div}(\rho\mathbf{u}\psi) = \text{div}(\Gamma\text{grad}\psi) + S \quad (1)$$

Where ρ is the fluid density, and $\mathbf{u}$ is a velocity vector. ψ represents a generic variable. Γ is the diffusion coefficient and S is the general source term.

The different terms of the governing equations, when considering the κ - ε turbulence model, are illustrated in Table 3. The parameter p is pressure, u , v and w are the velocity components in x , y and z -direction, respectively. μ_{eff} is the effective viscosity, which is the sum of the fluid viscosity μ and the Reynolds apparent viscosity μ_t . The turbulent kinetic energy κ and turbulent dissipation rate ε are related to μ_t by the expression $\mu_t = \rho C_\mu \kappa^2 / \varepsilon$. C_μ , $C_{1\varepsilon}$ and $C_{2\varepsilon}$ are empirical constants. G_k is the generation of turbulent kinetic energy κ due to the mean velocity gradients. σ_κ and σ_ε represent the turbulent Prandtl numbers corresponding to κ and ε , respectively.

The turbulent model constants have the default values of: $C_\mu = 0.09$; $C_{1\varepsilon} = 1.44$; $C_{2\varepsilon} = 1.92$; $\sigma_k = 1.0$ and $\sigma_\varepsilon = 1.3$.

Table 3. The terms of the governing equations considering the κ - ε model

Equation	ψ	Diffusivity Γ	Source term S
Continuity	1	0	0
x-momentum	u	$\mu_{eff} = \mu + \mu_t$	$-\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}(\mu_{eff} \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y}(\mu_{eff} \frac{\partial v}{\partial x}) + \frac{\partial}{\partial z}(\mu_{eff} \frac{\partial w}{\partial x}) + S_u$
y-momentum	v	$\mu_{eff} = \mu + \mu_t$	$-\frac{\partial p}{\partial y} + \frac{\partial}{\partial x}(\mu_{eff} \frac{\partial u}{\partial y}) + \frac{\partial}{\partial y}(\mu_{eff} \frac{\partial v}{\partial y}) + \frac{\partial}{\partial z}(\mu_{eff} \frac{\partial w}{\partial y}) + S_v$
z-momentum	w	$\mu_{eff} = \mu + \mu_t$	$-\frac{\partial p}{\partial z} + \frac{\partial}{\partial x}(\mu_{eff} \frac{\partial u}{\partial z}) + \frac{\partial}{\partial y}(\mu_{eff} \frac{\partial v}{\partial z}) + \frac{\partial}{\partial z}(\mu_{eff} \frac{\partial w}{\partial z}) + S_w$
Kinetic energy	κ	$\mu + \frac{\mu_t}{\sigma_k}$	$G_k + \rho\varepsilon$
Turbulent dissipation rate	ε	$\mu + \frac{\mu_t}{\sigma_\varepsilon}$	$\frac{\varepsilon}{k}(C_{1\varepsilon}G_k - C_{2\varepsilon}\rho\varepsilon)$

Grid independence test

The whole computational domain was meshed using a mix of structured hexahedral and unstructured tetrahedral grids. Figure 3 presents the overall physical domain and some zoomed views of the final grid used. The grid density is especially high nearby solid boundaries, especially in the vicinity of the opening roller and the rotor wall. In reality, four different grid sizes were considered. The number of cells employed for each case was 668740, 1409506, 2001986 and 3087363, respectively. For each mesh, the same boundary conditions were employed. Pressure and velocity distributions were evaluated at two different positions along the transfer channel central axis (A-A), and across the inner rotor diameter (B-B), 3 mm above the bottom, see Figure 3. The results are presented in Figures 4 and 5.

Figure 4 presents the relative pressure and velocity magnitude along the transfer channel central line for the four grids evaluated. It is observed that slight differences are to be spotted between the different grid configurations, which indicates that this line may not be the optimum one to evaluate the grid independency. For this reason, relative pressure, velocity magnitude and tangential velocity are assessed at the rotor central diameter, see Figure 5. From this figure, it can clearly be seen that for coarse meshes, pressure and velocity values obtained are not having the required accuracy. Once the mesh overcomes 2 million cells, the results are independent of the number of cells. When using 3 million cells, the results obtained have minor differences versus the ones achieved by using 2 million cells. However, the computational time required for 3 million cells is 50 hours and for 2 million cells it is 31 hours, in both cases using an i5-2.5GHz computer. Since the results obtained are almost identical and computational time required is much shorter for 2 million cells, all simulations presented are undertaken using the 2 million cells mesh.

Figure 3. Positions used to compare the different grid results and zoom views of the grid used.

Figure 4. a) Relative pressure and b) velocity magnitude along the transfer channel central axis, A-A.

Figure 5. a) Relative pressure, b) velocity magnitude and c) tangential velocity across the rotor central diameter, B-B.

At this point, it can easily be checked the hypothesis of flow incompressibility. From Figures 4 and 5, it is observed that the maximum velocity is about 110 m/s,

therefore the maximum Mach number has to be around 0.32. The error when using incompressible fluid equations instead of compressible ones and when the air velocity is 110 m/s, is about 2.56%, which is acceptable in most engineering problems, and so will be in the present case, especially when considering that this maximum speed just appears at the transfer channel outlet and the external rotor diameter.

Another interesting point is that the tangential velocity graph, Z velocity, presented in Figure 5, clearly demonstrates that inside the rotor the fluid is turning, generating a forced vortex. Notice that fluid velocity increases with the radius increase, and according to Euler's equation, the minimum pressure will be found in the vortex center.

Results and discussion

Main generic flow characteristics

In the spinning process, the rotor spinning unit is almost closed, air enters the trash extraction chute and leaves through the rotor outlet, which is connected to a suction pump. Therefore, when the suction pump extracts the air inside the rotor, a negative pressure zone is formed within the whole rotor, see Ref. [17]. Due to the transfer channel inlet dimensions and shape, under the standard working conditions, a vortex is generated on one side of its upstream region, see Figure 6. The existence of the vortex located at the transfer channel inlet reduces the channel mass flow and greatly affects the movement and parallelization of the fibers inside the channel. Fibers peeled off from the opening roller are vulnerable to curling when in contact with the severe vortex.

Figure 6. Streamlines in the plane $z = 0$ mm of Case 2. A-A: along the transfer channel central axis; B-B: across the rotor center in $y = 3$ mm plane; C-C: across the vortex center; D-D: across the transfer channel inlet; Section S: fiber separation region.

When the airstream coming from the transfer channel reaches the rotor, due to the fact that air enters the rotor laterally, and considering as well that the rotor turns at a very high speed, 100000 rpm, air rotates in the x - z plane around the internal rotor part as defined in Figure 5. The flow leaves the rotor towards the suction pump through

the rotor outlet while the yarn leaves through the yarn guiding mouth. Inside the rotor, the main vortex appears in the x - y plane, and it is located at the opposite side versus the transfer channel outlet, see Figure 6. The detailed discussions on the airflow characteristics inside the rotor were presented in Ref. [17]. Therefore, the present paper will focus on how the parameters defined in Tables 1 and 2 will influence the airflow characteristics of the transfer channel, and as a result, how yarn properties are expected to be modified.

Effect of transfer channel length

In this section, the effect of transfer channel length on the spinning unit flow characteristics and the likely fiber behavior inside the rotor spinning unit is studied.

Figure 7a) presents the velocity towards the y direction at the transfer channel inlet width (line D-D in Figure 6). The velocity appears to be maximum nearby the walls, while in the center the flow remains rather static. This indicates that most of the fibers escape from the opening roller pins at the right-hand side of the transfer channel inlet, called fiber separation region (S) (see Figure 6), rather than in the center region. This is due to the reason that the small aerodynamic force in the channel inlet center, where velocity vertical component is nearly zero, is not likely to overcome the metal/fiber frictional force and the component of the centrifugal force, which tends to drag the fibers along with the opening roller.

To better understand the flow behavior, Figure 8 presents the velocity streamlines along the transfer channel. It is seen that a negative vortex, increasing as the transfer channel length increases, appears at the transfer channel inlet left-hand side. As can be observed in Figure 7a), when the channel length is 28 mm, a small positive vortex appears on the transfer channel inlet right-hand side, shifting the mainstream flow slightly towards the left. These vortices not only reduce the effective width, defined as the width of a cross section containing forward flow [13], but also increase the possibility of fiber buckling, leading to the deterioration of yarn properties. Also, they are slightly reducing the mass flow through the transfer channel, as shown in Figure 7b). The decrease in mass flow to the rotor will increase the fiber transportation time through the transfer channel. This will increase the risk of fiber collision with the wall, thus decreasing fiber configuration.

It is observed from Figure 7a) that the maximum velocity of the mainstream flow located at the inlet right-hand side increases as the transfer channel length increases, until reaching a length of 52 mm. A further transfer channel length increase tends to reduce the maximum velocity and the mass flow. To understand this effect, two main parameters need to be considered. The first one is the positive vortex appearing at small transfer channel lengths, and disappearing as the transfer channel length increases. The disappearance of this particular vortex is responsible for the velocity and mass flow increase as transfer channel length increases from 28 to 52 mm. The shear stresses, which is the second parameter to be considered, have a secondary effect. A further increase of the transfer channel length reduces the stream velocity and therefore the mass flow. Because no positive vortex exists, the flow is driven by the shear stresses effect, and shear stresses increase with the length increase.

As a conclusion from Figures 7 and 8, it can be stated that the maximum mass flow is to be accomplished when using transfer channel lengths between 40 and 52 mm. Smaller transfer channels generate two vortices at the inlet, a negative one on the left-hand side and a positive one at the right-hand side. The second one is strongly affecting the mainstream flow, but both vortices tend to promote fiber entangling.

Figure 7. a) Vertical velocity distribution and b) mass flow for different transfer channel lengths.

Figure 8. Vortices generated at the transfer channel inlet for different transfer channel lengths.

Effect of opening roller speed

Figures 9 to 11 represent the effect of opening roller speed on the flow key features. From Figure 9a), it is observed that as roller speed increases, the vertical velocity peak, located on the transfer channel inlet right side, keeps decreasing, being as well displaced towards the left side. This effect is understood when evaluating the flow characteristic presented in Figure 10. As roller speed increases, the positive vortex generated on the transfer channel inlet right-hand side gradually enlarges, shifting the mainstream flow towards the left and at the same time decreasing the vertical velocity. This positive vortex also reduces the effective width at the fiber separation region (S), especially when the roller speed is 8400 rpm, the effective width is only about 50% of the geometric width. Regarding the positive vortex, it is relevant to highlight that the opposite effect was appearing when increasing the transfer channel length, see Figure 8. This vortex disappeared as length increased. According to Lawrence and Chen [10], fibers usually retain the configuration they have when entering the transfer channel, then the aerodynamic forces in the transfer channel help in maintaining the original fiber form. Therefore, if a fiber is removed from the opening roller pins in a bent form, it will hardly be straightened in the channel.

Regarding the main vortex located on the transfer channel inlet left-hand side, it is

seen from Figure 9b) that this particular vortex is not suffering considerable modifications, although slightly decreases in size with the increase of roller speed. The vertical lines in Figure 9b) indicate the likely location of the vortex right-hand side. It is also observed that as roller speed increases, maximum vortex tangential velocity tends to decrease at first and then increase, see the intersection between each curve and the vertical lines in Figure 9b), indicating that the vortex turning speed as well as its intensity may suffer a slight increase after a small decrease. This effect is negative when considering the likelihood of fibers being trapped in this section.

Figure 9. Velocity distributions at the sections D-D and C-C as a function of opening roller speed.

Figure 10. Vortices generated at the transfer channel inlet for different opening roller speeds.

Figure 11. a) Ratio of the mean velocity magnitude of the transfer channel inlet V_m to the circumferential velocity of the opening roller V_c , and b) mass flow for different opening roller speeds.

Figure 11a) introduces the ratio of the transfer channel average inlet velocity magnitude V_m , section D-D, to the circumferential velocity of the opening roller V_c . The ratio V_m/V_c declines as roller speed increases, showing that larger roller speeds have a weaker ability to peel off the fibers from the roller pins. From the point of benefiting fiber separation, it is desirable that the air velocity flowing into the transfer channel should be greater than roller circumferential velocity [10]. According to the results presented, it seems that air velocity may not be sufficient to peel off fibers from the opening roller. But in reality, the airflow on the roller right-hand side channel is about 40 m/s, and the tangential opening roller velocity is about 28 m/s when turning at maximum speed. This indicates air velocity is always higher, at the peeling point, than tangential velocity.

Notice that the velocity V_m in Figure 11a) is the mean velocity across the entire inlet width, which according to Figure 9a) slightly decreases with roller speed increase, while tangential velocity V_c increases linearly as roller speed increases. Figure 11b) clearly illustrates that mass flow decreases with the increase of roller speed, indicating that smaller roller speeds would be desirable. Considering that the intensity of the main vortex is higher for smaller roller speeds, see Figure 9b), it seems that a roller speed of 6000 rpm would be the optimum choice, which is close to the optimum value, 6000 to 6500 rpm, reported by Marino et al [18].

Effect of opening roller internal diameter

Four different opening roller internal diameters, 60, 65, 70 and 80 mm, were evaluated. The rest of the parameters were kept constant as defined in Table 1, see Cases 2, 8, 9 and 10. For each case, the external diameter was always 4 mm bigger than the internal one.

At this point, it is important to highlight that the effect generated by the opening roller internal diameter increase is expected to be similar to the one produced by the opening roller speed increase, since both increases generate an increase of the tangential velocity. Despite the expected similarity, when comparing the flow behavior exhibited in Figure 12a) and Figure 9a), it is seen from the right-hand side of Figure 12a) that as the opening roller diameter increases, the maximum velocity keeps increasing until reaching a diameter of 70 mm. A further diameter increase generates a low-intensity vortex at the transfer channel inlet right-hand side, displacing the flow towards the left and reducing the maximum Y-axis velocity. This effect can also be seen in Figure 14. Returning to Figure 9a), an opposite effect is observed. The maximum Y-axis velocity on the channel inlet right-hand side decreases with the increase of roller speed. This opposite effect is likely to be caused by the positive vortex produced on the channel inlet right-hand side.

Figure 12. Velocity distributions at the sections D-D and C-C as a function of opening roller internal diameter.

As displayed in Figure 12b), the size and strength increase of the main vortex located on the channel inlet left-hand side is produced when decreasing the roller diameter. However, from Figure 13, the mass flow fluctuates smoothly when the diameter is between 60 to 70 mm, whereas it undergoes a sudden drop when the diameter increases to 80 mm. This effect appears to be due to the vortex generated on the fiber separation region, shown in Figure 14. As a consequence, it seems unfavorable to adopt a roller diameter too small or too big in size. Therefore a diameter of around 70 mm could be the most desirable choice when considering both the productivity and yarn properties.

Figure 13. Mass flow for different opening roller internal diameters.

Figure 14. Vortices generated at the channel inlet for different opening roller internal diameters.

Effect of rotor outlet relative pressure

The vertical velocity distributions along the transfer channel inlet section D-D and across section C-C as a function of rotor outlet relative pressure is presented in Figure 15. It is seen that the velocity peaks on both sides of the transfer channel inlet increase

with the decrease of rotor outlet pressure, while the velocity distributions are highly similar in the channel inlet center. Although the size and strength of the vortex on the left side of the channel go up slightly (Figure 15b)), the velocity on the effective pathway for the fibers experiences a significant increase as the rotor outlet pressure decreases.

The mass flow trend presented in Figure 16 coincides with the velocity distributions in the transfer channel (Figure 15). Decreasing the rotor outlet pressure leads to a linear mass flow growth. However, it requires a higher energy consumption, and it seems that the transfer channel effective width constantly reduces as the rotor outlet pressure decreases (Figure 15b)). Since a higher mass flow increases the fiber transportation velocity along the transfer channel, more straightened fibers can be achieved. Furthermore, a higher amount of fibers can be fed and a corresponding increase in rotor speed is required, as a result, decreasing the rotor outlet relative pressure is beneficial to the productivity. However, at the same time, it boosts the risk of fiber buckling during its transfer to the rotor groove. This risk was proved by the results reported by Bauer et al. [19], showing that the Uster-CV slightly grew as the rotor negative air pressure increased from -60 mbar to -90 mbar. Therefore, the rotor outlet pressure cannot be further decreased without considering the adverse effects associated.

Figure 15. Velocity distributions for different rotor outlet relative pressures at sections D-D and C-C.

Figure 16. Mass flow for different rotor outlet relative pressures.

Rounding the transfer channel inlet corner with different inlet widths

For this particular section, three different inlet widths were studied. The original one corresponding to Case 2 had an inlet width of 18.5 mm. The other two corresponded

to Case G and Case J, with inlet widths of 20.5 mm and 15.5 mm respectively. Three different corner radii r were applied for each inlet width case, namely, 0, 1.5 and 3 mm.

Figure 17 presents the contours of streamlines in the transfer channel for all the cases studied in this section. For the original cases ($r = 0$ mm), it can be seen that as the transfer channel inlet width increases, the main vortex located on the channel inlet left-hand side has a minor increase. Besides, when the channel inlet width is 20.5 mm, a second vortex appears in the fiber separation region. As previously described, it is of vital importance for the fiber separation region to be free of vortices that may cause fiber choking, accumulation and inlet available area decrease. Hence a small channel inlet width is recommended to obtain high yarn quality. This conclusion coincides with that reported by Zhang [12] and Kong [14].

When rounding the inlet left-hand side corner with a radius of 1.5 mm, and maintaining the original dimensions, the main vortices in all studied cases as well as the small vortex appearing in Case G ($W = 20.5$ mm, $r = 0$ mm) disappear. Instead, a new very low-intensity vortex appears at the center of the field, although this new vortex is not affecting the mainstream flow. As a result, the mass flow increases significantly as illustrated in Figure 18. When further increasing the corner radius to 3 mm, the undesirable vortex arises again in the fiber separation region, with a general decreasing trend in its size as the inlet width increases. Increasing the corner radius from 1.5 mm to 3 mm has a small influence on mass flow for the inlet width of 18.5 mm. However, when the inlet widths are 15.5 mm and 20.5 mm, a decrease in mass flow can be seen. This variation can be understood when observing the vortices in the fiber separation region. It is important to consider that the higher the mass flow, the larger amount of fibers can be transported into the rotor. Thus, more fibers can be fed and the rotor speed can be increased to spin the yarns without increasing the negative pressure inside the rotor. Additionally, if vortices can be reduced to a minimum, the yarn quality will also be improved. Based on the above analysis, rounding the channel inlet left corner is meaningful. However, the corner radius should be appropriately determined according to the channel dimensions. So far from the nine cases studied, the optimum one corresponds to an inlet width of 15.5 mm and a radius of 1.5 mm.

Figure 17. Vortices generated at the channel inlet as a function of inlet width and inlet corner radius.

Figure 18. Mass flow for different inlet corner radii in different transfer channel inlet widths.

Effect of bypass

Another method that may contribute to the decrease of the vortex strength and size is to add a bypass near the transfer channel inlet. In this section, three different bypass inlet velocities were set at 2, 5 and 8 m/s. A case in which the bypass inlet was connected to atmospheric pressure was also considered. This particular case is relevant since it involves no extra energy usage.

Figure 19 shows the streamlines of the transfer channel for three different bypass inlet velocities and a case with bypass inlet atmospheric pressure. With the introduction of the bypass, in all cases the main vortex appearing on the left-hand side channel inlet is reduced compared with the original case (Case2), see Figures 6 and 19. For the cases where the initial bypass inlet velocity is ≥ 2 m/s, it is seen that when increasing the initial velocity, the main vortex on the channel inlet left-hand side is gradually reduced. Notice as well that the vortex in the transfer channel inlet center does not exist in any of these three cases. However, a new positive vortex appears on

the inlet right side for a bypass inlet velocity of 8 m/s. This may be due to the following reason. As air flows into the bypass at a low speed, it joins the main stream from the roller surrounding housing to form a stronger air stream flowing into the transfer channel. If the bypass inlet air velocity is high enough, it will be able to shift the mainstream towards the roller, generating a downstream low-pressure field, and therefore a positive vortex.

When atmospheric pressure boundary is set at the bypass inlet, the main vortex is smaller and the main stream flow is smoother, having a higher velocity magnitude than what was obtained in Case D (bypass inlet velocity = 2 m/s). As a result, the overall mass flow shown in Figure 20 is higher than that obtained in the other cases. Furthermore, the adoption of a bypass is surely improving the overall mass flow.

Figure 19. Effect of bypass inlet velocity and atmospheric pressure on the transfer channel inlet vortices.

Figure 20. Transfer channel outlet mass flow as a function of bypass inlet condition. Q_{m1} is the overall mass flow directed to the rotor from the transfer channel outlet, and Q_{m2} is the mass flow without considering that from the bypass. ΔQ_m is the mass flow contributed by the bypass.

Considering the impact of the bypass on the mass flow distribution, the results are promising. The overall mass flow directed to the rotor Q_{m1} is composed of two parts, one is from the bypass ΔQ_m and the other is contributed by the trash extraction chute

Q_{m2} . As observed from Figure 20, when injecting air into the bypass at a constant speed, Q_{m2} reduces as the bypass inlet speed increases. This demonstrates that an increase in ΔQ_m would lead to a decrease in Q_{m2} . A much noticeable difference is seen in the case where pressure inlet boundary is used. Since the bypass inlet and the trash extraction chute inlet are both set as atmospheric pressure, 101000 Pa, the shorter the distance from the inlet to the outlet, the larger the mass flow. As some fibers from the feeding roller (not presented in the current study) are beyond holding by the opening roller, these fibers will be transported by the air. Thus, the decrease of Q_{m2} will reduce the amount of these fibers. Moreover, the reduction of Q_{m2} indicates a decrease in air velocity in the trash extraction chute, which will decrease the risk of trash flowing back to the opening roller chamber. Nevertheless, the contribution of ΔQ_m will increase the mass flow in the fiber separation region, and consequently, the velocity of fibers being transported through the transfer channel is expected to increase. Under this conditions, crimped and hooked fibers will undergo more effective straightening through the accelerating air in the transfer channel.

Rounding the inlet left-hand side corner and using the bypass.

To take advantages of both, rounding the transfer channel inlet corner and using a bypass, a new transfer channel design combining a rounded corner and the bypass is presented in this section, see Figure 2b). As in the previous section, three different bypass inlet velocities of 2, 5 and 8 m/s are evaluated, bypass inlet atmospheric pressure as a boundary condition is also considered.

From Figure 21, it can be clearly seen that rounding the channel inlet corner and using a bypass can improve the velocity streamlines in the transfer channel. Compared to the cases without a rounded corner, the main vortex located at the transfer channel inlet left-hand side disappears. When the bypass inlet velocity exceeds 5 m/s, the vortex on the channel inlet right-hand side appears. Once the bypass inlet velocity reduces to 2 m/s, all vortices are eliminated, and the whole streamlines are rather smooth. When the pressure inlet boundary condition is used, the streamlines are also favorable, providing a desirable surrounding for fibers.

Figure 21. Streamlines modification when using inlet corner rounding and bypass.

Figure 22. Transfer channel outlet mass flow as a function of bypass inlet condition.

The mass flow distribution is shown in Figure 22, and the parameters involved are the same as the ones already presented in Figure 20. As the bypass inlet velocity increases from 2 to 8 m/s, the overall mass flow directed to the rotor shows a slight increase, whereas Q_{m2} is seen to decrease significantly. It is observed that when using pressure boundary at the bypass inlet, maximum mass flow is obtained, while Q_{m2} , mass flow from extraction chute, is reduced to the minimum value. Under the authors' point of view, this is the optimum case for the rotor spinning.

In Table 4, the geometries of the optimum transfer channel in the present study are compared with the ones of some commercial models of Schlafhorst and Rieter. The dimensions of the transfer channel length, inlet width and inlet height vary in these models. The basic structural design criterion of the transfer channel is that it is tapered to enable effective fiber drafting. Nevertheless, the modifications applied on the transfer channel in this study could be extended to other models.

Table 4. Comparison of different transfer channel geometries and dimensions.

Type	Schlafhorst		Rieter	Proposed model
	BD7	Coro box SE-12	R36	
Channel Length L (mm)	50	98	35	40
Inlet width W (mm)	15	20	14	18.5
Inlet height H (mm)	11	10	5	11.5
Overall Shape	Tapered			
Inlet shape	Approximately rectangle			
Outlet shape	Approximately oval			
Corner Radius (mm)	-	-	-	1.5
Bypass	-	-	-	YES

Experimental test rig

To further validate the results presented, simulation results are to be compared with the experimental ones. Figure 23 shows the test rig system employed to perform the measurements. The test rig consists of a suction pump, a control valve, which function is regulating the rotor outlet relative pressure, and two digital manometers. One of the manometers is used to measure the rotor outlet pressure, and the other one is connected to a Pitot tube, which will be employed to measure the pressure at the rotor internal chamber, see Figure 23. The Pitot tube was put inside the rotor from the yarn

guiding mouth to measure the pressure at different positions. The test points were set in two directions, along the rotor central vertical axis, and around the rotor circle. Nine different vertical positions measured from the rotor center bottom to a vertical distance of 8 mm were evaluated. Each position had a spacing of 1 millimeter from its neighbors. Eight equidistant measurement points on a circle of 10 mm in the x - z plane at $y = 3$ mm were selected, as presented in Figure 23 b).

The experiments were undertaken under the following conditions that both the rotor and opening roller were kept static. It is prudent to mention that our experimental validation was only limited to measurement of pressure under static opening roller and rotor. There are discrepancies associated with this scenario because under industrial scale, the opening roller and the rotor revolve at high speed. This adjustment was implemented in both simulation and experimental procedure to improve precision of CFD validation. The tests were performed at two different rotor outlet relative pressures, -6000 and -7000 Pa. Simulations were carried out with the same boundary conditions to match the experiments.

Figure 23. a) Test rig scheme and rotor internal side view; b) different positions inside the rotor where pressure measurements were performed. Measurements were taken on a circle of 10 mm in the x - z plane at $y = 3$ mm, and also along the central y -axis.

When considering the measurements undertaken at different angular positions, it was initially seen that the experimental results gave pressures inside the rotor slightly higher than the simulated ones. In reality, the simulated results presented pressures inside the rotor which were almost the same as the pressures outside the rotor. This fact directed the attention of the authors to the rotor outlet and the opening roller pins. In fact, the rotor outlet of 1 mm used in the previous sections was an estimation when considering the dimensions of a brand new machine. In reality, the rotor spinning system utilized for the experiments was a very old one. Based on the CFD results, the authors realized that the rotor outlet dimension was, for this particular machine, much smaller than the theoretical one. This is why in Figure 24 the experimental results were compared with the CFD ones for a rotor outlet of 0.4 mm and an overall wall roughness of 0.05 mm. Both conditions were matching the reality.

Regarding the measurements undertaken along the rotor central vertical axis, Figure 24a), they have a perfect agreement with the simulated ones. The CFD and experimental results around the circle of 10 mm in the x - z plane, show quite a large deviation (see Figure 24b) when compared to the results measured along the vertical axis. In fact, the maximum pressure difference between CFD and experimental results

is obtained when outlet pressure is -7000 Pa, and it is of 436 Pa, which characterizes a deviation of 6.58%. A small pressure deviation versus the desired one in the manometers used could account for such difference between experimental and CFD results. Besides the limitations due to the measurement technique, the pressure loss generated at the roller could also contribute to such deviation.

To better understand the pressure-flow characteristics of the channel between the trash extraction chute and the rotor, and also between the rotor and the outlet, the characteristic losses of these two channels are obtained based on CFD results. Such constants are presented in Table 5 and obey to the generic equation $\Delta P = kQ^2$ (pressure P is measured in Pa and volumetric flow Q is given in m³/s).

Comparing the trash extraction chute-rotor channel constant k_1 with the rotor-outlet constant, defined as k_2 in Table 5, it is noticed that the first one is much bigger than the second one. In reality, due to the roller pins, k_1 is expected to be even bigger, which for a given flow will further reduce the pressure inside the rotor, reducing as well the small gap between experimental and CFD results.

Figure 24. Comparison of the simulation pressures with the experimental results: a) the vertical positions and b) positions on a circle of 10 mm in the x-z plane at $y = 3$ mm.

E-Experiment, and S-Simulation.

Table 5. Friction and contraction constants of the trash extraction chute-rotor channel and the rotor-outlet by CFD.

Rotor outlet pressure P_{outlet} (Pa)	-6000	-7000
Trash extraction chute pressure P_{inlet} (Pa)	0	0
Pressure inside the rotor P_{rotor} (Pa)	-5675.04	-6624.32
Pressure difference $P_{\text{rotor}} - P_{\text{outlet}}$ (Pa)	324.96	375.68
Volumetric flow (m ³ /s) $\times 10^{-4}$	5.5193	5.9666
Friction and contraction (Constant) k_1 Pa/(m ³ /s) ² $\times 10^{10}$	1.8630	1.8608
Friction and contraction (Constant) k_2 Pa/(m ³ /s) ² $\times 10^9$	1.0668	1.0553

Spinning experiments

In this section, the conventional JWF1612-2157A rotor spinning unit was modified such that the transfer channel inlet corner was rounded using a globet hand file tool and a bypass channel was utilized. Figure 25 shows a schematic diagram of the conventional transfer channel and the modified one. Yarns were produced using the conventional and modified rotor spinning units at three different rotor speed conditions. The specifications of the raw material used and spinning conditions are

shown in Table 6.

Figure 25. Schematic diagram of the original and modified transfer channel.

Table 6. Specifications of the raw material and spinning conditions.

Material	Cotton
Fiber length	29 mm
Yarn count	58 tex
<i>Speeds of the rotor spinning unit</i>	
Opening roller (rpm)	6000
Rotor (rpm)	38500, 48600, 60400
Yarn delivery (m/min)	59.16, 75.71, 92.86

Physical properties (tenacity, elongation, hairiness, evenness and imperfections) of each yarn produced were tested for ten samples to get the average tested properties. Yarn strength was tested on the YG061 yarn strength tester at a clamp speed of 500 mm/min and gauge length of 500 mm. Yarn evenness and imperfections were measured on a Changling CT3000 yarn evenness tester at a speed of 100 m/min. Yarn hairiness was tested on the YG172A yarn hairiness test device at a speed of 30 m/min. All the tests were conducted at a temperature of $20\pm 2^{\circ}\text{C}$ and humidity of $65\pm 4\%$.

As shown in Figure 26, the tensile properties of the rotor spun yarns made by the two spinning systems are decreased with the increase of the rotor speed. Tenacities of the yarn made by the modified rotor spinning unit are 12.929 cN/Tex, 12.748 cN/Tex and 12.631 cN/Tex respectively, increasing by a margin of 4.88 %, 5.09 % and 7.28 % with respect to that of the conventionally rotor spun yarns. Table 7 presents the comparison of the evenness and hairiness of the spun yarns. With the increase in rotor speed, the CVm (%), thin places (-50%), thick places (+50%) and neps (+280%) of the modified spun yarns are fewer than that of the original yarns. While the yarn hairiness between the two has little difference under the different rotor speed conditions.

Figure 26. Comparison of the yarn tensile properties, a) tenacity (cN/Tex) and b) elongation (%).

Table 7. Yarn evenness and hairiness in different rotor spinning units.

Yarn type	38500 rpm		48600 rpm		60400 rpm	
	Co	Mo	Co	Mo	Co	Mo
CVm (%)	12.58	12.85	13.3	12.02	12.5	12.19
-50% thin (number·km ⁻¹)	14	36	36	20	24	12
+50% thick (number·km ⁻¹)	14	12	14	12	20	10
+280% neps (number·km ⁻¹)	4	4	4	2	10	2
Hairiness ≥ 3mm (number·m ⁻¹)	10.5	4.56	6.6	9.4	15.8	16.7

Co: conventional rotor spinning unit; Mo: modified rotor spinning unit.

According to the above analysis, it was concluded that tenacity, yarn evenness and imperfection of the yarn produced by the modified rotor spinning unit improved. Based on the tests performed, it can be stated that the transfer channel modification has a promising future in the yarn producing industry.

Conclusions

In this paper, three-dimensional CFD models of the rotor spinning unit have been developed to study the effects of geometric and spinning parameters on the transfer channel airflow characteristics. New transfer channel designs were proposed to reduce the vortices generated in the transfer channel to improve fiber configuration and hence yarn properties.

According to the above analysis, vortices produced in the transfer channel inlet are real obstacles for fiber-straightening. A transfer channel length between 40 and 52 mm is recommended, as minimum vortices and maximum mass flow are obtained. Giving consideration of both yarn properties and system productivity, a roller speed of 6000 rpm and a diameter around 70 mm would be desirable choices. Decreasing the rotor outlet relative pressure will benefit yarn production, however, it also increases the vortex intensity on the transfer channel inlet left-hand side, leading to a decrease in fiber straightening and therefore yarn properties.

It is possible to dissipate the vortices located at the transfer channel inlet via rounding left-hand side corner, using a bypass or combining both modifications. By rounding the channel inlet corner, the main vortex disappears whereas a low-intensity vortex appears at the channel inlet center. With the introduction of the bypass, the strength and area of the vortex can decrease rapidly if the bypass inlet is properly set.

A combination design of using the rounded corner and bypass can take advantages from both concepts. A conventional rotor spinning unit was modified in which a rounded corner and a bypass channel were utilized. The yarn properties such as tenacity, CVM%, thin places, thick places and neps of the modified yarn improved compared to that of the conventional yarn. The proposed transfer channel modifications can improve rotor spun yarn quality.

Acknowledgements

This work was supported by the Key Grant Project of Ministry of Education of the People's Republic of China (Grant 113027A).

References

1. J. Kwasniak. An Investigation of a New Method to Produce Fancy Yarns by Rotor Spinning. *J Textile Inst* 1996; 87: 321-334.
2. Y. I. Matsumoto, S. Fushimi, H. Saito, et al. Twisting Mechanisms of Open-End Rotor Spun Hybrid Yarns. *Textile Res J* 2002; 72: 735-740.
3. K. B. Cheng and R. Murray. Effects of Spinning Conditions on Structure and Properties of Open-End Cover-Spun Yarns. *Textile Res J* 2000; 70: 690-695.
4. K. R. Salhotra and P. Balasubramanian. Influence of Carding Technique on Yarn Tenacity Response to Increasing Rotor Speed. *Textile Res J* 1985; 55: 381-382.
5. L. X. Kong, R. A. Platfoot, and X. Wang. Effects of Fiber Opening on the Uniformity of Rotor Spun Yarns. *Textile Res J* 1996; 66: 30-36.
6. A. Peyravi, S. Eskandarnejad, and M. B. Moghadam. Dual-feed rotor spinning of cotton fiber: trash separation and yarn properties. *J Textile Inst* 2013; 105:377-382.
7. C. F. J. Kuo, W. L. Lan, and Feriyonika. A study of processing parameters in open-end spinning process for organic cotton (Ne 24). *Textile Res J* 2012; 82: 1560-1568.
8. C. F. J. Kuo, C.-Y. Kao, and H.-J. Wei. Optimization of Open-end Rotor Spinning Frame Parameter and Estimation of Relevant Quality Characteristics. *Polym-plast Technol* 2011; 50: 923-930.
9. C. A. Lawrence and K. Z. Chen. A Study of the Fibre-transfer-channel Design in Rotor-spinning. Part II: Optimization of the Transfer-channel Design. *J Textile Inst* 1988; 79: 393-408.
10. C. A. Lawrence and K. Z. Chen. A Study of the Fibre-transfer-channel Design in Rotor-spinning. Part I: The Fibre Trajectory. *J Textile Inst* 1988; 79: 367-392.
11. S. Eskandarnejad. A study of fiber behavior in the transfer zone of an open-end spinning system. PhD Thesis, University of Manchester, UK, 1991.
12. L. H. Zhang and B. X. Zhang. A Study of Fiber Transfer Channel in Rotor Spinning. *J. China Textile Univ.* 1991; 17:16-26.
13. S. Seyed, S. Eskandarnejad, and A. Emamzadeh. Effect of geometry of end of the fibre transport channel with slotted exit on rotor spun yarn quality. *J Textile Inst* 2015; 106: 564-570.
14. L. X. Kong and R. A. Platfoot. Two-Dimensional Simulation of Air Flow in the Transfer Channel of Open-End Rotor Spinning Machines. *Textile Res J* 1996; 66: 641-650.
15. L. X. Kong and R. A. Platfoot. Fibre transportation in confined channel with recirculations. *Comput Struct* 2000; 78: 237-245.

16. A. C. Smith and J. W. W. Roberts, Straightening of Crimped and Hooked Fibers in Converging Transport Ducts: Computational Modeling. *Textile Res J* 1994; 64: 335-344.
17. H. Lin, Y. Zeng, and J. Wang. Computational simulation of air flow in the rotor spinning unit. *Textile Res J* 2015; 86: 115-126.
18. P. N. Marino, J. Carpintero, A. M. Manich, et al. The Influence of the under-Pressure in the Rotor on the Properties of Open-End-Spun Cotton Yarns at Different Values of the Rotor Speed and Opening-Roller Speed. *J Textile Inst* 1985; 76: 86-102
19. W. Bauer, H. Muller, and S. Tabibi. Fiber Flow and its Effects on the Characteristics of OE Spun Yarns. *Textil Praxis Int.* 1989; 44: 15-17.