**A Novel Compact Spinning with Lattice Apron Achieved through Airflow Simulations**

**SATY Malik Yonis Hassan**, **AKANKWASA Nicholus Tayari**, **WANG Jun**

**College of Textiles, Donghua University, Shanghai 201620, China**
**Key Laboratory of Textile Science & Technology of Ministry of Education, College of Textiles, Donghua University, Shanghai 201620, China**
**College of Engineering Technology of Industries, Sudan University of Science and Technology, Khartoum, Sudan**

Received 10 October 2020; accepted for publication 1 April 2021

**Abstract**

Compact spinning with a lattice apron is commonly used in commercial yarn spinning despite the high energy consumption due to high negative pressure generated during operation. In the current study, a numerical and experimental approach is used to study the impact of airflow behavior on yarn properties. Three-dimensional (3-D) printed guiding devices were used to control the airflow mechanism and negative pressure. Airflow analysis showed a significant decrease in the negative pressure and the rise in the total airflow velocity and streamlines distribution. This phenomenon eliminated the spinning triangle thus enhancing the spun yarn qualities and energy efficiency. The novel design is thus commended for further improvement and commercialization.

**Key Words**: Compact spinning, Numerical simulation, Airflow, Energy consumption

---

1. **Introduction**

In compact yarn production, the most widely used technique nowadays is pneumatic compact spinning [1-2]. This technique is implemented using negative pressure airflow to condense the fiber bundle for eliminating the spinning triangle and improve the yarn qualities [1-3]. Compact spinning with lattice apron is the most widely used pneumatic compact spinning system at present, mainly including three-line compact spinning (TLCS) and four-line compact spinning [4-8]. Nevertheless, the compact spinning systems with lattice apron is considered expensive due to the high energy consumption during the spinning process [1].

In this study, we employed numerical simulation techniques to study the airflow field based on computational fluid dynamics (CFD) to optimize different guiding devices to reduce the negative pressure in the compact spinning system with a lattice apron. In the next sections, 3-D physical models for the condensing zone, boundary conditions, simulation results and discussion and conclusions are presented.

2. **Three-dimensional physical models for the condensing zone**

The physical dimensions of the compact spinning machine were measured directly and the virtual 3-D model of the condensing zone was generated with CAD software. Fig. 1 (a) and (b) illustrate the 3-D model and the guiding device respectively. In Table 1, the geometrical parameters of the condensing zone are given.

| Table 1 | The geometrical parameters of the condensing zone. |
|---------|----------------------------------------------------|
| Front top roller diameter (mm) | Output top roller diameter (mm) | Length of suction slot (mm) | Width of suction slot (mm) |
| 28 | 40 | 20 | 1.5 |

* Corresponding author: E-mail : junwang@dhu.edu.cn, Tel : +86 (21) 6237 9160
parameters of the condensing zone are presented in details. In these models, the output direction of the fiber strand is defined as a negative direction of the X-axis. In contrast, the transverse condensing direction of the fiber strand is defined as Y-axis while the thickness direction of the fiber strand is defined as Z-axis. The mid-point of the lower end of the suction slot is taken as the origin as shown in Fig. 1 (a). The fiber strand volume is smaller than the condensing zone and hence will be ignored in this model.

2.1 Boundary conditions

The flow field in the condensing zone is assumed to be incompressible and expressed by Reynolds equations. The turbulence model adopts $k - \varepsilon$ two-equation model and other equations have been expressed in our previous works [8]. In this model, the pressure outlet was assumed as the atmospheric pressure (101325 Pa). In actual design, the collecting pipe is linked with the centrifugal fan, and the air inside the collecting pipe is sucked from one side of the pipe. Therefore, the face of the pipe is set as a pressure outlet boundary with a value of -2000 Pa (without a guiding device) and -1000 Pa (with a guiding device). The static pressure determined by the centrifugal fan is pumped down and the other faces of the model are considered solid walls. The condition of the non-slip boundary is observed on the solid wall, that is, the velocity of the wall is zero. The model set up in Fluent was set with single-precision implicit split operator solver, while the coupling problem between pressure and velocity was resolved with a SIMPLEC Algorithm with convergence precision $1 \times 10^{-4}$.

2.2 Meshing

In this paper, the dedicated grid generation software ICEM CFD of ANSYS 14.5 was used to mesh the calculation area of the airflow field in the condensing zone. By applying a program-controlled Triangular surface mesh. Unstructured tetrahedral cells were utilized in the computational field as well as in the application of refinement meshing on grids in the area where the collection slot is located that were denser than the other areas (see Fig. 2 below).

2.3 Porous jump boundary condition

To simulate the effect of the lattice apron on the airflow characteristics, the plane covered by the grid circle is set to a porous jump boundary (Fig. 1 (a)). The pressure changes ($\Delta p$) above and below the plane can be defined by Darcy’s law and an inertia loss term:

$$\Delta p = -\left(\frac{\mu}{\alpha} + C_2 \frac{1}{2} \rho v^2\right) \Delta m$$

In the above formula, $\mu$ is the laminar viscosity; $\alpha$ is the permeability; $C_2$ is the pressure jump coefficient; $v$ is the normal velocity, and $\Delta m$ is the plane thickness. The effect of grid circles of different thicknesses and porosity on the airflow can be simulated by setting the permeability and the thickness of the porous plane. According to the actual situation of the grid circle, the permeability and the thickness of the porous plane are set at $1 \times 10^{-7}$ and 0.09 mm, respectively. The details of the equations of airflow modelling in ANSYS were used as expressed in literature [9].
3. Results and discussion of numerical simulations

3.1 Validation of the numerical code and grid independence

The mesh independence test is done by using three different grids with various elements and nodes. Grid 1 (823956 elements); Grid 2 (496761 elements); and Grid 3 (341480 elements). Grid 2 was chosen for further simulation. A grid independence test is performed to ensure that the simulation results from accuracy are not influenced by the mesh fineness (see Fig. 3).

3.2 Simulations of the flow field

The guiding device used in the simulation is shown in Fig. 1 (b), which is obtained from our previous work [8]. By using fluent software, the results of numerical simulation for a 3D flow field without a guiding device and with the guiding device in the condensing zone of the compact spinning method with a lattice apron are presented in Fig. 4. It is confirmed that the flow velocity of airflow is the highest on the suction slot and at the front and the back of the condensing zone. Velocity gradually decreases in other areas as demonstrated by simulation results. The airflow velocity streamlines distributions are denser and the active area of the negative pressure increased with the surging negative pressure (see Fig. 4 (a) and (b)). This phenomenon increases fiber condensing thus improving their alignment along the fiber axis and mechanical properties. When the guiding device was used, the active area of negative pressure was more than that without a guiding device because the guiding device prevents the decrease in negative pressure with respect to atmospheric pressure. Therefore, using the guiding device allows the airflow inwards from two holes on top of the guiding device which is more likely beneficial for fiber condensing directly. Fig. 4 (c) and (d) show the streamline diagram of the flow velocity in the y–z section with the velocity vector colored according to the velocity magnitude in m/s. When the guiding devices were used, the air velocity increased more than the measured air-flow in the cases without a guiding device, a phenomenon that can be attributed to the dispersion of the flow field distribution. As a result, the air velocity increases more with a guiding device than when the device is not used.

4. Conclusions

In our study, we have presented a new technique to optimize the high energy consumption in compact spinning with a lattice apron. The optimization process reducing the negative pressure by using the airflow guiding device. This concept will be further investigated for commercialization in our next works.

References

[1] Altas S, Kadoğlu H (2012) Journal of Engineered Fabrics & Fibers, 7, 87-100. https://doi.org/10.1177/155892501200700110
[2] Zhuan YZ, Yun DZ, Zhi HH, Yan W, Long DC (2009) Textile Research Journal, 80, 712-719. https://doi.org/10.1177/0040517509342314
[3] Dou H, Liu S (2011) The Journal of the Textile Institute, 102, 713-718. https://doi.org/10.1080/00405000.2010.515728
[4] Xuzhong S, Weidong G, Xinjin L (2016) The Journal of The Textile Institute, 107, 811-824. https://doi.org/10.1080/00405000.2015.1061786
[5] Xuzhong S, Xinjin L, Xiaoyan L (2018) International Journal of Clothing Science and Technology, 30, 363-379. https://doi.
[6] Xinjin L, Wenlong L, Hong Z, Xuzhong S (2015) The Journal of The Textile Institute, 106, 431-442. https://doi.org/10.1080/00405000.2014.925198

[7] Ting F, Jianping Y, Guangwei C, Nanlian C, Yiping Q (2017) Textile Research Journal, 88, 2499-2505. https://doi.org/10.1177/0040517517723029

[8] Saty MY, Akankwasa NT, Wang J (2021) Textile Research Journal, 91, 1389-1398. https://doi.org/10.1177/0040517520982586

[9] Akankwasa NT, Lin HT, Zhang Y, Wang J (2016) Textile Research Journal, 88, 237-253. https://doi.org/10.1177/0040517516677230