Three-dimensional Simulation of Turbulent Rectangular Jet Spreading Technology of Carbon Fiber Bundle

Jinliang Chen¹, Xuejuan Niu*,¹,² and Zhun Zhang¹
¹Mechanical Engineering Institute, Tiangong University, 300387 Tianjin, China
²Advanced Mechatronics Equipment Technology Tianjin Area Major Laboratory, Tiangong University, Tianjin 300387, China
Email: niuxuejuan@tjpu.edu.cn

Abstract. In order to smoothly spread large-sized carbon-fiber bundle, hot-jet blower is added to a turbulent spreading platform to conduct pre-spreading process. To achieve the best spreading performance, CFD model of the blower are simulated based on the Reynolds average Naiver-Stokes equation. The influence of the relevant geometric parameters and process parameters on the internal flow field are analyzed. The distributions of streamline, surface temperature and air pressure under different working condition are characterized and compared parametrically. The effective area of the nozzle directly contacts with fiber wall surface. Simulating model has been built and crossover experiments have been carried out on the spreading platform to investigate and optimize the influence of jet distance, nozzle air pressure and hot-jet temperature on the spreading performance of fiber bundle. According to the experimental data and simulating results, relevant geometric parameters and process parameters of the nozzle are optimized.

1. Introduction
As a kind of high performance conforming material with low density, high specific strength, high specific modulus, good heat resistance and chemical stability, carbon fibers are widely used in aerospace, automobile manufacturing, building reinforcement and other fields [1-3]. In order to meet the needs of different working conditions such as fiber fabrics, thin layer treatment of carbon fibers is very important. At present, there are many ways to thin carbon fibers, according to the relevant technical description given by Irfan [4].These methods can be divided into three types, one is the active method, the other is the passive method, and the third one is a mixture of the two. For the active methods, energy is used to spread the fiber bundle, such as fiber bundle through small gaps which depending on airflow (pressure or suction) or under (ultra-)sonic waves and vibration effects [5, 6]. The passive approach is applying tension and constant motion on different geometries to spreading the fiber bundle, such as using fiber rods, convex structures or geometric guiding elements to make the fiber bundle become the fiber tow [7-9]. Compared with other methods, jet spreading technology uses hot air as the working fluid. And hot air flow plays the following two important roles. On the one hand the perpendicular airflow make the force on the fiber bundle more concentrated. On the other hand impinging jet can produce high heat transfer rate around an impinging position on an impinging wall which can make the binder on the surface of fiber bundles can be softened sufficiently. Therefore, In order to make ensure the carbon fiber spreading smoothly and evenly in the spreader, it is necessary to pre-spreading.
   Baucom [5] first used the airflow and completed the broadening of the fiber bundle with the help of air jet technology. The spread fiber bundle, here we call fiber tow, can be better fuse carbon fiber and
resin powder. Chen [6] used the planar airflow fiber-spreading method to simulate the internal flow field of the spreader based on the Reynolds average Naiver-Stokes equation, and obtained the flow field characteristics inside the pneumatic spreader. It is concluded that the greater the air velocity, the more favorable the expansion of the fiber bundle. Sihn [7] used air flow negative pressure method, which fiber bundle was passed through the fiber guide equipped with a duct and a vacuum pump. The vacuum pump sucked air downward through the duct, and the fiber bundle can be easily diffused without damaging. El-Dessouky [8] used secondary airflow fiber-spreading technology. The fiber bundle was firstly spreading through a Type 1 fiber guide obtain the pre-spreading tow, and then passed through a Type 2 spreader to obtain the final fiber tow. By reducing the conventional 12k carbon fiber bundle by increasing the tow by 5 mm to 25 mm, the weight per unit area is reduced by about 500%. Ren [9] used mechanical-pneumatic spreading method, combing the advantages of the pneumatic spreading and the traditional mechanical device by machining air groove on the spreading pin. It showed that when the airflow pressure is 0.2 MPa, the prepreg produced by carbon fiber after fiber spreading has good performance. Huang [10] spread the fiber bundle by multi-stage airflow spreading system, and simulated the internal flow field characteristics of the spreader by finite element method (FEM). Structural parameters of the facilitate fiber spreading are obtained, and the width of the tow can be stretched more than three times. However, most researchers analyzed the velocity of the air stream and the effect of the structure of the spreader on the fiber spreading, and most of them are nearly neglected the influence of the temperature of the air flow on the spreading of the fiber bundle.

In this work, a new pneumatic spreading, hot-jet spreading technology, are designed as a pre-spreading system which is very essential for large-sized fiber bundles to ensure the fully spread of carbon fiber bundles. By establishing the three-dimensional CFD model of hot-jet blower, the flow field in it is simulated and characterized. The influence of the relevant geometric parameters and process parameters, such as nozzle-to-fiber distance, nozzle pressure and temperature, on spreading performance of carbon fiber bundle have been studied. Based on the simulation data, hot-jet pre-spreading experiments have been carried out. Through comprehensive analysis, the optimum parameters for pre-spreading fibers are obtained.

2. Experimental Method

2.1. Materials

The raw materials used were carbon fiber (CF) bundle (purchased from Toray), and the size of the bundle is 24k which means the fiber bundle have 24000 filaments. The properties of carbon fiber (CF) tows as received is shown in Table 1.

| Fiber type    | No. of CF filaments/tow | Tensile strength (GPa) | Tensile modulus (GPa) | Elongation (%) | Diameter of CF (μm) | Density (g/cm³) | Size type | Size content (%) |
|---------------|-------------------------|------------------------|-----------------------|----------------|---------------------|----------------|-----------|------------------|
| CT24-5.0/270-E100 | 24,000                  | 5.0                    | 270                   | 1.90           | 6.9                 | 1.79           | epoxy     | 1.0              |

2.2. Experimental Setup

The schematic experimental platform designed is shown in Fig.1. The jet pre-spreading is mainly consists of four parts, namely a sizing loosening system, a tension and speed controlling system, a fiber guiding system and a winding system. The components and operation parameters and functions are summarized in Table 2. The structure of the experimental platform is constructed of aluminum profiles. At the beginning of the experiment, a constant tension fiber is used, and two pairs of press rolls ensured that the tension of the fiber tow is constant. The tension sensor (JZHL-L1-5N) is used to determine the tension of the fiber tow between the two pairs of rolls. The hot air blower (FH6273) has the advantages of high power and convenient operation. The height of the hot air blower is adjustable to accommodate different fiber distances. The distance is adjusted according to the characteristics of the jet flow field.
When the tension sensor reaches the expected value, the two pairs of rollers are pressed to make sure that the tension of the fiber bundle will not change. The hot air blower is turned on and the airflow is directed from the inlet of the fan to the fiber bundle. After the airflow encounters the obstruction of the fiber bundle, forming a certain pressure on the surface of the fiber and fiber bundle will be in a tension-free state. With the help of pressure, the fiber bundle is spreading to be the fiber tow.

### 2.3. **The Principle of Jet Flow Fiber Spreading**

To better understand the principle of jet spreading, Fig. 2 shows the distribution of fibers under jet airflow. Under the action of the air pressure, the fiber bundle moves rapidly to both sides, and the fiber bundle is easily widened and thinned. The initial state of the carbon fiber tow is shown in Fig. 2(a). Due to the blocking action of the carbon fiber tow, the air flow velocity $v_1$ should be less than the air flow velocity $v_2$ on both sides of the fiber bundle, therefore the pressure of $P_1$ is higher than the pressure of $P_2$. As shown in Fig. 2(b), under the action of pressure and the hot air flow, the cohesion of the binder between fibers is softened, and the external fibers break away from the fiber bundle one by one, and the carbon fiber bundle is unfolded. The same principle apply to Fig. 2(b), due to the blocking of the fiber tow, $v_1 < v_2 < v_3$, so $P_1 > P_2 > P_3$. As the airflow time increases, the width of the fiber expands to a stable level. As shown in Fig. 2(c), the fiber distribution finally reached a steady state.
Figure 2. The principle of jet air flow fiber spreading

2.4. Collection of Experimental Data
For the convenience of experiments, the pressure drop and temperature of the fan are separately calibrated before the start of the experiment. During the experiment, the Mp115 differential pressure gauge is used to measure the pressure drop between the fan outlet and atmospheric pressure. Which range is $-500 \text{mbar}$ to $500 \text{mbar}$, and a resolution of $0.1 \text{mbar}$. Change the gears of different speeds, measure the pressure drop three times for each gear position, and finally use the average as the pressure drop of the gear position. The TES1310 handheld thermometer, with a range of $223.15 \text{K}$ to $1573.15 \text{K}$, is used to measure the inlet temperature of the fan. Use the same way, by changing the different temperature gears, the inlet temperature of the fan is measured three times at each gear position. As shown in Table 3, the average value is calculated and used as the final value.

Table 3. Relationship of hot air blower and temperature

| Temperature Gear position | Temperature(K) |
|--------------------------|----------------|
| Fist level               | 323            |
| Second level             | 343            |
| Third level              | 363            |

3. Numerical Study

3.1. Governing Equations
The continuity equation and the Naiver-Stokes equation used in establishing the jet flow field CFD model are as follows:

\[
\begin{align*}
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) &= 0 \\
\frac{\partial \mathbf{u}_i}{\partial t} + \frac{1}{\rho} \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \mathbf{u}_i \mathbf{u}_j \right) - \frac{2 \rho}{3} \left[ \mathbf{v} \left( \frac{\partial \mathbf{u}_i}{\partial x_j} + \frac{\partial \mathbf{u}_j}{\partial x_i} \right) \right] &= 0
\end{align*}
\]

Where: $\rho$ is the density; $t$ is the time; $\mathbf{u}_i (i = 1, 2, 3)$ is the velocity; $p$ is the pressure; $T$ is the temperature; $\mathbf{v}$ is the dynamic viscosity; $\mathbf{u}_i$, $\mathbf{u}_j$ is the velocity $\mathbf{u}_i$ and $\mathbf{u}_j$ of the pulsation value; $\mathbf{u}_i'\mathbf{u}_j'$ is the Reynolds stress.

\[
t_{ij} = -\rho \mathbf{u}_i \mathbf{u}_j = \mu_t \left( \frac{\partial \mathbf{u}_i}{\partial x_j} + \frac{\partial \mathbf{u}_j}{\partial x_i} \right) - \frac{2}{3} \left( \rho k + \mu_t \frac{\partial \mathbf{u}_i}{\partial x_j} \right) \delta_{ij}
\]

Where $k$ is kinetic energy, $k = \frac{u_i' u_j'}{2} = \frac{1}{2} \left( u'^2 + v'^2 + w'^2 \right)$, $\mu_t$ is the turbulent viscosity; $\delta_{ij}$ is the "Kronecker delta" symbol ($\delta_{ij} = 1$ when $i = j$; $\delta_{ij} = 0$ when $i \neq j$).

The internal air velocity of the jet flow field is less than $100 m/s$, which is an incompressible air, and the turbulence model adopts the Realizable $k$-$\varepsilon$ model. The transport equation of the turbulence model is as follows:
\[
\begin{aligned}
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_i} &= \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial k}{\partial x_j} \right) + \frac{\mu_l \sigma_i}{\sigma_f} \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M \\
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} &= \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{\mu_l \sigma_\varepsilon}{\sigma_f} \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 \varepsilon^2 - \rho C_2 \frac{\varepsilon^2}{k + \varepsilon} + C_\varepsilon \frac{\varepsilon}{k} C_{3\varepsilon} G_b 
\end{aligned}
\]  

(3)

Where: \( \mu_t = \rho \mu \frac{k^2}{\varepsilon}, C_{1\varepsilon}, C_{2\varepsilon}, C_\mu, C_2 \) are empirical constants; \( \sigma_k \) and \( \sigma_\varepsilon \) are the Prandtl numbers corresponding to the turbulent energy \( k \) and the dissipation rate \( \varepsilon \), respectively; \( G_k \) is caused by the average velocity gradient; The term of the kinetic energy \( k \), \( G_k = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} \)

According to the recommended values of Launder [14], the turbulence model parameters in Fluent 15.0 are shown in Table 4.

Table 4. Realizable k-\( \varepsilon \) turbulence model parameters in Fluent 15.0

| \( C_{1\varepsilon} \) | \( C_{2\varepsilon} \) | \( C_{3\varepsilon} \) | \( C_2 \) | \( \sigma_k \) | \( \sigma_\varepsilon \) |
|---|---|---|---|---|---|
| 1.44 | 1.9 | 0.09 | 1.9 | 1.0 | 1.2 |

There are three states of flow: laminar, turbulent and in the transitional phase. The flow state can be specified by the value of Reynolds number \( Re \) which is defined as the ratio of inertial and viscous forces. Generally the Reynolds number \( Re < 2100 \) is a laminar flow state, \( Re > 4000 \) is a turbulent state, and \( 2100 < Re < 4000 \) is a transition state. And Reynolds number can be defined as:

\[
Re = \frac{\bar{u} D}{\nu}
\]  

(4)

\[
D = \frac{4A}{C} = \frac{4ab}{2(a+b)} = \frac{2ab}{a+b}
\]  

(5)

Where: \( \bar{u} \) is average velocity; \( D \) is hydraulic diameter; \( \nu \) is kinematic viscosity; \( A \) is cross sectional area of the inlet; \( C \) is wetted perimeter of the jet; \( a \) is the length of rectangle; \( b \) is the width of rectangle.

3.2. Mathematical Model

The 3D model of the jet spreading device was carried out by the Cartesian orthogonal coordinate system. In order to facilitate the establishment of the CFD model of the jet flow field, the jet flow field is simplified and modeled. The whole model is simplified into a cuboid with a length of 400\( mm \), a width of 300\( mm \) and a height of 100\( mm \). According to the mouth shape of the blower, the jet nozzle is simplified to a rectangle of 60\( mm \) length and 18\( mm \) width.

The CFD model of the jet flow field is shown in Fig. 3. The \( x-axis \) is the direction which the fiber bundles are spreading. The \( y-axis \) is moving direction of the fiber bundle, and the \( z-axis \) is the direction in which the jet air is incident. The jet nozzle is the inlet in the jet flow field, and uses the pressure inlet as a boundary condition. The wall, \( W_i \), is the impacted wall by the jet air, and its boundary condition is a fixed and no-slip. According to the actual working condition, the periphery walls are connecting with the atmosphere. Therefore, it is most appropriate for them to set their BCs as Pressure outlet boundary conditions. The diameter of carbon fiber bundle is 6.9\( um \), the thickness of fiber bundle is 0.48\( mm \), and the original width is 7\( mm \). Compared with the flow filed of spreader, the size of carbon fiber bundle is too small and can be neglected. Therefore, the simulation is based on non-fiber simulation.
Since the simplified CFD model is a cuboid, it is divided into blocks by using multiple blocks technology in ICEM, and structured grid is used for meshing. To better observe the flow field distribution characteristics at the entrance and near the impact wall, mesh refinement technology is applied in these areas. The number of nodes and elements in the fluid domain are 288288 and 304271, respectively. Among them, there are 271825 with units type HEXA_8 and 32446 with units type QUAD_4. Convergence checks are made on the mesh to ensure that the number of meshes.

The SIMPLE algorithm was used to treat the pressure-based coupling. The convective terms and the diffusion terms of the governing equations were discretized with the upwind scheme both with second order accuracy. By solving with double precision, the convergence criteria for the continuity, momentum and the energy equations were defined to reach the scaled residues of $10^{-5}$, $10^{-4}$ and $10^{-6}$, respectively. All simulations were performed on a computer with ten 2.4GHz processors and 64GB of main system RAM.

### 3.3. Mesh Sensitivity and Model Validation

In order to check the mesh dependence, three cases with 189973, 304271 and 640387 mesh elements, respectively, were calculated for the CFD model. The air velocity along the nozzle axis and the temperature distribution perpendicular to radial direction at $z = 80\text{mm}$ are shown in Fig. 4(a) and (b), respectively. With the pressure perpendicular to radial direction at $z = 80\text{mm}$ shown in Fig. 4(c). As shown in Fig.4, there are little differences in the results with the increase of grid number. Therefore, we can conclude that the grid dependence is acceptable, and the mesh with 304271 elements is used in the following simulations.
In order to verify the correctness of the simulation model and reliable results, the velocity on the jet axis measured at different spray distances was compared with the simulation data. The experimental equipment in this study consists of a rectangular made of synthetic plastics with a hydraulic diameter of $28\, nm$. As shown in Fig. 5. It can be visually seen that the velocity on the central axis of the jet flow field is through a constant velocity zone, and then the velocity begins to decrease gradually. The velocity of this phase is in the free jet zone; when approaching the fiber wall surface, the velocity is drastically reduced, and finally the attenuation on the wall is zero. As we can see that the simulation results and the experimental data are match well which indicating that the simulation is reliable. However, the simulated results are slightly larger than the experimental data. This is due to the difference in temperature of the surrounding environment during the experiment test. While in the process of simulation, the ambient temperature is set to be constant and will not be affected by external factors.

![Figure 4](image1)

**Figure 4.** Grid dependence check ((a) air velocity along the nozzle axis; (b) air temperature along theradial direction at $z = 80$ m; (c) air pressure along the radial direction at $z = 80$ m)

![Figure 5](image2)

**Figure 5.** The velocity distribution of the jet central axis

![Figure 6](image3)

**Figure 6.** Lines selected for plotting results
3.4. Data Reduction
Numerical results are obtained after solving the governing equations. The results are presented in non-dimensional form. The non-dimensional pressure coefficient is drawn along lines A-A and B-B. Figure 6 shows the lines selected for plotting the results.

4. Results

4.1. Computational Results

4.1.1. Analysis of jet flow field characteristics
For the convenience of description, some parameters are defined. Such as the inlet temperature, $T$. The jet nozzle pressure, $P_l$. The distance, $D$, from the nozzle to the impact wall surface, the distance from the nozzle to the fiber bundle is called the fiber distance. The outlet pressure, $P_o$. And the pressure drop, $P_d$, is the difference between inlet and outlet pressure. Therefore, $P_d = P_l - P_o$. As shown in Fig.7, the jet flow field has a constant velocity flow area, a free jet area, a sluggish area and a near wall jet area. When the air flow from the nozzle is directed towards the wall surface, a constant velocity area and a free jet area are formed successively. Along with the airflow approaches the wall, the velocity of the airflow gradually decays to zero. In the meantime, the dynamic pressure is also drop down to zero, and a sluggish area is formed at the near wall surface. When the air flow is blocked by the wall, the airflow begins to spread to the surroundings, and the flow direction of the airflow changes, forming a near-wall jet region around the impact region.

![Figure 7. Characteristics of jet flow field](image)

![Figure 8. Velocity changes in the centerline of different sections](image)

4.1.2. Discussion on the fiber distance of the jet flow field
For the sake of appropriate fiber distance under the jet flow field, a simulation of the jet with a spray distance of $120mm$ is carried out. When the boundary condition is set as: $P_l = 101645Pa$, $P_o = 101325Pa$, $T = 323K$, the jet flow field is simulated and analyzed.

The flow field adopts fiber-free simulation. To find a suitable fiber-optic distance, it is assumed that the distance between the nozzles on the $z - axis$ and the fiber bundle on the $H$-plane is evenly distributed. By analyzing the velocity on the $y - axis$ centerline of the $H$-plane, the force of the fiber in the flow field can be easily judged. Therefore, the cross section, $H$, in the $z - axis$ direction of the jet flow field is selected as the fiber surface, the $x - axis$ direction is the direction of the fiber bundle spreading, and the $y - axis$ is the moving direction of fiber bundle. Select the fiber cross section in the positive direction of the $z - axis$. The distance from jet nozzle to the selected fiber cross section are $40mm$, $50mm$, $60mm$, $70mm$, $80mm$, $90mm$, $100mm$, $110mm$. The velocity at the centerline of the section ($y = 0$) is analyzed as shown in Fig.8. When the fiber cross section changes from $40mm$ to $80mm$, the velocity on the centerline has hardly changed. The velocity between $-50mm$ and $50mm$ is a constant value, which is benefit for fiber bundle spreading. When the fiber cross-section is changed from $80mm$ to $110mm$, the air flow velocity is V-shaped in the range of $-50mm$ to $50mm$. The intermediate air flow velocity is small, and the air
flow velocity on both sides is large. This phenomenon is not conducive to the uniform deployment of the fibers, which causes the fibers to alternately float and the spacing between the fibers becomes larger. Therefore, fiber spacing is most suitable for selection in the constant velocity area, that is to say a choice from 40 mm to 80mm is advantageous for the fiber bundle spreading uniformly.

4.1.3. Analysis of the influence of pressure drop on jet spreading

In order to study the influence of pressure drop on the jet flow field, the jet flow field under the fixed fiber distance is simulated and analyzed. As can be seen from Fig.8, the suitable fiber spacing is 40 – 80mm. In this section, the fiber wall distance of 80mm is selected as the impact wall surface of the jet flow field. The boundary conditions are set as: $P_l = 101645 Pa$, $P_o = 101325 Pa$, $T = 323 K$. The flow field of the jet spreading flow is simulated under different pressure drops.

Fig.9 shows the detail distribution of the air velocities in the A-A cross section and B-B cross section (shown in Fig.6). Compared Fig.9a and Fig.7, it can be seen that the air velocity under the inlet of the flow field is constant. This area is called the undisturbed jet core. As approaching the fiber surface at different pressure drops, air flow goes through the free jet area and the stagnation zone, and its velocity decreases gradually. In the Stagnation Zone, the velocity rapidly decreases to zero and a negative pressure is formed. With the increase of the pressure drop ($P_d$), the initial velocity of the nozzle airflow increases from 16 m/s to 25.6 m/s. There are obvious turbulence phenomenon in the near fiber area, which cause the vibration of the fibers and contribute to disperse of the fiber bundle greatly. Under the impact of the hot-air jet flow, the cohesive material in fiber bundle are soften and the fibers are dispersed rapidly toward the clapboards on each side.

From the velocity cloud chart of Fig.9, it can be seen that the velocity of the airflow is reduced to zero near the fiber wall surface, forming a negative pressure. The fiber bundles are forced to unfold under the action of pressure. In order to better describe the pressure on the fiber wall surface, the pressure distribution diagram of the fiber wall under different pressure drops are shown in Fig.10. The pressure distribution of the fiber wall surface is an elliptical shape, the pressure distribution ranges from −50mm to 50mm in the x direction and from −30mm to 30mm in the y − axis direction. The range of pressure in the $x − axis$ direction is significantly larger than the $y − axis$ direction, which is closely related to the shape of the jet nozzle. The $x − axis$ direction is the direction which the fiber bundle spreading, the direction of fiber movement is the $y − axis$. Therefore, the faster the pressure gradient changes along the x-axis, the more advantageous the fiber bundle broadening. From Fig. 10(a) to Fig. 10(d), as the pressure increases, the pressure distribution in the $x − axis$ direction does not change significantly. This may be due to the effect of fiber spacing and nozzle shape. Therefore, a reasonable pressure needs to be verified according to the experiment.
4.1.4. Analysis of the effect of temperature on jet spreading

In order to study the action mechanism of thermal jet impact on fiber spreading, the wall surface of different temperature airflow was analyzed. The fiber spacing is 80mm, the boundary condition are $P_t = 101645 Pa$, $P_0 = 101325 Pa$, $P_d = 320 Pa$. The set temperatures are 323K, 343K, and 363K, respectively. The results of the analysis are shown in Fig. 11.

Fig. 11(a) and Fig. 11(b) are temperature distribution diagrams of the fiber wall surface at temperatures of 323K and 343K, respectively, and the temperature diffusion on the fiber wall surface is relatively uniform, and the temperature gradient is 2K. The temperature of Fig.11(c) is 363K, the change of temperature gradient is 5K, and the speed of diffusion is relatively fast. It can be seen that the higher the nozzle temperature, the more obvious the exchange heat exchange with the surrounding gas. After encountering the gas with lower temperature, the temperature gradient drops faster. Fig. 11(d) is a graph showing the temperature diffusion in the positive direction of the x-axis of the fiber wall at different temperatures. Under the influence of the jet nozzle, the temperature change is relatively slow from 0mm to 50mm, and the temperature drops sharply from 50mm to 200mm.

4.2. Experiment Results

Combined with the results of the simulation, when the boundary conditions are set as, $D = 120mm$, $P_t = 101645 Pa$, $P_0 = 101325 Pa$, $P_d = 320 Pa$, the experimental results are shown in Fig.10. When the temperature is constant, the width of the fiber bundle broadening tends to increase first and then decrease as the fiber-to-nozzle distance increases. When the temperature is 323K, the width of the fiber broadening reaches a maximum at a fiber-to-nozzle distance of 50mm, and the maximum broadening width is 43mm. When the temperature is 343K and 363K, the width of the fiber broadening reaches a maximum at a fiber-to-nozzle distance of 80mm, and their maximum values are 54mm and 66mm, respectively. It can be seen that when the fiber is fixed from the nozzle, the width...
of the carbon fiber spreading increases remarkably as the temperature increases. When the fiber-to-nozzle changes from 80 mm to 100 mm, as the temperature and the fiber distance increases, the width of the fiber bundle unfolds or even decreases. This is because the fiber bundle is in the free jet region, which the jet velocity changes relatively, the fiber bundle easy to overlap.

5. Conclusion
Through the three-dimensional simulation analysis of the hot-jet flow field, the data which is favorable for jet spreading. And the spreading under different boundary conditions are obtained. The simulation results are verified by experiments, and the optimal boundary conditions for jet spreading are obtained.

1) Through the simulation analysis of the hot-jet flow field, it can be seen that there is a constant velocity area, a free jet area, a sluggish area and a wall jet area in the flow field.

2) When the spray distance \( D = 120 \text{mm} \), the optimal fiber distance is \( 40 \text{–} 80 \text{mm} \), and the constant velocity area of the jet in this area is beneficial to the homogenization of the fiber.

3) On the basis of the simulation, the hot-jet spreading test is carried out to obtain the pressure drop and temperature which are favorable for the jet spreading. When the temperature is \( 363 \text{K} \), the pressure drop is \( 320 \text{Pa} \), and the fiber-to-nozzle distance is \( 80 \text{mm} \), the fiber bundle spreading effect is optimal, which can be spread to \( 65 \text{mm} \).

6. Acknowledgment
Financial support from the Natural Science Foundation of Tianjin (No: 18JCYBJC89000) is acknowledged.

7. References
[1] Lu R, He X M 2012 The Application of Carbon Fiber Concrete in the Intelligent Bridge Reinforcement. Advanced Materials Research. 594-597: 1539-1542.
[2] Chen, Yan Y 2014 The Application of Carbon Fiber Materials in Sports Equipment. Applied Mechanics and Materials. 687-691: 4240-4243.
[3] Wang F F 2017 Study on the Application of Carbon Fiber Composite Materials in High-Speed Trains. Materials Science Forum. 893: 31-34.
[4] Irfan M S, Machavaram V R, Mahendran R S, Shotton-Gale N, Wait C F, Paget M A, Hudson M, Fernando G F 2011 Lateral spreading of a fiber bundle via mechanical means. J. Compos Mater. 46 (3): 311-330.
[5] Sebaey T A, Mahdi E, Manufacturing 2016 Using thin-plies to improve the damage resistance and tolerance of aeronautical CFRP composites. Compos. Pt. A- Appl. Sci. Manuf. 86: 31-38.
[6] Borg C 2015 An introduction to spread tow reinforcements: Part 1 – Manufacture and properties. Reinforced Plastics. 59 (4): 194-198.
[7] Park S M, Kim M S, Choi Y S, Lee E S, Yoo H W, Chon J S 2015 Carbon Fiber Tow Spreading Technology and Mechanical Properties of Laminate Composites. Composites Research. 28 (5): 249-253.
[8] Baucom R M, Marchello J M 1990 LaRC powder prepreg system. 21: 4.
[9] Chen J C, Chao C G 2005 Numerical simulation and experimental investigation for design of a carbon fiber tow pneumatic spreading system. Carbon. 43 (12): 2514-2529.
[10] Sihn S, Kim R, Kawabe K, Tsai S 2007 Experimental studies of thin-ply laminated composites. Compos. Sci. Technol. 67 (6): 996-1008.
[11] El-Dessouky H M, Lawrence C A 2013 Ultra-lightweight carbon fiber/thermoplastic composite material using spread tow technology. Compos. Pt. B-Eng. 50: 91-97.
[12] Ren F, Yu Y, Cao M, Li Y, Xin C, He Y 2017 Effect of pneumatic spreading on impregnation and fiber fracture of continuous fiber-reinforced thermoplastic composites. J. Reinf. Plast. Compos. 36 (21): 1554-1563.
[13] Huang B, Yuan S, Xue J 2015 Spreading process simulation and experimental investigation for a carbon fiber pneumatic spreader. Journal of Xian Jiaotong University.
[14] Launder B E, Spalding D B 2014 The numerical computation of turbulent flows. Comput. Meth. Appl. Mech. Eng. 3 (2): 269-289.