ABSTRACT

CAI, YIYUN, Computer Modeling of Fiber Motion in High-speed Airflow (Under the Direction of Dr. William Oxenham.)

Today high-speed airflow is widely used in many processes in textile industry. Even though the interactions between fibers and airflows have attracted many researchers’ interest, there have been few published studies that focus on the computer modeling of fiber motion in airflow fields.

The present research was aimed at developing a model that can effectively simulate the interactions between fibers and airflows, thus providing clearer understanding of the behavior of textile fibers in certain processing machines.

A three-dimensional model of an aerodynamic component of a textile machine was developed. A commercial computation fluid dynamics (CFD) software package was used to compute the airflow field of this model and the results were analyzed to study the airflow field’s characteristics. Resultant data were used as input for the fiber movement model by using a one-way coupling method.

The mathematical model of fiber movement was constructed by integrating the governing equations with a model that describes the fiber configurations. A numerical method was developed to solve these equations and visualization programs were established to illustrate and animate the simulated fiber movements. The results obtained were studied and compared under different initial and boundary conditions.

Fiber bending and twisting properties were integrated into the computer model. Their influences on the fiber movement were simulated and analyzed.
The present research successfully demonstrates the effectiveness of computer modeling for studying the fiber motion in high-speed airflow. It can provide better understanding of fiber behavior in airflow fields and its potential and prospect in the research of textile processes, in which airflow plays an important role, are very promising.
COMPUTER MODELING OF FIBER MOTION IN HIGH-SPEED AIRFLOW

By

YIYUN CAI

A dissertation submitted to the graduate faculty of North Carolina State University in partial fulfillment of the requirements for the Degree of Doctor of Philosophy

FIBER AND POLYMER SCIENCE

Raleigh
March 2003

Approved by

Dr. George Hodge  
Member of Advisory Committee

Dr. Yiping Qiu  
Member of Advisory Committee

Dr. Douglas Reeves  
Member of Advisory Committee

Dr. William Oxenham  
Chair of Advisory Committee
To My Parents and My Wife
BIOGRAPHY

Yiyun Cai was born in Tangjiang, Jiangxi Province, China on January 4, 1974. He received his Bachelor degree in Thermal Energy Engineering from Beijing University of Aeronautics and Astronautics in 1995. He worked in Shanghai Scientific Instrument Factory, China Academy of Space Technology for one year, and then he received his M.S. degree in Fluid Machinery and Engineering from Beijing University of Aeronautics and Astronautics in 1999. He came to College of Textiles, North Carolina State University in 1999; he received a Master of Textiles degree in 2001 on his way to a Ph.D. degree in Fiber and Polymer Science.
ACKNOWLEDGEMENTS

The Author wishes to express his appreciation to his committee chair, Dr. William Oxenham, for his guidance, encouragement, and patience throughout the whole course of the research and the preparation of this dissertation. The author also wishes to express his appreciation to Drs. George Hodge, Yiping Qiu, and Douglas Reeves for their valuable suggestions.

The author is also indebted to the National Textile Center for the financial support for Project F99-NS06 “Fiber Motion and Yarn Forming in High-Speed Air Flows” that enable the present research to be conducted.
TABLE OF CONTENTS

List of Figures .................................................................................................................. viii
List of Tables .................................................................................................................. xii
List of Symbols ............................................................................................................... xiii

1. Introduction ................................................................................................................. 1

2. Overview of Literature ............................................................................................... 3

2.1 Introduction ........................................................................................................... 3

2.2 Studies on Airflow in Textile Processes ............................................................... 3

2.2.1 Rotor Spinning ............................................................................................... 3

2.2.2 Air-Jet Spinning ............................................................................................. 7

2.2.3 Air-Jet Texturing ............................................................................................ 9

2.2.4 Air-Jet Weaving ........................................................................................... 17

2.2.5 Applications in Nonwovens ........................................................................ 20

2.3 Studies on airflow acting on textile fiber / filament ........................................... 21

2.4 A Overview of Methods Used for Fiber-Airflow Interaction Modeling ............. 24

3. Computation of Airflow Field .................................................................................. 28

3.1 Introduction ........................................................................................................... 28

3.2 CFD-ACE+ .......................................................................................................... 29

3.2.1 CFD-GEOM ................................................................................................ 30

3.2.2 CFD-ACE and CDF-GUI .......................................................................... 30

3.2.3 CFD-VIEW .................................................................................................. 32

3.3 CFD Results Processing and Integration with Fiber Movement Model ............. 33

4. Aerodynamics Analysis of Physical Models ....................................................... 35

4.1 Introduction ........................................................................................................... 35
4.2 Geometrical Configurations and Dimensions ....................................................... 36
4.3 Boundary and Initial Conditions for CFD .......................................................... 37
4.3 Analysis based on One-dimensional Aerodynamics Estimation ....................... 40
4.4 Outlet Back Pressure ...................................................................................... 42
4.5 Channel Outlet Size ....................................................................................... 43
4.6 Opening Roller’s Influence to Airflow at the Channel Inlet ......................... 45
4.7 Re-circulation Zone ....................................................................................... 45
4.8 CFD Results Analysis .................................................................................... 46
  4.8.1. CFD Study on Channel Outlet Size ............................................................ 47
  4.8.2. CFD Study on Channel Inlet Airflow Velocity ............................................ 54
  4.8.3. CFD Study on Channel Outlet Back Pressure ............................................ 59
  4.8.4. CFD Study on Re-circulation Zone inside Channel .................................. 63
5. Basic Equations of Fiber Modeling ...................................................................... 70
  5.1 Introduction ................................................................................................. 70
  5.2 Basic Equations ........................................................................................... 70
  5.3 Model of Fiber Configuration ....................................................................... 73
  5.4 Fiber Movement Model ............................................................................... 75
  5.5 Discrete Equations Integrated with the Fiber Movement Model .................. 79
6. Model’s Numerical Solution and Results ............................................................ 84
  6.1 Introduction ................................................................................................. 84
  6.2 Methods Used for Solutions ......................................................................... 84
  6.3 Results of Simulation .................................................................................. 91
    6.3.1 Fiber’s Initial Position ............................................................................. 93
    6.3.2 Fiber’s Initial Speed .............................................................................. 100
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.3.3 Airflow Field Velocity Distribution</td>
<td>102</td>
</tr>
<tr>
<td>7. Modeling of Fiber Bending and Twisting Properties</td>
<td>105</td>
</tr>
<tr>
<td>7.1 Introduction</td>
<td>105</td>
</tr>
<tr>
<td>7.2 The Definitions of Fiber Bending and Twisting Parameters</td>
<td>105</td>
</tr>
<tr>
<td>7.3 The Integration of Bending and Twisting Properties into the Three-</td>
<td>107</td>
</tr>
<tr>
<td>dimensional Mathematical Model</td>
<td></td>
</tr>
<tr>
<td>7.3.1. Numerical Methods for Computing Bending and Twisting moments</td>
<td>107</td>
</tr>
<tr>
<td>7.3.2. The Integration of Bending and Twisting Moments with the</td>
<td>111</td>
</tr>
<tr>
<td>Governing Equations</td>
<td></td>
</tr>
<tr>
<td>7.4 Results of Simulations with bending and Twisting Properties</td>
<td>114</td>
</tr>
<tr>
<td>8. Summary and Conclusions</td>
<td>121</td>
</tr>
<tr>
<td>9. Recommendations for Future Work</td>
<td>125</td>
</tr>
<tr>
<td>10. References</td>
<td>127</td>
</tr>
<tr>
<td>Figure 2.1</td>
<td>Definition of the half angle .......................................................... 4</td>
</tr>
<tr>
<td>Figure 2.2</td>
<td>Rotor spinning box ................................................................. 5</td>
</tr>
<tr>
<td>Figure 2.3</td>
<td>Pressurized airflow used to disturb the fiber flow .......................... 6</td>
</tr>
<tr>
<td>Figure 2.4</td>
<td>Murata type air-jet spinning ...................................................... 7</td>
</tr>
<tr>
<td>Figure 2.5</td>
<td>Measurement of the axial velocities of a trumpet-shaped nozzle ....... 10</td>
</tr>
<tr>
<td>Figure 2.6</td>
<td>Schematic diagram of the feed and delivery zones, showing the frictional forces acting on the filaments ........................... 12</td>
</tr>
<tr>
<td>Figure 2.7</td>
<td>Incremental textiles fiber in airflow ............................................. 12</td>
</tr>
<tr>
<td>Figure 2.8</td>
<td>Illustration of the variation of the filament projected area with its position in the nozzle ..................................................... 14</td>
</tr>
<tr>
<td>Figure 2.9</td>
<td>Control volumes in cylindrical nozzles as used in mathematical modeling .......................................................... 15</td>
</tr>
<tr>
<td>Figure 2.10</td>
<td>Rectangular shape air-jet texturing nozzle ..................................... 17</td>
</tr>
<tr>
<td>Figure 2.11</td>
<td>Analysis of loop storage .............................................................. 19</td>
</tr>
<tr>
<td>Figure 2.12</td>
<td>Fiber configuration used in Smith and Robert’s research ................ 26</td>
</tr>
<tr>
<td>Figure 2.13</td>
<td>Contours of streamlines and velocity distribution ............................ 26</td>
</tr>
<tr>
<td>Figure 2.14</td>
<td>Fiber configuration used in Kong and Platfoot’s research ............... 27</td>
</tr>
<tr>
<td>Figure 3.1</td>
<td>Process flow of CFD-ACE+ .......................................................... 30</td>
</tr>
<tr>
<td>Figure 3.2</td>
<td>Physical model imported to CFD-GEOM ........................................ 31</td>
</tr>
<tr>
<td>Figure 3.3</td>
<td>Grid generated by the CFD software ............................................. 31</td>
</tr>
<tr>
<td>Figure 3.4</td>
<td>Calculating of the airflow field ..................................................... 32</td>
</tr>
<tr>
<td>Figure 3.5</td>
<td>CFD result shows the velocity distribution of the airflow field .......... 33</td>
</tr>
<tr>
<td>Figure 4.1</td>
<td>The layout of a rotor-spinning box ............................................... 36</td>
</tr>
</tbody>
</table>
Figure 4.2. Structure of the fiber transfer channel simulated with CFD software ...... 38
Figure 4.3. Fiber transfer channel CFD model grid generated by software package and boundary conditions............................................................. 39
Figure 4.4. Simplified fiber transfer channel for estimation ........................................ 40
Figure 4.5. Airflow velocity distribution along the center line (4x4 mm² outlet) ...... 49
Figure 4.6. Airflow velocity distribution along the center line (6x4 mm² outlet)...... 49
Figure 4.7. Airflow velocity distribution along the center line (8x4 mm² outlet)...... 50
Figure 4.8. Airflow velocity distribution along the center line (10x4 mm² outlet)..... 50
Figure 4.9. Airflow velocity distribution along the center line (12x4 mm² outlet)..... 51
Figure 4.10. Airflow velocity distribution along the center line (14x4 mm² outlet)..... 51
Figure 4.11. Change of center line maximum speed with outlet size ...................... 52
Figure 4.12. Speed distributions along the Z direction center lines at outlet cross-sections ...................................................................................................... 52
Figure 4.13. Speed contours of V (Velocity at Y-axis direction).......................... 53
Figure 4.14. Airflow velocity distribution along the center line (6500 rpm)......... 56
Figure 4.15. Airflow velocity distribution along the center line (7000 rpm)........ 56
Figure 4.16. Airflow velocity distribution along the center line (7500 rpm)........ 57
Figure 4.17. Airflow velocity distribution along the center line (8000 rpm)........ 57
Figure 4.18. Change of center line maximum speed with inlet airflow speed .......... 58
Figure 4.19. Speed distributions along the Z direction center lines at outlet cross-sections ...................................................................................................... 58
Figure 4.20. Airflow velocity distribution along the center line (60 mbar).......... 60
Figure 4.21. Airflow velocity distribution along the center line (70 mbar).......... 60
Figure 4.22. Airflow velocity distribution along the center line (90 mbar).......... 61
Figure 4.23. Airflow velocity distribution along the center line (100 mbar) ................. 61
Figure 4.24. Change of center line maximum speed with outlet back pressure .......... 62
Figure 4.25. Speed distributions along the Z direction center lines at outlet cross-
sections .................................................................................................................. 62
Figure 4.26. Three-dimensional structure of a re-circulation zone .............................. 63
Figure 4.27. V speed distribution in I-planes near the channel inlet ............................ 64
Figure 4.28. Static pressure distribution along the channel center line .................... 65
Figure 4.29. $V_y$ distribution along the center chord line 7 ................................... 66
Figure 4.30. $V_y$ distribution along the center chord line 19 ................................... 67
Figure 4.31. $V_y$ distribution along the center chord line 7 ................................... 68
Figure 4.32. $V_y$ distribution along the center chord line 19 ................................... 69
Figure 5.1. Forces acting on a fiber element .............................................................. 71
Figure 5.2. Different fiber modeling methods ............................................................. 74
Figure 5.3. Forces acting on one node ....................................................................... 75
Figure 5.4. Forces acting on one rod ......................................................................... 77
Figure 5.5. Drag coefficient for spheres ..................................................................... 82
Figure 6.1. Simulation result shows a bended fiber was straightened while passing through the channel ................................................................. 92
Figure 6.2. Computer modeling result shows a straight fiber changed its orientation after passing through the channel ................................................. 92
Figure 6.3. Selected starting points of fiber initial position ......................................... 94
Figure 6.4. The configurations of fibers of different initial positions at the beginning of movements ................................................................. 95
Figure 6.5. Results of simulations show the fiber movements starting from different initial positions along the J-direction ...................................... 97
Figure 6.6. Fiber tip’s speed at z direction as initial position moving close to channel wall. ................................................................. 98

Figure 6.7. Fiber straightening while moving in the channel.......................................... 99

Figure 6.8. Fiber acceleration while moving through the fiber transfer channel. ..... 101

Figure 6.9. Fiber acceleration with different initial speeds....................................... 101

Figure 6.10. Fiber acceleration with different channel outlet back pressures......... 103

Figure 6.11. Fiber configurations different channel outlet back pressures ............ 104

Figure 7.1. Calculation of fiber curvature at a point. .............................................. 109

Figure 7.2. Calculation of fiber torsion at a point. ................................................ 110

Figure 7.3. Bending and twisting moment acting on the fiber............................... 111

Figure 7.4. Forces and moments acting on one rod of the modeled fiber.......... 112

Figure 7.5. Fiber straightening-factor vs. fiber flexural rigidity ......................... 115

Figure 7.6. Fiber-straightening factor vs. fiber torsional rigidity....................... 116

Figure 7.7. Fiber-straightening factor vs. different combinations of flexural and torsional rigidities.................................................. 116

Figure 7.8. Simulated fiber movements with different flexural rigidities .......... 117

Figure 7.9. Simulated fiber movements with different torsional rigidities ......... 118

Figure 7.10. Simulated fiber movements with different combinations of flexural and torsional rigidities........................................... 119

Figure 7.11. The fiber movement when it has extremely large rigidities. .......... 120
# LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.1</td>
<td>Channel outlet size and convergent angle</td>
<td>48</td>
</tr>
<tr>
<td>4.2</td>
<td>Inlet airflow velocity</td>
<td>55</td>
</tr>
<tr>
<td>4.3</td>
<td>Outlet back pressure</td>
<td>59</td>
</tr>
<tr>
<td>6.1</td>
<td>Air and fiber properties used in simulation programs</td>
<td>91</td>
</tr>
<tr>
<td>6.2</td>
<td>Fiber-straightening factor as fibers starting from different positions</td>
<td>100</td>
</tr>
<tr>
<td>7.1</td>
<td>Flexural and torsional rigidities for simulations</td>
<td>107</td>
</tr>
<tr>
<td>7.2</td>
<td>The combinations of Flexural and Torsional rigidities for simulations</td>
<td>114</td>
</tr>
</tbody>
</table>
LIST OF SYMBOLS

a: Acceleration

$C_D$: Drag coefficient

$F_i$: Force

$f$: Force

$k$: Specific heat ratio

$\kappa$: Curvature

$L$: Dimension (length)

$M$: Torque

$m$: Mass

$Ma$: Mach number

$P$: Pressure

$\rho$: Density

$\rho_i$: Fiber linear density

$Re$: Reynolds number

$R_B$: Flexural rigidity

$R_T$: Torsional rigidity

$t$: Time

$\tau$: Torsion

$V_i$: Velocity

$v$: Velocity

$\nu$: Kinematic viscosity

$x, y, z$: Coordinates
1. INTRODUCTION

Airflow has been used in many processes in textile industry for a long time, for example, in the transport of fiber tufts and assisting moisture removal from fibers, yarns, and fabrics. While developments in science and technology (e.g. introduction of microcomputers) have brought about radical changes in the textile industry, this has been accompanied by a significant growth in the use of high-speed airflow in many textile processes. The optimum use of high-speed airflow is a critical component of many "new" processes such as:

- Rotor, air-jet, vortex and friction spinning;
- Intermingling, air texturing, and splicing;
- Advanced fiber and yarn testing equipment;
- Melt blown and spun bond nonwovens;
- Air-jet weaving.

For example, in the area of staple yarn manufacture, there are five essential steps, namely: — separating, cleaning, drafting, condensing and twisting. Airflow can be used to achieve all of these steps because of its high speed and energy. However recent incursions into processes such as jet spinning have clearly shown that there can be disadvantage associated with the use of air in fiber processing and in particular its different interaction with different fibers. Furthermore concerns about the economics of jet spinning, due to inefficient use of the air, has resulted in ongoing "improvements" such as the substitution of air twisters by mechanical twistes.
Even though there is reported experimental work regarding airflow in fabric and yarn processing there is a scarcity of publications on the modeling of fiber and yarn motion in airflow. Furthermore very little of the reported work relates to high-speed airflow.

The goal of this research is to develop a clearer understanding of the behavior of textiles in high-speed airflow using computer modeling. Emphasis will be placed on establishing computer models that can elucidate the interaction of fibers with airflows.

The computer modeling is based on the following parts:

1). Calculating the airflow field in a particular component of the textile machinery.

2). Constructing the mathematical description of the physical phenomena of fiber motion in airflow field.

3). Developing computer programs to generate the numerical solution of the equations from the previous step.

4). Developing computer programs for result analyses and model visualization.
2. OVERVIEW OF LITERATURE

2.1 Introduction

There have been many research projects about airflow on textile processes, including rotor spinning, air-jet spinning, and air-jet texturing, etc., these research activities were carried out both experimentally and theoretically. In this chapter, an overview of these research projects was presented and their methodologies were analyzed. This overview attempts to provide a basic outline on the research and development of applications of airflow on textile machinery.

In addition, from this overview, it can be seen that one direction of this type of research is the development in methods of computer modeling; this forms the basis of the present research, which was aimed at determining the effectiveness and prospect of airflow modeling as applied to textile processes.

2.2 Studies on Airflow in Textile Processes

2.2.1 Rotor Spinning

In 1968, Edberg published his study in cotton fiber behavior subjected to the aerodynamic forces \(^{(1)}\). Fiber behavior in laminar airflow and turbulent airflow were both studied. Experiments were carried out in a wind tunnel, and the parallelization effect of the airflow on the fiber was tested at different flow velocities. In addition, configuration of the wind tunnel was changed by varying the half angle of convergence (see Figure 2.1). In experiments, velocities were measured by using a hot wire anemometer when the wind tunnel was run empty. Any velocity change caused by the fibers was neglected, and
while this is not strictly true, it is an assumption also used in computer modeling to enable solution to be more easily achieved.

![Figure 2.1 Definition of the half angle](image)

The results from the experiments showed that:

1. Fiber became more parallel to the airflow as the velocity of the airflow increased.

2. As the half angle of the convergence of the wind tunnel increased, the fiber parallelization was also increased.

However, Edberg did not give out a clear explanation about the action that produced the fiber straightening. A review of the paper shows that Edberg did not compare a velocity gradient at the direction which is perpendicular to the axis of the airflow, he indicated a parameter, named “mean acceleration per meter” \( (a_m) \), which was defined as:

\[
a_m = \frac{V_2 - V_1}{\text{convergent length}} \text{ sec}^{-1}
\]

Where \( V_1 \) is the inlet airflow velocity and \( V_2 \) is the outlet airflow velocity.

Much of the research on rotor spinning has been based on experimental studies. Acar and King used high-speed photography to study the fiber alignment and straightening properties in rotor spinning \(^{(2)}\). The draft was defined as the ratio of the speed of fibers leaving the opening roller to the speed of fibers 28mm downstream from the opening...
roller (see Figure 2.2). They found that as the draft increases, the degree of fiber straightness improves along the axis of airflow. They also concluded that the reason was that the higher drafting forces produce wider differences between the speeds of the leading and trailing ends of the fibers. Therefore, at constant airflow speed, as the speed of the opening roller is increased, fiber straightness and degree of alignment along the axis of flow deteriorate. At constant opening roller speed, as airflow speed increases, fiber straightness and degree of alignment along the axis of airflow improve. However, they did not include the consideration of the impact of the shape of the fiber transport channel on the resultant airflow field.

![Transport Channel and Opening Roller Outlet](image)

**Figure 2.2**    Rotor spinning box \(^{(2)}\)

In rotor spinning, airflow need not only be used to transport paralleled fibers. Kwasniak used an additional pressurized airflow to disturb the airflow field and then to produce fancy yarns \(^{(3)}\). At first, pressurized airflow was directed to different locations inside the rotor spinning box to evaluate the most suitable way to disturb the fiber. These locations are (see Figure 2.3)

(1). Rotor groove,
(2). Rotor bottom,

(3). Opening roller,

(4) Fiber transport channel.

Kwasniak found that the best location, at which the pressurized airflow was placed to disturb the fiber flow so fancy yarn can be produced, was in the fiber transfer channel. A specific location in the transport channel was found to give the optimal effect; the direction of the pressurized airflow was against the direction of the fiber flow (see Figure 2.3).

![Diagram of fiber processing system](image)

Figure 2.3  Pressurized airflow used to disturb the fiber flow

A series of experiments was carried out to test the configuration of this new method, and this included the influence of the following parameters on the characteristics of the fancy yarn:

(1) Nozzle,

(2) Rotor diameter and rotor groove,

(3) Opening roller speed,

(4) Rotor speed.

In addition, continuous blowing and intermittent blowing were used to test different fancy yarn effects. A theoretical analysis was carried out in another paper by Kwasniak.
and Peterson (4). Kwasniak also tested the method in commercial rotor spinning machines (5). The most commonly used methods for producing fancy yarns on rotor spinning are feeding excess material to the rotor, but the disadvantage is that effects can not be shorter than the rotor circumference. Kwasniak and Peterson developed the technology to make it possible to divide long effects into short ones by blowing pressurized air to the fiber transportation channel (6).

2.2.2 Air-Jet Spinning

Airflow is used in air-jet spinning to generate the action of “false twist” to the drafted sliver that goes into the twist chamber (see Figure 2.4). This is achieved by a high-speed airflow that is injected from the nozzle. A vortex is formed by the circular shape of the twist chamber and the direction of the nozzle’s outlet.

Much research has been done about the properties of the air-jet spun yarn. Grosberg, Oxenham and Miao compared air-jet spun yarn properties on three different arrangements of air-jets (7), the results show that Murata type twin-jet arrangement (see Figure 2.4) produces well-wrapped, strong yarns. Based on microscopy observations, the yarn structure was classified into four categories.

Grosberg, Oxenham and Miao carried out a theoretical analysis to the twisting kinematics (8) of the yarn. An assumption was made with regard to the rate of twist flow to make a prediction agreeable with experimental results.
By modifying the jet design, Oxenham and Basu studied the influence to the strength of cotton air-jet spun yarns \(^9\). A significant improvement in yarn tenacity was achieved by using a proper jet angle.

Chasmawala, Hansen and Jayaraman studied the structure and properties of the air-jet spun yarn \(^{10}\). Yarn structure was divided to five fiber categories: core, wrapper, wild, core – wide, and wrapper – wild. The relationship between yarn structure and first nozzle pressure and the draft ratio was investigated. They found that the number of core fibers in the air-jet spun yarn decreases with the increase of the first nozzle pressure and the draft ratio. With increasing first nozzle pressure and draft ratio, yarn properties, such as breaking load, breaking elongation, hairiness, and irregularity, register higher values. One of the relationships between the yarn structure and yarn properties that has been found is that as the number of core fibers increases, the yarn strength and yarn hairiness decrease.

There is little published about computer simulation of air-jet spinning. Rajamanickan, Hansen and Jayaraman used a computer to carry out a parameter research on air-jet spinning \(^{11}\). They studied the relationship of the following interactive factors using “computer simulation”:

- Yarn count and fiber fineness
- Fiber tenacity and fiber friction
- Fiber length and fiber friction
- Number of wrapper fibers and wrap angle

The authors gave out suggestions of yarn engineering guidelines to optimize yarn strength using the results of each of the four simulations.
2.2.3 Air-Jet Texturing

By using high speed, high pressure airflow, the multi-filament yarn can be textured in the air-jet nozzle, as yarn is “overfed” into the nozzle, longitudinal displacements of yarns and loops will be formed in the extremely violent airflow stream.

There is much literature discussing the mechanism of the air-jet texturing process. Acar and Wray gave a comprehensive review of the development of understanding of air-jet texturing technology (12). A discussion of air-jet texturing mechanism was presented based on important opinions and observations from previous research. The previous work can be briefly summarized as:

Wray and Enstwistle postulated that the turbulent air stream’s false-twist action untwisted the twisted yarns. Their analysis was based on high-speed photography (13, 14).

Sen and Wray suggested an alternative hypothesis of the loop formation of the filament during the air-jet texture process (15, 16, 17). That is that the end of the yarn withdrawn from the jet exit was subjected to an alternative force at right angle to its axis due to the momentum of the blown-out yarn. The false-twisting effect was created, and the structure is opened. Then the extremely violent natures of the airflow blew the yarn into a looped and entangle state. For the first time, Sen showed the shock wave occurred out side the nozzle.

Sivakumar concluded that the existence of shock waves played a very important role in loop formation (18). “Pressure barrier” formed by the shock waves retards the overfed yarn, yarn tension suddenly is reduced and filaments snarl and form loops.

It is important to note that the above hypotheses are all based on a pre-twisted supply yarn, and it is claimed that the theories are not applicable when the supply yarn is not pre-
twisted (12). Based on their experiments, Bock and Lünenschloss also concluded that there is a force within the air stream that bends the filaments (19, 20), and the “pressure barrier” of the shock waves causes this force.

Acar, Turton, and Wray carried out their study on the mechanism of air-jet texture by using a scaled-up model of the HemaJet nozzle (21). They claimed that in their experiments a geometrically similar model was also a dynamically similar model. So experimental information obtained from the scaled-up model was claimed to be a representative of the original nozzle.

They used a Pitot-tube system to measure the velocity and pressure distribution. Axial-velocity distribution was measured and the effects of the trumpet-shaped diverging exit (Figure 2.5) were investigated. However, their attempt to measure the air velocity with filaments inside the nozzle failed due to the filaments’ interference to the measuring probes.

Figure 2.5  Measurement of the axial velocities of a trumpet-shaped nozzle (21)

The variation of local air velocity caused longitudinal displacements of the filaments relative to each other. Since it was claimed that the aerodynamic force acting on scattered
filaments was a function of the square of the local velocity, they suggested that this variation was essential to the texturing process.

They also used shadowgraphs to study the free airflow and disturbed airflow (21). These shadowgraphs showed that even though shock waves could exist in free airflow, the shock waves in an airflow disturbed by filaments were destroyed.

Based on their findings, they concluded contrary to earlier work that the shock waves did not play any significant role in the texturing process.

Acar, Turton, and Wray used high-speed still and cine-photographic techniques to study the behavior of filaments during texturing with an industrial texturing nozzle (22). Filaments’ motion induced by airflow was investigated. High-speed photos confirmed that the mechanism of “loop formation by shock wave” is invalid. The separation of the filaments in the yarn is essential for effective texturing. They also found that the average yarn speed in the airflow increases with increasing air pressure. The average yarn tension in the delivery zone, texturing speed, air pressure, and overfeed ratio were also studied in their experiments, and the results obtained were considered useful information to evaluated the effectiveness of the texturing and the stability of loop formation.

Aerodynamic forces acting on filaments during air-jet texturing were also studied theoretically by Acar, Turton, and Wray (23). They established a model used to describe the forces acting on the filaments \((F_r)\), which was defined as a summation of air drag from primary flow \((F_p)\), secondary flow \((F_s)\), and frictional forces \((F_t)\) (see Figure 2.6).

For simplifying the problem, the air drag on an incremental section of a filament was analyzed. In addition, it is assumed that this section is rigid and the movement is two-dimensional (see Figure 2.7).
Figure 2.6  Schematic diagram of the feed and delivery zones, showing the frictional forces acting on the filaments \(^{(23)}\)

\[
F_f = f_1 + f_2 + f_3 \\
F_r = F_p - F_s - F_f
\]

Figure 2.7  Incremental textiles fiber in airflow \(^{(23)}\)

The velocities of the fiber are:

\[
V_n = (u - u_g) \sin \phi + (v - v_g) \cos \phi + r \phi \\
V_t = (u - u_g) \cos \phi + (v - v_g) \sin \phi
\]
\[ V_t = (u - u_g) \cos \Phi + (v - v_g) \sin \Phi \]

\( V_n \) and \( V_t \) are the normal and tangential velocity components relative to the airflow. \( u \) and \( v \) are the velocity components of the airflow at the fiber’s mass center. \( u_g \) and \( v_g \) are the velocity components of the fiber’s mass center.

The air drags acting on the fiber are:

\[ dD_p = \frac{1}{2} c_p \rho d_f V_n^2 dl \]

So:

\[ D_p = \rho \frac{d_f}{2} \int_{-L/2}^{L/2} C_p V_n^2 dl \]

\[ D_f = \rho \frac{d_f}{2} \int_{-L/2}^{L/2} C_f V_f^2 dl \]

Then:

\[ D_x = D_p \sin \phi + D_f \cos \phi \]

\[ D_y = D_p \cos \phi + D_f \sin \phi \]

The factors that affect air drag were discussed in that study, these factors includes air supply pressure, design of nozzle, filament’s cross-section, position across the nozzle, and orientation.

The projected area of the filament in the airflow is determined by the position of the filament in the nozzle (Figure. 2.8). Higher aerodynamic force will act on a filament that is closer to the centerline of the nozzle. Since aerodynamic forces acting on filaments at different positions are different, there is a longitudinal velocity difference along the filaments. Filament cross-section was also studied in that paper (23). For a filament with circular cross-section, a larger diameter causes greater air drag to act on the filament. Its
inertial resistance to the air drag increases proportionally to the square of the filament diameter. In addition, the bending stiffness also increases with fiber diameter. Therefore, they concluded that supply yarns composed of finer filaments could be textured more satisfactorily than those composed of coarser filaments could.

After comparing filaments with circular, elliptical, and hollow-circular cross-section shape, they concluded that with the same cross-section area, supply yarn composed of filaments that have smaller second moment of area and polar second moment of area require smaller forces to bend and twist them.

![Illustration of the variation of the filament projected area with its position in the nozzle](image2)

Figure 2.8  Illustration of the variation of the filament projected area with its position in the nozzle

A detailed mathematical model used to describe the airflow within an air-jet texture nozzle was derived by Acar et al. (24, 25). They divided the air-jet nozzles into two
categories: converging-diverging nozzle, and cylindrical nozzles. The airflow in cylindrical nozzles was modeled.

The airflow in a cylindrical nozzle was considered to have three control volumes, and the airflow in each of the three volumes was assumed isentropic and one-dimensional, as shown in Figure 2.9.

![Control volumes in cylindrical nozzles as used in mathematical modeling](image)

Volume 1 consisted three inlet bores. Those bores were actually convergent nozzles. Because the operation pressure of airflow is much higher than critical pressure, the airflows at the throats of these inlets were considered to have reached their critical conditions, which can be determined by one-dimensional aerodynamics equations.

Airflows from inlets mixed in control volume 2. These are considered to be in the same critical conditions as they left control volume 1. It was assumed that, in control volume 2, the airflow did not expand. Then the airflow in control volume was assumed incompressible. The areas of primary flow and secondary flow, and the mass rate of each flow were calculated with the theory of collision and mixing of jets, which considers the
mass flow conservation and momentum change alone the direction of the axis of the nozzle.

In control volume 3, the outgoing primary flow and secondary flow were treated as a sudden expansion of an abrupt enlargement. The Mach numbers of both flows was calculated based on the assumption that the static pressures at the exits of the nozzle were atmospheric. Then other properties of the flows can be determined.

Acar et al. also carried out a series of trials using experimental cylindrical nozzles that were larger than industrial nozzles and had no trumpet-shaped diverging exit. The experiments’ results demonstrated a good agreement with the mathematical model.

Versteeg, Bilin, and Acar compared airflow characteristics and texturing performance of nine nozzles (26), trying to find out their relationships with the geometries of nozzles. Their results showed that the variation in strength properties and increase in linear density is very small even thought those nozzles’ geometries were very different. The fact that some nozzles with very weak shock waves produce good texturing indicated the presence of shock waves was not a necessary condition for good texturing. In addition, they did not find correlation between mean total pressure in the exit region and texturing quality. However, these nozzles did produce textured yarns with different visual appearances.

Bilgin et al. continued their research on influences of nozzle geometry on air-jet texturing performance (27). Air-jet nozzles with rectangular cross section (Figure 2.10) were developed to investigate the relationships between air-jet geometric parameters and texturing performance. Their earlier tests showed that yarns textured by rectangular nozzles had similar properties and qualities to yarns textured by conventional nozzles.
They concluded that higher stabilizing zone tension, increased linear density, and lower instability were effective indicators of better texturing qualities.

![Figure 2.10 Rectangular shape air-jet texturing nozzle](image)

Results from their experiments showed that with longer primary flow length (shown in Figure 2.10 as $L_p$) inside the nozzles, and a higher stabilizing zone tension, the increases in linear density were higher, and instability was lower. These suggest that longer primary flow length could help in achieving better texturing qualities. The effects of air inlet angles were not conclusive. It was found that a jet that had an air inlet very close to the exit, or a jet that did not have a curved diverging exit, did not produce stable process.

Techniques similar to air-jet nozzles were also applied to intermingling nozzles (25, 28).

**2.2.4 Air-Jet Weaving**

The effect of nozzle design on the air velocity for an air-jet filling insertion system has been studied by Mohamed and Salama (29). Experimental nozzles were used to study the influence from parameters of the nozzle structure on the airflow characteristics at the nozzle exit. Theoretical analysis based on one-dimensional flow was carried out to
explain the nozzle performance. Relationships among air velocity, turbulence, and flow rate at the nozzle exit and nozzle structure parameters, such as air tube length and air tube diameter were reported.

Salama and Mohamed also studied the air velocity distribution along the air guide system of an air-jet loom (30). Experimental apparatus was designed and built to study the different factors involved in air-jet filling insertion. The influences of the dimensions and configurations of the air guiding system on the air velocity distribution were reported. The effects of slots inside the guide system were discussed.

Adanur and Mohamed carried out theoretical models for yarn motion in air-jet filling insertion (31). The model was built based on the following assumptions: The yarn has uniform properties, such as constant diameter, constant linear density, and inextensible; the yarn moves along a straight line, which means no fluctuation due to turbulent and unsteady flow; and the effect of gravitational force was neglected.

Two models were postulated, one for drum storage, the other for loop storage, both models developed equations that describe the motion of yarns in the airflow.

In the model for drum storage, the total force was considered consist of air-friction force and yarn tension from the yarn guide.

In the model for loop storage, the total force was considered to consist of air-friction force and force acting on the loop. The loop was assumed to keep the same shape during the yarn motion.

Both models derived the acceleration of the yarn during the process. Both are first-order, non-linear, ordinary differential equations. The results from both models only
showed the acceleration of yarns during those two kinds of air-jet filling insertion processes, but the deformation of yarns was not considered.

![Diagram of loop storage](image)

Figure 2.11  Analysis of loop storage (31)

The above theoretical models were validated by experiments (32). The results show that:

1. Yarns with high linear density had longer insertion times
2. Loop storage gives higher yarn velocities than drum for all yarns
3. Textured yarn has relatively high yarn velocity with loop storage, but there is no significant change in yarn velocity in the drum case.

These experimental results showed discrepancies from the computer model results. One reason is that the computer model assumed that the yarn moved along a straight line inside the tube, so the flexibility and possible bulking of the yarn was neglected.

Adanur and Mohamed also studied the parameters that affect the yarn velocity of the weft insertion (33). They found that in air-jet weft insertion, the yarns are not positively pulled, but inserted by air-drag force, which is provided by the friction between the airflow and the yarn surface.
They reported that with the increased air-supply pressure, initial loop length, and running speed, the average yarn velocity increases. Increased linear density and twist of the yarn increase the insertion time. Open-end spun yarns and Murata air-jet spun yarns have higher yarn velocities than ring spun yarns. Textured yarns have higher velocities than the flat filament yarns.

Natarajan, Prasil, Egrt, and Hrus introduced a new air-jet weft insertion system (34), which introduced plain reeds and relay nozzles to replace the expensive profiled reeds. They used Pitot tube, Schlieren photography, and Laser Doppler Anemometer to test the airflow field variables such as air velocities and pressures. Constructional and setting parameters, for example, the diameter and length-diameter ratio $l/a$ of the nozzle holes, and the distance between adjacent nozzles, were studied.

They found that the diameter and the length-diameter ratio $l/a$ affect the air consumption and reliability of weft insertion. The distance between adjacent nozzles greatly affects the air consumption and overall economic viability. They concluded that the air velocity profile repeated itself along the weft-insertion axis. This new system’s air velocity along the weft-insertion axis was much less than conventional system. The vertical component of the air velocity near the jet axis was more than one third of its horizontal component.

**2.2.5 Applications in Nonwovens**

Airflow plays very important roles in nonwovens industry, for some cases, the filaments or fibers are in a melted status, such as in spun bond and melt blown process. This characteristic is very different from the status of filaments or fibers used in other conventional textile processes, for this reason the research on interaction between airflow
and fiber/filament in nonwovens process is not discussed here. It is believed to be very complicated because during this process, the air drag not only moves the filament/fiber, but also causes changes of internal structure and physical properties of filament/fiber, such as orientation and crystallinity, because of the high tensile stress, high tensile strain rate, and very high heat transfer rate at the hot surface of melted filament/fiber in these processes.

2.3 Studies on airflow acting on textile fiber / filament

Before the establishment of an appropriate computer model, the basic relationship between fiber / filament and airflow is needed to be determined by experiments.

Aerodynamic forces acting on a body is usually described by establishing the mathematical relationship between the Reynolds number ($Re$) and drag coefficient ($C_D$), Reynolds number is defined as:

$$Re = \frac{UL}{v}$$

$U$: Relative velocity between air and the body

$L$: Characteristic dimension of the body

$v$: Kinematic viscosity of air

For a sphere or an infinite long cylinder that is placed in a plane that is normal to the direction of the airflow, the drag coefficient is usually defined as:

$$C_D = \frac{D}{\frac{1}{2} \rho U^2 A}$$

$D$: Drag force on the body.

$\rho$: Body density
$U$: Relative velocity between air and the body

$A$: the area of the body projected to a plan that is normal to the direction of the airflow

For a fiber/filament that is placed parallel to airflow, the air drag acting on it can usually be considered as being dominated by frictional force, and then the drag coefficient can be derived from the frictional drag coefficient ($C_f$):

$$C_f = \frac{\tau}{\frac{1}{2} \rho U^2}$$

$\tau$: Shear stress

$\rho$: Body density

$U$: Relative velocity between air and the body

Gould and Smith used wind tunnel to measure the air-drag on filaments and yarns with the radius Reynolds number ranging from 10 to 200\textsuperscript{(35)}. The filament’s diameter used in experiment was less than 300 $\mu$m. A function of radius Reynolds number and drag coefficient was derived, it is expressed as:

$$C_D = 0.27Re_a^{-0.64}$$

$$Re_a = \frac{Ua}{\nu}$$

$$C_D = \frac{D}{\frac{1}{2} \rho U^2 \pi 2aL}$$

$a$: Fiber/filament radius

$L$: Fiber/filament length immersed in airflow
For $Re_a < 10$, the predictions of Glauert and Lighthill for laminar axisymmetric flow is very close to the experiment results $^{(36)}$, and the relationship between $\log_{10}(\nu x / Ua^2)$ and $\log_{10}(F/\mu U)$ was given in their solutions, in which $x$ is the axial distance along a long cylinder, $F$ is frictional force per unit length, and $\mu$ is the viscosity. These solutions were approximated by White as $^{(37)}$:

$$C_D \approx \frac{4}{Re_a} \left(1 / G + 1.5772 / G^2 + \ldots\right)$$

$$G = \ln \left(4 \frac{Re_L}{Re_a^2}\right)$$

$$C_f \approx \frac{4}{Re_a} \left(1 / G + 0.5772 / G^2 + \ldots\right)$$

$$G = \ln \left(4 \frac{Re_x}{Re_a^2}\right)$$

$$Re_a = \frac{Ua}{v}$$

$$Re_L = \frac{UL}{v}$$

$$Re_x = \frac{Ux}{v}$$

$a$: Fiber/filament radius

$L$: Fiber/filament length immersed in airflow

$x$: the axial distance along the fiber / filament

For $Re_a > 200$, White carried out a theoretical analysis for fully developed axisymmetric turbulent flow $^{(37)}$. The following equations were recommended in his paper:

$$C_D \approx 0.0015 + \left[0.30 + 0.015(L/a)^{0.4}\right] Re_L^{-1/3}$$
\[ C_f \approx 0.0015 + [0.20 + 0.016(x/a)^{0.4}] Re_{\ell}^{-1/3} \]

In order to decide the lateral aerodynamic force acting on the fiber, the drag coefficient in the direction transverse to the fiber/filament axis is needed. If the fiber/filament is considered to be made up of a series of spheres (the idea was initially used by Yamamoto and Matsuoka\(^{(38)}\)), the drag coefficient of a sphere is needed to calculate the aerodynamic force. Streeter and Wylie listed the experimental results of these drag coefficients\(^{(39)}\).

### 2.4 A Overview of Methods Used for Fiber-Airflow Interaction Modeling

Considering the complexity of interactions between fibers and airflows, currently available models simplify these interaction to some degree. According to different methods used in the modeling, they can be categorized to several approaches, with the following representative examples.

The first approach can be seen as the separated study of fiber/yarn configuration and airflow. The interaction between fiber and airflow was not coupled by mathematical means. The airflow field was calculated with theories of one-dimensional aerodynamics. An example of this approach is the research on air-jet texturing nozzles carried out by Acar et al, which was discussed in section 2.1.3.

The second approach can be seen as airflow parameters calculated using one-dimensional aerodynamics and the filament/fiber is considered with fixed configurations to some degree, which will simplify the coupling of air drag and fiber/filament movement. Then the interaction between the airflow and fiber/filament was calculated with the above results. An example of this approach is the research on yarn motion in air-jet filling insertion carried out by Adanur and Mohamed, which was discussed in section 2.1.4.
The third approach can be seen, along as the development of computer science, as trying to model the fiber configuration in airflow field in a more detailed way. This means the introduction of the airflow field from a two-dimensional estimation, or a two-dimension airflow field from a real configuration of an aerodynamic component of textile machinery, and the definition of air drag on fiber from relatively more flexible and accurate ways.

This approach includes research carried out by Smith, and Robert in a general study of fiber movements in an assumed convergent channel\(^{(40)}\), and by Kong, and Platfoot in rotor spinning\(^{(41, 42)}\). In both of their models, the airflow fields were two-dimensional, which is not as uniform as one-dimensional cases, and differential equations were built to describe the fiber movements at different time steps.

In Smith and Robert’s research, they calculated the velocity distribution of airflow field inside a convergent duct using the following analytical equation:

\[
v_f = \frac{|Q|}{\rho_a \alpha \sqrt{(x-x_d)^2 + y^2}}
\]

in which \(Q\) is the mass flow rate of air, \(\rho_a\) is the air density, \(\alpha\) is the wedge angle of that duct.

In that model the fiber was considered as many rods connected together, as shown in the figure 2.12.

They established differential equations describe the fiber movement inside the duct. The fiber movement was modeled based on above configuration. The equations were coupled with the result from the equation used to calculate the airflow speed mentioned above. Discrete equations were solved by considering the fiber section as cylindrical rod,
and using corresponding equations to calculate the air drag acting on the rods. Fiber configurations during the movement were calculated and results for achieving a better effect of fiber straightening were discussed.

![Figure 2.12](image)

Figure 2.12  Fiber configuration used in Smith and Robert’s research

Kong and Platfoot studied the fiber movement in fiber transfer channel using a two-dimensional configuration. They applied commercial CFD software, FIDAP, to calculate the airflow field in the channel, and evaluate the channel’s performance by studying the velocity distributions in different configurations.

![Figure 2.13](image)

Figure 2.13  Contours of streamlines and velocity distribution

Kong and Platfoot coupled the fiber into the airflow field by using a fiber configuration that is shown in Figure 2.14 The air drag was calculated by an assumed rod (in the figure, from $i-1$ to $i$ and from $i$ to $i+1$). Movements of the fiber were solved by calculating the movements of the nodes (in the figure, node $i-1$, $i$, and $i+1$).
Figure 2.14  Fiber configuration used in Kong and Flatfoot’s research\textsuperscript{(42)}
3. COMPUTATION OF AIRFLOW FIELD

3.1 Introduction

Computational fluid dynamics (CFD) played a very important role in this research. In this chapter, the utilization of CFD software and its functions are discussed and its process flow is presented.

“Computational Fluid Dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation”\(^{(43)}\). During the early years, fluid mechanics only had two branches: theoretical and experimental. Because of the mathematics difficulties, theoretical fluid mechanics can only solve simple problems. Experimental fluid mechanics, on the other hand, is relatively expensive, and it is often difficult to observe details in some flow fields. Computational fluid dynamics, which has been developing very fast since 1970’s, provides a tool that can simulate flow phenomena in some flow fields that cannot be solved by theoretical fluid mechanics and are very difficult to be observed by experimental fluid mechanics. The flow field simulations in the present research were solved by using CFD technology.

Different pneumatic components in textile machinery have different shapes and dimensions, for example, the airflow fields inside nozzles for air-jet texturing, spinning chambers for vortex spinning, or transfer channels for rotor spinning box, have very different flow pattern.

In addition, airflow parameters of these airflow fields are varied in a wide range, like the air pressure of an air-jet texturing nozzle.
Considering above facts, developing computer programs to calculate the airflow field for every single case is impracticable. Fortunately, there is computational fluid dynamics (CFD) software available now. Such as:

1. CFD-ACE+
2. FLUENT
3. ANSYS/FLOTRAN

In general, the difference between them is that first two packages use finite-volume method and the ANSYS/FLOTRAN use finite-element method to solve the flow field. The following criteria were considered for selecting the software:

1. The CFD software selected in this research should be able to compute the airflow field in the pneumatic components of textile machinery.
2. Results from the software should be accurate and stable.
3. It should have integrated pre-and pose-processing packages or functions.
4. Software costs should be acceptable according to project budgets.
5. Customer support and software upgrade.

Based on above considerations, CFD-ACE+ was selected from the several CFD software packages. CFD-ACE+ version 5.0 was initially purchased; it was upgraded, through several versions, to version 2002.

3.2 CFD-ACE+

The CFD-ACE+ software consists of several components, mainly CFD-GEOM, CFD-GUI, CFD-ACE, and CFD-VIEW. The general process flow of these components is shown in Figure 3.1.
3.2.1 CFD-GEOM

The first part of this software is the CFD-GEOM as a pre-processor. This component is used for solid model construction and grid generation, mainly it is capable of structured grid generation and unstructured grid generation. In addition, the CAD software, such as Pro/Engineer or AutoCAD, is usually used to construct solid model of more complex components. The model can then be imported to CFD-GEOM in IGES or DXF format. The grid of the flow field, which is generated in CFD-GEOM, is based on this solid model after faces, edges, and blocks that contain the airflow field have been specified, part of the boundary conditions can be setup before the model is put into the CFD solver (Figure 3.2 and Figure 3.3).

3.2.2 CFD-ACE and CDF-GUI

The second part of this software is CFD-ACE, integrated with a graphic user interface CDF-GUI. Boundary conditions, initial conditions, and physical properties of the fluid in flow field (volume conditions) are set up through CFD-GUI and the solver (CFD-ACE) uses them to calculate the airflow field (Figure 3.4).

CFD-ACE is a general purpose CFD solver, which utilized the pressure-based finite volume method to solve the flow field. It uses the Semi-Implicit for Pressure-Linked Equations Consistent (SIMPLEC) algorithm for pressure correction, and Conjugate
Gradient Squared (CGS) method and Algebraic MultiGrid (AMG) method for solving the equations obtained.

Figure 3.2 Physical model imported to CFD-GEOM

Figure 3.3 Grid generated by the CFD software

Before putting the model into computation, control parameters of this solver, like spatial differencing methods, relaxation, etc., must be specified, and they are very
important for getting results with high accuracy, along with grids generated for computing the flow field.

This software package provides several models for simulations, such as heat transfer and chemistry, etc. In this research, only the flow model was utilized, mainly for simulating velocity field and pressure field.

The limitation of this module is that it is not suitable for simulating higher supersonic flows which have Mach numbers higher than 2, because pressure-based method is not ideal for them. In this research, the airflow speeds were not higher than this limitation.

![Figure 3.4 Calculating of the airflow field](image)

### 3.2.3 CFD-VIEW

The third part is the post-processor, CFD-VIEW. This component is an interactive program for post-processing the CFD results, for example, airflow field visualization, data probation, and many other functions (Figure 3.5).
3.3 CFD Results Processing and Integration with Fiber Movement Model

Data points calculated by the software are contained in the DTF (Data Transfer Format) files. Information of these data points is exported to the computer program that simulates the fiber movement in this airflow field. This program, which is not a part of the CFD software, is based on the model that describes fiber movement in airflow field. It has developed specific codes to import and locate (or interpolate, if necessary) the data from DTF files. An algorithm ensured that, during every time step, every position of key points of the fiber that is modeled is mapped into the airflow field, and the flow field parameters surround them are known. So, the one-way coupling between the fibers and airflow field can be achieved by establishing the dynamic equations.

One important point is that in this research one-way coupling between fiber and the airflow field is considered, which means that while there is air drag acting on every fiber, the existence of fibers will not interfere with the airflow field.
The reasons for choosing one-way coupling to implement the fiber-airflow interactions are:

1) If the traditional two-phase flow approach is used to solve the two-way coupling, the fiber need be treated as continuum. It will be impossible to study the movement and configuration of a single fiber, and obviously, this assumption is far from the real situation, for example, to describe fiber movement inside fiber transfer channel of rotor spinning box, there are not many fibers in the channel at the same time.

2) Theoretically, fibers’ interference to the airflow field at a particular position can be calculated, but the generation of the grid will be impracticable if we consider the dimension of a single fiber, the whole airflow field and the whole movement of these fibers, which means during each time step when fibers are moving, the airflow field needs to be re-calculated.
4. AERODYNAMICS ANALYSIS OF PHYSICAL MODELS

4.1 Introduction

Playing a key roll among all the parts of a rotor-spinning box, the fiber transfer channel has already drawn the interest of many researchers (see the literature review in Section 2.2.1). In the present research, the rotor-spinning fiber transfer channel was studied by CFD technique. The fiber motion inside this channel is a typical example about interactions between fibers and high-speed airflows in textile procedures. This was also an exploration and demonstration of the potential and prospect of applying CFD in research and development of textile technologies.

In this chapter, a three-dimensional computer model of the fiber transfer channel of a rotor-spinning box was constructed, its geometrical parameters were analyzed, and the optimization of airflow field parameters was discussed. Before applying it to the CFD calculation, a simplified aerodynamics analysis was carried out in order to discuss the basic trends of a fiber transfer channel.

The CFD software, which was discussed in the previous Chapter 3, was used in the construction and computation of the flow field (with the aid of CAD software as AutoCAD).

The dimensions of the channel were specified using a channel configuration described in research papers published by Lawrence and Chen\textsuperscript{45, 46}, along with comparison with commonly used industrial products as references. A part of parameters was varied to construct different channel shapes in order to compare these shapes and look for an optimization.
Initial conditions of the airflow field of the fiber transfer channel, which are mostly decided by the operation pressure and rotation speed of opening roller, were also be specified by commonly used industrial parameters. Different initial conditions were applied to the model, in order to compare the performance of the channels under different working conditions.

The rotor-spinning box described in this research has a functional configuration as depicted in Figure 4.1.

![Figure 4.1. The layout of a rotor-spinning box](image)


**4.2 Geometrical Configurations and Dimensions**

The fiber transfer channel of a rotor-spinning box commonly has a convergent flow passage shape; this is because it is considered that this configuration has an aerodynamic
characteristic that can satisfy the requirement of fiber movement during this procedure, in which fibers from the opening roller can be accelerated and straightened before they reach the rotor groove.

The configurations and dimensions of the fiber transfer channel studied in this chapter were firstly adopted from Lawrence and Chen’s research\(^{45}\), in which it was claimed as an optimized fiber transfer channel. This channel was constructed as a framework in AutoCAD, Consequent work for grid generation was finished in CFD-GEOM, the engineering drawing in Figure 4.2 shows its shape and dimensions.

4.3 Boundary and Initial Conditions for CFD

1. Boundary conditions:

   The generated grid and number of faces are shown in Figure 4.3. Since the airflow in these convergent shape channels can only be subsonic or sonic, so only two parameters, air density and velocity, were specified at the inlet as required by the software package, and only back pressure was specified at the outlet.

   Face 1: Inlet of air and fibers. Air density and velocity were specified.

   Face 2 to Face 5: Solid walls of the fiber transfer channel. On these walls, no-slip wall conditions are used.

   Face 6: Outlet of air and fiber. Air back pressure was specified.

2. Initial conditions:

   Initial volume conditions inside the channel were specified before the model was calculated.

   Initially the airflow field was calculated by using laminar flow model.
Figure 4.2. Structure of the fiber transfer channel simulated with CFD software
Figure 4.3. Fiber transfer channel CFD model grid generated by software package and boundary conditions

- Face 1: Inlet of air and fiber
- Face 2 ~ 5: Solid walls of the fiber transfer channel
- Face 6: Outlet of air and fiber
4.3 Analysis based on One-dimensional Aerodynamics Estimation

Even though CFD technology provides a powerful tool for studying the airflow field characteristics of the fiber transfer channel discussed in this chapter, a reasonable estimation could be very helpful in providing a basic understanding on the channel’s structure and working parameters. So firstly, estimation on the fiber transfer channel’s working conditions at its outlet was carried out based on the consideration that airflow in the channel can be assumed as pipe flow. The estimation was based on one-dimensional steady flow. Then the affects from friction and three-dimensional flow were discussed. The airflow field is since simplified as shown in Figure 4.4. Airflow with static pressure $P_1$ and speed $V_1$ enters the channel at the inlet cross-section $I-I'$, then exits the channel at outlet cross-section $2-2'$, with static pressure $P_2$, and speed $V_2$.

![Figure 4.4. Simplified fiber transfer channel for estimation](image)

According to the one-dimensional pipe flow theory, subsonic airflow entering the convergent channel at inlet $I-I'$ is accelerated to a speed which has a maximum Mach number $Ma$ of one. Assuming the back pressure at the channel outlet was $P_b$, if the airflow is under sub-critical status or critical status at the outlet cross-section, the ratio between $P_b$ and outlet total pressure $P_2^*$ should satisfy:
\[
\frac{P_b}{P_2} \geq \beta_{cr}
\]  \hspace{1cm} (4.1)

\(\beta_{cr}\) is the critical pressure ratio, \(\beta_{cr} = \left(\frac{2}{k+1}\right)^{\frac{k}{k-1}}\), \(k\) is the specific heat ratio, for air, \(k = 1.4\), so \(\beta_{cr} = 0.528\).

If the airflow inside the channel is considered as adiabatic and isentropic, \(P_2^*\) then equals to the inlet total pressure \(P_1^*\), in addition, the work from the opening roller was neglected because the total pressure gained from its action to accelerate the air was relatively low. So, at the inlet opening of the fiber transfer channel, the inlet total pressure can be considered as the atmosphere static pressure \(P_{atm}\), which equals to \(1.0133 \times 10^5\) Pa. In order to satisfy the inequality (4.1), back pressure \(P_b\) at the outlet should be:

\[
P_b \geq \beta_{cr} P_2^* = 0.528 \times 1.0133 \times 10^5 \text{ Pa}
\]

\[
= 5.3502 \times 10^4 \text{ Pa}
\]

\(P_{cr}\) is the critical pressure at the outlet cross-section.

Therefore, if the back pressure at the outlet is maintained higher than \(5.3502 \times 10^4\) Pa, the channel is under sub-critical status. This also means that if the maximum air speed at the outlet cross-section, which equals to sonic speed, is wanted to be reached, the back pressure \(P_b\) should lower than the critical pressure \(P_{cr} = 5.3502 \times 10^4\) Pa.

Under real working conditions, the channel has a status that is not close to the result predicted by above estimations. The reason is that the airflow inside a real fiber transfer channel cannot be calculated as a one-dimensional adiabatic, isentropic pipe flow. The average air speed at the outlet cross section is much lower than sonic speed even if the
back pressure reached the above calculated critical pressure. There are mainly two factors that are contributing to the difference between the real conditions and the above estimation:

1) Friction between the airflow and the channel wall. Friction causes the dissipation of the airflow’s kinetic energy. The boundary layer’s existence causes the flow rate at outlet to be lower than the idealized conditions, which means that the average speed is lower.

2) Effects coming from three-dimensional airflow. Real fiber transfer channels have complex structures and the airflow inside the channel is three-dimensional. In addition, airflow at the inlet is not parallel to the center line of the channel. It is thus difficult to keep the airflow parallel to the channel center line so it can be considered as one-dimensional flow. The airflow at the outlet cross-section cannot reach the critical status if the back pressure $P_b$ is equals to $P_{cr}$, a lower back pressure is still able to increase the average air speed at the outlet cross-section.

Once the air at the outlet cross-section reached its critical status, the airflow speed equals the local sonic speed, and won’t increase as the back pressure decreases.

Results from CFD were analyzed in the following sections. It can be seen that, from a three-dimensional computation, the working effectiveness of the channel is worse than the results estimated with one-dimensional aerodynamic theories. This can be explained by the possibility that the frictional force was underestimated.

**4.4 Outlet Back Pressure**

From the above analysis, when the channel outlet cross-section was working under sub-critical conditions, a lower outlet back pressure resulted in high airflow speed at the
outlet. CFD calculations showed the same tendency. But if the channel outlet cross-section was working under critical or super-critical conditions, lower outlet back pressure cannot increase the airflow speed at the outlet. In this case, this means that more energy was required to keep the system under the pressure, but without increasing the outlet airflow speed and airflow rate.

4.5 Channel Outlet Size

The size of the channel outlet has to be a compromise between the rotor size and the aerodynamic efficiency of the channel.

The first, the outlet size is limited by the structure and size of the rotor groove.

The second issue was actually reflected by the ratio between the inlet and outlet size

For isentropic pipe flow, there is:

\[
\frac{\rho^*}{\rho} = \left(1 + \frac{k-1}{2} Ma^2 \right)^{\frac{1}{k-1}}
\]  \hspace{1cm} (4.2)

\(\rho\) and \(\rho^*\) are air density and total density respectively. From the above equations, if \(Ma = 0.3\), \(\rho \approx 0.96 \rho^*\), it is clear that when the Mach number \(Ma\) is low, the air density can be deemed as incompressible.

From the continuity equation:

\[
\rho_1 A_1 V_1 = \rho_2 A_2 V_2
\]  \hspace{1cm} (4.3)

If the Mach number is low, the outlet air speed decreases as the outlet cross-section area increases.

The one-dimensional aerodynamics analysis in the previous section showed that under commonly used industrial conditions, even if the back pressure \(P_b\) at the outlet equals to the critical pressure \(P_{cr}\), the outlet cross-section is under sub-critical status, the
air speed at the outlet is not supersonic. The following CFD results also revealed that the
air speed at the outlet is much lower than the sonic speed. This means under current
working conditions, a channel with a larger outlet size results in lower outlet air speed.
Obviously, this is going to cause drawbacks to the rotor spinning process:

1) Larger fiber transfer channel outlet size causes the problem of how to fit the
   structure and size of rotor grooves.

2) Lower outlet air speed causes lower fiber speed, and has an important impact on
   the fiber orientation. This is discussed in the following chapters.

3) Lower fiber speed means fewer fiber transfer rate, which affects the whole
   system’s productivity of yarn.

On the other hand, the channel outlet size also cannot be too small. Because currently
the fiber transfer channel designs in industry have a convergent profile, assuming the
initial conditions, it is impossible for the airflow to reach a supersonic speed inside the
channel. Therefore, under the same initial conditions and boundary conditions, an
improperly reduced outlet size will possibly cause the channel to choke, but offers no
help in improving the fiber’s speed and orientation.

The study on outlet size was actually a study on the ratio of the outlet and the inlet
size. The inlet size is a compromise with the structure and size of the opening roller; the
discussion above about the outlet size was based on the pre-condition that the inlet
remained unchanged, then this can satisfy the requirement for the one-dimensional
aerodynamic estimation carried out above.
4.6 Opening Roller’s Influence to Airflow at the Channel Inlet

The opening roller inside the rotor spinning box has very compact structure, its teeth are fit very close to the inner wall of the box, so the airflow speed at the channel inlet cross-section may be specified as the peripheral speed of the opening roller. The direction of this speed maybe not parallel to the center line of the fiber transfer channel.

For the same rotor-spinning box structure, Estimations from one-dimensional aerodynamic theories in previous sections and calculated CFD cases showed that, if the Mach number is not high so air can be considered uncompressible, the airflow speed at the outlet cross-section increases as the inlet airflow speed increases. The inlet airflow velocity is also an important factor that causes the re-circulation zone at the beginning of the channel. A higher inlet speed with an improper angle may cause a re-circulation zone across a larger region; this is discussed in the next section. Since the rotation speed of the opening roller has a direct influence on the inlet airflow speed, it is an important factor that affects the airflow speed at the outlet cross-section. In addition, a very high rotation speed may damage fibers.

4.7 Re-circulation Zone

The velocity at the channel inlet may not be parallel to the center line of the channel, but inclined an angle to the tangent line at the point at which the fiber transfer channel was connected to the opening roller. If the initial speed at the inlet was high and the angle with the tangent direction was increased to a certain degree, re-circulation zones will occur at the beginning of the channel. For most of cases, the re-circulation zone plays a negative role for improving fiber orientation and speed. These are discussed in next chapters.
The formation of re-circulation zone has three important contributing factors:

1) The channel’s geometrical structure. Because of the three-dimensional structure of the channel, the re-circulation zone may occur at two directions.

2) Airflow speed at the channel inlet. Lower speed at the inlet is less likely to cause the re-circulation zone.

3) Back pressure at the channel outlet. Lower back pressure is less likely to cause the re-circulation zone.

These are discussed in the following sections along with the CFD results.

**4.8 CFD Results Analysis**

The model used for CFD simulation is shown in Figure 4.2. Inside the model, working media is air. airflow simulated before entering the channel has parameters initially specified as the boundary conditions and initial conditions that were as discussed in section 4.3.

Working temperature was set as 300K. Airflow speed at inlet was set as $V_x = V_y = 20.14$ m/s, $V_z = 0$ m/s, This approximated an opening roller rotation speed of 8500 rpm, directions of x, y, z are shown in Figure 4.2, right-hand convention is used. The back pressure at the outlet was specified as 80 mbar since this vacuum is commonly used in industry. Since a value was required for calculating the inlet density by the software package, and for conducting a simulation that is more similar to the real machine operation, the boundary pressure was initialized as 1 atm at the channel inlet (calculated inlet pressures will be shown according to solutions).

The above method was implemented to specify the boundary conditions and initial conditions during the simulations used to study the effects from outlet size, inlet airflow
speed, and outlet back pressure. These results are presented in the following sections. Because, for most textiles processes except nonwovens, the velocity field of the airflow is considered as the decisive factor that affects the fibers’ configuration and movement in that airflow field, in the simulations velocity distributions were paid more attentions. The existence of re-circulation zone in these airflow fields and their characteristics and contributing factors were also studied.

4.8.1. CFD Study on Channel Outlet Size

In these simulations, the outlet width, shown in Figure 4.2 as W, was varied from 4 mm to 14 mm; other initial conditions and boundary conditions were as same as the conditions discussed in above section. Six models were calculated: W = 4mm, 6mm, 8mm, 10mm, 12mm, 14mm. Wide outlet sizes, like 12X4 and 14X4 mm$^2$, are actually not practical in industry. These wider sizes were used in this research because they can serve as evidence that evaluated the effects of the ratio of inlet size and outlet size of the channel, on other words, convergent angle (half angle) of the channel. They could be used to study the case that if the inlet size and outlet size are both small. Table 4.1 shows the outlet size parameters.

From Figure 4.5 to Figure 4.10$^1$, which plots out the velocity distributions along the center lines of channels, the CFD results clearly show that airflow was being accelerated in those channels that had a convergent structure.

$^1$ In all the figures that show the airflow velocity distributions from CFD calculations, since the legends of these data were automatically generated by the CFD software package, they are different from the symbols used in this dissertation. In these figures, U stands for the airflow velocity in x direction ($V_x$); V stands for the velocity in y direction ($V_y$); W stands the velocity in z direction ($V_z$). x, y, z directions are defined in Figure 4.2. Right hand convention is used.
### Table 4.1. Channel outlet size and convergent angle

<table>
<thead>
<tr>
<th>Outlet Size (mm²)</th>
<th>4X4</th>
<th>6X4</th>
<th>8X4</th>
<th>10X4</th>
<th>12X4</th>
<th>14X4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Convergent angle (°)</td>
<td>7.1</td>
<td>6.4</td>
<td>5.7</td>
<td>5.1</td>
<td>4.4</td>
<td>3.7</td>
</tr>
</tbody>
</table>

Those results show that under the same specified initial conditions and boundary conditions, the speed along the center line reached its maximum at the outlet cross-section; and as the outlet width W increased, this maximum speed decreased. This maximum speed along the channel center line ranged from 93.53 m/s (outlet size 4X4 mm²) to 21.13 m/s (outlet size 14X4 mm²). This is the same trend as the estimation in Section 4.5 predicted.

It also can be seen that, especially in Figure 4.15, in which the highest outlet airflow speed was observed in the 4X4 mm² outlet, the acceleration of airflow was done mostly in the second half of the channel. One of the reasons is believed to be that there was re-circulation zone in the first half of the channel; airflow velocity inside a re-circulation zone could be in the opposite direction compared with the main stream. Another reason could be that at the beginning of the channel, airflow was not flowing into the channel parallel to its line of the channel, but at an angle. Compared with this configuration of inlet airflow velocity, the airflow at the second half of the channel had a velocity that was more parallel to the channel center line.

Figure 4.11 showed the decrease of the speed along with the outlet size (area), this relationship was not linear. Figure 4.12 showed distributions of the y-axis component of the velocity along the z-axis direction center line at the outlet cross-section (the x-coordinate of that figure was the number of grid nodes, ranged from 1 to 26, along this center line, not the chord length).
Figure 4.5. Airflow velocity distribution along the center line (4x4 mm$^2$ outlet)

Figure 4.6. Airflow velocity distribution along the center line (6x4 mm$^2$ outlet)
Figure 4.7. Airflow velocity distribution along the center line (8x4 mm$^2$ outlet)

Figure 4.8. Airflow velocity distribution along the center line (10x4 mm$^2$ outlet)
Figure 4.9. Airflow velocity distribution along the center line (12x4 mm$^2$ outlet)

Figure 4.10. Airflow velocity distribution along the center line (14x4 mm$^2$ outlet)
Figure 4.11. Change of center line maximum speed with outlet size

Figure 4.12. Speed distributions along the Z direction center lines at outlet cross-sections
Figure 4.13. Speed contours of $V_y$ (Velocity at Y-axis direction). (a) is a contour of the plane cutting through the channel center line in Z-axis direction; (b) is a contour of the plane cutting through the channel center line in X-axis direction. Outlet size: 4X4 mm$^2$. Outlet back pressure: 80 mbar. Inlet airflow velocity: $V_x = V_y = 20.14$ m/s, $V_z = 0$ m/s.
From these airflow velocity distributions of channels, it can be observed that a channel with smaller ratio of inlet/outlet size (smaller convergent angle) was less efficient in accelerating the air flowing through it.

In addition, it is necessary to point out that, because of the three-dimensional structure of the channels studied, the maximum speed along the channel center line did not mean that it was the maximum speed in the whole airflow field; some place at the outlet cross-section may have a higher speed. The velocity distribution in Figure 4.12 indicated that the maximum speed of the outlet cross-section was not on the center line of that channel, air that was closer to the wall (Face 2 and Face 3, see Figure 4.3) had a higher acceleration. This can also be seen from the Figure 4.13(a), which shows the velocity contour taken from a plane cutting through the channel’s center line at Z-axis direction, the speed near the convergent wall (but outside the boundary layer) was higher.

This could be explained as that, suppose there is a stream tube that has a convergent angle and has the channel’s center line as its own center line. Such a stream tube with a larger convergent angle has a higher average speed than a stream tube with a smaller convergent angle, that means airflow that is more distant from the center line has a higher speed.

4.8.2. CFD Study on Channel Inlet Airflow Velocity

If other conditions were the same, the opening roller rotation speed will have an important influence on inlet airflow velocity. This influence to the airflow field was studied with a connection to the opening roller rotation speed. Since the distance between the fiber transfer channel’s inlet and the opening roller is very small, considering the non-slip wall condition, the opening roller’s rotation speeds were used to simulate the inlet
airflow speeds. The opening roller diameter was specified as 64 mm, which is common in industry. Nominal rotation speeds from 6500 rpm to 8500 rpm were used in the simulation; channel outlet size was specified as 4X4 mm²; other conditions were the same as those of the previous section. Table 4.2 shows the input speeds.

<table>
<thead>
<tr>
<th>Rotation speed (rpm)</th>
<th>6500</th>
<th>7000</th>
<th>7500</th>
<th>8000</th>
<th>8500</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V_x$ (m/s)</td>
<td>15.40</td>
<td>16.59</td>
<td>17.77</td>
<td>18.96</td>
<td>20.14</td>
</tr>
<tr>
<td>$V_y$ (m/s)</td>
<td>15.40</td>
<td>16.59</td>
<td>17.77</td>
<td>18.96</td>
<td>20.14</td>
</tr>
<tr>
<td>$V_z$ (m/s)</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Table 4.2. Inlet airflow velocity

Figure 4.14 ~ Figure 4.17 showed the velocity distributions along the center lines of channel, the distribution of 8500 rpm was shown in Figure 4.5. In these simulations, the maximum speeds along the channel center line all were reached at the channel’s outlet cross-section, from 72.53 m/s (according to nominal rotation speed of 6500 rpm) to 93.53 m/s (according to nominal rotation speed of 8500 rpm). The maximum speed increased when the inlet airflow speed increased, as shown in Figure 4.18, this was coincident as what the estimation predicted in Section 4.6.

From Figure 4.18 it also could be seen that the relationship between the inlet airflow speed and the outlet airflow speed at the center line was close to linear, but there is no reason to confirm or deny that they have a linear relationship.

Figure 4.19 showed the speed distributions along the Z-axis direction center line at the outlet cross-section. Compared with Figure 4.12, in which it can be seen that larger outlet size resulted a speed distribution profile that is not as even as smaller outlet size did, these distributions in Figure 4.19 had similar profiles.
Figure 4.14. Airflow velocity distribution along the center line (6500 rpm)

Figure 4.15. Airflow velocity distribution along the center line (7000 rpm)
Figure 4.16. Airflow velocity distribution along the center line (7500 rpm)

Figure 4.17. Airflow velocity distribution along the center line (8000 rpm)
Figure 4.18. Change of center line maximum speed with inlet airflow speed

Figure 4.19. Speed distributions along the Z direction center lines at outlet cross-sections
4.8.3. CFD Study on Channel Outlet Back Pressure

Outlet back pressure has a very important influence on the channel’s airflow field, as discussed in Section 4.4. In this section, the effect of outlet back pressure was simulated for the values shown in Table 4.3. Outlet size was 4X4 mm$^2$, other conditions were the same as those in Section 4.8.1.

<table>
<thead>
<tr>
<th>Outlet back pressure (mbar)</th>
<th>60</th>
<th>70</th>
<th>80</th>
<th>90</th>
<th>100</th>
</tr>
</thead>
</table>

Table 4.3. Outlet back pressure

Figure 4.20 to Figure 4.23 showed that the airflow speed distribution along the channel center line, the distribution of 80 mbar was shown in Figure 4.5. Under all these back pressures, the speed was increasing along the center line, and reached the maximum at the outlet, lower back pressure resulted in higher maximum outlet speed (see Figure 4.24). Moreover, Figure 4.25 showed that the average speed at the outlet was also higher with a lower outlet back pressure. The simulation results were coincident with the estimation of sub-critical status in Section 4.4. From these results and discussions in Section 4.4, it was indicated that in above outlet pressure range (60 mbar ~ 100 mbar), the simulated channel outlet did not reach the critical status and was not choke, that means it is still possible to research a higher outlet speed by decreasing the outlet back pressure. There is a report of experimental research that had reached a negative pressure of 30 mbar\(^{(48)}\).

Since the energy used to maintain the rotor spinning machine’s vacuum counts for 25% ~ 40% of the overall energy consumption, selections of a proper back pressure could be very helpful for improving the spinning system’s economics. In addition, the control and regulation of pressure has an important impact on the resulted yarns’ qualities\(^{(48)}\).
Figure 4.20. Airflow velocity distribution along the center line (60 mbar)

Figure 4.21. Airflow velocity distribution along the center line (70 mbar)
Figure 4.22. Airflow velocity distribution along the center line (90 mbar)

Figure 4.23. Airflow velocity distribution along the center line (100 mbar)
Figure 4.24. Change of center line maximum speed with outlet back pressure

Figure 4.25. Speed distributions along the $Z$ direction center lines at outlet cross-sections
4.8.4. CFD Study on Re-circulation Zone inside Channel

In the simulations, the existence of re-circulation zones inside the channel was observed. Re-circulation zone’s structure, characteristics and reasons of its formation were investigated in this research; comparisons under different conditions were also studied.

These re-circulation zones have a three-dimensional structure, which is not possible to be observed by using two-dimensional simulations. As shown in Figure 4.26 (a) and (b) by one of streamlines calculated by CFD, airflow of this re-circulation zone not only circulated along the channel’s longitudinal direction (Figure 4.26(a)), but also at the channel’s lateral direction (Figure 4.26(b)).

![Figure 4.26](image-url)

Figure 4.26. Three-dimensional structure of a re-circulation zone

In these simulations, all re-circulation zones happened near the inlets of channel models calculated, similar to what demonstrated in Figure 4.26. Re-circulation zones stopped at some distance from the channel inlet, as the $V_y$ speed distributions along I-planes shown in Figure 4.27, (I-planes cut through channels from lateral side. They were

63
introduced when the solid models were put into grid generation and used for the computation of finite volume method. Most model simulated in this research had 86 I-planes.) In these distributions, there were regions that had velocities in the opposite direction of the main stream, shown in the figure as negative speed. These regions were smaller as they researched further distance from the channel inlet. At last, disappeared and merged into the main stream.

Figure 4.27. \( V_y \) speed distribution in I-planes near the channel inlet

Figure 4.28 showed the static pressure distribution along the channel’s center line (outlet size 4X4 mm\(^2\), outlet back pressure 80mbar, inlet speeds \( V_x = V_y = 20.14 \text{ m/s}, V_z = 0 \text{ m/s} \)), from the inlet the pressure actually increased for a distance, then decreased to the back pressure at the outlet. The pressure increases at the beginning of the inlet causing the airflow to slow down, once the airflow does not have enough energy to overcome the increasing pressure, it starts to move in the opposite direction to the main stream. Therefore, a re-circulation zone is formed.

64
Two factors were studied regarding their relationship with the re-circulation zone and they are inlet air speed and outlet back pressure.

During these simulations, geometrical parameters were kept the same, with outlet width $W = 4$ mm. Airflow speed at the $Y$ direction ($V_y$) was studied in different cases. Two center chord lines (perpendicular to the width $W$ direction, as shown in Figure 4.27) at two different $I$-planes: $I$-plane 7 and $I$-plane 19 were picked up to study the speed distribution along the chord length. Namely, these two center chord lines were called center chord line 7 and center chord line 19. Basically $I$-plane 7 was considered as well inside the re-circulation zone and $I$-plane 19 was considered as at or near the boundary of the re-circulation zone.

When the nominal opening roller rotation speed is increased from 6500 rpm to 8500 rpm, which means the increase of inlet airflow speed, with a fixed outlet back pressure 80
mbar, both Figure 4.29 and Figure 4.30 show that higher inlet airflow speed caused stronger re-circulation. This conclusion is based on the following observations:

The first, as shown in Figure 4.29, on center chord line 7, as the nominal rotation speed increases, the negative $V_y$ speed, which means the air is flowing at the opposite direction of main stream, decreased (or increased in magnitude). In that figure, the region with negative $V_y$ speed (the plots that is under 0 m/s) can be considered as inside a re-circulation zone because $V_x$ and $V_z$ do not have high values in these simulations.

The second, as shown in Figure 4.30, on center chord line 19, airflow is considered to be out of (or almost out of) the re-circulation zone. Results show that the lowest $V_y$ for 6500 rpm is 0.429 m/s, but for 8500 rpm, the lowest $V_y$ is –0.044 m/s, which means that higher inlet speed cause a re-circulation zone that could cover a larger space inside the airflow field.

![Figure 4.29. $V_y$ distribution along the center chord line 7](image-url)
Figure 4.30. $V_y$ distribution along the center chord line 19

CFD results show that outlet back pressure has an important influence on re-circulation zone’s configuration. In these simulations, initial conditions and boundary conditions are the same as conditions used in Section 4.8.3.

In Figure 4.31, it can be seen that along the center chord line 7, the higher the outlet back pressure, the stronger is the re-circulation zone. This can be evaluated by the negative $V_y$ value. The chord length that has a negative $V_y$ is longer when the outlet back pressure is higher, in addition, there is a general trend that the negative speed is going to be higher. Note that when back pressure reaches 100 mbar, the highest negative $V_y$ speed decreases a little, but from the figure it can be seen that under such a back pressure, the $V_y$ is negative along a much larger portion of the center chord line than it is under the other four outlet back pressure values. For the case of a 60 mbar back pressure, there is even no negative $V_y$ along the center chord line 7, this indicates that lower outlet back pressure can cause less stronger re-circulate zone, or even avoid it. It is reasonable that,
similar to the explanation to Figure 4.28, if the back pressure is not high, then there could not be enough to slow and force the airflow back.

Results presented in Figure 4.32, in which $V_y$ distribution along the center chord line are plotted according to different outlet back pressures from 60 mbar to 100 mbar, support the same conclusion as above. When the back pressure is 90 mbar and 100 mbar, part of this center chord line is still inside a re-circulation zone but it is not if under other back pressure like 60 mbar and 70 mbar. This means that under lower back pressure, there can be no re-circulation zone, or the re-circulation zone only covers a smaller space.

Figure 4.31. $V_y$ distribution along the center chord line
Figure 4.32. $V_y$ distribution along the center chord line 19
5. BASIC EQUATIONS OF FIBER MODELING

5.1 Introduction

For fiber movement simulations, in addition to the airflow fields that were solved by CFD as discussed in the previous chapter, it is also necessary to establish mathematical equations used to describe the fiber movement and configuration. For establishing such a set of equations that can finally be utilized in computer program codes to simulate the whole movement, it is necessary to:

1. Establish basic governing equations;
2. Determine the choices of specific models regarding fiber configurations for simulations;
3. Combine the basic equations and the fiber configuration models to create the equations for the simulation.

In the work described in this chapter, fibers’ bending and twisting stiffness were excluded in order to emphasize the fiber-airflow interaction. These properties are discussed in Chapter 7.

5.2 Basic Equations

Figure 5.1 represents the element, $ds$, of a fiber moving inside an airflow field. The basic governing equation of that element are written in Equation 5.1:

$$ \rho \ddot{s} ds = \vec{F}_s - \vec{F}_{surf} + \vec{F}_{body} $$

(5.1)

$\rho$: fiber linear density

$a$: element acceleration of speed

$F_s$: concentrated force
For fiber movements in airflow field by computer simulations, the concentrated forces $F_s$ and $F_{s+ds}$ on this element were considered as interactions from elements that connected to it; surface force $f_{surf}$ was air drag, and body forces $F_{body}$ were gravity and buoyancy acting on the fiber. Equations (5.1) can also be written as:

$$\rho \frac{\partial \vec{v}}{\partial t} = -\frac{\partial \vec{F}}{\partial s} + \vec{f}_{surf} + \vec{f}_{body}$$

in which $v$ is element velocity and $f_{body}$ is body force of unit length.

Figure 5.1. Forces acting on a fiber element

Theoretically, elements could be specified in any shape, but if they were represented by simple and uniform configurations, this could be helpful in solving the simulations in a more effective way, for example, if elements’ shape is consider as a function of $s$, then it would be very complicated problems in solving air drag acting on the fiber.

Studies on fiber crimp suggested parameters used to study the fiber crimp configurations\(^{(49, 50, 51)}\). One important parameter was the fiber Effective Wave Number (EWN), which was defined as the number of waves of unit length of a fiber, and it was used to describe the number of crimps per unit length\(^{(49)}\). In another research, it was
shown that the EWN of cotton fibers varied from 2.8 per centimeter to 4.6 per centimeter, and if there was a load at the end of the fiber, the EWN decreased as the load increased\(^{(51)}\).

A study on fiber crimp geometry used a concept that experimentally obtains fibers’ curvature and torsion by collecting data of positions of small segments that compose one fiber\(^{(52)}\). This concept of considering the fiber geometry as a series of “wave” or segment was also extended to studies on fibers’ mechanical properties. As the model used for studying on fiber crimp removal\(^{(53)}\), in which fibers were assumed to have a configuration as rods connected by springs.

In these cases, fibers were actually studied in configurations that had discrete forms; where “discrete form” means that a fiber is assumed composed of a series of small segments that have a fixed shape. In computer modeling, this is a very important assumption that will finally led to the formulation of discrete equations for computing fiber movements. One reason is that, theoretically, air drag acting on a fiber is a continuously distributed force along the fiber’s surface, but in computer modeling, it is practically impossible to establish such an equation that can be used to describe this continuous force; a more realistic method is to describe it in discrete form according to each fiber segment.

If fibers were modeled from discrete configurations, dimensions of these discrete segments could be specified by some estimation that utilizes results from studies on fiber crimp. A reasonable estimation would be that the modeled fiber has about the same, or more, segments than two times of the number of crimp waves, this could keep both the
accuracy that is needed to describe the fiber configurations and the efficiency of computations.

Actually, as the fiber becomes increasingly straight, like cases in which fiber-straightening effects inside a convergent channel were studied, the straightening procedure means that fiber’s crimp diameters and/or number of crimp waves was decreasing.

### 5.3 Model of Fiber Configuration

The selection of an applicable model to describe the fiber configurations is the key that links those governing equations discussed in previous section to computer simulations. In other words, it means that, in order to implement the governing equations, it is essential to choose the correct model for describing the fiber configurations.

An importance aspect that must be specified in the model was how to compute aerodynamic forces acting on the fiber. There are the following considerations:

1. Because the action between airflow and fiber studied was assumed as one-way coupling (see Section 3.3), during the progress of fiber moving through the airflow field, airflow velocity at a position is known for the corresponding space coordinates.

2. During the CFD process, the grid was generated in such a manner that the whole length $(ds)$ of a fiber segment actually can not be located inside any single cell of the airflow field calculated. Thus, it should be evident that different fiber segments will be under different aerodynamic forces.

3. In order to solve the model used for simulating fiber movements and configurations, it is essential to decide on the shape or/and the orientation of fiber
segments before the known airflow velocity can be used to calculate aerodynamic forces.

Based on above discussions, some applicable models were considered in the this research, the basic concepts of these methods were shown in Figure 5.2, the main configurations and characteristics of these methods were:

1) Spheres (Figure 5.2 a)
   - Spheres are connected with inflexible rods of zero diameters and zero mass.
   - The air drag only acts on the spheres.

2) Spheres + Rods (Figure 5.2 b)
   - Fiber mass is concentrated in the spheres
   - Air drag acts on the rods.

3) Rods (Figure 5.2 c)
   - Fiber mass is distributed evenly through the whole fiber.
   - Air drag acts on the rods.

**Figure 5.2. Different fiber modeling methods**

In above three models, the first one, in which the fiber were considered to be inextensible, was selected for simulations in this research. This method, compared with the other two, is more effective in computation and also capable of describe fiber movements in enough details. In addition, this model can be used to study fiber’s bending
and twisting characteristics in airflow under several assumptions, which are discussed in Chapter 7.

### 5.4 Fiber Movement Model

Regarding the model selected in previous section, to describe the movement of one single node of the fiber in the airflow field, related to Equation 5.1, the following dynamics equation can be used (see Figure 5.3, right-hand system was used):

\[
m\ddot{a}_i = \ddot{f}_{i-1,i} + \ddot{f}_{i+1,i} + \ddot{f}_a + \ddot{f}_b + \ddot{f}_g
\]

or

\[
m\ddot{a}_i = \ddot{f}_{i-1,i} + \ddot{f}_{i+1,i} + \ddot{f}_a + \ddot{f}_b + \ddot{f}_g
\]  \hspace{1cm} (5.3)

![Figure 5.3. Forces acting on one node](image)

- \(m\): mass of a single node
- \(a_i\): velocity of the node
- \(t\): time
- \(f_a\): air drag
- \(f_b\): buoyancy
- \(f_g\): gravity
- \(f_{i-1,i}\): action from node \(i-1\) to \(i\)
$f_{i,i+1}$: action from node $i$ to $i+1$

The direction of gravity was assigned as the negative direction of y-axis.

According to the assumption that fibers modeled are inextensible, which means that each fiber segment is inextensible during the simulated movements, so the length $ds$ of every segment should remain the same during the whole computation process, for the model in Figure 5.2a, this means:

$$L_{i}^{j+1} = L_{i}^{j} = \text{const} \quad (5.4)$$

$L_{i}^{j}$: Length of the $i^{th}$ segment at the $j^{th}$ time step.

For three-dimensional simulations, equations for node $i$ can be written as:

$$m \frac{dv_{ix}}{dt} = f_{ax} + f_{i-1,i,ix} + f_{i+1,i,ix}$$

$$m \frac{dv_{iy}}{dt} = f_{ay} + f_{b} + f_{g} + f_{i-1,i,y} + f_{i+1,i,y} \quad (5.5)$$

$$m \frac{dv_{iz}}{dt} = f_{az} + f_{i-1,i,z} + f_{i+1,i,z}$$

According to the assumption of the selected model that spheres are connected by rods of zero mass and zero diameters, this means that forces acting on one rod were balanced along its axis, and there is no air drag acting on the rod. Moreover, because in this chapter, the scenarios simulated here are that bending and twisting are neglected, this means that there is no bending or twisting moment acting on the rod. Therefore, from these analyses it can be seen that the rod actually is in a force equilibrium status.

The equations about the interactions between node $i$ and $i+1$ can be written as Equations 5.6. The derivation of these equations is discussed as the following, together with an analysis illustrated in Figure 5.4.
From the force equilibrium of the rod, $\vec{F}_{\text{Right}} = -\vec{F}_{\text{Left}}$, in addition, there is no bending or twisting moment acting on the rod, so the two forces are acting exactly on the same line, which is the rod’s centerline. This can be explained as the following: $\vec{F}_{\text{Right}}$ is decomposed into $\vec{F}_{R1}$ and $\vec{F}_{R2}$. $\vec{F}_{R1}$ can be defined in the same direction as the direction from node $i$ to node $i+1$ ($\vec{L}_i$) and $\vec{F}_{R2}$ is defined in a plane which is perpendicular to that direction, as shown in Figure 5.4. If node $i$ is assumed as a rotation center, then because the rod has zero mass, which means that its moment of inertia is zero, so torque $\vec{F}_{R2} \times \vec{L}_i = 0$, otherwise the rod will gain an infinite angular acceleration, which is impossible. The fact that this torque is zero proves that $\vec{F}_{R2} = 0$ since $\vec{F}_{R2}$ is defined in a plane that is perpendicular to $\vec{L}_i$. From the above analysis it can be concluded that these two forces acting on the two ends of a rod are along the rod’s centerline, and have the same magnitude, but opposite directions. Therefore, the equations describing the action between node $i$ and node $i+1$ via the rod can be written as in Equations 5.6.

Figure 5.4. Forces acting on one rod
\( \theta_{i,i+1x}, \theta_{i,i+1y}, \) and \( \theta_{i,i+1z} \) are the inclination angles between the rod connecting nodes from node \( i \) to node \( i+1 \) and the x-axis, y-axis, and z-axis directions respectively.

\[
f_{i,i+1x} = f_{i,i+1} \cos \theta_{i,i+1x} \\
f_{i,i+1y} = f_{i,i+1} \cos \theta_{i,i+1y} \\
f_{i,i+1z} = f_{i,i+1} \cos \theta_{i,i+1z} \tag{5.6}
\]

\[
\cos \theta_{i,i+1x} = \frac{x_{i+1} - x_i}{L_i} \\
\cos \theta_{i,i+1y} = \frac{y_{i+1} - y_i}{L_i} \tag{5.7} \\
\cos \theta_{i,i+1z} = \frac{z_{i+1} - z_i}{L_i}
\]

\( x_i, y_i, \) and \( z_i \) are the \( x, y, z \) positions of node \( i \).

Because for every segment during the movement the length keeps the same, so from Equation 5.4, at any time step, the movement should satisfy:

\[
\sqrt{(x_{i+1} - x_i)^2 + (y_{i+1} - y_i)^2 + (z_{i+1} - z_i)^2} = L_j \tag{5.8}
\]

Node \( i \)'s movement between time step \( n \) and \( n+1 \) is:

\[
v_{ix} = \frac{dx_i}{dt} \\
v_{iy} = \frac{dy_i}{dt} \tag{5.9} \\
v_{iz} = \frac{dz_i}{dt}
\]
5.5 Discrete Equations Integrated with the Fiber Movement Model

When a fiber is moving in an airflow field, solving equations 5.1 or 5.2 can get the position and velocity of each single point on that fiber, but obviously, because of complex initial conditions and boundary conditions, numerical solutions are what are required in this research in order to carry out simulations. In order to get the numerical solution of the model for simulating fibers’ movements in airflow, discrete equations of Equation 5.2 can be written as Equations 5.10, in these equations all surface forces and body forces are acting on unit length of the fiber. Concentrated forces acting on the two ends of a fiber element are respectively $f_i$ and $f_{i+1}$ in these equations, subscripts $i$ denote element series, and superscripts $n$ denote time steps. Subscripts $a$, $b$, and $g$ have the same meanings as those in Equation 5.3 (the negative direction of gravity is defined as the positive $y$-axis direction).

$$\rho \frac{y_{xi}^{(n+1)} - y_{xi}^{(n)}}{\Delta t} = \frac{f_{i-1x}^{(n)} - f_{ix}^{(n)}}{\Delta s} + f_{aix}^{(n)}$$

$$\rho \frac{y_{yi}^{(n+1)} - y_{yi}^{(n)}}{\Delta t} = \frac{f_{i-1y}^{(n)} - f_{i+1y}^{(n)}}{\Delta s} + f_{aiv}^{(n)} + f_k + f_g$$

Equation 5.10

$$\rho \frac{y_{zi}^{(n+1)} - y_{zi}^{(n)}}{\Delta t} = \frac{f_{i-1z}^{(n)} - f_{iz}^{(n)}}{\Delta s} + f_{aiz}^{(n)}$$

The above equations are multiplied by $\Delta s$, then $\Delta s$ is substituted by $L$, which is the length of rods in the model discussed in the previous section, $f_i$ is written as $-f_{i+1,i}$, so Equation 5.10 can be written as the following Equations 5.11. The body forces and surface forces described in these equations are like those forces that are described in Equation 5.3, they act on the whole length $L$ of one element, instead of a unit length.
Therefore, when integrated with the model analyzed from Equations 5.3 to 5.9 in Section 5.4, which provide additional conditions and constraints, the discrete equations for solving fiber movements are:

\[
\begin{align*}
  m \frac{v_{x_{i}i}^{(n+1)} - v_{x_{i}i}^{(n)}}{\Delta t} &= f_{a_{x_{i}i}}^{(n)} + f_{x_{i}i}^{(n)} f_{x_{i+1}i}^{(n)} + f_{x_{i}i}^{(n)} \\
  m \frac{v_{y_{i}i}^{(n+1)} - v_{y_{i}i}^{(n)}}{\Delta t} &= f_{a_{y_{i}i}}^{(n)} + f_{y_{i}i}^{(n)} f_{y_{i}i}^{(n)} f_{y_{i+1}i}^{(n)} + f_{y_{i}i}^{(n)} + f_{y_{i}i}^{(n)} \\
  m \frac{v_{z_{i}i}^{(n+1)} - v_{z_{i}i}^{(n)}}{\Delta t} &= f_{a_{z_{i}i}}^{(n)} + f_{z_{i}i}^{(n)} \\
  f_{x_{i}i}^{(n)} &= f_{x_{i}i}^{(n)} \cos \theta_{x_{i}i}^{(n)} \\
  f_{y_{i}i}^{(n)} &= f_{y_{i}i}^{(n)} \cos \theta_{y_{i}i}^{(n)} \\
  f_{z_{i}i}^{(n)} &= f_{z_{i}i}^{(n)} \cos \theta_{z_{i}i}^{(n)} \\
  \cos \theta_{x_{i}i}^{(n)} &= \frac{x_{i}^{(n)} - x_{i-1}^{(n)}}{L} \\
  \cos \theta_{y_{i}i}^{(n)} &= \frac{y_{i}^{(n)} - y_{i-1}^{(n)}}{L} \\
  \cos \theta_{z_{i}i}^{(n)} &= \frac{z_{i}^{(n)} - z_{i-1}^{(n)}}{L} \\
  x_{i}^{(n+1)} &= x_{i}^{(n)} + \frac{1}{2} (v_{x_{i}i}^{(n)} + v_{x_{i}i}^{(n+1)}) \Delta t \\
  y_{i}^{(n+1)} &= y_{i}^{(n)} + \frac{1}{2} (v_{y_{i}i}^{(n)} + v_{y_{i}i}^{(n+1)}) \Delta t \\
  z_{i}^{(n+1)} &= z_{i}^{(n)} + \frac{1}{2} (v_{z_{i}i}^{(n)} + v_{z_{i}i}^{(n+1)}) \Delta t \\
  \sqrt{(x_{i}^{(n+1)} - x_{i-1}^{(n+1)})^2 + (y_{i}^{(n+1)} - y_{i-1}^{(n+1)})^2 + (z_{i}^{(n+1)} - z_{i-1}^{(n+1)})^2} &= L
\end{align*}
\]
In Equations 5.11, air drag acting on fibers is calculated by the following method: as discussed in Section 5.3, fibers are modeled as several spheres connected together by rods of zero diameters, and then air drag acting on those spheres is calculated. As described in that model, the air drag acting on that sphere is considered as air drag acting on the whole element. Calculation can be carried out by the following method and equations:

If Reynolds number is smaller than five, the air drag coefficient is calculated by Oseen Equation, which is:

\[
C_D = \frac{24}{Re} \left(1 + \frac{3}{16} Re\right)
\]

The Reynolds number in these equations is defined as \( Re = \frac{U_\infty D}{\nu} \), the characteristic length used in these equations is the diameter of spheres: \( D \). \( U_\infty \) is the incoming airflow speed, and \( \nu \) is air kinetic viscosity.

When the Reynolds number is larger than five, the air drag is calculated by interpolated experimental data (see Figure 5.5).

Air drag \( F_a \) is calculated by the following equation:

\[
F_a = C_D \pi r_0^2 \left(\frac{1}{2} \rho U_\infty^2\right)
\]

in which \( r_0 \) is the radius of the sphere.

In these equations, velocities are actually the relative velocities between the airflow and the moving fiber, relative velocity is calculated as the difference between the velocity of a sphere and the velocity of incoming airflow to that sphere, which is at the same position. This velocity of airflow was obtained from CFD results.
Air drag calculated with above methods has a direction that is parallel to the velocity of the initial undisturbed incoming airflow to that sphere.

Figure 5.5. Drag coefficient for spheres\(^{(54)}\)

Introducing the following dimensionless items in Equations 5.16, they are dimensionless velocity, forces, and coordinates:

\[
V = v \left( \frac{\Delta t}{L} \right)
\]

\[
F = f \left( \frac{\Delta t^2}{mL} \right)
\]

\[
X = x \left( \frac{1}{L} \right)
\]

\[
Y = y \left( \frac{1}{L} \right)
\]

\[
Z = z \left( \frac{1}{L} \right)
\]

\(5.16\)
The velocity, forces, and coordinates in Equations 5.11 to 5.15 can be substituted by dimensionless items, resulting in the dimensionless equations:

\[
V_{ix}^{(n+1)} - V_{ix}^{(n)} = F_{ai}^{(n)} + F_{i-1,ix}^{(n)} + F_{i+1,ix}^{(n)}
\]

\[
V_{iy}^{(n+1)} - V_{iy}^{(n)} = F_{ai}^{(n)} + F_{i-1,iy}^{(n)} + F_{i+1,iy}^{(n)} + F_g + F_b
\] (5.17)

\[
V_{iz}^{(n+1)} - V_{iz}^{(n)} = F_{ai}^{(n)} + F_{i-1,iz}^{(n)} + F_{i+1,iz}^{(n)}
\]

\[
F_{i-1,ix}^{(n)} = F_{i-1,iy}^{(n)} \cos \theta_{i-1,ix}^{(n)}
\]

\[
F_{i-1,iy}^{(n)} = F_{i-1,iy}^{(n)} \cos \theta_{i-1,iy}^{(n)}
\] (5.18)

\[
F_{i-1,iy}^{(n)} = F_{i-1,iz}^{(n)} \cos \theta_{i-1,iz}^{(n)}
\]

\[
\cos \theta_{i-1,ix}^{(n)} = X_i^{(n)} - X_{i-1}^{(n)}
\]

\[
\cos \theta_{i-1,iy}^{(n)} = Y_i^{(n)} - Y_{i-1}^{(n)}
\] (5.19)

\[
\cos \theta_{i-1,iz}^{(n)} = Z_i^{(n)} - Z_{i-1}^{(n)}
\]

\[
X_i^{(n+1)} = X_i^{(n)} + \frac{1}{2} (V_{xi}^{(n)} + V_{xi}^{(n+1)})
\]

\[
Y_i^{(n+1)} = Y_i^{(n)} + \frac{1}{2} (V_{yi}^{(n)} + V_{yi}^{(n+1)})
\] (5.20)

\[
Z_i^{(n+1)} = Z_i^{(n)} + \frac{1}{2} (V_{zi}^{(n)} + V_{zi}^{(n+1)})
\]

\[
\sqrt{(X_i^{(n+1)} - X_{i-1}^{(n+1)})^2 + (Y_i^{(n+1)} - Y_{i-1}^{(n+1)})^2 + (Z_i^{(n+1)} + Z_{i-1}^{(n+1)})^2} = 1
\] (5.21)

The methods used to solve those equations are discussed in Chapter 6.
6. MODEL’S NUMERICAL SOLUTION AND RESULTS

6.1 Introduction

In this chapter, numerical methods for solving equations in Chapter 5 are discussed. Solutions of the simulated models are obtained when the equations are solved by coupling them with the data from the airflow field, which are discussed in Chapter 4. The results obtained for the simulations are presented following these discussions.

Two possible methods were considered for solving discrete equations from Equations 5.16 to Equations 5.21. The first method was the well-known Newton’s method, and the second one used an algorithm that was developed from non-linear optimization theory. Solutions of simulations were obtained by using the second method.

The results from simulations’ were studied for the following three factors: fiber initial position, fiber initial velocity, and airflow field velocity distribution. Their effects on simulated fibers’ configurations during their movements in the fiber transfer channel are presented and discussed.

6.2 Methods Used for Solutions

The dimensionless discrete equations in the previous Chapter 5 are non-linear equations. For non-linear equations:

\[
\begin{align*}
    f_1(x_1, x_2, \cdots, x_n) &= 0 \\
    f_2(x_1, x_2, \cdots, x_n) &= 0 \\
    &\vdots \\
    f_n(x_1, x_2, \cdots, x_n) &= 0 \\
\end{align*}
\]  

(6.1)

usually solutions can be achieved by using the Newton’s method, which was explained as the following:
Assume non-linear equations in Equations 6.1 have solutions:

\[ S = (s_1, s_2, \ldots, s_n)^T \]  

(6.2)

Then assume \((x_1^{(0)}, x_2^{(0)}, \ldots, x_n^{(0)})^T\) that is near \(S\), from Taylor series, there is:

\[
\begin{align*}
&f_1(x_1^{(0)}, x_2^{(0)}, \ldots, x_n^{(0)}) + \frac{\partial f_1}{\partial x_1}(x_1 - x_1^{(0)}) + \frac{\partial f_1}{\partial x_2}(x_2 - x_2^{(0)}) + \cdots + \frac{\partial f_1}{\partial x_n}(x_n - x_n^{(0)}) + \cdots = 0 \\
&f_2(x_1^{(0)}, x_2^{(0)}, \ldots, x_n^{(0)}) + \frac{\partial f_2}{\partial x_1}(x_1 - x_1^{(0)}) + \frac{\partial f_2}{\partial x_2}(x_2 - x_2^{(0)}) + \cdots + \frac{\partial f_2}{\partial x_n}(x_n - x_n^{(0)}) + \cdots = 0 \\
&\vdots \\
&f_n(x_1^{(0)}, x_2^{(0)}, \ldots, x_n^{(0)}) + \frac{\partial f_n}{\partial x_1}(x_1 - x_1^{(0)}) + \frac{\partial f_n}{\partial x_2}(x_2 - x_2^{(0)}) + \cdots + \frac{\partial f_n}{\partial x_n}(x_n - x_n^{(0)}) + \cdots = 0
\end{align*}
\]  

(6.3)

If the high order items in Equations 6.3 were neglected, they can also be written as the following equations:

\[
\begin{bmatrix}
\frac{\partial f_1}{\partial x_1} & \frac{\partial f_1}{\partial x_2} & \cdots & \frac{\partial f_1}{\partial x_n} \\
\frac{\partial f_2}{\partial x_1} & \frac{\partial f_2}{\partial x_2} & \cdots & \frac{\partial f_2}{\partial x_n} \\
\vdots & \vdots & \ddots & \vdots \\
\frac{\partial f_n}{\partial x_1} & \frac{\partial f_n}{\partial x_2} & \cdots & \frac{\partial f_n}{\partial x_n}
\end{bmatrix}
\begin{bmatrix}
x_1 - x_1^{(0)} \\
x_2 - x_2^{(0)} \\
\vdots \\
x_n - x_n^{(0)}
\end{bmatrix}
= \begin{bmatrix}
f_1(x_1^{(0)}, x_2^{(0)}, \ldots, x_n^{(0)}) \\
f_2(x_1^{(0)}, x_2^{(0)}, \ldots, x_n^{(0)}) \\
\vdots \\
f_n(x_1^{(0)}, x_2^{(0)}, \ldots, x_n^{(0)})
\end{bmatrix}
\]  

(6.4)

Written in vector format, this equation set becomes:

\[ f'(\bar{x}^{(0)})\Delta\bar{x}^{(0)} = -f(\bar{x}^{(0)}) \]  

(6.5)

in which \(f'\) is the Jacobian matrix of functions \(f\) about vector \(x\) (see Equation 6.5).

Above equations now are a set of linear equations, after \(\Delta\bar{x}^{(0)}\) is directly solved, define:

\[ \bar{x}^{(1)} = \bar{x}^{(0)} + \Delta\bar{x}^{(0)} \]  

(6.6)

as the next initial value of these linear equations, which replaces \(\bar{x}^{(0)}\) in Equations 6.4 to solve the \(\Delta\bar{x}^{(1)}\) for the next step. This iteration is repeated until \(\|\Delta x^{(i)}\| \leq \varepsilon\) has been
satisfied, ($\epsilon$ is a predetermined value that is small enough), and then $\bar{x}^{(i)}$ is deemed as one solution for the equation set (see Equation 4.3).

When the above Newton’s method is used to solve non-linear equations, the following problems should be considered:

1. It is more complex to solve the equation set as the number of unknowns increases. For example, in a two-dimension model, for a fiber that is modeled as consisting of 11 nodes (10 segments), there are 54 unknowns for a single time step. In three-dimensional cases, which are the major focus discussed in this dissertation, the complexity is even much higher.

2. Another set of linear equations must be solved at every iteration as initialized in Equation 6.6;

3. It could be difficult to guess a proper initial value for the equation set, which means that, in Equation 6.5, the initial $\bar{x}^{(0)}$ should be chosen to be close to the real solution $S$ (Equation 6.2); and in consequence, a stable convergent solution is difficult to achieve.

At first, this method was used to solve the equation set from Equations 5.17 to Equation 5.21, but with the occurrences of above problems, it was believed that, for the equations used to describe this model, a numerical method that can obtain more robust solutions was needed.

The second numerical method was used to solve those equations; which is based on the idea of non-linear optimization methods.

From the Taylor series, there are:

$$f(x + \Delta) \approx f(x) + \Delta'g(x)$$

(6.7)
\[ g(x) = \left( \frac{\partial f}{\partial x_1}, \ldots, \frac{\partial f}{\partial x_n} \right) \]

\[ \Delta = (\Delta x_1 \cdots \Delta x_n) \]

In the direction of \(-g(x)\), the function \(f(x + \Delta)\) gets the fastest decreasing speed, so in that direction a one-dimension linear search is made to find a position of \(x^{(i)}\) where function \(f(x)\) has the minimum value along this direction. Then \(x^{(i)}\) is set as the starting position of the next iteration; the gradient function \(g(x)\) and the linear search are computed again from \(x^{(i)}\). The above process is repeated to find the next minimum until the error can satisfy a pre-determined value that is small enough.

The objective function used to solve the equations of the model was defined as:

\[ f_{OBJ} = \sum_{i=1}^{k} (L_i^{(n+1)} - 1)^2 < C \quad (6.8) \]

This function was selected because during the whole movement, the extension of a fiber is neglected and so the length of every single segment should remain as a constant value. In Equation 6.9 \(C\) is a predetermined value, as a convergence criterion for simulations, \(C\) was defined to be close to zero during the computation, So once it had been satisfied, the solution of the equations was achieved.

For a model in which the simulated fiber was considered as consisting of ten segments (11 nodes, numbered from 0 to 10), Equation 6.8 can be written as the following equation:

\[
\sum_{i=0}^{9} \left[ \left( X_{i+1}^{(n)} - X_i^{(n)} + V_i^{(n)} - V_{i+1}^{(n)} \right)^2 + \left( Y_{i+1}^{(n)} - Y_i^{(n)} + V_{i+1}^{(n)} \right)^2 + \left( Z_{i+1}^{(n)} - Z_i^{(n)} + V_{i+1}^{(n)} \right)^2 \right]
\]

\[
- 0.5 F_{i+1,x}^{(n)} \cos \theta_{i+1,x}^{(n)} - 0.5 F_{i+1,y}^{(n)} \cos \theta_{i+1,y}^{(n)} - 0.5 F_{i+1,z}^{(n)} \cos \theta_{i+1,z}^{(n)} - 0.5 F_{i,x}^{(n)} \cos \theta_{i,x}^{(n)} - 0.5 F_{i,y}^{(n)} \cos \theta_{i,y}^{(n)} - 0.5 F_{i,z}^{(n)} \cos \theta_{i,z}^{(n)} \right]
\]

87
\[ + 0.5F_{w_i+1z}^{(n)} - 0.5F_{ax}^{(n)} - 0.5F_{i-1y}^{(n)} \cos \theta_{i-1z}^{(n)} + F_{i,j+1}^{(n)} \cos \theta_{i,j+1z}^{(n)} - 0.5F_{i+1,j+2}^{(n)} \cos \theta_{i+1,j+2z}^{(n)} \right]^{1/2} - 1 \right| < C \]

(6.9)

The interactions between nodes were selected as controlled variables for Equation 6.9, so the \(-g(x)\) is:

\[-\left( \frac{\partial F_{OBJ}}{\partial F_{0,1}}, \frac{\partial F_{OBJ}}{\partial F_{1,2}}, \ldots, \frac{\partial F_{OBJ}}{\partial F_{i-1,i}} \right)^T \]

(6.10)

Equation 6.8 controlled that, at every next time step, fiber section lengths should keep the same length as the previous time step (a constant), and this explained the physical meanings of variables in the gradient function actually coming from the previous time step, which are the interactions between each two connected nodes.

\[ \frac{\partial F_{OBJ}^{(n+1)}}{\partial F_{i+1}} = -\left[ (\sqrt{L_{i-1}^{(n+1)} + L_{i-1}^{(n+1)}} + L_{i}^{(n+1)} - 1)(L_{i-1}^{(n+1)} + L_{i}^{(n+1)} + L_{i}^{(n+1)} - 1)(L_{i}^{(n+1)} + L_{i}^{(n+1)} + L_{i}^{(n+1)} - 1)(L_{i}^{(n+1)} + L_{i}^{(n+1)} + L_{i}^{(n+1)} - 1) \right] + \]

\[ + 2\left[ (L_{i}^{(n+1)} + L_{i}^{(n+1)} + L_{i}^{(n+1)} - 1)(L_{i}^{(n+1)} + L_{i}^{(n+1)} + L_{i}^{(n+1)} - 1)(L_{i}^{(n+1)} + L_{i}^{(n+1)} + L_{i}^{(n+1)} - 1) \right] - \]

\[ + \left[ (L_{i}^{(n+1)} + L_{i}^{(n+1)} + L_{i}^{(n+1)} - 1)(L_{i}^{(n+1)} + L_{i}^{(n+1)} + L_{i}^{(n+1)} - 1)(L_{i}^{(n+1)} + L_{i}^{(n+1)} + L_{i}^{(n+1)} - 1) \right] + \]

\[ L_{i+1}^{(n+1)} \cos \theta_{i+1}^{(n)} + L_{i+1}^{(n+1)} \cos \theta_{i+1}^{(n)} \]

(6.11)

\[ L_{ix}^{(n)} = X_{i+1}^{(n)} - X_i^{(n)} \]

\[ L_{iy}^{(n)} = Y_{i+1}^{(n)} - Y_i^{(n)} \]

(6.12)

\[ L_{iy}^{(n)} = Y_{i+1}^{(n)} - Y_i^{(n)} \]

\[ i = -1, \ldots, 10, \]

For \( \frac{\partial F_{OBJ}^{(n+1)}}{\partial f_{0,1}} \), define \( L_{-1x}^{(n)} = L_{-1y}^{(n)} = L_{-1z}^{(n)} = 0 \)
For $\frac{\partial f^{(n+1)}_{OBJ}}{\partial f^{(n)}_{9,10}}$, define $L^{(n)}_{10x} = L^{(n)}_{10y} = L^{(n)}_{10z} = 0$

The following steps outlined the procedure for solving the equations for simulations using the method discussed above, this is also the workflow used by the computer programs developed for this research.

1. Program reads in the following parameters as part of the initial conditions:
   i. Fiber properties such as density, length, and diameter;
   ii. Airflow properties such as density and viscosity;
   iii. Parameters used in computing such as time step length.

2. Program reads and specifies the following initial conditions:
   i. Airflow field parameters calculated by CFD software;
   ii. Fiber initial configurations such as shape, velocity, and position;
   iii. Initializing the starting time step and interactions between nodes.

3. Program calculates the air drag acting on each node. Based on positions and velocities of the nodes in the flow field, firstly, the airflow parameters at the respective positions need to be located, and then the relative velocities between airflow and nodes are calculated and used as the velocities for calculating the air drag (see Section 5.5).

4. After the interactions between nodes were initialized, the gradient function is calculated from Equations 6.10 and 6.11 to get a direction for linear search on fixed step length, the linear search stops when a minimum is found along that direction.
(5). The position at which the linear search stops is set as initial values used for the gradient function of the next iteration. Step (4) is repeated until a set of values of those interactions between nodes that can satisfy Equation 6.8 is found.

(6). Fiber position and velocity can be calculated after Step (5) by substituting these interactions between nodes into Equation 5.17 to 5.21. These values then can be used as fiber initial configurations such as positions and velocities for the next time step.

(7). At each time step, resultant data from the fiber movement model, such as velocity and position, are saved; and these data are utilized by the visualization program.

(8). Computations are repeated from Step (3) to Step (7). The program will stop if:
   i. Fiber hits the solid wall; or
   ii. Fiber has moved out of the channel outlet; or
   iii. Fiber has been moving for a time that is longer than a specified period.

During all computations that were carried out in this research, the $C$ value, which was defined in Equation 6.9, had been reached. This means that, even though theoretically there is a possibility of the solution converging to a local minimum that is far from zero, this method has given a robust solution to the equations for simulations.

Table 6.1 gives a set of parameters of air and fiber properties that were used in programs for the simulations. These parameters of air were selected from published data. The parameters for the simulated fibers were initially selected in such a way that they were in a range that is close to those of cotton fibers.

---

2 In computation this could be indicated by that the $C$ cannot be reached along with the progress of iterations and the error is convergent to some other certain value. This is the case in which initial values should be guessed again.
Other parameters include time step length and fibers’ initial positions and velocities. They can be varied for each computation. If bending and twisting properties are simulated, then bending stiffness and twisting stiffness should also be considered in the programs, this is discussed in the Chapter 7.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Air density:</td>
<td>1.205 kg/m³</td>
</tr>
<tr>
<td>Air Viscosity:</td>
<td>1.57×10⁻⁵ m²/s</td>
</tr>
<tr>
<td>Fiber Density:</td>
<td>1.55×10³ kg/m³</td>
</tr>
<tr>
<td>Fiber Diameter:</td>
<td>1.5×10⁻⁵ m</td>
</tr>
<tr>
<td>Fiber Length:</td>
<td>2.0×10⁻² m</td>
</tr>
</tbody>
</table>

Table 6.1. Air and fiber properties used in simulation programs

6.3 Results of Simulation

Initially two-dimensional models were put into the computation. The following two snapshots in Figure 6.1 and 6.2 were obtained from those simulations. The modeled channel, which has a convergent shape, has an effect that accelerates and straightens the simulated fiber in the airflow field (see Figure 6.1). In addition, in Figure 6.2 it can be seen that the airflow field changed the direction of a straight fiber, which was put to lower part of that channel. These results show the same tendency as predicted by earlier research(1,40).

In this dissertation, the discussions of results are focused on three-dimensional simulations. The simulated fiber movements were computed in the airflow field of the modeled fiber transfer channel, which is discussed in Chapter 4. So that the model investigated not only gave an understanding of the general characteristics of fiber-airflow interactions, but also represented the fiber movements in a three-dimensional model of a real component in textile machinery.
Figure 6.1. Simulation result shows a bended fiber was straightened while passing through the channel

Figure 6.2. Computer modeling result shows a straight fiber changed its orientation after passing through the channel

These simulations were carried out for the following factors: fiber’s initial position, fiber’s initial velocity, and airflow field distribution.

The results of simulations were used as inputs for the visualization program, which was developed using OpenGL. This program displays fiber movements within the contour of the three-dimensional model. Fiber is repeatedly displayed once at a position corresponding to a certain time with a fixed time interval. Animation programs were also developed
6.3.1 Fiber's Initial Position

The fiber’s initial position has very important impact on the movement inside the channel. For a three-dimensional model, this movement could be much more complex than a two-dimensional model such as shown in Figure 6.1 and 6.2. The following two reasons can partly explain this complexity.

The first, the CFD simulations in Chapter 4 showed that the airflow field of the channel was not evenly distributed; especially at the channel inlet, the flow pattern was far from uniform. The velocity of the airflow may be not parallel to the axial direction of the channel. It could change dramatically, as demonstrated by the existence of recirculation zone.

The second, as the fiber enters the channel at some initial position, it is very possible that the fiber will hit the walls of the channel during the movement. After the fiber hits the wall, in addition to air drag, the friction between the wall and fiber then has an influence on fiber’s trajectory. In some situation, this influence could be negative, for example, it is going to decrease the fiber’s speed.

The airflow field data for the simulations was selected from the CFD result of the airflow field with a 4×4 outlet, which was discussed in Section 4.8.1. In order to study the influence of different initial positions as an independent factor, the initial speed of fiber was defined as zero, and then the fiber’s initial inertia was zero and did not have a contribution to the fiber’s trajectory inside the airflow field.

The initial positions for the simulation were selected at the following points. These points were used to define the coordinates of the last node at the modeled fiber’s tail. The coordinates of other nodes were then defined with respect to the fiber’s initial shape.
Figure 6.3 shows the I-, J- and K-directions at the fiber transfer channel’s inlet cross-section (the cross-section of I-plane 1, see Figure 4.27 regarding the definition of I-planes).

![Diagram of I-, J-, and K-directions](image)

Figure 6.3. Selected starting points of fiber initial position

At first, three points were selected as the starting position of the modeled fibers. These three points are located at the J-plane 13, between I-planes 1 and 2, as the points I, II, and III shown in Figure 6.3. Their z coordinates in computation equal the z coordinate of the grid at I = 1, J = 13, K = 1. Their x and y coordinates equal the average values of x, y coordinates of grid at I = 1, J = 13, K = 1 and 2 (for point I), K = 2 and 3 (for point II), K = 3 and 4 (for point III).

Simulation results showed that starting from the beginning of the movement, the tails of the simulated fibers were “pushed down” by airflow if the selected initial positions were point II or point III, as displayed in Figures 6.4(b) and (c). However, if the simulated fiber started from point I, the airflow was carrying the fiber moving through the channel, as displayed in Figure 6.4(a).
Figure 6.4. The configurations of fibers of different initial positions at the beginning of movements.

In all these three situations, the simulated fiber started with a straight shape. (a) shows the movement started in point I, the interval between two snapshots was 0.001 second. (b) shows the movement started in point II, the interval between two snapshots was 0.0005 second, the time length shown in the figure was 0.002 second. (c) shows the movement started from point III, the interval between two snapshots was 0.0003 second, the time length was 0.0009 second.
In Chapter 4, the CFD results demonstrate the existence of the re-circulation zone starting from the inlet of the channel. From Figure 4.26 it can be seen that the air near the center of the inlet cross-section actually does not flow in the direction that is nearly parallel the channel’s centerline, but in a direction that is close to the K-direction. Points II and III are located close to this region where the airflow is almost parallel to the K-direction mentioned above, because of the action of the airflow at the re-circulation zone, the tails of those simulated fibers have the tendency of flapping down instead of moving forward. Point I is located much closer to the channel’s wall, at this location the airflow has already turned to a direction that is more parallel to the channel centerline direction. Therefore, the air drag does not cause the fiber’s tail to flap down.

The above results and analysis indicated that, at some initial positions, the existence of a re-circulation zone at the beginning of the airflow field could cause the fiber to bend. This is coincident with the two-dimensional simulation results by Kong and Platfoot(42).

Figure 6.5 compares the movements of fibers starting from three different initials positions distributed along the z direction in computations. These three points are shown in Figure 6.3 as points IV, V, and VI. The distance at z direction is 0.002 m between IV and V, 0.001 m between V and IV.

It can be seen from Figures 6.5(a), (b), and (c) that, even though it is possible that the fiber hits the walls of the fiber transfer channel, at the direction that the channel has a convergent contour, or in other words, the cross-section from inlet to outlet has a negative gradient, the airflow has a tendency to carry the fiber away from the wall toward the channel exit.
Figure 6.5. Results of simulations show the fiber movements starting from different initial positions along the J-direction. All simulated movements shown in above figures lasted for 0.007 second. The interval between two snapshots was 0.001 second.
Figure 6.6 compares the z direction speeds of the tips of fibers starting from these three different initial positions. All these speeds were obtained when the simulated fiber had moved for 0.007 second. This figure shows that if the initial position is closer to the channel’s wall, the fiber receives a larger action from airflow field that could turn it to the centerline toward the exit of the channel, as indicated by the increasing z direction speed. However, if the fiber started too close to the wall, it could hit the wall before it exits the channel.

![Graph showing fiber tip's speed at z direction as initial position moving close to channel wall](image)

The convergent channel’s effect on straightening bended fibers has been studied in some earlier theoretical and experimental research. Figure 6.7 shows zigzag shape fibers moving through the channel, these simulated fibers started from different initial positions as defined in Figure 6.5. From these simulations, it can be found that before the fibers hit the channel’s wall, they were straightened while moving through the channel.

In order to compare the straightening effects, a fiber-straightening factor $C_s$ was defined as:
Figure 6.7. Fiber straightening while moving in the channel.

All simulated movements shown in above figures lasted for 0.004 second. The interval between two snapshots was 0.002 second.
\[ C_s = \frac{L_0}{L_1} \] (6.13)

in which \( L_0 \) is the distance between fibers two ends at the beginning of the movement, \( L_1 \) is the above distance at a certain time during the movement. A smaller \( C_s \) means the fiber is more straightened by the airflow during the movement. For the simulations shown in Figure 6.7, the \( C_s \) were:

<table>
<thead>
<tr>
<th>Initial Position:</th>
<th>IV</th>
<th>V</th>
<th>VI</th>
</tr>
</thead>
<tbody>
<tr>
<td>( C_s )</td>
<td>0.73</td>
<td>0.76</td>
<td>0.78</td>
</tr>
</tbody>
</table>

Table 6.2. Fiber-straightening factor as fibers starting from different positions

From Table 6.2, it can be seen that starting from initial position IV, the bended fiber becomes straighter compared with position V and VI, even thought the differences are not big. It could be explained from Figures 6.5 and 6.6 that since as the fiber was starting from a position that was closer to the wall, its leading end was been turning toward the centerline of the channel by the airflow.

6.3.2 Fiber's Initial Speed

Apparently, because the airflow in the channel is accelerating (see the discussions in Chapter 4), generally the fiber, which is carried by the airflow, is also accelerating from the inlet to the outlet. Figure 6.8 shows the speed increasing as the fiber moving across the channel; the fiber initial configurations were the same as the configurations shown in Figure 6.5 (a).

In real situations, fibers enter the fiber transfer channel at some certain speeds. The following simulations studied the fiber acceleration in the fiber transfer channel starting at different initial speeds. In all these simulations, fibers started from the same initial position, as shown in Figure 6.5 (a).
Figure 6.8. Change of fiber speed while moving through the fiber transfer channel

The selected initial speeds were: 8 m/s, 10 m/s, 12 m/s, Figure 6.9 compares the changes of speed of the leading ends of the simulated fibers.

Figure 6.9 shows that with a higher initial speed, the fiber gains an acceleration that is higher, at least not less, than the fiber with a lower initial speed. Even though the air drag is computed by the relative speed between the airflow and the fiber, it could be explained
as the following. In the fiber transfer channel, airflow always has a higher acceleration than the fiber (see Chapter 4 about the speed increasing along the channel’s centerline), the fiber with a higher initial speed could enter the region where air has a higher speed within an earlier time, so with the same time length, the fiber could gain more acceleration.

6.3.3 Airflow Field Velocity Distribution

Besides the initial conditions of the simulated fibers, such as initial position and speed, the airflow field plays an importance role in the determination of fiber movements. According to the mathematical model established in Chapter 5, the fiber-airflow interaction is calculated by the relative velocity between these two phases, so the velocity field’s variations affects the fiber movements in the airflow field even if the simulated fibers have same initial conditions.

CFD simulations have been used to analyze variation of velocity field with respect to the variation of the channel outlet back pressure in Section 4.8.3. The data of velocity distribution from those CFD results was utilized as input for the simulations. Fiber movements were simulated in the airflow fields that were under the channel outlet back pressures of 60 mbar, 70 mbar, 80 mbar, and 100 mbar. The initial configurations of simulated fibers were the same as those of the simulation shown in Figure 6.5 (a).

Figure 6.10 compares the fiber accelerations in the channel under different back pressures. The results reveal that, in the simulated back pressure range, the fiber has a higher acceleration if the channel has a lower outlet back pressure. The results discussed in Section 4.8.3 show that as the back pressure decreases, the acceleration of airflow
increases in the fiber transfer channel (see Figures 4.20 ~ 4.25), thus air drag acting on the fiber is higher given that the simulated fiber all has the same initial configurations.

Figure 6.10. Change of fiber speed with different channel outlet back pressures

In Section 6.3.1, the simulation results reveal that starting from some initial positions, it is possible that the simulated fibers could “flap down” because of the action from the re-circulation zone (See Figure 6.4 (b) and (c)). Fiber movement was simulated again using the same fiber initial conditions as those of the simulated fiber in Figure 6.4 (b), but the channel outlet back pressure was decreased from 80 mbar to 60 mbar.

A comparison between the results from these two simulations is demonstrated in Figure 6.11. It shows the following differences between the fiber movements under these two back pressures. For the 80 mbar outlet back pressure, the fiber’s tail was flapping down, and after about 0.002 seconds, it hit the wall; but for the 60 mbar outlet back pressure, even though at the beginning the fiber was also flapping down, but the effect was not as pronounced as the previous example. The fiber did not hit the wall, it moved away toward the channel outlet.
Figure 6.11. Fiber configurations at the beginning of movements with different channel outlet back pressures. In both figures, the interval between two snapshots is 0.00025 second.

The relationship between the re-circulation zone and the channel outlet back pressure has been discussed in Section 4.8.4. CFD results indicate that under a lower outlet back pressure, the re-circulation zone covers smaller space or indeed there may be total absence of a re-circulation zone. So when the simulated fiber starts its movement at the channel inlet, even though its tail is located in a region in which the airflow velocity is in a direction that is not parallel or even transverse to the channel’s centerline, the main part of this fiber is still submerged in a region that the airflow velocity has already become more parallel to the channel’s centerline. This enables the fiber to gain enough acceleration to cause its tail to move toward the channel outlet in order to avoid hitting the wall at the beginning, as displayed in Figure 6.11 (b). This simulation result can be supported by the earlier experimental work about fiber draft ratio\(^{(2)}\).
7. MODELING OF FIBER BENDING AND TWISTING PROPERTIES

7.1 Introduction

In the fiber movement model discussed in Chapter 5, it was assumed that the fiber was completely flexible and the bending and twisting properties were excluded from these simulations. This assumption is based on the consideration that for one single fiber in the simulated scenarios, its stiffness is not large enough to affect the pattern of fiber movement in the studied airflow field to a notable degree.

In real terms, bending and twisting properties are of the basic factors that are usually used to reflect the textile material’s mechanical characteristics. It is thus believed that the computer model integrated with these properties can be not only more accurate for simulating and predicting the fiber movement in the fiber transfer channel discussed in previous chapters, but also potentially more expandable to other textile processes.

Therefore, this research was extended to include the integration of fiber bending and twisting properties in the computer model in order to investigating their influences on fiber movement. This chapter discusses the methods used to implement bending and twisting into the mathematical model. The simulation results were compared with respect to the impact of different flexural rigidities and torsional rigidities.

7.2 The Definitions of Fiber Bending and Twisting Parameters

The fiber flexural rigidity is defined as the change of bending moment to cause change of fiber unit curvature:

\[
R_b = \frac{\Delta M_b}{\Delta \kappa} \quad (7.1)
\]
in which \( R_B \) is the fiber flexural rigidity, \( M_b \) is the bending moment, and \( k \) is the curvature. Larger flexural rigidity means more bending load is needed to bend the fiber to the same degree.

The fiber torsional rigidity is defined as the change of twisting moment to cause the unit angular change along unit fiber length:

\[
R_T = \frac{\Delta M_T}{\Delta \tau}
\]

(7.2)

in which \( R_T \) is the fiber torsional rigidity, \( M_T \) is the twisting moment, and \( \tau \) is the fiber angular change along unit length. Larger torsional rigidity means more twisting torque is needed to twist the fiber to the same degree.

A fiber’s flexural rigidity and torsional rigidity are determined by the fiber’s modulus, shape, and density, etc., usually fibers with lower modulus or smaller diameter have lower values of flexural and torsional rigidities. Some environment variables such as humidity and temperature have important influence on them; the increasing of humidity or temperature will cause the certain fiber rigidities to decrease.\(^{57, 58}\) This is largely because of the decreasing of fiber modulus\(^{59, 60}\).

There have been many experimental studies investigating fiber flexural and torsional rigidities of different textile fibers\(^{61, 62, 63}\). The resultant data from these studies were not completely coincident with each other, or even the definitions were slightly different, for example, compared with the above definition of torsional rigidity, some researchers used number of turns instead of the angular change.

The present research does not attempt to make a choice of a specific value for flexural or torsional rigidity, but simulates and compares their influence on fiber movements in the airflow filed discussed in previous chapters by using a series of values for these
parameters. These values were selected in such a way that they well cover the range of the proposed values of cotton fibers from the resultant data of those earlier experimental investigations. They are listed in Table 7.1.

a. Flexural rigidity \((\times 10^{-12} \text{ N m}^2)\):

| 5 | 10 | 15 | 20 | 25 |

b. Torsional rigidity \((\times 10^{-12} \text{ N m}^2)\):

| 2 | 4 | 6 | 8 | 10 |

Table 7.1. Flexural and torsional rigidities for simulations

7.3 The Integration of Bending and Twisting Properties into the Three-dimensional Mathematical Model

7.3.1. Numerical Methods for Computing Bending and Twisting moments

There were many earlier studies on the problems of three-dimensional large-scale elastic bending and twisting curves\(^{(64, 65)}\). For most cases, the research was actually focused on the static equilibrium problems of the deformed curves by using different approaches\(^{(66, 67)}\). The present research attempted to look for a numerical method that can be integrated conveniently and effectively into the mathematical model for simulating the fiber movements.

Regarding the bending and twisting of a fiber in three-dimensional space, the first problem that needs to be solved is to determine the geometrical curvature and torsion of a point at the deformed space curve of the studied fiber. The geometrical curvature is connected with two effective curvatures with respect to the two principal axes of inertia of the cross-section of the fiber, and theoretically, there should be two flexural rigidities
about these two inertia axes. The torsion corresponds to torsional rigidity about center of inertia of the fiber cross-section.

Then based on the fibers’ stress-strain relationship discussed in the previous section, the equations connecting the bending and twisting moments and the effective curvatures and torsion can be written as Equations 7.3:

\[ M_{b1} = R_{b1} \cdot \kappa_1 \]
\[ M_{b2} = R_{b2} \cdot \kappa_2 \]
\[ M_T = R_T \cdot \tau \]

in which the subscripts 1 and 2 denote the two different inertia axes for the bending moments acting on the fiber.

From above analysis, it can be seen that in order to solve equations (7.3), it is necessary to calculate two effective curvatures and the torsion of a point at the curve of the simulated fiber. The problem is that for some textile fibers, such as cotton, their cross-sections are very irregular, so it could be difficult to precisely decide these two effective curvatures. In addition, assumptions on the two principal flexural rigidities are needed (shown as \( R_{b1} \) and \( R_{b2} \) in Equations 7.3), for example, a specific ratio could be assumed between these two flexural rigidities\(^{(69)}\).

In the present computer simulations, it is assumed that the fibers have circular cross-sections and are isotropic, so the bending moment can be calculated using one inertia axis, which is projected at the same direction of the bi-normal direction of a point at the curve of the simulated fiber. Then the geometrical curvature of the fiber at that point can directly be used for the calculations, thus Equations 7.3 becomes:
\[ M_b = R_b \cdot \kappa \] (7.4)

\[ M_T = R_T \cdot \tau \]

For a space curve, its curvature \( \kappa \) at a point is the rate of turn of the tangent with respect to arc at that point\(^{(68)}\). The curvature is written as:

\[ \kappa = \left| \vec{r}' \times \vec{r}'' \right| \] (7.5)

in which \( \vec{r} \) is the position vector of that curve, \( \vec{r}' = dt/ds \), \( \vec{r}'' = d^2t/ds^2 \), \( s \) is the arc length.

In the computer model, \( \vec{r}' \) and \( \vec{r}'' \) can be written as discrete format as:

\[ \vec{r}'_i = \frac{\vec{r}_{i+1} - \vec{r}_{i-1}}{2\Delta s} \]

and

\[ \vec{r}''_i = \frac{\vec{r}_{i+1} - 2\vec{r}_i + \vec{r}_{i-1}}{\Delta s^2} \],

the directions of the above two vectors are shown in Figure 7.1 as \( \vec{a} \) and \( \vec{b} \) respectively. They are perpendicular to each other.

![Figure 7.1. Calculation of fiber curvature at a point](image)

Since \( \vec{r}' \) and \( \vec{r}'' \) are perpendicular to each other, the curvature at position \( i \) then can be written from Equation 7.5 as:

\[ \kappa_i = \left| \vec{r}'_i \times \vec{r}''_i \right| \] (7.6)

From Figure 7.1, \( |\vec{r}'| = \cos \frac{\theta}{2} \) and \( |\vec{r}''| = 2\sin \frac{\theta}{2}/\Delta s \), so \( \kappa_i = \frac{\sin \theta}{\Delta s} \), when \( \Delta s \) is small.
enough, the angle $\theta$ between the two neighboring positions is close to zero. In this case, there is:

$$\kappa_i = \frac{\theta}{\Delta s} \quad (7.7)$$

The space curve’s torsion $\tau$ at a point is the rate of turn of the bi-normal with respect to arc at that point\(^{(68)}\). In order to keep with the definition of the torsional rigidity in section 7.2, in the computer simulations the torsion at a point $i$ is calculated by using the following method as illustrated in Figure 7.2.

At first the bi-normal directions of the neighboring positions $i-1$ and $i+1$, shown in Figure 7.2 as $\vec{b}_{i-1}$ and $\vec{b}_{i+1}$, are projected to the normal plane of $i$, then the angle between these two projections is calculated as the rotation of the bi-normal vector about the tangent direction of position $i$ at the fiber curve. In other words, the change of angle between the two neighboring osculating planes projected on the normal plane of $i$ is considered as the angular change of the bi-normal vector about the tangent direction of point $i$.

After the above calculations about curvature and torsion at a point of the fiber curve, the bending and twisting moments are calculated by using Equation 7.4. Then they can be
integrated into the governing equations of fiber movement in airflow discussed in Chapter 5.

### 7.3.2. The Integration of Bending and Twisting Moments with the Governing Equations

Figure 7.2 illustrates the moments acting on the two ends of a fiber element, which is the same element as discussed in Section 5.2. In the configuration of the moments shown in this figure, the computer model does not assume that there is any moment distributed along the element.

![Diagram showing bending and twisting moments](image)

**Figure 7.3.** Bending and twisting moment acting on the fiber

When these bending and twisting moments are integrated into the fiber movement model discussed in Section 5.4, they can be totaled as the torque \( \tilde{M} \) acted on one end of the rod, as shown in Figure 7.4. \( \tilde{M} \) is projected to directions as \( \tilde{M}_a \) and \( \tilde{M}_b \). \( \tilde{M}_a \) is in the direction that is parallel to the rod’s centerline (\( \vec{L}_i \) direction), and \( \tilde{M}_b \) is at a plane that is perpendicular to the rod’s centerline, its direction parallel is to \( \vec{L}_i \times (\tilde{M} \times \vec{L}_i) \).

In Section 5.4, it has been proven that the force \( \vec{F}_{r2} \), which is at a plane that is perpendicular to \( \vec{L}_i \), equals zero, but with the integration of bending and twisting moments, \( \vec{F}_{r2} \) is not zero if \( \tilde{M}_b \) does not equal zero. Because the rod shown in Figure
7.4 has zero mass, for it there must be \( \vec{F}_{R2} \times \vec{L}_i = \vec{M}_b \). This means that static equilibrium of all torque on the rod must be satisfied; otherwise, the rod will gain infinite rotation acceleration. So from the above analysis:

\[
|\vec{F}_{R2}| = \frac{|\vec{M}_b|}{L} 
\]  

(7.8)

the direction of \( \vec{F}_{R2} \) is parallel to \( \vec{L}_i \times \vec{M}_b \). For equilibrium of the forces (see Section 5.4), the other end of the rod, shown in Figure 7.4 as the end connected to node \( i \), is acted on by a force that counteracts the \( \vec{F}_{R2} \).

![Figure 7.4. Forces and moments acting on one rod of the modeled fiber](image)

For each sphere connected to two rods, the force acting on it from each rod caused by bending and twisting has the same value as the force it exerted on the rod. They are expressed in the Equations 7.9:

\[
|\vec{F}_{hi-1,j}^{(n)}| = \frac{|\vec{M}_{hi-1,j}^{(n)}|}{L} 
\]  

(7.9)

\[
|\vec{F}_{hi+1,j}^{(n)}| = \frac{|\vec{M}_{hi+1,j}^{(n)}|}{L} 
\]
in which \( \vec{f}_{ibti}^{(n)} \) and \( \vec{f}_{bi+1,i}^{(n)} \) are bending and twisting action from rods \((i-1, i), (i, i+1)\) to node \(i\). Their directions are \( \vec{M}_{bi-1,i}^{(n)} \times \vec{L}_{i-1,i} \), \( \vec{L}_{i,i+1} \times \vec{M}_{bi,i+1}^{(n)} \), \( \vec{M}_{bi-1,i}^{(n)} \) and \( \vec{M}_{bi,i+1}^{(n)} \) have the same meaning as the \( \vec{M} \) in Equation 7.8, they acted on rods \((i-1, i), (i, i+1)\).

The effect of \( \vec{M}_a \) is not considered, because the rod is modeled as zero mass and zero diameter, the rotation about its own axis does not have significant physical meaning. In addition, for the model’s assumption that the moment of inertia is neglected, omitting \( \vec{M}_a \) from the governing equations could be deemed as an adjustment.

Along with Equation 7.9, the following Equations 7.10 replaces the Equations 5.11 in Section 5.5 and serves as the discrete governing equations integrated with fiber bending and twisting properties.

\[
\frac{m \Delta t}{\Delta t} \begin{align*}
\dot{y}_{ix}^{(n+1)} - y_{ix}^{(n)} &= f_{aix}^{(n)} + f_{i-1,ix}^{(n)} + f_{i+1,ix}^{(n)} + f_{bi-1,ix}^{(n)} + f_{bi+1,ix}^{(n)} \\
\dot{y}_{iy}^{(n+1)} - y_{iy}^{(n)} &= f_{aix}^{(n)} + f_{i-1,iy}^{(n)} + f_{i+1,iy}^{(n)} + f_b^{(n)} + f_g^{(n)} + f_{bi-1,iy}^{(n)} + f_{bi+1,iy}^{(n)} \\
\dot{y}_{iz}^{(n+1)} - y_{iz}^{(n)} &= f_{aix}^{(n)} + f_{i-1,iz}^{(n)} + f_{i+1,iz}^{(n)} + f_{bi-1,iz}^{(n)} + f_{bi+1,iz}^{(n)}
\end{align*}
\tag{7.10}
\]

To accommodate the change of governing equations, the other equations in the dimensionless format (Equations 5.17), the objective function (Equation 6.9) and the gradient functions (Equations 6.11) should be also modified.

In addition, it is necessary to point out that, before entering the simulated fiber transfer channel, the fibers are considered completely relaxed irrespective of their original shapes. This means that at the instant that the computation starts, there is no stress from bending or twisting in the fiber.
7.4 Results of Simulations with bending and Twisting Properties

Simulations for studying the effects on fiber movements, including the influence of fiber bending and twisting properties, were computed by using the program that is based on the governing equations integrated with bending and twisting moments. The flexural rigidities used for simulations are listed in Table 7.1 (a), the torsional rigidities used for simulations in Table 7.1 (b). All the simulations have the same fiber initial configurations and airflow field conditions as the simulation shown in Figure 6.7 (c).

In the initial simulations, the fibers’ bending and twisting properties were investigated separately, which means when bending properties were studied, the simulated fiber’s torsional rigidity was set as zero, and vice versa. Then a series of combinations of flexural and torsional rigidities were used for simulations. These combinations are shown in Table 7.2. The selection of these combinations was based on the observation on the data from earlier studies, which showed that the values of fibers’ torsional rigidities are always several times smaller than the flexural rigidities.

<table>
<thead>
<tr>
<th>Combination number</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexural rigidity ((\times 10^{-12} \text{ N m}^2))</td>
<td>5</td>
<td>10</td>
<td>15</td>
<td>20</td>
<td>25</td>
</tr>
<tr>
<td>Torsional rigidity ((\times 10^{-12} \text{ N m}^2))</td>
<td>2</td>
<td>4</td>
<td>6</td>
<td>8</td>
<td>10</td>
</tr>
</tbody>
</table>

Table 7.2. The combinations of Flexural and Torsional rigidities for simulations

From the results of the simulations, fibers’ straightening factor \(C_s\) (See section 6.3.1 for the definition), and configurations during the movements were studied. Figure 7.5 – 7.7 shows the variation of the fiber-straightening factor \(C_s\) with the change of fiber rigidities, these \(C_s\) were obtained after the simulated fibers moved in the fiber transfer
channel for 0.005 second. There is no significant change in $C_s$ found for the range of rigidities used in all the three situations simulated. Figure 7.8 – 7.9 show the snapshots of the fibers moving inside the channel. The very small differences between the fiber configurations shown in these figures also indicate that, in the simulated range, the flexural and torsional rigidities do not play an important role on the fibers’ pattern of movement in the channel.

Two possible reasons can be used to explain the findings of these simulations:

1. For a single textile fiber, its tiny diameter, 15 micron as in the present research, means that it can not have very large bending and twisting stiffness, and it can be considered to be nearly completely flexible.

2. In the modeled fiber transfer channel, the simulated moving time of a fiber is very short. As shown in Figure 7.8 – 7.9, the movements were only for 0.005 second. Thus if the rigidities are not very large, in very short time the effects of bending and twisting properties to the fibers’ movements maybe not revealed.

Figure 7.5. Fiber straightening-factor vs. fiber flexural rigidity
Figure 7.6. Fiber-straightening factor vs. fiber torsional rigidity

Figure 7.7. Fiber-straightening factor vs. different combinations of flexural and torsional rigidities
Figure 7.8. Simulated fiber movements with different flexural rigidities. The rigidities are (a): $5 \times 10^{12}$ N·m², (b): $15 \times 10^{12}$ N·m², (c): $25 \times 10^{12}$ N·m². In all these simulations, torsional rigidity was set as zero. The time interval between two snapshots in above figures is 0.0025 second; the time length is 0.005 second.
Figure 7.9. Simulated fiber movements with different torsional rigidities. The rigidities are (a): $2 \times 10^{-12} \text{ N}\cdot\text{m}^2$, (b): $6 \times 10^{-12} \text{ N}\cdot\text{m}^2$, (c): $10 \times 10^{-12} \text{ N}\cdot\text{m}^2$. In all these simulations, flexural rigidity was set as zero. The time interval between two snapshots in above figures is 0.0025 second; the time length is 0.005 second.
Figure 7.10. Simulated fiber movements with different combinations of flexural and torsional rigidities. The flexural and torsional rigidities are (a): $5 \times 10^{-12}$ N·m² and $2 \times 10^{-12}$ N·m², (b): $15 \times 10^{-12}$ N·m² and $6 \times 10^{-12}$ N·m², (c): $25 \times 10^{-12}$ N·m² and $10 \times 10^{-12}$ N·m². The time interval between two snapshots in above figures is 0.0025 second; the time length is 0.005 second.
The above results indicated that, inside the fiber transfer channel, for the range of values selected, the fiber’s bending and twisting properties do not have an important effect on its movement in the airflow field. They also prove that the assumption used for the simulated scenarios in the present research, i.e. that the simulated fiber is completely flexible, is reasonable. However, it does not mean that fiber’s bending and twisting properties can be neglected in all conditions. Figure 7.11 shows the simulation modeling a fiber that is moving inside the channel with extremely large rigidities, which are $1.0 \times 10^{-6}$ N·m² for flexural rigidity and $1.0 \times 10^{-9}$ N·m² for torsional rigidity. It can be seen that the fiber moves like a rigid body, with almost no deformation along the movement. The simulation means that large stiffness does affect the fiber movements and the model discussed in the previous Section 7.3 has the capability to model this situation.

![Figure 7.11. The fiber movement when it has extremely large rigidities](a) (b)

Figure 7.11. The fiber movement when it has extremely large rigidities

---

3 These are extreme values. They could not be associated with textiles fibers.
8. SUMMARY AND CONCLUSIONS

In the present research, studies have been carried out on the computer modeling of fiber movements in high-speed airflow. The research attempted to develop computer models that can simulate the interactions between textile fibers and the airflow fields of aerodynamic components in textile machines, and thus provide clearer understandings of the behavior of textile fibers inside the airflow fields.

Earlier research projects on fiber–airflow interactions in textile processes were surveyed. These processes include rotor spinning, air-jet spinning, air-jet texturing, and air-jet weaving. In all these textile processes, high-speed airflow plays crucial roles. An overview of research literature indicates that most of these earlier studies were focused on experimental investigations. Moreover, even though there were mathematical discussions, they were mostly qualitative. Numerical analyses, especially those on methods for computer modeling, are very few; and no research on three-dimensional modeling of real aerodynamic components was found. Based on the review of these earlier efforts, the different modeling methods were categorized into three approaches.

For studying the airflow fields in the present research, a commercial computational fluid dynamics (CFD) software package, CFD-ACE+ from the CFD Research Corporation, was used to simulate the airflow fields. The CFD software provides effective and powerful tools for computing, analyzing, and visualizing the studied airflow fields. The resultant data, such as the velocities at different positions, are used to integrate with a fiber movement model as an input by using one-way coupling approach, this
means that while there is air drag acting on every fiber, the interferences from fibers to
the airflow field is not considered.

A three-dimensional model of the fiber transfer channel in a rotor-spinning box was
constructed in the present research. Initially, one-dimensional pipe flow theory was used
to make qualitative estimations of the characteristics of the fiber transfer channel, and
then it was calculated by using the CFD software package under different conditions. The
results from these simulations yielded the following conclusions:

1. Under the initial conditions and boundary conditions for the simulations, the
   airflow speed along the channel’s centerline is accelerating. It reaches its
   maximum at the channel’s outlet cross-section; and the acceleration is mainly
   achieved in the second half of the channel.

2. Under the same initial and boundary conditions, the above maximum speed
decreases as the channel’s outlet size increases.

3. Within the range of parameters used in these simulations, the above maximum
   speed increases as the initial airflow speed at the channel’s inlet increases,
   provided other conditions are the same.

4. Within the range of parameters used in these simulations, the above maximum
   speed increases as the back pressure at the channel’s outlet decreases, provided
   other conditions are the same.

5. Under the initial conditions and boundary conditions for the simulations, an
   airflow re-circulation zone exists at the fiber transfer channel’s inlet. The structure
   of this re-circulation zone is three-dimensional. With higher initial inlet velocity,
   the re-circulation zone covers a larger space inside the channel. With lower
channel outlet back pressure, the re-circulation zone covers a smaller space or even does not exist.

For the three-dimensional model that simulates the fiber movements in the airflow field, basic governing equations have been established. In order to implement the governing equations to computations, the model for describing the fiber’s configuration was chosen. This model describes the fiber as a series of spheres that are connected by inflexible rods of zero diameters and zero mass. The dynamic equations of this model were studied and they were integrated into the basic governing equations. In consequence, a set of dimensionless equations were derived and applied to computation.

Numerical method for solving the above equations, which is non-linear, was developed based on non-linear optimization method. For the simulated cases, robust solutions have been obtained from this method. These simulations can be illustrated and animated by visualization programs that were developed using OpenGL. The results found can be summarized as:

1. The airflow in the simulated channel has an effect that straightens the fiber moving inside it.
2. The airflow has an effect that accelerates the fiber moving inside the channel. In the simulated speed range, fibers with higher initial speeds have accelerations that are not lower than fibers with lower initial speeds if other conditions are the same.
3. The fiber’s initial position at the airflow field affects its pattern of movement inside the channel; one of the reasons is the complex re-circulation zone at the channel inlet.
4. The airflow field’s own conditions have important influences on the simulated fiber’s pattern of movement in the channel. Lower channel outlet back pressure could result in a higher fiber acceleration.

Fiber bending and twisting properties were studied and integrated into the computer model. The numerical methods used to facilitate fiber flexural and torsional rigidities into bending and twisting moments were developed based on considerations of differential geometry. Then the governing equations, integrated with bending and twisting properties for fiber movements, were constructed. The results from the computations indicate that for the simulated scenarios, fiber bending and twisting stiffness do not have important impact on the fiber movement.

All the methods and results developed and obtained in the present research clearly demonstrate that the computer modeling is a practical, effective, and powerful tool for studying the fiber motion in high-speed airflow. It can elucidate the fiber-airflow interaction and provide better understanding of fiber behavior in airflow fields. It has a very good potential and prospect in the research of textile processes in which airflow plays important roles.
9. RECOMMENDATIONS FOR FUTURE WORK

For future research on computer modeling of fiber movements in airflow fields, the current model used for describing the fiber configuration, which is modeled as a series of spheres connected by rods of zero diameter and zero mass, could be developed into more complicated and sophisticated models. For example, it could be a model that describes the fiber as a series of rods of certain diameters. It is anticipated that the description could have a better integration with fiber bending and twisting properties along with the introduction of rotation inertia of the rods.

The model could be applied into more textile processes, such as air-jet spinning. For some of these processes, the assumption of one-way coupling used in this present research may not be very applicable, for example, for air-jet spinning, the yarn’s existence in the rotation chamber will make it necessary to consider its interference on the airflow field. This could require further assumptions regarding the airflow field’s configuration.

More challenging problems could be found in air-jet texturing process, in which continuous filaments, instead of staple fibers, are textured by very violent airflow, this could require dramatic changes of boundary conditions used for fibers in the present research. In addition, in many air-jet texturing processes, water is used. It is expected that this may largely affect the friction between the filaments and the airflow.

It is believed that the quality and accuracy of computer modeling could be improved by using more computing power. This is very important when the airflow field becomes more complicated and/or when more sophisticated fiber movement model is adopted. The simulation will be more accurate when more details about the airflow field of the
aerodynamic component of textile machines are computed. Dividing the fiber into more elements and decreasing the time step length will also make results from simulations more close to the fibers’ configurations in the real world. These could be especially important when fiber bending and twisting properties are considered.
10. REFERENCES


44. www.cfdrc.com


