KINEMATIC MODEL FOR YARN MOVEMENT IN TURBULENT AIR FLOWS

Philip Jungbecker, Gunnar Seide, Thomas Gries

Institut für Textiltechnik of RWTH Aachen University, Aachen, Germany,
Eilfschornsteinstr. 18, 52062 Aachen,
Tel: +49 (0) 241 80 95-621, Fax: +49 (0) 241 80 92-149,
philip.jungbecker@ita.rwth-aachen.de, gunnar.seide@ita.rwth-aachen.de, thomas.gries@ita.rwth-aachen.de

Abstract:

In the textile industry a tool is needed that can predict fibre and yarn movement in turbulent air streams. The Institut für Textiltechnik of RWTH Aachen University (ITA) has developed a yarn model that can be used to study the movement of single fibres and yarns in turbulent air flows. The kinematic model is described in this article. Special attention is paid to the aerodynamic forces that determine the flight path of fibres and yarns. The coefficient of drag tangential to the fibre axis \( c_t \) was studied thoroughly using computational fluid dynamics (CFD). It is shown that the diameter has a strong influence on the wall shear stress. Neglecting this effect for thin fibres can lead to errors in the coefficient of drag of a factor of 500. The turbulence intensity also has an important influence on the boundary layer development, which also determines the coefficient of drag. The assumptions made for the yarn model were tested in an experiment in which yarn flight paths were detected with a high-speed video camera. The comparison to the simulation results confirms the usability of the yarn model.

Keywords:

CFD, yarn model, simulation, boundary layer, coefficient of drag, coefficient of friction, high-speed video

Introduction

The demand for a yarn model that can predict the flight path and the deformation of fibres and yarns in turbulent air streams is growing in the textile industry. Current models assume a number of simplifications; in particular, the textile nature of the fibres and yarns is not taken into account. Such a model could help to analyse different kinds of processes in which fibres and yarns are influenced by the ambient air. Such processes can be air-jet weaving, air-jet texturing, or aerodynamic non-woven lay down. In these processes the air is used to guide the process. In other processes such as carding or ring spinning, fast moving machine parts induce an air flow that can heavily influence process stability.

At the Institut für Textiltechnik of RWTH Aachen University (ITA), a model was created to simulate fibre and yarn movement in turbulent air streams. One of the most important features of the model is its ability to represent different yarn and fibre types. It is for example possible to simulate the movement of a single staple fibre as well as a fluffy wool yarn. The interaction of the yarn flow and the air flow is important for applications in which the fibre material influences the air flow field. Of special interest are the aerodynamic forces acting on the fibres and yarns.

The complex boundary layer development around a fibre is determined not only by the flow velocity and fibre length but also by the fibre surface, diameter, and fibre shape. Analyses of a three-dimensionally crimped staple fibre have revealed that separation plays an important role for the boundary layer [1]. Especially at high Reynolds numbers, boundary layer separation was dominant. Depending on the local Reynolds number, the boundary layer was observed to be laminar or turbulent. Analyses of fibres in a compressible flow at up to supersonic speed show the boundary layer development and the change from a laminar to a turbulent boundary layer. Such high speeds occur in textile processes in which compressed air is used, such as air-jet weaving or air-jet texturing [2]. To verify the model, a test setup was built in which yarns were aerodynamically transported through a duct. The yarn movement was filmed with a high-speed video camera and compared to the yarn movement in the simulation. The measurements show a very satisfying correlation with the simulation results. The experimental setup and the results are presented in the following sections.

Figure 1. Different methods for the generation of mesh around a fibre. From left to right: body fitted mesh, immersed boundary layer, unchanged grid.
Model description

The model described in the previous section is based on a kinematic model that represents the fibre or yarn and the forces acting on it. All characteristics of the yarn that determine its flight path and the forces are set as parameters. Thus it is possible to use only one model to simulate all kinds of fibrous materials, such as staple fibres, yarns, and filaments.

Grid generation

For the simulation of fibres in a flow there are three different methods of creating a grid around a fibre. The three methods are illustrated in Fig. 1. Exemplary grid sizes are shown for arbitrary flow direction and velocity.

One method is to use a body fitted mesh: a fine mesh around the surface of the body. With this method the boundary layer of the object can be resolved very accurately. The determination of the flow characteristics near the surface is also favoured as the mesh is parallel to the flow. A flow separation can be predicted with high precision with this method, especially for turbulent flows. The fact that the fibre is deformed and also moves during the examination requires a moving mesh that is adapted after each time step. This method is inexpedient for industrial usage as the large number of volume cells around the fibres results in too much computational effort. In the area of science this method can be used to study the boundary layer of single fibres or yarns.

The second method is an immersed boundary layer approach as described by Lai and Peskin [3]. Here a Cartesian grid is refined in the region of the fibre. The square cells are subdivided into smaller units until the desired mesh size is reached around the fibre. This method is much faster than using a body fitted mesh and is also applicable to moving objects. For high Reynolds flows where the boundary layers become very thin this method produces improper values for the wall shear stresses.

At ITA the method of choice for the fibre model is a mesh that is untouched by the fibre position. The information about the underlying air flow field is imprinted on the fibre. The reacting force of the fibre on the air flow field must be merged with the original air flow field in a second step. A coupled solution can thus be achieved. A precondition for this method is that the forces on the fibre or yarn can be determined only from the air flow field and the fibre properties. Unlike the other two meshing methods, the boundary layer that is responsible for the wall shear stress is not resolved and must have been determined in previous experiments [4].

Kinematic model

One of the first fibre models to be used at ITA consists of fibres represented by spheres. The sphere diameter and its density were adapted to obtain a flow behaviour similar to that of fibres. The first simulations with this model showed good agreement with the measurements [5]. This model worked well as many fibres agglomerated and had roughly the form of a sphere. This model enables the simulation of fibre flight paths and fibre distributions as long as the fibres are much smaller than the flow volume. For simulations in which fibres move through a small duct or a narrow machine part, their shape and position are important for the flight path. In this case a fibre model is needed that represents the fibre as a long, slender, flexible object.

In contrast to the spherical fibre model, the current model by Seide [4] is characterised by a higher grade of complexity. In the kinematic model the fibres are represented by a chain of slender cylinders which are connected at their ends by massless joining elements. A cylinder can thus be defined by its diameter d, its mass m, and the position of its endpoints. A fibre or a yarn can be defined by a number of cylinders and interconnecting springs that reach from one joining element to the next but one. Each cylinder has a velocity and a force acting on it. The forces can be divided into inner and external forces. Inner forces are spring forces between the cylinders which define the bending stiffness of the fibre. External forces are the aerodynamic force, the gravitational force, and contact forces of walls or other fibres. Other forces such as Coriolis, Basset, Magnus, or Saffmann forces, pressure forces, and forces from virtual masses [6] can also be added but are neglected in further investigations. The forces for each cylinder are split into one component in the normal direction \( F_n \) and one in the tangential direction \( F_t \), as shown in Fig. 2.

The gravitational force as an external volume force is given by:

\[
F_g = m_{cylinder} \cdot g
\]

The internal force on each cylinder is the sum of two spring forces and two damping forces. The spring forces are proportional to the spring compression, and the damping forces are proportional to the bending velocity. Those forces determine the bending stiffness and the damping behaviour.

Figure 2. Fibre represented by a chain of cylinders and the force components on the cylinders.
of the yarn. The bending stiffness of fibres is described in detail by Morton and Hearle [7] and the research group around Kawabata [8]. It must be mentioned that the springs and damping elements are used only for the bending of the fibres and not for the stretching behaviour. If necessary the stretching can be calculated in a second step once the forces for each element are known.

The aerodynamic force $F_{\text{aero}}$ is composed of one component that is normal to the cylinder axis, $F_{\text{aero,n}}$, and one component tangential to it, $F_{\text{aero,t}}$. The normal and tangential components can be written as:

$$F_{\text{aero,n}} = \frac{\mu_0}{A} \cdot \frac{\rho}{2} \cdot d \cdot l \cdot |\dot{u}|^2 \cdotp c_n$$

$$F_{\text{aero,t}} = \frac{\mu_0}{A} \cdot \frac{\rho}{2} \cdotp \pi \cdotp d \cdotp l \cdot |\dot{u}|^2 \cdotp c_t$$

The velocities $u_0$ and $u_1$ are the relative velocities between the fibre and the air flow field, in the normal and tangential directions respectively. $c_n$ is the coefficient of drag that is normal to the fibre, $c_t$ is the corresponding tangential coefficient, $\rho$ is the air density, and $d \cdot l$ and $\pi \cdot d \cdot l$ are the effective surface areas for the normal and the tangential components respectively. It is important to note that the coefficients $c_n$ and $c_t$ are not constant values but depend on the velocity, fibre length, fibre diameter, and turbulence intensity.

Other external forces such as wall or fibre contact forces depend on the intensity of rebound and frictional forces. The wall contact method chosen is the penalty force method. This method tracks elements which have moved beyond the flow bounds after a time step. Additional forces act on the according element that is accelerated towards the flow volume. After this operation the element is again inside the flow volume but has a new velocity.

The sum of all forces acts on the centre of the cylinder. From the forces the new cylinder positions can be derived by Newton’s 2nd law. As only one resultant force acts on each cylinder, it is not possible for a single cylinder to rotate. In Fig. 3 there are three fibre elements on which different forces act. Considering the same initial velocity for all three cylinders, the displacement is proportional to the resultant force on the cylinders. In a first step all cylinders are translated separately in the direction of the resultant force. This results in a state in which the cylinders are not connected. In a second step the cylinders are again translated and rotated to reconnect them. This is done by an algorithm that is explained in [9].

The reconnecting cylinder method does not include the rotational energy conservation of the elements. If an appropriate time step size is chosen, for example, an element rotation smaller than some degree per time step, this method is accurate as the translational energy of the elements is usually larger than the rotational energy. Therefore it must be considered that the rotational energy is indirectly expressed by the translational energy of the elements as shown in Fig. 3. In a first step this method was chosen for its simple implementation.

**Coefficients of drag, $c_n$ and $c_t$**

The aerodynamic forces on a fibre or yarn are dominant for their flight path. Thus special attention has to be paid to the determination of the coefficients. It is easily possible to describe a single fibre or a monofilament as a slender cylinder. The coefficient of drag has been studied well for a cylinder in a cross flow for a wide range of Reynolds numbers. Figure 4 shows the dependency of the coefficient $c_n$ according to different authors [10], [12]. Especially in the area where low Reynolds numbers occur for fibres, the divergence between the two authors is evident. For example, a relative velocity between air and fibre of 0.1 m/s and a fibre diameter of 15 µm would lead to a Reynolds number of 1.

![Figure 4. Coefficient of drag $c_n$ for a cylinder in a cross flow according to different authors.](http://www.autexrj.org/No4-2008/o299.pdf)
simulations of different yarn diameters and different velocities have revealed that the influence of the diameter is significant especially in the region of fibre and yarn diameters. A flat plate was simulated by setting the diameter to larger values. For values over 10 mm it is possible to use the simplification of a flat plate for a cylinder. Neglecting the effect of small diameters below this diameter can lead to errors in the coefficient of drag of a factor up to 500. A consequence is a drag force that is too small. The cause of the strong influence is the curvature of the boundary layer, as already studied by Bui and Cebeci [17]. The diameter can be divided into three sections, where \( d \) is the boundary layer thickness:

\[
d_{\text{aero}} \gg d \quad \text{As long as the boundary layer thickness } d \text{ around a cylinder is very small compared to its diameter, the curvature of the boundary layer is negligible. The aerodynamic force is proportional to the plate width or the cylinder diameter. } c_t \text{ is a constant value. For an infinite sized cylinder the assumptions for a flat plate can be applied.}
\]

\[
d_{\text{aero}} \sim d \quad \text{When the boundary layer thickness } d \text{ becomes of the same order as the cylinder diameter, the influence of the diameter on the coefficient of drag } c_t \text{ is significant. The ratio of the aerodynamic force to the surface area of the cylinder decreases with decreasing diameter. Therefore } c_t \text{ is a function of } d_{\text{aero}}^{-1}.
\]

\[
d_{\text{aero}} \ll d \quad \text{When the diameter becomes significantly smaller than the boundary layer thickness, a further decrease in } d_{\text{aero}} \text{ does not result in a higher wall shear stress. } c_t \text{ is a constant value again but it is some orders higher than in the case where } d_{\text{aero}} \gg d.
\]

In Fig. 5 the dependence of \( c_t \) on \( d_{\text{aero}} \) is shown for an example of one velocity setting and fibre length.

![Figure 5. Coefficient of drag \( c_t \) for a cylinder against cylinder diameter.](http://www.autexrj.org/No4-2008/0299.pdf)

**Influence of a turbulent boundary layer on the coefficient of drag \( c_t \)**

Further simulative investigations were done to study the coefficient of drag \( c_t \) at high velocities. High velocities of up to 600 m/s are used in different textile processes in which pressured air guides the process. Some examples are air-jet texturing, air-jet weaving, and air-jet spinning. The differences from low velocity flows are the compressibility of air and the turbulent boundary layers. For low velocities of up to 20 m/s, a laminar boundary layer was observed for the whole fibre or yarn length. At higher velocities, depending on the diameter and the length, a point was found at which the boundary layer changed from laminar to turbulent. This point was found at different positions on the yarn depending on the turbulence intensity. At low intensities the wall shear stress decreases with the distance from the leading edge. When the boundary layer becomes turbulent the wall shear stress increases and stays almost constant until the end of the yarn. At high turbulence intensities the beginning of the turbulent boundary layer begins much earlier. The wall shear stress does not increase again but with the beginning of the turbulent boundary layer the decrease of wall shear stress is slower. A long distance away from the leading edge, the wall shear stresses for a low turbulence flow and for a high turbulence flow converge. Wall shear stresses depending on the distance from the leading edge and the turbulence intensity are shown in Fig. 6.

![Figure 6. Wall shear stress on a fibre depending on distance from leading edge and turbulence intensity](http://www.autexrj.org/No4-2008/0299.pdf)

**Experiments**

**Experimental setup**

The theoretical model was verified by experiments. A setup was chosen in which yarn movement could be observed. A non-uniform air flow was needed so that different forces would act on the yarn during its flight. A square duct was chosen that made a 90° turn, as shown in Fig. 7. From the air and fibre inlet a 200 mm long duct leads to the sharp corner. Behind this corner a recirculation zone develops. The air was vacuumed out of the duct. This setup ensures a steady turbulent flow in the first part of the Re duct ranging from 10,000 to 35,000 depending on the volume flow. The strong direction change of the flow and the recirculation zone are the areas in which non-uniform forces act on the yarn. Due to the inertia of the yarn, it does not follow the streamlines but has a wall contact, that is another external force deflecting the fibre. The fibre movement was observed in the area of the corner with a high-speed video (HSV) camera. The duct was made of Plexiglas with the rear wall taped with black foil so that the fibres were visible. A frame rate of 9000 1/s was used to detect the fibre motion.

Three different yarn types and one type of fibre were investigated. The fibres were too light and stuck to the wall once they had wall contact. The yarns were a 178 tex ePTFE monofilament, a 7.31 tex cotton ring yarn with little hairyness, and a 700 dtex f140 PA 6.6 multifilament yarn. The yarns were cut into 30 mm long pieces and released at the inlet. The best results in terms of interesting yarn movement were observed with the ring yarn. The two other yarns were too stiff,
so that their shape was only slightly changed even at wall contact.

All yarn pieces reached the mean air velocity when they moved into the observed area. Due to their inertia they had intensive wall contact and rebounded. It was observed that many yarns partly moved into the recirculation zone. Here the low velocities were in the opposite direction to the mean flow. The yarn part that was in the free stream experienced an aerodynamic force that led to a rotation of the yarn. Some yarns even did a double rotation before they left the observed area.

**Simulation of the air flow and the yarn movement**

Before the simulation of the yarn movement could be performed, the air flow field in the duct had to be simulated. The mean air velocity in the duct could be determined from measurements with an anemometer. The volume flow that was derived from the measurements served as the outlet boundary condition. The air flow field was used to determine the forces on the yarns. A coupled solution was not needed here as the volume fraction of the yarn was negligible. The fibre properties of the yarn were entered into the yarn model. The simulated air flow field is shown in Fig. 8.

The simulation showed some weaknesses at wall contact. Here the bouncing behaviour could not be depicted correctly, so the subsequent yarn flight paths also did not match the measurements. Further simulations were performed that started just after the first wall contact. The yarn was given the position and the initial velocity taken from one of the HSV measurements. In this case the results of the simulation of the yarn rotation were in very close agreement with the experimental measurements, as shown in Fig. 9.

**Conclusions**

Comparison of the experimental results with the simulation results confirms the usability of the yarn model. Although the experimental setup did not represent a process step in textile production it was possible to test and to verify the yarn model.

Hence it will also be possible to apply the yarn model to any process in which fibres or yarns are processed by an air stream. Even the event of wall contact can be depicted. Through the open structure of the kinematic model it is possible to apply additional forces on the yarn.

Further investigations are in progress that enable the simulation of different yarn characteristics. In this way it is possible to include even more textile properties that influence the coefficients of drag $c_d$ and $c_l$.

Application of the yarn model to different textile processes is also in progress. Utilisation of the model can help to better understand the processes and to improve process stability, efficiency, and product quality.

**References:**


http://www.autexrj.org/No4-2008/0299.pdf


Received: 7.1.2008 Reviewed: 15.01.2009