TIME-DEPENDENT CALCULATION OF THE VELOCITY OF A YARN LAUNCHED BY THE MAIN NOZZLE OF AN AIR-JET LOOM

Delcour Lucas[1], **Peeters Jozef**[2], **Van Langenhove Lieva**[3], **Degroote Joris**[1, 4]
[1] *Department of Flow, Heat and Combustion Mechanics, Ghent University, Ghent, Belgium*
[2] *Picanol NV, Ieper, Belgium*
[3] *Department of Materials, Textiles and Chemical Engineering, Ghent University, Ghent, Belgium*
[4] *Flanders Make, Ghent, Belgium*
lucas.delcour@ugent.be

ABSTRACT

In air-jet weaving looms the yarn is initially accelerated by the main nozzle. To obtain a high yarn velocity a high air velocity is required which results in complex flow patterns. Consequently, predicting the influence of a change in geometry or inlet pressure on the yarn velocity is not straightforward. In this research a fast time-dependent fluid-structure interaction framework is used to model the acceleration of a yarn during launch. Initially, the performance of the framework is assessed by considering a smooth monofilament yarn. A suggestion is also made and tested to deal with the surface texture of hairy/multifilament yarns.

Key Words: air-jet weaving, main nozzle, computational fluid dynamics, fluid-structure interaction

1. INTRODUCTION

In air-jet weaving looms the yarn velocity (and thus the picks per minute of the machine) largely depends on the air flow inside the main nozzle. One of the first studies about the dynamical behavior of a yarn inside the main nozzle was performed by Uno [1]. His study relied on an empirical model for the air velocity distribution and the use of force coefficients to obtain the forces on the yarn. In his model the yarn motion was restrained to axial motion along the centerline and the yarn itself was considered inextensible. Similar approaches were adopted by e.g. Adanur and Mohamed [2, 3], comparing drum and loop storage, and Celik and Babaarslan [4] who also accounted for the reed, relay nozzles and a stretching nozzle.
However, accurate models for the air velocity distribution are hard to obtain, especially when one wants to consider variations of the geometry or inlet pressure. Using computational fluid dynamics (CFD) one can make accurate predictions of the flow and even obtain more data than can be obtained experimentally. Furthermore, it has been shown that the value of the force coefficient varies substantially with the relative air velocity [1, 5, 6]. Tabulating the data for the force coefficient as a function of relative air velocity is a delicate process.
For a smooth contact surface CFD allows the calculation of forces on that surface based on the law of the wall, which is more general than the use of force coefficients. For rough surfaces this law of the wall is adapted based on a roughness height. Generally, adding a roughness will increase the frictional force at the contact surface. Although this wall roughness model was initially developed for pipe flow [7], in this research the same concept is applied to alter the frictional force experienced by the yarn in the simulation. Research using CFD to evaluate the main nozzle flow has for example been performed by Oh, Kim, and Song [8], Kim et al. [9] and Osman [10].
In the case of a yarn launched by a main nozzle, the motion of the yarn depends on the flow but the yarn motion also influences the flow. When the mutual interaction between a structure (in this case the yarn) and a flow (in this case the main nozzle air flow) is taken into account in a simulation it is referred to as a fluid-structure interaction (FSI) simulation. Typical FSI simulations consider a flexible structure as was for example done by Wu et al. [11] and Osman,

Malengier et al. [12]. Taking into account the flexibility of the structure implies an increase in computational time as typically a 3D flow solver and a separate structural solver are required. In this research the yarn is considered to be rigid and to move only axially along the centerline of the main nozzle. The velocity of the yarn is calculated based on the fluid force (obtained from the flow solver) and an inertial force. The inertial force is obtained from the structural solver, which in this case corresponds to a few lines of code in the CFD software.

2. EXPERIMENTAL SETUP

A sketch of the experimental setup is displayed in Figure 1. The setup consists of a main nozzle supplied from a 5-litre pressure reservoir. The yarn is initially stored in a yarn supply device which was selected to minimize yarn ballooning and friction so that these forces can be neglected in the structural model. A high speed camera is used to record the yarn movement at the main nozzle inlet. Regularly spaced black markings are applied to the yarn so that a yarn velocity can be retrieved from the high-speed footage.

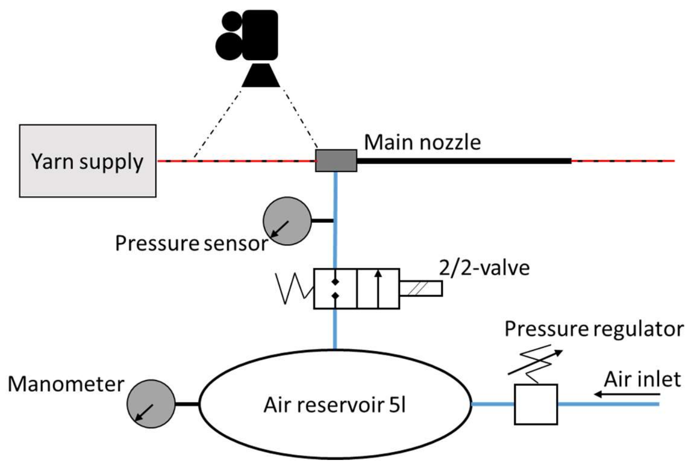

Figure 1: Sketch of the experimental setup.

3. METHODLOGY

As mentioned in the introduction, FSI simulations are performed in this research. This implies that both structural and flow equations are solved with mutual interaction. Simplifications are applied to both the structural and flow model to reduce the computational cost as much as possible while maintaining reasonable accuracy.

3.1 Flow model

A 2D-axisymmetric model is used for the flow simulations. A sketch of the flow domain with the applied boundary conditions and the mesh is provided in Figure 2. At the pressure inlet (which corresponds to the air supply of the main nozzle) a time-dependent total pressure is imposed corresponding to the one measured by the pressure sensor in the experimental setup (Figure 1). The “moving wall” represents the yarn wall and is thus offset from the axis. The yarn velocity, as obtained from solving the structural equations, is imposed at this wall. The main nozzle itself is about 30 cm long and the domain extends 0.5 m downstream of the nozzle end.

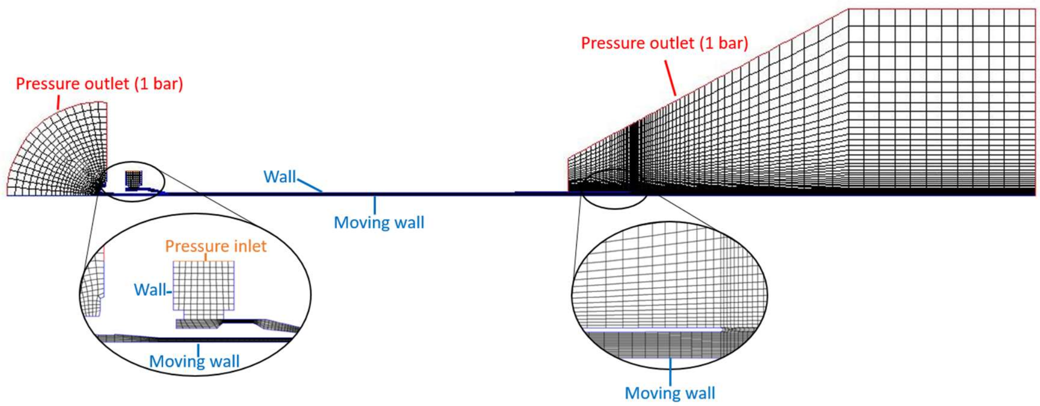

Figure 2: Mesh of the flow domain and its boundary conditions.

Due to the high speed of the flow, the compressiblity of the fluid has to be taken into account and a small time step size has to be used (5e-06 s). A pressure-based solver is employed and turbulence is modelled using the k-ω SST model. The flow simulations are performed in Fluent 17.2.

3.2 Structural model

The structural model takes as an input the force exerted by the fluid on the yarn and calculates a yarn acceleration. A representation of the structural model is displayed in Figure 3.

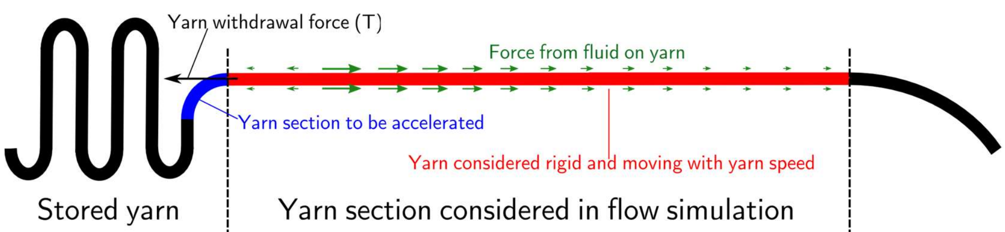

Figure 3: Representation of the structural model.

The structural model takes into account 3 contributions to Newton's second law of motion (discretized in time):

1. The inertia associated to acceleration of the yarn from time t to time $t + dt$
2. Acceleration of a new section of yarn (located in the storage device at time t, blue section in Figure 3) to the current yarn velocity. This is referred to as the yarn withdrawal force.
3. The force exerted by the fluid (air) on the yarn. This force will depend on both the velocity of the yarn and the flow field.

3.3 Coupling

In previous research it was verified that an explicit coupling suffices for the case at hand [13]. This implies that in each time step the flow and structural equations are solved once.
The structural equation (discretized in time) can be represented as:

$$m|_{t+dt} \cdot x_{dd}|_{t+dt} = F - \rho_A x_d^2|_{t+dt} \quad (1)$$

with:

t = current time

dt = time step

x_d = yarn velocity

x_{dd} = yarn acceleration

ρ_A = yarn linear density [kg/m]

F = force from the fluid on the yarn

The term $\rho_A x_d^2|_{t+dt}$ corresponds to the yarn withdrawal force mentioned previously. As can be seen from Formula (1) the yarn mass is considered to be time dependent; the mass varies as more yarn is set into motion.

4. RESULTS

The yarn velocities as obtained from the simulations are compared to those observed in the experiment. Two different nozzles are considered. Both nozzles have a conical jet entry and a conical acceleration tube but differ in terms of throat area, conicity and diameter. These nozzles will be referred to as CON1 and CON2. For both nozzles the experiment is performed once with an inlet pressure of 3 bar gauge and once with an inlet pressure of 5 bar gauge.

To perform the simulations, initial conditions have to be specified. The initial velocity and acceleration of the yarn are set to 0. The flow field is initialized with a velocity of 0 m/s, a uniform pressure of 1 bar and a uniform temperature of 300 K. Additionally, a value for the yarn mass [m in formula (1)] at time 0 has to be specified. In the simulation this initial mass was chosen so that it corresponds to the yarn length in between the nozzle exit and the yarn storage device multiplied by the yarn linear density. The mesh and time step sensitivity study have been reported in previous research [13].

The FSI simulations require, on average, 13 hours of computational time on a single core of a 12-core Intel Xeon E5-2680v3 2.5 GHz CPU. The computational time could be decreased by parallelizing the code and running the simulation on multiple CPUs.

4.1 Polymer coated yarn

At first, experiments and simulations are performed for a thin, monofilament, polymer coated yarn. Because the surface of this yarn is rather smooth, its implementation in the flow solver should not require any modification or tuning to obtain the correct fluid forces. The results of the experiments and the simulations are displayed in Figure 4. The experimental values are represented by a stepwise plot as only a limited number of data points is available. The polymer coated yarn has an average diameter of 0.21 mm and an average linear density of 76 tex.

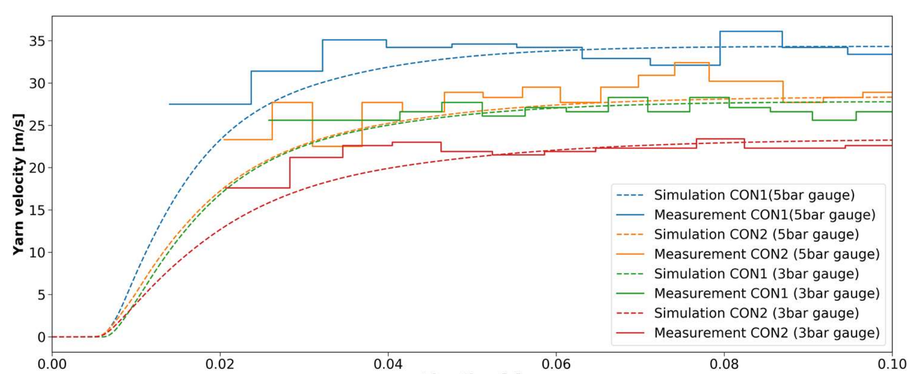

Figure 4: Experimental and simulated yarn velocity for a polymer coated yarn.

Figure 4 illustrates that the simulated velocity corresponds rather well to the experimental values. It can be observed that the simulated yarn accelerates somewhat slower in the beginning. This acceleration is, however, largely determined by the choice of the initial mass. Currently, the simulation considers the entire section in between the nozzle exit and the storage device as a rigid body that has to be accelerated. In reality the yarn has some elasticity and will also sag a bit.

4.2 Cotton yarn

The employed cotton yarn has an average diameter of 0.5 mm and an average linear density of 100 tex. Contrary to the polymer coated yarn, the cotton yarn does not have a smooth surface. For a similar flow velocity this will increase the force exerted by the fluid on the yarn. In order to accommodate for this increased force the yarn wall is assigned a roughness height in the simulation.

The value of the roughness height is considered to be a property of the yarn. Its value is determined based on a single experiment. In Figure 4 it can be seen that the yarn velocity tends to reach a constant value. To determine the appropriate value for the roughness height, a section of yarn is taken from the bobbin and launched as before. From the experiment its regime velocity is determined. A steady-state variant of the current simulation setup is then used. Such a steady-state simulation takes less than 5 minutes on a single core. The value of the roughness height is then tuned until the simulated yarn velocity matches the one that was experimentally determined. The same value is then used for all other simulations using that type of yarn. For the cotton yarn considered here a value of 0.074 mm was determined by tuning the simulation to the experiment on the CON1 nozzle at 5 bar gauge.

For the polymer coated yarn it was observed that an initial mass corresponding to the yarn length in between nozzle exit and yarn supply device resulted in a too slow acceleration of the yarn. Therefore, for the cotton yarn simulations, the initial mass was chosen to correspond to the length in between the nozzle entrance and the yarn supply device. In Figure 5 the results of these simulations are compared to experimental data.

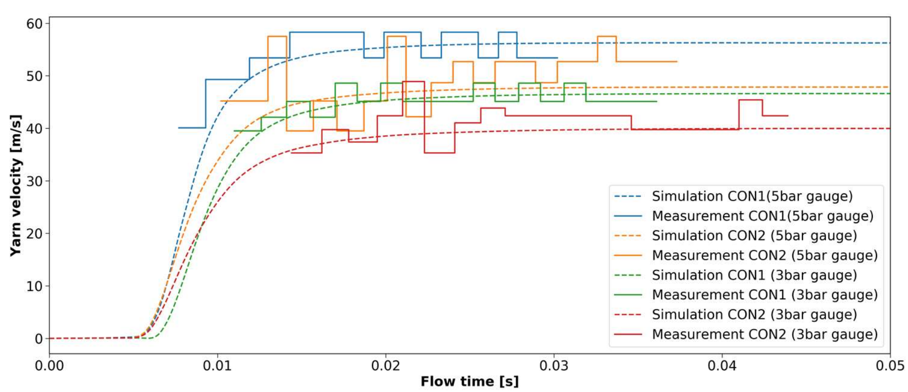

Figure 5: Experimental and simulated yarn velocity for a cotton yarn.

It should be remarked that at higher yarn velocities the experimental method to determine the velocity becomes more prone to error as it is based on video analysis with a fixed framerate. Additional data points to obtain a smoother experimental curve are desired to allow for a better comparison with the experimental data. Nevertheless, the results agree reasonably well with the experiment. It can also be seen that the new choice of initial mass yields better results for the acceleration of the yarn. As the yarn velocity is also matched quite well for the second nozzle and when altering the inlet pressure, it can be stated that yarn surface texture can to some extent be treated as a wall roughness in CFD.

5. CONCLUSION

In this research FSI simulations, with a low computational cost, were performed to calculate the yarn velocity as it is launched by a main nozzle. Overall, reasonable agreement with experimental data was observed, but the experiments could benefit from a more accurate method to determine the yarn velocity.
For yarns with a smooth surface no special measures need to be taken in the simulation. For hairy and/or multifilament yarns the surface texture needs to be incorporated into the simulation. To do so the authors proposed to alter the law of the wall in the CFD by treating the yarn wall as a rough surface. The roughness parameters can be tuned based on a single experiment and can then be considered to be a property of the yarn.

6. REFERENCES

1. Uno M. A Study on Air-Jet Loom with Substreams Added, *Journal of the textile machinery Society of Japan,* 1972, Vol.25, 48-56.
2. Adanur S. and Mohamed M.H. Analysis of yarn tension in air-jet filling insertion, *Textile Research Journal,* 1991, Vol.61, 259-266.
3. Adanur S. and Mohamed M.H. Analysis of yarn motion in single-nozzle air-jet filling insertion, *The Journal of the Textile Institute,* 1992, Vol.83, 45-55.
4. Celik N. and Babaarslan O. A mathematical model for numerical simulation of weft insertion on air-jet weaving machine, *Textile Research Journal*, 2004, Vol.74, 236-240.
5. Anderson S.L. and Stubbs R. Use of air currents for tensioning fibres, *Journal of the Textile Institute Transactions*, 1958, Vol.49, 53-57.
6. Selwood A. The Axial air-drag of Monofilaments, *Journal of the Textile Institute Transactions*, 1962, Vol.53, 576.
7. Moody L.F. Friction factors for pipe flow, *Transactions of the American Society of Mechanical Engineers*, 1944, Vol.66, 671-681.
8. Oh T.H., Kim S.D., and Song D.J., A numerical analysis of transonic/supersonic flows in the axisymmetric main nozzle of airjet loom, *Textile Research Journal*, 2001, Vol.71, No.9, 783-790.
9. Kim H.-D., Lim C.-M., Lee H.-J. and Chun D.-H. A study of the gas flow through air jet loom, *Journal of Thermal Science*, 2007, Vol.16, No.2, 159-163.
10. Osman A. *Computational fluid-structure interaction study of the aeroelastic behavior of a wire in transonic and supersonic flows* (Doctoral dissertation), Ghent University, Ghent, 2017.
11. Wu Z., Chen S., Liu Y., and Hu X. Air-flow characteristics and yarn whipping during start-up stage of air-jet weft insertion, *Textile Research Journal*, 2016, Vol.86, 1988-1999.
12. Osman A., Malengier B., De Meulemeester S., Peeters J., Vierendeels J. and Degroote J. Simulation of air flow-yarn interaction inside the main nozzle of an air jet loom, *Textile Research Journal*, 2017, Vol.88, 1173-1183.
13. Delcour L., Peeters J., Degroote J. and Vierendeels J. Fast computation of a smooth yarn's velocity in a main nozzle, *Proceedings of the 6th European Conference on Computational Mechanics (Solids, Structures and Coupled Problems) [ECCM 6] and the 7th European Conference on Computational Fluid Dynamics [ECFD 7]*, Glasgow, 2018, 985-996.