An optimum design study of interlacing nozzle by using Computational Fluid Dynamics

Makhsuda Juraeva, Kyung -Jin Ryu, Sang Dug Kim and Dong Joo Song

School of Mechanical Engineering, Yeungnam University, 214-1 Dae-dong,
Gyeongsan, Gyeongbuk, Korea, 712-749
Phone: (82)-53- 810-2449
Email: djsong@yu.ac.kr

Key words: CFD parallel computation, interlacing nozzle

ABSTRACT

Air interlacing serves to protect the yarn against damage, strengthens inter-filament compactness or cohesion, and ensures fabric consistency. The air interlacing nozzle is used to introduce intermittent nips to a filament yarn so as to improve its performance in textile processing. The effect of various interlacing nozzle geometries on the interlacing process was studied. The geometries of interlacing nozzles with single or multiple air inlets located across the width of a yarn channels are investigated. The basic case is the yarn channel, with a perpendicular main air inlet in the middle. Other cases have main air inlets, slightly inclined double sub air inlets. The yarn channel cross sectional shapes are either semicircular or rectangular shapes. The compressed impinging jet from the main air inlet hole hits the opposing bottom wall of the yarn channel, is divided into two branches, joins with the compressed air coming out from sub air inlet at the bottom and creates two free jets at both ends of the yarn channel. The compressed air movement in the cross-section consists of two opposing directional vortices. The CFD-FASTRAN flow parallel solver was used to perform steady simulations of impinging jet flow inside of the interlace nozzles. The vortical structure and the flow pattern such as pressure contour, particle traces, velocity vector plots inside of interlace nozzle geometry are discussed in this paper.

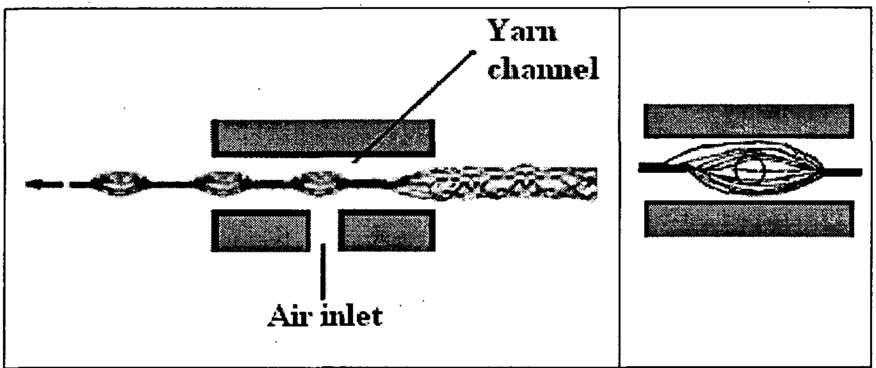
Introduction

Interlacing is the process of yarn treatment, at the beginning of the process, filament yarns are flat and the filament bundle is completely open. The individual filaments of the yarn, while entering the air interlacer, are separated by the airflow and are set in rotation as shown in Figure1. Through this an accumulation of false-twist sections is created at the entrance as well as at the exit of the compressed air coming out from the air inlet, and these sections are called knots, or nodes. As the interlaced yarn is moving through the compressed air, the formed knot moves towards the air inlets and, consequently, the rotation of the filaments is stopped. This process repeats itself periodically.

Interlacing influences the yarn structure and it is one of the advantage of interlacing. Schubert [1] found that a single air inlet leads to a higher nip frequency than multiple air inlets with the same air consumption. He found that an oblique air inlet increases nip frequency but disturbs yarn tension. Most researchers used simple air inlet configuration as a starting point to improve nip frequency.

There are different air interlacing techniques, tangling or intermingling, soft-interlacing, continuous interlacing.

A new Y- interlacing technology is based on special flow air inlet geometry and its location in the air inlet. The two small side branches of the Y-cross sectional air flow transport the yarn into the middle of the channel to the front of the main air inlet in the interlacing zone [2].



a) Side view | b) Top view

Figure 1. Yarn channel with single air inlet

In case of high performance interlacing jets and an average Polyester filament dpf (Denier per filament) of 3, up to 2,200 knots per second are formed. In case of micro-filaments this may be even 3,500 knots per second or more. Air interlacing provides assurance in the downstream performance of the yarn in weaving, knitting or warp knitting, without changing the other yarn's properties.

Computational approach

The CFD-GEOM provides the geometry and modifies the geometry read through IGES or Plot3D data formats [4]. CFD-GEOM is used for initial problem set-up, definition of the prescribed motion of the geometry and structure grid topology of the problem, the geometry manipulation, and grid. CFD-FASTRAN is a parallel, implicit, Euler/Navier-Stokes Flow solver. This code is used to solve the high speed, highly separated flow problems. CFD-FASTRAN capabilities represent the culmination of the growing experiences of developing advanced simulation software and associated physical models [5]. CFD-VIEW is the 3D graphical post-processor that analyze CFD [6]. One of the challenges in computational modeling is that each simulation generates a large volume of data that must be reduced to extract useful information that can be applied to practical science and engineering problems. CFD-VIEW uses surface-based visualization. It can display one or more computational planes, cutting planes, walls, vector quantities by mapping small vector arrows on surface, particle tracing, surface contours, flooded contours and point probe.

Figure 1 shows the various computational tools used in this approach. The results shown below for each case are creased using CFD-VIEW.

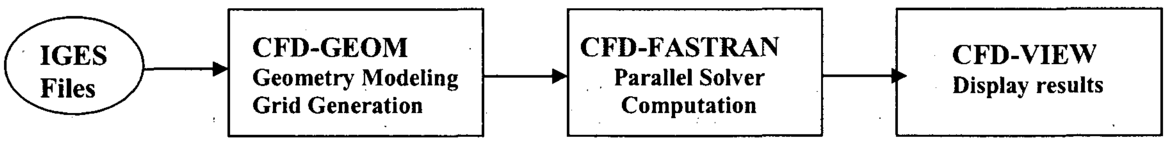


Figure 2 Overview of Computational approach

Results and Discussion

The computational results of four cases are presented. The width and depth of yarn channel and the width of the sub air inlets are chosen case by case. Vortices of the compressed air movement are investigated. When yarn channel has main air inlet, the compressed air coming out from the air inlet hits the opposing wall of the yarn channel, divides into two branches, and creates two free jets at both ends of the yarn channel. If there are doubled sub air inlets the compressed air coming out from main air inlet joins with the compressed air coming out from the sub air inlets and creates vortices in the yarn channel. These directional vortices are important to improve nip frequency and nip regularity. The first case has single (main) air inlets and it is used as basic interlacing nozzle to develop new interlacing nozzles.

Conclusions

The compressed air coming out from main air inlet hits the opposing wall of the yarn channel, divides into two branches, and forms vortex motion for case 1.

In case 2, the compressed air coming out from sub air inlet reduces the compressed air coming out from main air inlet and makes sliding vortex shape. The sub air inlets are quite big and not near to the side wall of yarn channel.

If we compare with case 1, in case 3 the compressed air coming out from sub air inlets moves flow to the center of the yarn. We can observe the compressed air coming out from sub air inlet flows strongly through the side wall of the yarn channel and creates high vortex motion then case 2. If we compare the vorticity strength of cases 3 and 4 the vorticity strength of case 4 is relatively smaller than case 3.

Case 4 has the inclined air inlet which guides flow to go through the side wall of yarn channel. The compressed air coming out from the sub air inlet hits the opposing side wall of the yarn channel and makes vortical motion. The right side of the sub air inlet has dead zone.

REFERENCES

[1] Schubert, G., Vortexting of Flat Yarns, Chemiefas, Textilind. 30/82(10), 820-823. E92-93 (1980)

[2] M. A. Zenses, *Y and CI air jets – air interlacing technology for different textile processes*, Chemical fiber international, Volume 53, February 2003.

[3] Spalart, P.R. and Allmaras, S.R. "A one-equation turbulence model for aerodynamic flows". *AIAA Paper 92-0439*, American Inst. of Aeronautics and Astronautics, 1992.

[4] CFD-GEOM Version 2004 User's Manual, CFD Research Corporation, Huntsville, AL

[5] CFD-FASTRAN Version 2002, User's Manual, CFD Research Corporation, Huntsville, AL

[6] CFD-VIEW Version 2002, User's Manual, CFD Research Corporation, Huntsville, AL