텐터기 노즐의 최적설계를 위한 수치해석적 연구
An Optimum design study of nozzle for tenter machine

쥬레바막슈다, 김용대, 박시우, 이기풍, 전두환[1]

(재)한국섬유기계연구소 , [1]영남대학교 섬유패션학부

Abstract

텐터의 성능을 결정하는 요인은 여러 가지가 있으나, 섬유의 종류 또는 가공 공정의 목적에 따라 적절한 건조속도 및 효율성, 원단의 전 폭에 걸친 건조 균일도, 습윤 공기의 자동 배출, 원단의 장력 및 오버피드, 그리고 각종 자동화 제어 장치의 활용 등으로 구분되어 질 수 있으며, 그러므로 텐터기 챔버 내부의 공간구조에 따른 비효율적 유로형상과 공기 분사노즐 정확한 압력 및 온도분포에 대한 현장 기초자료를 확보하여 텐터의 에너지 절약 및 건조 원단의 품질을 향상시킬 수 있는 구조의 설계가 요구된다.

1. Introduction

Tenter machines were used in the process of making woollen cloth. After the cloth had been woven it still contained oil from the fleece and some dirt. It was cleaned in a fulling mill and then had to be dried carefully as wool shrinks. To prevent this shrinkage, the wet cloth would be placed on a large wooden frame, a "tenter", and left to dry outside [1]. The lengths of wet cloth were stretched on the tenter using hooks all around the perimeter of the frame to which the cloth's edges were fixed so that as it dried the cloth would retain its shape and size.

2. Results and Discussions

Computational Fluid Dynamics (CFD) technologies have become a vital part of design and analysis in both research and commercial industries. CFD offers the ability to make a very detailed and through study of flow-fields and is capable of solving a broad range of flow problems. The ANSYS Workbench provides the geometry and modifies the geometry read through data formats and generates computational grid. CFX-Pre is used for initial problem set-up, definition of the prescribed motion of the geometry. CFX-Solver solves the high-speed, highly separated flow problems and capabilities represent the culmination of the growing experiences of developing advanced simulation software and associated physical models. CFX-Post is the 3D graphical post-processor that analyzes CFD [2]. Figure 1 shows the geometries which are used for computations.

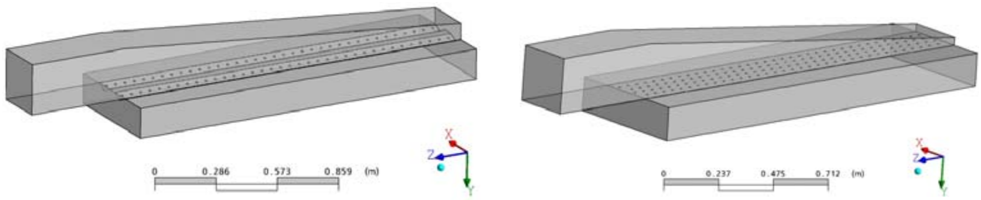

Fig. 1. Simulation geometries of Tenter machine.

3. Conclusion

A purpose was to have velocity uniformity at all nozzles. A nozzle duct sizes are changed and studied to solve vortex motion at the end part of the duct for two geometries. A nozzle velocity distribution is shown in Fig 2.

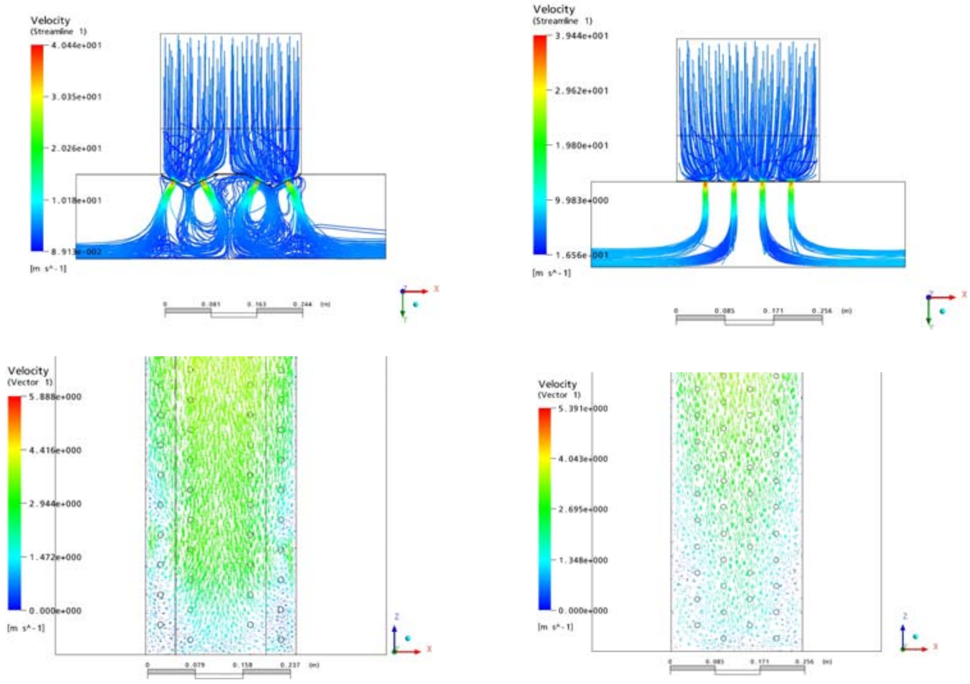

Fig. 2. Nozzle velocity distribution.

Acknowledgment

본 연구는 2007년도 지식경제부 지역진흥사업 기초기술개발사업(70002166)의 지원으로 수행되었으며, 이에 감사드립니다.

Reference

1. ANSYS Workbench, CFX-Pre, CFX-Solver, CFX-Post, Version 2005 User's Manual.

Tel.: +82-53-819-3128; e-mail: mjuraeva@kotmi.re.kr