Simulating the Flow Dynamics in Hydroentangling Nozzles: Effect of Cone Angle and Nozzle Aspect Ratio

H. VAHEDI TAFRESHI AND B. POURDEYHIMI

Nonwovens Cooperative Research Center, North Carolina State University, Raleigh, North Carolina 27695, U.S.A.

ABSTRACT

Flow behavior in various nozzle shapes from the hydroentangling cone-capillary family is modeled in this paper. The work studies the effects of cone angle and nozzle aspect ratio (the ratio of the capillary section to its diameter) on water-jet formation. In particular, two sets of nozzles are considered. The first set consists of nozzles that have an aspect ratio of one and cone angles of 19, 15, 11, 7, and 3°, respectively. The nozzles in the second set have their inlet and outlet diameters fixed, but their aspect ratios vary from 1 to 6. The results from a two-phase axisymmetric steady-state model of turbulent nozzle flow reveal that decreasing the cone angle or increasing the nozzle aspect ratio does not affect the hydroentangling water-jet characteristics. The computational scheme is validated by comparing part of the results with the available experimental data in the literature.

Hydroentangling, inspired by the needle-punching technology, exploits the energy transfer abilities of high-pressure water-jets to entangle loose fibers carried and supported by a moving screen. The water jets issue from a thin-plate strip with forty to fifty orifices per inch. The geometry of these fine nozzles plays a crucial role in the hydrodynamic behavior of the water jets. It is important that the water jets maintain their kinetic energy downstream of the orifice for an appreciable distance. However, depending on their nozzle geometry, water jets may break up into spray soon after the outlet [6, 8]. Once a water jet turns into spray, its kinetic energy is divided among millions of very fine droplets. Broken water jets have practically no utility, and consequently are not able to entangle fibers efficiently. For this reason, nozzle geometry is an important parameter of hydroentangling efficiency.

In order to improve the quality and uniformity of the surface texture of hydroentangled nonwoven fabrics, it may be desirable to reduce the distance between the water jets as well as their diameters. This means that a greater number of holes per inch can be placed on the orifice strip. Hydroentangling orifices typically consist of a capillary section (about 0.12 mm in diameter) connected to a conical part (0.3 mm larger diameter). As discussed previously [9], the diameter of such water jets is close to the size of the capillary diameter. Therefore, the limitation to increasing the number of holes per inch on an orifice strip is partially due to the size of the base in the conical section of the nozzle. It may therefore be desirable to explore possible ways of reducing the cone angle without disturbing water-jet functionality.

A second concern in hydroentangling is the orifice erosion that renders the strip ineffective after a relatively short time. Eroded nozzles do not have sharp-edged inlets. Once the inlets lose their sharpness, the orifices are no longer able to generate constricted water jets with highly intact lengths [9]. A practical solution to this problem is polishing the nozzle surface, which extends the lifetime of the nozzle strip and can potentially reduce the cost of hydroentangling. For this technique to be feasible, the nozzle capillary section should have enough material (thickness) to accommodate polishing without compromising the strength and utility of the strip. It is therefore desirable to examine the feasibility of using hydroentangling nozzles with high aspect ratios (the ratio of the capillary length to its diameter). A typical hydroentangling nozzle has an aspect ratio of about one and a cone angle of around 19°.

In this paper, we provide a brief overview of the numerical technique used to model fluid flow through these orifices. Next, we present the simulation results and compare our findings with the available experimental data. Finally, we present a generalized discussion followed by some key conclusions.

Computational Scheme

We consider a manifold pressure of 120 bars for all simulations. This corresponds to the highest pressure examined experimentally by Ghassemieh *et al.* [3], whose data we use for validating our simulation results. We define the Reynolds number in terms of the manifold pressure:

Textile Res. J. **73**(8), 700–704 (2003) 0040-5175/$15.00