Simulating Cavitation and Hydraulic Flip Inside Hydroentangling Nozzles

H. VAHEDI TAFRESHI AND B. POURDEYHIMI

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

ABSTRACT

Hydroentangling owes its success to the peculiar properties of coherent water jets. For hydroentangling to be feasible at higher pressures, it is extremely important that water jets maintain their collimation for an appreciable distance downstream of the nozzle. However, water-jet breakup accelerates at high pressures. Recent studies have shown that cavitation severely affects the integrity of high-pressure water jets. Investigating cavitation experimentally is not trivial. Computational fluid dynamics simulations offer appropriate tools as a first step. This paper discusses the results of an unsteady-state simulation, which shows the inception and time-evolution of a cavitation cloud inside a hydroentangling nozzle. Under certain conditions, the cavity cloud extends to the nozzle outlet, resulting in the so-called hydraulic flip. Once hydraulic flip occurs, cavitation suddenly vanishes because the downstream air moves upward into the nozzle and fills the cavity. This air envelops the water flow inside the nozzle, which results in the depletion of cavitation-induced instabilities from the jet surface and elongates the jet breakup length. Moreover, our simulations reveal the approximate time scales of cavity growth through the nozzle. This information is highly relevant for experimental visualization of nozzle cavitation. The discharge and velocity coefficient obtained from the simulation are in a good agreement with published experimental data.

Hydroentangling exploits the energy transfer capabilities of high-pressure water jets to entangle loose assemblies of fibers carried and supported by a moving screen. Hydroentangling nozzles are traditionally made up of two sections: a cylindrical section (capillary part) with a typical diameter of about 100 micron, connected to a slim cone with an angle of about 18°. Hydroentangling water jets issue from thin plate strips with a thickness of about 1 mm. These jet strips can be as much as 5 to 6 meters long with 1600 to 2000 orifices per meter. The use of popular cone-capillary geometry extends back to the early days in the development of the hydroentangling process.

The geometry of these fine nozzles plays a crucial role in the hydrodynamic behavior of the water jet. It is important that water jets maintain their kinetic energy, at least for a certain distance downstream of the orifice. However, water jets can break up into spray soon after leaving the nozzle. Once a water jet turns into a spray, its kinetic energy is divided between millions of very fine droplets. Broken water jets have practically no power and are unable to entangle fibers.

Water-jet breakup can be attributed to many factors such as hydrodynamic properties of the flow, aerodynamic interactions between the water jet and the ambient air, and nozzle internal flow. Traditionally, disintegration of the liquid jet has been related mainly to its aerodynamic interaction with the ambient air. However, recent studies by Hiroyasu [10, 11] and Tamaki *et al.* [18] have proven that, in nozzle geometries that favor cavitation, the main cause of water-jet breakup may be the strong disturbances that appear in the flow because of cavitation inside the nozzle. Cavitation refers to the condition where bubbles form in liquid flow. When the pressure of a body of liquid decreases at a constant temperature, eventually a state will be reached at which vapor bubbles (or cavities) become visible and grow in the system [12]. Nozzle cavitation depends strongly on nozzle shape. A dynamic reduction in pressure occurs in a nozzle flow whenever the nozzle wall turns away from the flow direction. In the case of a sharp turning angle (*e.g.*, the inlet of a sharp-edged orifice), boundary layer separation will cause the main flow to follow a curved path around a separated but liquid-filled region. If the flow velocity is high enough to cause the pressure on the separated region to drop to water vapor pressure, vaporization will occur and a cavitation pocket will form. When cavitation occurs, vapor regions form at the nozzle inlet, move toward the nozzle exit, and collapse in high-pressure zones. This can strongly disturb the flow steadiness inside the nozzle and affect its instantaneous velocity profile. The ambient air will then amplify these cavitation-