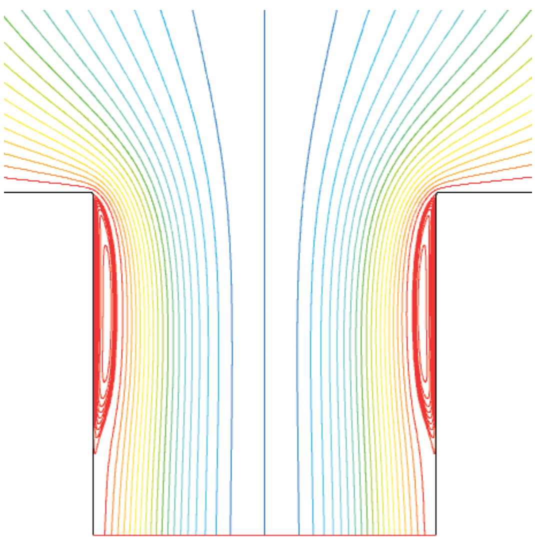
Cavitation and Hydraulic Flip

By H. Vahedi Tafreshi and B. Pourdeyhimi, Nonwovens Cooperative Research Center, North Carolina State University, Raleigh, NC

An unsteady simulation of a sharp-edged orifice ($r/d = 0.01$, where r is the inlet radius of curvature and d the inlet diameter) conducting a waterjet under high pressure (150 bar) has been performed to examine the onset and evolution of cavitation clouds inside a nozzle. Using FLUENT, the simulation showed that if the pressure is high enough and the inlet edge sufficiently sharp, the cavitation cloud grows and reaches the nozzle outlet. As the cloud reaches the outlet, the downstream ambient air finds a way to flow into the nozzle, resulting in a so-called hydraulic flip. Once the hydraulic flip condition occurs, cavitation immediately stops because the cloud region becomes filled with air, separating the waterjet from the nozzle wall. This effect keeps the waterjet surface from cavitation- and friction-induced instabilities. Constricted waterjets, enveloped by air inside a nozzle, stay intact for a significantly greater distance than non-constricted jets[1]. These waterjets have diverse applications, including nonwoven fabric manufacturing via a process called hydroentangling. This process is used for mechanically bonding a web of loose fibers to form uniform entangled sheets of fibers[1]. The impact of the waterjets with the fibers displaces and rotates them with respect to their neighbors. During these relative displacements, some of the fibers twist and entangle around others and inter-lock with them through fiber-to-fiber friction.



Contours of stream function at 26 μs show the presence of cavitation

The cavitation model in FLUENT 6 was used for a 17,000-cell axisymmetric simulation of flow through a sharp-edged nozzle. The cavitation model tracks two interpenetrating fluids (liquid and vapor) using a volume fraction equation and a single momentum equation. Bubbles form when the local pressure becomes less than the vaporization pressure, and these bubbles may grow and form cavities. Pressure inlet and pressure outlet boundary conditions were used along with the RNG k-ε turbulence model. The two-layer zonal method was used for the wall treatment, with y^+ values close to unity in the cells adjacent to the solid surface.

For the cavitation simulation, the bubble number density (BND) should be known in advance. Acquiring such information is difficult because it requires sophisticated experimental facilities. Instead, a minimum BND that results in the occurrence of a hydraulic flip was determined and used in the simulation. To determine the value, a series of simulations were run with different BND values. Starting from a low value, the BND was gradually increased until the hydraulic flip occurred. The final value (6×10^9 bubbles/m^3) was in agreement with a range of values (10^8 to 10^{12} bubbles/m^3) reported in the literature[2]. (Any value greater than 6×10^9 bubbles/m^3 would cause the hydraulic flip to occur as well.) To simplify the problem, the pressure outlet was set up with a vapor (rather than air) volume fraction of unity. This means that any backflow through the outlet enters the calculation domain in the form of vapor rather than air. This approximation is valid since the densities of vapor and air are similar when compared to that of liquid water.

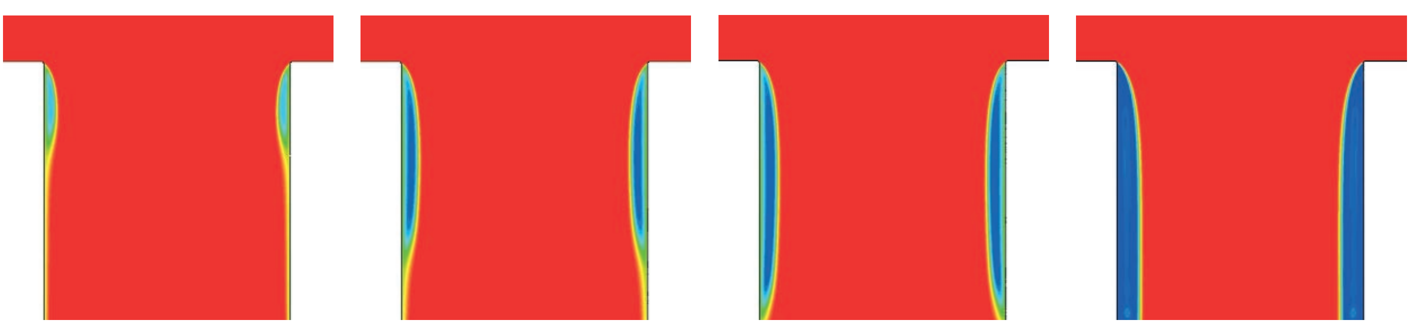
The CFD simulations were very successful in predicting the hydraulic flip. The discharge coefficient obtained from the simulation, $C_d = 0.63$, defined as the ratio of the actual flow rate from a nozzle to that calculated by inviscid one-dimensional theory (the Bernoulli equation), was found to be in excellent agreement with experimental data. ■

references:

1 H. Vahedi Tafreshi and B. Pourdeyhimi, Experiments in Fluids, **35**(4), p. 364-371, 2003.

2 R.A. Bunnell, S.D. Heister, C. Yen, and S.H. Collicott, Atomization and Sprays, **Vol.9**, p. 445-465, 1999.

3 H. Vahedi Tafreshi and B. Pourdeyhimi, Textile Research Journal, **Vol.74**(4), 2004.



Contours of mixture density inside the nozzle after 10, 30, 50, and 60 μs of operation