Large Eddy Simulation of Micro Water-Jet in a Coaxial Swirling Gas Flow

Meirong Wei¹, Yan Liu⁴*, Tao Zhang¹, Hui Li²
¹School of Naval Architecture & Ocean Engineering, Huazhong University of Science and Technology
²School of Power and Mechanical Engineering, Wuhan University
Wuhan, Hubei, China

ABSTRACT

Wall-Modeled Large Eddy Simulation (WMLES) was performed to investigate the mechanism of the stability of waterjet injected into a surrounding coaxial swirling gas flow. The study takes into consideration the distribution of pressure and velocity in the cavity on the influence of initial disturbance of the jet stability. Estimation of the breakup length is of paramount importance in applications of micro-waterjet guided laser, needleless injectors, hydroentangling waterjets, etc. With sufficient grid resolution, simulations of three different boundary conditions were performed and results shown that low pressure channel is conducive to the stabilization of waterjet.

KEY WORDS: WMLES; coaxial swirling gas flow; breakup length; waterjet.

INTRODUCTION

There are abundant studies investigating the stability of the waterjet, especially in the application of atomization. However, a limited number of studies have paid attention to the coaxial swirling gas flow effects on the stability of waterjet. The distance of the end of the continues liquid column from the nozzle outlet prior to breakup is called breakup length. The longer the breakup length, the better the stability of waterjet. Experimental studies by Tamaki et al.(1998,2001) and Hiroyasu et al.(2000) show that the occurrence of cavitation inside the nozzle makes a substantial contribution to the breakup of the exiting liquid jet. Many scholars study the mechanisms of atomization of liquid jet surrounded by gas flow, including swirling gas flow (YAN Chun-ji,2004) and non-swirling airstreams(Aliseda, A et al.,2008, Varga, CM, et al.,2003, Tian, XS et al,2015 ). Pierce C and Moin P(1998) performed large eddy simulation of a coaxial jet combustor with swirling fluid. It was shown that confined swirling flows can be very sensitive to downstream boundary conditions. Xiao, F. et al.(2014) studied liquid jet primary breakup in coaxial air flow using a developed two-phase flow CLSVOF LES methodology. It is found that the turbulence vortices under the condition of high Reynolds number are the main cause of initial interface disturbance, and the mean shear stress of gas flow is the primary cause of initial interface instability. The research of Cadavid, R. et al(2005) showed that reducing the environmental pressure around the water jet with light gases such as helium, can increase the coherence length of the water jet. Fenn, RW at el(1969) found that for larger weber numbers aerodynamic pressure forces become important and lead to reduced stability of the jet. Eddingfield, D and Albrecht M (1979) found that air-injected shroud on the high-speed waterjet can increase the breakup length.

Although many scholars have studied the coaxial gas-water jet, and determined that the coaxial gas improve the stability of the water jet through experiments or simulations, the influence of coaxial rotating airflow on the stability of the water jet has not been studied in detail. Many scholars have studied the coaxial gas-water jet, and determined that coaxial gas can improve the stability of water jet through experiments or simulations, but the influence of coaxial swirling gas flow on the stability of water jet has not been studied in detail. The effects of coaxial swirling gas on the waterjets are almost invisible due to the millimeter-scale swirling gas in cavity, so computational fluid dynamics (CFD) was used to help understand the procedure in this paper.

MODELING

Physical Modeling

According to the structure of jet device, the computational domain can be extracted from the inner flow channel as shown in Fig.1. The simulation domain and the boundary condition are shown in Fig.2. The primary water jet has a diameter of $3 \times 10^{-4}$ meters which is taken as the reference length scale with the computational domain and outer radius of cavity is 650 μm. The whole cavity is 13.45 mm high, is oriented vertically with gravity acting in the direction of the flow.