Numerical simulations on sheet region of the spray cooling process of the pressure-swirl nozzle

H. Ding¹, P. Yangmin¹, Y. Luo*¹

¹Tianjin University, China

Abstract

The commercial software Fluent 15.0 is employed to carry out the numerical simulation on the internal and external flow fields of the pressure-swirl nozzle. The axisymmetric 3-D flow field is represented by an equivalent 2-D grid. The VOF multiphase flow model and Reynolds Stress Model (RSM) are chosen. Numerical simulations on flow fields are performed in two different circumstances: Ⅰ. Gas phase is specified as air and there is no heat and mass transfer between phases; Ⅱ. Gas phase is saturated steam and heat and mass transfer exists between phases. Lee, a computational model embedded in Fluent 15.0, is specified as the phase-transition model of heat transfer. Comparisons between CFD simulations and experiment are launched. The internal and external flow fields are analyzed based on simulation datum. Results indicate that a air core forms inside the nozzle due to the helical motion of liquid phase, velocity of which increases sharply at the junction of contraction section and orifice's straight pipe section of the nozzle. Furthermore, comparisons are also performed between circumstance Ⅰand Ⅱ. Numerical simulation results indicate that when heat and mass transfer exists between phases (i.e. in case of circumstance Ⅱ), (1) pressure of the flow fields is slightly lower and peak velocity is larger; (2) heat transfer coefficient of liquid film decreases gradually along the flow direction; (3)the film is thicker due to the vapor condensation, and liquid film breakup length is larger.