Computers and Fluids 154 (2017) 216–223

Contents lists available at ScienceDirect

Computers and Fluids

journal homepage: www.elsevier.com/locate/compfluid

Benchmark solutions

Mixing of hollow-cone water spray in a confined high-temperature gas crossflow

Wenjie Wang, Haibin Zhang, Zhongchuang Zhao, Qinghai Zheng, Bofeng Bai*

State Key Laboratory of Multiphase Flow in Power Engineering, Xi'an Jiaotong University, Shaanxi 710049, China

ARTICLE INFO

Article history:
Received 23 March 2016
Revised 21 November 2016
Accepted 12 June 2017
Available online 12 June 2017

Keywords:
Mixing
Evaporation
Hollow-cone spray
Numerical simulation

ABSTRACT

The mixing process of hollow-cone spray in a confined high temperature gas crossflow accompanied with evaporation is studied numerically. A fundamental insight into the mixing is obtained. The characteristics of the mixing flow field are investigated and the mixing under different conditions are quantitatively evaluated based on the cross-sectional temperature distribution using a newly proposed spatial uniformity index. It is shown that the temperature pattern of the mixing flow field depends primarily on the initial droplet distribution and the subsequence droplet dispersion dynamics. Several pairs of small vortices are generated near the wall due to the interaction between the crossflow and the droplet swarm. These vortices promote the evaporation of the entrained droplets and dissipate quickly due to the vapor convection. The influences of factors such as spray angles, axial injection angles, tangential injection angles and position of nozzle on the mixing are analyzed, showing that an optimum mixing can be achieved with appropriate spray angle.

1. Introduction

Hollow-cone spray is extensively encountered in relevant industrial applications, such as cooling, evaporation, mixing and fuel burning. In the water ramjet propulsion systems, sea water (also works as working substance addition) is injected into the combustion chamber by hollow-cone spray nozzles and then the spray droplets mix with the fuel gas until the droplets evaporate completely. A good mixing between the spray and the fuel crossflow is critical for a good engine performance. Therefore, a well understanding of the resultant mixing flow structures related with such mixing enhancement is required. However, the mixing process which involves the droplet collision and evaporation, droplet/wall impingement and various vortices is highly complex and the details of them are still not well understood due to the difficult experimental challenges.

Previous studies on hydro-reactive metal fuel engine mainly focus on the metal fuel [1–7], propulsion of the engine [8–12], thermodynamic performance [13,14]. For the spray/jet in crossflow, researchers paid more attentions on the vortices structures and the droplet dispersion dynamics [15–20]. By using large eddy simulation (LES) turbulent model, Salewski and Fuchs [21] found that the counter-rotating vortex pair (CVP) is the major vortex structure in the mixing flow field and the droplet dispersion was greatly affected due to the development of the CVP. Bai et al. [15,22,23] and Sun et al. [24] confirmed that the CVPs was the dominating structure in the mixing flow field in their experiments of the hollow-cone spray in a confined crossflow at the ambient temperature and pressure by using PIV visualization system. The coherent structures which also impose greater influences on the mixing was discovered. The dynamics of spray droplet dispersion and the interaction between the droplets and the large-scale vortices were analyzed and the influences of different factors on the mixing were also discussed.

However, compared with that of liquid jet in crossflow (LJIC), the literature on hollow-cone spray in crossflow is relatively few, and among the available sources very few studies have dealt with the interaction between hollow-cone spray and a confined high temperature crossflow. The mixing process of hollow-cone spray in a confined high temperature crossflow which involves the break-up and atomization of the spray, the droplet dispersion, the droplets evaporation as well as the droplet-wall impingement is very complex. Kachhwaha [25,26] experimentally studied the droplet dispersion and evaporation of a hollow-cone spray in a moderate temperature air crossflow. Zhang and Bai [27] studied the characteristics of the acetone droplet evaporation injected by a hollow-cone spray nozzle in high temperature gas crossflow in a confined mixing space. The temperature pattern of the mixing flow field and its formation mechanism were qualitatively analyzed and they concluded that a good mixing can be achieved with a proper spray penetration depth. However, further details of droplet dispersion

* Corresponding author.
E-mail address: bfbai@mail.xjtu.edu.cn (B. Bai).

http://dx.doi.org/10.1016/j.compfluid.2017.06.007

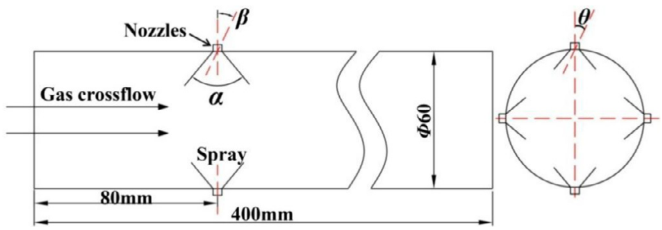

Fig. 1. Geometrical structure of simulation system.

and evaporation and the interactions among the droplets, the gas crossflow and the vapor were not given. Furthermore, in the real engine, the parameters such as the temperature and pressure are much higher than that in the laboratory conditions and the well understanding of the real mixing process is still difficult by experimental methods.

Therefore, a three-dimensional numerical simulation of the hollow-cone spray in a confined crossflow under high temperature and pressure conditions is carried out in this paper. Firstly, based on the former numerical studies on hollow cone spray interacting with crossflow at normal temperature [28–30] and the investigations on droplet evaporation models for modeling spray in crossflow [31] as well as the droplet-wall impingement researches, the numerical model for the simulation of the mixing of hollow cone spray in a confined high temperature crossflow is proposed, and then the characteristics of the mixing process and the mixing flow field are studied numerically. The main objectives of these simulations are:

(1) To get the features of droplet distribution injected by the hollow-cone spray in the crossflow confined in a roundness space and the characteristics of droplet evaporation;
(2) To explore the evaluation of the temperature patter with the development of the mixing and the vortices;
(3) To systematically obtain the influences of factors such as spray angle, axial injection angle and tangential injection angle on the mixing, and the principle for a good mixing arrangement.

This paper is organized as follows: in Section 2 the models of the turbulent gas crossflow and the hollow-cone spray as well as the procedures of the numerical implementation are described and the model validation is presented. The characteristics of the mixing flow field are described in Section 3 and in Section 4 the influences of spray angel, axial injection angle and tangential injection angle on the mixing are discussed. The main findings and the major conclusions are given in Section 5.

2. Numerical approach

2.1. Physical system

The geometrical structure of the computational domain is a tube with diameter of 60 mm and length of 400 mm (see Fig.1). The four nozzles are symmetrically located on the circumference of the cross section. The distance between the nozzles and the inlet is 80 mm. The spray angle, axial injection angle and tangential injection angle of the nozzles are denoted as α, β, and θ and are show in Fig.1.

The droplets are injected by the hollow-cone spray nozzle into the high temperature crossflow and then the mixing is carried out. The temperature of the inlet crossflow is 1500 K and the pressure is 3 MPa, the mass flow of a single nozzle is 3 g/s.

2.2. Simulation model

The Eulerian method is used to solve the continuous phase and the droplets are tracked in a Lagrangian way. The realizable $k-\varepsilon$ turbulent model is applied in the simulation, because the realizable $k-\varepsilon$ turbulent model contains a new formulation for the turbulent viscosity and a new transport equation for the dissipation rate ε. The dissipation rate is derived from an exact equation for the transport of the mean-square vortices fluctuation, exhibiting a superior performance for the flows involving rotation and complex structure. On the basis of mass and momentum conservation, continuous phase model equations of continuity, momentum, k and ε can be unified as:

$$\frac{\partial}{\partial t}(\rho_g\phi)+\frac{\partial}{\partial x_j}(\rho_g u_j\phi)=\frac{\partial}{\partial x_j}(\Gamma_j\frac{\partial\phi}{\partial x_j})+S_\phi^g+S_\phi^p \tag{1}$$

where, S_ϕ^g is the source item of gas phase, S_ϕ^p the source item of discrete phase, u_j the gas velocity of direction j, Γ the generalized diffusion coefficient, ρ_g the density of gas phase, ϕ can be a variable as 1, u_i, T, k and ε, respectively. The source terms are listed in Table 1.

To model the primary break-up process of the hollow-cone spray, the LISA (Linearized Instability Sheet Atomization) model which was commonly recommended [29,32–35] in many previous researches was utilized in the present work. Several secondary break-up models were proposed till now, such as the TAB (Taylor Analogy Break-up) model, E-TAB (Enhanced-TAB) model, I-TAB (Improved-TAB) model. The hybrid LISA-TAB model can perform a relatively better prediction for the hollow-cone spray at normal pressure condition [36–38]. After that, the validation of performing the LISA-TAB to simulate the hollow-cone spray in crossflow has been approved [29,39]. Therefore, the hybrid LISA-TAB model was employed in our study.

The dispersed spray droplets are tracked in groups called "parcels". Drag force and gravity of droplets are considered and the droplets are assumed spherical. Equations of the droplet motion are as follows:

$$\frac{du_p}{dt}=(\boldsymbol{u}_g-\boldsymbol{u}_p)/\tau_p+\boldsymbol{g} \tag{2}$$

$$\tau_p=\frac{4\rho_p d_p^2}{3\mu_g C_D Re_p} \tag{3}$$

$$C_D=\begin{cases}\left(\frac{24}{Re_p}\right)\left(1+0.15Re_p^{0.687}\right), & Re_p<1000\\ 0.424, & Re_p\geq 1000\end{cases} \tag{4}$$

where $\boldsymbol{u}_p$ and $\boldsymbol{u}_g$ is the velocity of liquid phase and the gas phase, respectively. τ_p is the relaxation time of droplets velocity. $\boldsymbol{u}_g$ is the dynamic viscosity of gas phase. ρ_p is the density of liquid phase. C_D is the coefficient of drag force. Re_p is the particle Reynolds number.

The high temperature of gas phase induces a strong heat convection transfer between liquid phase and gas phase and thus promote the heating rate of droplets. Hence, the effect of droplet internal temperature gradient can be ignored and the isothermal calculation method can be used to consider the rate of droplets evaporation with the assumption of equal rate between heat transfer and mass transfer.

$$\dot{m}_p=-2\pi D_p\frac{\lambda_g}{c_{p,g}}Nu\ln(1+B_M) \tag{5}$$

$$Nu=2+0.6Re^{1/2}Pr^{1/3} \tag{6}$$