International Journal of Heat and Mass Transfer 85 (2015) 1003–1011

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Numerical study of heat transfer deterioration of turbulent supercritical kerosene flow in heated circular tube

Guoxin Dang, Fengquan Zhong *, Yongjiang Zhang, Xinyu Zhang

State Key Laboratory of High Temperature Gas Dynamics, Institute of Mechanics, Chinese Academy of Sciences, Beijing 100190, China

ARTICLE INFO

Article history:
Received 30 October 2014
Received in revised form 8 January 2015
Accepted 17 February 2015
Available online 7 March 2015

Keywords:
Supercritical kerosene
Convective heat transfer
Turbulent flow
Numerical study

ABSTRACT

Turbulent flow and convective heat transfer of supercritical kerosene flow in an axisymmetrically heated circular tube with a diameter of 2 mm and at a mass flow rate range of 0.0015–0.015 kg/s and a wall heat flux range of 0.15–2.0 MW/m^2 are numerically studied using Reynolds averaged Navier–Stokes method with a two-layer turbulence model. The thermophysical and transport properties of kerosene are determined by a 10-species surrogate with the Extended Corresponding State method. Mesh dependency is first investigated and numerical results of fuel and wall temperatures are compared with experimental data for validations. The results show that flow properties such as velocity and Reynolds number increase significantly along the axial direction as the fuel temperature rises and kerosene undergoes the state transition from liquid to supercritical state. Deterioration of convective heat transfer is found to occur when the wall heat flux exceeds a critical value and at the same time, the wall temperature approaches the pseudo-critical temperature of kerosene. The present results show that deterioration of heat transfer are attributed to the development of turbulent properties in the near-wall region based on the results of turbulent kinetic energy and turbulence production term. The relation between the critical heat flux (q_{wc}) for occurrence of heat transfer deterioration and the mass flux (G) is studied and a fitting formula of q_{wc} and G is obtained.

1. Introduction

Regenerative cooling using onboard hydrocarbon fuels as coolant is considered to be one of the most effective methods for rocket and scramjet thermal protections [1,2]. In a regenerative cooling system, hydrocarbon fuel flows through cooling channels and absorbs heat of engine wall via heat convection before being injected into combustor. Characteristic of convective heat transfer of hydrocarbon fuels is one of the most important fundamental issues for cooling design and optimization. It has been known that the working pressure of fuel is usually higher than its critical value (P_c) and along the cooling path, fuel temperature would increase and exceed its critical temperature (T_c) so that hydrocarbon fuel becomes supercritical without going through a two-phase dome region. Compared to gaseous or liquid state, supercritical fluid presents many of unique features [3–5]. It has gas-like transport property, relatively large density similar to liquid and very high solubility. The thermophysical and transport properties of supercritical fluid such as density, specific heat, viscosity, and thermal conductivity have remarkable variations with temperature and pressure near the critical point and there exists a peak of specific heat at a temperature as so called "pseudo-critical temperature" (T_{pc}). Therefore, flow and heat transfer properties of supercritical hydrocarbon fuels are significantly different from those of simple liquids and could result in heat transfer deterioration or enhancement. It is worth noticing that understanding heat transfer deterioration or enhancement is critical for cooling design. Therefore, it is imperative to study characteristics of convective heat transfer of supercritical hydrocarbon fuels at flow conditions relevant to rocket or scramjet applications.

Most of previous studies on convective heat transfer of supercritical fluids are focused on simple fluids such as water [6], carbon dioxide [7,8], and hydrocarbons of small molecules such as methane [9,10]. Ward et al. [11] numerically investigated flow and heat transfer properties of mildly cracked alkanes at supercritical conditions. Urbano and Nasuti [12] numerically studied heat transfer of supercritical methane. The critical value of ratio of wall heat flux to specific mass flow rate for the onset of heat transfer deterioration and its dependency on pressures were investigated. In another recent paper [13], Urbano and Nasuti studied

* Corresponding author at: Bei-Si-Huan-Xi Road #15, Institute of Mechanics, Chinese Academy of Sciences, Beijing 100190, China. Tel.: +86 10 82543838.
E-mail address: fzhong@imech.ac.cn (F. Zhong).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.02.052
0017-9310/

flow and wall heat flux conditions for the occurrence of heat transfer deterioration for varied light hydrocarbons including methane, ethane and propane. More recently, Ruan et al. [14] calculated turbulent flow and heat transfer of *n*-decane with pyrolytic reaction at supercritical pressures. Very limited studies about heat transfer of supercritical hydrocarbon fuels such as kerosene or diesel have been reported, of which the majority are experimental studies. Linne et al. [15] studied supercritical heat convection and thermal cracking of JP-7 fuel with an electrically heated pipe system. Zhong et al. [16] investigated heat transfer of China RP-3 aviation kerosene at a Reynolds number range of 3×10^3–3×10^5 and a wall heat flux range of 50–300 kW/m^2 with a two-stage heating facility. The deterioration and enhancement of heat transfer of kerosene flow were observed. Hu et al. [17] experimentally inspected heat transfer of kerosene at high wall heat fluxes (5–10 MW/m^2). They identified the occurrence of heat deterioration in the region where the inner wall temperature is close to the pseudo-critical temperature. Due to difficulties in the measurement of fluid temperature inside small pipe with a diameter of a few millimeters, most of the experiments discuss only overall performance of convective heat transfer in terms of the measured inlet and outlet fuel temperatures and mass flow rate, lacking of details of the change in flow and heat transfer properties along the pipe length. At the same time, due to hundreds of species in kerosene and its complex thermophysical properties, numerical simulation of kerosene flow, especially turbulent flow, encounters many of difficulties and so far, very few relevant numerical works have been reported.

In this paper, turbulent convective heat transfer of supercritical aviation kerosene in a heated, straight and horizontally placed circular tube is numerically studied. The Reynolds averaged Navier–Stokes equations are solved with a two-layer turbulence model, including the RNG k–ε turbulence model and the Wolfstein one-equation model in the near-wall region. Thermal and transport properties of kerosene are determined by a 10-species surrogate proposed in our previous work [16] for China RP-3 aviation kerosene with the Extended Corresponding State method (ECS) [18]. The present numerical method is validated via study of mesh dependency and comparisons of wall and fuel temperatures with experimental data. Based on the present results, deterioration of convective heat transfer of kerosene are observed and the associated turbulent flow mechanisms are discussed. Effects of mass flow rate and wall heat flux on heat transfer properties are studied as well as the issue of critical heat flux for occurrence of heat transfer deterioration. The present study is expected to provide further insights into mechanisms of supercritical heat transfer and reference data for design and optimization of cooling system for rocket and scramjet applications.

2. Numerical procedures

2.1. Computational domain and boundary conditions

Kerosene flow in a straight and horizontally-placed circular tube with an inner diameter of 2 mm is studied. The flow is turbulent since the minimum Reynolds number Re_d (at the entrance of heated section of the tube) is 7000, larger than the critical value of 2300 for turbulent pipe flow. The gravity effect is not included in the present study because the maximum ratio of Grashof number Gr to Re_d^2 is less than 6×10^{-4} that is smaller than the threshold value of 0.001 [19] for the consideration of gravity effect. Another issue worth noticing is that for China RP-3 aviation kerosene, pyrolysis occurs when the fuel temperature reaches approximately 800–850 K at pressures of 3–5 MPa [20]. Hence, heat transfer data analyzed in the paper are limited in the flow region where the wall and the fuel temperatures are both below 800 K.

Fig. 1 gives a sketch of the tube with heat loading. Due to axisymmetry of circular tube flow and wall thermal boundary condition, two-dimensional computation is adopted. Dimensions and boundary conditions of the computational domain are given as below.

(1) The diameter of the tube is 2 mm and the length is varied from 1 to 2.2 m according to different cases with varied wall heat flux. As shown in Fig. 1, a 0.1 meter-long flow region without heat loading is calculated upstream of the heated section to obtain a fully developed turbulent pipe flow.
(2) The inlet fuel temperature is 350 K and the inlet pressure is 3 MPa, greater than the critical value of approximately 2.4 MPa.
(3) The mass flow rate is in a range of 0.0015–0.015 kg/s and correspondingly, the mass flow rate per unit area, i.e. G ($G = \frac{\dot{m}}{A}$) is varied from 475 to 4750 kg/(m^2 s). An exponential velocity profile according to fully developed turbulent pipe flow is imposed at the inlet as boundary condition.
(4) The outflow boundary condition with zero gradients of flow velocity along the axial direction [21] is applied at the outlet for fully developed tube or duct flows. The no slip and no penetration boundary are used for the tube wall and a constant heat flux is imposed on the wall of the heated section. The heat flux is changed from 0.15 to 2.0 MW/m^2.

2.2. Numerical method and computational mesh

The Navier–Stokes equations are solved by finite volume method provide by the density based solver of Fluent 6.3. The 2nd-order upwind scheme is applied for convective terms and 2nd-order central scheme for viscous terms. The RNG $k-\varepsilon$ model is applied for turbulence modeling and the Wolfstein turbulence model [22] is adopted in the near-wall region characterized with low Reynolds number properties. The SIMPLE algorithm is employed to resolve the coupling between velocity and pressure. The implicit Gauss–Seidel iteration is used to calculate the time advancement.

Stretched grids are constructed along the radial direction in the near-wall region. The first grid point to the wall is constructed to ensure $\Delta y^+ \leqslant 1$ ($\Delta y^+ = \frac{\Delta y}{y_\tau}$, where y_τ is the viscous length scale of turbulent flow [23]) and there are more than 10 grids within the viscosity-dominant near-wall region (i.e. $y^+ \leqslant 11$). Since thermophysical properties of kerosene change dramatically at temperatures close to the critical value, the grid size in the vicinity of the wall should be refined to ensure accuracy of the numerical scheme. A uniform mesh is first constructed along the flow direction (for example, the total gird number is $N_x = 2000$) and computation is conducted. The adaptive grid technique based on density gradient of the computational result is then used to refine the grid spacing in the region with large variations of fuel properties and the grid number after the adaption process is increased to 2500.

The mesh dependence is first investigated. Four meshes are studied. The total grids numbers are 100,000 (case a), 150,000 (case b), 240,000 (case c) and 960,000 (case d) respectively. The tube diameter of the test case is 2 mm and the wall heat flux is

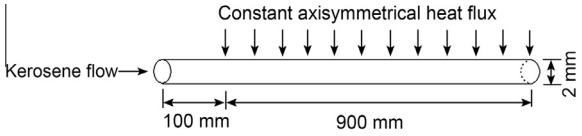

Fig. 1. Sketch of the heated circular tube.