Simulation Study of LPG Cooking Burner

Mana Wichangarn¹, Anirut Matthujak¹*, Thanarath Sriveerakul¹, Sedthawatt Sucharitpwatskul² and Sutthisak Phongthanapanich³

¹Combustion and Jet Application Research Laboratory (CJARL) Department of Mechanical Engineering, Ubon Ratchathani University, 34190 Ubon Ratchathani, Thailand
²Computer-Aided Engineering Laboratory, Design and Engineering Research Unit, National Metal and Materials Technology Center (MTEC), 12120 Pathum Thani, Thailand
³Department of Mechanical Engineering Technology, King Mongkut's University of Technology North Bangkok, 10800 Bangkok, Thailand
*Corresponding author E-mail: Anirut.m@ubu.ac.th

Abstract

The objective of this paper is to numerically study the flow feature and combustion phenomena of an energy-saving cooking burner using three-dimensional computational fluid dynamics (CFD). Combustion temperatures were experimentally and numerically investigated in order to not only validate the CFD model, but also describe the combustion phenomena. From the temperature comparison, the CFD model was good agreement with the experiment, having the error of less than 5.86%. Based upon the insight from the CFD model, the high temperature of 1,286 K occurred at the middle of the burner. The high intensive vortex of the flow being enhanced the combustion intensity and the heat transfer coefficient is obvious observed near the burner head inside the ring. Therefore, it is concluded that the burner ring is the major part since it controls flame structure, high temperature region, intensive combustion region, heat loss and suitable flow feature. However, heat transfer to the vessel should be further clarified by the CFD model.

Keywords: Energy-saving cooking burner, CFD, combustion temperature, flow feature, validation.

1. Introduction

LPG cooking burners are widely used as domestic heating appliances because of convenience and safety. In view point of energy saving and pollutant emission control, improvement of thermal efficiency is the most important research topic. Recently, it is known well that there have been many studies to improve the thermal efficiency with low pollutant emission. Much attention has been focused on improving KB cooking burner by experimental study [1-5], while there has been very limited simulation study.

In 2014, Boggavarapul, P. [6] studied both experimentally and numerically KB burner using LPG and PNG (piped natural gas). Three-dimensional computational fluid dynamic (CFD) modeling of the steady-state flow and combustion was reported. Design modifications of the burner based on the insights from CFD modeling were proposed. From experiment, the improvement in burner thermal efficiency of 2.5% and 10% for LPG and PNG, respectively, was achieved. However, validation of the CFD model with the experiment was not presented.

Recently, there is new model of LPG burner in Thailand, which its thermal efficiency is about 45% [7], which is higher than conventional KB burner, being 35% from experimental testing. It is called that energy-saving cooking burner as shown in Figure 1. However, the insight of flow feature leading to the high thermal efficiency of the energy-saving cooking burner was not clarified.

In this paper, the flow feature and temperature distribution of the energy-saving cooking burner are preliminarily studied by three-dimensional CFD modeling. Moreover, validation of the CFD modeling with the experiment is presented.

Figure 1: Energy-saving cooking burner. [7]
2. Numerical Modeling

The objective of this paper is to preliminarily study and validate the flow feature and combustion of the energy-saving cooking burner. Thus, heat transfer to the vessel is not modeled yet. Three-dimensional computational fluid dynamic modeling of the steady state flow and combustion was performed by FLUENT and the computational grid was prepared by GAMBIT. The energy-saving cooking burner used in this modeling is shown in Figure 1. The 3D computation domain and the grid are shown in Figure 2a. Structured meshes were used wherever possible and tetrahedral mesh was used for meshing complicated geometrical features.

In this study, the LPG flow rate used in CFD modeling was measured experimentally at pressure of 1 bar. Thus, the turbulence and eddy dissipation combustion models for volumetric reactions were used to solve the species transport equations. A three-step reaction mechanism [8] involving four reactions was used.

Table 1: Inlet data from cold flow simulation. [9]

<table>
<thead>
<tr>
<th>Detail</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass flow rate of mixture</td>
<td>0.003952742</td>
</tr>
<tr>
<td>Mass fraction of C3H8</td>
<td>0.0392758</td>
</tr>
<tr>
<td>Mass fraction of CH4</td>
<td>0.0392758</td>
</tr>
<tr>
<td>Mass fraction of N2</td>
<td>0.7095291</td>
</tr>
<tr>
<td>Mass fraction of O2</td>
<td>0.2119193</td>
</tr>
</tbody>
</table>

Figure 2b shows the boundaries of the model. The boundaries in ambient were set as pressure outlets. A wall boundary of burner head was shown in Figure 2c. The steady-state problem is solved. The mass flow rate of the LPG-air mixture and its mass fraction of propane, butane, nitrogen and oxygen at the inlet were specified at nozzle pressure of 1 bar [9] as shown in Table 1. Gravitational forces were included to simulate the buoyancy driven flow around the burner head. The discrete ordinates radiation model was used for the radiation effect.

A two-step solution procedure was conducted in this study [6]. In the first step, a cold flow simulation was done where the fluid dynamic solution was obtained without reactions by switching off the energy equation. In the second step, the energy equation was switched on. The ignition was initiated in the combustion zone. The solution was considered converged when the residual value was set to $10^{-6}$ for all models.

3. Results and Discussion

Figure 3 shows flame shape from experiment and CFD model at LPG pressure of 1 bar. From photograph of flame in Figure 3a, the flame shape seems spearhead. It is highly luminous and long in the middle region. Its periphery near the ring is short because of suitable design of burner holes and heat loss. The highly luminous flame in the middle shows the high temperature and combustion intensity. It can be confirmed by temperature contour from CFD in Figure 3b. The flame shape from CFD is similar to the experiment.
cooking burner should be done in the further study. Moreover, improving thermal efficiency of this burner.

Acknowledgement

This research is financially supported by Ubon Ratchathani University.

References


