International Journal of Engineering & Technology, 7 (3.7) (2018) 142-144

International Journal of Engineering & Technology

Website: www.sciencepubco.com/index.php/IJET

Research paper

Simulation Study of LPG Cooking Burner

Mana Wichangarm[1], Anirut Matthujak[1*], Thanarath Sriveerakul[1], Sedthawatt Sucharitpwatskul[2] and Sutthisak Phongthanapanich[3]

[1]Combustion and Jet Application Research Laboratory (CJARL) Department of Mechanical Engineering, Ubon Ratchathani University, 34190 Ubon Ratchathani, Thailand
[2]Computer-Aided Engineering Laboratory, Design and Engineering Research Unit, National Metal and Materials Technology Center (MTEC), 12120 Pathum Thani, Thailand
[3]Department 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.

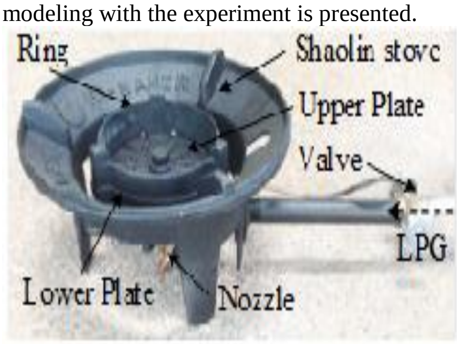


Figure 1: Energy-saving cooking burner. [7]

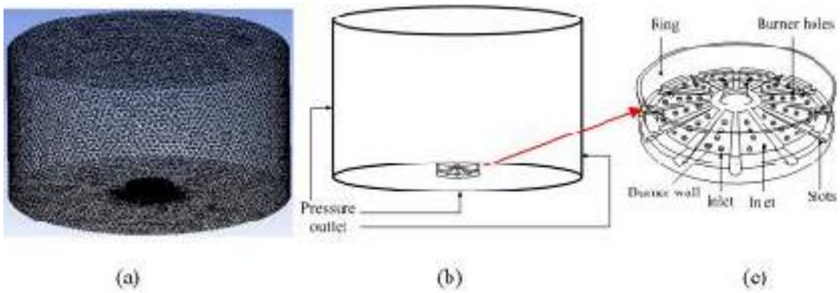


Figure 2: (a) computational domain, (b) boundary conditions and (c) burner head (upper plate and ring)

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]

Detail	Value
Mass flow rate of mixture (kg/s)	0.003952742
Mass fraction of C_3H_8	0.0392758
Mass fraction of C_4H_{10}	0.0392758
Mass fraction of N_2	0.7095291
Mass fraction of O_2	0.2119193

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.

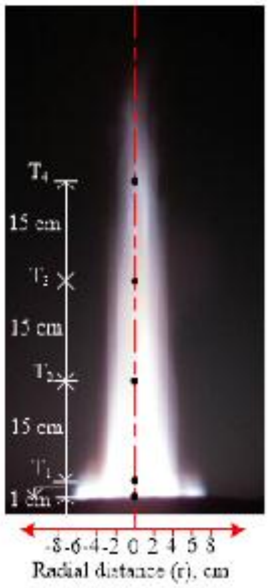
A two-step solution procedure was conducted in this study [6]. In the first step, a cold flow simulation was done where the fluid dynamic solution was obtained without reactions by switching off the energy equation. In the second step, the energy equation was switched on. The ignition was initiated in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

3. Results and Discussion

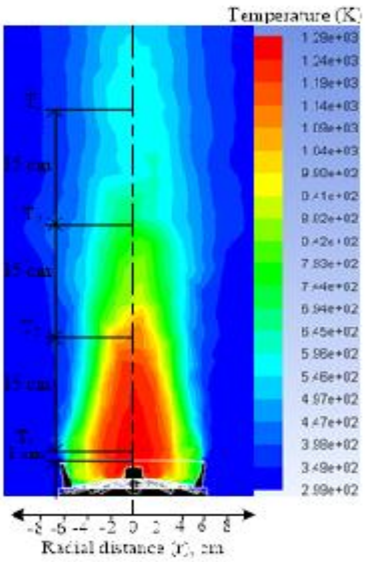
Figure 3 shows flame shape from experiment and CFD model at LPG pressure of 1 bar. From photograph of flame in Figure 3a, the flame shape seems spearhead. It is highly luminous and long in the middle region. Its periphery near the ring is short because of suitable design of burner holes and heat loss. 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. The temperature distribution being obviously seen is thoroughly described in the next section.

A radial temperature distribution of the cooking burner in various heights from the burner head is shown in Figure 4. K-type thermocouple was used to measure the temperature at various positions associated to Figure 3a. Temperatures at the center of burner (r = 0 cm) were the highest value at all heights, being related with the photograph of flame in Figure 3a and temperature distribution in Figure 3b. The temperature decreased when the radial distance and the height increased. The maximum temperature obtained from measurement was 1,210 K at height of 1 cm and radial distance of 0 cm. In this study, the temperatures were used to validate the CFD modeling. From comparing at all positions, the temperatures from CFD model were good agreement with the experiment, being the errors of less than 5.86%. Hence, the CFD modeling was able to apply for understanding the flow feature and combustion phenomena of the burner.

Figure 5 shows temperature distribution at the burner head in the mid-plane. The flame shown by the high temperature is in the restrict region inside the ring region, having the high temperature of 1,286 K. The flame did not disperse out, but focused into the middle region. It affected on reducing heat loss, which it can be observed the low temperature of hot gas being escaped into the atmosphere. Thus, it is implied that the ring is the main part to control the flame structure. This may be one reason that the energy-saving cooking burner has higher thermal efficiency than KB burner as the previous mention.



(a) Experiment



(b) CFD model

Figure 3: Flame shape.

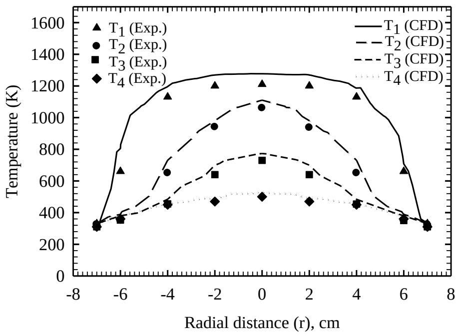


Figure 4: Temperature distribution obtained from experiment and CFD model.

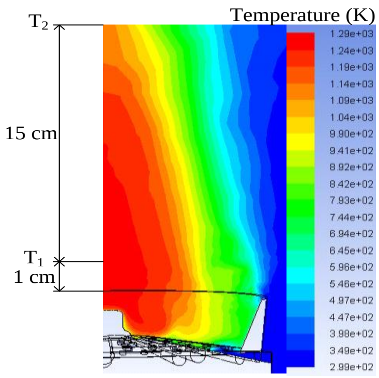


Figure 5: Temperature distribution at the burner head in the mid-plane.

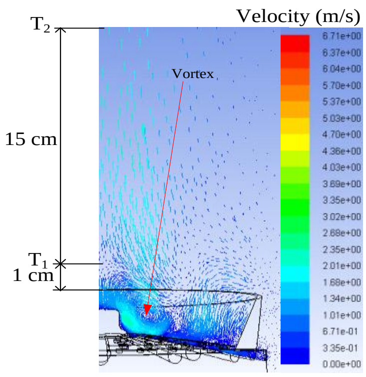


Figure 6: Velocity field at the burner head in the mid-plane.

Figure 6 shows velocity field at the burner head in the mid-plane. The LPG-air mixture coming out of the burner holes and an entrained secondary air can be observed. The maximum velocity of 6.7 m/s occurs in the vicinity of the burner holes which are the smallest length scales related with the flow. It can be obviously observed that the secondary air is naturally entrained from ambient coming into combustion region via the slots, which enhances the combustion efficiency.

This may be the second reason why the combustion of the burner can provide the blue flame, the complete combustion, at all combustion regions or gas flow rates. Moreover, high intensive vortex of the flow near the burner head inside the ring is obvious observed. It is known well that the combustion intensity and the heat transfer coefficient were enhanced by the vortex. This may be another reasons of the higher thermal efficiency of this burner.

4. Conclusion

Simulation study of an energy-saving cooking burner was presented in this paper. The temperatures of combustion from measurement in radial distance at various heights were applied to validate the CFD modeling. It was found that the CFD results were good agreement with the experimental results, having the errors of less than 5.86%. Based upon the insight of the CFD model, the high temperature region, intensive combustion region, low heat loss and suitable flow feature were understood. The ring with suitable design of burner holes is the major part of the burner to control flame structure. However, phenomena of heat transfer to the vessel have to be clarified by the CFD modeling in the further study. Moreover, improving thermal efficiency of the energy-saving cooking burner should be done in the further study.

Acknowledgement

This research is financially supported by Ubon Ratchathani University.

References

[1] Jugjai, S., et al., Parametric Studies of Thermal Efficiency in a Proposed Porous Padiant Recirculated Burner (PRRB): A Design Concept for the Future Burner. Proceedings of RERIC International Energy Journal Vols, 1996(18): p. 97-111.

[2] Jugjai, S., et al., Thermal efficiency improvement of an LPG gas cooker by a swirling central flame. International Journal of Energy Research Vols, 2001(8): p. 657–674.

[3] Shuhn-Shyurng, H., et al., Influence of oblique angle and heating height on flame structure, temperature field and efficiency of an impinging laminar jet flame. International Journal of Energy Conversion and Management Vols, 2005(46): p. 941–958.

[4] Li, H.B., e al., Thermal performances and CO emissions of gas-fired cooker-top burners. International Journal of Applied Energy Vols, 2006(83): p. 1326–1338.

[5] Shuhn-Shyurng, H., et al., Efficiency and emissions of a new domestic gas burner with a swirling flame. International Journal of Energy Conversion and Management Vols, 2007(48): p. 1401–1410.

[6] Boggavarapu, P., et al., Thermal Efficiency of LPG and PNG-fired burners: Experimental and numerical studies. International Journal of Fuel Vols, 2014(116): p. 709–715.

[7] Matthujak, A., et al., Study on thermal efficiency improvement of energy-saving gas stove by swirling flow. in Conference on Energy Network Thailand (ENETT 12) (in Thai) Forum. 2016.

[8] Anetor, L., et al., Reduced Mechanism Approach of Modeling Premixed Propane-Air Mixture Using ANSYS Fluent. Engineering Journal Vols, 2012(16).

[9] Wichangarm, M., et al., Simulation of flow characteristics in an energy-saving high-pressure gas stove. in Conference of Mechanical Engineering Network Thailand (ME-NETT 30) (in Thai) Forum. 2016.