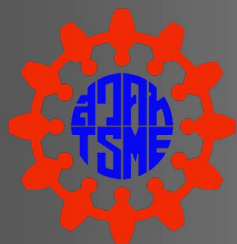
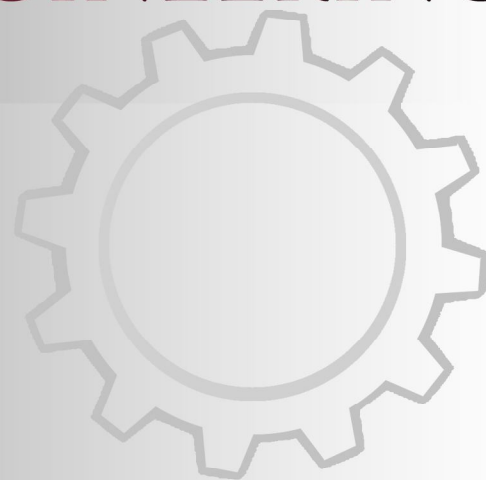


AN INTERNATIONAL JOURNAL OF
THE THAI SOCIETY OF MECHANICAL ENGINEERS



JOURNAL OF RESEARCH
AND APPLICATIONS IN
MECHANICAL ENGINEERING



CONTENTS

- | | | | |
|----|--|----|--|
| 1 | Letter from the Editor | 25 | Novel Boundary Conditions for Turbulent Flows Enclosed by Porous Media
<i>Arpiruk Hokpunna, Barbara Wohlmuth and Somchai Wongwises</i> |
| 3 | Nanostructures and Oxidation Kinetics of Diesel Particulate Matters
<i>Preechar Karin, Hiroshi Oki, Katsunori Hanamura
and Chinda Charoenphonphanich</i> | 31 | Prediction of Total Equivalence Ratio for a Self-Aspirating Burner
<i>Apinunt Namkhat and Sumrerng Jugjai</i> |
| 9 | The Prediction of Nominal and Off-design Characteristics of Economised
Vapour Injection Scroll Compressor-driven Heat pump
<i>Minh Q. Nguyen and Neil J. Hewitt</i> | 37 | The Effect of Baffles on Fluid Sloshing inside the Moving Rectangular Tank
<i>Krit Threepopnartkul and Chakrit Suvanjumrat</i> |
| 15 | An Experimental Study of Thermo-syphon Solar Water Heater
in Thailand
<i>Nicha Sritong, Kritsarapond Krajangkul,
Supachart Chungpaibulpatana and Bundit Limmeechokchai</i> | 43 | The Two-Phase Microchannel Flow Study of Chicken Blood on Lab-on-a-Chip
<i>A. Pramuanjaroenkij, K. Thanomsit, B. Chaiburi, G. Phonong,
A. Khanaphol, W. Phutthasawong, T. Maturros, T. Lomas, A. Tuantranont,
S. Kakaç and C. Thipyopas</i> |
| 19 | Effect of Tube Length on Combustion Characteristics of a Self-aspirating
Radiant Tube Burner (SRTB)
<i>Chanon Chuenchit and Sumrerng Jugjai</i> | 49 | Time to Seizure Comparisons between Different Plain Journal Bearings
<i>James Moran and Phonlathep Tangsukkasemsan</i> |

Editor

Tawit Chitsomboon

International Advisory Board

Adrian Bejan, Duke University, USA	Chairman
Jens N. Sørensen, Technical University of Denmark, Denmark	Member
Withaya Yongchareon, Chulalongkorn University, Thailand	Member
Sylvie Lorente, National Institute of Applied Sciences, France	Member

Editorial Board

Mongkol Mongkolwongrojn

Phadungsak Rattanadecho

Pongjate Promwong

Somchai Wongwises

Somnuek Theerakoolphisoot

Sujin Bureerat

Sumrerng Jugjai

Tanongkiat Kiatsiriroat

Worawut Wisutmethangoon

Assistants to the Editor

Atit Koonsrisuk

Chalothorn Thumthae

Published in Thailand by
Journal of Research and Applications in Mechanical Engineering
School of Mechanical Engineering, Institute of Engineering
Suranaree University of Technology,
111 University Avenue, Nakhon-Ratchasima 30000, Thailand
Tel. 66-44-224410, Fax. 66-44-224613
E-mail : jrame@sut.ac.th Website : <http://eng.sut.ac.th/me/JRAME>

ISSN 2229-2152

Letter from the Editor

Dear Readers,

It has been about a year since the appearance of our inaugural issue. This historic 2nd issue is in print at the time of ICOME 3rd meeting in Chaingrai, Thailand. In this issue all the papers are in English; this is per the advice of a TSME committee meeting.

Three papers were submitted directly from authors; the rest were selected from the 2011 ICOME Krabi conference. We based our selection on papers that are well-rounded in areas of ME, have potential high impacts in applications as well as research, well written and proper in English.

We hope that the readers will find articles in this issue to be useful in practice as well as in promoting new ideas. We further hope that the readers will subscribe to JRAME and also contribute to JRAME by submitting their manuscripts.

As per the advice of TSME, from now on JARME will be upgraded into a quarterly journal and we will need many more manuscripts from the readers. Please join us so that together we can better serve the ME community.

Sincerely,

JRAME Editor and Editorial Staffs

Nanostructures and Oxidation Kinetics of Diesel Particulate Matters

Preechar Karin^{*1}, Hiroshi Oki², Katsunori Hanamura² and Chinda Charoenphonphanich³

¹ International College, King Mongkut's Institute of Technology Ladkrabang, Bangkok, Thailand 10520

² Faculty of Mechanical and Control Engineering, Tokyo Institute of Technology, Tokyo, Japan 152-8552

³ Faculty of Engineering, King Mongkut's Institute of Technology Ladkrabang, Bangkok, Thailand 10520

Abstract

Diesel particulate matters (PMs) must be removed from the exhaust gas emitted from diesel engines to protect the environment and human health; therefore, regulation of vehicle emissions has become increasingly strict. The nanostructures of diesel particulate matters emitted from an actual diesel engine and a diffusion flame burner were investigated by using a transmission electron microscopy (TEM) for better understanding. The single particulate's sizes of both engine and burner were approximately 20-80 nm. The various size of particulate might be strongly related to drag and shear forces of fluid flow, Brownian motion force of gases molecules and electrostatic forces of charges carbon elements, even though such forces is the order of Pico-Newton. Thermo-gravimetric analysis (TGA) was used to investigate chemical kinetics of PM oxidation. The apparent activation energies of engine's PM oxidation were approximately 105kJ/mol and 248kJ/mol for hydrocarbon and carbon zones, respectively. On the other hand, the apparent activation energies of lamp's PM oxidation were approximately 139kJ/mol and 218kJ/mol, respectively. Consequently, much amount of soluble organic fraction (SOF) emitted from an actual engine may be strongly affected to the low apparent activation energy at the low temperature oxidation zone. Similarly, an internal combustion engine operates with very high temperature and pressure. Structure of soot emitted from diesel engine may be strong carbon bonding resulting in increasing of the apparent activation energy of carbon oxidation.

Keywords: Diesel particulate matter, Diesel particulate filter, Hydrocarbon, Apparent activation energy.

1. Introduction

Among internal combustion engines, diesel engines have the highest thermal efficiency. However, particulate matters (PMs) must be removed from exhaust gases that are emitted by diesel engines to protect the environment and human health. Diesel particulate matters consist of a solid fraction (SOL) and a soluble organic fraction (SOF). Primary particles, composed of carbon and metallic ash, are coated with SOF and sulfate. The mean diameter of the primary and agglomerated particles is usually in the range of 20–80 nm and 80–300 nm, respectively. The composition of particles from a diesel engine may vary widely depending on the operating conditions and fuel composition [1–5].

The nanostructures of primary soot particles have been characterized using transmission electron microscopy (TEM) to understand them in detail. A primary soot particle has two distinct parts: an inner core and an outer shell. The inner core has a diameter of 10 nm and it is located at the central region of the primary particle [6, 7].

The electric charge of carbon particles sampled from combustion flames and diesel engines have also been reported. Carbon particles in the later stage of growth are assumed to be charged by thermionic emission and electron capture. The number of charges per particle is varies but average is two. The potential energy increases with increasing charge when the radius of the soot particle is fixed, and decreases with increasing radius for a constant charge [8, 9].

Diesel particulate filters (DPFs) play an important role in particulate trapping and oxidation (regeneration of DPFs). A DPF is generally made of ceramic materials, such as cordierite or silicon carbide, consisting of many rectangular channels with alternate channels blocked using cement at each end. The exhaust gas is forced to flow through a channel wall having numerous micron-

scale pores that trap the PM. Furthermore, the collected PM must be oxidized to regenerate the DPF and reduce the back pressure on the diesel engine. A number of studies have been performed to simulate particulate trapping and chemical reactions inside DPFs and provide valuable information in designing them [10–14].

Catalytic fuel additives and catalyst DPFs have been proven to reduce the overall activation energy required for particulate oxidation, resulting in lower-temperature oxidation compared to non-catalyst DPFs [15, 16]. The oxidation reaction of soot has been investigated using thermogravimetric analysis (TGA) and temperature program oxidation (TPO) to evaluate catalytic activities [17–20]. Moreover, the oxidation of soot on conventional and under developing DPFs were also reported [21–23].

Although structure and oxidation of particulate involve complex behaviors, very few information is available to aid in understanding such phenomena. In the present study, micro-, nano-structures and oxidation behaviors are investigated by using TEM and TGA to aid in better understanding and future designing of an ideal DPF configuration.

2. Experimental Procedure

Figure 1 shows an experimental procedure of nano-structures and oxidation behaviors are investigated by using TEM and TGA. A diesel fuel lamp (conventional diesel fuel of JIS K2204 No. 2) was used in the soot generation process. Part of the soot inside the diesel flame was introduced into PM collector through a bypass line. Similarly, a diesel engine (Robin SDG2200, manufactured by Subaru Co. Ltd.) with a displacement of 230 cm³ and a rated output of 2.8 kW was also used in the soot generation process. The fuel injection pressure was approximately 50 MPa. The engine was operated under a constant load of 2.0 kW at a constant speed of 3000 rpm throughout the experiment. Finally, particulate of each group are investigate by TEM, TGA and CHN. For the thermal and

* Corresponding Author: E-mail: kkpreech@kmitl.ac.th, Tel: 02 329 8261-2, Fax: 02 737 2580,

oxidation analysis of oxidizable particulates, the operating conditions are as follow: temperature increases at a rate of 10 °C per minute from 15 °C to 700 °C with oxygen of 10% and nitrogen of 90% atmosphere.

Lamp ➡ PM collector ➡ TEM, TGA, CHN

Engine ➡ PM collector ➡ TEM, TGA, CHN

Fig. 1 Particulate emitted from a lamp and an engine are captured for characterization.

3. Results and Discussion

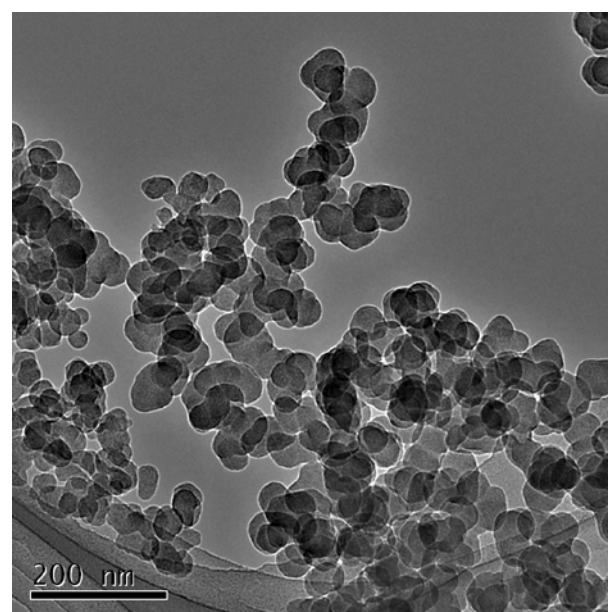
3.1 Structure of PM

Figure 2a and 2b shows TEM images of diesel particulates emitted by the diesel fuel lamp and the diesel engine, respectively. The uniform size of primary (single) of ultrafine particulates can be clearly seen. The single particulate's sizes of both engine and lamp were approximately 20-80 nm. Nanostructures such as density of crystallite and length of each crystallite of pure carbon inside diesel lamp and diesel engine PMs are very similar, as shown in Fig.3a (diesel lamp) and 3b (diesel engine). Figure 4a shows TEM image of homogenous carbon crystallite. The crystallite of carbon was approximately 3–5 nm × 1 nm in length and thickness, respectively. A crystallite of carbon consists of approximately 3,000 atoms. A single particle of PM contains approximately 0.1 to 10 million atoms. A primary soot particle has two distinct parts: an inner core and an outer shell. The inner core has a diameter of 10 nm and it is located at the central region of the primary particle. The size of each crystallite and agglomeration of many crystallites and size of Primary particle might be strongly related to Brownian force of gas molecule, drag and shear forces of fluid dynamics, electrostatics forces of charge elements.

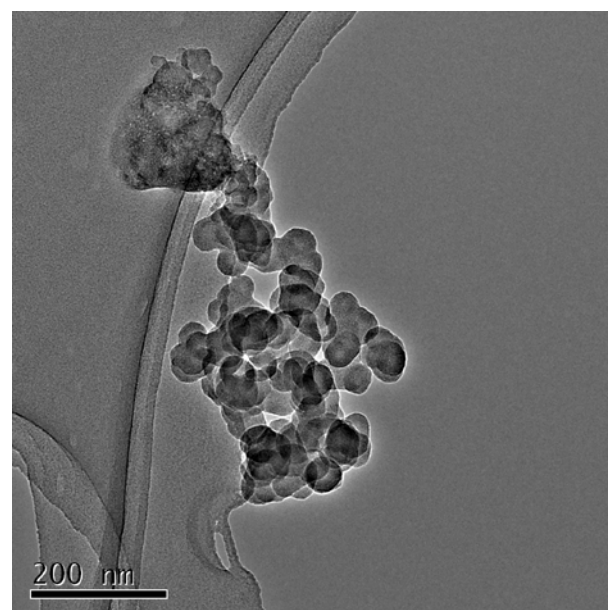
Moreover, some soot was agglomerated with ash (calcium), which included in lubricant oil, as shown in Fig.2b. It was clearly observed the different nanostructure of carbon and calcium. The pattern of carbon crystallite looks like a curve line (4a), whereas the pattern of ash crystallite looks like a straight cross line (4b). Generally, ash is unburned additives of fuel itself and engine lubrication oil.

3.2 Composition of PM

The composition of each PM was characterized by CHN analyzer (oxidized gas analysis) for better understanding. The amount of unburned HC was estimated by using CHN analyzer during oxidation of PM emitted from the diesel engine compared to that on the diesel fuel lamp, as shown in Fig. 5. The mass fraction of C, H, N and others (ash, sulfate etc.) inside PMs were converted from gas analysis during combustion. The mass fraction of C inside PM emitted from the diesel engine smaller than that from the diesel fuel lamp, whereas the mass fraction of H and ash were largest. As a result, particulate emitted by the diesel engine might be composed of largest amount of adsorbed SOF compared to that by the diesel lamp. Moreover, much amount of other composition (ash, sulfate etc.) on particulate emitted from the diesel engine was also observed.

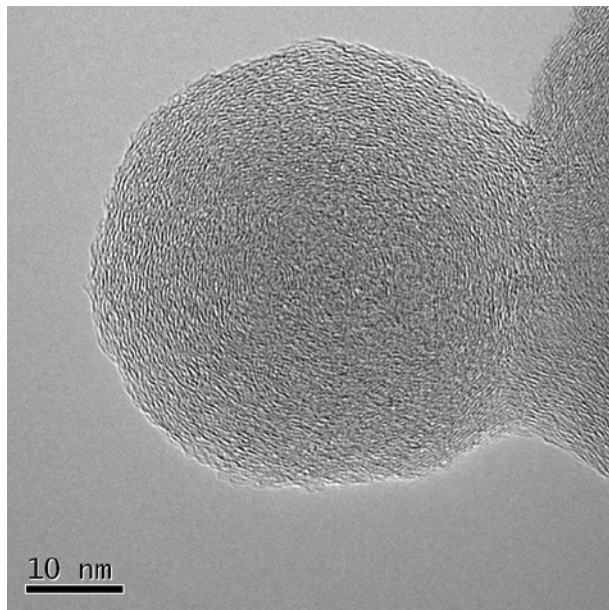


(a)

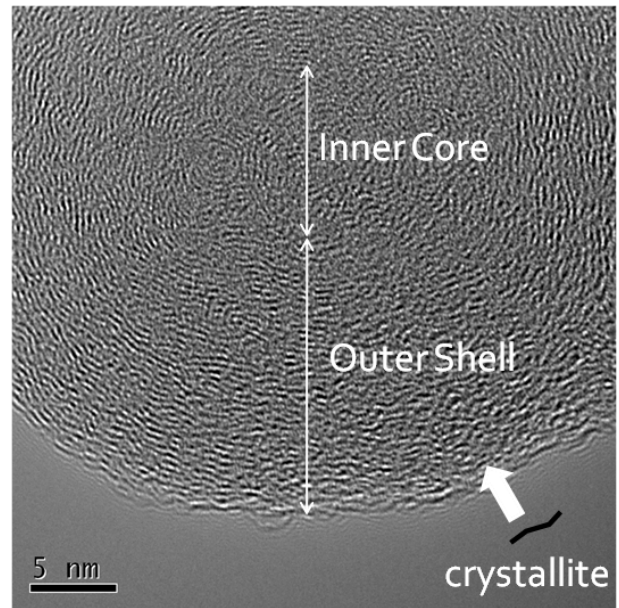


(b)

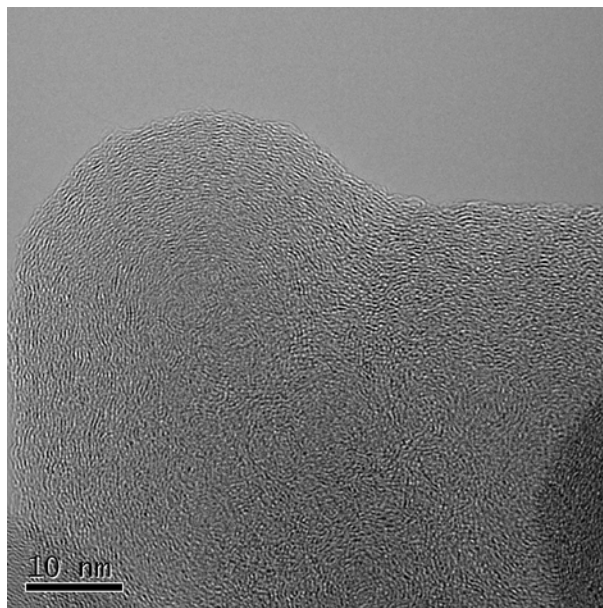
Fig. 2 TEM images of (a) lamp and (b) engine diesel fine particles (accumulate) of PMs.



(a)

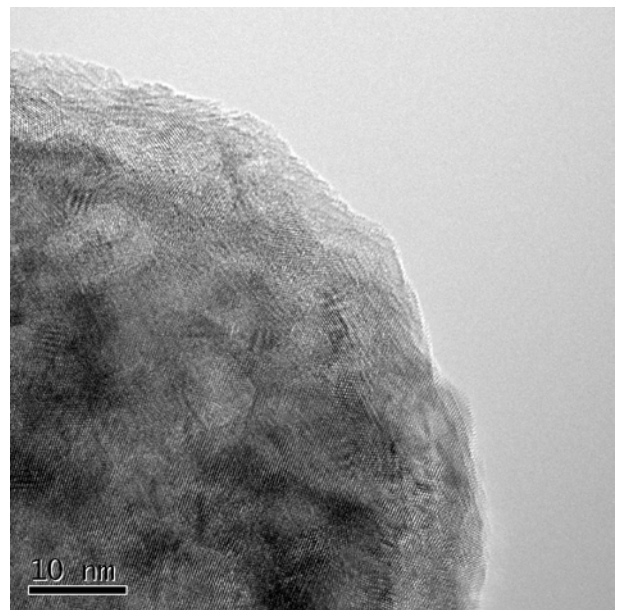


(a)



(b) [23]

Fig. 3 TEM images of (a) lamp and (b) engine diesel ultra fine particles (primary) of PMs.



(b)

Fig. 4 TEM images of (a) carbon (soot) and (b) ash nanostructures of PMs.

The mass conversion during oxidation of their oxidizable PM was also analyzed using the TGA. Figures 6a and 6b show the particulate mass conversion and conversion rate, respectively, with respect to time. The mass of particulate emitted from the diesel engine decreased faster than that from the diesel fuel lamp. There are two peak of mass decreasing rate for PM emitted from the diesel engine, whereas, only one peak of PM mass decreasing rate for the diesel fuel lamp—which very high purity of carbon compared to that on the diesel engine.

The particulate conversion curves may be divided into three regions. In the first region, from 50 °C to 100 °C, it can be considered the evaporation of water molecules. In the second region, from 100 °C to 450 °C, that is assumed to be hydrocarbon oxidation. In the third region, from 450 °C to 700 °C, that is assumed to be carbon oxidation. In this region, the weight of particulate is decreased larger than that of moisture and hydrocarbon regions.

Figure 7a shows the Arrhenius plots used to obtain the apparent activation energy for the lamp's PM oxidation. The Arrhenius plots of PM emitted from the lamp may be divided into two regions. In the first region, as shows in the temperature of 425 to 515 °C, the activation energy is 139.4 kJ/mol. It can attribute to the reaction between strong bonding of hydrocarbon and oxygen. In the second region, shows as the temperature above 515 °C, the activation energy is 218.9 kJ/mol. It can attribute to the reaction between carbon and oxygen.

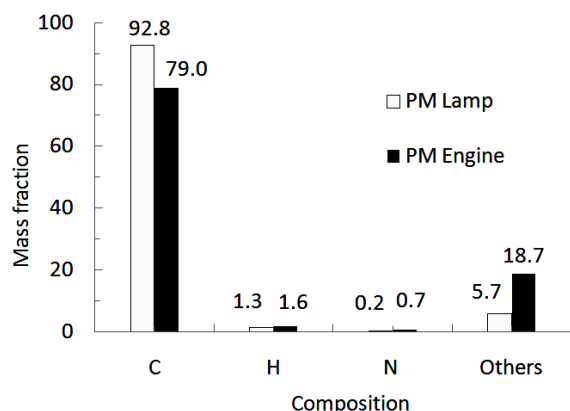
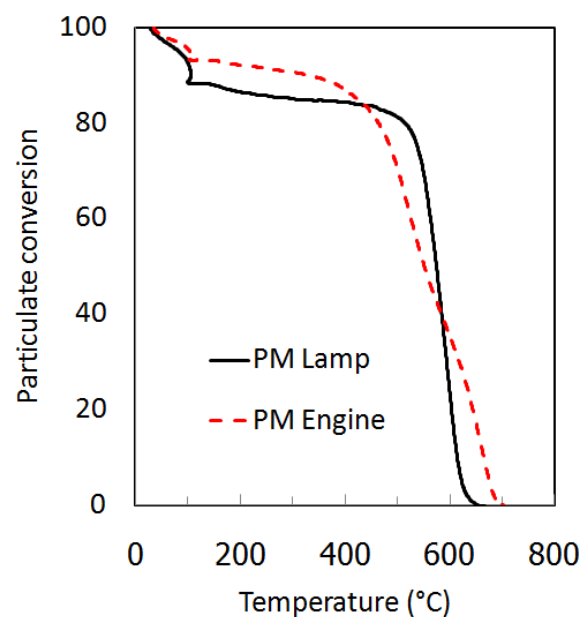
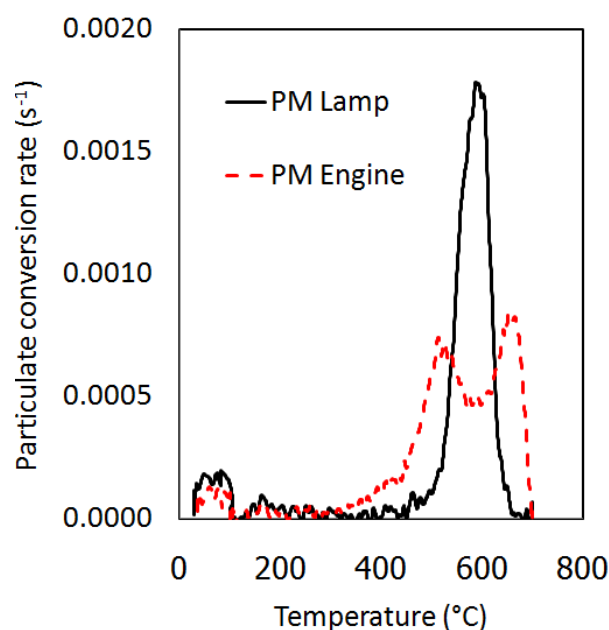


Fig. 5 Compositions of particulate using CHN (Carbon Hydrogen Nitrogen) gas analysis.

Figure 7b shows the Arrhenius plots used to obtain the apparent activation energy for the engine's PM oxidation. The Arrhenius plots of PM emitted from the engine may be divided into three regions. In the first region, as shows in the temperature lower than 425 °C, the activation energy is 48.5 kJ/mol. In the second region, shows as the temperature between 425 to 515 °C, the activation energy is 104.9 kJ/mol. It can attribute to the oxidation of very weak hydrocarbon (unburned fuel) and weak hydrocarbon in the first and second regions, respectively. In the third region, as shown in the temperature above 515 °C, the activation energy is 248.3 kJ/mol. It can attribute to the oxidation of very strong carbon. Table 1 summarizes the apparent activation energy of each PM oxidation.



(a)



(b)

Fig. 6 (a) Particulate conversion and (b) particulate conversion rate by TGA.

Table 1 Apparent activation energy calculated from oxidation rate of each PM.

PM Type	E _a (kJ/mol)		
	<425 °C	425-515 °C	>515 °C
Lamp	-	139.4	218.9
Engine	48.5	104.9	248.3

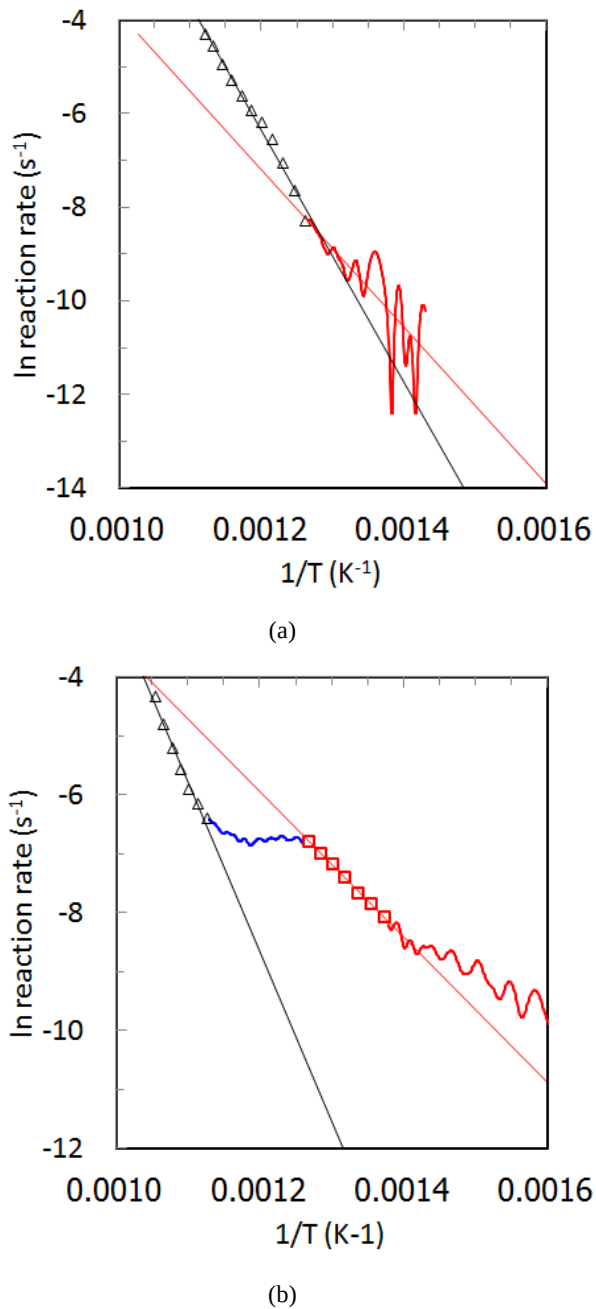


Fig. 7 Arrhenius plot of (a) particulates emitted by (a) lamp and (b) engine.

4. Conclusion

1. Nanostructures of pure carbon inside diesel lamp and diesel engine PMs are very similar, even though, some part of soot was agglomerated with ash, which included in lubricant oil of diesel engine.

2. The mass fraction of carbon inside PM emitted from the diesel engine is smaller than that from the diesel fuel lamp, whereas the mass fraction of H was largest. As a result, particulate emitted by the diesel engine might be composed of largest amount of adsorbed SOF compared to that by the low temperature and pressure of diesel lamp diffusion flame.

3. Soluble organic fraction (SOF) emitted from an actual engine (incomplete combustion) might strongly

affected the low temperature oxidation zone resulting in low apparent activation energy.

4. The strong carbon bonding of engine soot (high temperature and pressure operation) might strongly cause the oxidation apparent activation energy higher than that of lamp soot (low flame temperature and pressure).

5. Acknowledgement

The authors gratefully acknowledge support from Isuzu Research Center and NGK Insulator.

6. References

- [1] Heywood, J.B. (1998). Internal Combustion Engine Fundamental, McGraw-Hill series in mechanical engineering, Singapore.
- [2] Smith, O.I. (1981). Fundamentals of soot formation in flames with application to diesel engine particulate emissions, *Progress in Energy and Combustion Science*, Vol. 7, pp.275-291.
- [3] Maricq, M.M. (2007). Review Chemical Characterization of particulate emissions from diesel engine: A review, *Journal of Aerosol Science*, Vol.38, pp.1079-1118.
- [4] Kittelson, D.B. (1998). Engines and nanoparticles: A review, *Journal of Aerosol Science*, Vol.29, pp.575-588.
- [5] Majewski, W.A. and Khair, M.K. (2006). Diesel Emissions and Their Control, SAE Order No.R-303, SAE International. Warrendale USA.
- [6] Ishiguro, T. Takatori, Y. and Akihama, K. (1997). Microstructure of Diesel Soot Particles Probed by Electron Microscopy: First Observation of Inner Core and Outer Shell, *Combustion and Flame*, 108, pp.231-234.
- [7] Vander Wal, R.A. Yezerets, A. Currier, N.W. Kim, D.H. and Wang, C.H. (2007). HRTEM Study of diesel soot collected from diesel particulate filters, *Carbon*, 45, pp.70-77.
- [8] Ball, R.T. and Howard, J.B. (1971). Electric charge of carbon particles in flames, *Symposium (International) on Combustion*, Vol.13, Issue 1, pp.353-362.
- [9] Maricq, M.M. (2006). On the electrical charge of motor vehicle exhaust particles, *Journal of Aerosol Science*, Vol.37, pp.858-874.
- [10] Konstandopoulos, A.G. Kostoglou, M. Vlachos, N. and Kladopoulos, E. (2005). Progress in Diesel Particulate Filter Simulation, *SAE Technical paper*, 2005-01-0946.
- [11] Konstandopoulos, K.G. Zarvalis, D. Kladopoulou E. and Dolios, L. (2006). A multi-reactor assembly for screening of diesel particulate filters, *SAE Technical paper*, 2006-01-0874.
- [12] Tushima, S. Nakamura, I. Sakashita, S. Hirai, S. and Kitayama, D. (2010). Lattice Boltzmann simulation on particle transport and captured behaviors in a 3d-reconstructed micro porous DPF, *SAE Technical paper*, 2010-01-0534.
- [13] Hanamura, K. Karin, P. Cui, L. Rubio, P. Tsuruta, T. Tanaka, T. and Suzuki, T. (2009). Micro- and macroscopic visualization of particulate matter trapping and regeneration processes in wall-flow diesel particulate filters, *International Journal of Engine research*, Professional Engineering Publishing, Vol.10, No.5/2009, pp.305-321.
- [14] Karin, P. Cui, L. Rubio, P. Tsuruta, T. and Hanamura, K. (2009). Microscopic Visualization of PM Trapping and Regeneration in Micro-Structural Pores of

a DPF Wall, *SAE International Journal of Fuels and Lubricants*, SAE International, Vol.2, No.1, pp.661-669.

[15] Fino, D. and Specchai, V. (2008). Review open issues in oxidative catalysis for diesel particulate abatement, *Powder technology*, Vol.180, pp.64-73.

[16] Suzuki, J. and Matsumoto, S. (2004). Development of catalysts for diesel particulate NOx reduction, *Topics in Catalyst*, Vol.28, pp.171-176.

[17] Neeft, J.P.A. Nijhuis, T.X. Smakman, E. Makkee, M. and Moulijn, J.A. (1997). Kinetics of the oxidation of diesel soot, *Fuel*, Vol.76, No.12, pp.1129-1136.

[18] Darcy, P. Costa, P.D. Mellottee, H. Trichard J.M. and Mariadassou, G.D. (2007). Kinetics of catalyzed and non-catalyzed oxidation of soot from a diesel engine, *Catalysis Today*, Vol.119, pp.252-256.

[19] Yezerets, A. Currier, N.W. and Eadler, H.A. (2003). Experimental Determination of the Kinetics of Diesel Soot Oxidation by O₂ – Modeling Consequences, *SAE Technical paper*, 2003-01-0833

[20] Lorentzou, S. Pagkoura, C. Zygogianni, A. Kastrinaki, G. and Konstandopoulos, A.G. (2008). Catalytic nano-structured materials for next generation diesel particulate filters, *SAE Technical paper*, 2008-01-0417.

[21] Karin, P. and Hanamura, K. (2010). Particulate Matter Trapping and Oxidation on Catalyst-Membrane, *SAE International Journal of Fuels and Lubricants*, SAE International, Vol.3, No.1, pp.368-379.

[22] Karin, P. and Hanamura, K. (2010). Particulate Matter Trapping and Oxidation on a Diesel Particulate Filter, *The First TSME International Conference on Mechanical Engineering*, 2010.

[23] Oki, H. Karin, P. and Hanamura, K. (2010). Visualization of Oxidation of Soot Nanoparticles Trapped on a Diesel Particulate Membrane Filter, *SAE International Journal of Engine*, SAE International, 2011-01-0602.

The Prediction of Nominal and Off-design Characteristics of Economised Vapour Injection Scroll Compressor-driven Heat Pump

Minh Q. Nguyen^{1*} and Neil J. Hewitt²

¹College of Technology, University of Danang, Danang City, Vietnam

²Centre for Sustainable Technologies, University of Ulster, UK

Abstract

Economised vapour injection technology based on the integration of a hermetic scroll compressor and a brazed plate heat exchanger is capable of greatly improving vapour compression heat pump efficiency. However there has been a large difference in increased level of performance between off-design and nominal conditions due to the overall heat pump system being designed and built from several components which certainly interact with each other during operation process. The size and configuration of each component are particularly important factors causing the overall system performance to be deviated from nominal condition-based values and subsequently energy losses because of components to be either oversized or undersized unexpectedly. Experimental investigation was conducted to estimate these efficiency deviations in each individual component as well as the whole system under a wide range of working conditions. In comparison with the performance reference predicted by the Copeland Selection Software, the experimental results show a tendency of increasing deviation between the measured and predicted performances when the compressor power is increased while the experimentally measured heating capacities are slightly deviated from the nominal values throughout the operating range.

Keywords: Heat pump, Scroll compressor, Economised vapour injection, Heating capacity, Coefficient of performance

1. Introduction

Electric heat pumps are one of a range of heating devices that can be thought of as energy-efficient and environmentally friendly systems. They offer large energy savings due to the fact that a well-designed system will deliver considerably more heat energy than the primary energy used to drive it while also contribute to significant reduction of CO₂ emissions compared to other heating systems. If the electricity is produced from renewable sources, this benefit is further increased. However the energy savings and potential environment benefits offered by electric heat pump have been insufficient to enable them to become the first-choice heating system for domestic and commercial buildings because of inherent energy losses of vapour compression cycle on which heat pump is operated. Therefore improved vapour compression cycles including economised vapour injection (EVI) technique have been developed to overcome restrictions and limitations of the conventional vapour compression cycle in order to eliminate thermodynamic losses and enlarge the operation range of heat pump.

2. Heat pump with EVI technology

The economised vapour injection technology applying to vapour compression heat pump is a novel design concept using either an internal subcooling heat exchanger or a flash tank connected to an immediate pressure injection line of single two-stage scroll compressor. By subcooling the liquid refrigerant the flash gas formed during the expansion process is reduced and subsequently inducing a lower quality of refrigerant entering the evaporator contributing to improved heat transfer performance in the evaporator. On the other hand, by injecting cooled vapour into the scroll compressor at immediate pressure, the compressor work input and discharge temperature are reduced, resulting in

decreased power consumption of the compressor and increased heat transfer effectiveness of the condenser. Figure 1 describes schematic principle of such two EVI heat pump cycles with internal subcooling heat exchanger and flash tank and its P-h diagrams.

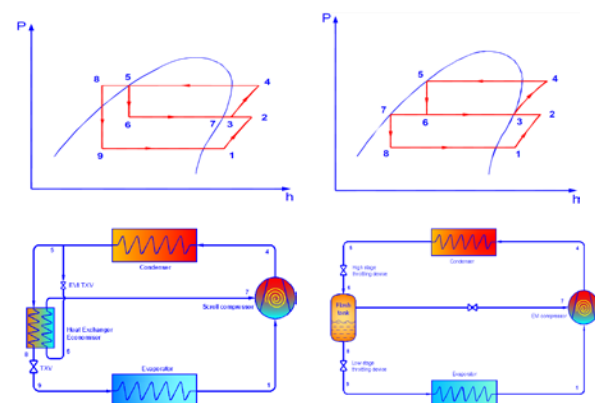


Fig. 1 EVI cycles and P-h diagrams

Many studies have been conducted both theoretically and experimentally to enhance the performance of conventional heat pump by using EVI technology integrated with either internal heat exchanger or flash tank [1-5]. The results showed significant large augmentation in overall efficiency, especially when operating in high temperature lift conditions which are disadvantageous for traditional heat pump. Some other intensive researches also demonstrated much further benefits obtainable from refrigerant injection as well as the effect of varying injection conditions on overall performance and compressor stability [6-7]. However the experimental systems developed in above mentioned

*Corresponding Author: E-mail: ngminh@ud.edu.vn, Tel: +8420X511 3894883, Fax: + 84 20X 511 389 4884

studies are probably constructed and tested based on off-design conditions which are entirely different from nominal values provided from manufacturer. The present paper predicts experimentally this deviation between nominal and off-design performances of EVI air-source heat pump for heating applications.

3. Experimental apparatus and data reduction

The heat pump prototype tested in the present study was developed based on vapour compression refrigeration system which consists of a Copeland EVI scroll compressor, two brazed plate-type heat exchangers for condenser and internal subcooler, a cylindrical finned tube heat exchanger for evaporator and two thermostatic expansion valves for main and subcooling circuits. Refrigerant R407C was selected with a critical temperature at 86.74°C corresponding to a critical pressure of 46.191bar. The instrumentation was designed to measure temperature, pressure, humidity and flow rate of refrigerant, water and air as well as to monitor operational stability of the overall system during test processes. Data collection and system control were achieved by using a data acquisition system connected to a personal computer. A schematic diagram of such a test apparatus and measuring equipment as well as component layout in the test bed are illustrated in Figures 2 and 3.

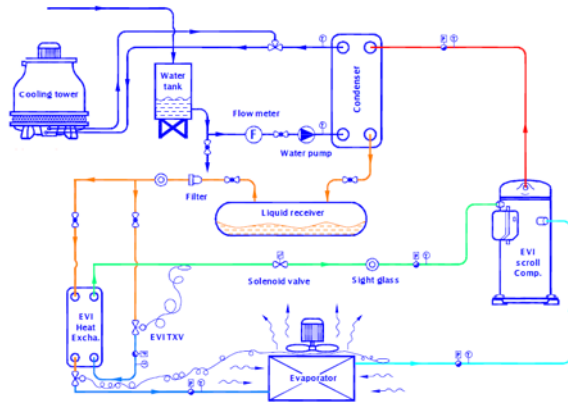


Fig. 2 Schematic diagram of test apparatus

Test conditions and procedures are in accordance with European Standards EN14511-2 and EN14511-3 respectively, regarding the rating and performance of an air-to-water heat pump with an electrically driven compressor. Details of test conditions are listed in Table I.

The operating effectiveness of heat pump system is represented by heating capacity Q_h^w and coefficient of performance COP_h on the water-side that both are calculated based on measured data as follows:

$$Q_h^w = m_w \int C_p^w dT_w = m_w (t_w^{out} - t_w^{in}) \quad (1)$$

$$COP_h^w = \frac{Q_h^w}{W} \quad (2)$$

Table I. Test conditions

Order of test	Air-on temperature (°C)	Water-off temperature (°C)
1	-12	60
2	-12	50
3	-7	60
4	-7	50
5	-2	50
6	2	60
7	2	50
8	2	35
9	7	60
10	7	50
11	15	60

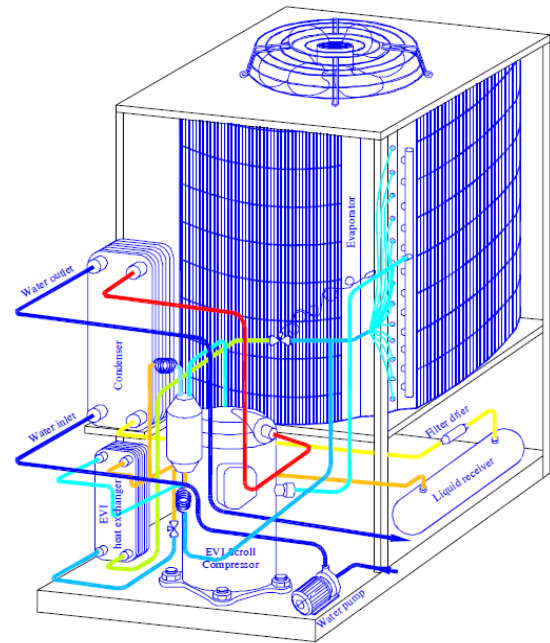


Fig. 3 Layout of components in test bed

In order to verify the validity of the measured data on the water-side, an energy balance using the thermodynamic laws is established in each test to calculate heating capacity Q_h^r and coefficient of performance COP_h^r on the refrigerant-side as follows:

$$Q_h^r = m_r \int C_p^r dT_r = m_r (i_r^{out} - i_r^{in}) \quad (3)$$

$$COP_h^r = \frac{Q_h^r}{W} \quad (4)$$

The error in heating capacities calculated on the water-side and refrigerant-side can be determined by the following equation and should be less than 5%:

$$\Delta Q(\%) = \frac{Q_h^r - Q_h^w}{Q_h^w} \times 100 \quad (5)$$



Fig. 4 Test bed of EVI heat pump

Figure 4 shows a photograph of the experimental apparatus installed and commissioned in a test chamber along with all measuring equipment which are calibrated before embedding to the test system.

4. Results and Discussion

A heat pump system in practice is normally designed and assembled from a number of components provided by several different manufacturers. For instance, with test facilities of heat pump system in this study, EVI scroll compressor is provided by Copeland while SWEP brazed plate heat exchangers are used as the main condenser and internal subcooler. Evaporator and expansion valves are a circular shaped coil heat exchanger and Alco TISE types, respectively. So the operating parameters of each individual component are measured in association with the operation of the overall system. Their performance will be compared with the reference data available in the Copeland Selection Software (CSS) at the same working conditions in order to evaluate whether the operation of each component reaching to its nominal characteristics.

Figure 5 illustrates a comparison of compressor power between the actual measured values and reference values predicted by the CSS. Experimental measurements show higher measured values compared to the predicted values derived from the CCS. The tendency has larger deviation when the compressor power is increased.

It was also found that on average the electricity actually used by compressor is approximately 7% greater than that specified by manufacturer. This is due to higher mass flow rate of refrigerant through EVI subcooler that leads to higher EVI capacity in comparison with the reference value obtained from the CSS. Figure 6 shows this capacity increase which means that EVI subcooler is slightly oversized for integrating to the current EVI compressor under off-design conditions.

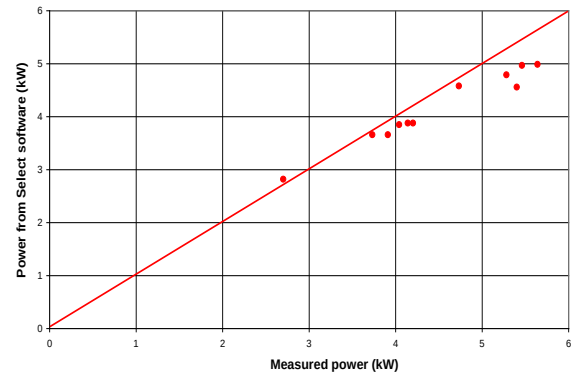


Fig. 5 Measured and predicted compressor power

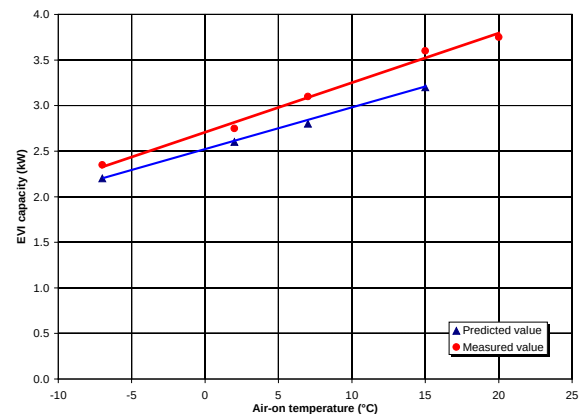


Fig. 6 Measured and predicted heat capacities of EVI subcooler

The performance assessment of SWEP brazed plate heat exchanger was also carried out over a wide range of operating condition. Due to the temperature glide of the refrigerant R407C, the refrigerant temperature for performance evaluation is rated at mid-point state. Figure 7 shows the variation of condenser mid-point temperature difference at several different test conditions of water-off temperature. Due to the condenser designed at the water temperature of 60°C corresponding to the condensing mid-point temperature of 62°C, the measured performance behaviour of the brazed plate heat exchanger have not changed significantly from off-design condition-based performance.

The deviation of condenser mid-point temperature difference over wide range of test conditions is relatively small. This means that the plate heat exchanger used for condenser is designed and sized properly under off-design conditions despite slightly

oversized EVI subcooler causing higher refrigerant flow rate through both EVI compressor and condenser that is not still much to affect the condenser operating behaviour.

The comparison of heating capacity basing on the measured values and the reference data predicted by the CSS is illustrated in Figure 8. Both measured and predicted values seem to remain approximately the same during the range of test conditions. The deviation rate is seen to be small, approximately around $\pm 3\%$.

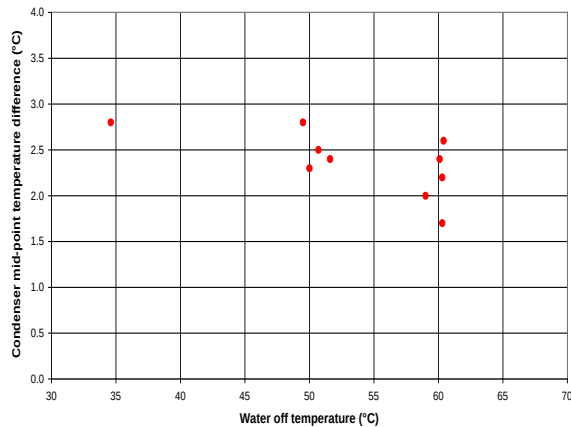


Fig. 7 Condenser mid-point temperature difference

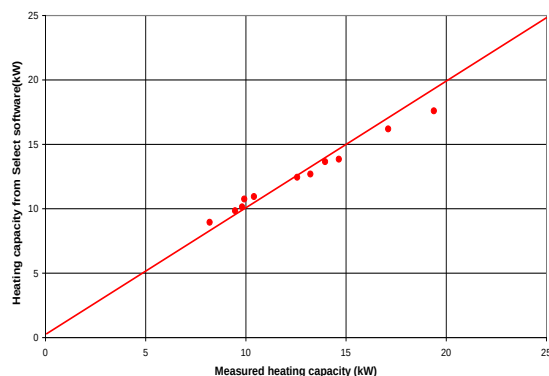


Fig. 8 Measured and predicted heating capacities of condenser

However the deviation behavior is different for every different test condition. For the cases of high heating capacity which is greater than 12kW, the measured value were found to deviate by maximum 3% over the predicted value while the inverse tendency was observed at the range of low heating capacities. This may be contributed by the performance of EVI subcooler which is more oversized when operating in high air-on temperature conditions.

Figure 9 describes the deviation in the overall system performance between actual COP calculated from measured data and nominal COP predicted by the CSS. The results show a significant degradation in the actual performance throughout the range of test conditions. As seen in Figure 5, on average the actual COP is reduced about 5.5% compared to the nominal COP specified by manufacturer. The reduced performance of the overall system is certainly induced by highly increased actual

input of compressor power over the predicted power value as already presented above, while the actual heating capacity seems to be kept approximately the same as the predicted value.

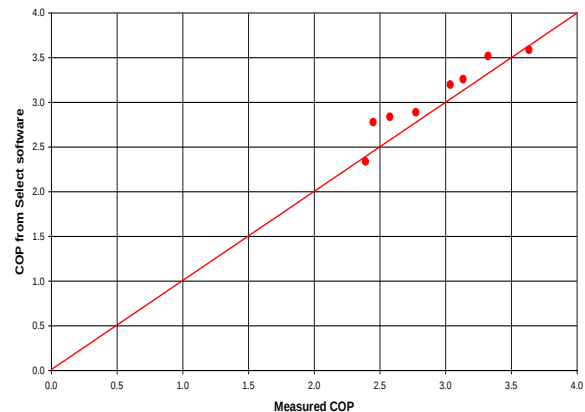


Fig. 9 Actual and predicted COPs of overall system

5. Conclusions

The economised vapour injection technique utilizing in vapour compression cycle has contributed to significant improvements in the performance of air-source heat pump, especially when operating at high temperature lift. However actual increased levels of performance are entirely not similar to those specified by manufacturer under nominal conditions. Although all components of the heat pump system are designed and commissioned basing on off-design conditions which are the same as nominal conditions, the overall system performance is still deviated significantly from the nominal value. This is due to operational interaction among components of system that induces degradation in performance of each individual component and subsequently the overall system. The system modification after design and commission processes is therefore extremely essential to obtain the highest possible efficiency.

6. References

- [1] Beeton, W. and Pham, H. (2003). Vapour-injected scroll compressor, *ASHRAE Journal*, pp.22-27.
- [2] Guayuan, M., Qinhu, G., and Yi, J. (2003). Experimental investigation of air-source heat pump for cold regions, *International Journal of Refrigeration*, vol. 26, pp. 12-18.
- [3] Huang, M., Hewitt, N., and Nguyen, M. (2007). Field testing of an economised vapour injection heat pump, Paper presented in *the International Congress of Refrigeration*, Beijing, China.
- [4] Winkler, J., Wang, X., and Radermacher, R. (2008). Simulation and validation of two-stage flash tank cycle using R410A as a refrigerant, Paper presented in *the International Conference of Refrigeration*, Purdue Univ., USA.
- [5] Guayuan, M. and Huaxia, Z. (2008). Experimental study of a heat pump system with flash-tank coupled with scroll compressor, *Energy and Buildings*, vol. 40, pp. 697-701.
- [6] Winandy, E. and Lebrun, J. (2002). Scroll compressors using gas and liquid injection: experimental analysis and modelling, *International Journal of Refrigeration*, vol. 25, pp. 1143-1156.

- [7] Dutta, A., Yanagisawa, T., and Fukuta, M. (2001). An investigation of the performance of a scroll compressor under liquid refrigeration injection, *International Journal of Refrigeration*, vol. 24, pp. 577-587.
- [8] Hewitt, N., Huang, M., Aderson, M., and Quinn, M. (2011). Advanced air source heat pumps for UK and European domestic, *Applied Thermal Engineering*, vol. 31, pp. 3713-3719.

An Experimental Study of Thermo-syphon Solar Water Heater in Thailand

Nicha Sritong, Kritsarapond Krajangkul, Supachart Chungpaibulpatana and Bundit Limmeechokchai*

Mechanical Engineering Program, Sirindhorn International Institute of Technology, Thammasat University
P.O.Box 22, Thammasat Rangsit Post Office, Klong Luang, Patum Thani 12121, Thailand

Abstract

The thermo-syphon solar water heater (TSSWH) is probably the most widespread application of solar thermal energy during the past years. This paper presents the experimental study and the mathematical models for prediction of hot water temperature produced from the TSSWH. Results are presented of collector temperature, storage temperature and thermal efficiency of the TSSWH. From the experimental study, it is found that the average TSSWH thermal efficiency was about 56% during the test days. The characteristics of this TSSWH are then used in simulation to predict its annual performance in Thailand using the three-year weather data during 2006-2008. Results from the simulation showed that the average year-round thermal efficiency is about 48%, average collector temperature is 51°C, and average hot water in the storage tank is 46°C. Finally, the technical and economic parameters of the TSSWH are used to assess its cost effectiveness under conditions in Thailand. Results from economic analysis revealed that TSSWH is competitive to the conventional electric heaters, especially when electricity price is higher, utilization factor is higher and the first cost of the TSSWH is lower.

Keywords: Thermo-syphon solar water heater, Thermal efficiency, Collector temperature, Storage tank temperature

1. Introduction

Solar energy is a primary source of Earth's energy commonly used to supply heat and electricity. The flat-plate solar thermal collectors absorb and diffuse radiation from solar energy and transform them into heat. They are usually fastened on rooftop or wall facing the sun to absorb the maximum radiant possible. It should be tilted at specific tilt-angle to maximize the solar radiation collected from the flat-plate solar thermal collector. In Thailand it is best to be tilted at 14°, facing south [1]. Flat-plate solar collector is made of a simple glass topping on the insulated box with a flat solar absorber made from sheet metal attached to copper pipes painted in black. The cover plates of the box are used to reduce convection and radiation heat losses to atmosphere while back insulation is used to reduce conduction losses. This absorbed solar energy is then used for heating the working fluid. The efficiency of the solar water heater depend on the design of the solar water heater and the amount of solar radiations [2].

The TSSWH is a system used to produce hot water. It is considered as an appropriate technology for general heating purposes. The three-year weather data from Thai Meteorological Department during 2006-2008 show that the average solar energy received for 14 hours a day, is 16.65 MJ/m² and the maximum solar energy receive from the whole year is 20.20 MJ/m²

There are two main types of solar water heater using today [3], which depend on the amount of solar energy at different locations. The first type is called the "active system". The active system includes a pump to circulate water between the collector and the storage tank, and a controller to turn the pump on and off. This pump operates only when the water in the collector is hotter than the water in the storage tank. Both pump and controller are driven by electricity. The second type is called the "Thermo-syphon", or sometime called "passive system", as shown in Fig. 1. This type of solar water heater does not depend on the use of pump but by natural convection for water to flow through the system. Here the tank itself acts

as solar storage. They are pressurized and usually depended on gravity flow and convection of hot water to circulate water from collector to tank

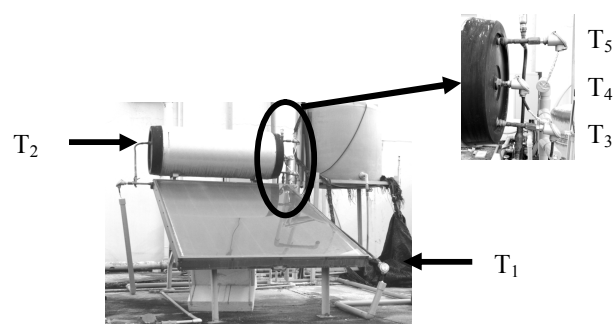


Fig. 1 The thermo-syphon solar water heater.

However, the system in this experiment falls into the second type of solar water heater, i.e. the passive system. The system is closed so the flow of water is due to the difference in water densities.

2. Experimental Setup

The TSSWH consists of one 2.16 m² flat-plate solar thermal collector, a storage tank with capacity of 150 litres placed horizontally above the solar thermal collector and the connecting pipes. Connecting pipes allow water to flow through the flat-plate collector to the storage tank and out of the storage tank to the collector for reheating. The pipes are covered with an insulated material wrapped with another layer of aluminum-foil sticky tape to prevent heat losses.

Water temperatures from different points and ambient temperature are measured by type K thermocouple sensors. These thermocouple wires run from the thermocouple sensors to the cabinet put up on the wall nearby. The cabinet stored two temperature displays. The upper one shows temperatures from point 1 to point 4 and

*Corresponding Author: E-mail: bundit@siit.tu.ac.th, Tel: +662-986-9009, Fax: +662-986-9113

the lower one shows temperature at point 5. The ambient temperature is also collected. Thermocouples from point 1 to 5 measure temperatures of water at different locations see Fig. 1. The temperature at point 1 (T_1) is the temperature of water entering the flat-plate solar thermal collector. The second point (T_2) is the temperature of water exiting from the flat-plate solar thermal collector. Point 3, 4 and 5 measured the temperatures of water stored in the storage tank where T_3 is the temperature of water at the bottom of the tank, T_4 is the temperature of water at the middle of the tank and T_5 is the temperature at the upper part of the tank. The last point is the temperature of air, ambient temperature T_a , where the thermocouple sensor is set up in a caged device (see Fig. 2).

The solar intensity is measured by a pyranometer shown in Fig. 2. It is tilted at the same angle with the flat-plate thermal solar collector.

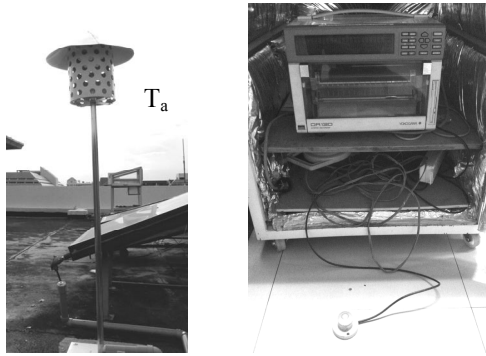


Fig. 2 Data collection system.

The experiment was done on sunny clear days. Before collecting any data, the temperatures of water storage measured by the three thermocouple sensors (T_3 , T_4 and T_5) must show the same temperatures. Then water in the storage tank is drained out because water left inside has higher temperature than the tap water. After the storage tank was emptied, it was then filled with tap water. As T_3 , T_4 and T_5 are equal the valve is closed and so is the system. The first set of temperatures and solar intensity were recorded in the morning times. During the start of the day, inlet temperature (T_1) and ambient temperature (T_a) usually show very close temperatures. All temperatures including solar intensity (T_1 , T_2 , T_3 , T_4 , T_5 , T_a and I_t) are recorded every 15 minutes, and continued for nine hours. The last set of data recorded is at 5:30 pm.

The outlet water temperature (T_2) increases during the experiment. The hot water caused a decrease in density of water. Hot water moved to the top of the storage tank. Cooler water is then pushed down to the bottom part of the storage tank. Tube connected at the bottom part of the tank allowed cooler water to circulate back into the flat-plate solar thermal collector to be reheated so water in the storage tank could reach its maximum temperature. This circulation continued until T_2 stop increasing or solar intensity is zero. These results were recorded for three days in January.

3. Analysis

This experiment could be evaluated and solved for parameters needed to calculate for the efficiency [4]. To find the collector efficiency, several equations are needed. The useful energy (Q_u) collected by the collector per unit time can be determined by mass flow rate m , inlet

temperatures (T_1) and outlet of the collector (T_2), which can be expressed by Eq.(1).

$$Q_u = m c_p (T_2 - T_1) \quad (1)$$

where c_p is the heat capacity of water $c_p = 4186 \text{ J/kg}\cdot\text{K}$. The useful energy can be found by the relation of energy change in the storage tank and the heat losses:

$$\text{Energy change in the storage tank} = (F \times \text{Useful energy from the collector}) - \text{Heat Losses} \quad (2)$$

where the control function F is either 1 or 0 depending on whether the outlet temperature of the collector (T_2) is greater or less than the storage tank temperature (T_s) respectively.

$$\begin{aligned} T_2 < T_s, & \quad F = 0 \\ T_2 > T_s, & \quad F = 1 \end{aligned}$$

where T_s is the current storage tank temperature or the average temperature of T_3 , T_4 and T_5 .

$$T_s = (T_3 + T_4 + T_5)/3 \quad (3)$$

The energy change in the storage tank per unit time Q_s is found by:

$$Q_s = \frac{\rho V C_p}{\Delta t} (T_s^+ - T_s) \quad (4)$$

where ρ is density of water (kg/m^3), V is the volume of water in the storage tank (150 litres), Δt is the time interval in second ($15 \times 60 = 900\text{s}$). T_s^+ is the storage tank temperature in the next period ($^\circ\text{C}$) that will be evaluated further. The heat losses defined in Eq.(2) come from two parts:

Connecting pipe,

$$\text{Pipe loss} = hA_p \left[\left(\frac{T_3 + T_1}{2} \right) - T_a \right] \quad (5)$$

where, hA_p is the heat loss coefficient of the pipe. The heat loss coefficient per meter long, hA_p/l_p is $0.764 \text{ W/m}\cdot^\circ\text{C}$ and the l_p , total pipe length, measured from the tested thermo-syphon solar water heater, is 1.8 m long. As l_p is cancelled out, hA_p is equal to $1.3752 \text{ W/}^\circ\text{C}$.

Storage tank,

$$\text{Storage tank loss} = (UA)_s (T_s - T_a) \quad (6)$$

where $(UA)_s$ is the overall heat loss coefficient, $(UA)_s = 1.836 \text{ W/}^\circ\text{C}$.

The collector efficiency η is defined as the ratio of the useful heat gain over the same specified

time period to the incident solar intensity over the same time interval.

$$\eta = \frac{\int Q_u dt}{A_c \int I_t dt} \quad (7)$$

where Q_u is the useful energy from the collector (W), A_c is the collector area (2.16 m^2) and I_t is the total hourly radiation (W/m^2).

The solar intensity I_t as read from the calibrated pyranometer showed the unit in voltage (mV). To transform unit from mV to W/m^2 , a conversion factor ($1 \text{ mV} = 991.1 \text{ W/m}^2$) of the pyranometer is needed. T_c is the average temperature of T_1 and T_2 as shown in Eq.(8).

$$T_c = \frac{T_1 + T_2}{2} \quad (8)$$

The efficiency function f_c is given in Eq.(9).

$$f_c = \frac{(T_c - T_a)}{I_t} \quad (9)$$

The efficiency curve shown in Fig. 3 is obtained by plotting out the efficiency function values f_c against the collector efficiency values η .

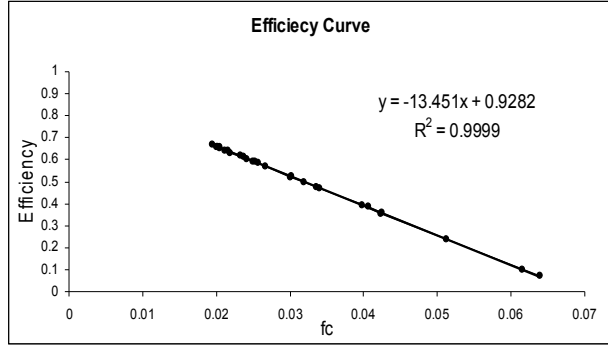


Fig. 3 Efficiency curve of the tested TSSWH.

The useful energy from the collector Q_u is evaluated by using Eq.(10).

$$Q_u = A_c(\tau\alpha)I_t - A_cU_L(T_c - T_a) \quad (10)$$

where $\tau\alpha = 0.9387$ and $U_L = 13.61 \text{ W/m}^2$

The predicted temperatures of the storage tank in the next period can be determined from Eq. (13).

From Eq. (2)

$$mC_p(T_s^+ - T_s) = F \times Q_u - \text{Losses} \quad (11)$$

$$mC_p(T_s^+ - T_s) = \{F[A_c(\tau\alpha)I_t - A_cU_L(T_c - T_a)\Delta t] - hA[(T_3 + T_1)/2 - T_a] - (UA)_s(T_s - T_a)\} \quad (12)$$

After rearranging, T_s^+ is obtained.

$$T_s^+ = T_s + \frac{\Delta t}{mC_p} \{F[A_c(\tau\alpha)I_t - A_cU_L(T_c - T_a)] - hA[(T_3 + T_1)/2 - T_a] - (UA)_s(T_s - T_a)\} \quad (13)$$

4. Results and Discussion

This experiment determines the system efficiency of the tested TSSWH, and predicts the year-round efficiency using three-year weather data in Thailand. The energy change in the storage tank Q_s is found from Eq.(4). To find the useful energy from the collector Q_u , losses from Eq.(5) and Eq.(6) must be added to the energy change in the storage tank Q_s . Next the efficiency function f_c is found from Eq.(9) and the collector efficiency η from Eq.(7). The efficiency curve in Fig. 3 is plotted. Linear equation was applied to this efficiency curve to find the coefficients in the equation, where the new effective absorptance $\tau\alpha$ is shown as the y-intercept and the new overall heat loss coefficient U_L is shown as the slope of the graph. The new $\tau\alpha$ is found to be 0.9282 while U_L is found to be $13.451 \text{ W/m}^2\cdot^\circ\text{C}$.

Fig. 4 shows average temperatures of water in the storage tank against the time recorded of the whole day. The four lines represent the average water temperatures of the storage tank T_s , the water temperature of the lower part of the storage tank T_3 , the water temperature of the middle part of the storage tank T_4 and the water temperature of the upper part of the storage tank T_5 . This graph shows that the temperatures in the storage tank tend to increase gradually with increasing time. T_3 has the lowest temperature among other temperatures while T_5 shows the highest temperature.

Fig. 5 shows the average solar intensity and average ambient temperature against the time recorded during the day. The solar intensity curve shows that the flat-plate solar thermal collector absorbs high solar intensity during 12pm to 2pm. The least amount of solar intensity collected is during the start and the end of the day.

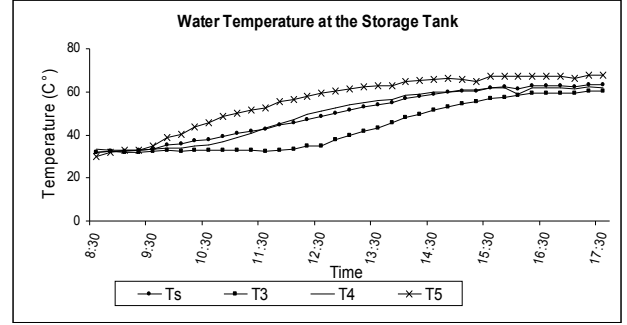


Fig. 4 Water temperatures in the storage tank.

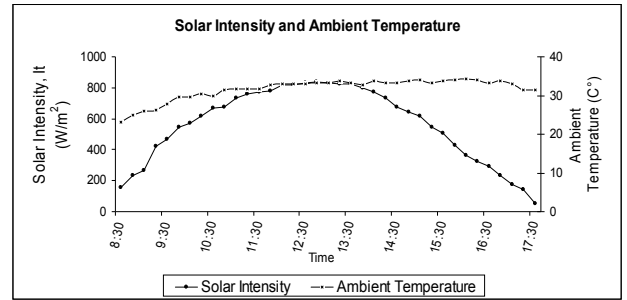


Fig. 5 Solar intensity and ambient temperature.

The $\tau\alpha$ and U_L were used to find the useful energy from the collector Q_u and estimated for the storage tank temperatures in the next period T_s^+ . To find the estimated value T_s^+ , the solar intensity I_t and the ambient temperature T_a of the specified period must be known. In this experiment the specified period is during 2006 - 2008. The solar intensity and the ambient temperature used during these years were taken from the Meteorological Department. The useful energy from the collector Q_u is found from Eq. (10). After the calculation is completed the efficiency function f_c is found from Eq.(9) while the collector efficiency η is found from Eq. (7). These values of η were plotted in Fig. 6 showing the trend of the percentage of the average efficiency during the year. The period with the maximum percentage of efficiency is in October and the least percentage of efficiency is in September due to raining season.

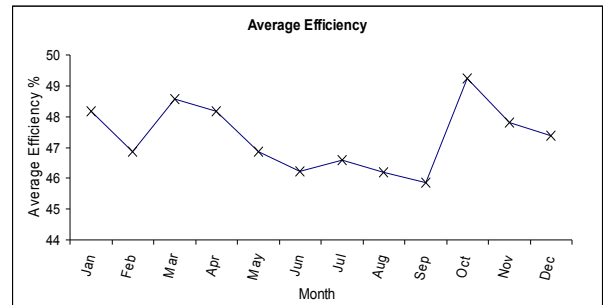


Fig. 6 Average annual efficiency.

Fig. 7 shows the average output temperature of hot water. It shows that the highest temperature of the outlet water T_2 is in March and April while the least temperature of the outlet water is in November.

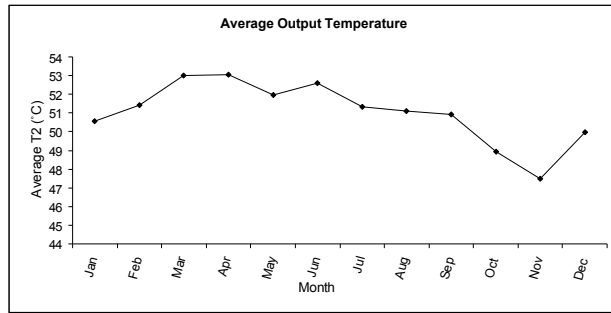


Fig. 7 Average outlet temperature of collector.

5. Economic Analysis of TSSWH

Before TSSWH could be put to work, it should be proved that the TSSWH is worth of investment as compared to the electric water heater (EWH) [5]. In economic analysis, three main costs in this study are investment and installation cost, maintenance cost, and the electricity cost. These costs are then summed to find the total cost of the TSSWH and EWH for a lifetime of 15 years.

Table. 1 Costs for TSSWH and EWH

Cost	TSSWH	EWH
Investment + Installation	40,000 Baht	10,000 Baht
Electricity Cost /per year	0 Baht	4,599 Baht
Total Present Worth of Cost	40,000 Baht	44,980 Baht

The investment and the installation cost of the TSSWH is higher than the EWH as presented in Table 1. The maintenance cost of the TSSWH is 20,000 Baht [6] for 15 years and the EWH are assumed to be zero since the lifetime expectancy of the EWH is approximately more than 15 years. The last cost component is the electricity cost. The TSSWH needs no electricity to operate so electricity cost is not included in the calculation while calculations are done for the electricity cost of EWH. The EWH maximum power is about 3,000 W. Load factor of 0.6 is included in the calculation. With load factor of 0.6 we find the number of units of electricity per day by multiplying the power of EWH (in kW) with the number of continuous hours used per day which is 2 hours per day. It is the same time used with the TSSWH, and 7 days per week. The number of units per day [7] is 3.6 kWhs. From this value, the number of unit per year is found to be 1,314 kWhs. The average electricity price is 3.5 Baht/kWh in Thailand. Eventually the electricity cost of EWH per year is 4,599 Baht.

The total present worth (PW) [8] of costs are then calculated for both TSSWH and EWH for 15 years. The total cost of EWH is found to be higher than the total cost of TSSWH (see Table 1).

6. Conclusion

The objectives of this study are to determine the solar collector efficiency, to predict the storage tank temperatures, and to assess economics of TSSW in Thailand. This paper shows mathematical model of the fat-plate solar thermal collector. The effective absorptance $\tau\alpha$ and the overall heat loss coefficient U_L were found to be 0.9282 and 13.451 W/m²·°C, respectively. These two parameters are used to find the predicted storage tank temperature with the use of three-year weather data from the Thai Meteorological Department. The weather data from 2006 to 2008 were applied to the TSSWH system in this experiment. It shows that the most useful heat gained from the solar intensity occurred during 12pm to 2pm because the maximum solar intensity gained during this period. Results of the daily efficiency calculation ranged from 45-50%, and the average temperature output is found to be 47-53 °C. The economic analysis shows that the PW of costs are 40,000 Baht and 44,980 Baht for TSSWH and EWH, respectively.

7. Acknowledgement

Authors would like to thank Dr. Thananchai Leephakpreeda, Mr. Artite Pattanapongchai and Mr. Prachuab Peerapong for the comments and helps of the experimental setup and Sirindhorn International Institute of Technology Thammasat University for the partial financial support to this study.

8. References

- [1] Runsheng T, Yuqin Y, Wenfeng G. (2011). Comparative studies on thermal performance of water-in-glass evacuated tube solar water heaters with different collector tilt-angles, *Solar Energy*, vol.85, April 2011, pp. 1381-1389.
- [2] M.S. Hossain, R. Saidur, N.A. Rahim, M.R. Islam, J.U. Ahamed, M.M. Rahman. (2011). Review on solar water heater collector and thermal energy performance of circulating pipe, *Renewable and Sustainable Energy Reviews*, vol.15, June 2011, pp. 3801-3812.
- [3] Wikipedia The Free Encyclopedia, (2011) Passive and active systems, URL: <http://en.wikipedia.org>
- [4] Sirindhorn International Institute of Technology Thammasat University, Mechanical Engineering Faculty (2009), Mechanical Engineering Laboratory II Solar Water Heater.
- [5] Solar Direct, Florida, USA (2010), Compare Solar to Conventional Water Heater, URL: <http://www.solardirect.com>
- [6] Digital Focus, Thailand (2009), Price of Solar Water heater, URL: <http://www.thaienergysaving.com>
- [7] Thaigoodview, Thailand (2011), Calculation of Electricity cost, URL: <http://www.thaigoodview.com>
- [8] Wikipedia The Free Encyclopedia, (2011), Time value of money, Present value of future sum, URL: <http://en.wikipedia.org>

Effect of Tube Length on Combustion Characteristics of a Self-aspirating Radiant Tube Burner (SRTB)

Chanon Chuenchit¹ and Sumrerng Jugjai^{2,*}

^{1,2} Combustion and Engine Research Laboratory (CERL), Department of Mechanical Engineering, Faculty of Engineering, King Mongkut's University of Technology Thonburi, Bangkok, Thailand

Abstract

Radiant tube burners (RTBs) are widely used in heating process of industrial production such as batch annealing and heat treating process. Most of the RTBs operate at high capacity that use a lot of fuel and combustion air to yield high gas temperature. Therefore, the air compressor is needed in order to supply the sufficient air for completely combustion. The objective of this research is to study the effect of the tube length on combustion characteristics of a self-aspirating radiant tube burner (SRTB) equipped with an energy recirculation using a porous heat exchanger concept. Three different lengths of U-shaped tube of 10 cm in diameter were selected, i.e. 0.8 m, 1.0 m and 1.25 m, respectively, which are called as tube S, M and L in the study. Because the combustion air is naturally entrained through the venturi burner by the momentum of fuel jet, therefore the air compressor is not used in this method. Temperature distribution along the SRTB and emission (NO_x & CO) were measured. The results show that the temperature distribution of the tube L is higher than the other tubes at the same firing rate (*FR*). The flame can stabilize inside the SRTB for the *FR* in the range of 3.66-8.28 kW. The maximum temperature inside SRTB is 1,481°C. The NO_x and CO emission are less than 336 ppm and 450 ppm, respectively.

Keywords: Radiant tube burner; Self-aspirating; Heat recirculation, Porous heat exchanger.

1. Introduction

Radiant tube burners (RTBs) appear well suited for the industrial processes that require a high and homogeneous temperature distribution within the furnace and the exhaust gas does not contact to the product. Increasing of energy efficiency is the most important topics in order to reduce fuel consumption and pollutant emissions. Preheating of combustible mixture by recycled heat from exhaust gas has been considered as an effective method for fuel conservation. This type of combustion has been called “excess enthalpies” or “super-adiabatic flame temperature” combustion in which the reactants (or the combustion air alone) are preheated using heat “borrowed” from beyond the flame zone, without mixing the two streams [1]. The heat recirculation systems have been extensively applied for increasing the combustion air temperature. Moreover, it expands the combustion stability. The new type of glass melting regenerative furnace at Nippon Furnace Kogyo (NFK) makes it possible to preheat the combustion air, typically above 1,000°C, by recirculating the heat from the hot exhaust gas resulting in fuel saving up to 50% over cold air of the conventional systems, as well as a significant decrease of greenhouse gas CO_2 and NO_x emissions [2].

Heat-recirculating combustion based on the porous medium technology has now been considered as the emerging furnace design methodology for the next generation of high performance combustion systems. Based on the prominent feature of the porous medium in effectively converting energy between flowing gas enthalpy and thermal radiation, the development of a high performance heat exchanger using a porous medium was proposed by Echigo [3]. The basic concept consists of a pair of porous mediums separated by a solid wall as shown in Fig. 1.

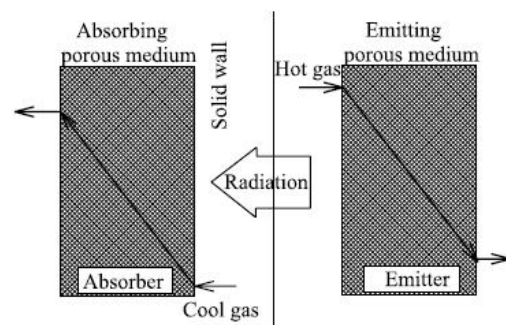


Fig. 1. Energy conversion by porous medium.

Enthalpy of the hot flowing gas (combustion products) is effectively converted to thermal radiation emitted by the high temperature (heating) side porous medium designated as the “emitter” and directed to the solid wall. In thermal equilibrium, the reversed conversion from the incident thermal radiation emitted from the solid wall to the cool gas (or combustion air) enthalpy takes place in the low temperature (cooling) side porous medium designated as the “absorber”, leading to an efficient method in preheating the combustion air. Jugjai and Rungsimuntuchart [4] applied the heat recirculation principle to a self-aspirating burner by employing a porous heat exchanger in combination with the swirling central flow burner (SB). The preheating temperature of the mixture can be increased up to 300°C without flashback. The results showed that the thermal efficiency of the proposed burner is higher than that of a conventional radial flow burner (CB) by about 30 percentage points (50% relative). Scribano et al [5] have been introduced the heat recirculation system by using of exhaust gas recirculation (EGR) techniques in order to enhance the performance of the industrial radiant tube burner. It was shown that the

* Corresponding Author: E-mail: sumrueng.jug@kmutt.ac.th Tel: (662) 4709128, Fax: (662) 4709111

NO_x emissions at the exhaust significant decrease up to 50% with respect to the original burner without the EGR. However, because almost all of the RTBs operate at high capacity that use a lot of fuel and combustion air to receive high temperature (>1,200°C), therefore, the air compressor is needed in order to supply a sufficient air for completely combustion. In recent years, Chuenchit and Jugai [6] applied the heat recirculation principle to the a self-aspirating radiant tube burner (SRTB) by employing a porous heat exchanger integrated with a self-aspirating burner and radiant tube in one unit, leading to an innovation for a radiant tube burner. The results showed that the temperature profile along the tube is higher than that of conventional radiant tube burner without the heat recirculation.

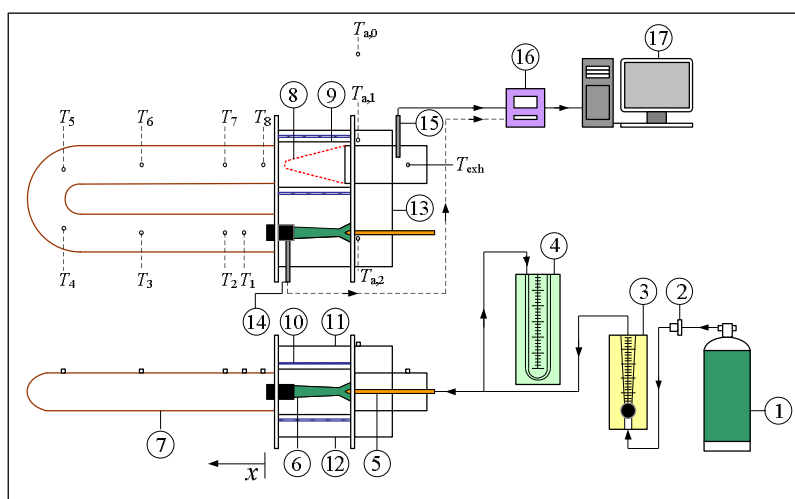
The main objective of this research is to study the effect of tube length on combustion characteristics of the SRTB equipped with the energy recirculation of a porous heat exchanger concept. The tube length significantly affects the pressure drop of radiant tube and thus optimization of the tube length is required. A unique advantage of the self-aspirating burner is that the combustion air is naturally entrained through the venturi burner by the momentum of fuel jet. Therefore, the air compressor is no longer desired by using this burner. Moreover, in order to save energy, the energy recirculation principle using porous medium technology is selected in this work. The SRTB is developed for using in a small-scale drying process and heating process at lower firing rate (lower than 10 kW), which has never been studied before.

2. Experimental Set-up

Fig. 2 shows the schematic diagram of the experimental set-up of the SRTB, whereas Fig.3 shows its photograph. The flow rates of LPG (1) (40% vol. of propane and 60% of butane) with low heating value of about 46 MJ/kg are controlled by a pressure regulator (2). The flow meter (3) is installed downstream of the pressure regulator in order to measure the firing rate (*FR*) of the

fuel. A mercury manometer (4) is used to measure the fuel inlet pressure. Then the fuel emerges from an injector orifice (5). On leaving the injector, the fuel entrains the primary air by a momentum-sharing process between the emerging fuel and the preheated primary air (blue arrow). The fuel/air mixture enters the venturi burner (6) and entrains the secondary air (black arrow) followed by combustion (red arrow) in the U-shaped, stainless radiant tube (7) having 0.1 m inside diameter and 1 mm in thickness. In order to simulate and understand the heat recirculating combustion without thermal loading, the radiant tube was covered by ceramic fiber insulator of 30 mm in thickness to reduce heat lose. The oxygen concentration in the mixture and the exhaust gas compositions are measured by the sampling probe (14) and (15), respectively, and then carried to the portable emission analyzer, Messtechnik Eheim model Visit01L (16). The measuring range is 1-10,000 ppm for CO and 0-4,000 ppm for NO_x with a measuring accuracy of about ± 5 ppm and a resolution of 1 ppm for both CO and NO_x. All measurement of emissions in the experiment is those corrected to 0% excess oxygen and dry basis. All data are recorded by computer (17).

The temperature T at different locations on the axial axis of the SRTB are also recorded as shown in Fig. 2. There are 9, 10, and 8 locations of thermocouples from the venturi burner outlet ($x=0$) to the radiant tube exit for three different tube length as specified by S, M and L, respectively. $T_1 - T_3$ are measured by B-type and the other locations are measured by N-type thermocouples. End of the U-tube is connected to the porous heat exchanger (13), the structure of which is composed of the stainless steel porous emitter (8) having a conical shape with several holes of 2 mm in diameter drilled on its surface for converting a part of gas enthalpy to thermal radiation to the separating wall (9) and then reradiated to the porous absorber (10) for preheating the primary air [3].



- | | | |
|--|---------------------|--------------------------------|
| 1. Fuel (Liquefied Petroleum Gas, LPG) | 7. Radiant tube | 13. Porous heat exchanger |
| 2. Pressure regulator | 8. Porous emitter | 14. Mixture sampling probe |
| 3. Flow meter | 9. Separating wall | 15. Exhaust gas sampling probe |
| 4. Mercury manometer | 10. Porous absorber | 16. Gas analyzer |
| 5. Injector orifice | 11. Upper cover | 17. Computer |
| 6. Venturi burner | 12. Lower cover | |

Fig. 2. Schematic diagram of the SRTB.

The porous absorber (10) was made of a layer of stainless steel wire mesh with 34 meshes per inch. It is cylindrical in shape with 17.7 cm in diameter and 24.7 cm in length. The energy that absorbs at the porous absorber is transferred to the in-coming air (black arrow). $T_{a,0}$, $T_{a,1}$, $T_{a,2}$ and T_{exh} are N-type thermocouple to measure the ambient temperature, air temperature at the porous absorber outlet, the preheated primary air temperature, and the exhaust gas temperature, respectively.



Fig. 3. Photograph of the SRTB.

The upper cover (11) and the lower cover (12) are made of steel sheath of 1 mm in thickness, which protect the heat loss from the heat recirculation section. The porous heat exchanger (13) was constructed such that the ambient air is induced from below to the inlet of the venturi burner (in case of with heat recirculation). In case of without heat recirculation, the porous emitter (8) and the porous absorber (10) are removed.

3. Experimental Results

3.1 Temperature distributions

Fig. 4 shows effect of the tube length on temperature distributions of the SRTB with porous heat exchanger at typical $FR = 7.52$ kW. Among the three tube lengths, tube L gives the highest combustion

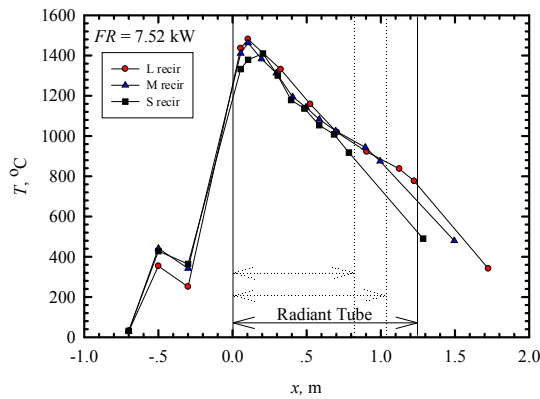


Fig. 4. Temperature distribution.

temperature despite its lowest preheated temperature. This may be attributed to a near stoichiometric combustion for the tube L as will be shown later. The longer the tube length, the more heat loss to the environment, leading to a low exit temperature of the exhaust gas with a decrease in the preheat temperature for tube L. The temperature drop of the last two temperatures at the exit as shown in Fig. 4 for each tube length means part of an enthalpy of the exhaust gas is re-circulated to the primary air by the porous heat exchanger (13), resulting in an increase in the inlet temperature $T_{a,2}$ of the venturi burner.

3.2 Total equivalence ratio ϕ

The total equivalence ratio is defined as the ratio of the fuel-to-oxidizer ratio to the stoichiometric fuel-to-oxidizer ratio and can be calculated from Eq. (1)

$$\phi = \frac{21 - \%O_2}{21} \quad (1)$$

where $\%O_2$ is the oxygen concentration in the exhaust gas at the exit. The total equivalence ratio ϕ is used to indicate quantitatively whether a fuel-oxidizer mixture in the combustion zone of the SRTB is rich, lean, or stoichiometric with the secondary air is taken into consideration.

Fig. 5 shows effect of FR on ϕ at the three tube lengths and with or without heat recirculation by the porous heat exchanger. Increasing FR causes the total equivalence ratio increased in every case, because the higher temperature causes thermal expansion of mixture and increase in its viscosity [7]. In case of with heat recirculation (solid line), the total equivalence ratio for FR of tube L varying from 3.66 to 7.52 kW is near stoichiometry ($\phi = 1$) as compared to the other tubes. This is a desirable condition for the design of the SRTB where a near stoichiometric combustion is achieved whilst an acceptable preheating effect is maintained. According to this reason, the combustion temperature of tube L is the highest despite its relatively low preheat temperature (Fig. 4). In case of without heat

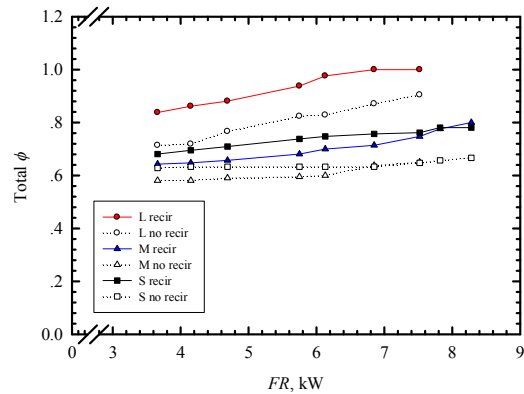


Fig. 5. Total equivalence ratio.

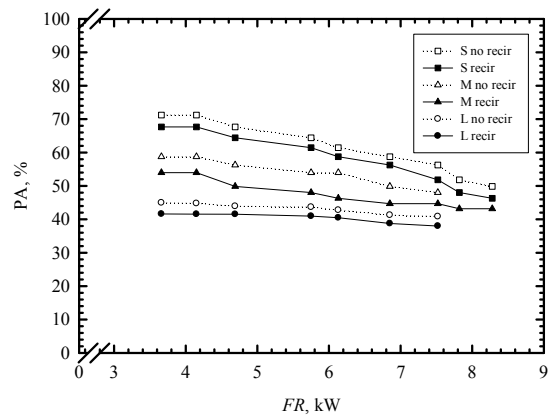


Fig. 6. Primary aeration.

3.3 Primary aeration PA

The primary aeration PA is used to indicate quantitatively whether a fuel-oxidizer mixture in the venturi burner of the SRTB is rich, lean, or stoichiometric prior to combustion without taking the secondary air into consideration. It is defined as the ratio of the oxidizer-to-fuel ratio to the stoichiometric oxidizer-to-fuel ratio, which could be expressed in the form

$$PA = \frac{\%O_2}{(A/F)_{\text{stoi}} \times (21 - \%O_2)} \times 100 \quad (2)$$

where $\%O_2$ is the oxygen concentration of the mixture in the venturi burner, $(A/F)_{\text{stoi}}$ is the air-fuel ratio of LPG at stoichiometric condition.

Fig. 6 shows variation of PA of SRTB in case of with heat recirculation (solid line) and without heat recirculation (dash line). Within the range of FR studied, PA in all cases is almost linearly decreased with increasing FR , due to an increasing of air viscosity [7]. Therefore, the primary air is difficult to be entrained into the venturi burner. In case of with heat recirculation, therefore, all PA values are even lower than that of without heat recirculation, because the preheating effect causes expansion of the mixture and an increase in its viscosity [7]. Among the three tubes, tube S gives the highest PA because of good entrainment of the secondary air into the relatively short radiant tube with relatively low pressure drop. On the other hand, tube L yields the lowest PA because of relatively high viscous flow caused by relatively high combustion temperature with a relatively high pressure drop of the relatively long tube length.

3.4 Emission characteristics

Fig. 7 shows the effect of FR on CO emission for the three tube lengths and for the cases with and without heat recirculation. In any cases and tube lengths, increasing FR increases the CO emission due probably to a decrease in PA that results in incomplete combustion (Fig. 6). It is clear, however, that the CO emissions for the case with heat recirculation are higher than that without heat recirculation, because of relatively low PA for the cases with heat recirculation. At any particular FR , the CO emission for the case without heat recirculation shows insensitive to the tube length,

whereas an opposite result occurs for the case with heat recirculation. Among the three tubes, it is interesting to note that for the case with

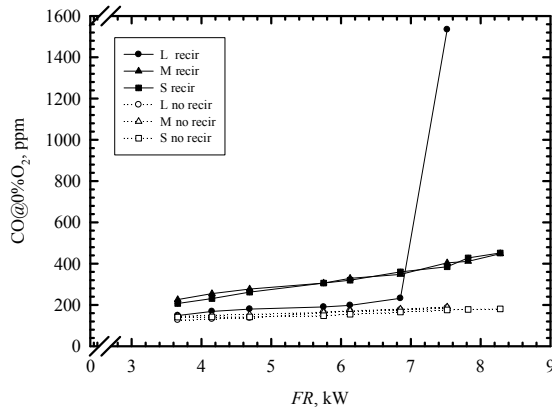


Fig. 7. CO emission.

heat recirculation and FR of lower than 7.5kW, tube L yields the lowest CO emission despite its relatively low PA , because of a long residual time with relatively high temperature for complete combustion. If FR is larger than 7.5kW, CO abruptly increases. This may be attributed to a significant decrease in PA as explained before.

Fig. 8 shows the corresponding NO_x emission of the SRTB. In any cases and tube lengths, increasing FR yields an increase in NO_x emission due probably an increase in the combustion temperature and the reduction in the combustion air. It is considered that, NO_x emission in the present SRTB may arise from the combination of thermal NO_x and prompt NO_x , which are expected to be a major contribution of the measured value according to Sullivan and Kendall [8]. They reported that the thermal NO_x is sensitive to the firing rate whereas the prompt NO_x is much more sensitive to the theoretical air percent. This statement is justifiable for the present SRTB where the firing rate (FR) and the theoretical air percent (or total equivalence ratio ϕ) are dependent on each other and are the dominating parameters controlling the combustion characteristics.

At relatively low FR , relatively low NO_x emission is observed due probably to relatively low thermal NO_x and prompt NO_x caused by relatively low combustion temperature at relatively lean mixture. On the other hand, a relatively high NO_x emission appears at a relatively high FR , due probably to relatively high thermal NO_x and prompt NO_x caused by relatively high combustion temperature at relative rich mixture.

It is also observed that NO_x emission for the case with heat recirculation is higher than that without the heat recirculation particularly at the relatively high FR , the same reason as stated above. Among the three tubes, it is interesting to note that, for the case with the heat recirculation and FR of lower than 7.5 kW, tube L yields the lowest NO_x emission, the value of which is comparable to that of the cases without the heat recirculation. This may be attributed to the significant reduction in the thermal NO_x caused by less N_2 available at a relatively rich mixture for a relatively large tube length despite its relatively high combustion temperature.

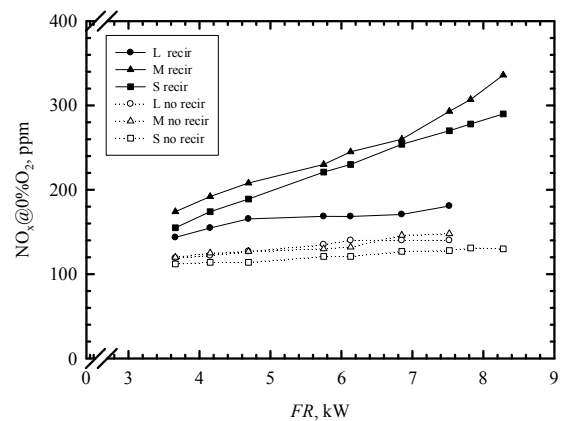


Fig. 8. NO_x emission

3.5 Carnot efficiency

Carnot efficiency is the highest efficiency a heat engine operating between the two thermal energy reservoirs at temperatures T_L and T_H can have, which could be expressed in the form

$$\eta_{\text{th}} = 1 - \frac{T_L}{T_H} \quad (3)$$

where T_L and T_H , respectively, are absolute temperatures of exhaust gas temperature (low temperature reservoir) and log-mean temperature of gas temperature at the positions that transfer heat from radiant tube (high temperature reservoir) to the surrounding. The log-mean temperature could be expressed in the form

$$T_H = \frac{T_{\max} - T_{\text{ex}}}{\ln\left(\frac{T_{\max}}{T_{\text{ex}}}\right)} \quad (4)$$

where $T_{\max}$ is the maximum gas temperature inside the SRTB and T_{ex} is the exhaust gas temperature (for the case without heat recirculation) or the gas temperature at the radiant tube exit before the gas enters the porous emitter (for the case with heat recirculation).

Fig. 9 shows the effect of FR on the Carnot efficiency. The result shows that Carnot efficiency in case of with heat recirculation is higher than that without heat recirculation because of heat re-circulating combustion, leading to an increase in the maximum gas temperature. In addition, the exhaust gas temperature decreased due to part of heat from exhaust gas is used to preheat the primary air. Among the three tubes, tube L gives the highest value of Carnot efficiency because it has the highest T_H and the lowest T_L .

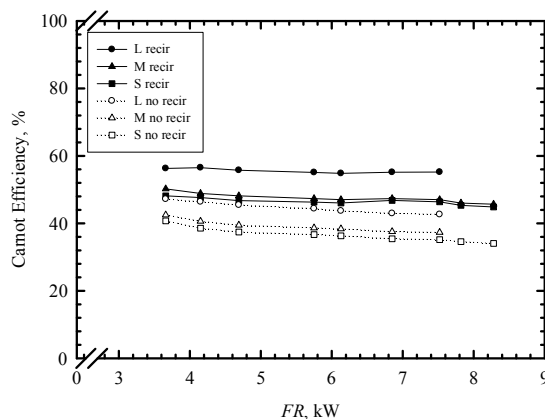


Fig. 9. Carnot efficiency.

4. Conclusions

An experimental study of the effect of the tube length on combustion characteristics of a self-aspirating radiant tube burner with and without heat recirculation has been carried out. Burner performance, combustion characteristic and effect of tube length of SRTB were clarified through measured thermal structure in terms of temperature distribution along the centerline of the SRTB and the emissions at the exit pipeline. The following conclusions can be drawn from the experimental results;

1. An innovative SRTB equipped with the energy recirculation of a porous heat exchanger is successfully developed. It can operate with FR in range of 3.66 to 8.28 kW and without flame flashback into the venturi burner.
2. According to the results of temperature distribution, the NO_x and CO emission and the Carnot efficiency, it can be concluded that the tube L is the most suitable length to apply and use in practical applications.
3. The SRTB with the energy recirculation in case of having external thermal load is the topic of the study in the near future.

5. References

- [1] Weinberg, F.J. (1971). Combustion temperatures: the future, *Nature*, 233, pp. 239–241.
- [2] Katsuki, M. and Hasegawa, T. (1998). The science and technology of combustion in highly preheated air, in: *27th Symposium (International) on Combustion/ The Combustion Institute*, pp. 3135–3146.
- [3] Echigo, R. (1982). Effective energy conversion method between enthalpy and thermal radiation and application to industrial furnace, *Proceedings of the 7th Heat Transfer Conference*, Munich, 6, pp. 361–366.
- [4] Jugjai, S. and Rungsimuntuchart, N. (2002). High efficiency heat-recirculating domestic gas burners, *Experimental Thermal and Fluid Science*, 26(5), pp. 581–592.
- [5] Scribano, G., Solero, G., and Coghe, A. (2006). Pollutant emissions reduction and performance optimization of an industrial radiant tube burner, *Experimental Thermal and Fluid Science*, 30, pp. 605–612.
- [6] Chuenchit, C. and Jugjai, S. (2011). Self - aspirating radiant tube burner, paper present in the *4th International Conference on Sustainable Energy and Environment (SEE 2011)*, Bangkok, Thailand.
- [7] Namkhat, A. and Jugjai, S. (2010). Primary air entrainment characteristics for a self-aspirating burner: model and experiments, *Energy*, 35(4), pp. 1701–1708.
- [8] Sullivan, J.D. and Kendall, R.M. (1992). Thermal performance and NO_x emissions from porous surface radiant burners, *International Gas Research conference*, pp.1626–1634

Novel Boundary Conditions for Turbulent Flows Enclosed by Porous Media

Arpiruk Hokpunna^{1*}, Barbara Wohlmuth², Somchai Wongwises¹

¹Department of Mechanical Engineering, Faculty of Engineering, King Mongkut's University of Technology, Bangkok, Thailand 10140

²M2 - Zentrum Mathematik, Technische Universität München, Garching, Germany 85748

Abstract

In this paper we present the new boundary conditions for the turbulent flows enclosed by porous medium. Such flows play a crucial role in many areas such as filters, oil wells, heat exchangers, catalytic reactors, ground water pollution, scouring and deposition of pollutions at river bed. Since the porous media always consist of multi-scale structures starting from the dimension of the porous medium itself down to pore scale. Resolving all of these structures is too expensive considering the uncertainty of the porous media geometry representation. Under certain conditions, the effects of the porous media to the turbulent flows can be modeled as boundary conditions. However, the approach adopted in [1], [2] and [3] lacks some physical property of the flow at the porous media interface because in those works, either the slip velocity or the interface-normal velocity is assumed zero. The boundary conditions proposed in this work, are validated against the pore scaled simulation in which the whole porous media are resolved. The proposed boundary conditions deliver excellent normalized mean velocity and fluctuations. The numerical simulation using boundary conditions only uses 0.92 Million grid cells and it is computed on a simple workstation compared to 230 Million grid cells of the pore scaled simulation which is computed on a cluster with 512 nodes. The proposed boundary conditions allow accurate predictions of such flows to be accessible by the computing resources available in Thailand.

Keywords: Boundary conditions, Turbulent flows, Direct numerical simulations, Flow over porous interface, Multi-physics

1. Introduction

Transport phenomena at the interface between porous medium and free flow play a crucial role in many areas. Filters, oil wells, heat exchangers, catalytic reactors, ground water pollution, benthic boundary layer are classic applications. In general, one can model the solid phase of a porous medium as an impermeable solid and solve the Navier-Stokes Equations (NSE) in the free flow part and in the void of the porous medium. It is possible to solve this problem with direct numerical simulation and obtain a full dynamic information of the flow. However, this approach is very expensive especially when turbulent flows are involved. Together with uncertainty in the representation of the porous media, the accuracy of the prediction is unlikely to justify the cost of the simulation. Therefore, this problem are usually solved by two alternative strategies: (i) two domains and (ii) one domain approaches. In two domains approach, each domain is governed by its respective equation, for example NSE in the free flow and Darcy's Law in the porous media. In one domain approach, the whole system is considered to be a single continuum and the Volume-Averaged Navier-Stokes equations (VANS) [4] are used to solve in both domains.

On the other hand, if the coupling between these two domains is local and the main interest is the fluid side, the porous media can be removed from the computation and we can model the effects of the interface by a certain set of boundary conditions. This third approach is thus called boundary conditions approach and the interface of the porous medium is modeled as a porous wall. The investigation of fluid flow above porous layer has a long history. The early investigation of the modern days started in the 60s and

most of the studies were concentrated on Stokes' flows (see [5,6 and 7]).

This is partly due to the earlier interests are focused on ground water flows. Only recently, the focus starts to shift to the Navier-Stokes equations such as [8, 9, 10 and 11] among others. In many situations, the flow in this coupling system can become turbulent and the validity of the tools developed for Stokes flows are thus can be questioned.

There are two boundary conditions proposed for permeable wall. The first approach by Jimenez et al. [1]. They proposed that for a porous medium with low porosity, the tangential velocity can be neglected and the interface-normal velocity can be computed from Darcy law. The conditions read

$$u = v = 0 \text{ and} \\ w = -\beta p'$$

where u , v and w are the velocities in the streamwise, spanwise and interface-normal directions, respectively. These condition math with perforated surface such as those aiming at controlling fluid flow on airfoil. The second approach [2] assumes that follows similar concept of Beavers-Joseph conditions [5] in combination with Darcy law. These conditions read

$$\left. \frac{\partial u}{\partial z} \right|_{\text{wall}} = \frac{\alpha}{\sqrt{k}} (u_{\text{wall}} - Q), \\ \left. \frac{\partial v}{\partial z} \right|_{\text{wall}} = (v_{\text{wall}} - Q) \text{ and.} \\ w = 0$$

* * Corresponding Author: E-mail: arpiruk.hok@kmutt.ac.th Tel: +66-2-4709264, Fax: +66-2-4709111,

This approach assumes that there is no mass flux across the interface. These two approaches are on the extreme ends. One end assumes no flow in the normal direction, and the other end assumes now flows in the tangential direction. For typical porous media, fluid can flow in every direction. It is thus necessary to conduct a detail study on the relationship between the flow condition above and at the porous interface. The information from this study can then be used to construct a new type of boundary condition.

This paper aims at investigating the relationship between the flow conditions just above the porous media and the conditions at the interface. To this end, we use direct numerical simulation of a turbulent flow above fully resolved porous media. We then present a novel boundary condition for turbulent flow enclosed by porous medium. These boundary conditions allow prediction of the mean flow and its fluctuation at a much lower cost in comparison to the fully resolved simulation where the flow inside the porous media is simulated. The paper is organized as follows. First, the setting of the direct numerical simulation where the new boundary conditions

2. Setup of fully-resolved simulation

The model porous media used here consists of eleven layers of the representative element in vertical. This number has been chosen desperately to ensure that the flow on the bottom part of the porous media is approximately laminar.

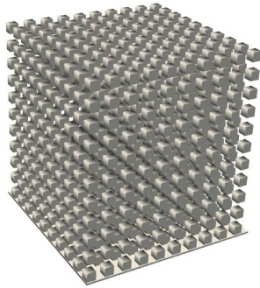
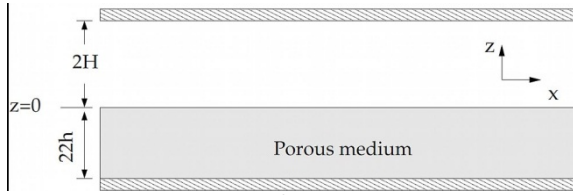


Fig. 1 Setup of the numerical experiment.

In Fig.1 we display a group building blocks of the model enclosing the bottom domain over 10D×10D area on the streamwise-spanwise plane. Note that we do not consider sub-pore-scaled roughness, e.g. irregularities on the surface of the grain. Consequently, all surfaces of these cubes are treated as smooth no-slip walls.

The dimension of the computational domains normalized by the channel half-width (H), together with the number of grid cells are listed in Tab.1. In order to compare the flow under a comparable conditions, we fixed the bulk flow Reynolds number (Re_H). This requires a higher driving pressure gradient for the porous case. The ratio of the driving pressure gradient of each simulation normalized by that of the smooth wall case

(p_x/p_{x0}) is shown in the table. The computational domain for the smooth wall case is the same as the one used in [12]. Smaller domains are used for the porous wall case due to the autocorrelation of the velocities from our previous study (on a larger domain) drop to zero well within the size used here. The number of the cubes representing the porous media grain is 1500 where each cubes is resolved by 1200 grid cell.

A similar configuration was also used in [13], but with a larger pore relative to the channel width. The size of the computational domain in term of the roughness height in these simulations is significant larger than previous studies e.g. [14, 15 and 16].

Numerical grids are listed in Tab.1 along with the numerical grid for turbulent channel flow at $Re = 5600$ which is used as a reference test case to compare the effects of the bottom porous wall. The grids used here are well smaller than the recommended values.

Table 1: Domain size and number of grid cells used in the simulations for porous media case (POR) and the reference smooth wall case (SMT).

Case	P_x/P_{x0}	L_x/H	L_y/H	N_x	N_y	N_z	$N \times 10^6$
Smooth(SMT)	1.00	12.56	4.20	256	160	144	5.9
Porous(PER)	2.20	4.91	2.95	1000	600	410	246.0

3. Numerical Approach

We solve the Navier-Stokes equations for incompressible Newtonian flows:

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j^2} + S_i$$

$$\frac{\partial u_i}{\partial x_i} = 0$$

Navier-Stokes equations (NSE) are integrated within the standard framework of finite volumes method on staggered Cartesian grid. The spatial approximations are second-order accurate using centered interpolations and differentiations. Time integration is performed via a fractional step method with a third-order Runge-Kutta scheme. The pressure is obtained by a projection method at the end of each substep. The Poisson equation is solved by Stone's strongly implicit procedure (SIP). See [17] for a review on these standard methods. Detail information and accuracy of the code can be found in [18].

The pressure cells are positioned carefully such that the surfaces of the cubes coincide with the pressure cell and the region occupied by these cubes will be blocked from the computation. Since we use staggered grids, the momentum cells cannot be made aligned to the cubes surfaces. The effect of these solid surfaces to the momentum are represented by the immersed boundary method (IBM). The basic concept of our immersed boundary method is a functional fitting. A certain approximate function $f(x,y,z)$ is assumed to represent the velocity field and boundary condition locally near the interpolating position. This function is determined by the method of undetermined coefficients.

Once the function is obtained, velocity components at a given point can be extracted. This gives a second-order smooth representation of the curved surface in our Cartesian grid.

In this work we use the cubic spline for the fitting function. Detail information about the immersed

boundary method can be found in [19] The IBM used here have been used successfully in many turbulent flow investigations e.g. [20 and 21] among others.

4. Results for fully-resolved pore scale simulation.

In all cases, the flow is driven by the a constant pressure gradient which is adjusted such that the Reynolds number based on the bulk flow velocity and the channel width $ReH = 2U_b H / \nu$ are approximately 5600. The flows are let to develop and balance themselves without further mass control mechanism. The computations are performed on NEC Nehalem cluster at High performance computing center Stuttgart. The number of processor used for the smooth- and the porous walls are 32 and 512, respectively. The larger case took about a month to complete the simulation. The flow statistics have been sampled for the time period of 500 and $352H/U_b$ in the smooth-, rough- and porous walls. These are corresponding to 40, and 72 through flow. A snapshot from the simulation of porous wall case is shown in Fig.2 where an ejection of low momentum fluid (A) and an inrush of the high energy fluid (B) from and into the porous medium, respectively.

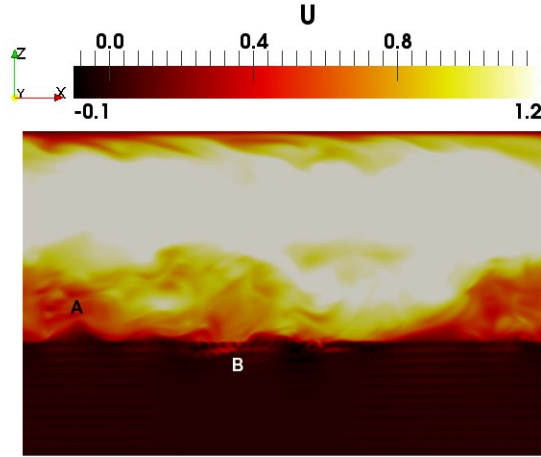


Fig.2: Snapshot of streamwise velocity showing ejection of low momentum fluid (A) and sweeping of high energy fluid (B).

4.1 Mean streamwise velocity

The overview of the mean streamwise velocity profiles sampled and averaged in time, streamwise and spanwise directions is displayed in Fig.3(a). The center of the channel flow with the porous wall is shifted closer to the top wall due to the higher skin friction at the bottom wall which requires a thicker boundary layer thickness (1.42H) to compensate. The boundary layer thickness here is measured from the nominal bottom wall (top of the cubes) to the point where the total shear stress vanishes. The mean streamwise velocity on the top ($z=0$) of the porous walls is $0.145U_b$. In Fig.3(b) we plot the normalized mean streamwise velocity profile without shifting the zero plane. The profile in the smooth wall case agrees well with the universal logarithmic profile. This is not the case for the boundary layer above the porous wall. In a usual boundary layer, the mean flow profile is consisting of viscous sublayer ($z^+ < 5$), buffer layer ($5 < z^+ < 50$) and the outer layer ($z^+ > 50$). The region of the log-law usually starts from $z^+ > 30$ and spans to $z/\delta < 0.3$ before entering the wake region. The inflection point which is usually observed in the buffer region that

connects the viscous sublayer and the log-layer are disappeared on the porous wall. It is still possible to fit the log law using $3.08 \ln(z^+) - 7.34$ for $100 < z^+ < 360$, but the power-law fit the profile on a larger range i.e. $7 < z^+ < 360$.

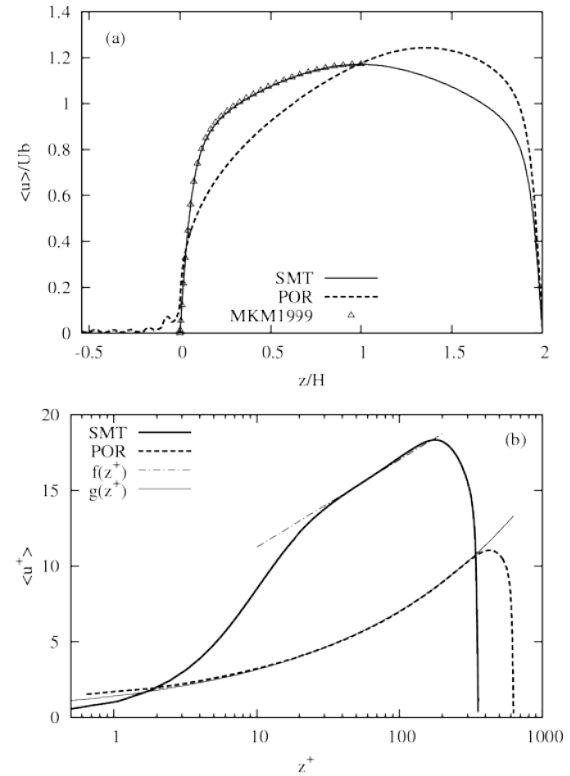


Fig.3: Mean velocity profile normalized by bulk flow velocity (a) and friction velocity at the bottom surface (b). The reference data from [12] is shown as MKM1999. The fitting functions according to the log-law and the power-law are $f(z^+) = 2.5 \log(z^+) + 5.5$ and $g(z^+) = 1.418 (z^+)^{0.348}$, respectively.

4.2 Correlation of the interface condition and the condition at $1 z^+$ above the interface

According to Beavers and Joseph [5], we know that, in laminar flow, the velocity is a continuous function, but the shear stress is not. To investigate the behavior of the volume averaged velocity and gradient in the vicinity of the surface under turbulent conditions, we plot the values averaged on the plane of the representative element volume (REV) and show them in Fig.4. The x-axis is the value sensed at $1 z^+$ above the porous surface and the y-axis is the value at the interface. Note that REV is the smallest building block of the porous media. In this experiment it is the volume $8h^3$ enclosing the cubes. According to the figure, it is clear that the momentum at the interface is a linear function to the momentum at $1 z^+$. The non-vanishing constant in Fig.4(a) is simply a result of the seepage flow within the porous media due to the driving pressure gradient which was applied throughout the domain. In Fig.4(b) the correlation passes the zero-coordinate which implies that, there will be now flow across the interface, if the flow is laminar. It is interesting to note that, the condition here is not exactly zero-gradient. Because, if it was the case, the flow coming into and out of the porous media will have to influence at all to the tangential components (due to mass-conservation). The shear stress, on the other hand,

exhibits quadratic relation. It is important to note that, calculating the shear stress using the extrapolation from the information at the surface gives a broader scatter (larger error). This means the jump condition of Beavers and Joseph [5] is less accurate.

Having obtained the correlation of the conditions between the flow above and at the interface, we can proceed to the simulation where we remove the porous media from the computational domain and replace the effects of complex interaction at the interface by a simple set of boundary conditions. The result of this approach is presented in the next section.

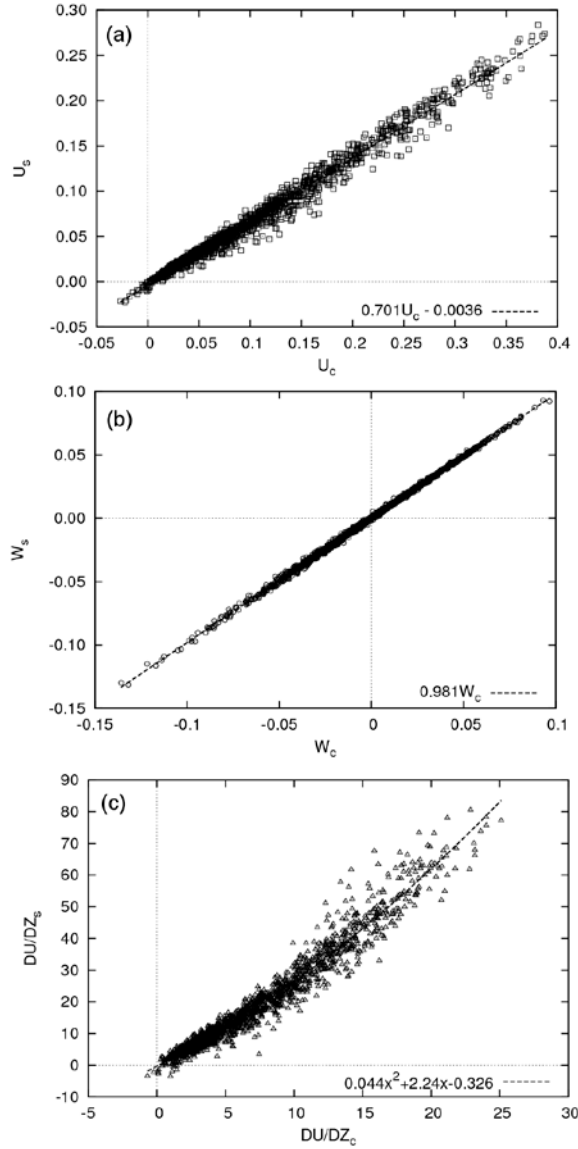


Fig.4 Correlation of the velocity and the streamwise velocity gradient at subsurface and the velocity at $1 z^+$: (a) streamwise velocity, (b) wall-normal velocity and (c) streamwise velocity gradient in z -direction (du/dz). The x variable in (c) stands for du/dz .

5. Results from boundary condition approach

The goal of this study is to find the boundary condition which mimics the behavior of porous media surface. The final goal is that we can conduct DNS of

fluid flow with porous medium at normal DNS grid resolution. For example, 6M grid cell instead of 246M grid cells. To evaluate the potential of this approach, we conduct a DNS of turbulent channel flow, driven by the same gradient as the POR case with the conditions obtained in the previous section. The computational domain in x, y and z is $[6.28, 2.4, 2]$ with 0.9M grid cell. The simulation is conducted on 16 processor for 4 hours.

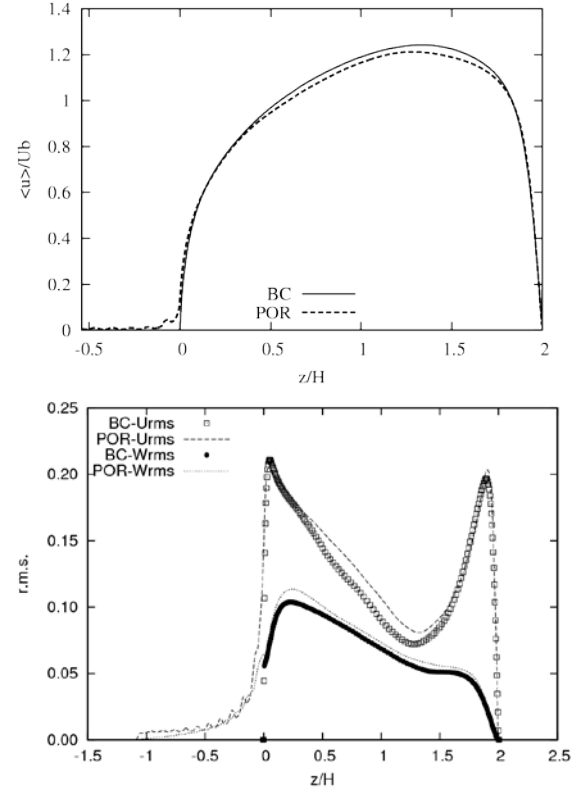


Fig.5 Mean streamwise velocity (top) and r.m.s. of velocity fluctuations (streamwise and wall-normal components) of the boundary conditions approach compared to pore-scale simulation.

The results in Fig.5(a) shows an excellent agreement between the boundary condition approach and the pore-scale simulation. Note that the boundary condition approach over predicts the mass flow by 2.4%. A correction of this mass flow (scaling the plots) will bring the two curves on top of each other. The more interesting results are shown in Fig.5(b). The r.m.s. of the velocity fluctuations between two simulations are very close. This similarity indicates that the flow structures in these two simulations are quite similar.

6. Conclusion and outlook

Despite the huge difference in grid resolution (273X) and the required computing resources, the boundary condition approach is able to predict the first- and the second-order statistics of the flow accurately. This new approach allows the studies to be conducted in Thailand where supercomputing facilities are not available. The next step of the research in this direction is to find the dependency of the constant on the Reynolds number, pressure gradient inside the porous media and the channel.

7. Acknowledgement

The authors A. Hokpunna and B. Wohlmuth would like to thank the German Research Foundation (DFG) for financial support of the project within the Cluster of Excellence in Simulation Technology (EXC 310/1) at the University of Stuttgart.

8. References

- [1] J. Jimenez, M. Uhlmann, A. Pinelli, and G. Kawahara, *Journal of Fluid Mechanics* (2001). Vol.442, pp. 89
- [2] S. Hahn, J. Je, and H. Choi, *Journal of Fluid Mechanics* (2002). vol.450, pp. 259
- [3] C. Wagner and R. Friedrich, *Intl. Jour. of Heat and Fluid Flow* (2002). vol.21(5), pp. 489
- [4] S. Whitaker (1996), *The Method of Volume Averaging*, Springer Verlag.
- [5] G. S. Beavers and D. D. Joseph, Boundary conditions at a naturally permeable wall, *Journal of Fluid Mechanics Digital Archive* (1967). vol.30
- [6] P. Saffman, On the boundary condition at the surface of a porous medium, *Stud. Appl.Math.*(1971). vol.50(93).
- [7] S. Whitaker, Flow in porous media I: A theoretical derivation of Darcy's law, *Transport in Porous Media* (1986). vol.1(3).
- [8] M. Sahraoui and M. Kaviany, Slip and no-slip temperature boundary conditions at the interface of porous, plain media: Convection, *International Journal of Heat and Mass Transfer* (1994). vol.37, pp.1029.
- [9] S. Whitaker, The Forchheimer equation: A theoretical development, *Transport in Porous Media* (1996). Vol.25(27).
- [10] A. Shvidchenko. and G. Pender, Macroturbulent structure of open-channel flow over gravelly beds, *Water Resource Research* (2001). vol.37, pp.709.
- [11] D. Pokrajac, C. Manes, and I. McEwan, Peculiar mean velocity profiles within a porous bed of an open channel, *Physics of Fluids* (2007). vol.19, pp. 098109.
- [12] Moser, Kim, and Mansour, Direct numerical simulation of turbulent channel flow up to $Re=590$, *Physics of Fluids* (1999). vol.11, pp. 943.
- [13] W. P. Breugem, B. J. Boersma, and R. E. Uittenbogaard, The influence of wall permeability on turbulent channel flow, *Journal of Fluid Mechanics* (2006). vol.562, pp.35.
- [14] K. Bhaganagar, J. Kim, and G. Coleman, Effect of Roughness on Wall-Bounded Turbulence Flow, *Turbulence and Combustion* (2004). vol.72, pp. 463
- [15] T. Ikeda and P. A. Durbin, Direct simulations of a rough-wall channel flow, *Journal of Fluid Mechanics* (2007). vol.571, pp.235.
- [16] O. Coceal, A. Dobre, T. G. Thomas, and S. E. BELCHER, Structure of turbulent flow over regular arrays of cubical roughness, *Journal of Fluid Mechanics* (2007). vol.589, pp.375.
- [17] J. Ferziger and M. Peric, *Computational Methods for Fluid Dynamics*, Springer, 1996.
- [18] M. Manhart, F. Tremblay, and R. Friedrich., Direct and large-eddy simulation of flow around a circular cylinder at subcritical Reynolds numbers, PhD thesis, Fachgebiet Strömungsmechanik, Technischen Universität München, 2001.
- [19] N. Peller, A. L. Duc, F. Tremblay, and M. Manhart, High-order stable interpolations for immersed boundary methods, *International Journal for Numerical Methods in Fluids* (2006)., vol.52, pp. 1175
- [20] J. Gradl, H.-C. Schwarzer, F. Schwertfirm, M. Manhart, and W. Peukert, Precipitation of nanoparticles in a T-mixer: Coupling the particle population dynamics with hydrodynamics through direct numerical simulation, *Chemical Engineering and Processing* (2006). vol.45, pp.908.
- [21] H. A. Khaledi, V. D. Narasimhamurthy, and H. I. Andersson, Cellular vortex shedding in the wake of a tapered plate at low Reynolds number, *Physics of Fluids* (2009).vol. 21, pp.013603.

Prediction of Total Equivalence Ratio for a Self-Aspirating Burner

Apinunt Namkhat and Sumrerng Jugjai *

Combustion and Engine Research Laboratory (CERL), Department of Mechanical Engineering, Faculty of Engineering,
King Mongkut's University of Technology Thonburi (KMUTT), Bangkok, Thailand 10140

Abstract

A total air entrainment affects the burner performance in terms of completeness of combustion. In recent years, the prediction of a value of the air entrainment for a self-aspirating burner deals only with the primary air, and is also limited to the without preheat case. This study provides a combined theoretical and experimental investigation into the prediction of the total air entrainment for a self-aspirating burner, which is corresponding to the total equivalence ratio. Both with and without preheat of combustion air cases are carried out the primary air entrainment. The oxygen concentration in the mixture is measured by the oxygen sensor. It reveals the level of primary air within this mixture. Calculation method for predicting the total equivalence ratio was developed by using the momentum and energy conservation principles. It is observed that the levels of both primary and secondary air entrainment are increased with increasing the heat input due to the high momentum jet. The preheated case yields a lower primary and secondary air entrainment because of the preheating effect, the fluid in the mixing tube has more viscosity. The levels of total equivalence ratio are in the range of 0.5 to 6. The rich condition is occurred at lower heat input because of the low total air entrainment. At higher heat input, the high momentum jet entrains an excess air. Therefore, the total equivalence ratio becomes very low under lean conditions. The optimum operation of this burner occurs at heat input equals to 7.5 kW, in which the stoichiometric combustion is achieved. As a result, it may be helpful in predicting the thermal efficiency in the future especially in combustion with preheated air.

Keywords: Self-aspirating burner, Total equivalence ratio, Preheating effect, Air entrainment

1. Introduction

Self-aspirating burners designed to be fueled by liquefied petroleum gas (LPG) are widely used in households for cooking in Thailand. During recent decades, the total of LPG consumption is increasing continuously [1] due to rapid development and industrialization. It is found that, the cooking sector accounts for about 50% of the country's total LPG consumption, as shown in Fig. 1. Because the large number of self-aspirating burners involved in our daily life, even a slight improvement in

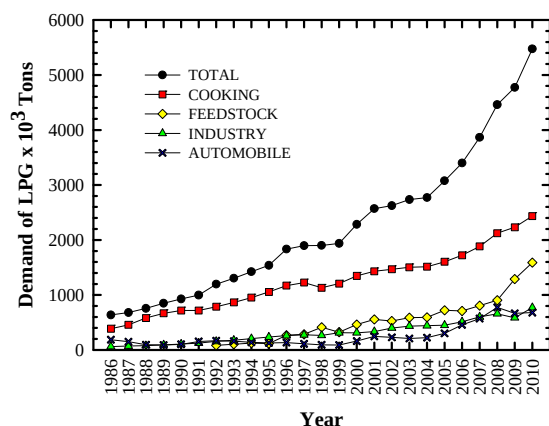


Fig. 1 Consumption of LPG in Thailand [1]

their thermal performance would result in a significant impact on the total energy consumption and the environment.

Much research on self-aspirating burners has been carried out to improve thermal performance and to reduce pollutant emissions by developing the burner port geometry and the combustion system [2-4]. The results showed that the thermal efficiency of the proposed burner is higher than that of a conventional burner. However, through the

preheating effect of the primary air [4], the amount of air entrainment is further reduced because of an increase in the flow resistance [5]. Therefore, incomplete combustion is taken place with a high level of CO emissions. The level of the primary air entrainment is of importance for more complete combustion of the self-aspirating burners. Many studies have been conducted with the self-aspirating burners to predict the level of the primary air entrainment [6-9]. They found that primary air entrainment is a function of fuel gas flow rate, type of fuel gas, injector geometry, mixing tube geometry, and burner port geometry. However, most of the studies mentioned above are limited to the cold test case (without combustion). In recent years, theoretical and experimental studies on primary air entrainment of a self-aspirating burner with hot test case (with combustion) were conducted. Its also performed for both without preheat [10,11] and preheat cases [12] to gain more understanding of primary air entrainment characteristics. A first correlation of primary air entrainment without preheated air is proposed. The results showed that the preheat case of $T_{pre} = 300^{\circ}\text{C}$ gives about a 14 percentage point (33% relative) lower primary aeration value than that of the without preheat case, because the preheating effect causes expansion of the mixture and an increase in its viscosity.

It is clear from the above review that the theoretical equations are not sufficient to use for the prediction of primary air entrainment with preheated air. Moreover, there is no information in the literature about the prediction of total equivalence ratio for a self-aspirating burner. Therefore, our present experimental work was devoted to study the prediction of primary air and secondary air entrainment with preheated air, which corresponds with the total equivalence ratio. It is importance and necessary to assess the behavior and performance of a given burners. This information may be helpful in calculating the adiabatic

* Corresponding Author: E-mail: sumrueng.jug@kmutt.ac.th Tel: (662) 4709128, Fax: (662) 4709111

flame temperature and the thermal efficiency of burners in the future.

2. Methodology

Fig. 2 shows a schematic diagram of the primary aeration measurement using the oxygen sensor which is applied for both with and without preheat conditions. It is composed of a self-aspirating burner with cross-sectional area of injector $A_i = 0.64 \text{ mm}^2$, cross-sectional area of throat $A_t = 254.47 \text{ mm}^2$, and overall cross-sectional area of the burner port $A_p = 245.44 \text{ mm}^2$. Liquefied petroleum gas (LPG) is used as a fuel in the experiment, and the oxygen sensor is used to measure oxygen concentration with an accuracy of about 0.05%. An uncertainty analysis was carried out with the method proposed by Kline and McClintock [13]. Using a 95% confidence level, the maximum and minimum uncertainties in the presented primary aeration were found to be 3.3% relative and 1.6% relative, respectively.

In the without preheat case, the flow rates of LPG 1 are controlled by a pressure regulator ②. The flow meter ③ is installed downstream of a pressure regulator. The volume flow rates are measured in order to calculate the heat input (q) of the fuel gas. A mercury manometer ④ is used to measure the fuel pressure. Then, the fuel gas is injected by the injector orifice ⑤ into the mixing tube 6. Meanwhile, the primary air from the air compressor ⑦ is supplied to the air storage tank ⑧, which is provided to maintain the primary air temperature. The dimensions of the air storage tank are 40 x 40 x 40 cm. The air storage tank walls are covered with 4 cm thickness of ceramic fiber. The pressure within the air storage tank is maintained at an ambient condition, as measured by the water manometer ⑨. The primary air will be entrained into the mixing tube by a momentum-sharing process between the injected gas and the surrounding air. The mixture is distributed uniformly to the multiple port burner 10 with premixed flame.

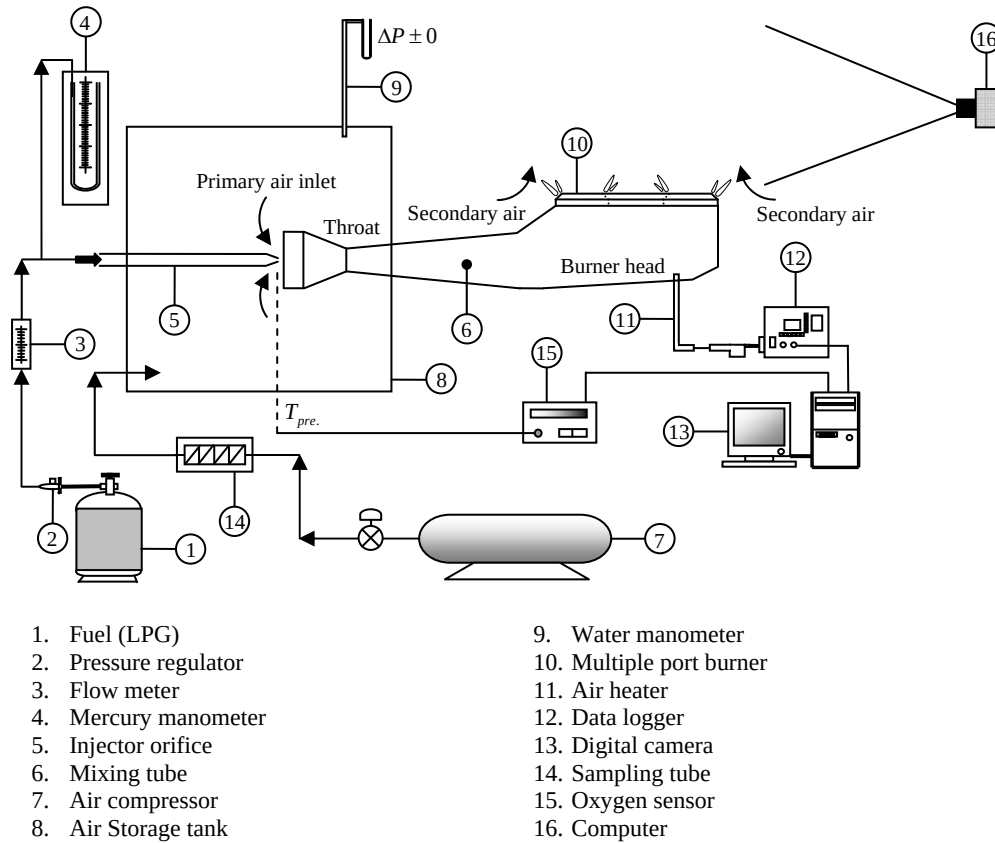


Fig. 2 Schematic diagram of primary aeration experiment

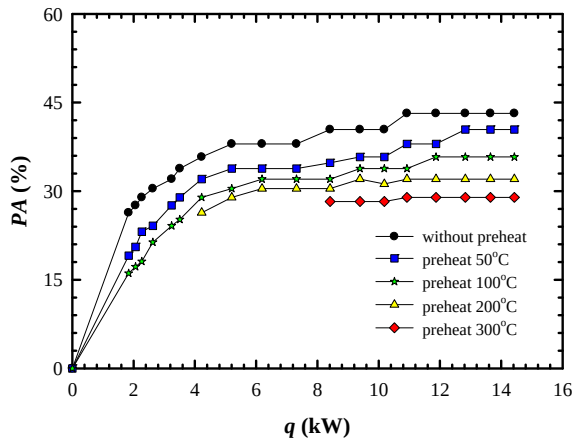


Fig. 3 Typical primary aeration of a self-aspirating burner

Meanwhile, the oxygen concentration in the mixture is measured by using a sampling gas line ⑪, which is located under the burner head, and then carried to the oxygen sensor ⑫ and is displayed by the computer ⑬. The primary aeration can be calculated from the following equation:

$$PA = \frac{\%O_2}{(A/F)_{stoi.} \times (21 - \%O_2)} \times 100 \quad (1)$$

For the preheat case, a similar experiment is performed using the same procedure as described for the without preheat case, but with a preheated primary air. The air heater ⑭ is installed downstream of the air compressor and provided to control preheated primary air temperatures. In this study, four preheated air temperatures ($T_{pre.}$) are selected for the experiments, i.e., 50°C, 100°C, 200°C, and 300°C. These temperatures are performed with N-type thermocouple, which is located at the primary air inlet, and then carried to the data logger ⑮. Meanwhile, the flame images are captured with a digital camera ⑯ to measure the height of the flame.

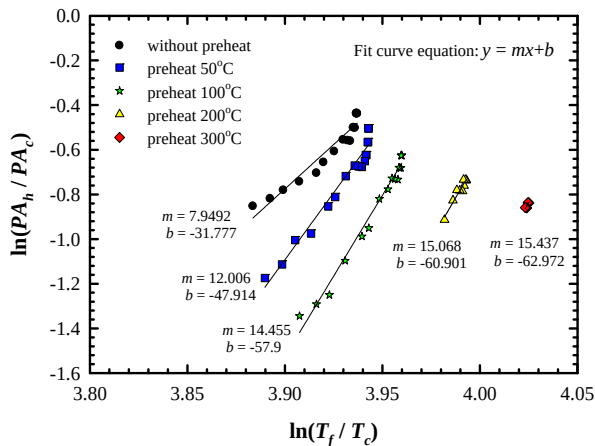


Fig. 4 Linearized plot of dimensionless PA with respect to dimensionless temperature

3. Results and discussion

3.1 Primary aeration and its variation with q

Fig. 3 shows typical primary aeration of a self-aspirating burner for both with and without preheated cases. It is clear that the primary aeration in both cases rapidly increases at the early stage with an increasing heat input (q). After that, the primary aeration is constant and no longer dependent on the heat input, due to limitations of mixing tube and burner port sizes. The primary aeration decreases with an increasing preheated air temperature ($T_{pre.}$).

It is interesting to note that, the preheated air temperature of $T_{pre.} = 300^\circ\text{C}$ gives about a 14

percentage point (33% relative) lower PA value than that of the without preheat case due to the preheating effect causes expansion of the mixture and an increase in its viscosity.

3.2 Correlation for primary aeration (PA)

To the authors' knowledge, earlier studies [6-8] pertaining to models and experiments predicting primary aeration for a self-aspirating burner were conducted only as cold tests (PA_c). Moreover, a preliminary experimental study [11] proposed a first correlation between the hot test and the cold one for primary air entrainment. However, the preheating effect caused by combustion with preheated air was overlooked. In this study, the general models or empirical correlations are revealed which are capable of predicting primary aeration as a function of preheated primary air temperature.

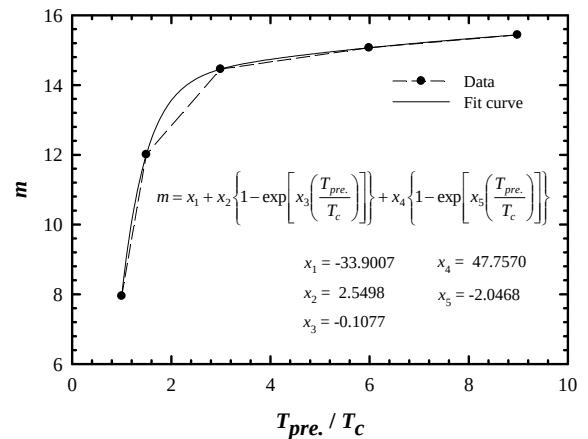


Fig. 5 Variation of the slope (m) with respect to $T_{pre.} / T_c$

The experimental data in the present study were used to develop an empirical expression for the primary aeration PA . The primary aeration for the hot test PA_h can be estimated from the cold test PA_c [11]. But, PA_h must be corrected to account for the effect of the temperature rise of the preheated primary air at the mixing tube inlet. The rising of temperature significantly affects the adiabatic flame temperature. Best-fit empirical correlation for PA_h as a function of PA_c and temperature has been determined and proposed with the formula:

$$PA_h = PA_c \left[\frac{T_f}{T_c} \right]^m e^{\lambda b} \quad (2)$$

where T_f and T_c respectively, are adiabatic flame temperature and ambient temperature.

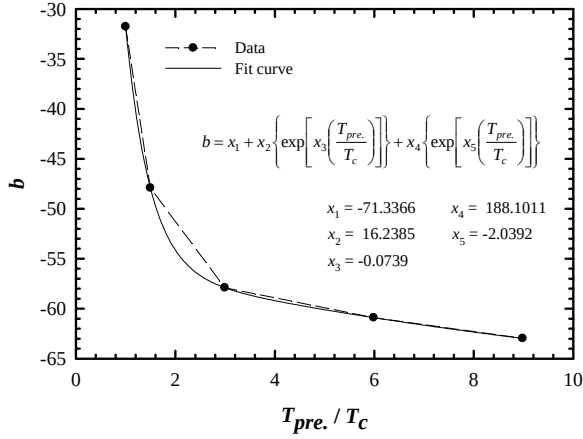


Fig. 6 Variation of the intercept (b) with respect to $T_{pre.} / T_c$

In addition, T_f is the flame temperature based on the primary equivalence ratio of the cold test with the primary air preheated. λ is a stage of completion. $\lambda = 0$ means without combustion, whilst with combustion referring $\lambda = 1$. The variable m and b in equation (2) are slope and intercept, respectively, as shown in Fig. 4. These variables also depend on the preheated primary air temperature and have been determined from Fig. 5 to 6 that can be expressed in the form

$$m = -33.9007 + 2.5498 \left[1 - e^{\left\{ -0.1077 \left(\frac{T_{pre.}}{T_c} \right) \right\}} \right] + 47.7570 \left[1 - e^{\left\{ -2.0468 \left(\frac{T_{pre.}}{T_c} \right) \right\}} \right]$$

$$b = -71.3366 + 16.2385 \left[e^{\left\{ -0.0739 \left(\frac{T_{pre.}}{T_c} \right) \right\}} \right] + 188.1011 \left[e^{\left\{ -2.0392 \left(\frac{T_{pre.}}{T_c} \right) \right\}} \right]$$

where $T_{pre.}$ is the preheated primary air temperature.

This correlation is valid for $30 < T_{pre.} < 300^\circ\text{C}$.

Equation (2) provides the best value in comparison with the experimental results. This is the first correlation obtained from this study. It can be concluded that when the mixing tube dimensions and the type of mixture are specified, the level of primary aeration will depend on the preheated primary air temperature ($T_{pre.}$). Fig. 7

shows a parity plot of PA_h obtained using equation (2) and by experimenting with a surface roughness of the mixing tube of $\mathcal{E} = 0.25$ mm. It was found that the maximum deviation between the correlation and the experiment is within $\pm 10\%$.

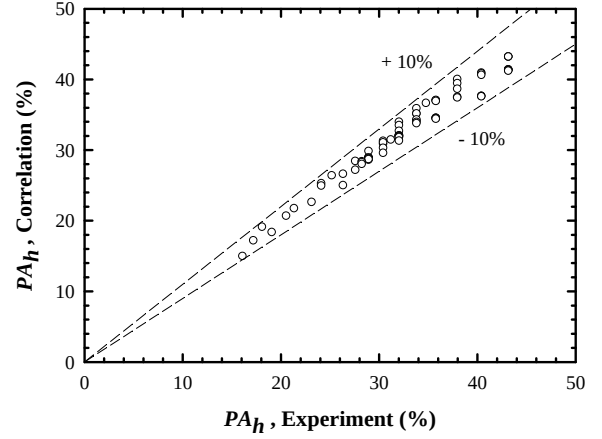


Fig. 7 Parity plot showing comparison between correlation and experimental data

3.3 Correlation for secondary aeration (SA)

The majority of the self-aspirating burners used in domestic LPG appliances are the blue flame. In this burner, the primary air will be entrained into the burner by the flow of a LPG jet through an orifice. The momentum force of this flow cannot normally entrain all the air needed to burn the mixture completely. The rest of the air required, called secondary air, comes from the air surrounding the flame as the mixture burns on the burner ports.

There is no information in the literature about the prediction of secondary air entrainment for a self-aspirating burner. However, previous study has been conducted with the free jets to achieve the desired objectives. Ricou and Spalding [14] presented an experimental technique for measuring the secondary air entrainment in the turbulent free jets. A correlation for secondary air entrainment has been determined and proposed with the formula:

$$\frac{\dot{m}_m + \dot{m}_{sa}}{\dot{m}_m} = 0.32 \left(\frac{\rho_{sa}}{\rho_m} \right)^{0.5} \frac{h_f}{D_p} \quad (3)$$

As shown in equation (3), the mass flow rate of secondary air $\dot{m}_{sa}$ is a function of the mass flow rate of mixture ($\dot{m}_m$), density of secondary air (ρ_{sa}), density of mixture (ρ_m), flame height (h_f), and diameter of burner port (D_p).

Since $\dot{V}_{sa} = \dot{m}_{sa} / \rho_{sa}$, from equation (3), the volume flow rate of secondary air is given by

$$\dot{V}_{sa} = \frac{\dot{m}_m}{\rho_{sa}} \left[\left(0.32 \left(\frac{\rho_{sa}}{\rho_m} \right)^{0.5} \frac{h_f}{D_p} \right) - 1 \right] \quad (4)$$

As a result, the secondary air entrainment (or secondary aeration) is expressed in the form

$$SA = \frac{\dot{V}_{sa}/\dot{V}_g}{(A/F)_{stoi.}} \times 100 \quad (5)$$

where $\dot{V}_g$ and $(A/F)_{stoi.}$ respectively, are the volume flow rate of gas fuel and stoichiometric air-fuel ratio.

Fig. 8 shows typical secondary aeration of a self-aspirating burner obtained using equation (5). It was found that the levels of SA for both with and without preheat cases increase with heat input q , because of a strong momentum effect caused by the high volume flow rate of the mixture as q increases. At $q < 6$ kW, the effect of higher buoyancy force is that there is more entrained secondary air, thus increasing the levels of SA for the preheat case. However, the higher q increases the effect of momentum force. As a result, the preheat case gives lower SA value than that of the without preheat case at $q > 6$ kW, because the preheating effect causes low primary aeration and a decrease in the momentum force of mixture.

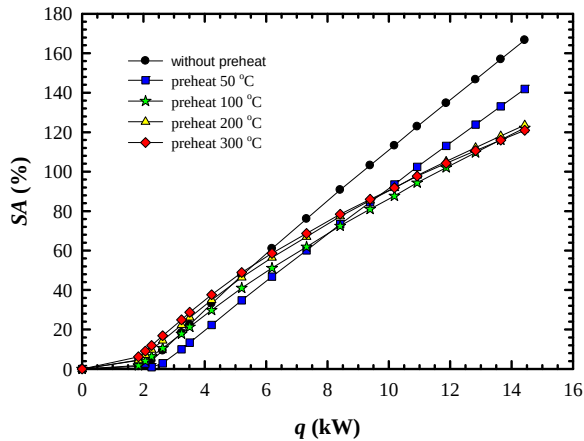


Fig. 8 Typical secondary aeration of a self-aspirating burner

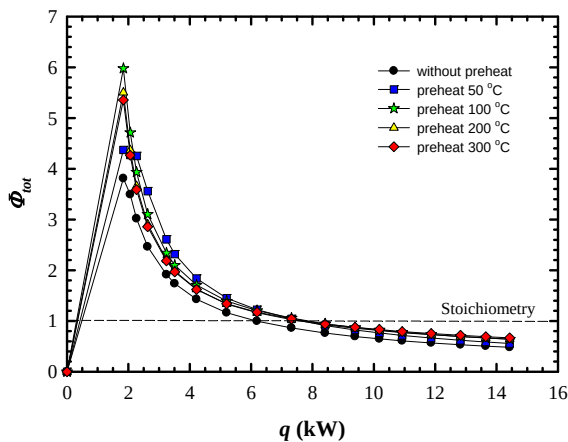


Fig. 9 Typical total equivalence ratio of a self-aspirating burner

3.4 Total equivalence ratio (ϕ_{tot})

The total equivalence ratio is commonly used to indicate quantitatively whether a fuel-oxidizer mixture is rich, lean, or stoichiometric. The total equivalence ratio is defined as

$$\phi_{tot} = \frac{100}{(PA + SA)} \quad (6)$$

In many combustion applications, the total equivalence ratio is the single most important factor in determining a system's performance [15]. Fig. 9 shows typical total equivalence ratio of a self-aspirating burner obtained using equation (6). It was found that the levels of total equivalence ratio are in the range between rich to lean conditions. The rich condition is occurred at lower heat input ($q < 7$ kW) because of the low total air entrainment. At higher heat input, the high momentum jet provides an excess air. Therefore, the total equivalence ratio becomes very low under lean conditions. The optimum operation of this burner occurs at heat input which is indicative of stoichiometric combustion.

4. Conclusions

1. The levels of both PA and SA are increased with increasing q due to the high momentum jet. Moreover, the preheated case gives a lower PA and SA values than that of the without preheat case, because the preheating effect causes expansion of the mixture and an increase in its viscosity.
2. The empirical formula for predicting PA_h in this study is accurate, finding the maximum deviation between the correlation and the experiment to be within $\pm 10\%$.
3. A first correlation for total equivalence ratio of a self-aspirating burner is proposed. The optimum operation of this burner occurs at q which is indicative of stoichiometric combustion.

5. Acknowledgement

This research was sponsored by the Ministry of Education, Royal Thai Government under the project of Thailand National Research Universities (NRU).

6. References

- [1] Energy Policy and Planning Office, Ministry of Energy, Thailand (2011). *Energy database*, URL: <http://www.eppo.go.th/info/index.html>, access on 08/06/2011
- [2] Jugjai, S., Tia, S. and Trewetaskorn, W. (2001). Thermal efficiency improvement of an LPG gas cooker by a swirling central flame, *International Journal of Energy Research*, vol. 25(8), May 2001, pp. 657 – 674.
- [3] Hou, S.S., Lee, C.Y. and Lin, T.H. (2007). Efficiency and emissions of a new domestic gas burner with a swirling flame, *Energy Conversion and Management*, vol. 48(5), January 2007, pp. 1401 – 1410.
- [4] Jugjai, S. and Rungsimuntuchart, N. (2002). High efficiency heat-recirculating domestic gas burners, *Experimental Thermal and Fluid Science*, vol. 26(5), April 2002, pp. 581 – 592.

- [5] Maughan, J.R., Cahoe, J.R. and Ghassemzadeh, R. (1992), *Autoregulation of primary aeration for atmospheric burners*, US Patent Number 5104311, USA, URL: <http://www.patentstorm.us/patents/5104311.html>, access on 02/11/2007
- [6] Pritchard, R., Guy, J.J. and Connor, N.E. (1977). Handbook of industrial gas utilization: engineering principles and practice, Van Nostrand Reinhold Co., New York.
- [7] Singh, G., Sundararajan, T. and Shet, U.S.P. (1999). Entrainment and mixing studies for a variable density confined jet, *Numerical Heat Transfer, Part A: Applications*, vol. 35(2), September 1998, pp. 205 – 223.
- [8] Singh, G., Sundararajan, T. and Bhaskaran, K.A. (2003). Mixing and entrainment characteristics of circular and noncircular confined jets, *Journal of Fluids Engineering*, vol. 125(5), September 2003, pp. 835 – 842.
- [9] Namkhat, A. and Jugjai, S. (2007). Primary air entrainment characteristics of a self-aspirating burner, paper presented in the 21st Conference of Mechanical Engineering Network of Thailand, Cholburi, Thailand.
- [10] Namkhat, A. and Jugjai, S. (2009). Fundamental Study on Primary Air Entrainment of a Self-aspirating Burner: Comparison on Hot Test and Cold Test Case, paper presented in the World Renewable Energy Congress 2009-Asia, The 3rd International Conference on “Sustainable Energy and Environment (SEE 2009)”, Bangkok, Thailand.
- [11] Namkhat, A. and Jugjai, S. (2010). Primary air entrainment characteristics for a self-aspirating burner: Model and experiments, *Energy*, vol. 35(4), April 2010, pp. 1701 – 1708.
- [12] Namkhat, A. and Jugjai, S. (2010). The effect of primary air preheat on the primary aeration of a self-aspirating burner, paper presented in the 1st TSME International Conference on Mechanical Engineering, Ubon Ratchathani, Thailand.
- [13] Kline, S.J. and McClintock, F.A. (1953). Describing uncertainties in single-sample experiments, *Mechanical Engineering*, vol. 75, January 1953, pp. 3 – 8.
- [14] Ricou, F.P. and Spalding, D.B. (1961). Measurement of entrainment by axisymmetrical turbulent jets, *J. Fluid Mechanics*, vol. 11, 1961, pp. 21 – 32.
- [15] Turns, Stephen R. (2000). *An Introduction to Combustion: Concepts and Applications*, 2nd edition, ISBN: 0-07-230096-5, McGraw-Hill, Singapore.

The Effect of Baffles on Fluid Sloshing inside the Moving Rectangular Tank

Krit Threepopnartkul and Chakrit Suvanjumrat *

Department of Mechanical Engineering, Faculty of Engineering, Mahidol University, Nakorn Pathom, Thailand 73170

Abstract

Sloshing of fluid inside a container such as tanker trucks is the important phenomenon that develops pressure on tank wall which can cause structural damage or loss of the maneuvering stability. Decreasing the severe sloshing, the baffles are required to be included into the tank at appropriate locations. The FVM (Finite Volume Method) is the favorite method for analyzing the fluid sloshing. The effect of baffles on the fluid sloshing can be investigated using the computational models. This research improved C++ code of the fluid sloshing inside the movable tanks with and without baffles. These codes were implemented in the Open Source software that was the OpenFOAM. The computational results were validated with the experiment results which had a good correlation. The different results between simulation and experiment were digitized using the image processing techniques and had the average error less than 3.73%. This error can be proved that the computational method can be used to evaluate the effects of the baffles to control the sloshing of fluid inside the tanker trucks..

Keywords: Sloshing, Tank, FVM, Baffles, OpenFOAM

1. Introduction

Because of the liquid sloshing, partially filled with liquid inside the tanker trucks can perform many troubles during transporting whether it is the structural damage or the maneuvering stability. The amplitude of liquid sloshing depended on the nature amplitude and frequency of the tank motion, liquid-fill depth, liquid properties and tank geometry [1]. Many researches attempted to describe the liquid sloshing phenomenon using computational methods there was the Computational Fluid Dynamics (CFD) methods [2-4]. The CFD method might be comprised of Finite Element Methods (FEM), Finite Difference Method (FDM) or Finite Volume Methods (FVM). Whole methods were compared in [5]. The FVM was extensively used to describe the liquid sloshing. The partially liquid sloshing inside the moving tank was complicated phenomenon to write the governing equation because of there was the non-conservative and multi-phase flow. Previous research, [6], we validated the FVM of the half liquid-filled sloshing inside a rectangular tank by implemented using C++ code in the Open Source Field Operation and Manipulation (OpenFOAM). For this research we studied more complicated FVM that was the addition of a baffle inside the tank. The effect of a baffle would be studied and the CFD results would be used to be the patterns for reducing the severe sloshing of liquid inside tanks.

2. Mathematical models

The fluid flow was governed by Navier-Stokes equation [7]. For the multiphase flow, the aggregated fluid treated both phases of fluid as one aggregated fluid with varying properties and used one continuity and momentum equation for the mixture of both phases, whereas segregated fluid models used a continuity and momentum equation for each phase.

2.1 Continuity and Momentum Equations

The mass conservation was applied on an arbitrary part, Ω , of the flow domain with the boundary, S , and

outward directed normal vector $\vec{u}$. The mass conservative equation was given by

$$\int_{\Omega} \frac{\partial \rho}{\partial t} d\Omega + \oint_S (\rho \vec{u}) \cdot \vec{n} dS = 0 \quad (1)$$

where $\vec{u}$ was velocity and ρ was density.

By applied the Gauss' divergence theorem, the integrate over $S = d\Omega$ could be written as the integral over the volume, Ω . Therefore, mass conservation could also be written in the conservative partial differential form as

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0 \quad (2)$$

or in the non-conservative notation form as

$$\frac{D\rho}{Dt} + \rho \nabla \cdot \vec{u} = 0 \quad (3)$$

When the velocity field was not divergence free ($\nabla \cdot \vec{u} \neq 0$) this would result in a varying density for compressible flow. For incompressible flow the velocity field was always divergence free ($\nabla \cdot \vec{u} = 0$). Similar to the mass conservation equation, the momentum equation was applied to a part Ω of the flow domain with boundary S . It was given by

$$\int_{\Omega} \frac{\partial (\rho \vec{u})}{\partial t} d\Omega + \oint_S \rho \vec{u} (\vec{u} \cdot \vec{n}) dS + \oint_S p \vec{n} dS - \oint_S [\mu (\nabla \vec{u} + \nabla \vec{u}^T) - \frac{2}{3} \mu \nabla \cdot \vec{u}] \vec{n} dS - \int_{\Omega} \rho \vec{F} d\Omega = 0 \quad (4)$$

where p was pressure, μ was dynamic viscosity and $\vec{F}$ was external force, respectively. The terms in momentum equation were characterized as the time derivative, the convective term, the pressure gradient, the viscous term and the body force term. By applying Gauss' theorem, also the momentum equation could be written in the conservative divergence form as

$$\frac{\partial (\rho \vec{u})}{\partial t} + \nabla \cdot (\rho \vec{u} \vec{u}) + \nabla p - \nabla \cdot [\mu (\nabla \vec{u} + \nabla \vec{u}^T) - \frac{2}{3} \mu \nabla \cdot \vec{u}] - \rho \vec{F} = 0 \quad (5)$$

* Corresponding Author: chakrit.suv@mahidol.ac.th, Telephone 662 889 2138 ext 6416, Fax. 662 889 2138 ext. 6429

The non-conservative form of the momentum equation was obtained by substituting the continuity, Eq. (2), into the momentum, Eq. (5), equation.

$$\rho \frac{\partial \bar{u}}{\partial t} + \bar{u} \frac{\partial \rho}{\partial t} + \rho(\bar{u} \cdot \nabla) \bar{u} + \bar{u} \nabla \cdot (\rho \bar{u}) + \nabla p - \nabla \cdot [\mu(\nabla \bar{u} + \nabla \bar{u}^T)] - \frac{2}{3} \mu \nabla \cdot \bar{u} - \rho \bar{F} = 0 \quad (6)$$

After dividing the remaining terms in the momentum Eq. (6) by the density, the continuity and non-conservative momentum equation were given by

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \bar{u}) = 0 \quad (7)$$

$$\frac{\partial \bar{u}}{\partial t} + (\bar{u} \cdot \nabla) \bar{u} + \frac{1}{\rho} \nabla p - \frac{1}{\rho} \nabla \cdot [\mu(\nabla \bar{u} + \nabla \bar{u}^T)] - \frac{2}{3} \mu \nabla \cdot \bar{u} - \bar{F} = 0 \quad (8)$$

2.2. Free surface equation

The volume of fluid (VOF) method, [8], was solved for one momentum equation and one continuity equation. These equations were the same for two phases. The physical properties of one fluid were calculated as weighted averages base on the volume fraction of the two fluids in one cell. The VOF in a cell was computed as

$$F_{vol} = \gamma V_{cell} \quad (9)$$

where V_{cell} was the volume of a computational cell and γ was the fluid fraction in a cell. The values of γ in a cell should be ranged between 0 and 1. If the cell was completely filled with fluid then $\gamma=1$ and if it was filled with the void phase then its value should be 0. At the interface, the value of γ was between 0 and 1. The scalar function γ could be computed from a separate transport equation that taken the form as

$$\frac{D\gamma}{Dt} = \frac{\partial \gamma}{\partial t} + \nabla \cdot (\gamma \bar{u}) = 0 \quad (10)$$

The necessary compression of the surface was achieved by introducing an extra artificial compression term into the VOF equation as follow:

$$\frac{D\gamma}{Dt} = \frac{\partial \gamma}{\partial t} + \nabla \cdot (\gamma \bar{u}) + \nabla \cdot (\gamma(1-\gamma) \bar{u}_\gamma) = 0 \quad (11)$$

where $\bar{u}_\gamma$ was a velocity field suitable to compress the interface. This artificial term was active only in the interface region due to the term $\gamma(1-\gamma)$.

The density, ρ , at any point in the domain was calculated as a weighted average of the volume fraction of the two fluids, γ , as

$$\rho = \gamma \rho_1 + (1-\gamma) \rho_2 \quad (12)$$

The surface tension force, F_s , which takes place only at the free surface, was computed as

$$F_s = \sigma K(x) \bar{n} \quad (13)$$

where $\bar{n}$ was a unit vector normal to the interface that could be calculated from

$$\bar{n} = \frac{\nabla \gamma}{|\nabla \gamma|} \quad (14)$$

and $K(x)$ was the curvature of the interface that could be calculated from

$$K(x) = \nabla \cdot \bar{n} \quad (15)$$

2.3 The motion equations

The movement of the control volume could be estimated by solving the external force,

$$\bar{F} = \frac{D(m\bar{u})}{Dt} \quad (16)$$

where m was the mass of the control volume

The velocity in every equation was the relative velocity which referenced to the moving coordinate and relative to the fix coordinate as shown in Fig.1

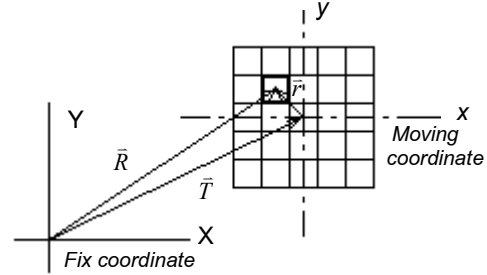


Fig. 1 The relation of the moving coordinate in the fix coordinates.

The motion of the rectangular tank was a direct consequence of the flow induced forces acting on its wall. The evaluation of 1-DOF motions was applied to the entire control volume which comprised sinusoidal translation as

$$T = A \left[\sin \left(\frac{2\pi}{T_p} + \phi \right) \right] \quad (17)$$

where T was the translation of a tank or the entire control volume, T_p was the translation period and ϕ was the translation phase

3. Experiment Setup

The research methodology comprised of the experimental and numerical methods of the fluid sloshing. The experimental setup had been developed to investigate the surface wave and displacement of liquid (water) as shown in Fig.2. The clear acrylic rectangular tank without baffle had dimension of 300×240×50 mm (width×height×depth). The tank with baffle had same dimension as the tank without baffle. A baffle had dimension of 50×120 mm (depth× height) and installed at middle of the width of the tank. The water-filled inside the tank at three levels that are 1/4, 1/2 and 3/4 of the tank volumes. The motion of the partially water filled tanks (all of levels of water) were controlled by the servo motor and the ball screw with velocity of 200, 300 and 400 mm/s, respectively. The velocity and position of the tank were controlled by voltage supplied to the servo motor. The input voltage controlled the time function of position for this experiment as shown by graph in Fig.3. The water sloshing inside the tank would be carried out using a Logitech webcam, model “Quickcam Pro for Notebook” which moved with the tanks and connected to a computer via a USB port.

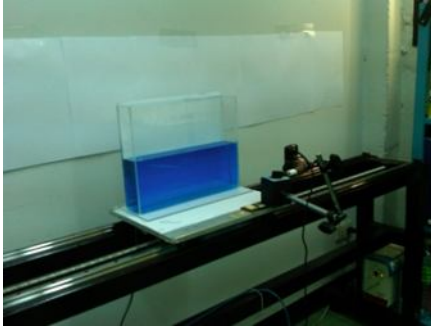


Fig. 2 Apparatus of experimental device for sloshing

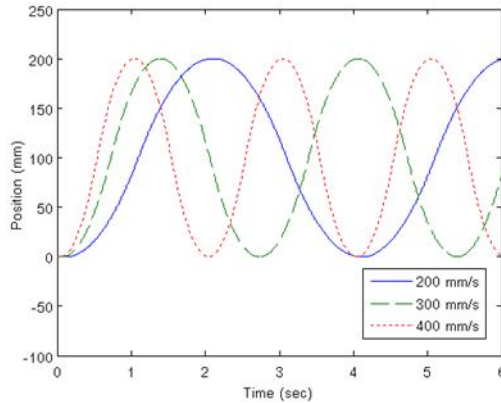


Fig. 3 The input signals to control the position of the rectangular tanks.

The governing Eqs. (1) – (17) were discretized and implemented using C++ language in three dimensions space of OpenFOAM. The 1,920 small parallelepiped control volumes were created and combined to be the tank which dimension is 300×240×50 mm (width×height×depth) as shown in Fig.4a. The 1,920 small parallelepiped control volume were also created and combined to be the tank with baffle had dimension 50×120 mm (depth× height) as shown in Fig 4b.

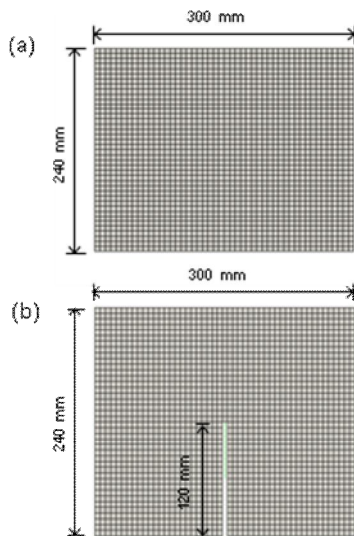


Fig.4 (a) FV model of the tank without baffles and (b) FV model of the tank with a baffle.

The volume of water at initial condition ($t=0$) performed by selecting cells to fill water and defined γ in Eq. (12) to be unity for these cells. The computer graphic of the initial condition of the partially water-filled were shown in Fig. 5. The red and blue color-filled in cells instead water and air, respectively. For solving the pressure and velocity of fluid inside the moving tank, the Pressure Implicit with Splitting of Operators (PISO) algorithm was used in this research. The laptop (specification: Pentium Core™2 Duo E7500-2.93 GHz and 2.0 GB RAM) was used for the simulated process.

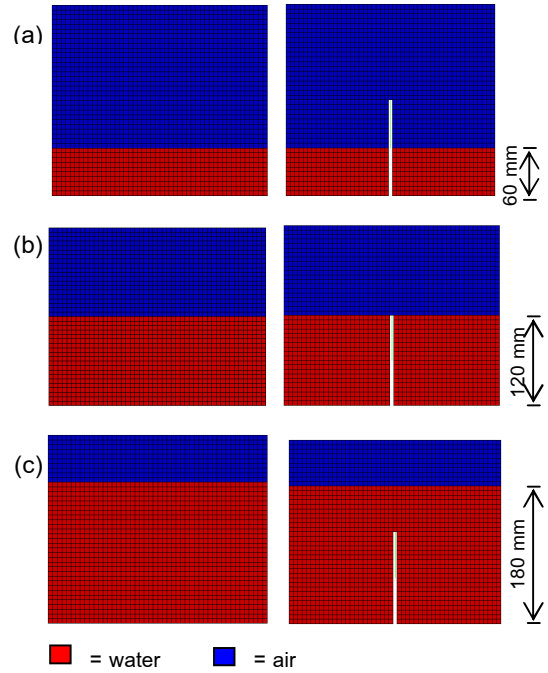


Fig. 5 The initial conditions for the FV models of the tanks with and without a baffle when the water-filled were (a) 1/4, (b) 1/2 and (c) 3/4 of tank volumes.

4. Results and Discussion

The tanks were controlled to move back and forth from left to right with velocity of 200, 300, 400 mm/s and positions according to the input signals (Fig. 3). The water sloshing inside the moving tank was recorded. Examples of the image series for the surface wave of the water-filled 1/2 of the tank volume for the tank with and without baffle moving 400 mm/s were shown in Fig.6. The height of surface wave developed on the wall of the tank with a baffle expressed that the baffle could reduce the severe of the surface wave when compared the image of the height of the surface wave on the wall of the tank without baffle.

The computational results were displayed using the third-party Open Source Software (ParaView) of OpenFOAM which could convert the numerical results to the graphic results. The image series of the FVM at the same time of the experiment were shown in Fig.7. The surface of water inside both tanks. (with and without a baffle) had a good correlated movement when compared with the surface wave of the experiments.

The sequences of the surface wave could express the maximum height of the sloshing water inside the rectangular tank which moved from the left to the right. The height of the surface wave at the tank wall would lift up and fall down corresponding to the movement of the tank.

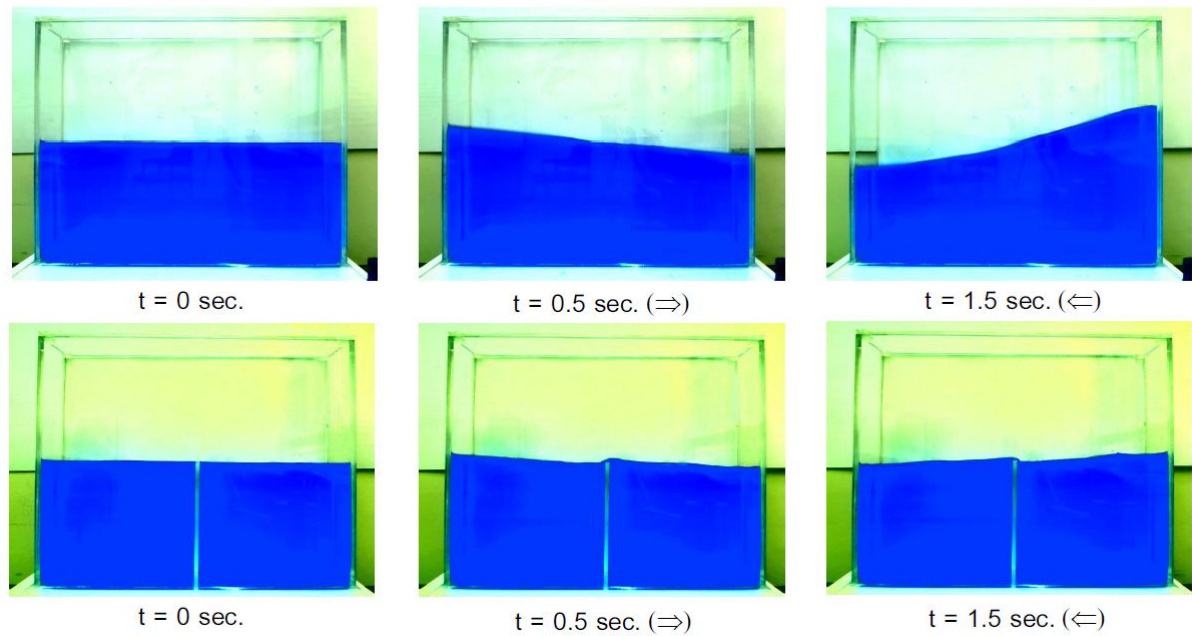


Fig. 6. Sequential images of the experimental surface wave inside the tank with and without a baffle at the initial condition and move to right ($\Rightarrow$) and left ($\Leftarrow$), respectively.

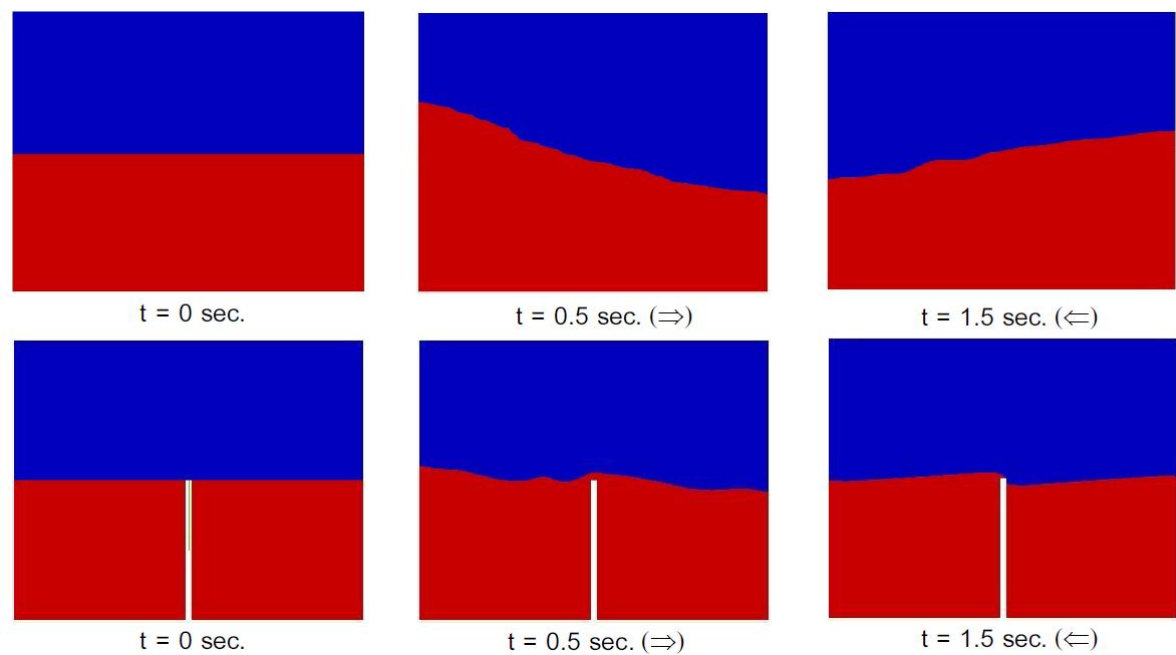


Fig. 7. Sequential images of the FVM surface wave inside the tank with and without a baffle at the initial condition and move to right ($\Rightarrow$) and left ($\Leftarrow$), respectively.

The “SloshDetector” software, [9], was used to determine the coordinates of the surface wave height at the left wall of tank. The variable heights of the surface wave on the left wall of tank could be plotted to compare between CFD and experimental results as shown in Figs. 8-13. The average error of the CFD when compared with the experiment results for the volume of water were 1/4, 1/2 and 3/4 of the tanks for every velocity of the tanks movement were 6.22%, 2.67% and 2.29%, respectively. Graphs in Figs. 8-10 shown for increasing of water levels and velocities of tank movements directly affected to the sloshing phenomena which had the severe sloshing. Decreasing the violent sloshing, a baffle was required to be installed into the tank. The effects of a baffle for reducing the severe sloshing were shown in Figs. 11-13. The comparisons of the maximum surface heights on the left wall of the tank with and without a baffle were described in Table. 1. The free surface height at the same levels of the baffle height had excellent results for every velocity of the tank movements.

5. Conclusion

The FVM technique for the multi phases flow had been used to simulate partially liquid sloshing in the moving rectangular tank with and without a baffle. The average error of 3.73% when comparing CFD to experimental results confirmed the good performance of FVM. The liquid sloshing on a baffle of tank exhibited the prominent behavior that the displacements of free surface wave were decreased and become linearity. The appropriated levels of water inside the tanks were equal the height of a baffles. The shallow water flow across the tip of a baffle during tank moved from the left to right cause the most reducing of the severed sloshing.

These effects of baffle that received from the experiment and FVM results for the liquid sloshing were the good study case. Furthermore, the FVM in OpenFOAM software would be used to design the good efficiency baffles inside tank.

Table.1 Max. different height of the wave surface in the tanks with and without a baffle

Initial surface height (mm)	Velocity of tanks (mm/s)	Different height between the wave surface and initial surface.	
		Tank without baffles (mm)	Tank with a baffle (mm)
60	200	14.62	5.81
	300	21.37	6.45
	400	32..84	11.81
120	200	5.00	2.7
	300	11.45	4.4
	400	68.71	5.5
180	200	7.55	5.85
	300	7.91	11.22
	400	47.68	27.55

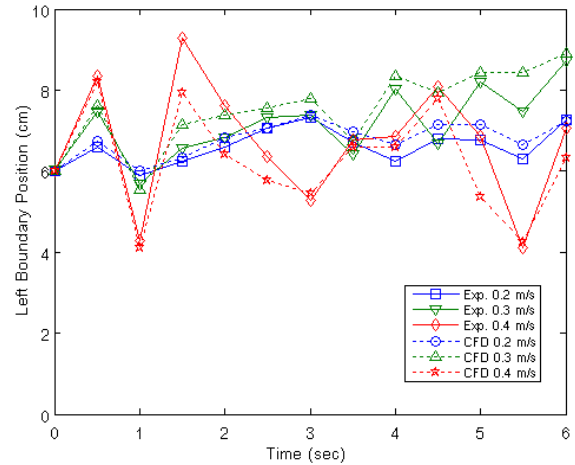


Fig. 8 Plot of the surface heights on the left wall of tank without baffle vs. time comparing CFD to experiments of the water 1/4 tank volume.

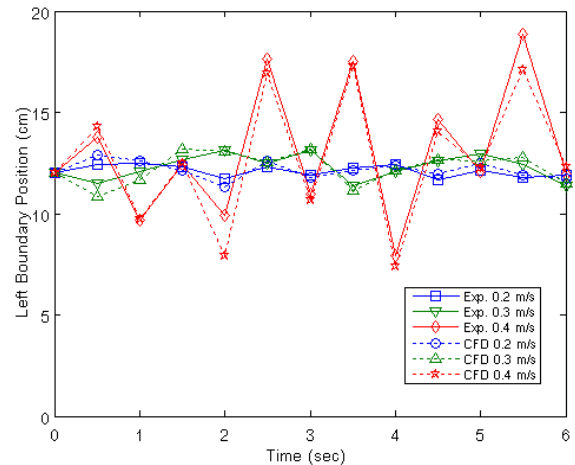


Fig. 9 Plot of the surface heights on the left wall of tank without baffle vs. time comparing CFD to experiments of the water 1/2 tank volume.

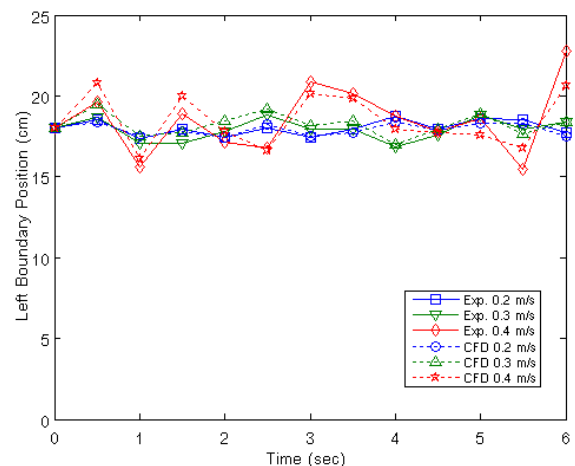


Fig. 10 Plot of the surface heights on the left wall of tank without baffle vs. time comparing CFD to experiments of the water 3/4 tank volume.

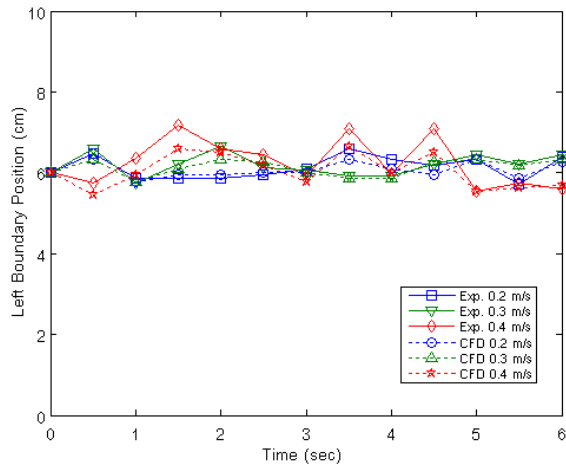


Fig. 11 Plot of the surface heights on the left wall of tank with baffle vs. time comparing CFD to experiments of the water 1/4 tank volume.

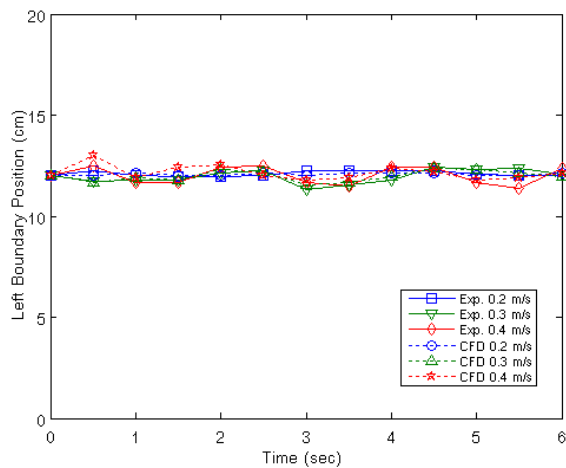


Fig. 12 Plot of the surface heights on the left wall of tank with baffle vs. time comparing CFD to experiments of the water 1/2 tank volume.

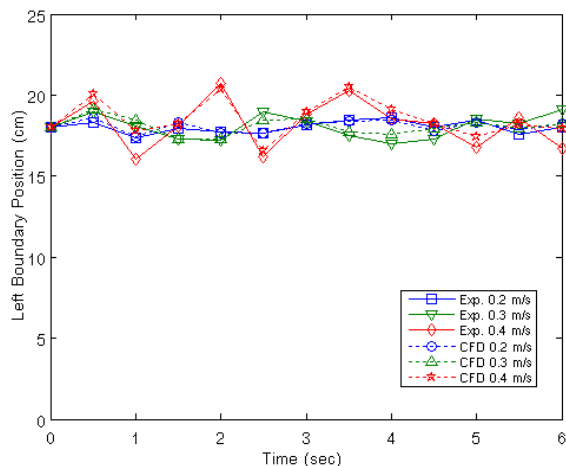


Fig. 13 Plot of the surface heights on the left wall of tank with baffle vs. time comparing CFD to experiments of the water 3/4 tank volume.

6. Acknowledgement

The authors wish to thank Asst. Prof. Tumrong Puttapitukporn on the used of the experimental devices at Mechanical Engineering Department, Kasetsart University.

7. References

- [1] Akyildizan, H. and Erdem U.N. (2006). Sloshing in a three-dimensional rectangular tank: Numerical simulation and experimental validation, *Ocean Engineering*, vol. 33, March 2006, pp. 2135–2149.
- [2] Liu, D. and Lin, P. (2008). A numerical study of three-dimensional liquid sloshing in tanks, *Journal of Computational Physics*, vol. 227, December 2007, pp. 3921–3939.
- [3] Cho, J.R., Lee, H.W. and Ha, S.Y. (2005). Finite element analysis of resonant sloshing response in 2-D baffled tank, *Journal of Sound and vibration*, vol. 288, April 2005, pp. 829–845.
- [4] Eswaran, M., Saha U.K. and Maity, D. (2009). Effect of baffles on a partially filled cubic tank: Numerical simulation and experimental validation, *Computers and Structures*, vol. 87, December 2008, pp. 198–205.
- [5] Rebouilliant, S. and Liksonov, D. (2010). Fluid-structure interaction in partially filled liquid container: A comparative review of numerical approaches, *Computers & Fluids*, vol. 39, January 2010, pp. 739–746.
- [6] Threepopnartkul, K. and Suvanjumrat, C. (2010). Development and validation of a C++ object oriented CFD codes for two dimensional fluid sloshing simulations, paper presented in *The 36th Congress on Science and Technology of Thailand*, Bangkok, Thailand.
- [7] Anderson Jr., J.D. (1995). *Computational fluid Dynamics: The basics with applications*, McGraw-Hill, New York.
- [8] Hirt, C.W. and Nichols, B.D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries, *Journal of Computational Physics*, vol. 39, November 1979, pp. 201–225.
- [9] Suvanjumrat, C. and Puttapitukporn, T. (2010). Sloshing surface monitoring using image processing, paper presented in *The first TSME International Conference on Mechanical Engineering*, Ubon Ratchathani, Thailand.

The Two-Phase Microchannel Flow Study of Chicken Blood on Lab-on-a-Chip

A. Pramuanjaroenkij^{1,*}, K. Thanomsit¹, B. Chaiburi¹, G. Phonong¹, A. Khanaphol¹,
W. Phutthasawong¹, T. Maturos², T. Lomas², A. Tuantranont², S. Kakaç³ and C. Thipyopas⁴

¹Kasetsart University, Chalermphrakiat Sakon Nakhon Province Campus, Sakon Nakhon 47000 Thailand

²Nanoelectronics and MEMS Laboratory, National Electronics and Computer Technology Center,
Pathumthani 12120 Thailand

³Department of Mechanical Engineering, TOBB University of Economics and Technology,
Ankara, 06560 Turkey

⁴Department of Aerospace Engineering, Kasetsart University, Bangkok 10900 Thailand

Abstract

In previous study, we introduce problems in assembly the polydimethylsiloxane (PDMS) lab-on-a-chip (LOC) devices from literatures; a manual assembly by using acrylic plates and an adhered assembly by using a plasma cleaner. Since electrodes in the devices had been designed to provide a strong dielectrophoretic force by using sputtering and electroplating techniques, the thick electrodes were used. The PDMS LOC was also aimed to use with biological samples to study the different flow patterns of the samples prepared from chicken blood and to solve a PDMS hydrophobic problem from our previous study where the sample cannot flow through a channel of the adhered assembly LOC. Since the chicken blood contains of whole blood and plasma, the sample can be considered as a combination of particle and fluid parts or the two-phase flow. The channels on the PDMS LOC having dimensions of the order of microns can be considered as microchannels; Knudsen number is used to distinguish flow regimes. From the experiment study, the flow problem has still existed in the adhered method but the flow problem did not exist on the manual method with a new-acrylic-plate set. Slip flow occurred at the leak areas because interactions between the sample and walls became significant. Different flow patterns occurred in the channels of workable LOCs in different channel distances. No slip (continuum) flow occurred at the middle path where pressure drop was decreased. The hydrophobic problem was solved by using Bovine Serum Albumin solution to reduce PDMS hydrophobic behavior and this solution did not affect red blood cells because of none damaged cells.

Keywords: LOC, Lab-on-a-Chip, Chicken Blood, Two-Phase Flow, PDMS, DEP Force.

1. Introduction

Lab-on-a-chip or LOC devices are miniature laboratories built on a thin glass or plastic chip of several micrometers in dimensions. These small devices can duplicate the specialized functions as their room-sized counterparts in clinical diagnoses; the advantages of these devices include significantly reduced reagent consumption, short analysis time, automation, and portability [1]. Researches on LOC systems have been developed by the aim to miniaturize, integrate, and automate biochemical assays. The development of the LOC systems and devices requires the integration of multiple fluidic functions onto a single system or device with the ability of a well defined fabrication process to fabricate cost efficient and portable devices. Since there is the large diversity in microfluidic developments, collaborations, and supply chains of different companies forming new solutions by combining off-the-shelf components are still out of reach [2]. Microfluidic platforms provide a variety of design components and fabrication technologies that are well tuned on to each other [3]. In a microfluidic platform, basic fluidic functions (unit operations), are reagent storage, reagent release, fluid transport, fluid metering, fluid mixing, flow controlling, and separation or concentration of molecules or particles [2].

Many experimental and analytical researches have been performed to understand fluid flow at

microscale for the design and the development of micro devices. However, there are no generalized solutions for the determination of flow characteristics in designing micro devices. On the other hand, the predictions of the friction coefficients and velocity profiles in microchannels are important in the application of electronics where such channels formed in the micro devices. Experimental and theoretical results for liquid flows cannot be considered or modified for gaseous flows in the microchannels because the flow regime boundaries are significantly different as well as flow characteristics. Surface to volume ratio in microstructures are greater than it is for macrostructures so surface effects become dominant [4]. This effects result an important pressure drop and greater mass flow rate than that is predicted with the continuum theory. The velocity in microchannels is not very high due to large pressure drops. Because of small dimensions, Reynolds number becomes smaller. In the transition from laminar to turbulent flows viscous effects and friction factors may be higher than those are predicted by conventional theory. All these discrepancies affect both the fluid flow and the convective heat transfer in microchannels [4, 5]. The high pressure gradients in microchannels lead to low flow rates. In small devices, the ability of the fluid stream to carry the heat away for a given temperature rise becomes limited but reducing the flow length of the channels and increasing the liquid flow rate must be

*Corresponding Author: anchasa@gmail.com, Tel. +6642725033, Fax. +6642725034

matched with their performances. As a result, employing multiple streams with short paths in a microdevice was recommended. One should note that, in the transition and turbulent flow regions, the reduced flow length reduces the pressure drop, multiple inlets enlarging channel area where the heat transfer rate is higher, and turbulent flow can help to increase heat transfer coefficients in these regions [6]. When interactions between fluid molecules and channel walls become as frequent as intermolecular collisions, the boundaries and the molecular structure become more effective on the fluid flow, these effects are called rarefaction effects [4]. The local Knudsen number is used to determine the degree of rarefaction and the degree of deviation from continuum model.

$$Kn = \frac{\lambda}{L} \quad (1)$$

where L is a characteristic flow dimension such as channel hydraulic diameter (D_h) and λ is the mean free molecular path corresponding to the distance travelled by the molecules between the collision. In continuum flow regime, no-slip flow where $Kn \leq 0.001$, continuum assumption which is widely used for macroscopic problems becomes valid. In the slip flow regime where $0.001 < Kn \leq 0.1$, continuum model is applicable except in the layer next to the wall which can be identified as Knudsen layer. For the Knudsen layer, slip boundary conditions should be considered. If the flow is in the transition regime, where $0.1 < Kn \leq 10$ and continues into free molecular flow regime, molecular approach should be used. In other words, Boltzman equation should be considered for atomic level studies of gaseous flow in transition regime [5]. Many experimental investigations on convective heat transfer of single-phase liquid flows in microchannels have been in the continuum regime [4].

Our previous study [7] presented problems and results in fabricating the microfluidic devices for biological sample manipulation at Kasetsart University, Chalermphrakiat Sakon Nakhon Province Campus. Electrodes obtained from electroplating and sputtering techniques were used to produce dielectrophoretic force on the devices. In the work, the electrodes were fabricated with three different current conditions, the higher current in electroplating process, the coarser electrode surfaces. The polydimethylsiloxane (PDMS) lab-on-a-chip (LOC) design from a literature was considered specifically as a microfluidic device prototype. The PDMS LOCs were adhered by using the plasma cleaner at Nanoelectronics and MEMS Laboratory, National Electronics and Computer Technology Center (NECTEC) and were fabricated manually at Chalermphrakiat Sakon Nakhon Province Campus but all components were prepared at NECTEC. The finished LOC with the coarse surface electrodes showed assembly problems in both assembly methods. Moreover, in the work, the biological sample could not flow through a channel of the adhered assembly LOC showing a hydrophobic problem. The LOC fabricated locally and manually without using plasma cleaners can be reassembled, considered as the main advantage. However, the re-assembly requires skills and techniques to arrange all components to be on exact positions. The completed-workable LOCs were examined with two sets of biological samples, prepared from chickens with poor and good health conditions. The results showed that red blood cells in the sample from the good health condition

could flow and be accelerated pass the electric field from the electrodes. But the red blood cells in the sample from the poor health condition could not flow smoothly to pass the electric field and some of them were moved forward to the electrode. Therefore, the last feasibility study showed that there was possibility to develop the LOC technology locally with helps from Nanoelectronics and MEMS Laboratory [7].

In this current study, the leak LOC problems were examined and solved by observing biological samples flow in the LOCs. The best electrode condition from the previous work was implemented in the current work. The biological sample was prepared [7] from healthy chicken blood. Sample flow patterns in the microchannels were observed to investigate flow behaviors, since the biological sample flow can be distinguished as two-phase flow in the microchannel. The hydrophobic problem was investigated and solved.

2. Experimental Study

The LOCs were fabricated according to the literature [7]; a manual assembly by using acrylic plates and an adhered assembly by using a plasma cleaner. The mask for fabricating electrode was designed according to Cetin design [8] by using L-Edit software. In the fabrication process, the silicon wafer was used to produce a master prototyping of the PDMS microstructure and was patterned by using the negative photo-resist (SU-8 25, MicroChem Co., Newton, MA) technique. The dielectrophoretic chamber was made from PDMS prepared by mixing the precursors sylgard with a curing agent at a ratio of 10:1 by volume. The prepolymer mixture was degassed at 20-50 mTorr at room temperature in desiccators pumped with a mechanical vacuum pump for 10 minutes to remove any air bubbles in the mixture. The PDMS mixtures were gradually poured onto the patterned silicon wafer or a mold. After the PDMS was cured at 100°C for 30 minutes on the mold, the molded polymer samples were peeled off and punched holes in order to create chambers. Because electroplating technique can vary electrode surface conditions [9], the chromium electrode array was patterned on glass slides by only DC sputtering technique through microshadow masking. The chromium were sputtering under argon plasma with sputtering pressure, sputtering current, and time of 3×10^{-3} mbar, 0.2 A, and 2 minutes, respectively. The sputtering was conducted at room temperature.

For the adhered assembly by using a plasma cleaner [10-20], the chamber and electrodes were treated under oxygen plasma (Harrick scientific corp. model PDC -32 G) before being attached to each others. The microelectrodes were inserted into the PDMS electrode chambers manually under the microscope. In a manual assembly [9], acrylic plates were investigated to improve a leak problem and searched for a new design.

On the LOCs, channel width and height were 50 μm and 25 μm , respectively. With these dimensions, the channels can be considered as the microchannels. In the workable LOCs, the sample flow patterns were observed to investigate flow behaviors, since two-phase flow in the microchannels can be different from liquid flow in the microchannels.

With the PDMS hydrophobic problem, Bovine Serum Albumin solution as a new solution was mixed with the sample and the PDMS sheets were kept under water to solve the hydrophobic behavior. The Bovine

Serum Albumin solution was inspected for cell damaging while it flowed with the sample.

3. Results and Discussion

The thick electrodes affected both ways of LOC assembly, the thinner electrodes from only applying the sputtering technique were used in the LOCs. The sample flowing through the adhered LOC with the thinner electrode showed a problem, Fig.1, collapse of channel ceiling. In the manual assembly, the former (simple) acrylic plates (Fig. 2) were replaced by a new-acrylic-plate set as shown in Fig. 3. The LOCs from both assembly methods were workable.

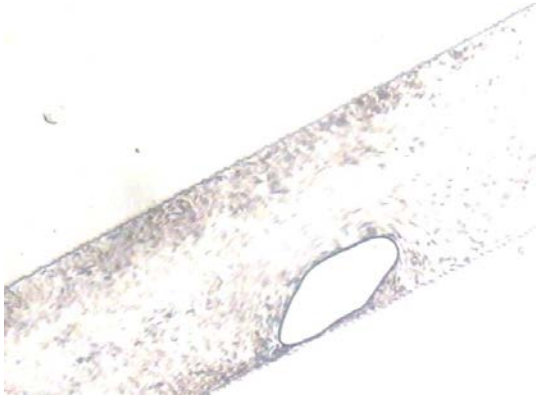


Fig. 1 Collapse of channel ceiling in the adhered LOC.



Fig. 2 The first acrylic plates used in the previous work [7].

Different flow patterns were observed in the channels for both workable LOCs and leak LOCs. From the observations, slip flow occurred at the leak areas as in Fig. 4–5 because friction between the flow and the surfaces was increased, interactions between the sample and the surfaces became significant and higher pressure drop occurred because higher friction force at boundaries. In the workable LOCs, the LOCs were inclined, before the sample loaded, to let the gravitational force drove the sample flow. No slip (continuum) flow occurred in the workable LOC at the middle path (Fig. 6) because of less friction at the boundaries which implied that lower pressure drop and smaller mean free molecular path (less friction to pull molecules apart), hence, lower Knudsen number took place.

In this study, we found the sample flowed in the microchannels pass the electrode area, in the workable LOC, with the velocity of 25.28×10^{-6} m/s. At low velocity, the particles flowed smoothly along the liquid part with no-slip condition between the particles and the

liquid. From the flow patterns and the velocity, the sample including particle and liquid parts flowed in the same patterns as single-phase liquid.

Next, the hydrophobic problem was investigated and solved by choosing Bovine Serum Albumin to reduce the PDMS hydrophobic behavior as shown in Fig. 7. From Fig. 7(b), the Bovine Serum Albumin did not affect red blood cells as it did not damage any red blood cells. Another experiment to reduce the PDMS hydrophobic behavior was keeping the PDMS sheets under water, then the sheets were dried by air blasting before they were assembled with other components. Since the sample flowed inside the LOCs with the submerged PDMS as shown in Fig. 8, keeping the sheets under the water can be another solution to solve the hydrophobic behavior.

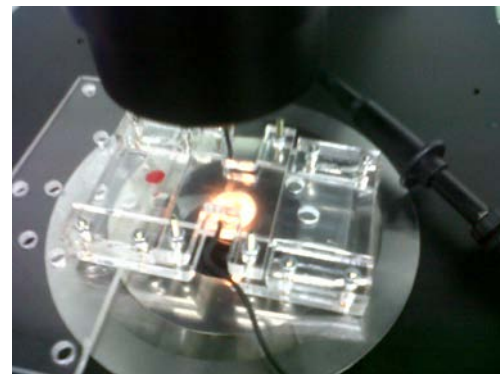
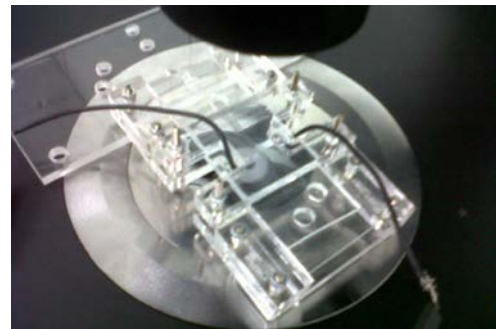


Fig. 3 A new-acrylic-plate set used in the current work.



Fig. 4 Slip flow profile occurred at entrance of a leak LOC because of high friction force at boundaries.



Fig. 5 Slip flow profile occurred at the middle path of the channel because of leaking next to an electrode

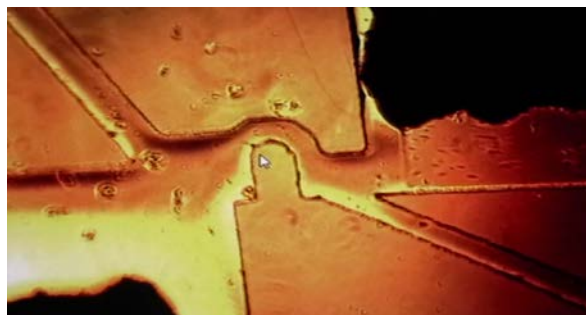


Fig. 6 No slip flow or continuum flow profile occurred at the middle path of the channel because of less pressure drop.



(a)



(b)

Fig. 7 Biological sample flowing with Bovine Serum Albumin (black particles) (a) overview picture and (b) zoomed in picture.

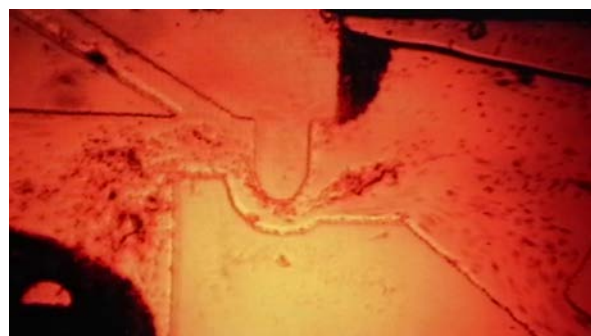


Fig. 8 The workable LOC with a submerged PDMS sheet.

4. Conclusion

From the experiment study, the problem in the adhered LOC has still existed. The sample flow in the manual LOC was improved with a new-acrylic-plate set. Slip flow occurred at the leak areas because higher friction. The sample flow in our study was a two-phase flow with low velocity, so no-slip condition between particles and fluid. Therefore, the sample flow in the microchannels behaved the same trend as the liquid flow in the microchannels, a generalized solution for the determination of flow characteristics as mentioned in a literature [21]. The hydrophobic problem was solved by keeping the PDMS sheets in water and using Bovine Serum Albumin solution to reduce PDMS hydrophobic behavior. The latter solution did not affect red blood cells because of none damaged cells.

5. Acknowledgement

This work has been supported by Research and Development Institute Chalermphrakiat Sakon Nakhon Province Campus, Faculty of Science and Engineering, Kasetsart University, Chalermphrakiat Sakon Nakhon Province Campus, and Nanoelectronics and MEMS Laboratory, National Electronics and Computer Technology Center, Thailand.

6. References

- [1] Li, D. (2010). Microfluidics lab-on-a-chip devices for biomedical application. In: S. Kakaç, B. Kosoy, D. Li, and A. Pramuanjaroenkij, eds., *Microfluidics Based Microsystems Fundamentals and Applications*, Springer: Netherland, pp. 377 – 397.
- [2] Mark, D., Haeberle, S., Roth, G., Von Stetten, F. and Zengerle, R. (2010). Microfluidic lab-on-a-chip platforms: Requirements, characteristics and applications. In: S. Kakaç, B. Kosoy, D. Li, and A. Pramuanjaroenkij, eds., *Microfluidics Based Microsystems Fundamentals and Applications*, Springer: Netherland, pp. 305 – 376.
- [3] Haeberle, S. and Zengerle, R. (2007). Microfluidic platforms for lab-on-a-chip applications, *Lab Chip*, vol.7, 1094 – 1110.
- [4] Kakaç, S., Liu, H. and Pramuanjaroenkij, A. (2012). *Heat Exchangers: Selection, Rating, and Thermal Design*, ISBN: 1-43984-990-0, CRC Press, Boca Raton.
- [5] Zhang, M. Z. (2007). *Nano/Microscale Heat Transfer*, ISBN: 0-07143-674-X, McGraw-Hill, New York.
- [6] Kandlikar, S., Garimella, S., Li, D., Colin, S. and King, M. R. (2005). *Heat transfer and fluid flow in minichannels and microchannels*, ISBN-13: 978-0080445274, Oxford, UK.

- [7] Pramuanjaroenkij, A., Napinij, U., Teingtit1, W., Boonthueng, S., Maturos, T., Tanom, L., Tuantranont, A. and Kakaç, S. (2011). *Feasibility study on Lab-on-a-chip fabrication in Kasetsart University, Chalermphrakiat Sakon Nakhon Province Campus*, The 2nd TSME-ICoME International Conference on Mechanical Engineering, Sheraton Krabi Beach Resort, krabi, Thailand.
- [8] Cetin, B. (2009). Microfluidic continuous separation of particles and cells by AC-dielectrophoresis, *Ph.D. dissertation*, Graduate School of Vanderbilt University, Tennessee, USA.

Time to Seizure Comparisons between Different Plain Journal Bearings

James Moran^{1,*} and Phonlathep Tangsukkasemsan²

¹Department of Mechanical Engineering, Faculty of Engineering, Chiang Mai University, Chiang Mai, Thailand 50200

²SCG-DOW Group, Asia Industrial Estate, Rayong, Thailand 21130

Abstract

This paper is a continuation of the research which was presented at last year's TSME conference. In that paper [1], two new concepts for plain journal bearings were presented. Their design takes advantage of the fact that the coefficient of friction is not just a function of the specific materials sliding together but also a function of time, load and environment. The coefficient of friction will increase with time until a steady state value is reached, where the rate of particle generation matches the rate at which particles are removed. The new designs took advantage of this information by incorporating particle traps and compliant surfaces to reduce friction. Since then an experiment has been designed and built to test both of the new design concepts against a regular plain journal bearing. The experimental data did not produce absolute data on the friction coefficient but it did allow a relative comparison between the different bearings. A time to seizure was produced in every case. The new concepts produced time to seizures ranging from 3 times greater than the regular bearing to over 5 times greater.

Keywords: Alumina, Nanofluids, Thermal Conductivity, TPS, Sensor

1. Introduction

Journal bearings are widely used in engineering applications. Typically they are used in low cost situations where the performance requirements are not as stringent as in the case with ball bearings [2]. Journal bearings are simple and inexpensive but can suffer from variable friction and high wear in comparison with the lubricated or ball bearing types [3]. An example of a plain journal bearing is shown in Figure: 1.



Figure: 1 Plain journal bearing on a shaft

In a paper published at last year's TSME conference [1] two novel inexpensive bearing concepts were proposed. They both took advantage of fundamental studies on the nature of friction by Suh et al. [4]. Other researchers have examined the specific components of friction, such as adhesion [5], asperity interaction [6] and plowing [7]. Both design proposals were intended to reduce the effects of friction from each of these three components. This is achieved through having less surface contact, compliant surfaces and particle traps. In this paper an experiment was designed and built in order to compare the performance of these bearings with that of a regular plain journal bearing. When a bearing generates enough particles on its surface it can seize. Eventually, given enough time and use all journal bearings seize. By developing a test which runs the

bearings until they seize a relative comparison can be made.

Both of the new designs shall be briefly described once more before describing the experiment. More complete details of their design can be found in [1].

2. Concentric Ring Design

This proposed design consists of a series of concentric brass rings. Each ring has an inner diameter of 12.1mm and a thickness of 1mm. They are spaced 2.5mm apart and there are 9 rings in total. The shaft fits through the center of these rings. They are supported in a housing of polyurethane. This assembly is enclosed in a 2mm thick stainless steel sheath, as shown in Figure: 2

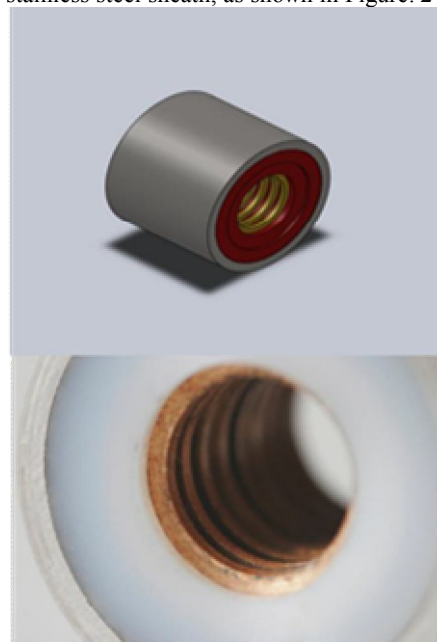


Figure: 2 Concentric ring design concept

*Corresponding Author: E-mail: jamesmoran12@gmail.com, Tel: 053 944 146 Ext. 923, Fax: 053 944 145

Adhesion should be reduced because there is less surface contact between the shaft and bearing. Particles between the shaft and the bearing surface are not trapped there. They can easily fall out via the particle traps as shown in Figure: 3.

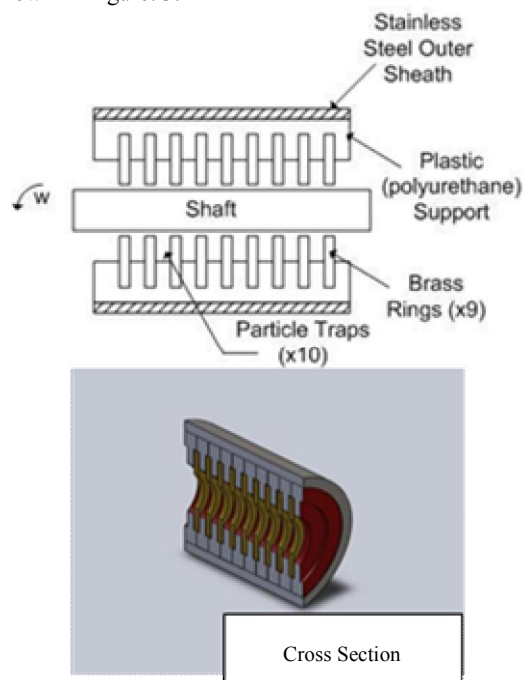


Figure: 3 Particle traps in concentric ring configuration

The component from plowing is therefore considerably reduced. If there are any large asperities on the shaft the ring can move as the shaft rotates against it. It can compress against the compliant surface.

Figure: 4 shows a picture of the completed design. The bearing length is 26mm and is designed for a shaft diameter of 12mm.



Figure: 4 Concentric ring bearing

3. Inner Spring Design

In an effort to reduce cost and yet maintain many of the advantages of the concentric ring design an alternative design concept was introduced. This is simply a stainless steel helical spring coil which the shaft is mounted through. The outside of the spring is supported with a steel sheath. The plastic support structure may be used in the same way as the previous design, if necessary. The spring used in Figure: 5 was manufactured from 3mm diameter wire and has a 4mm pitch. Two other spring bearings were manufactured with

2mm diameter stainless steel wire and a 5mm and 3mm pitch respectively.



Figure: 5 Inner spring bearing

If there is a large asperity on the shaft, in theory it should only make contact with the spring once per revolution. In a regular bearing this asperity would be in constant contact with the surface. Figure: 6 displays this mechanism.

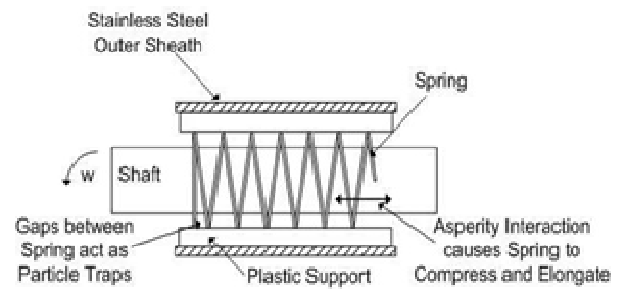


Figure: 6 Asperity interactions with inner spring design

The manufactured prototype for this design is shown in Figure: 5. The length is 32mm and it accommodates a shaft diameter of 12mm. A provisional patent was filed in Thailand, (patent pending #1001000704) for these designs.

4. Test Set Up

The next step was to design and build a test bed to test these bearings. Their performance should be compared with the plain bronze bearing shown in Figure: 1. A preliminary test bed was constructed. It applied a load to the bearing under test and a shaft was rotated inside the bearing. The power going into the motor was recorded along with the angular shaft speed. The time before the bearing eventually seized was recorded.

The test bed that was built is shown in Figure: 7. It consisted of the following basic components:

- 12 mm stainless steel shaft for testing
- Motor for driving the shaft (A salvaged NA4565D motor from NISCA motors)
- Bearing under test
- Belt/pulley system for power transmission
- Power Supply (Acbel 330 Watt power supply obtained from a salvaged computer)
- Aluminum frame
- 2 supports for the shaft
- Standard weight for applying the load
- Voltmeter/Ammeter
- Digital Tachometer



Figure: 7 Test Bed

5. Test Procedure

1. The bearing under test was setup and the shaft was aligned through it
2. A 2kg load was placed on the experimental bearing housing
3. The power supply was turned on and the shaft speed was adjusted until the angular speed was 750RPM
4. The voltage and current going to the motor was recorded every minute
5. The test was stopped when the motor stalled or overheated

6. Results

The principle behind the testing procedure was to be capable of comparing the relative performance of several bearings. The power going associated with the spinning shaft is given by:

$$P_{shaft} = \bar{\tau} \cdot \bar{\omega} \quad (1)$$

Where τ is the torque on the shaft in Newton Meters and ω is the shaft angular speed in radians per second. The torque is given by:

$$\bar{\tau} = \bar{r} \times \bar{F} \quad (2)$$

Where r is the displacement vector and F is the force on the shaft. Since the angle between the radius and the applied force is 90° and only the magnitudes are of interest, (2) becomes $\tau = r.F$ The force F , comes from the weight placed on the shaft, and the friction coefficient, μ .

They are related through (3).

$$F = \mu N \quad (3)$$

Combining Eqs.(1) – (3) gives:

$$P_{shaft} = \mu.N.r.\omega \quad (4)$$

Or the friction coefficient is given by

$$\mu = \frac{P_{shaft}}{N.r.\omega} \quad (5)$$

Ideally the force required to turn the shaft would be measured with a digital torque meter. However given budget constraints this was not a feasible option. Instead the power going into the motor was estimated by measuring the voltage, V and current I

$$P_{motor} = \bar{V} \cdot \bar{I} \quad (6)$$

The power going into the motor is not going to be equal to the power going to the shaft. For a start there are losses in the motor and losses associated with the transmission system. If an assumption is made that the total losses, o are a constant percentage of the incoming power then:

$$K_{loss} P_{motor} = P_{shaft} \quad (7)$$

K_{loss} could be up to 50% or more of the input power. For this experiment we do not know what

K_{loss} is. Substituting (7) into (5) gives the final value for the magnitude of the friction coefficient:

$$\mu = \frac{K_{loss} V.I}{N.r.\omega} \quad (8)$$

All of the variables in (8) are measured during the test, except for the loss coefficient. If this loss coefficient remains constant during separate tests, a reasonable assumption since the motor efficiency is constant, the relative friction coefficients between the different bearings may be obtained. Valuable qualitative data may still be extracted from the experiments, even though the precise value for the friction coefficient eludes us.

Figure: 8 displays the data for one set of experiments. The conditions for each experiment are the same. The plot shows the friction coefficient divided by the loss coefficient on the y-axis. On the x-axis is the number of rotations. The data with the diamond symbol is the original plain journal bearing, see Figure: 1. It begins at 0.8 and seizes up completely after approximately 7500 revolutions. The maximum μ/K_{loss} that the motor can

deliver is 1.8. The light blue line is the data for the 'ring' bearing. It starts out at a slightly lower value of 0.7 and stays fairly constant over the course of the experiment, 11250 revolutions. The next 3 data sets come from 3 separate 'spring' designs. They each contain a slightly different spring. The brown line has a 2mm stainless steel spring with a 3mm pitch. The green line has a 2mm diameter stainless steel spring with a 5mm pitch. The purple line has a 3mm diameter stainless steel spring with a 4mm pitch. All appear to be fairly flat over

the course of the experiment with the 2mm diameter 3mm pitch having the average lowest value A durability test was done on the 3 different designs. This was a similar experiment except that the shaft was spun inside the bearing until it seized. The length of time it took to completely seize up was recorded. The results

are shown in Figure: 9. At a value of $\mu/K_{loss} = 1.8$ the motor seized. For the plain journal bearing the time it took to seize was approximately 10 minutes. The time it took for the spring bearing with the 2mm diameter 3mm pitch spring inside to seize was approximately 37 minutes. The experiment for the ring design was run for 55 minutes and then stopped as the motor was getting hot. At no point during the 55 minutes did the motor come close to stalling

7. Discussion

In this paper two designs have been presented and initial testing carried out. The experiment allowed for the relative performance between the three different bearings to be observed. This has provided useful information.

The first conclusion is that for a regular bearing the friction coefficient is not constant. As can be seen from Figure: 8 the coefficient of friction depends on time as well as material properties of both interacting surfaces. This is consistent with the conclusions from Prof. Suh [8]. From the same figure it is possible to conclude that the new bearing designs have a friction coefficient that is more constant in time when compared with the regular bearing. The ring bearing has a lower coefficient of friction but the spring bearing initially has a higher friction coefficient. There is no satisfactory explanation at the moment as to why this is so.

After each test the stainless steel shaft was examined. It containing surface marks and scratches and was very rough. This was the case no matter which bearing was used. The shaft was replaced for each test. Each new test always started with two clean new surfaces.

Durability testing was run with the results displayed in Figure: 9. It shows that the best performing spring bearing lasted 3.7 times longer than the regular bearing. The ring bearing had outlasted the regular bearing by a factor of 5.5 before the test had to be stopped. It is unknown how much longer it would have been capable of running for.

There were issues with the experimental procedure. The most obvious was an inability to measure the shaft power directly. It had to be measured indirectly by monitoring the power to the motor. Also the test was not automated. Each piece of data had to be manually written down. This took some time and the conditions of the test may have changed as data was being collected. This probably accounts for some of the fluctuations in the data. The fluctuations were not commented on as it is impossible to tell if they are real or an artifact of the data recording time delay. The reason for these issues was due to financial and time constraints.

The next step is to design and build an experiment that takes into account these deficiencies and allows the collection of more accurate data. The experiment will need to measure the power on the shaft directly and take the data automatically. The exact friction coefficient and the effect of different spring size, ring size, plastic support and housing material needs to be characterized. The present results are very encouraging and can be used as a guide for further research. Presently this work is being undertaken at Chiang Mai University.

8. References

- [1] Moran J., Tangsukkasemsan P. and Asadatorn S. New Design Concepts for Low Friction Plain Journal Bearings. *TSME International Conference on Mechanical Engineering*. Ubon Ratchani : TSME, 2010.
- [2] Avalonne E., Baumeister T. and Sadegh A. Mark's Standard Handbook for Mechanical Engineers. NY : McGraw-Hill, 2006.
- [3] Slocum, A.H. Precision Machine Design. NY : Prentice-Hall, 1992.
- [4] Suh, N.P. and Sin, H.C. The genesis of friction. *Wear*, 1981, Vol. 69, pp. 91-114.
- [5] Chowdhury, S.K. and Gosh, P. Adhesion and adhesion friction and the contact between solids. *Wear*, Vol. 174, pp. 9-19.
- [6] Kim, D.E., Cha, K.H. and Sung, L.H. 1, 2002, Design of surface micro-structures for friction control in micro-systems applications. *CIRP Annals - Manufacturing Technology*, Vol. 51, pp. 495-498.
- [7] Mosleh, M., Saka, N. and Suh, N.P. 2002, A mechanism of high friction in dry sliding bearings. *Wear*, Vol. 252, pp. 1-8.
- [8] Suh, N.P. Tribophysics. Prentice-Hall, 1986.
- [9] Evans, C.J. and Bryan, J.B. 1999, Structured, textured or engineered surfaces. *Ann. CIRP*, Vol. 48, pp. 541-556.
- [10] Wang, A. 2001, A unified theory of wear for ultra-high molecular weight polyethylene in multi-directional sliding. *Wear*, Vol. 248, pp. 38-47.

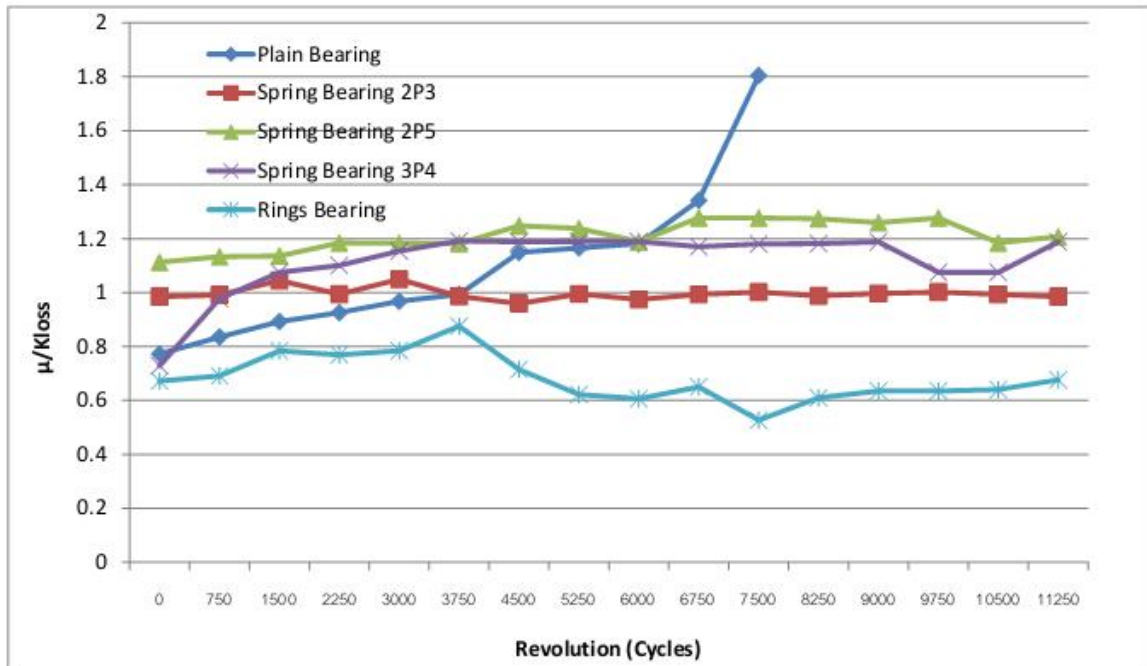


Figure: 8 Friction coefficient divided by loss coefficient for several bearings versus the number of revolutions

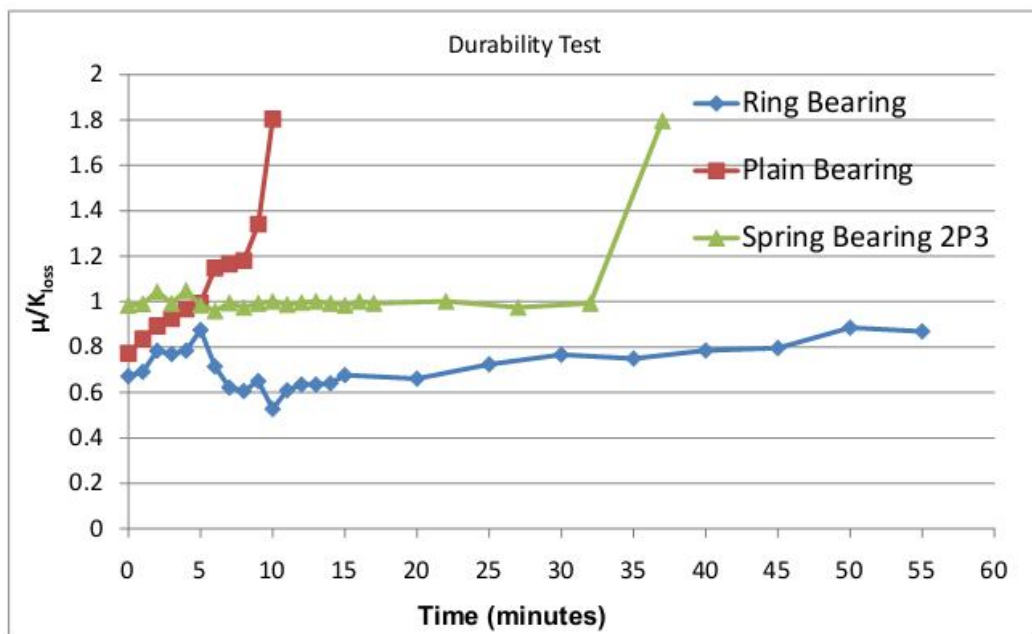


Figure: 9 Durability Test for three Bearing Styles

Subscription to JRAME

JRAME is “not for profit” but for a creation of novel ME information for the betterment of the world society at large. We need your paid subscription to continue this idealism in the long term.

The rate of subscription is only 125 Thai baths per issue. Subscription is accomplished easily by transferring money into the following bank account:

Account name: TSME journal
Savings Account No: 707-249035-6, Siam Commercial Bank Plc.,
Suranaree University of Technology Branch.

Then please inform us of your transaction via email to: jrame@sut.ac.th. Do not forget to provide your account number, full name, mailing address and telephone number.

Journal of Research and Applications in Mechanical Engineering
School of Mechanical Engineering, Institute of Engineering
Suranaree University of Technology,
111 University Avenue, Nakhon Ratchasima 30000, Thailand
Tel. (INTL)-66-44-224410, Fax. (INTL)-66-44-224613
E-mail : jrame@sut.ac.th Website : <http://eng.sut.ac.th/me/JRAME>

GUIDE FOR AUTHORS

The Journal of Research and Applications in Mechanical Engineering publishes results of research, applications, ideas and innovations related to mechanical engineering issues. Papers submitted to the journal must be (1) original (2) substantial (3) of significant importance.

PREPARATION OF MANUSCRIPT

Manuscripts submitted to JRAME must be written in English and should be formatted in the same form and style as that of the final journal article. A template for the article's style can be downloaded from <http://eng.sut.ac.th/me/JRAME>

The title should contain the article title, authors' names and affiliations and the address for manuscript correspondence (including e-mail address, telephone and fax numbers). Following the title is a single paragraph abstract summarizing the main findings of the paper in less than 150 words. After the abstract a list of up to 6 keywords that will be useful for indexing or searching should be included. The main body of the paper should start right after the keywords. Experimental procedure, data and computational methods must be explained in sufficient detail to permit replication by other researchers.

References

References may be cited in the text by Arabic numerals in brackets, e.g., [1], [2], or by the use of author-date, e.g. Johnson (2011); the former style is preferred. References cited in the text should be in numeral order but this is not an absolute requirement. Journal titles may be listed in full; abbreviations are also allowable.

Figures

Figures should be numbered consecutively with Arabic numerals. Lettering on drawings should be professional quality or generated by high-resolution computer graphics. Illustrations in color can be accepted only if the authors defray the cost. TIF or EPS files are required for figures.

Tables

Tables should be numbered consecutively with Roman numerals in order of appearance in the text.

SUBMISSION OF MANUSCRIPT

Files should be saved as Word (.doc or .docx) before uploading, to ensure the equations translate properly. All manuscripts and enquiries should be submitted via email attachment to jrame@sut.ac.th.

Manuscripts are accepted for review with the understanding that the same work has not been and will not be submitted elsewhere, and that its submission for publication has been approved by all of the authors and by the institution where the work was carried out. Any person cited as a source of personal communications must approve such citation.

Articles published in JRAME represent the opinions of the author(s) and should not be construed to reflect the opinions of the Editor and the Publisher. Copyright in the published manuscripts, including the right to reproduce the article in all forms and media, shall be assigned exclusively to the Publisher.