



3rd IS-FMTS

PROCEEDING: 3RD INTERNATIONAL SYMPOSIUM ON FLUID MECHANICS AND THERMAL SCIENCES (3RD ISFMTS 2019)

The Everly Putrajaya, Malaysia

14-14 September 2019

eISBN: 978-967-16649-1-9

Published By:





3rd International Symposium on Fluid Mechanics and Thermal Sciences (3rd ISFMTS 2019)
eISBN 978-967-16649-1-9
The Everly Putrajaya, Malaysia
14-15 September 2019

Copyright © 2019
Global Academic Excellence (M) Sdn Bhd

All rights reserved. No part of this proceeding may be reproduced in any form, except for the inclusion of brief quotations in review, without permission in writing from the author/ publisher.

eISBN 978-967-16649-1-9

GLOBAL ACADEMIC EXCELLENCE

PUBLISHED BY:
GLOBAL ACADEMIC EXCELLENCE (M) SDN BHD
(1257579-U)
KELANTAN
MALAYSIA

Contents

1. INVESTIGATION ON GENERALISED TRAPEZOIDAL DIFFERENCING TIME-MARCHING SCHEME FOR MODELLING OF ACOUSTICAL WAVE	1
2. INVESTIGATION OF THE EFFECT OF AWNING USING SUNLIGHT SENSOR TO REDUCE COOLING LOAD IN THE ROOM	10
3. ENERGY SAVING BY APPLYING DIFFERENT WALL THERMAL INSULATIONS ON A ROOM AT MALAYSIAN INSTITUTION.....	20
4. RECENT STUDIES ON LITHIUM-ION BATTERY THERMAL BEHAVIOUR FOR ELECTRIC AND HYBRID ELECTRIC VEHICLES: A REVIEW.....	31
5. DEVELOPMENT OF HEAT PIPE HEAT EXCHANGER TECHNOLOGY AS UTILIZATION OF LOW ENTHALPY GEOTHERMAL ENERGY FOR COFFEE DRYING PROCESS	42
6. STUDY ON THE AIRFLOW INDUCED BY MOVEMENT OF ELEVATOR CAR WITHIN THE ELEVATOR SHAFT USING PIMPLE ALGORITHM AND K-EPSILON TURBULENCE MODEL50	
7. PREPARATION, THERMAL AND MECHANICAL PROPERTIES OF MYCELIUM AS BIO-MATRIX IN PRODUCTION BIO-COMPOSITE	62
8. EVALUATION OF THERMAL PERFORMANCE OF HYBRID NANOFLUIDS IN PULSATING HEAT PIPE	71
9. COMPUTATIONAL METHODS FOR PREDICTING A PICO-HYDRO CROSS-FLOW TURBINE PERFORMANCE	83
10. EFFECT OF ANNULUS AND INLET HEIGHT RATIO IN STRAIGHT BLADE UNDERSHOT WATERWHEEL TURBINE BY COMPUTATIONAL METHOD	91
11. EFFECT OF BLADE DEPTH ON THE ENERGY CONVERSION PROCESS IN CROSSFLOW TURBINE	99
12. STAGNATION POINT FLOW OVER A STRETCHING/SHRINKING SHEET IN A CARBON NANOTUBES WITH SUCTION/INJECTION EFFECTS	108
13. GRAVITY-AIDED ULTRASONIC SEPARATION OF NANOPARTICLES FROM LIQUID USING MACRO-SCALE SEPARATOR.....	117
14. EFFECTS OF MOISTURE CONTENT AND TEMPERATURE ON THE DIELECTRIC STRENGTH OF TRANSFORMER INSULATING OIL.....	124
15. A REVIEW ON DEVELOPMENT OF LIQUID COOLING SYSTEM FOR CENTRAL PROCESSING UNIT (CPU).....	134
16. NUMERICAL INVESTIGATION OF TAPERED DL-MCHS WITH ALTERNATING FLOW CONFIGURATION	156
17. TEMPERATURE MINIMIZATION ON THE SUBSTRATE OF A HEAT SINK BY RIB-GROOVE MICROCHANNEL HEAT SINK WITH EFFECTIVE ENERGY CONSUMPTION: GROOVE GEOMETRY PARAMETER EFFECTS	167
18. A STUDY ON 2-BUTANOL DROPLET: ANALYSIS OF INTRODUCED CONSTANT IN THE NEW APPROACH ON FILM STAGNANT MODEL	180
19. MIXED CONVECTIVE STAGNATION POINT FLOW OF HYBRID NANOFLUID OVER A PERMEABLE STRETCHING/SHRINKING CYLINDER WITH THERMAL STRATIFICATION EFFECT	189
20. EFFECT OF BUOYANCY FORCE ON THE FLOW AND HEAT TRANSFER AROUND A THIN NEEDLE IN Al_2O_3 - Cu HYBRID NANOFLUID	202
21. STABILITY ANALYSIS OF MHD FLOW AND HEAT TRANSFER PASSING A PERMEABLE EXPONENTIALLY SHRINKING SHEET WITH PARTIAL SLIP AND THERMAL RADIATION214	
22. MAGNETOHYDRODYNAMIC (MHD) SLIP DARCY FLOW OF VISCOELASTIC FLUID OVER A STRETCHING SHEET AND HEAT TRANSFER WITH THERMAL RADIATION AND VISCOUS DISSIPATION	223
23. NUMERICAL EXAMINATION ON TWO-EQUATIONS TURBULENCE MODELS FOR FLOW ACROSS NACA 0012 AIRFOIL WITH DIFFERENT ANGLE OF ATTACK	235
24. EVALUATING THERMAL PERFORMANCE OF POUCH TYPE LITHIUM POLYMER BATTERY CELL	254
25. EFFECT OF SUCTION/INJECTION ON STAGNATION POINT FLOW OF HYBRID NANOFLUID OVER AN EXPONENTIALLY SHRINKING SHEET WITH STABILITY ANALYSIS	261
26. CHARACTERIZATION OF LIQUID OIL PRODUCT FROM HIGH DENSITY POLYETHYLENE WITH PYROLYSIS USING L-SHAPE HEAT PIPE HEAT EXCHANGER AS LIQUID COLLECTION SYSTEM	273



27. MAGNETIC FIELD EFFECT ON MIXED CONVECTION HEAT TRANSFER IN A LID-DRIVEN RECTANGULAR CAVITY	282
28. BOUNDARY LAYER FLOW AND HEAT TRANSFER OF BLASIUS AND SAKIADIS PROBLEMS IN NANOFLUIDS WITH PARTIAL SLIP AND THERMAL CONVECTION.....	291
29. CHARACTERIZATION OF U-BEND HEAT PIPE HEAT EXCHANGER TO APPLY AS DEHUMIDIFIER IN THE ELECTRONIC FACTORY	302
30. AVERAGE FRICTION FACTOR BETWEEN INLET AND OUTLET FOR LAMINAR FLOW OF GAS IN MICROCHANNELS.....	311
31. UNCERTAINTY OF TEMPERATURE MEASURED BY THERMOCOUPLE	319
32. NUMERICAL ANALYSIS OF CALCIUM CARBONATE (CACO ₃) SUSPENSION FLOW IN PENTAGON SPIRAL PIPE WITH PITCH VARIATION.....	327
33. THE EFFECTS OF MAGNETIC BLOOD FLOW IN AN INCLINED CYLINDRICAL TUBE USING CAPUTO-FABRIZIO FRACTIONAL DERIVATIVES.....	334
34. NATURAL CONVECTION HEAT TRANSFER IN HYBRID RECTANGULAR-TRIANGULAR ENCLOSURES WITH ISOTHERMAL VERTICAL HEATING	346

INVESTIGATION ON GENERALISED TRAPEZOIDAL DIFFERENCING TIME-MARCHING SCHEME FOR MODELLING OF ACOUSTICAL WAVE

Priscilla Francesca¹
Wah Yen Tey^{1,2*}
Lit Ken Tan^{2,3}

¹Department of Mechanical Engineering, Faculty of Engineering, UCSI University, Kuala Lumpur, Malaysia

²Takasago i-Kohza, Malaysia-Japan International Institute of Technology, Universiti Teknologi Malaysia, Kuala Lumpur, Malaysia

³International Center, Tokyo City University, 1-28-1, Tamazutsumi, Setagaya-ku, Tokyo, 158-8557, Japan*

*Co-author's Email: teywy@ucsiuniversity.edu.my

Abstract: Generalised Trapezoidal Differencing Time-Marching (GTDTM) Scheme has been developed in this paper to solve non-linear wave equation for acoustical modelling. The method is indeed a combination between Euler explicit method and Euler implicit method. To investigate the computational accuracy for GTDTM, the explicit factor is reduced from 1 (fully explicit) to 0 (fully implicit). The results show that the lower the explicit factor, the propagated wave will be damped with a faster speed. As a remedy, the Courant number is reduced when the explicit factor is small. In fact, Euler implicit scheme is not an effective approach to solve the hyperbolic wave equation. However, the minimum Courant number required due to different explicit factor has been computed too for future implementation of GTDTM.

Keywords: Wave equation, Explicit time-marching, Implicit time-marching, First order time-marching, Acoustic wave

Introduction

Wave equation has been widely applied in engineering physics such as the investigation on tidal wave [1], modelling on ultrasonic pretreatment [2], vibration of quantum particles [3], simulation of mountain wave [4] and formation of Keller-Miksis equation which governs the high-speed bubble dynamics [5,6]. Since wave equation appears as hyperbolic partial differential equation, designing suitable numerical stencils for time-marching scheme is a critical issue among numerical scientists to ensure computational accuracy.

The most popular method in time discretisation of wave equation is Euler explicit method, in which the spatial derivative of the equation is presumed to hold current time step. This is probably due to its simplicity and excellent accuracy, provided that the grid size is small enough. The improvised versions of explicit method comprise Lax method [7,8], Delfim-Soares explicit method [9] and higher order explicit time-stepping method [10]. They are proven to be efficient in the computation of wave equation and other hyperbolic equations.

Nevertheless, the application of implicit time-stepping method in solving hyperbolic equation is not as favored as explicit scheme. Besides than its complexity, implicit scheme may lead to asymptotically obliterating amplitude [8,11]. To solve the problem, there are basically three types of approaches have been proposed. First method is through combination of the explicit and implicit scheme can be combined, and the example of this is trapezoidal differencing method [8]. Second approach is via the increment of higher order accuracy of Taylor's series expansion, and the examples for this comprise leap-frog method [8], dissipation-adaptive method [11], Lax-Wendroff method [12], high order Runge-Kutta

method, and Malkoti method [13]. The third way is via deploying of correction step. The methods which were developed using this method are MacComack method, Rusanov method and Warming-Kutler-Lomax method, and the details of these methods can be found in the work of Tannehill et al. [8]. Implicit-explicit multistep method is recently proposed by Gao and Mei [14] too, yet their numerical scheme, finite element Galerkin approach is applied instead of finite difference method.

However, for most of these improvised methods, their computational performance is left unexamined via practical numerical modelling of wave equation. In this paper, we will develop a generalized trapezoidal differencing method [8] to solve acoustic wave, inspired by the generalization of discretized heat equation from Reith et al. [15]. The possible way to set right the annihilation of wave will be investigated as well.

Formulation of Wave Equation

Consider a string as illustrated in Fig. 1, the equilibrium of forces in one dimensional domain can be formed from the Newton's second law:

$$F = ma = \rho Va$$

$$\rho \sqrt{\Delta x^2 + \Delta P^2} \frac{\partial^2 P}{\partial t^2} = F(x + \Delta x, t) \sin[\theta(x + \Delta x, t)] - F(x, t) \sin[\theta(x, t)] + F_e \quad (1)$$

where ρ , ΔP , F and F_e is the density of string, vertical displacement of the string, string tension and external force applied on the string respectively. By taking Δx to be infinitely close to zero, Eq. (1) can be simplified as in Eq. (2). Then Eq. (2) is differentiated with first derivative of x to form Eq. (3).

$$\rho \sqrt{\Delta x^2 + \Delta P^2} \frac{\partial^2 P}{\partial t^2} = F(x, t) \cos[\theta(x, t)] + F_e \quad (2)$$

$$\rho \left[\sqrt{1 + \left(\frac{\partial P}{\partial x} \right)^2} \right] \frac{\partial^2 P}{\partial t^2} = \cos[\theta(x, t)] \frac{\partial}{\partial x} [F(x, t)] + F(x, t) \frac{\partial}{\partial x} \{ \cos[\theta(x, t)] \} + \frac{\partial F_e}{\partial x} \quad (3)$$

Now, by applying the trigonometric equation as in Eqs. (4.1) and (4.2) into Eq. (3), the wave equation as in Eq. (5). The physical assumption made during the simplification are: (a) the displacement gradient $\partial P / \partial x$ is equivalent to 1; and (b) in vertical harmonic oscillation, the rate of change of force per unit distance across x -axis will be zero.

$$\cos \theta = \frac{1}{\sqrt{1^2 + \left(\frac{\partial P}{\partial x} \right)^2}} \approx 1 \quad (4.1)$$

$$\sin \theta = \frac{\frac{\partial P}{\partial x}}{\sqrt{1^2 + \left(\frac{\partial P}{\partial x} \right)^2}} \approx \frac{\partial P}{\partial x} \quad (4.2)$$

$$\rho \frac{\partial^2 P}{\partial t^2} = F \frac{\partial^2 P}{\partial x^2} \rightarrow \therefore \frac{\partial^2 P}{\partial t^2} = c^2 \frac{\partial^2 P}{\partial x^2} \Big|_{c = \sqrt{\frac{F}{\rho}}} \quad (5)$$

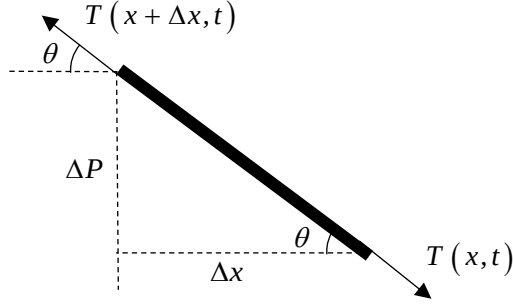


Fig. 1. The propagation of energy via oscillating string

In Eq. (5), c represents the wave speed, which can be further related with the wave frequency and wavelength as shown in Eq. (6). Wave speed for sound is dependent on the transmitting medium and the ambient temperature. For example, the wave of sound moves with the approximate speed of 344 m/s and 1500 m/s in air and water respectively when the ambient temperature is 20°C. The details of the speed of sound can be referred in the work of Müller and Möser [16].

Generalised Trapezoidal Differencing Time-Marching Scheme

The discretized equation of Eq. (5) using trapezoidal differencing time-marching scheme [8] is shown in Eq. (5).

$$\frac{P_i^{n+1} - 2P_i^n + P_i^{n-1}}{\Delta t^2} = c^2 \frac{1}{2} \left(\frac{P_{i+1}^{n+1} - 2P_i^{n+1} + P_{i-1}^{n+1}}{\Delta x^2} + \frac{P_{i+1}^n - 2P_i^n + P_{i-1}^n}{\Delta x^2} \right) \quad (6)$$

Note that for the spatial discretisation, half of the stencils are built based on the future time step, while the other half are holding the current time step. For a generalized equation, Eq. (6) can be further modified to form Eq. (7). Rearrangement of the stencil in Eq. (7) will form Eq. (8).

$$\frac{P_i^{n+1} - 2P_i^n + P_i^{n-1}}{\Delta t^2} = c^2 \left\{ (1 - \beta) \frac{P_{i+1}^{n+1} - 2P_i^{n+1} + P_{i-1}^{n+1}}{\Delta x^2} + \beta \frac{P_{i+1}^n - 2P_i^n + P_{i-1}^n}{\Delta x^2} \right\} \quad (7)$$

$$\begin{aligned} -\alpha(1 - \beta)P_{i+1}^{n+1} + [1 + 2\alpha(1 - \beta)]P_i^{n+1} - \alpha(1 - \beta)P_{i-1}^{n+1} \\ = \alpha\beta P_{i+1}^n - 2(1 - \alpha\beta)P_i^n + \alpha\beta(1 - \beta)P_{i-1}^n \end{aligned} \quad (8)$$

where, $\alpha = c^2 \frac{\Delta t^2}{\Delta x^2}$

In Eq. (8), β (≤ 1) is known as explicit factor, i.e. when β is 0, 1 and 0.5, then Eq. (8) will become fully implicit, fully explicit and trapezoidal differencing method respectively. Eq. (8) can be further re-expressed to yield a matrix as shown in Eq. (9) for the ease of computation.

$$\begin{aligned}
 & \begin{pmatrix} 1+2\alpha(1-\beta) & \alpha(\beta-1) & \cdots & 0 \\ \alpha(\beta-1) & 1+2\alpha(1-\beta) & \ddots & \vdots \\ \vdots & \ddots & \ddots & \alpha(\beta-1) \\ 0 & \cdots & \alpha(\beta-1) & 1+2\alpha(1-\beta) \end{pmatrix} \begin{pmatrix} P_1 \\ P_2 \\ \vdots \\ P_m \end{pmatrix}^{n+1} \\
 &= - \begin{pmatrix} P_1 \\ P_2 \\ \vdots \\ P_m \end{pmatrix}^{n-1} + \begin{pmatrix} \chi_1 \\ 0 \\ \vdots \\ \chi_2 \end{pmatrix} + \begin{pmatrix} 2(1-\alpha\beta) & \alpha\beta & \cdots & 0 \\ \alpha\beta & 2(1-\alpha\beta) & \ddots & \vdots \\ \vdots & \ddots & \ddots & \alpha(\beta-1) \\ 0 & \cdots & \alpha\beta & 2(1-\alpha\beta) \end{pmatrix} \begin{pmatrix} P_1 \\ P_2 \\ \vdots \\ P_m \end{pmatrix}^n \quad (9)
 \end{aligned}$$

where:

$$\chi_1 = \alpha(1-\beta)S + \alpha\beta S$$

$$\chi_2 = \alpha(1-\beta)P_{m-1}^{n-3} + \alpha\beta P_{m-1}^{n-4}$$

The solution of wave equation via Eq. (9) is named as Generalised Trapezoidal Differencing Time-Marching (GTDTM) scheme. The second term of the right-hand side of Eq. (9) is matrix which imposes the boundary condition. The first term and last term in the matrix represents the source of acoustic wave (Dirichlet boundary condition) and the end-wall boundary condition of the problem domain (Neumann boundary condition). In our problem, the Neumann boundary condition set will enforce a non-reflecting wall for wave. The symbol m represents the maximum nodes available in the problem domain, while the mathematical expression of S and α is shown in Eq. (10) and Eq. (11) respectively,

$$S = P_{\max} \sin(2\pi ft) \quad (10)$$

$$\alpha = \left(c \frac{\Delta t}{\Delta x} \right)^2 \quad (11)$$

where f and t is the wave frequency and instantaneous time of the wave propagation. The frequency and $P_{\max}$ set the current work is 20 kHz and $2 \times 10^5 \mu\text{Pa}$ respectively, while speed of sound c is set as 344 m/s. The grid number is 1400. To ensure time marching stability, the Courant-Friedrichs-Lewy condition must be fulfilled [17]. In our simulation the initial Courant number Co applied is 0.5, and its can be expressed mathematically as in Eq. (12). Upon computation of the results, the possible damped pressure will be translated as attenuation of sound pressure level L_p , which can be mathematically represented as in Eq. (13).

$$Co = \sqrt{\alpha} = c \frac{\Delta t}{\Delta x} \quad (12)$$

$$L_p = 20 \log \left(\frac{P^n}{P_{ref}} \right) \Bigg|_{P_{ref}=20 \mu\text{Pa}} \quad (13)$$

Results and Discussion

We found that with the increasing value of explicit factor, the damping of the sinusoidal wave can be apparently observed, as shown in Fig. 2. At $\beta = 1$, which lead to fully explicit scheme, shows an oscillation of wave without “damping” effect, while at $\beta = 0$, the solution is fully implicit and the wave propagation will be silenced soon after it travels away from the source.

This implies that the increment of weightage for the component of implicit time-marching in Generalised Trapezoidal Differencing Time-Marching Scheme will greatly deteriorate the results. This is in accordance with the findings by Tannehill et al. [8] and Bathe and Baig [18] that the implicit scheme is the source of numerical instability during the non-linear solution of time.

However, the change of β will not affect the wavelength, which is maintained as $0.0172\mu\text{m}$ all the time despite the trimming of wave amplitude.

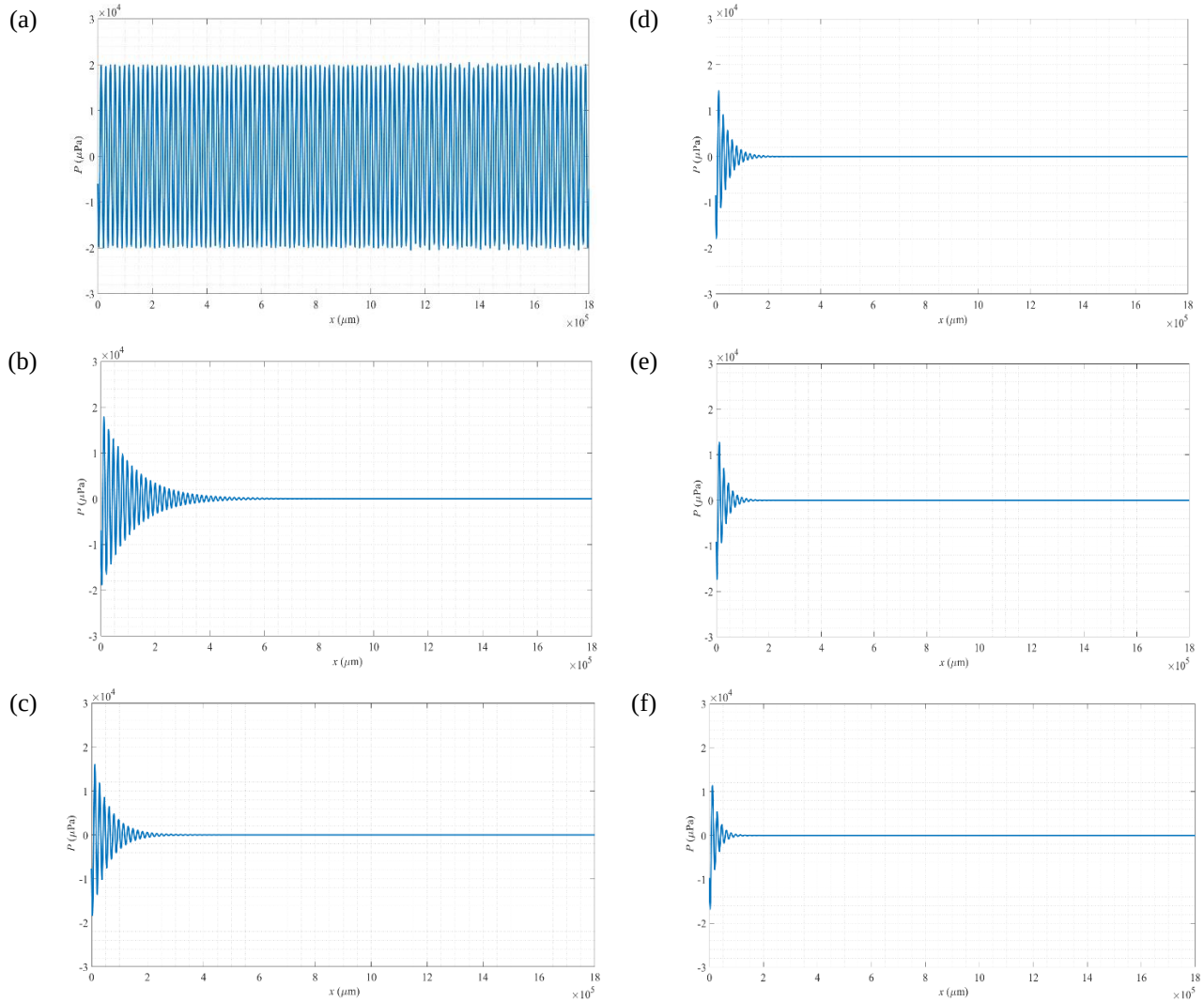


Fig. 2. Wave propagation when the explicit factor β is: (a) 1.0; (b) 0.8; (c) 0.6; (d) 0.4; (e) 0.2; (f) 0.0 when $Co = 0.5$

The attenuation of wave can be further quantified via the records on the maximum amplitude for every cycle of wave and the resultant sound pressure level reduction, as shown in Fig. 3(a) and 3(b) respectively. The maximum amplitude and resultant sound pressure level is reducing in exponential way and negative linear way respectively. Although the blue line as in Fig. 3 (a) which represents the maximum pressure for $\beta = 1$ appears to be unstable, yet there is no annihilation observed. The see-sawing of the results is possibly caused by the insufficient spatial grids.

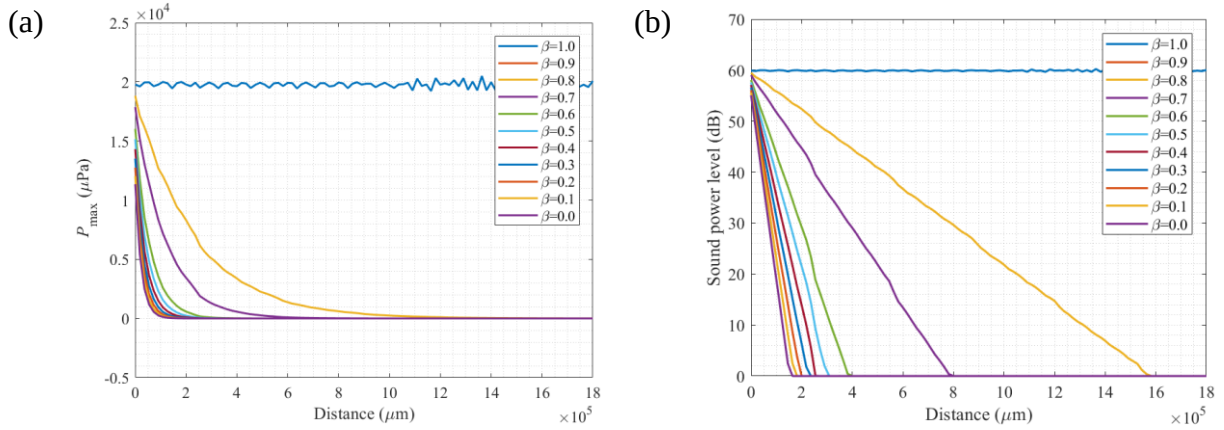


Fig. 3. Attenuation of wave propagation in terms of (a) maximum pressure per cycle; and (b) sound pressure level

To improve the GTDTM scheme, the remedy is endeavored by reducing the Courant number. Since the maximum annihilation happens at $\beta = 0$, the effect of Co to the results accuracy is further as investigated. The coefficient of determination R^2 and average error E for comparison between the computed pressure and the actual pressure can be analyzed using Eq. (14) and (15) respectively. The actual pressure is calculated by taking $\beta = 1$ with small Co of 0.05.

$$R^2 = 1 - \frac{\sum_{i=1}^n \left\{ (P_i)_{\text{computed}} - (P_i)_{\text{exact}} \right\}^2}{\sum_{i=1}^n \left\{ (P_i)_{\text{computed}} - P_{\text{average}} \right\}^2} \quad (14)$$

$$\text{Average Error} = \frac{1}{n} \sum_{i=1}^n \left| (P_i)_{\text{computed}} - (P_i)_{\text{exact}} \right| \quad (15)$$

Fig. 4(a) and (b) illustrates the effect of Co to the propagated wave pattern and the averaged error. For examination purpose, the spatial domain is zoomed into the distance only up to $6.16 \times 10^4 \mu\text{m}$, the total time is set only to $150 \mu\text{s}$ while the grid size is reduced only to 160. The Courant number is reduced until an undamped wave is obtained. Co is gradually decreased until the convergence is reached (i.e. the results will not change any more by increment of Courant number). From Fig. 4(a) and (b), it is clearly shown that by decreasing the Co , the averaged error will be reduced. However, the significant reduction of Co is required to obtain a result with R^2 approaches to 1.

In order to ensure the numerical accuracy for every β , Co is gradually reduced upon convergence. The maximum Co required and the CPU time required to a stable and undamped computation has been computed as in Table 1. The results are obtained with processor of Intel® Core™ i7-8700K CPU @ 3.40 GHz with RAM 32.0 GB. It can be clearly observed that the minimum Courant number required is very high while the CPU time is very expensive when $\beta = 0$. Moreover, the algorithm needs to store the field memory up to five time-steps due to the necessity of boundary condition.

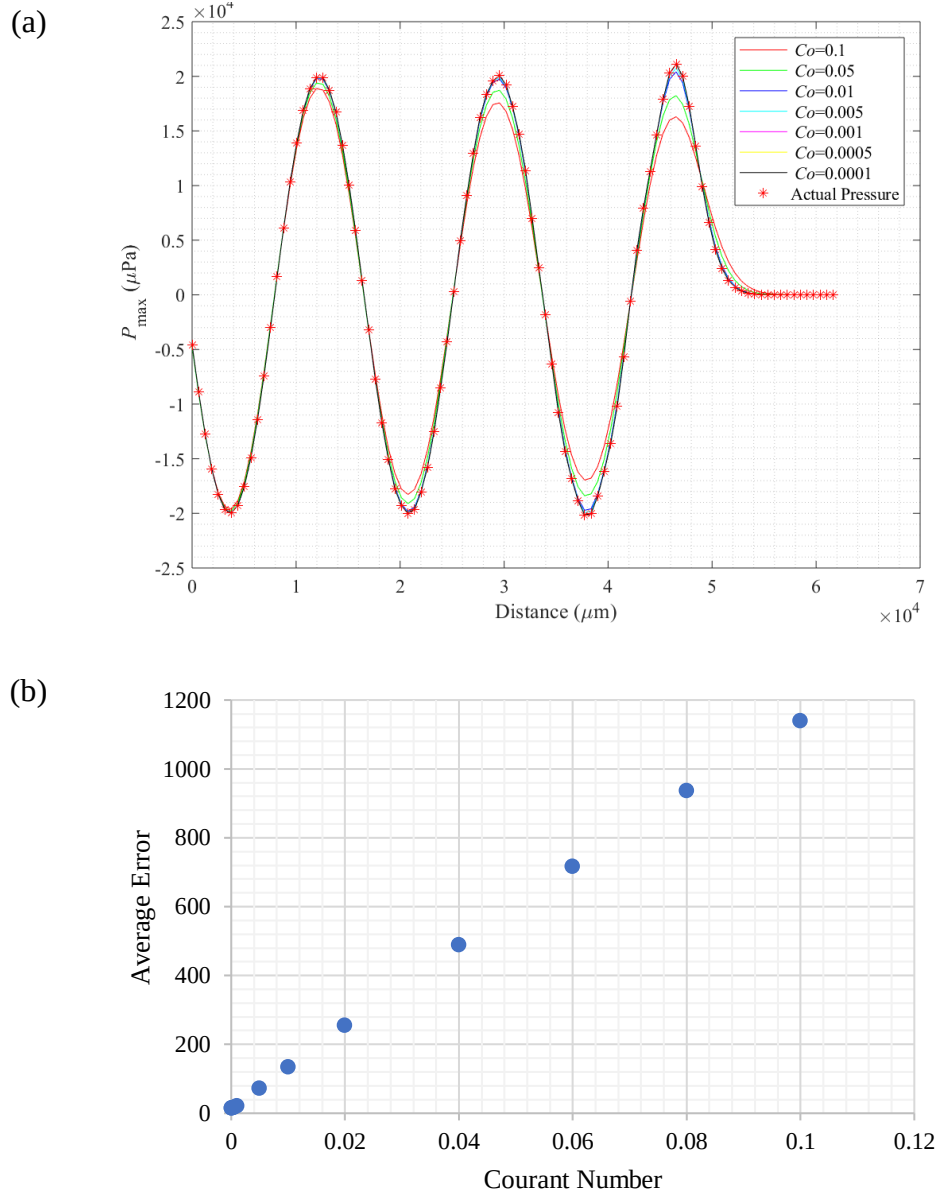


Fig. 4. Effect of reduction in Courant number to: (a) propagated wave pattern; and (b) average error

Indeed, despite implicit method is unconditionally stable, yet during the solution of hyperbolic equation, it does not give much advantages compared with fully explicit method. In fact, the time step required for implicit solver is much higher than the explicit solver. Moreover, the implementation of the boundary condition is difficult when Co has been decreased to deal with the implicit damping. This is possibly the main reason why in most of the available literature working on the improvement of time integration method for linear wave equation, modification based on explicit methods [9,10,19-22] are more preferable. Some modifications on implicit method [11,18,23] have been reported too, yet with the cost of computational complexity.

Table 1. Maximum Courant number for stable computation and CPU time required for different β

β	Co	R^2	CPU Time (seconds)
0.0	0.00005	1.00	10701.837
0.1	0.00080	1.00	583.662
0.2	0.00100	1.00	473.085
0.3	0.00200	1.00	240.449
0.4	0.00400	1.00	117.598
0.5	0.00500	1.00	95.535
0.6	0.00600	1.00	79.622
0.7	0.00800	1.00	59.667
0.8	0.01000	1.00	48.148
0.9	0.02000	1.00	23.800
1.0	0.05000	1.00	9.358

The value of Co in Table 1 would ensure an undamped wave propagation despite a coarse grid. However, with the increment of grid resolution, the Co can be increased, especially if the value of β is 1.0. Therefore, in solving GTDTM, Co independence study is required too besides than mesh independence study, so that the value of Co in which the result will not change anymore regardless of smaller value of Co can be identified.

Concluding Remarks

Generalised Trapezoidal Differencing Time-Marching (GTDTM) Scheme has been developed to solve the hyperbolic wave equation. With the decreasing value of explicit factor β , the resultant waves are oscillating with a damped manner, and upon some distance, the wave will be fully annihilated. To rectify the dissipative issue, the Courant number is reduced, and the maximum Courant number required for each β has been computed for the ease of implementation for GTDTM.

Acknowledgement

The authors would like to thank the support from Takasago Research Fund (Vote No: 4B314) from Malaysia-Japan International Institute of Technology, Universiti Teknologi Malaysia and the Pioneer Scientist Incentive Fund (PSIF) with vote number Proj-In-FETBE-038.

References

- [1] Khan, N., Kalair, A., Abas, N. and Haider, A. "Review of ocean tidal, wave and thermal energy technologies." *Renewable and Sustainable Energy Reviews* 72 (2017): 590-604.
- [2] Lais, H., Lowe, P. S., Gan, T. H. and Wrobel, L. C. "Numerical modelling of acoustic pressure fields to optimize the ultrasonic cleaning technique for cylinders." *Ultrasonics Sonochemistry* 45 (2018): 7-16
- [3] Umul, Y. Z. "Relativistic wave equation for a quantum particle with potential energy." *Optik* 180 (2019): 220-225.
- [4] Keller, M., Joseph, B. and Miksis. "Bubble oscillations of large amplitude." *Journal of the Acoustical Society of America* 68 (2): 628.
- [5] Tey, W. Y., Hou, J. T. T. and Kang H. S. "Numerical modelling of convective wave using fractional-step method." *International Journal of Engineering & Technology* 7 (2018): 99-103.

- [6] Yasui, K. (2018). *Bubble Dynamics - Acoustic Cavitation and Bubble Dynamics: Theoretical and Computational Chemistry*, page 37–97.
- [7] Lax, P. D. “Weak solutions of nonlinear hyperbolic equations and their numerical computation.” *Communications on Pure and Applied Mathematics* 7 (1954): 159-193.
- [8] Tannehill, J. C., Anderson, D. A. and Pletcher, R. H. (1997) *Computational Fluid Mechanics and Heat Transfer*, 2nd ed. United States of America: Taylor & Francis, page 112.
- [9] Delfim, S. “A novel family of explicit time marching techniques for structural dynamics and wave propagation models.” *Computer Methods in Applied Mechanics and Engineering* 311 (2016): 838-855.
- [10] Marcus, J. G. and Teodora, M. “High-order explicit local time-stepping methods for damped wave equations.” *Journal of Computational and Applied Mathematics* 239 (2013): 270-289.
- [11] Delfim, S. “An implicit family of time marching procedures with adaptive dissipation control.” *Applied Mathematical Modelling* 40 (2016): 3325-3341.
- [12] Lax, P. D. and Wendroff B. “Systems of conservation laws.” *Communications on Pure and Applied Mathematics* 13 (1960): 217–237.
- [13] Malkoti, A., Nimisha, V. and Ram, K. T. “A highly efficient implicit finite difference scheme for acoustic wave propagation.” *Journal of Applied Geophysics* 161 (2019): 204-215.
- [14] Gao Y. and Mei, L. “Implicit–explicit multistep methods for general two-dimensional nonlinear Schrödinger equations.” *Applied Numerical Mathematics* 109 (2016): 41-60.
- [15] Rieth, Á., Kovács, R. and Fülöp, T. “Implicit numerical schemes for generalized heat conduction equations.” *International Journal of Heat and Mass Transfer* 126 (2018): 1177-1182.
- [16] Müller, G. and Möser, M. (2013) *Handbook of Engineering Acoustics*, Springer Link.
- [17] Versteeg, H.K. and Malalasekera, W. (2007). *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*. Pearson Prentice Hall.
- [18] Bathe, K. J. and Baig, M. M. I. “On a composite implicit time integration procedure for nonlinear dynamics.” *Computers and Structures* 83 (2005): 2513-2524.
- [19] Hahn GD. “A modified Euler method for dynamic analysis.” *International Journal for Numerical Methods in Engineering* 32 (1991): 943-955.
- [20] Chung, J. and Lee, J.M. “A new family of explicit time integration methods for linear and non-linear structural dynamics.” *International Journal for Numerical Methods in Engineering* 37 (1994): 3961-3976.
- [21] Souza L. A., Carrer, J. A. M. and Martins, C. J. “A fourth order finite difference method applied to elastodynamics: finite element and boundary element formulations.” *Structural Engineering and Mechanics* 17 (2004): 735-749.
- [22] Delfim, S. and Francisco, C. A. “An explicit direct FEM–BEM coupling procedure for nonlinear dynamics.” *Engineering Analysis with Boundary Elements* 103 (2019): 94-100.
- [23] Chung, J. and Hulbert, G. M. “A time integration algorithm for structural dynamics with improved numerical dissipation: The generalized- α method.” *Journal of Applied Mechanics* 30 (1993): 371-375.

INVESTIGATION OF THE EFFECT OF AWNING USING SUNLIGHT SENSOR TO REDUCE COOLING LOAD IN THE ROOM

M.W. Muhieldeen¹

Y.R. Lim²

Sunil Govinda³

¹Faculty of Engineering, Technology and Built Environment, Mechanical Engineering, UCSI University, Cheras, Malaysia. (Email: al-gailani@ucsiuniversity.edu.my)

²Faculty of Engineering, Technology and Built Environment, Mechanical Engineering, UCSI University, Cheras, Malaysia. (Email: limmyyingran@gmail.com)

³Faculty of Engineering, Technology and Built Environment, Electrical and Electronics Engineering, UCSI University, Cheras, Malaysia.

Abstract: *In this paper, the potential of a motorized awning, coupled with sunlight sensor, attached to the window, in reducing the cooling load of a wooden room is investigated experimentally. The wooden room was built with a scaled down dimension according to a classroom in UCSI College that has a window facing approximately true East, hence fully exposed to the morning sun. A Light Dependent Resistor (LDR) was installed at the bottom of the window to detect the presence of sunlight hitting the window. The LDR then gives a signal to the microcontroller Arduino Uno 3 which controls the servo motor to lift or lower down the awning to block sunlight. The constructed system is then able to change the angle of awning periodically to provide constant shading to the window. A manual mode was also constructed to allow users to override and control the awning with satisfaction and flexibility. An experiment was conducted on the wooden room on both cases with and without awning, from 9am to 3pm daily. The surrounding temperature measurements were taken with an anemometer while surface temperatures were taken with an Infrared thermometer. Results showed that a potential reduction of average temperature and cooling load of 0.47 °C and 1.86 kW for the room with awning installed. Moreover, the projected monthly operation savings was RM 114 for 120 operation hours.*

Keywords: *Motorized Awning; Sun Light Sensor; Saving Energy; Cooling Load*

Introduction

Many researchers have done multiple different studies and development of external awnings and high-performance windows, along with their effect on cooling load, including the works of Juanico [1], Luo et al. [2], Muhieldeen, et al., [3], Hammad & Abu-Hijleh [4], Lee, et al., [5], Arce et al. [6], Farrington et al. [7], Samaan et al. [8], Winther et al., [9], Ghodsieh & Nadooshan [10], Muhieldeen and Kuang [11], Muhieldeen et al., [12] and Salman et al., [13].

However, the effectiveness of the awning in reducing direct insulation was not maximized due to it being fixed. Wong and Istiadji [14] investigated the effects of external awnings on temperature as well. The investigation demonstrated a reduction of indoor temperature of 0.6 – 0.88°C with a horizontal awning implemented. Al-Tamimi & Syed Fadzil [15] carried out a study of the potential of awnings in the reduction of temperature in high-rise buildings in Penang, Malaysia. Among the horizontal, vertical and egg-crate awnings, the findings showed that in unventilated rooms the egg-crate awning had a significant impact on the indoor temperature, with a maximum reduction of 5.1 °C. Wong and Li [16] investigated

the effectiveness of awnings on the consumption of cooling energy for windows on the East and West façade in Singapore. A 30 cm horizontal awning was applied to the window. The results showed a 2.6 – 3.2 % of savings on cooling load. The savings increased to 5.9 – 7.0 % when the depth of the awning was 60 cm.

Arifin & Denan [17] analysed the relationship of indoor air temperature with various external awnings in Malaysia. The awnings included in the analysis were vertical awning, horizontal awning and egg-crate awning. The location of the analysis was set at Klang Valley, Kuala Lumpur and the duration were seven days, from 13 to 20 February 2014, 1:00 pm. The analysis returned a positive result in reducing room temperature with awnings installed, with egg-crate awning achieving the lowest average indoor temperature, at 30°C. Rabczak et al., [18] investigated the influence of awning on cooling energy demand of a building in Lublin Providence, Poland. The results demonstrated an approximate of 4% of reduction of annual cooling demand when simple blinds are attached on the windows at an angle of 45°.

Gomez-Munoz & Porta-Gandara [19] proposed a mathematical model to analyze the relationship between external awnings and external walls. The setting of the study was in Northwestern Mexico, where it is hot and dry. The authors proposed a shading device design incorporating awnings and external walls. Results displayed that the method is reliable in creating an increased size of shadow on the façade to completely obstruct solar radiation onto the window with optimal parameter relations between the shading devices. With the current setup, in the Summer Solstice, the design achieved a 45% reduction of direct solar insolation. Gusdorf et al., [20] conducted an experiment to investigate the savings of cooling energy that an external awning can save. The awnings were manually operated and angled at 55° and the experiment was conducted from August to September 2010. Results proved a total saving of 3.186 kWh/day and 30% of net reduction of cooling load. Dubois et al., [21] studied the impact of the geometry of seasonal awning on the heating and cooling loads in an office room with 30% glazing-to-wall-ratio. The result returned a decent amount of energy saving of 12 kWh/m²-year can be obtained by employing simple awning during summer. The implementation of light dependent resistors in sensing systems are common and permits light to be a variable for several light-related investigations such as night lamp controllers [22–24]. The objective of this paper is to evaluate the potential reduction of cooling load in a room with and without an automated awning experimentally.

Methodology

Test room

The design of the test room was based on one of the classrooms in UCSI College Kuala Lumpur, with its window facing approximately true East, as shown in Fig 1. The approximate dimension of the designated classroom was (4.8 x 3.2 x 3.2) m and the window was (2.6 x 1.6) m. For this project, the test room was scaled down to (1.2 x 0.8 x 0.8) m with a window of (0.65 x 0.4) m, centered. Acrylic was used as the substitute material for the window as it is widely available in the market and share the opacity of reinforced glass. On the other hand, the dimension of the awning was (0.65 x 0.308) m, and made with a 2 mm thick plywood, so that it is light and would not cause extra load to the servo motor. The awning was attached to the window using door hinges that enables single-degree of freedom movements. The required tools and materials to build the test room, along with the measuring equipment are listed in Table 1

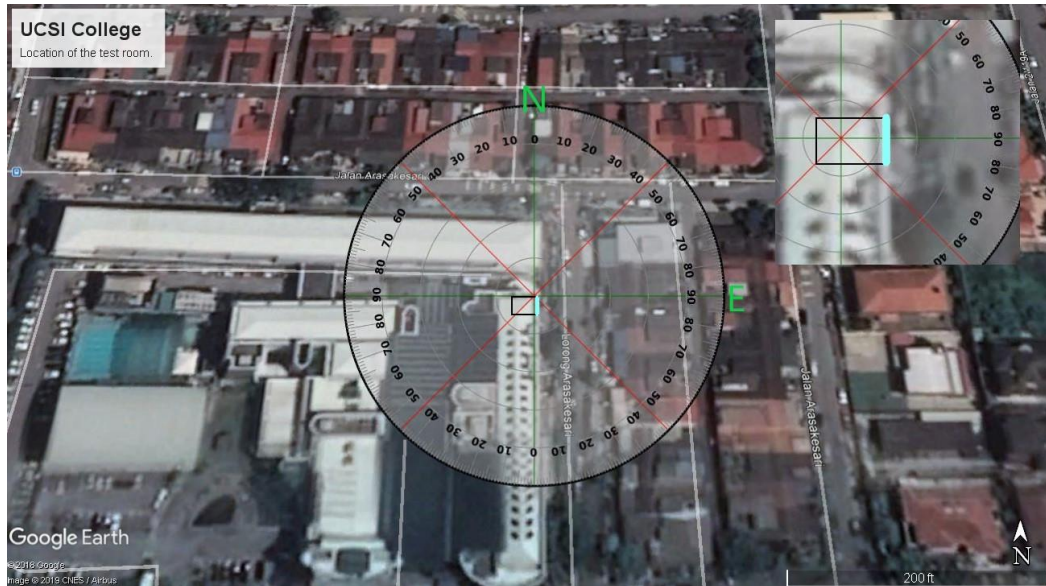


Fig 1 Location of the classroom in UCSI College, Kuala Lumpur.

Table 1: Tools, materials and devices.

Materials and apparatus	Quantity	Function
Plywood (122 x 244 x 0.9) cm	2	Walls of the room
5mm screws	1 packet/50	To hold the room together
Acrylic board (A2 and 1.5 mm thick)	1	The window of the test room
L-shaped metal brackets	33	Used with 5mm screws
Small door hinge	2	To connect awning with window
Electrical Jigsaw	1	To cut the plywood
Power drill	1	To drill holes
Electric screwdriver	1	To install brackets
Sandpaper	1	Smoothen cut edges
F-clamps	4	To hold the plywood during woodworking
Safety gloves	1	Safety precautions
Epoxy	1	To adhere the acrylic to the plywood
IR thermometer	1	To measure surface temperature
Anemometer	2	To measure surrounding temperature

Sensing system

The sensing system of this project was adapted from the studies by Jung et al., [25]. The system aims to detect the presence of sunlight using a simple light dependent resistor and control the awning by lifting or lowering it to a suitable angle, thus providing a constant shade to the window. The basic working principle and interaction of the electrical components are based on it. In addition, the servo motor was installed on top of the test room model to raise and lower the awning, granting a smooth control, while the LDR was installed at the bottom part of the window. Fig 2 shows the schematic diagram of the sensing system. Three major parts of sub-systems are designed to receive different input parameters that can be manipulated. R2 is the potentiometer where the pin of the wiper is connected to A0 pin on the Uno 3 as an analog input to detect varying resistance. The Uno 3 then manipulates this value and sends a PWM to the servo motor through pin D8, thus allowing the user to manually control the awning by

turning the knob of the potentiometer. Referring to Fig 2, PH2 represents the LDR, coupled with a 10k Ohm resistor connected in series, with an analog signal input connected to A1 pin of the Uno 3. As the varying amount of sunlight hits the LDR, it exhibits different resistance and in accordance to the voltage divider rule, the analog signal at A1 determines the amount of voltage drop the circuit receives. On the other hand, S1 represents an “on-off” switch that is connected to Ground on one end, and pin D9 on the other. Such connection of the switch enables the microcontroller to accept a digital signal of 1 or 0, which literally deduces the On or Off state.

Conclusively, the system accepts two modes of inputs, the LDR and the potentiometer, and produces only one output, which is the servo motor. The schematic diagram is then converted to a Printed Circuit Board (PCB) layout using DipTrace and fabricated. The PCB layout is shown in Fig 3.

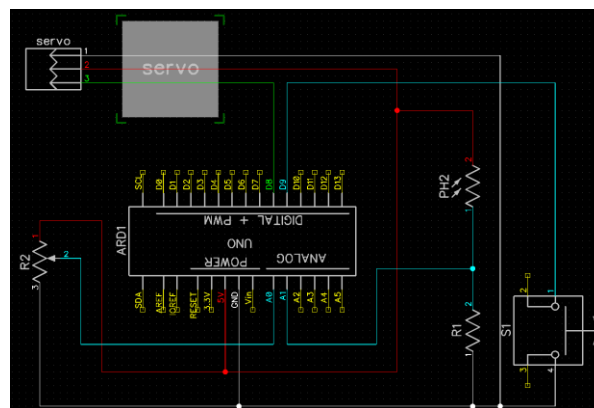


Fig 2 Schematic diagram.

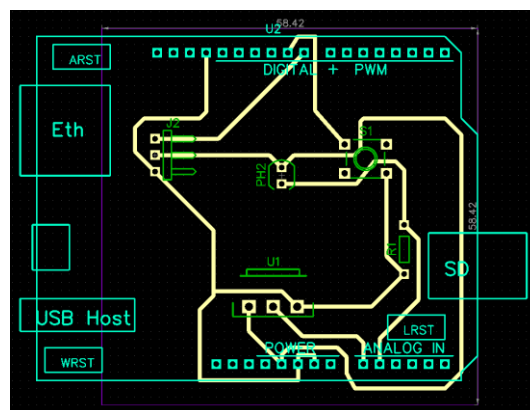


Fig 3 PCB layout.

Experimental setup

Fig 4 shows the final product of the test room, installed with the sensing system and awning. Data on the surface temperature of the awning and window, and the surrounding temperature of inside and outside the test room is collected daily from (9 am to 3 pm) for the whole month of March 2019. Measuring devices were mentioned in Table 1: Tools, materials and devices.

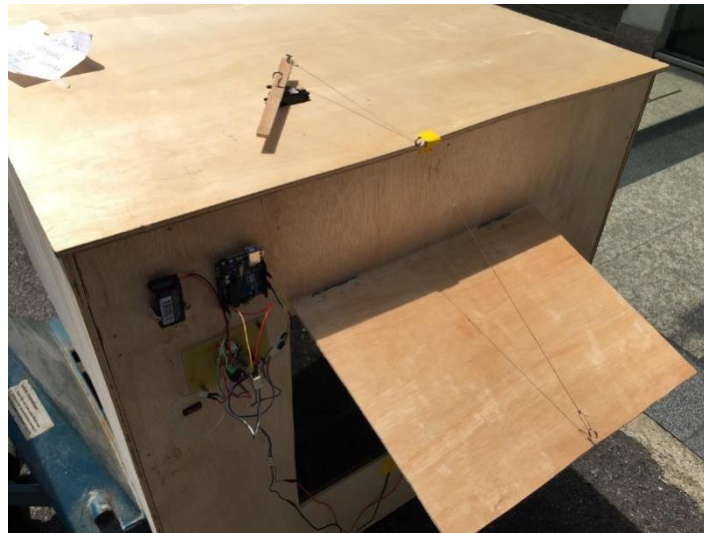


Fig 4 Final product of the test room.

Results

Experimental result

Fig. 5 shows the graph of average outdoor temperature. The temperature increases from 36.6°C at 9am as the sun changes its position and by 1 pm, the temperature rose to 43.1°C, which totals to a growth of 6.5°C. At 9 am, the distance from the sun is further than that at 1pm, which results in the sun rays or solar radiation to travel a further distance through the atmosphere, before hitting the test room model [26]. Therefore, occurs the loss of heat and reduction in surrounding temperature at morning hours. It is also noteworthy that the peak of the graph is at 1300 hours (1 pm) as oppose to 12 pm (noon). According to Trenberth et al., [26], higher levels of solar radiation is absorbed by the surface of the Earth, i.e. buildings and the ground as compared to the atmosphere, and the slow absorption process takes about one or two hours. The temperature at noon is higher than that in the morning, but in the afternoon, 1 pm in this study, as the heat is released from the Earth, the heat combined with the solar radiation develops a temperature profile higher than noon, thus explains the peak of this curve. As the day moves into the afternoon, the temperature drops to 41.8°C at 1500 hours (3 pm), hence the decline in the graph, after the peak.

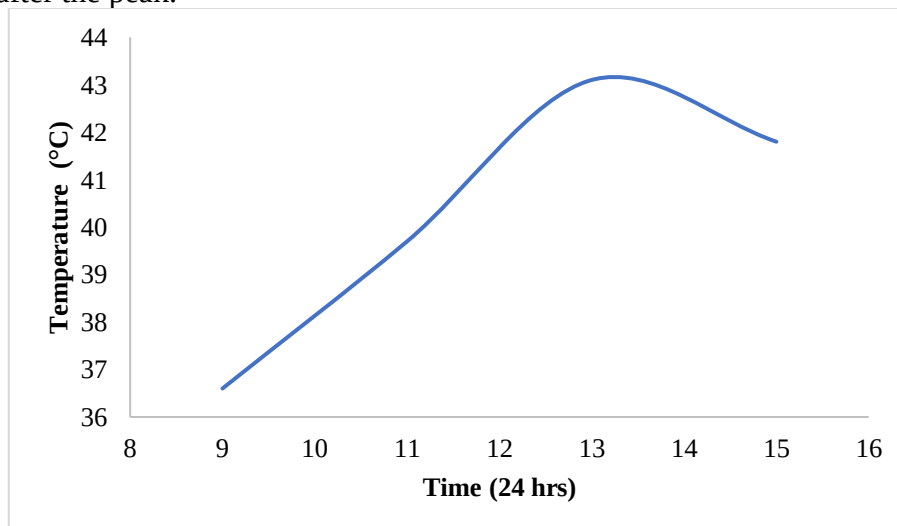


Fig. 5 Average Outdoor Temperature.

It is illustrated in Fig. 6 that the graph of outer surface temperature of the window (acrylic) shows different trends for the case of with and without awning installed. Both cases show a similar decline from 1300 hour to 1500 hour, which was mainly due to the position of the sun that was then above the test room model. This resulted the absence of direct solar insulation to the outer surface of the window, thus the reduction of temperature for both cases. Still, a constant gap was still present between the two cases, with the difference being 1.1°C at 1500 hour. In the morning hours from 0900 hours to about 1100 hours, the window without awning shows an enormous growth, with temperature reaching 43.2°C at 1100 hours, a shocking difference of 2.9°C as compared to the other trend. This would be caused by the absence of awning, whereby sun rays directly hit the surface of the window, on top of the absorbed heat by the window. Looking at the test room with awning however, since the window was constantly shaded, the increment of the surface temperature was swift but constant.

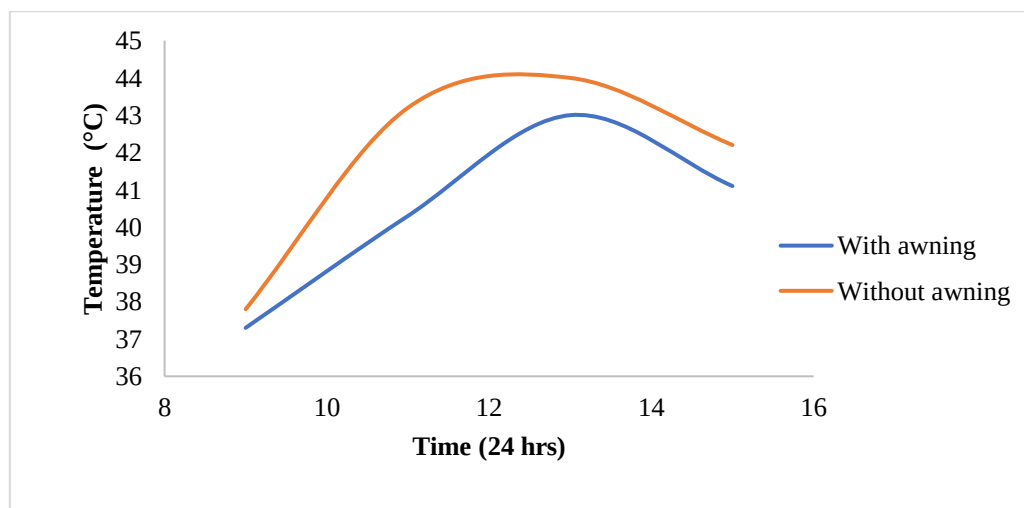


Fig. 6 Outer Surface Temperature of the Window With and Without Awning.

Fig. 7 shows the graph of indoor temperatures of the test room model in both cases, with and without awning. It can be observed that both cases exhibit similar curves and trends which is an accelerated growth from 0900 (9 am) hours to 1300 hours (1 pm) and a decline from therein onwards to the 1500-hour (3 pm) point. The temperature trend of the test room model without awning has been constantly shadowing the temperature profile of the test room model with awning. From 0900 hours to 1200 hours the trends showed an almost constant difference in temperature difference, with 0.5°C at 0900 and 0.4°C at 1100 hours respectively. By 1300 hours, the temperature of the room without awning has peaked at 41.3°C, as compared to the temperature of the room with awning, at 40.7°C. This peak has the most difference, of 0.6°C among the whole trend. Both trends start to converge after 1300 hour and ended with a temperature difference of 0.4°C. To sum up, the implementation of an awning to the test room model demonstrates an effective reduction in temperature along the experiment time period, especially during the morning hours when the sun rays are hitting the window.

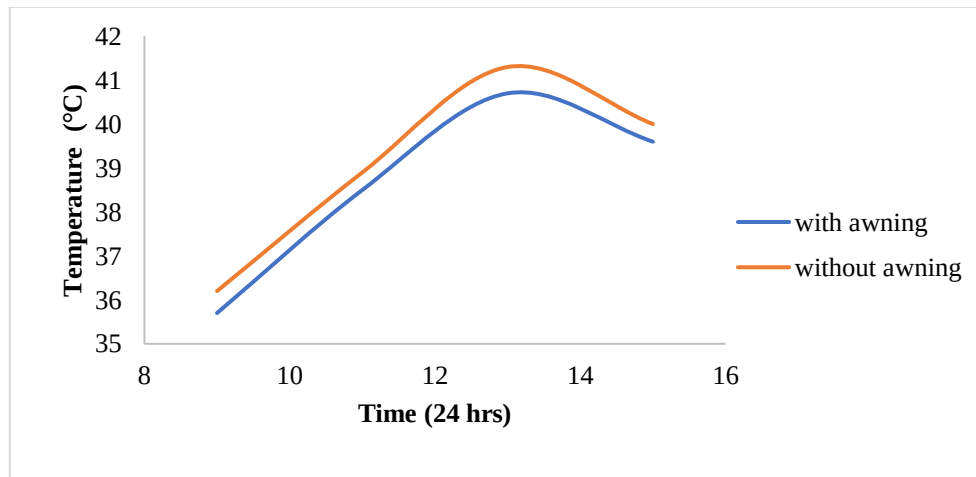


Fig. 7 Indoor Temperature of The Test Room Model With and Without Awning.

Fig. 8 illustrates the combined trends of outdoor temperatures and indoor temperatures, with and without awning. It is easily understood that all three trends had a constant and accelerated increment of temperature during the morning hours, however the indoor temperatures diverge from the outdoor temperature from 1300 hours onwards. It can be deduced that the temperature difference of the test room model with awning installed reached a peak of 1.5°C at 1300 hours, while the case of without awning only contributes to a 0.9°C of reduction at the same hour. This justified that the awning is a great shading device in reducing temperature and the cooling load of a room. In addition, looking at the climbing trend of both indoor temperatures, the indoor temperature trend with installed awning can be constantly below the case of without awning due to it is constantly shaded by the moving awning.

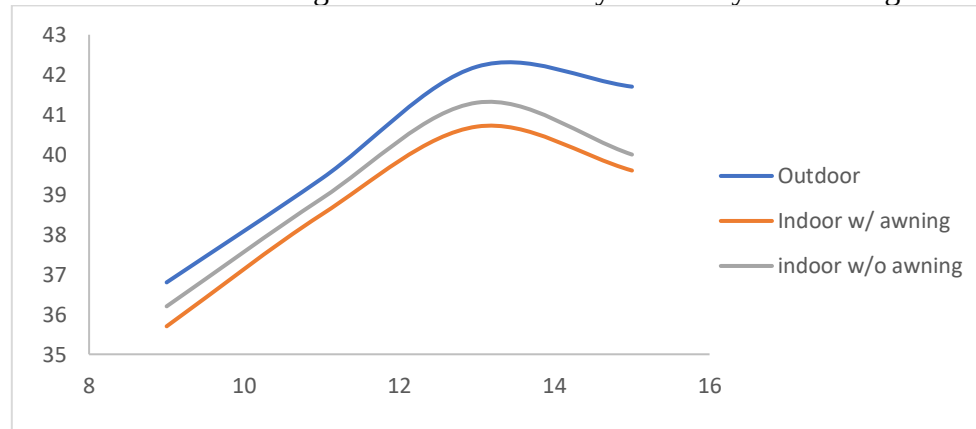


Fig. 8 Outdoor Temperature and Indoor Temperature With Awning.

Cooling load analysis

Table 2 summarizes the cooling load, temperature reduction and peak reductions. It is noticeable that the average temperature of the case with awning is 38.63°C, which is equivalent to a 0.47°C difference as compared to the case of without awning. The table also shows that the peak temperature reached by the test room with awning is 0.6°C lesser than the test room without awning. Looking at the cooling load of both cases, it is clear that the test room with awnings installed produces a 31% reduction of cooling load. Results for the experimental methods are acceptable as they display similar peaks and trends of outdoor and indoor temperature as mentioned by Wong and Li [16] and Arifin and Denan [17].

Table 2: Temperature reduction and cooling load.

Case	Average Temperature (°C)	Average Temperature reduction (°C)	Peak Temperature (°C)	Cooling load (kW)	Peak cooling load (kW)
Without awning	39.1	-	41.3	5.993	5.421
With awning	38.63	0.47	40.7	4.132	4.653

Cost analysis

It is assumed that the room operates for the duration of 6 hours from 9 am to 3 pm daily, 20 working days in a month.

Table 3 summarizes the energy consumption for each case and the percentage of reduction. It shows that the test room model with awning implemented contributes to a 1.86 kW of cooling load reduction, thus 223.32 kWh reduction of monthly energy consumption, which equals to a 31.05 % of reduction per month.

Table 3: Energy consumption.

Case	Cooling load (kW)	Monthly Consumption (kWh)	Energy Monthly Reduction (kWh)	Percentage of Reduction (%)
Without awning	5.993	719.16	-	-
With awning	4.132	495.84	223.32	31.05

On the other hand, Table 4 shows the calculated electricity cost for both cases, referred to the tariff by Tenaga Nasional Berhad Malaysia. The commercial tariff for low voltage usage is 43.5 cents per kWh for the first 200 kWh and 50.9 cents per kWh for 201 kWh onwards [27]. It is clear that the installation of the awning will introduce a 32.6 % of monthly savings due to the reduction of cooling load.

Table 4: Electricity consumption.

Case	Monthly Electricity Cost (RM)	Monthly Savings (RM)	Percentage of Savings (%)
Without awning	351.25	-	-
With awning	237.58	113.67	32.36

Conclusions

An experimental study on the potential of an automated awning coupled with a light sensor to reduce the cooling load inside a test room (1.2 x 0.8 x 0.8) m. The results suggested that a potential reduction of the average temperature and cooling load with the installation of awning to the window of the room was 0.47°C and 1.86 kW. It is therefore justified that an added awning to the window of the room can reduce the cooling load of a room. Moreover, calculation of cost estimated a monthly saving of RM113.67 based on the tariff by Tenaga Nasional Berhad. It is therefore clear to say that the objectives of this research have been achieved.

References

- [1] Juanicó L. E., “A new design of configurable solar awning for managing cooling and heating loads,” *Energy Build.*, vol. 41, no. 12, pp. 1381–1385, 2009.
- [2] Luo S., Li H., Mao Y., and Yang C., “Experimental research on a novel sun shading & solar energy collecting coupling device for inpatient building in hot summer and cold winter climate zone in China,” *Appl. Therm. Eng.*, vol. 142, pp. 89–99, 2018.
- [3] Muhieldeen M.W., Adam N. M., and Salman B. H., “Experimental and numerical studies of reducing cooling load of lecture hall,” *Energy Build.*, vol. 89, pp. 163–169, 2015.
- [4] Hammad F. and Abu-Hijleh B., “The energy savings potential of using dynamic external louvers in an office building,” *Energy Build.*, vol. 42, no. 10, pp. 1888–1895, 2010.
- [5] Lee E. S., DiBartolomeo D. L., and Selkowitz S. E., “Thermal and daylighting performance of an automated venetian blind and lighting system in a full-scale private office,” *Energy Build.*, vol. 29, no. 1, pp. 47–63, 1998.
- [6] Arce P., Castellón C., Castell A., and Cabeza L. F., “Use of microencapsulated PCM in buildings and the effect of adding awnings,” *Energy Build.*, vol. 44, no. 1, pp. 88–93, 2012.
- [7] Farrington R., Rugh J., and Barber G., “Effect of Solar-reflective Glazing on Fuel Economy. Tailpipe Emissions, and Thermal Comfort,” *SAE Tech. Pap.*, 2000.
- [8] Samaan M. M., Farag O., and Khalil M., “Using simulation tools for optimizing cooling loads and daylighting levels in Egyptian campus buildings,” *HBRC J.*, vol. 14, no. 1, pp. 79–92, 2016.
- [9] Winther F. V., Liu M., Heiselberg P., and Jensen R. L., “Experimental and Numerical Analysis of Modelling of Solar Shading,” *Energy Procedia*, vol. 132, pp. 472–477, 2017.
- [10] Ghodsieh S. and Nadooshan A. A., “The Impact of Awning on Consuming for Cooling and Heating Energies at A Residential Construction,” *Indian J. Fundam. Appl. Life Sci.*, vol. 6, pp. 91–101, 2016.
- [11] Muhieldeen M.W., Kuang Y.C. “Saving Energy Costs by Combining Air-Conditioning and Air-Circulation Using CFD to Achieve Thermal Comfort in The Building” *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 58, Issue 1 (2019) 84-99.
- [12] Muhieldeen M.W., Adam N.M., Dato’ Elias Salleh, Tang S. H., HR.Ghezavati. (2009). CFD Simulation on Use of Polyethylene Single Bubble to Reduce Radiant Heat on Lecture Hall International Advanced of Technology Congress (ATCi),PWTC, Malaysia. November 3-5, 2009.
- [13] Salman B.H., Hamzah M.Z., Purbolaksono b. J., Inayat-Hussain J.I., Mohammed H.A., Muhieldeen M.W. Determination of correlation functions of the oxide scale growth and the temperature increase. *Engineering Failure Analysis* 18 (2011) 2260–2271
- [14] Wong N. H. and Istiadji A. D., “Effects of External Shading Devices on Daylighting and Natural Ventilation, Agustinus Djoko Istiadji Department of Building, School of Design and Environment National University of Singapore , 4 Architecture Drive , Singapore 117566,” *Eighth Int. IBPSA Conf. Eindhoven, Netherlands August 11-14, 2003*, pp. 475–482, 2003.
- [15] Al-Tamimi N. A. and Fadzil S. F. S., “The potential of shading devices for temperature reduction in high-rise residential buildings in the tropics,” *Procedia Eng.*, vol. 21, pp. 273–282, 2011.
- [16] Wong N. H. and Li S., “A study of the effectiveness of passive climate control in naturally ventilated residential buildings in Singapore,” *Build. Environ.*, vol. 42, no. 3, pp. 1395–1405, 2007.

- [17] Arifin N. A. and Denan Z., “An Analysis of Indoor Air Temperature and Relative Humidity in Office Room with Various External Shading Devices in Malaysia,” *Procedia - Soc. Behav. Sci.*, vol. 179, pp. 290–296, 2015.
- [18] Rabczak S., Bukowska M., Proszak-Miąsik D., and Nowak K., “Influence of Shading on Cooling Energy Demand,” *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 245, no. 5, 2017.
- [19] Gómez-Muñoz V. M. and Porta-Gándara M. A., “Simplified architectural method for the solar control optimization of awnings and external walls in houses in hot and dry climates,” *Renew. Energy*, vol. 28, no. 1, pp. 111–128, 2003.
- [20] Gusdorf J., Armstrong M., Szadkowski F., and Sager J., “Summer Shading Performance of Awnings at the Canadian Center for Housing Technology” *Natl. Res. Counc. Canada*, 2011.
- [21] Dubois M., “The design of seasonal awnings for low cooling and heating loads in offices,” *Proc. 5th Symp. Build. Phys. Nord. Ctries.*, vol. 2, no. Källblad, pp. 505–512, 1999.
- [22] Govinda K., Mishra U., and Ekansh A., “Night lamp controller,” *Int. J. Sci. Res. Dev. Vol. 2*, pp. 238–241, 2014.
- [23] Maan Salim G., Ismail H., Debnath N., and Nadya A., “Optimal light power consumption using LDR sensor,” *IEEE Int. Symp. Robot. Intell. Sensors*, pp. 144–148, 2015.
- [24] Tharamuttam J. K. and Ng A. K., “Design and Development of an Automatic Solar Tracker,” *Energy Procedia*, vol. 143, pp. 629–634, 2017.
- [25] Jung W., Hong T., Oh J., Kang H., and Lee M., “Development of a prototype for multi-function smart window by integrating photovoltaic blinds and ventilation system,” *Build. Environ.*, vol. 149, no. October 2018, pp. 366–378, 2018.
- [26] Trenberth J., Fasullo K. E., Kiehl J. T., “Earth’s global energy budget,” *Bull. Am. Meteorol. Soc.*, vol. 90(3), pp. 311–324, 2009.
- [27] Tenaga Nasional Berhad Malaysia, “Pricing and Tariff,” 2014. [Online]. Available: <https://www.tnb.com.my/commercial-industrial/pricing-tariffs1/>. [Accessed: 20-Mar-2019].

ENERGY SAVING BY APPLYING DIFFERENT WALL THERMAL INSULATIONS ON A ROOM AT MALAYSIAN INSTITUTION

M.W. Muhieldeen^{1*}
Q.Y. Wong²
Z.A.R. Umni³
Wah Yen Tey⁴

^{1,2,3,4}Faculty of Engineering, Technology and Built Environment, Mechanical Engineering, UCSI University, Cheras, Malaysia.

*Email: al-gailani@ucsiuniversity.edu.my

Abstract: *In a hot and humid climate country, many passive methods have been presented to reduce cooling load of buildings. Nowadays, energy saving has become one of the popular topic over the world. The global warming has caused the high electricity consumption due to the high cooling load demand of the buildings. Where found exterior wall is an important source of heat transfer for the room because wall works as a store for heat. The aim of this study is to evaluate cooling load based on different types of wall thermal insulation. Three different types of material have been selected for wall insulation which are Polyethylene Aluminium Single Bubble (PASB), Expanded Polystyrene (EPS) foam and Rockwool, as well as without insulation. The insulating materials are placed on the front wall of a test room built with dimension 1.2m x 0.8m x 0.8m. Two measuring equipment have been used through this study, Anemometer to measure the air temperature, and Infrared Thermometer to measure the surface temperature of the walls. The data show that rockwool is a better insulation material to reduce the heating load inside the room, the potential temperature reduction inside the room was around 3.85%, where PASB insulation reduced only 2.49% and EPS insulation reduced only 0.89%, which is not much effective as compared to Rockwool. The results show that Rockwool has a potential saving of RM40 per month based on the prototype dimensions without using air condition unit.*

Keywords: *Saving Energy, Thermal Insulation, Cooling Load, Tropical Climate.*

Introduction

Thermal insulation is the reduction of the transfer of heat between objects in thermal contact. In the modern era, thermal insulation in building is a quite important factor to achieve thermal comfort and reduce heat gain or heat loss which can decrease the heating load and cooling load of the buildings [1-5]. The effectiveness of the thermal insulation is evaluated by its resistance value [6]. Venkiteswaran et al. [7] has researched the passive methods for reducing the cooling load which are wall thermal insulation by polystyrene, white painted roof and single low emissivity window glazing, and found that when all three methods applied together could produce the highest reduction in temperature and cooling load.

Many researchers also studied on how to optimize the indoor thermal comfort and energy efficiency by installing the external wall thermal insulation, including Hu et al. [8] studied the use of polystyrene for external brick wall insulation, Ding et al. [9] studied the use of vitrified microsphere inorganic thermal insulation on brick wall, Cao et al. [10] studied the use of composite polystyrene granule thermal insulation mortar in external wall, Noerwasito et al. [11] studied the optimum polystyrene thickness on brick wall, Kumar et al. [12] studied the

improvement of energy efficiency by using various type of glasses and building materials, Halim et al. [13] has studied the use of pitch and wall insulation on air-conditioned rooms, while Pruteanu et al. [14] and Mishra et al. [15] have studied about the expanded and extruded polystyrene rigid foam board on masonry wall. However, this papers [16-17] state that the performance of two most popular energy-efficient retrofitting measures, installation of insulation material and high reflectivity coating on the building wall. The researchers have found that the insulation board with 50mm thickness could give a satisfactory result. However, the thickness with more than 50mm could only get a limit additional profit.

Wang et al. [18] has studied the influence of the position of the thermal insulation in a building on indoor thermal comfort and energy consumption in Chongqing, China. The researchers found that the building with internal insulation have a very poor thermal stability. Yongson et al. [19] found that the position of the air conditioner in the room could affect the thermal comfort also. The direction of the airflow could change the path with the components and occupants of the room. Meaning that occupants should be positioned on a plane so that could be closer to the blower in order to feel cooler.

To prevent the overheating of the building wall in the extremely hot summer, the walls that facing the South were painted with light color to avoid absorption of the solar heat. While for the opposite wall of the building, the walls should be painted with darker color to maximize the heat absorption factor in order to retain heat during the winter season. The design showed the color and the quality of the surface of the building walls are the most important factors [20]. Nyers et al. [21] and Ma et al. [22] show that by increasing the thickness of the polystyrene as a wall thermal insulation layer would increase the material cost but it could reduce the cost of exploitation. Besides that, Bojic et al. [23] shows that a 5cm thick of polystyrene board insulation that facing inside the wall slabs could reduce the annual cooling load by 6.22%. Based on the study conducted by Muhieldeen et al. [24 & 25], the optimum thickness of the Rockwool insulation to reduce the indoor temperature without wasting over insulating cost is between 50 mm and 100 mm.

The solar heat gain through the wall could led to increase of indoor temperature that will lower down the thermal comfort inside the building and subsequently increase the cooling load demand. As a matter of choice, wall thermal insulation could be installed in the building in order to reduce the heat transfer rate from outdoor to indoor through the wall. This means that wall insulated buildings could save electricity bills effectively. The main objective of this study is to evaluate cooling load reduction based on different types of wall thermal insulation.

R-value

Wall is one of the important parts of a building as it occupied the largest area of the building envelope. Therefore, the thermal insulation value of the wall is crucial to have thermal comfort indoor environment and energy saving. Insulation performance is measured in R-values, which quantify the thermal resistance of a building. The greater the R-values, the higher the insulating performance. The R-value of the thermal insulation is highly depending on the type of insulation, thickness, and thermal conductivity. As shown in Figure 1, bulk insulation reduce the heat transmitted to the room by trapping dry air in bulky materials. Large amounts of air could be trapped in the medium and reduce the ability for heat to be transferred by conduction efficiently due to the poor thermal conductivity of air. The small pockets of trapped air could reduce the ability to transfer heat by convection as compared to rather than a large and contiguous volume of air.

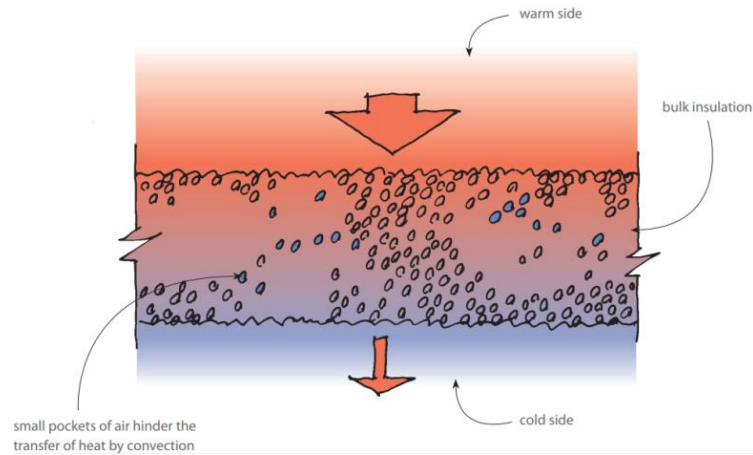


Fig. 1. Mechanism of heat transfer through insulation layer.

As shown in the material properties, Table 3, Rockwool has the highest thermal conductivity among the insulations however it could reduce the indoor air temperature the most. This can be explained by using the thermal resistance equation below:

$$R = \frac{L}{k} \quad (1)$$

Where R is the thermal resistance of the insulation ($m^2 \cdot K/W$), L is the thickness of the insulation (m), k is the thermal conductivity of the insulation ($W/m \cdot K$).

The R-value was calculated and tabulated in Table 1. The thermal resistance value of the system or material is not only defined by its thermal conductivity, it could be varying by its thickness and thermal conductivity as well. The thermal resistance is directly proportional to the thickness of the insulation and inversely proportional to the thermal conductivity of the insulation. Therefore, Rockwool has the highest R-value and higher performance in energy saving.

Table 1: Thermal resistance of each insulation

Insulation	Thickness (m)	Thermal conductivity ($W/m \cdot K$)	R-value ($m^2 \cdot K/W$)
PASB	0.005	0.032	0.156
EPS	0.023	0.033	0.697
Rockwool	0.052	0.044	1.182

Methodology

Test Model

One of the room at North Wing UCSI University, Malaysia was selected to be the test room. A wooden room is fabricated based on the laundry room in the North Wing UCSI University campus as shown in Figure 2 and 3.



Fig. 2 Test room



Fig. 3 UCSI laundry room

The dimension of the wooden room was 1.2m x 0.8m x 0.8m which is scaled down by a ratio 0.4 to the actual size of the laundry room. The material of the wooden room is made up of plywood, 9mm thick with a thermal conductivity of 0.13W/m.K with density 620 kg/m³. The properties of the plywood are shown in the Table 2. The wooden room has 5 sides which are front, back, left, right and top. In order to allow some air circulations inside the room, the bottom of the wooden room was left uncovered and placed on stands while conducting the experiment. The front wall surface of the wooden room which is exposed to the sun the most will be applied with a layer of thermal insulation during the experiment.

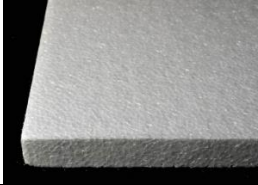
Table 2: Plywood specifications

Property	Value	Unit
Density	620	kg/m ³
Thermal Conductivity	0.13	W/m. K
Thickness	9	mm
Shear Modulus	6.96 - 8.55	GPa
Yield Strength	31.0 – 41.0	MPa
Poisson Ratio	0.3	-

Types of insulating materials

Three types of different thermal insulation were selected for conducting the experiment: Polyethylene Aluminium Single Bubble (PASB), Rockwool and Expanded Polystyrene (EPS) board. The properties of the thermal insulations are listed in Table 3.

Table 3: Properties of different types of insulations

Insulation	Types of insulation	Thickness (mm)	Thermal conductivity (W/m.K)	Density (kg/m ³)
PASB 	Reflective insulation	5	0.032	85
Rockwool 	Bulk insulation	52	0.044	90
EPS board 	Rigid foam insulation	23	0.033	15

Devices

Figure 4 and 5 show the measuring equipment used in the project which are infrared thermometer and anemometer. Infrared thermometer was used to measure the surface temperature of the external wall and internal wall that applied with the insulation layer. While anemometer was used to measure the indoor temperature and surrounding temperature. The tools and devices used in this experiment were listed in Table 4.



Fig. 4 Infrared thermometer



Fig. 5 Anemometer

Table 4: Tools

Item	No.	Function
Infrared thermometer	1	To measure wall surface temperature
Anemometer	2	To measure the air speed and air temperature inside and outside the room
Camera	1	To take photo of the progress
Screw	80	To build the wooden room
Screw driver	1	
Nail	4	To install the insulation
Insulation material	1	PASB
	1	Rockwool
	1	EPS board

Experimental setup

Firstly, the wooden room built was placed in the open area in the North Wing UCSI University campus as shown in Figure 6. The front wall surface of the wooden room was made sure to place under condition of fully exposed to the sunlight during the entire experiment. After that, the front wall of the test room is applied with the PASB insulation as shown in Figure 7. The outdoor and indoor temperature of the test room are measured by anemometer. While the surface temperature of the wall and insulation layer were measured by infrared thermometer from 10am to 4pm during the peak hours for one month. There were three different types of thermal insulation proposed in this experiment in order to evaluate their effect and performance for reducing the heat penetrating into the room. Once the measurement was done for the first thermal insulation, it will be removed and placed with the second type of thermal insulation material and same goes for the third thermal insulation material as shown in Figure 8 and 9.



Fig. 6 Completed wooden room



Fig. 7 Test room insulated with PASB



Fig. 8 Test room insulated with EPS board



Fig. 9 Test room insulated with Rockwool

Results and Discussion

Experimental result

This experiment has been done to study the effect of different types of wall thermal insulation in reducing temperature of buildings. For conducting the experiment, a wooden room was built as prototype model which was related to the size of the UCSI laundry room at North Wing. The work involved were measuring data for three different types of wall insulation and without insulation as well. Measurements for air temperature and walls surface temperature have been done for a period of one month in a sunny condition.

Figure 10 shows the average results between the outer air temperature and the inner air room temperature of the test room that included applying the PASB insulation with a thickness of 5mm on the front wall of the room. As the highest outdoor temperature was at 2 pm reaching to an approximate temperature of 35.03°C while the inner room temperature after applying PASB at the same timing was close to 33.5°C which shows a 0.3°C lower as compared to the inner room temperature without any insulation which was 33.8°C.

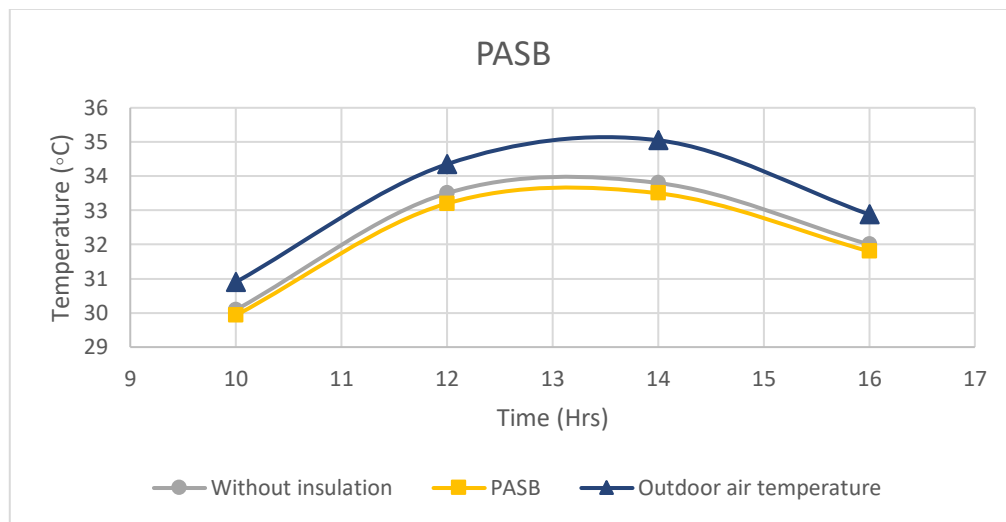


Fig. 10 Temperature Distribution Inside and Outside the Room with PASB Insulation

By referring to the Figure 11, where shows the average results between the outer air temperature and the inner air room temperature after applying EPS insulation with a thickness of 23mm on the front wall of the prototype room. As the highest outer temperature was at 2 pm reaching to an approximate temperature of 35.03°C while the inner room temperature after applying EPS insulation at the same timing was 32.96°C which shows a 0.84°C lower as compared to the inner room temperature without any insulation which was 33.8°C.

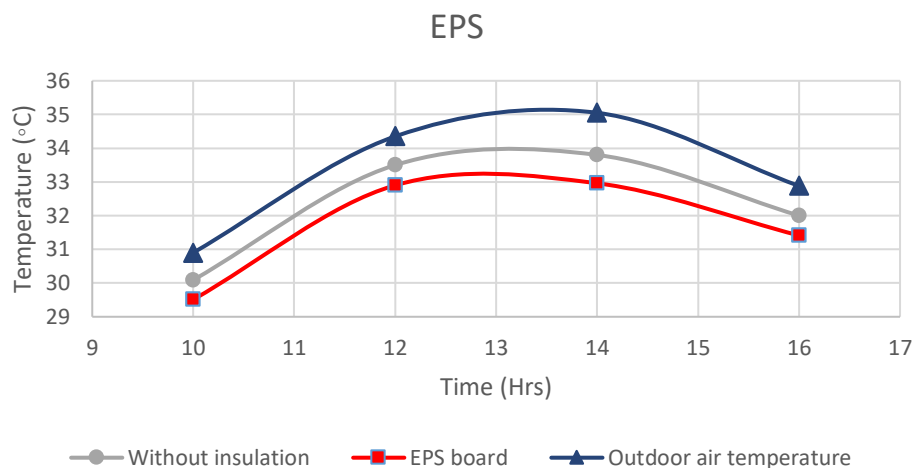


Fig. 11 Temperature Distribution Inside and Outside the Room with EPS Insulation

Figure 12 shows the average results between the outer air temperature and the inner air room temperature after applying Rockwool insulation with a thickness of 52mm on the front wall of the prototype room. As shown in the figure the highest outer temperature was 2 pm reaching to an approximate temperature of 35.03°C while the inner room temperature after applying Rockwool insulation at the same timing was 32.5°C which shows a 1.3°C lower as compared to the inner room temperature without any insulation which was 33.8 °C.

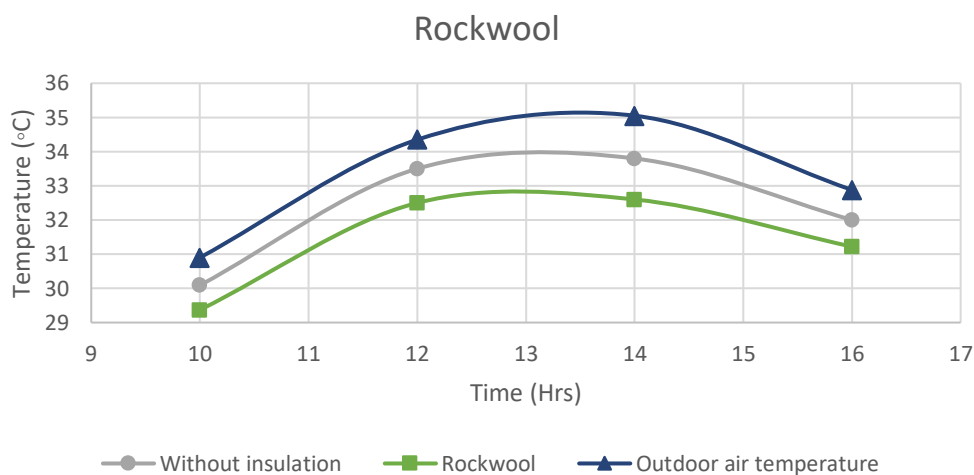


Fig.12 Temperature Distribution Inside and Outside the Room with Rockwool Insulation

Figure 13 shows that Rockwool insulation with thickness of 52mm has the best reduction in temperature as compared to the PASB and EPS insulation. During the highest outdoor temperature which was 2pm, Rockwool with thickness of 52mm could reduce the indoor temperature the most which was 1.3°C, following by EPS with thickness of 23mm which could reduce the indoor temperature by 0.84°C, while PASB with thickness of 5mm does not show significant indoor temperature reduction which was only 0.3°C.

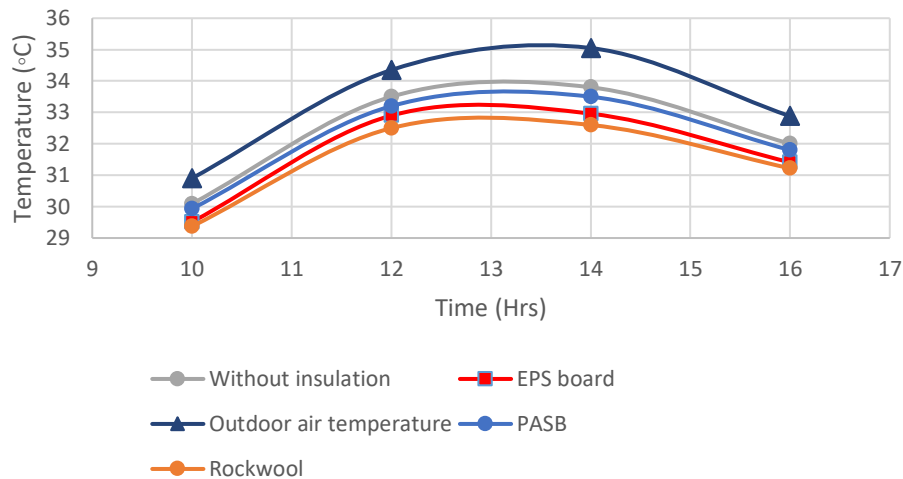


Fig.13: Temperature Distribution Inside and Outside the Room with All Insulations

Power consumption analysis

Table 5 shows the monthly power consumption reduction for each insulation applied. The power consumption was calculated by assuming the air condition units operate for 10 hours a day and there is 30 days in a month. The results show that the application of Rockwool insulation gave the highest power consumption reduction followed by EPS and PASB as justified by the temperature reduction.

Table 5: Energy consumption

Method	Monthly power consumption (kWh)	Monthly power reduction (kWh)	Percentage monthly power reduction (%)
Without insulation	432	-	-
PASB	427	5	1.16
EPS	370	62	14.35
Rockwool	360	72	16.67

The electricity cost was calculated based on the electricity tariff by Tenaga Nasional Berhad. The first 200kWh was 21.80 cent/kWh per month, for the next 100kWh was 33.4 cent/kWh per month. The calculated electricity cost was tabulated in Table 6. The results show that by applying Rockwool insulation could save the electricity cost the most while PASB insulation has the least electricity cost saved. This is due to Rockwool insulation has the highest power reduction followed by EPS and PASB insulation.

Table 6: Monthly electricity cost saving

Method	Monthly power consumption (kWh)	Monthly Electricity cost (RM)	Monthly saving (RM)	Percentage monthly saving (%)
Without insulation	432	241.93	-	-
PASB	427	238.73	3.2	1.32
EPS	370	207.02	34.91	14.43
Rockwool	360	201.17	40.76	16.85

Comparison between Rockwool and polystyrene foam insulation

The main idea of this paper is to evaluate the different types of wall thermal insulation performance based on the indoor air temperature reduction. The results show that Rockwool with a thickness of 52mm could reduce the indoor temperature by 1.3°C. The study done by Venkiteswaran et al. [6] in Malaysia showing that by using 50mm thick of polystyrene could reduce the room temperature by 0.81°C. The results of the using Rockwool as wall insulation were compared with the results by using polystyrene foam insulation that has been conducted in the previous studies under the same of tropical climate in Malaysia. The results show that Rockwool could achieve a 0.49°C lower as compared to polystyrene foam insulation.

Conclusions

In this study, the effect of the insulating materials on temperature reduction for the test room (1.2m x 0.8m x 0.8m) walls was experimentally investigated. The cooling load for three different types of insulating materials are calculated as well. The experimental results show that the insulating material Rockwool (thickness 52mm) could give the best reduction in temperature which could reduce the inner temperature of the test room by 3.85%, leading to a potential saving of RM40 per month without using air condition units or fan, compared to the results given by EPS insulation (thickness 23mm) which reduced 2.49% and to the PASB insulation (thickness 5mm) which reduced 0.89%.

References

- [1] Tabrizi T. B., Hill G., Aitchison M., The impact of different insulation options on the life cycle energy demands of a hypothetical residential building, *Procedia Engineering*. 180 (2017) 128-135.
- [2] Demharter A., Polyurethane rigid foam, a proven thermal insulating material for applications between +130°C and -196°C, *Cryogenics*. 38 (1998) 113-117.
- [3] Salman B.H., Hamzah M.Z., Purbolaksono b. J., Inayat-Hussain J.I., Mohammed H.A., Muhieldeen M.W. Determination of correlation functions of the oxide scale growth and the temperature increase. *Engineering Failure Analysis* 18 (2011) 2260–2271
- [4] Aktacir M. A., Buyukalaca O., Yilmaz T., A case study for influence of building thermal insulation on cooling load and air-conditioning system in the hot and humid regions, *Applied Energy*. 87 (2010) 599-607.
- [5] Muhieldeen M.W., Adam NM, Elias S, Tang SH, Ghezavati H. CFD Simulation on Use of Polyethylene Single Bubble to Reduce Radiant Heat on Lecture Hall. *International Advanced of Technology Congress (ATCi), PWTC, Malaysia*. November 3-5, (2009)
- [6] Incropera F. P., Dewitt D. P., Bergman T. L., Lavine A. S., *Principles of heat and mass transfer* 7th edition, Chapter 1: Introduction. (2012) 1-49.
- [7] Venkiteswaran V.K., Liman J., Alkaff S.A., Comparative study of passive methods for reducing cooling load, *Energy Build.* 142 (2017) 2689-2697.
- [8] Hu X.J., Zhu W., Analysis of application of external wall insulating technology in affordable housing in Tonglu country, *Energy Build.* 14 (2012) 488-492.
- [9] Ding Y., Xu L., Li B.Z., Liu X.Q., Experiment study on thermal performance of external insulated walls of intermittent air-conditioned rooms in hot summer and cold winter region, China, *Advances in Materials Science and Engineering*. (2014).
- [10] Cao B., Study on building energy consumption performance of composite polystyrene granule thermal insulation mortar, *Chemical Engineering Transaction*. 62 (2017).
- [11] Noerwasito V.T., Nirwansyaha R., Embodied energy efficient in the composition of bamboo walls and soil blocks wall, *Social and Behavioral Sciences*. 179 (2015) 330-338.

- [12] Kiran Kumar G., Saboor S., Ashok Babu T. P., Study of various glass window and building wall materials in different climatic zone of India for energy efficient building construction, *Energy Procedia*. 138 (2017) 580-585.
- [13] Abdul Halim N.H., "Thermal and energy performance of pitch and wall insulation for air-conditioned buildings in Malaysia", *Applied Mechanics and Materials*. 209-211 (2012) 1766-1769.
- [14] Pruteanu M., Maxineasa S.G., Vasilache M., Taranu N., A study on the use of expanded polystyrene for external masonry walls thermal insulation, *Architecture Section*. (2013) 31-42.
- [15] Mishra S., Usmani J. A., S. Varshney, Energy saving analysis in buildings walls through thermal insulation system, *International Journal of Engineering Research and Applications*. 2 (2012) 128-135.
- [16] Huang Y., Qi R., Mi L., Investigation on energy-efficient retrofitting measures on commercial building external walls in cooling-dominate cities, *Energy Procedia*. 205 (2017) 2973-2979.
- [17] Mahlia T. M. I., Taufiq B. N., Masjuki H. H.. Correlation between thermal conductivity and the thickness of selected insulation materials for building wall, *Energy Build*. 39 (2007) 182-207.
- [18] Wang D., The influence of thermal insulation position in building exterior walls on indoor thermal comfort and energy consumption of residential buildings in Chong Qing, *Earth and Environmental Science*. 40 (2016).
- [19] Yongson O., Badruddin I.A., Aswatha Marayana Z.A., Zainal P.A., Air-flow analysis in an air-conditioning room, *J. Build. Environ*. 42 (2007) 1531-1537.
- [20] Premier A., A new stucco coating based on pearlescent pigments for improving wall thermal insulation, 10th Conference on Advanced Buildings Skin. (2015).
- [21] Nyers J., Komuves P., Optimum of external wall thermal insulation thickness using total cost method, *Obuda University Budapest*. (2015).
- [22] Ma Y. & Si H., External wall insulation technology Research in building, *Information Technology Journal*. 13 (2014) 78-85.
- [23] Bojic M., Yik F., Sat P., Influence of thermal insulation position in building envelope on the space cooling of high-rise residential buildings in Hong Kong, *Energy and Buildings*. 33 (2001) 569-581.
- [24] Muhieldeen, M. W., Adam, N. M., & Salman, B. H. Experimental and numerical studies of reducing cooling load of lecture hall. *Energy and Buildings*, 89, (2015) 163–169.
- [25] Muhieldeen M.W. Kuang Y.C. Saving Energy Costs by Combining Air-Conditioning and Air-Circulation Using CFD To Achieve Thermal Comfort in The Building. *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 58, Issue 1 (2019) 84-99.

RECENT STUDIES ON LITHIUM-ION BATTERY THERMAL BEHAVIOUR FOR ELECTRIC AND HYBRID ELECTRIC VEHICLES: A REVIEW

Adam Maxwell Doumin¹

Zul Hilmi Che Daud^{2*}

Zainab Asus²

Izhari Izmi Mazali¹

Mohd Kameil Abdul Hamid¹

Mohd Farid Muhamad Said²

¹School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia

²Automotive Development Centre (ADC), School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia

*Corresponding author's E-mail address: hilmi@mail.fkm.utm.my

Abstract: Environmental issues related to excess greenhouse gases have led to the introduction of electric (EV) and hybrid electric vehicles (HEV). The power source of these vehicles is mainly supplied from a battery specifically Lithium-ion battery (LIB). Lithium-ion battery possesses the characteristic of high power rating, high energy density, and high cycle life. However, this battery suffers from temperature problems, and thus battery thermal management (BTM) is crucial. This paper reviews the existing works related to lithium-ion battery thermal behaviours such as battery heat generation, battery temperature distribution and battery thermal management. Analysis of battery heat generation and temperature distribution is focusing more on single cell temperature behaviour. For battery thermal management, cooling systems such as air-based, liquid based and solid-based are reviewed but special attention has been putting on air-cooling type.

Keywords: Lithium-Ion Battery, Thermal Behavior, Electric Vehicle, Hybrid Electric Vehicle

Introduction

Land vehicles that available on the road such as cars, busses, trucks, and even motorcycles are increasing in numbers as years passed. This include the increase of internal combustion engine (ICE) usage since most vehicles are powered by the mechanical stroke of an ICE [1]. However, the products produce by burning fossil fuels inside ICE is not environmentally friendly since it produce waste products of excess carbon dioxide during complete combustion and carbon monoxide during incomplete combustion, which will affect the health of humans and contributes to the addition of greenhouse gases [1, 2]. Build-up of greenhouse gases in the lower level of atmosphere traps the heat causing increase of earth temperature and lead to the catastrophic effect of global warming such as rising sea level, droughts, climate changes and flora fauna extinction [3]. Based on a report from Shaukat *et al.* [4], transportation sector contributes 31% of greenhouse gases emissions and the other percentage consist of electricity power sector, industry sector, residential commercial sector and lastly, renewable energy resources sector [4]. Because of this increasing greenhouse gas problem, solution has been made to counter the effects. Some example would be the emergence of green energy such as wind turbines, hydro dam power, and solar power for supplying electricity to the grid [5]. In

transportation sector, they have introduced electric vehicle (EV) and hybrid electric vehicle (HEV) that target in reducing the emission of greenhouse gas such as carbon dioxide [5, 6].

In all the architecture of EVs and HEVs, battery is the main component that provides electrical power for the whole systems. The requirements of becoming an electrical storage power supply for this system is to have a high power rating, high density of energy, high life-cycle and safe to be operate [7, 8]. Lithium-ion battery (LIB) types are widely used in the application of EV and HEV nowadays due to its good characteristic that meets the requirements of becoming a power supply storage, plus LIB does not contain any poisonous metals like cadmium, lead or mercury [7-9]. Although LIB possess good characteristic that is suitable for application for EV HEV energy storage, LIB has a major downside related to the effects of temperatures since it will have a great impact on the cell operating efficiency, contribution to cell degradation, reducing lifetime cycle of the battery and increase safety risks [10-13]. Figure 1 shows the relationship of various types of battery, heat responses and ways on managing the dissipation of battery heat.

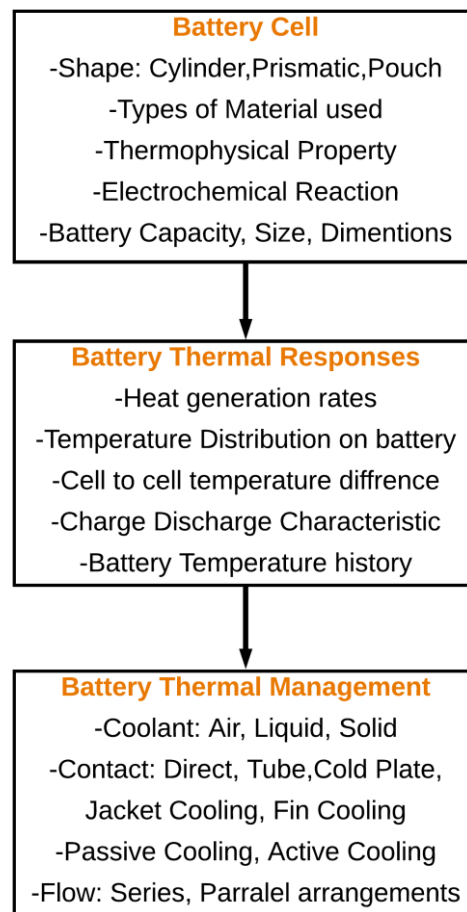


Fig. 1. Flow of Battery thermal management

In this paper, review process has been carried out in these scopes: 1) Introduction of Lithium-ion battery; 2) Battery heat generations; 3) Battery temperature distribution throughout the surface area; 4) Battery thermal management review on air, liquid and solid heat dissipation systems.

Battery Heat Generation

There are two sources that contribute to heat generation in lithium-ion battery: 1) Joule effects, where heat is generated when electric charge flows through a resistance. 2) Heat released by electro-chemical reactions [14]. There are also outside parameters that influence the heat generation of lithium-ion battery such as,

- Charge and Discharge current. When higher value of current used for charging and discharging, joule effect presents the strongest contributor of high temperature.
- The State of Charge (SOC) effects. SOC is defined as the percentage energy remains in the battery. SOC is linked to the electrochemical phenomenon and diffusion of lithium ions.
- Temperature of battery will also affect the electrochemical reactions that can result in the internal resistance becoming lower.
- The battery material chemistry because some materials have a great impact on heat.

Study of different current rates for charge and discharge in order to obtain the relationship with temperature is widely done since joule effect becomes the main heat contributor at higher current rates. The relationship obtained is, higher rates of currents circulate inside a battery, the average battery temperature increases [15-18]. The relationship of current rates and temperature is shown in figure 2. As currents pass through a certain resistance, some energy will be lost as heat; thus, when more current is supplied, more energy will be lost that will contribute to the heat generation from the battery.

Second parameter that influences heat generation of lithium-ion battery is the state of charge (SOC) of the battery. Knowing that an increase of current rates will lead to higher heat generation and the temperature increase is almost linear. When it comes to the end of discharge process or more specifically below 20% of SOC, there is a sudden increase of temperature as shown in figure 3 [19]. The sudden increase of temperature is due to a huge increase of internal resistance at the end of the discharge process [19].

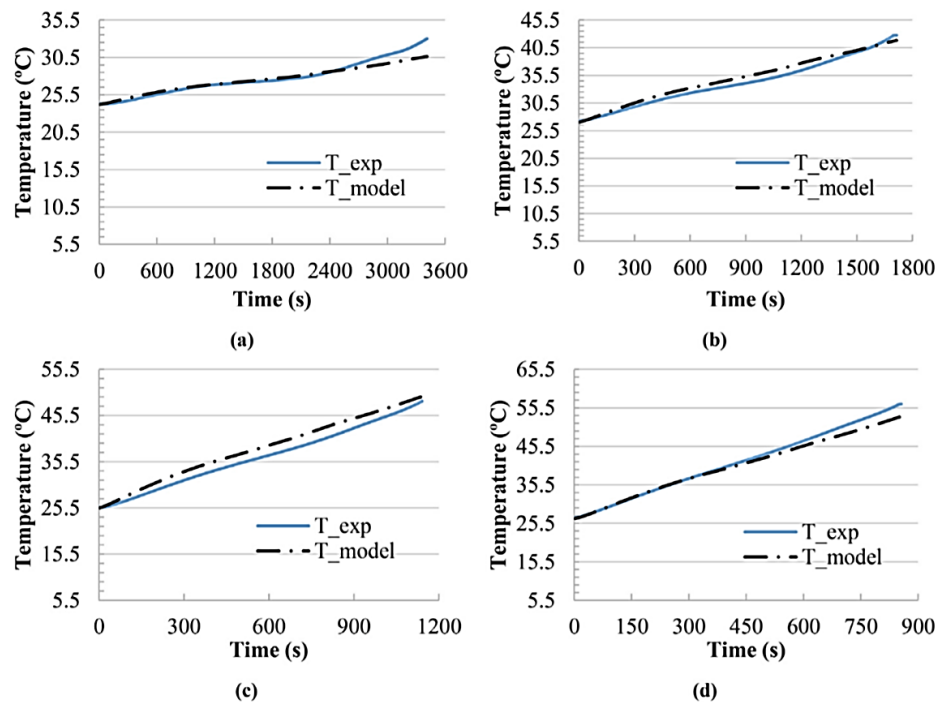


Fig. 2. Average temperatures of the battery pack
a) 1C, b) 2C, c) 3C, d) 4C [15]

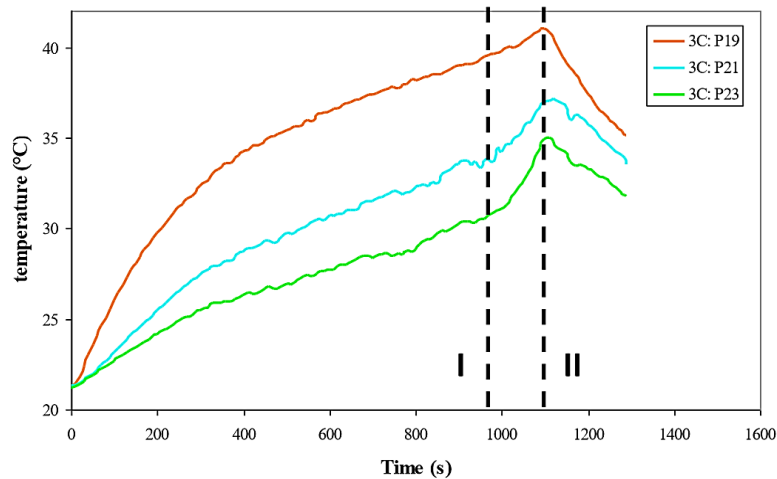


Fig. 3. Cell surface temperature evolution under 3C discharge rate at cooling air velocity of 2 m/s for abuse discharge [19]

Battery Temperature Distribution

Temperature at different points on the cell surface is different. This is because lithium-ion are able to move freely inside the electrolyte, thus at certain area that are pack with movement of ions, temperature is higher. Determining the cell-surface temperature distribution is one of the solutions on managing thermal issues on lithium-ion battery [20-22]. The purpose of this method is to find which area that represent higher cell surface temperature. The method is to arrange thermocouples on the cell surface, run the charge or discharge process, then record and compare the temperature of all thermocouples. In figure 4, 3 different types of thermocouples arrangement is presented. The arrangement is based on early hypothesis where heat generated more near the positive and negative terminal due to constrain path of packed lithium-ion. Based on figure 4 (a) and (b) represent the arrangement of 10 thermocouples on each side of the battery cell. Figure 4 (c) Represent the simplified thermocouple placing where only 3 thermocouples on each side of the battery is used. The simplified arrangement is due to early stage of hypothesis that state the area that near positive and negative terminal will experience higher temperature due to constrain path of current flow and high movement of lithium ions thus resulting higher heat generation on that area.

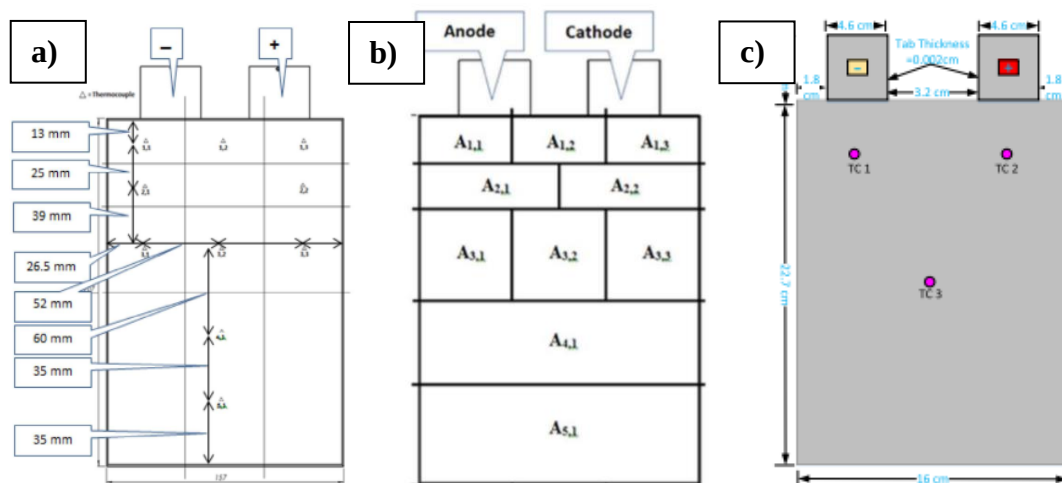


Fig. 4. Arrangement of thermocouple placing [20-22]

The battery is discharge with a constants current of 1C, 2C, 3C and 4C [21]. The result is shown in figure 5, where the battery is discharged at 3C current rate and surrounding temperature remains constant at 25°C. At the beginning of discharged process, temperature at all location is homogenous but as time passes the hot area become significant. At the end of the discharged process, it is conclude that the hot area is near the terminals area.

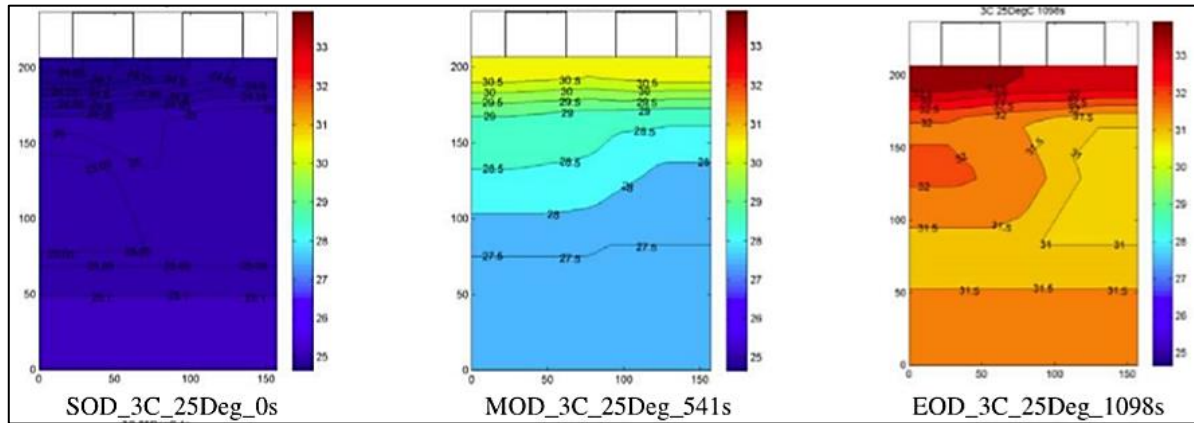


Fig. 5. Temperature distribution on LIB at 3C and 25°C [21]

However, between positive and negative terminal, the temperature is not the same. Based on the review, positive terminal experienced higher temperature compare to negative terminal [20, 21, 23]. In figure 5 and 6, at the end of discharged process, anode or positive terminal generate more heat and it is hotter compare to cathode terminal [20, 21, 23]. Reason due to this temperature variation is that the type of material used for positive and negative terminal. Different material has different value of specific heat capacity. Heat capacity is define as the amount of heat energy required to raise the temperature by one degree Celsius thus the higher the specific heat capacity value, the slower the heat increase. In positive terminal of lithium-ion battery, normally it consist of Cobalt, Nickel, and Manganese metals while on the negative terminal, lithium or graphite is used. Table 1 shows the types of metal and its specific heat capacity. Based on the table 1, positive terminal materials has lower value of specific heat capacity means that less heat energy required to increase the temperature by one degree Celsius thus the material will experience a temperature increase in a fast rates.

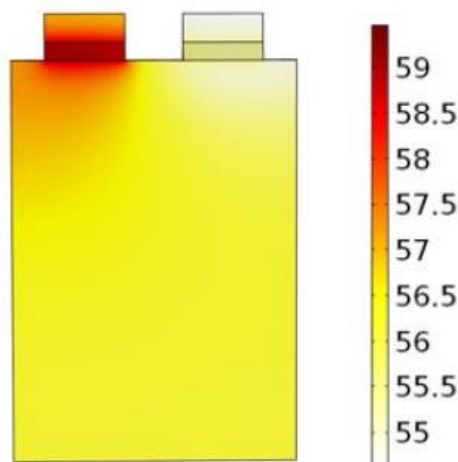


Fig. 6. Temperature on cell surface [23]

Table 1: Types of material and its specific heat capacity

Material types	Specific heat capacity J/(g°C)
Cobalt	0.41868
Nickel	0.50241
Manganese	0.47729
Lithium	3.55871
Graphite	0.72000

Battery Thermal Management

When a large amount of battery cells is combined in pack, it will become a battery module. Temperature in a battery module is required to be maintained in a suitable range with temperature between cells should be as uniform as possible. Hence, good heat dissipation is important to achieve this goal. There are three main types of cooling media for battery thermal management, which are air-based, liquid-based and solid-based such as phase change material (PCM). Air and liquid based cooling is an active cooling method because the coolant is the one that carry all the heat to be dissipate while solid-based is a passive cooling method since its only enhance the cooling method for an active medium such as air or liquid to dissipate the heat.

For active cooling, when it comes to a large-scale of cells plus discharging at high discharge rates, the cooling capacity of natural or forced convection with air are barely met the cooling requirements. That is when liquid cooling comes in since liquid-cooling efficiency can be much higher than air-based cooling [24, 25]. However, the main drawback of liquid cooling is its complexity, cost, high maintenance and potential leakage make manufacturers hesitate to use it [24-26] compared to air-based cooling system. In air-based cooling system the application is less complex, less cost needed, and less maintenance issues.

Passive cooling is normally based on solid material. A phase change material (PCM) or heat sink is largely used for enhancing the rate of heat transfer of active cooling. For a simple battery power application, PCM is great for cooling because PCM does not require any power source for cooling purpose however PCM is not suitable for higher temperature cooling and it need an additional support from active cooling to lower down the high temperature.

Selecting a proper battery thermal management is depend on the ease of use, adaptability, heat transfer efficiency, maintenance, duration of life, first cost and annual cost. Different types of cell used (prismatic, cylindrical, pouch) has different cooling setup.

Air-based cooling

The mechanism of air-based cooling used several fans to drive the air and directly cools the cells by flowing through the surfaces of cells. However, air-based cooling has poor heat capacity and low thermal conductivity, thus it is not proper to be used as cooling medium in BTM but some vehicles consider using this cooling method because of its simplicity of operating, design manufacturing and low cost used [12, 24]. The cooling configurations can be divided as series, parallel, mix series-parallel configurations [12, 24]. The configuration of series and parallel configuration is shown in figure 7 & 8.

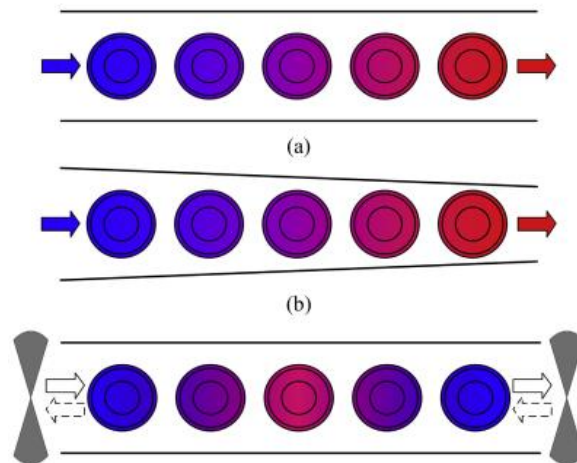


Fig. 7. Air-cooling configuration of Series configuration [24]

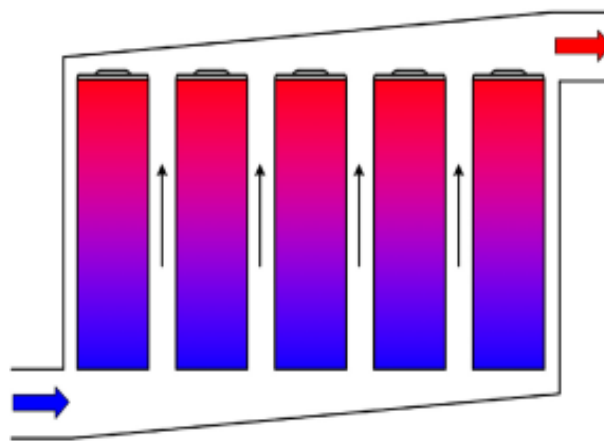


Fig. 8. Air-cooling configuration of Parallel configuration [24]

For series cooling configuration, the temperature of air-coolant may rise due to heat absorption when the air-coolant flows through the series arrangement of cells, and this will result to the cell temperature to cool down. However, for series arrangement, the air flow will pass through cell by cell resulting the first cell near the inlet to be cooled but the cell that is near to the exit is obviously higher since the air-coolant temperature already rise up due to heat dissipation from the cells before [24, 27]. Therefore, series arrangement still can reduce the cell temperature but temperature uniformity for all cells cannot be achieved.

The next idea came for pure parallel cooling configuration. The air-coolant flow passing the cells from bottom to top side of the cell or in the opposite way [24, 27]. Since the coolant flows past all of the cells at the same time, thus the temperature distribution between cells in a battery pack is more uniform. The key factor in order to achieve the uniform temperature is the flow distribution. An ideal design of the input distributor and outlet collector can evenly distribute the airflow throughout all the cells.

Chen *et al.*[28] has studied the effect of cooling air velocity to the surface temperature of lithium-ion battery. From this study, graph of air temperature versus time in the function of velocity has been plotted and shown in figure 9. Result shows that the temperature will decrease as the velocity of cooling air speed increase because the heat transfers dissipation rate increase. Heat transfer rate is affected by these three parameters, which are, velocity of moving air, dynamic viscosity of air, and thermal conductivity of air. Figure 9 shows that as the velocity

increase, temperature will decrease but the difference of temperature become smaller at higher air velocities. This is due to the effect of dynamic viscosity as temperature is low, the dynamic viscosity become high thus more force is needed to move the particle of air.

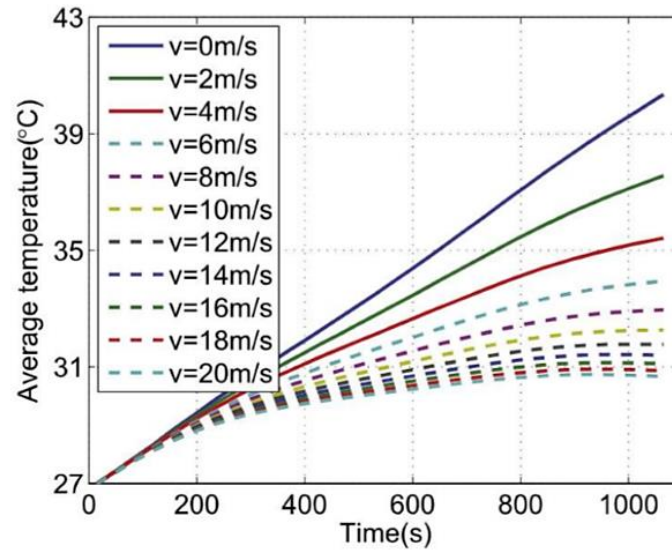


Fig. 9. Graph of air-cooling velocity with temperatures [28]

Arrangements of battery cells play a big role in designing an optimum cooling in battery pack. One parameter is about the cell-to-cell distance, since a battery pack has to be compact so it did not used to much space in hybrid electric vehicles. Yang *et al.* [29] assessed the effects of cylindrical battery arrangements in a pack in terms of temperature behaviour. The variable parameter in this assessment is the distance between the cells and the result is based on maximum temperature and temperature uniformity. Figure 10 shows the plotted graph at different range of gaps between cells that is arrange as in figure 11 and the temperature result.

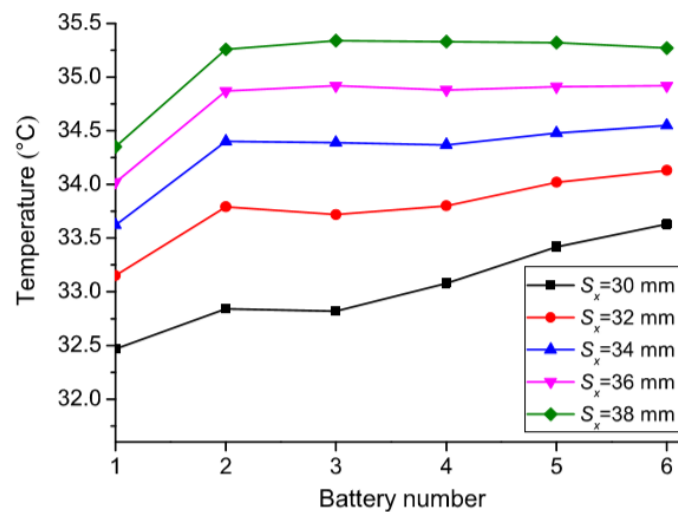


Fig. 10. Graph of batteries temperatures with different gap distances [29]

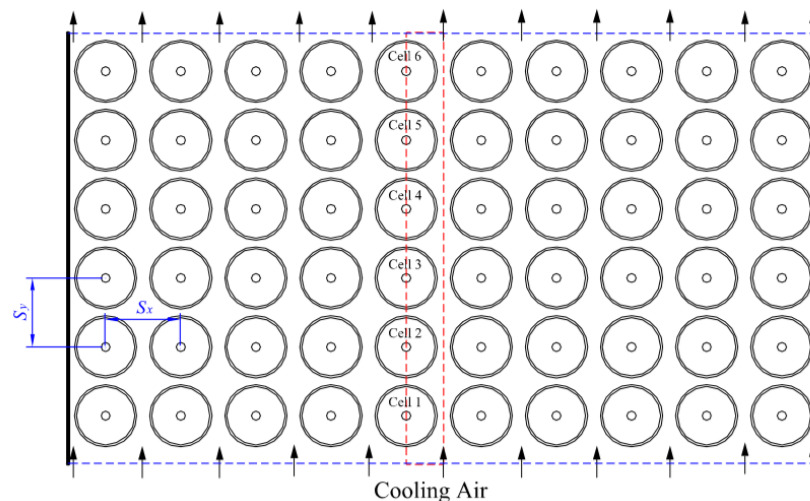


Fig. 11. Schematics of air cooling arrangement [29]

Figure 10 shows the temperature of batteries at the end of discharge. It shows that when the gap between cells become larger, the battery temperature is higher when compared to a smaller gap but the temperature uniformity is improved. Larger gap leads to decrease of air coolant velocity, which lowers down the heat dissipation rate of the battery.

Conclusions

Temperature control is crucial in operating lithium-ion battery. Reviews have been conducted on the existing works related to lithium-ion battery thermal behaviours such as battery heat generation, battery temperature distribution and battery thermal management. Battery heat generation trends shows that by increasing the value of current rates, temperature will increase and the major cause is due to joule effects. When an abuse discharged was carries out by discharging below 20% SOC, battery will experience a sudden temperature rise due to huge increase of internal resistance at the end of discharge process. Temperature distribution on a cell surface study shows that temperature is non-homogenous. Normally areas that are near the positive and negative terminal are have higher temperature compared to other location of the battery. Finally, battery thermal management that used cooling systems such as air-based, liquid based, and solid-based are reviewed with the focus is more on air-cooling system. Air-based cooling is a simple method to cool a battery pack and the heat transfer rate can be increase by varying the cooling air speed and optimizing the cells arrangement. Liquid-based cooling provide the best heat dissipation strategy but it require a complex design and require high maintenances as liquid may damage the cells if failure occurs. The performance of air and liquid can be enhanced by merging with a passive system using solid heat sink.

Acknowledgement

This research was funded by a grant from Research University Grant (UTM-TDR: PY/2018/03432).

References

- [1] Ferguson, Colin R, and Allan T Kirkpatrick. *Internal Combustion Engines: Applied Thermosciences*. John Wiley & Sons, 2015.
- [2] Olivier, Jos GJ, KM Schure, and JAHW Peters. "Trends in Global Co2 and Total Greenhouse Gas Emissions." *PBL Netherlands Environmental Assessment Agency* (2017): 5.

- [3] Ho, Daniel Tang Kuok. "Climate Change in Malaysia: Trends, Contributors, Impacts, Mitigation and Adaptations." *Science of the Total Environment* (2018).
- [4] Shaukat, N, B Khan, SM Ali, CA Mehmood, J Khan, U Farid, M Majid, *et al.* "A Survey on Electric Vehicle Transportation within Smart Grid System." *Renewable and Sustainable Energy Reviews* 81 (2018): 1329-49.
- [5] "Industry Week: Cars Pass Power Plants as Top Source of Us Carbon Emissions." Industry Week, 2017, accessed January, 2019, <https://www.industryweek.com/environment/cars-pass-power-plants-top-source-us-carbon-emissions>.
- [6] Lanzarotto, D, M Marchesoni, M Passalacqua, A Pini Prato, and M Repetto. "Overview of Different Hybrid Vehicle Architectures." *IFAC-PapersOnLine* 51, no. 9 (2018): 218-22.
- [7] Ehsani, Mehrdad, Yimin Gao, Stefano Longo, and Kambiz Ebrahimi. *Modern Electric, Hybrid Electric, and Fuel Cell Vehicles*. CRC press, 2018.
- [8] Larminie, James, and John Lowry. *Electric Vehicle Technology Explained*. John Wiley & Sons, 2012.
- [9] Hannan, Mohammad A, MS Hossain Lipu, Aini Hussain, and Azah Mohamed. "A Review of Lithium-Ion Battery State of Charge Estimation and Management System in Electric Vehicle Applications: Challenges and Recommendations." *Renewable and Sustainable Energy Reviews* 78 (2017): 834-54.
- [10] Wang, Qingsong, Ping Ping, Xuejuan Zhao, Guanquan Chu, Jinhua Sun, and Chunhua Chen. "Thermal Runaway Caused Fire and Explosion of Lithium Ion Battery." *Journal of power sources* 208 (2012): 210-24.
- [11] Mohammadian, Shahabeddin K, and Yuwen Zhang. "Thermal Management Optimization of an Air-Cooled Li-Ion Battery Module Using Pin-Fin Heat Sinks for Hybrid Electric Vehicles." *Journal of Power Sources* 273 (2015): 431-39.
- [12] Wang, Qian, Bin Jiang, Bo Li, and Yuying Yan. "A Critical Review of Thermal Management Models and Solutions of Lithium-Ion Batteries for the Development of Pure Electric Vehicles." *Renewable and Sustainable Energy Reviews* 64 (2016): 106-28.
- [13] Abdul-Quadir, Y., P. Heikkilä, T. Lehmuspelto, J. Karppinen, T. Laurila, and M. Paulasto-Kröckel. "Thermal Investigation of a Battery Module for Work Machines." Paper presented at the 2011 12th Intl. Conf. on Thermal, Mechanical & Multi-Physics Simulation and Experiments in Microelectronics and Microsystems, 18-20 April 2011.
- [14] Ianniciello, Lucia, Pascal Henry Biwolé, and Patrick Achard. "Electric Vehicles Batteries Thermal Management Systems Employing Phase Change Materials." *Journal of power sources* 378 (2018): 383-403.
- [15] Malik, M, M Mathew, I Dincer, MA Rosen, J McGrory, and M Fowler. "Experimental Investigation and Thermal Modelling of a Series Connected Lifepo4 Battery Pack." *International Journal of Thermal Sciences* 132 (2018): 466-77.
- [16] Lu, Zhao, Xiaoling Yu, Liyu Zhang, Xiangzhao Meng, Lichuan Wei, and Liwen Jin. "Experimental Investigation on the Charge-Discharge Performance of the Commercial Lithium-Ion Batteries." *Energy Procedia* 143 (2017/12/01/ 2017): 21-26. <https://doi.org/https://doi.org/10.1016/j.egypro.2017.12.642>. <http://www.sciencedirect.com/science/article/pii/S1876610217364068>.
- [17] Wang, Zhirong, Kai Liu, Jingjing Liu, Qingkai Luo, and Chi Ma. "Influence of the Charging and Discharging of the 18650 Lithium-Ion Battery Thermal Runaway." *Journal of Loss Prevention in the Process Industries* (2017).

- [18] Panchal, S, M Mathew, R Fraser, and M Fowler. "Electrochemical Thermal Modeling and Experimental Measurements of 18650 Cylindrical Lithium-Ion Battery During Discharge Cycle for an Ev." *Applied Thermal Engineering* 135 (2018): 123-32.
- [19] Daud, Zul Hilmi Che, Zainab Asus, Izhari Izmi Mazali, Mohd Kameil Abdul Hamid, and Fitri Yakub Doumin Adam Maxwell. "Experimental Study of Lithium-Ion Battery Thermal Behaviour under Abuse Discharge Condition." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* (2019).
- [20] Panchal, S, I Dincer, M Agelin-Chaab, R Fraser, and M Fowler. "Thermal Modeling and Validation of Temperature Distributions in a Prismatic Lithium-Ion Battery at Different Discharge Rates and Varying Boundary Conditions." *Applied Thermal Engineering* 96 (2016): 190-99.
- [21] "Experimental Temperature Distributions in a Prismatic Lithium-Ion Battery at Varying Conditions." *International Communications in Heat and Mass Transfer* 71 (2016): 35-43.
- [22] Panchal, Satyam, Ibrahim Dincer, Martin Agelin-Chaab, Michael Fowler, and Roydon Fraser. "Uneven Temperature and Voltage Distributions Due to Rapid Discharge Rates and Different Boundary Conditions for Series-Connected Lifepo4 Batteries." *International Communications in Heat and Mass Transfer* 81 (2017): 210-17.
- [23] Worwood, D, E Hosseinzadeh, Q Kellner, J Marco, D Greenwood, R McGlen, WD Widanage, A Barai, and P Jennings. "Thermal Analysis of a Lithium-Ion Pouch Cell under Aggressive Automotive Duty Cycles with Minimal Cooling." (2016).
- [24] Xia, Guodong, Lei Cao, and Guanglong Bi. *A Review on Battery Thermal Management in Electric Vehicle Application*. Vol. 367, 2017. doi:10.1016/j.jpowsour.2017.09.046.
- [25] Zhao, Chunrong, Antonio CM Sousa, and Fangming Jiang. "Minimization of Thermal Non-Uniformity in Lithium-Ion Battery Pack Cooled by Channeled Liquid Flow." *International Journal of Heat and Mass Transfer* 129 (2019): 660-70.
- [26] Saw, Lip Huat, Hiew Mun Poon, Hui San Thiam, Zuansi Cai, Wen Tong Chong, Nugroho Agung Pambudi, and Yeong Jin King. "Novel Thermal Management System Using Mist Cooling for Lithium-Ion Battery Packs." *Applied energy* 223 (2018): 146-58.
- [27] Wang, Tao, KJ Tseng, Jiyun Zhao, and Zhongbao Wei. "Thermal Investigation of Lithium-Ion Battery Module with Different Cell Arrangement Structures and Forced Air-Cooling Strategies." *Applied energy* 134 (2014): 229-38.
- [28] Chen, Dafen, Jiuchun Jiang, Gi-Heon Kim, Chuanbo Yang, and Ahmad Pesaran. "Comparison of Different Cooling Methods for Lithium Ion Battery Cells." *Applied Thermal Engineering* 94 (2016): 846-54.
- [29] Yang, Naixing, Xiongwen Zhang, Guojun Li, and Dong Hua. "Assessment of the Forced Air-Cooling Performance for Cylindrical Lithium-Ion Battery Packs: A Comparative Analysis between Aligned and Staggered Cell Arrangements." *Applied Thermal Engineering* 80 (2015): 55-65.

DEVELOPMENT OF HEAT PIPE HEAT EXCHANGER TECHNOLOGY AS UTILIZATION OF LOW ENTHALPY GEOTHERMAL ENERGY FOR COFFEE DRYING PROCESS

Imansyah Ibnu Hakim¹
Nandy Putra^{1*}
Kukuh Tri Margono²
Yohanes Gunawan³

¹Heat Transfer Laboratory, Department of Mechanical Engineering Universitas Indonesia, Kampus UI, Depok 16424, Indonesia

*(Corresponding author's E-mail: imansyah@eng.ui.ac.id)

Abstract: Coffee is one of Indonesia's main commodity which competes in international market. However, Indonesia's coffee production still faces some problems. One of the main problems is for the drying process. Moisture contents of coffee need to be reduced to 12%. All this time, most coffee producers use conventional method by using heat source from sunlight. However, uncertain weathers become one of the factors which slow down drying process. Therefore, the purpose of this research is to find and utilize an alternative energy to solve drying process problem. One of energy source which is possible to be used for those criteria is geothermal energy especially with low enthalpy that usually used for daily activity such as bathing, cooking, and warming houses. In this research, Heat Pipe Heat Exchanger (HPHE) which consists of 42 straight heat pipes with staggered configuration is used for utilizing geothermal energy. The heat pipe has a length of 700 mm OD 10 mm with filling ratio is 50% and added by 181 pcs of aluminum with dimension size of 76 x 345 x 0.105 mm as a fin. Geothermal fluid is simulated by water that heated by 9000 watt heater. Temperature of hot water with range of 50, 60, and 70 °C and air speed (0.2, 0.4, 0.6 m/s) are variations for the experiment. The result shows that the highest effectiveness of HPHE is 78.8 % with temperature 70 °C and 0.2 m/s air speed while the lowest effectiveness is 68.8% with temperature 50 °C and 0.6 m/s air speed. Other than that, drying process with temperature 70 °C and 0.6 m/s air speed is the fastest while the slowest is at 50 °C and air speed 0.2 m/s. This result prove that drying process is directly proportional with temperature and air speed.

Keywords: Drying process, coffee, heat pipe, geothermal

Introduction

Coffee is one of the most commonly consumed beverages in the world [1]. Besides of its taste, consuming coffee in certain dose can prevent hypertension, coronary heart disease, inflammation, and breast cancer[2][3]. Indonesia is the fourth biggest coffee producer which exports 11.1 million bags between 2016/2017. But, Indonesia's coffee productivity is lower than Vietnam which has smaller area [4]. Low productivity of coffee in Indonesia is caused by postharvest process especially drying which moisture content is reduced up to 12% (wb) [5]. Most coffee producer in Indonesia still use sunlight energy which has some limitations especially due to uncontrollable weather [6]. This uncontrollable situation causes drying process take longer drying period [7]. Also, open air sun drying has a potential of contamination with dust and insects [8]. Therefore, a new heat energy source is needed for drying process.

Table 1.1. Moisture contents of solar drying of various agricultural produces [9], [10]

Product	Moisture Content (%)		Max allowable temperature (°C)	Drying time (h)
	Initial	Final		
Onions	85	6	55	48
Tomatoes	95	7	60	36
Coffee	65	11		288
Tea	80	3		96
Corn	24	14		
Cabbage	80	4	55	
Chilies	80	5		48

Indonesia is a country with more than 200 volcanoes which are located along Sumatra, Java, Bali, and the eastern islands of the country with an expected potential of about 28 GWe composed of 312 geothermal potential locations [11][12]. Mostly, geothermal energy (GE) in Indonesia are characterized as low enthalpy geothermal and have shallow depth such as hot spring which is utilized for bathhouse, cooking, building heating, and agricultural drying [13][14][15]. Therefore, GE as a renewable energy can be used in drying processes as a heat source replacing the high energy costs of fossil fuels [16]. The distribution of GE in Indonesia is shown in figure 1.

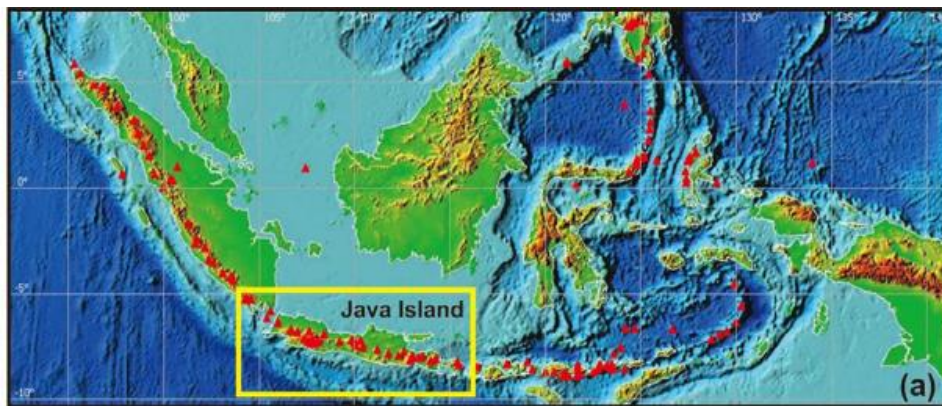


Figure 1. Geothermal in Indonesia [17]

To optimize the utilization of GE, a heat transfer technology that has high efficiency/effectiveness is needed. Heat Pipe Heat Exchangers (HPHEs) have become very efficient because they are very compact and have the possibility for adjusting the transferred heat flux [18]. HPHEs are heat transfer devices which transfer heat passively without any moving parts, that work by phase change and vapour diffusion principal [19]. It consists of closed tubes that are filled with a proper working fluid. The heat pipes can be classified as three types: a conventional heat pipe (CHP), a two phase closed thermos syphon (TPCT) and an oscillating heat pipe (OHP) [20].

Methodology

Coffee's moisture content measurement

Moisture content is defined as how much water is contained in a commodity. Coffee's moisture content is measured with ISO 6673 standard about coffee's moisture content measurement with 105°C temperature [21]. To calculate the moisture content there are two methods, those are dry

base and wet base. The formula for the calculation of moisture content is shown in eq. 1 and eq. 2

$$\text{Moisture Content (dry basis)} = \frac{\text{weight of wet coffee} - \text{weight of dry coffee}}{\text{weight of dry coffee}} \times 100\% \quad (\text{eq.1})$$

$$\text{Moisture Content (wet basis)} = \frac{\text{weight of wet coffee} - \text{weight of dry coffee}}{\text{weight of wet coffee}} \times 100\% \quad (\text{eq.2})$$

Based on the measurement that had been done, moisture content of coffee sample is 55.42% (wb) and 124.31% (db).

Experimental Design

The HPHE is used for this experiment is same as in the previous study [22]. The HPHE consists of 42 tubular heat pipes with staggered arrangement and 181 pcs of the fin. The heat pipe length is 70 cm OD 1 cm while the length of the condenser is 35 cm and the length of the adiabatic side is 10 cm. The heat pipes are made from copper with the wick in the form of sintered copper and the filling ratio of water as working fluid is 50%. Fin is made from aluminium with a thickness of 0.105 mm with a size area of 76 x 345 mm Glass wool and polyurethane is used as insulator in the adiabatic area. Figure 2 shows the HPHE used in this study.

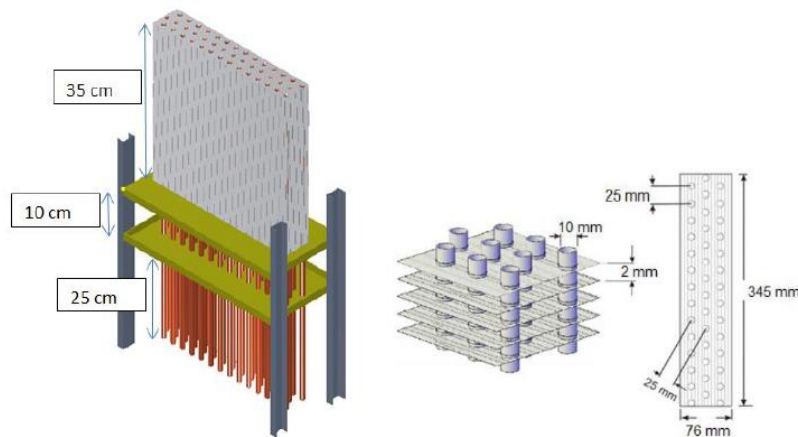


Figure 2. Heat Pipe Heat Exchanger's Design

Geothermal fluid is simulated by heated water with a heater capacity of 9 kilowatts and flowed with a pump which is connected to the inverter. Air for drying is flowed with a fan connected to the inverter so that air flow can also be adjusted. HPHE and all supporting equipment that have been arranged are then assembled in experiment tools as shown in Figure 3.

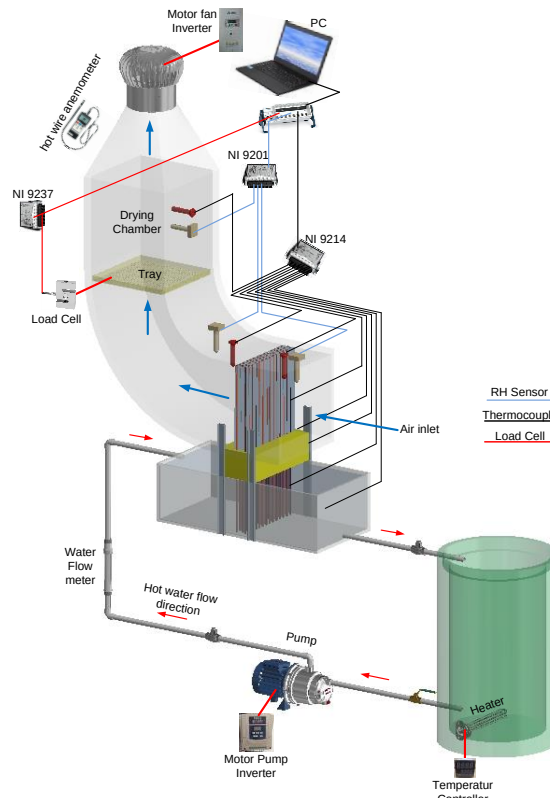


Figure 3. Schematic diagram of coffee drying experimental set up

The sensor used and its placement can be seen in Figure 4. Type K thermocouple connected to NI-cDAQ 9214 module is placed in the evaporator / inlet of the incoming hot water HPHE ($T_{\text{water},i}$), Evaporator outlet / hot water that exits HPHE ($T_{\text{water},o}$), condenser inlet / hot air enters HPHE ($T_{\text{air},i}$), condenser outlet / hot air that exits HPHE ($T_{\text{air},m}$), and air out drying tray ($T_{\text{air},o}$). The relative humidity sensor uses the THD-D Autonics type connected to NI-cDAQ 9201 module and placed in the HPHE condenser inlet ($RH_{\text{air},i}$), HPHE condenser outlet ($RH_{\text{air},m}$), and the drying chamber outlet tray ($RH_{\text{air},o}$). Load Cell with capacity of 1000 grams is connected to NI-cDAQ 9237 module and used for measuring the weight reduction of coffee in the drying tray. The tray area is $360 \times 360 \text{ mm}^2$. Labview software is installed and used for signal recorded from the DAQ in real time.

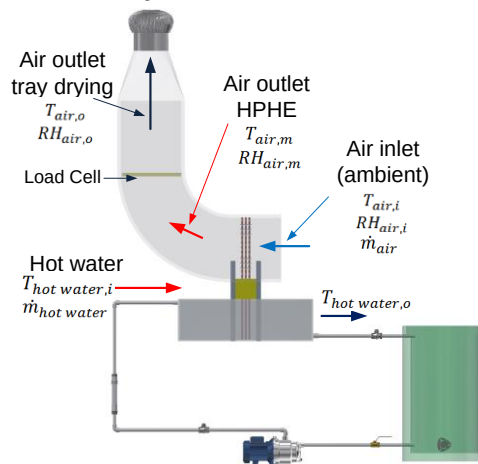


Figure 4. Schematic placement of measuring instruments

Experimental Procedure

Papandayan's coffee variety is chosen as sample because it is located near to geothermal source, Papandayan volcano. A hundred grams of coffee are put into the tray for drying process. This number of mass is chosen due to the dimension of the tray. This experiment is conducted for 4 hours after the experimental set up achieved stable temperature. The variable of this experiment is temperature (50, 60, and 70 °C) and air speed (0.2, 0.4, and 0.6 m/s). While the flow of hot water is set at a value of 18 liters/second. This value is the value for optimal hot water flow in previous studies [22]. The condition of wet and dried coffee on tray is shown in Figure 5.



(a)



(b)

Figure 5. (a) Wet coffee (b) Dried Coffee

HPHE Effectiveness

Effectiveness of heat exchanger is defined as the ratio of the actual heat transfer rate to the maximum possible heat transfer rate [23].

$$\varepsilon = \frac{\text{actual heat transfered}}{\text{maximum possible heat transfered}} \quad (\text{eq. 3})$$

With detail:

$$\varepsilon = \frac{\dot{m} \cdot C_p \text{ air} \cdot (T_h(\text{air}) - T_c(\text{air}))}{\dot{m} \cdot C_p \text{ min} \cdot (T_h(\text{water}) - T_c(\text{air}))} \quad (\text{eq. 4})$$

In this research, minimum $\dot{m}C_p$ is air's $\dot{m}C_p$. So the equation will be

$$\varepsilon = \frac{(T_h(\text{air}) - T_c(\text{air}))}{T_h(\text{water}) - T_c(\text{air})} \quad (\text{eq. 5})$$

Moisture Ratio

Moisture Ratio is used for determining the time needed for drying process. The formula for determining moisture ratio is

$$MR = \frac{M - M_e}{M_o - M_e} \quad (\text{eq. 6})$$

M	= Moisture content at certain time	(%,db)
M_e	= Moisture content equilibrium	(%,db)
M_o	= Moisture content initial	(%,db)
MR	= Moisture ratio	

While M_e is calculated by formula

$$M_e = \frac{(a_w M_m c k)}{[(1 - k a_w)(1 + c k a_w - k a_w)]} \quad (\text{eq. 7})$$

$$c = c_o \exp\left(\frac{c_1}{RT_{ab}}\right) \quad (\text{eq. 8})$$

$$k = k_o \exp\left(\frac{k_1}{RT_{ab}}\right) \quad (\text{eq. 9})$$

Where a_w is water activity, T_{ab} is absolute temperature (K), and R is universal gas constant (8.32 kJ mol⁻¹ K⁻¹). But, based on previous research that had been done by Purnomo et al [24], because of the value of RH is unstable during the experiment and the value of M_e is too small, M_e is negligible for finding MR. For coffee, the moisture ratio is reduced to 66.58%.

Results

Effectiveness of HPHE

The performance of the HPHE system can be seen from the value of its effectiveness. From Figure 6, it can be seen that the greatest effectiveness of HPHE occurs at variations of air velocity 0.2 m / s with a hot water temperature 70°C and flow: 18 lpm with the effectiveness 78.8%. While for the smallest effectiveness occurs at variations of air velocity 0.6 m/s with a hot water temperature 50°C and flow: 18 lpm with effectiveness 67.3%.

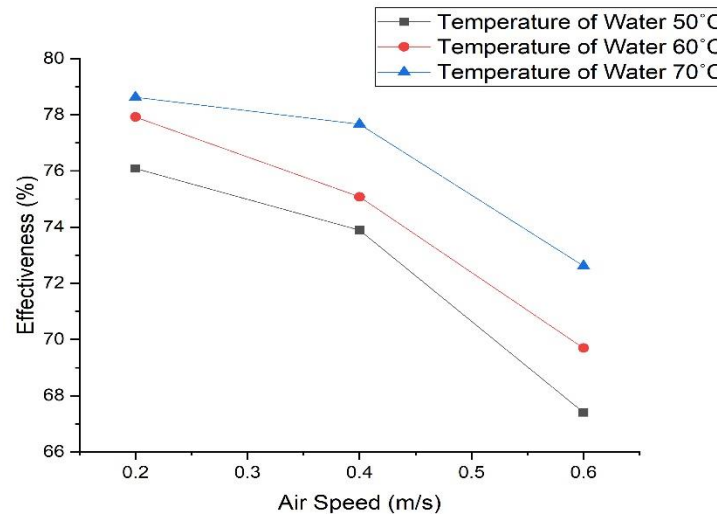


Figure 6. Effectiveness of HPHE

Reduction of Moisture Ratio

After do the experiment for 4 hours, the results shows at figure 7 that the fastest drying process occur with condition temperature 70°C with $v=0.6$ m/s while the slowest at temperature 50°C with $v=0.2$ m/s.

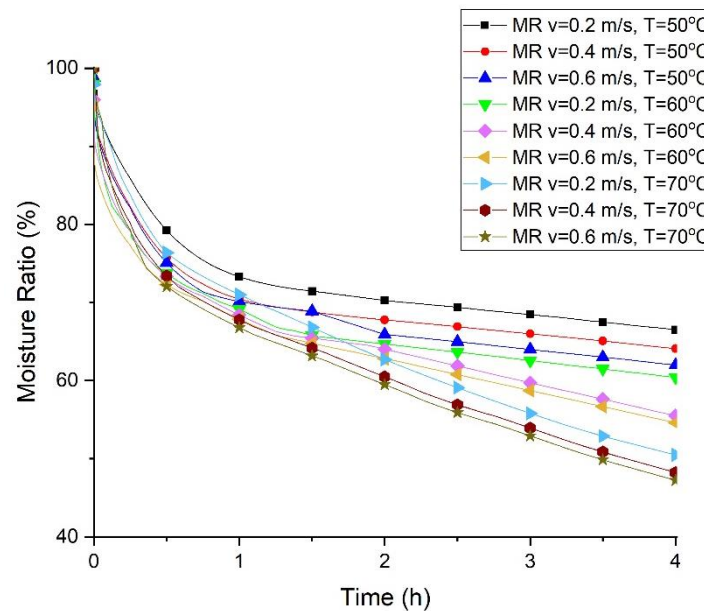


Figure 7. Reduction of moisture ratio

Conclusions

The using of Heat Pipe Heat Exchanger with geothermal as source of energy for drying process is suitable to be implemented to reduce the consumption of fuel and electricity as heat producer. The highest effectiveness of HPHE for drying process in this experiment is 78.8% while the lowest is 67.3%. Higher temperature and air speed will impact to less time needed for drying process.

Acknowledgement

The author would like to thank Directorate of Research and community Engagements Universitas Indonesia (DRPM UI), Kemenristek Dikti for funding this research through the PITTA 2018 scheme with contract number NKB-0739/UN2.R3.1/HKP.05.00/2019, and Research and Development Centre for Electricity, New, Renewable Iso 2452.

References

- [1] M. Montagnana, E. J. Favaloro, and G. Lippi, "Coffee intake and cardiovascular disease: Virtue does not take center stage," *Semin. Thromb. Hemost.*, vol. 38, no. 2, pp. 164–177, 2012.
- [2] C. Ciaramelli, A. Palmioli, and C. Airolidi, "Coffee variety, origin and extraction procedure: Implications for coffee beneficial effects on human health," *Food Chem.*, vol. 278, no. November 2018, pp. 47–55, 2019.
- [3] N. Stefanello *et al.*, "Coffee, caffeine, chlorogenic acid, and the purinergic system," *Food Chem. Toxicol.*, vol. 123, no. October 2018, pp. 298–313, 2019.
- [4] "International Coffee Organization: Annual Review," [Http://Www.Ico.Org/Documents/Cy2017-18/Annual-Review-Website-E.Pdf](http://www.ICO.Org/Documents/Cy2017-18/Annual-Review-Website-E.Pdf), 2016.
- [5] P. Ghosh and N. Venkatachalapathy, "Processing and Drying of Coffee - A review," *Int. J. Eng. Res. Technol.*, vol. 3, no. 12, pp. 784–794, 2014.
- [6] N. Ahmed, J. Singh, H. Chauhan, P. G. A. Anjum, and H. Kour, "Different Drying Methods : Their Applications and Recent Advances Different Drying Methods : Their Applications and Recent Advances," *Researchgate*, vol. 5, no. January 2013, 2015.
- [7] R. O. Lamidi, L. Jiang, P. B. Pathare, Y. D. Wang, and A. P. Roskilly, "Recent advances

- in sustainable drying of agricultural produce: A review,” *Appl. Energy*, vol. 233–234, no. July 2018, pp. 367–385, 2019.
- [8] H. U. Helvaci, A. Menon, L. Y. Aydemir, F. Korel, and G. G. Akkurt, “Drying of olive leaves in a geothermal dryer and determination of quality parameters of dried product,” *Energy Procedia*, vol. 161, no. 2018, pp. 108–114, 2019.
 - [9] O. Prakash, V. Laguri, A. Pandey, A. Kumar, and A. Kumar, “Review on various modelling techniques for the solar dryers,” *Renew. Sustain. Energy Rev.*, vol. 62, pp. 396–417, 2016.
 - [10] A. Fudholi, K. Sopian, M. H. Ruslan, M. A. Alghoul, and M. Y. Sulaiman, “Review of solar dryers for agricultural and marine products,” *Renew. Sustain. Energy Rev.*, vol. 14, no. 1, pp. 1–30, 2010.
 - [11] S. Mohammadzadeh Bina, S. Jalilinasrabady, H. Fujii, and N. A. Pambudi, “Classification of geothermal resources in Indonesia by applying exergy concept,” *Renew. Sustain. Energy Rev.*, vol. 93, no. June, pp. 499–506, 2018.
 - [12] F. Razzano, M. Cei, E. G. Power, and A. Pisano, “Geothermal Power Generation in Italy 2010-2014 Update Report,” *World Geotherm. Congr.*, no. April, pp. 19–25, 2015.
 - [13] G. Pikra, N. Rohmah, R. I. Pramana, and A. J. Purwanto, “The electricity power potency estimation from hot spring in Indonesia with temperature 70-80°C using organic Rankine cycle,” *Energy Procedia*, vol. 68, pp. 12–21, 2015.
 - [14] N. Prasetya, D. E. Umra Lubis, D. Raharjo, N. M. Saptadji, and H. B. Pratama, “Smart geo-energy village development by using cascade direct use of geothermal energy in Bonjol, West Sumatera,” *IOP Conf. Ser. Earth Environ. Sci.*, vol. 103, no. 1, 2018.
 - [15] P. Bayer, G. Attard, P. Blum, and K. Menberg, “The geothermal potential of cities,” *Renew. Sustain. Energy Rev.*, vol. 106, no. February, pp. 17–30, 2019.
 - [16] I. Dincer, A. Midilli, and H. Kucuk, “Progress in exergy, energy, and the environment,” *Prog. Exergy, Energy, Environ.*, pp. 1–1086, 2014.
 - [17] Y. Daud *et al.*, “Resistivity characterization of the Arjuno-Welirang volcanic geothermal system (Indonesia) through 3-D Magnetotelluric inverse modeling,” *J. Asian Earth Sci.*, vol. 174, no. January, pp. 352–363, 2019.
 - [18] A. Burlacu *et al.*, “Energy efficient heat pipe heat exchanger for waste heat recovery in buildings,” *Procedia Manuf.*, vol. 22, pp. 714–721, 2018.
 - [19] H. Jouhara, A. Chauhan, T. Nannou, S. Almahmoud, B. Delpech, and L. C. Wrobel, “Heat pipe based systems - Advances and applications,” *Energy*, vol. 128, pp. 729–754, 2017.
 - [20] W. Srimuang and P. Amatachaya, “A review of the applications of heat pipe heat exchangers for heat recovery,” *Renew. Sustain. Energy Rev.*, vol. 16, no. 6, pp. 4303–4315, 2012.
 - [21] 8589ISO, “International Standard International Standard : ISO 14644,” vol. 2007, 2013.
 - [22] I. I. Hakim, N. Putra, and Y. Gunawan, “Experimental Study for Heat Pipe Applications on Low Enthalpy Geothermal Energy Utilization for Agricultural Products Dryers,” vol. 1, no. 1, pp. 68–77, 2019.
 - [23] E. Firouzfar, M. Soltanieh, S. H. Noie, and M. H. Saidi, “Investigation of heat pipe heat exchanger effectiveness and energy saving in air conditioning systems using silver nanofluid,” *Int. J. Environ. Sci. Technol.*, vol. 9, no. 4, pp. 587–594, 2012.
 - [24] C. W. Purnomo and S. Indarti, “Modification of Indirect Solar Dryer for Simplicia Production,” *IOP Conf. Ser. Earth Environ. Sci.*, vol. 120, no. 1, 2018.

STUDY ON THE AIRFLOW INDUCED BY MOVEMENT OF ELEVATOR CAR WITHIN THE ELEVATOR SHAFT USING PIMPLE ALGORITHM AND K-EPSILON TURBULENCE MODEL

J H Ang¹
Y Yusup¹
S A Z Shaikh-Salim²
A Salehabadi¹
M I Ahmad¹

¹Environmental Technology Division, School of Industrial Technology, Universiti Sains Malaysia, 11800 Penang, Malaysia

²Department of Mechanical Precision Engineering, Malaysia-Japan International Institute of Technology, Universiti Teknologi Malaysia, 54100 Kuala Lumpur, Malaysia

*E-mail: mardianaidayu@usm.my

Abstract: Rapid development, population growth, and scarcity lead to the vertical development of buildings and infrastructures. Along with the increasing demand for vertical transport, aerodynamics study on elevator system is increasingly important. Various computational fluid dynamics (CFD) studies on elevator aerodynamics were done with their focus on acoustic, ride comfort, aerodynamics performance, and airflow effect. However, most of the studies do not discuss in detail on their solving principle and method, especially the working principle and mechanisms of numerical solver (algorithm) and turbulence model. Thus, this paper studies the airflow induced by the moving elevator car within the elevator shaft using PIMPLE algorithm and k-epsilon turbulence model in the openFOAM software. More details on the solving principles and methods were discussed, including computational domain set up, governing principles and equations, numerical methods and mechanisms, and boundary conditions. The assumptions made on the flow are unsteady, turbulent, incompressible, Newtonian and viscous. Pre-processing was done using Salome, meshing and solving were done using openFOAM, while post-processing was done using Paraview. The simulation result obeys the theories of fluid mechanics and physics. Thus, the results of this study are reliable to provide initial guess value to be input into the modeling and simulation using k-omega and k-omega Stress Transport Model (SST), as well as determine the significant measurement points in wind tunnel experiment.

Keywords: CFD, elevator system, PIMPLE algorithm, k-epsilon turbulence model, OpenFoam

Introduction

Rapid development, population growth, and scarcity lead to the vertical development of buildings and infrastructures. Along with the increasing demand for vertical transport, aerodynamics study on elevator system is increasingly important. Various studies on elevator aerodynamics were done with their focus on acoustic, ride comfort, aerodynamics performance, and airflow effect. Navier-Stokes equation and its derivatives are the main governing equation for these studies. For the study using computational fluid dynamics (CFD) software, different numerical solver and turbulence model were used based on the objective of the study. PIMPLE algorithm was preferable for the unsteady incompressible turbulent flow. Modeling and simulation approaches of Reynolds Average Navier-Stokes (RANS) and

Detached Eddies Simulation (DES) were commonly used in the past studies on elevator aerodynamics. There are several turbulence models classified based on the modeling and simulation approaches, for example, k-epsilon model and k-omega model are under RANS, whereas the k-omega Shear Stress Transport (SST) model is under DES.

The standard k-epsilon model is a high Reynolds number model. The application of wall function skips the necessity of the simulation of the buffer region in order to save the computational cost. This model is linear so has fast convergence and low memory demand, but it is less accurate when there is an adverse pressure gradient, strong curvature or jet flow. It is suitable for the flow with consistent pressure from inlet to outlet, and external flow around complex geometries bluff body. It provides a good initial guess for preliminary study before using other more complicated model or carrying out experiment [1,2]. Low Reynolds k-epsilon does not apply wall function, so it requires a denser mesh. It is usually used after getting the initial guess value from the preliminary study using standard k-epsilon model, in order to simulate the problem more accurately. For the case to study the overall airflow within elevator system, k-epsilon turbulence model was used. For example, the study on flow effect to ride comfort, and the aerodynamics performance of the elevator car [3,4].

The k-omega model is suitable for low Reynolds number flow yet the wall function can be applied. It is non-linear thus has a slower convergence. It is sensitive to the initial condition value, so it requires the preliminary study using less sensitive model such as k-epsilon to obtain initial guess values. K-omega is accurate in internal flow, and flow with strong curvature, separated flow, and jet flow. For the case to focus near wall interaction for smaller scale eddies, k-omega turbulence model was used. For example, the studies on the small scale eddies, which contribute to the aero-vibro acoustic factor were done to evaluate the ride quality and noise [5–7].

The advancement of the CFD solving method combines k-epsilon and k-omega as k-omega SST, using the appropriate model based on the location of the problem, which are k-epsilon for the free stream far from the wall, and k-omega for the near wall flow. Its resolution is equal to that of the k-omega and low Reynolds number k-epsilon, but without the weakness of pure k-epsilon and k-omega [1]. k-omega SST provides an accurate initial guess as k-epsilon, and simulate flow separation in adverse pressure gradient conditions as k-omega. The past study using k-omega SST was done by Singh [8]. Fluid Structure Interaction (FSI) was studied to analyze the airflow effect within the elevator system.

A point often overlooked, is that most of the past studies did not discuss in detail on the solving principle and method, especially the working principle and mechanisms of numerical solver (algorithm) and turbulence model. Many studies on aero-vibro and acoustic factors of the elevator system, which focus on the ride quality and noise of the elevator system did not even mention on the type of the numerical solver and turbulence model used [9–12]. Thus, this paper studies the airflow induced by the movement of the elevator car within elevator shaft using PIMPLE algorithm and k-epsilon turbulence model. More details on the numerical algorithm and turbulence model were discussed, including their computational domain set up, governing principles and equations, numerical methods and mechanisms, and boundary conditions. This study can provide initial guess value to be input into the modeling and simulation using k-omega and k-omega SST, as well as determine the significant measurement points in wind tunnel experiment.

Methodology

Pre-processing was done using Salome, meshing and solving were done using openFOAM, while post-processing was done using Paraview.

Computational domain and set up

Symmetry quarter of the elevator system was modeled to represent the elevator system. The computational domain meshed into grids consisting of 142527 cells, which are 141897 hexahedra and 630 polyhedra. Figure 1 shows the computational domain of the elevator system, which uses the static mesh approach. The downward movement of the elevator car at 10 m s^{-1} within the shaft was represented by the air inflow and outflow from the base of the shaft and top of shaft respectively at 10 m s^{-1} . Simulation period of 20 s with the time steps interval 0.003125 s was used since the preliminary test shows its sufficiency for the flow to be stable with a relatively constant parameters value and flow pattern.

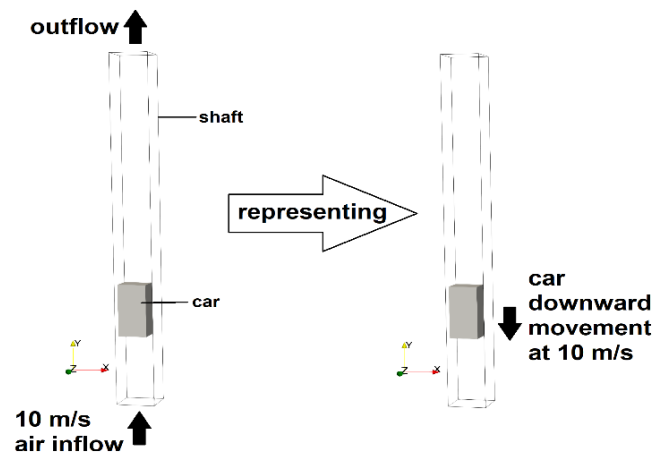


Fig. 1. Computational domain of the elevator system.

Framework of computation

Figure 2 shows the framework of computation for unsteady turbulent, incompressible, viscous, Newtonian flow using PIMPLE algorithm and k-epsilon turbulence model. Further discussion on the boundary conditions and turbulence model were shown in section 2.5.

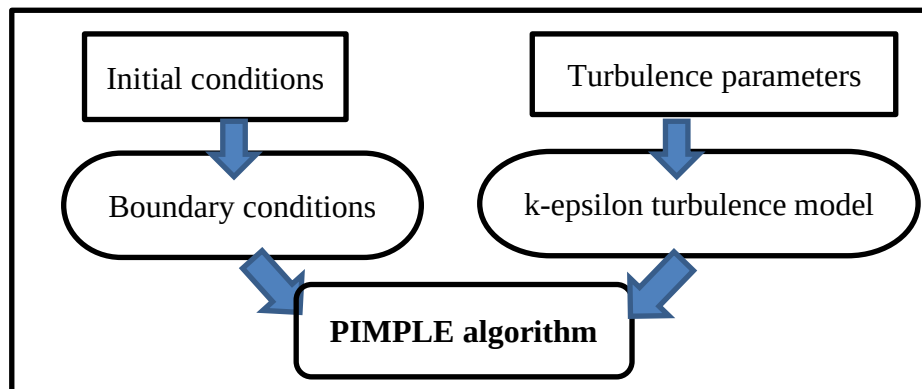


Fig. 2. Framework of computation

Numerical algorithm: PIMPLE algorithm

The atmospheric pressure and air density reduction can be neglected since most of the elevator shaft in high-rise buildings is less than 500 m [13,14]. Moreover, most of the HVAC system in typical buildings maintains the air properties at the ambient air condition [15]. In that case, incompressible flow is assumed in this study. Since there is no strong thermal source in the elevator shaft, the air within it is assumed as a Newtonian fluid, which has a constant viscosity of 1.864×10^{-5} . The unsteady turbulent incompressible Newtonian viscous flow induced by

the elevator car movement is governed by the continuity equation, time-averaged Navier-Stokes equations (Reynolds Average Navier-Stokes equations), and modified k-epsilon turbulence model equations. These equations works based on the conservation of mass and momentum, with constant air temperature and thus constant density and viscosity. Parameters simulated are air velocity in three directions, U_x , U_y , U_z ; kinematic static pressure, p ; turbulent kinetic energy, k ; and dissipation rate of turbulent kinetic energy, ϵ .

PIMPLE algorithm assumes an unsteady, incompressible, viscous and Newtonian flow. The simplified incompressible Navier-Stokes equations governing the PIMPLE algorithm are as in Eq. 1 and 2. These equations are input with the boundary conditions and initial conditions, as well as the turbulence parameters to solve a case.

$$\nabla \cdot \mathbf{u} = 0 \quad (1)$$

$$\frac{d}{dt} \mathbf{u} + \nabla \cdot (\mathbf{u}\mathbf{u}) + (v_{eff} \nabla^2 \mathbf{u}) = -\nabla p \quad (2)$$

Where $\mathbf{u}$ is the velocity (m s^{-1}), p is the kinematic static pressure (m s^{-2}), and v_{eff} is the effective kinematic viscosity.

It is the combination of PISO and SIMPLE algorithm. Initially, it uses the SIMPLE algorithm to initiate the flow and determine the guess value for the following simulation. It uses the SIMPLE steady state treatment to compute the steady-state solution for each time step, which enables the Courant number to be larger than one. However, it solves the weakness of SIMPLE algorithm in neglection of transient behavior and development of flow, which is caused by its large time step. It enables a larger time step yet maintaining the accuracy and accounting of temporal change as in PISO algorithm. This enables it to reduce the computational cost while accounting for the significant transient behavior and development of flow. As a result, the computational cost of each time step is more expensive but a larger time step is allowed in PIMPLE algorithm. It is compatible with all turbulence models in the openFOAM software.

It works based on the parameters of velocity, $u(\text{ms}^{-1})$ and kinematic pressure, $P(\text{m}^2\text{s}^{-2})$, which are input into the mechanism of pressure-momentum coupling with the following steps. Firstly, a momentum matrix is constructed using the guess value, then the pressure matrix is constructed based on this momentum matrix, and pressure is calculated. The velocity initial guess value is corrected with the newly calculated pressure value. The momentum matrix is then reconstructed using the new velocity value. The whole cycle is repeated again until the adjustment of the velocity. The number of cycles (loops) per time step depends on the `nOuterCorrector` set, which is usually 2. PIMPLE algorithm stabilizes the simulation using relaxation factor by mixing the current result with the previous one in between each time step to smoothen the convergence rate. It also moderates the convergence rate using residual control and `nOuterCorrector`.

Turbulence model: K epsilon model

k-epsilon model was used in this study since it is suitable to be applied in the case with high Reynolds (Re) and turbulence flow, which its pressure gradient from inlet to outlet is low [2]. Although it has a lower fidelity of the near-wall region, it is sufficient for this study, which aims to study the overall airflow within the elevator system. The results are sufficient to be used to determine the high velocity and low turbulence points, acting as the preliminary guide for experimental studies such as wind tunnel, as well as the initial guess to be input to a higher order accuracy solving approaches or resolutions. This study only involves study on a big

section of the shaft instead of focusing on specific near wall region. Low turbulence k-epsilon and k-omega will be used for further study after got the baseline study done using k-epsilon.

K epsilon model is a two-equation model with parameters of turbulence kinetic energy, k ($\text{m}^2 \text{s}^{-2}$); and turbulence dissipation rate, ϵ . k is the kinetic energy per unit mass of the turbulent fluctuations in a turbulent airflow. ϵ is the rate of dissipation of the turbulence kinetic energy per unit mass. For the Newtonian fluid, it is the negative rate of change of the turbulence kinetic energy because the energy was used to fluctuate the viscous stresses in resisting deformation of the fluid by the fluctuating strain rates [16]. It uses wall functions, which are $kqRWallFunction$ and $epsilonWallFunction$ to reduce the computational cost for the near-wall region. For the isotropic turbulence, the initial value of parameters can be estimated using the established formula as in Eq. 3, 4 and 5 [17,18].

$$k = 3/2 (uI)^2 \quad (3)$$

Where

u is the air velocity (m s^{-1})

I is the turbulence intensity

$$\epsilon = C_\mu^{\frac{3}{4}} \frac{k^{\frac{3}{2}}}{l} \quad (4)$$

Where

C_μ is the k epsilon model constant which typically valued as 0.09

k is the turbulence kinetic energy (m s^{-2})

l is the turbulence or eddy length scale

$$l = 0.07 L \quad (5)$$

Where

L is the characteristic length of the case

Standard k-epsilon model equations

Eq. 6, 7 and 8 show the equations of standard k-epsilon model, which are equation of turbulence kinetic energy, equation of dissipation rate of turbulence kinetic energy, and equation of turbulence viscosity respectively [19]. Most of the CFD software use these equations with the default value of the coefficients as shown in table 1. However, some software such as openFOAM use the modified equations of k-epsilon model.

$$\frac{D}{Dt} (\rho k) = \nabla \cdot (\rho D_k \nabla k) + G_k + G_b - \rho \epsilon + S_k \quad (6)$$

$$\frac{D}{Dt} (\rho \epsilon) = \nabla \cdot (\rho D_\epsilon \nabla \epsilon) + \frac{C_1 \epsilon}{k} (G_k + C_3 G_b) - C_2 \rho \frac{\epsilon^2}{k} + S_\epsilon \quad (7)$$

$$\nu_t = C_\mu \frac{k^2}{\epsilon} \quad (8)$$

where

$\frac{D}{Dt} (\rho k)$ is the mean-flow material derivative of turbulent kinetic energy (local derivative + advection)

$\nabla \cdot (\rho D_k \nabla k)$ is the turbulence transport of turbulence kinetic energy

$\rho \epsilon$ is the dissipation of turbulence kinetic energy

G_b is the buoyancy flux

S_k is the modulus of the mean rate-of-strain tensor

Modified k-epsilon model equations (applied in OpenFOAM)

Modified k-epsilon model equations incorporate the effect of Rapid Distortion Theory (RDT), but exclude the buoyancy factor, G_b . Its default values of model coefficients is similar as the standard k-epsilon model equations, with only the addition of the coefficient $C_{3,RDT}$. Table 1 shows the default values of model coefficients of both standard or modified version k-epsilon model equations. Eq. 9 and 10 shows the equations of modified k-epsilon model, which are equation of turbulence kinetic energy and equation of dissipation rate of turbulence kinetic energy respectively [18].

Table 1: Default value of the coefficients of k-epsilon model equations

Coefficients	Value
C_μ	0.09
C_1	1.44
C_2	1.92
$C_{3,RDT}$	0
σ_k	1
σ_ϵ	1.3

$$\frac{D}{Dt}(\rho k) = \nabla \cdot (\rho D_k \nabla k) + G_k - \frac{2}{3} \rho (\nabla \cdot u) - S_k \quad (9)$$

$$\frac{D}{Dt}(\rho \epsilon) = \nabla \cdot (\rho D_\epsilon \nabla \epsilon) + \frac{C_1 G_k \epsilon}{k} - \left(\frac{2}{3} C_1 - C_{3,RDT} \right) \rho (\nabla \cdot u) \epsilon - C_2 \rho \frac{\epsilon^2}{k} + S_\epsilon \quad (10)$$

Boundary conditions

Table 2 shows the boundary conditions (BC) of each parameter of this study using PIMPLE algorithm and k-epsilon turbulence model. Dirichlet BC and Neumann BC were applied alternately at inlet and outlet to optimize the flow continuity and stability [20]. The fixedValue BC was applied at velocity inlet so zeroGradient BC was applied at velocity outlet to make sure the flow is initiated and being continuous appropriately based on the initial conditions and Navier-Stoke equations. The same goes to the pressure inlet and outlet. However, the pressure inlet was set as zeroGradient while pressure outlet was set as fixedValue to avoid the contradiction between velocity and pressure fields. For turbulence parameters, the inlet BCs specialized for k-epsilon turbulence model was applied, and zeroGradient was applied at the outlet. Equally important, for velocity and pressure fields, noSlip BC was applied at solid wall. For turbulence parameters, k-epsilon wall function was applied. Uniquely, since only symmetrical quarter of the elevator system was modeled in this study, symmetry BC was used at the symmetry faces [7].

Table 2: Boundary conditions of each parameters.

	Inlet	Outlet	Car and shaft wall	Car and shaft symmetry region
Velocity, u (m s⁻¹)	fixedValue	inletOutlet	noSlip	symmetry
Kinematic static pressure, p (m² s⁻²)	zeroGradient	fixedValue	zeroGradient	symmetry
turbulence kinetic energy, k (m² s⁻²)	turbulentIntensity KineticEnergyInlet	inletOutlet	kqRWallFunction	symmetry
dissipation rate of turbulence kinetic energy, ε (m² s⁻³)	turbulentMixingLengthDissipationRateInlet	inletOutlet	epsilonWallFunction	symmetry

Results and discussion

Figure 3, 4, 5 and 6 show the development of 10 m s⁻¹ airflow until a relatively stable state. These figures show that the airflow was squeezed to pass through the narrow gap between the shaft and car side wall, from the upstream end (base) of the car until the downstream end (top) of the car. This causes the air velocity immediately upstream of the car decreased, air velocity within the narrow gap increased, and air velocity downstream the car decreased as shown in figure 3 and 7 (a). Conversely, the kinematic static pressure immediately upstream of the car increased, air velocity within the narrow gap decreased, and air velocity downstream the car increased as shown in figure 4. This is due to the blockage effects of the car within the shaft, which were reported by past studies [5,7,11]. The airflow disturbance creates the turbulence eddies, with increasing size of the turbulent wake region located downstream of the car (blockage of the flow). This turbulent wake region has a lower velocity magnitude but higher velocity fluctuation (turbulence). Nonetheless, in figure 3 and 4, fluctuation in velocity and pressure field of the airflow was observed even after reached the relatively stable state. This is caused by the persisting turbulent eddies within the airflow especially in the wake region. The same phenomenon was reported by Timo [20].

Based on figure 5, the turbulence kinetic energy increases with the flow development due to the increasing velocity and its fluctuations. Based on figure 6, the dissipation rate of turbulence kinetic energy, ϵ do not show significant changes, it is relatively higher at the region between the lower side wall of the car and the shaft during the early time, but remain relatively constant at a lower value until the end of the simulation. This is because initially, when the flow is developing and not stable, the turbulence kinetic energy was used to fluctuate the viscous stresses in resisting deformation of the fluid by the fluctuating strain rates. After the flow reached its constant velocity and was fully developed with a larger turbulence or eddy length scale, the loss of its turbulence kinetic energy became negligible compared to its high turbulence kinetic energy [21].

Insight into the turbulence aspect and turbulence model used, as the air velocity and Reynolds number increase, small eddies are produced in the airflow. However, this spatially (small size) and temporally (exist only for short period) small scale oscillations is computationally not worth to be resolved using ordinary Navier-Stokes equation, since the airflow velocity (u) over time contains only small local oscillations (u') that can be solved in time-averaged way, which is less computational costly. Moreover, these small local oscillations appear only in the wake region, instead of the high air velocity region that is significant to be studied [1]. So RANS was used in most of the studies to average up the air velocity along with the simulation time steps, assuming the simulation period is sufficiently long to capture the turbulent fluctuations. Based on figure 5 and 6, the simulated results show only medium size eddies, showing a smaller wake region and high-velocity region. The same results were reported by the studies using RANS k-epsilon by Li-qun [4], as well as RANS k-omega by Wang [5] and Ling [7]. This simulation approach is sufficient for the study that aims to study the average flow patterns and parameter values as an initial guess to guide the experimental study. For the studies simulated using the DES, all sizes of eddies were simulated, showing a larger wake region and high-velocity region due to a higher fidelity especially for the near-wall region Nishant Singh [22]. This type of simulation approach is required for the studies that aim to simulate all sizes of eddies and focus at the near-wall region, which will then be validated by experimental study.

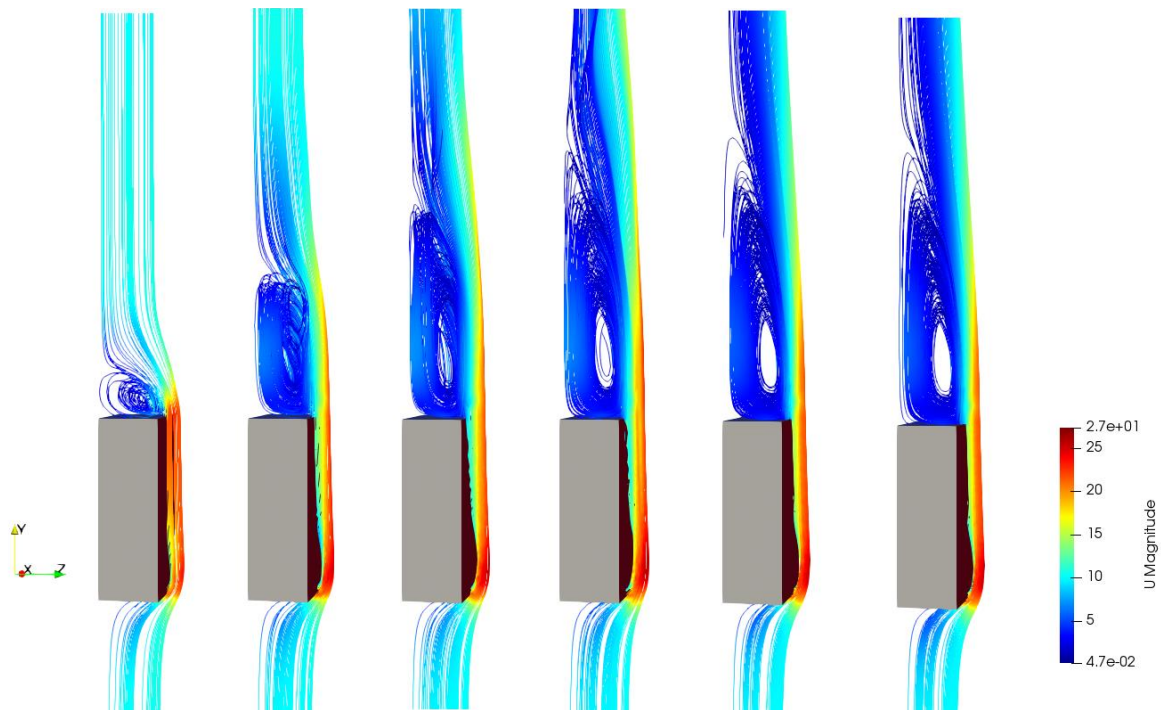


Fig. 3. Streamline plot of velocity field from the first until the last time steps of simulation.

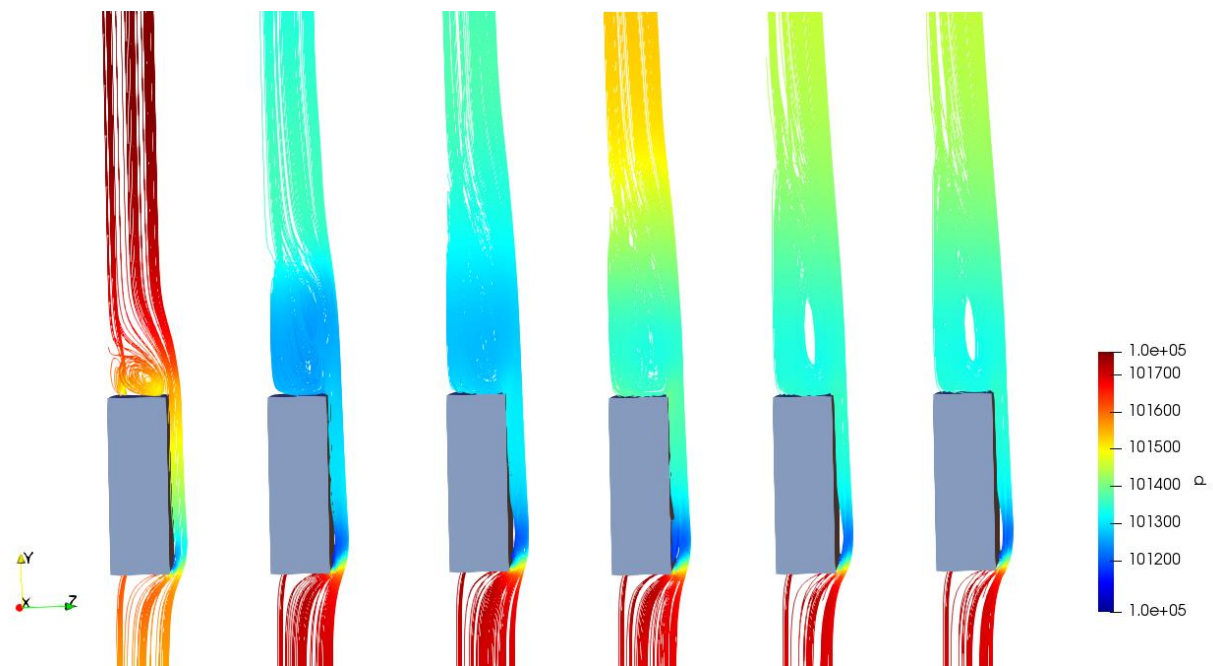


Fig. 4. Streamline plot of kinematic static pressure field from the first until the last time steps of simulation.

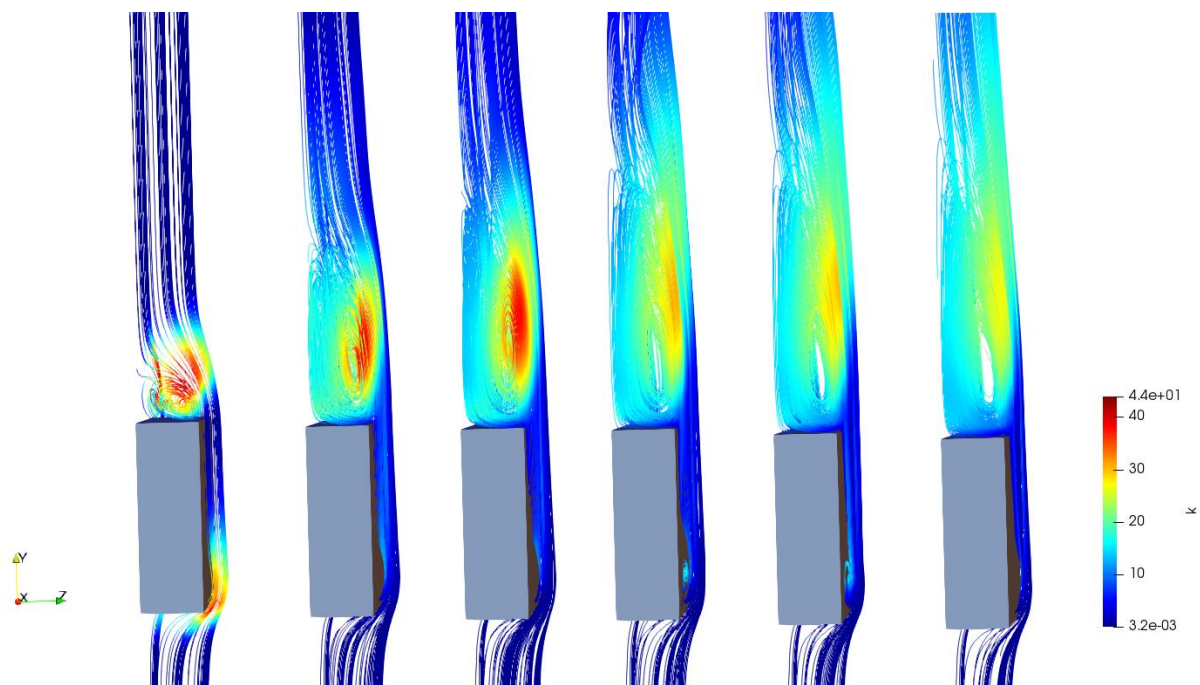


Fig. 5. Streamline plot of turbulence kinetic energy field from the first until the last time steps of simulation.

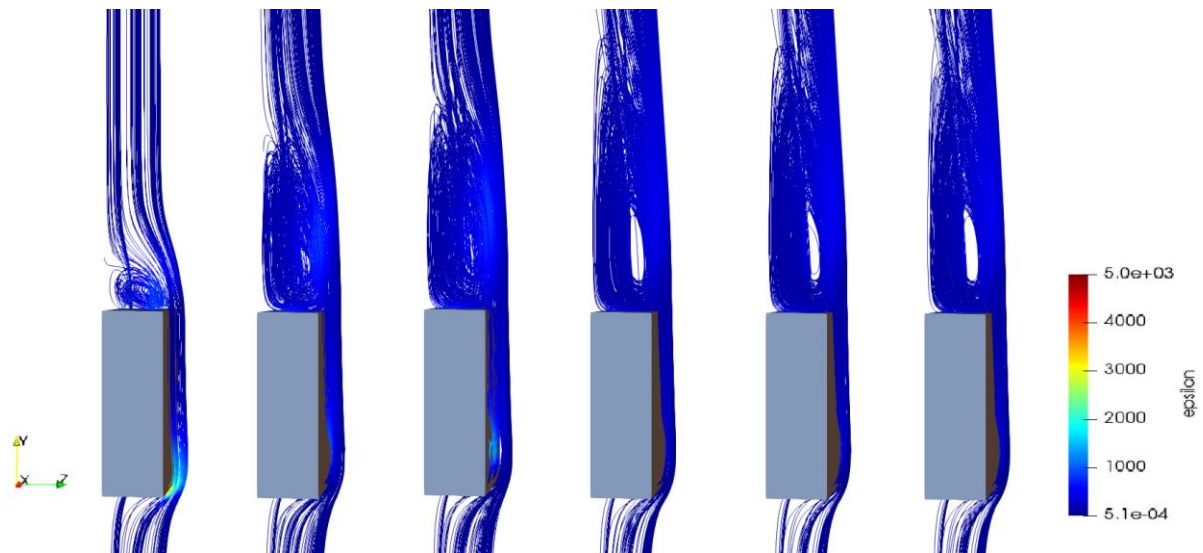


Fig. 6. Streamline plot of dissipation rate of turbulence kinetic energy field from the first until the last time steps of simulation.

Figure 7 (a) shows the velocity contour during stable state of the airflow, with its range of the airflow velocity in directions x, y, and z from 0.0470384 to 27.3158 m s⁻¹. Figure 7 (b) shows the significant measurement points (white dots) for experimental study, which has high velocity and low turbulence.

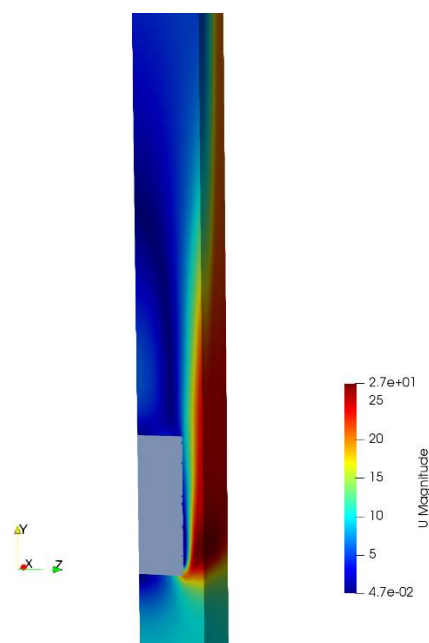


Fig. 7 (a). Velocity contour of the airflow

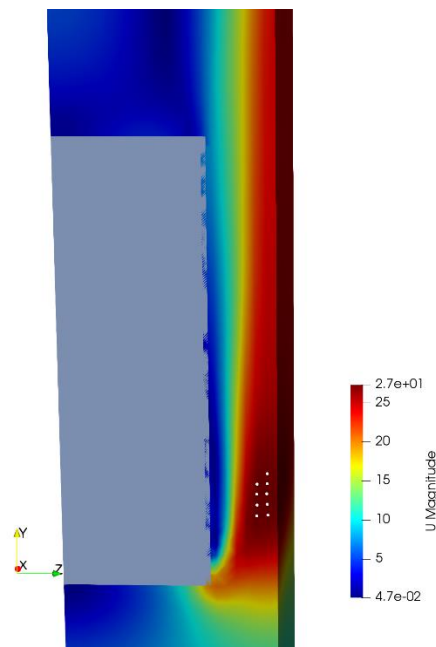


Fig. 7 (b). Points with high velocity and low turbulence

Conclusion

Generally, this study discussed the principles and working mechanisms of the PIMPLE algorithm, k-epsilon turbulence model, as well as the boundary conditions for this approach combination. Overall, the results show a good agreement with the theories of fluid mechanics and physics. Thus, the numerical results are reliable to be used as an initial guess input for

further study using DES approach, as well as reference data for the experimental study. Moreover, significant points with high velocity and low turbulence were determined in order to reduce the unnecessary measurement points in an experimental study.

Acknowledgments

This project is funded by the Trans-disciplinary Research Grant Scheme (TRGS) Ministry of Education Malaysia (203/PTEKIND/67610003) for the financial support in relation to this project. The first author would like to thank Universiti Sains Malaysia (USM) Fellowship scheme for the financial support related to her doctorate study.

References

- [1] W. Frei, Which Turbulence Model Should I Choose for my CFD Application?, COMSOL Blog. (2017) 1–8. <http://www.comsol.com/blogs/which-turbulence-model-should-choose-cfd-application/>.
- [2] C. D Scott-Pomerantz, The K epsilon model in the theory of turbulence, University of Puttsburgh, 2004.
- [3] K. Rantanen, Flow effects on ride comfort in super high speed elevators, Helsinki University of Technology, 2002.
- [4] S. Li-qun, L. Ying-zheng, J. Si-yu, C. Zhao-min, Numerical simulation of unsteady turbulent flow induced by two-dimensional elevator car and counter weight system, J. Hydrodyn. 19 (2007) 720–725. https://ac.els-cdn.com/S1001605808600098/1-s2.0-S1001605808600098-main.pdf?_tid=395cab6c-e5a6-11e7-bc77-00000aabb0f01&acdnat=1513788955_61c4ea55beaf05db0b16d5f066946f9e (accessed December 21, 2017).
- [5] X. Wang, Z. Lin, P. Tang, Z. Ling, Research of the blockage ratio on the aerodynamic performances of high speed elevator, 4th Int. Conf. Mechatronics, Mater. Chem. Comput. Eng. (ICMMCCE 2015). (2015) 3–7.
- [6] R. Wu, Z. Zhu, G. Cao, Computational fluid dynamics modeling of rope-guided conveyances in two typical kinds of shaft layouts, PLoS One. 10 (2015). doi:10.1371/journal.pone.0118268.
- [7] Z. Ling, P. Tang, X. Wang, Z. Lin, D. Tang, Research on the drags of high speed elevator with different height diversion cover, (2015) 2366–2370.
- [8] N. Singh, S. Kaczmarczyk, T. Ehrl, An analysis of airflow effects in lift systems, in: Lift Escalat. Symp. 7, 2017: pp. 2052–7225.
- [9] M. Pierucci, M. Frederick, Ride quality and noise in high speed elevators, in: Acoust. 08 Paris, San Diego State University, 2008: pp. 1615–1620.
- [10] A. Lehtinen, G. Roivainen, J. Tanttari, Aero-vibro acoustic noise prediction for high speed elevators, (2017) 317–322.
- [11] S. Mirhadizadeh, S. Kaczmarczyk, Tongue, H. Al-Jelawy, P. Feldhusen, W. Delk, K. Anderson, F. Dudde, Modelling and computer simulation of aerodynamic interactions in high-rise lift systems, in: Symp. Lift Escalat. Technol., The CIBSE Lifts Group, 2015: p. 117-127.
- [12] Syney Elevator, Study and improvement on aerodynamic of high-speed elevator, (2014). http://blog.sina.com.cn/s/blog_eb15d7810102v6h7.html.
- [13] P. Lotfabadi, High-rise buildings and environmental factors, Renew. Sustain. Energy Rev. 38 (2014) 285–295. doi:10.1016/j.rser.2014.05.024.
- [14] DHSS, Atmosphere and Gas Laws, 4th ed., Alaska Department of Health and Social Services, 2005.
- [15] S. Mijorski, S. Cammelli, Stack Effect in High-Rise Buildings : A Review Stack Effect

- in High-Rise Buildings : A Review, *Int. J. High-Rise Build.* 5 (2016) 327–338.
- [16] CFD Online, Introduction to turbulence / Turbulence kinetic energy, Cfd-Wiki. (2018) 1–13. https://www.cfd-online.com/Wiki/Introduction_to_turbulence/Turbulence_kinetic_energy#Transport_or_divergence_terms.
 - [17] CFD-Wiki, Turbulence kinetic energy, CFD-Wiki Turbul. Parameters. (2018). https://www.cfd-online.com/Wiki/Turbulence_kinetic_energy.
 - [18] OpenFOAM Documentation, k-epsilon Model equations OpenFOAM, (2018) 2–4. <https://www.openfoam.com/documentation/cpp-guide/html/guide-turbulence-ras-k-epsilon.html>.
 - [19] Wikipedia, Turbulence kinetic energy, Turbul. Parameters. (2018) 3–5. doi:10.1115/1.2912722.
 - [20] O. Timo, Aero-vibro acoustic simulation of an ultrahigh speed elevator, Tampere University of Technology, 2015.
 - [21] F. Cfd-wiki, CFD Online, Introduction to turbulence / Turbulence kinetic energy, Cfd-Wiki. (2018) 1–13. https://www.cfd-online.com/Wiki/Introduction_to_turbulence/Turbulence_kinetic_energy#Transport_or_divergence_terms.
 - [22] N. Singh, S. Kaczmarczyk, T. Ehrl, An analysis of airflow effects in lift systems An Analysis of Airflow Effects in Lift Systems, *Lift Escalat. Symp.* 7 (2017) 2052–7225. <https://liftsymposium.org/download/LiftandEscalatorSymposiumProceedings2017.pdf> (accessed December 21, 2017).

PREPARATION, THERMAL AND MECHANICAL PROPERTIES OF MYCELIUM AS BIO-MATRIX IN PRODUCTION BIO-COMPOSITE

Mohammad Aliff Shakir¹

Baharin Azahari²

Yusri Yusup¹

Mohd Firdaus Yhaya³

Ali Salehabadi¹

Mardiana Idayu Ahmad^{1*}

¹Environmental Technology Division, School of Industrial Technology, Universiti Sains Malaysia, 11800 Gelugor, Pulau Pinang, Malaysia

²Bio-resource, Paper and Coatings Technology Division, School of Industrial Technology, Universiti Sains Malaysia, 11800 Gelugor, Pulau Pinang, Malaysia

³Biomaterial and 3D Imaging (BioM3D) Laboratory, School of Dental Sciences, Universiti Sains Malaysia, 16150 Kubang Kerian, Kelantan, Malaysia

*(Email: drmia707@gmail.com)

Abstract: *The objective of this work is to access the possibility of mycelium as a new reinforcement and inexpensive bio-matrix in production of bio-composite board. In this study, mycelium from different inoculation time has been obtained. The potential bio-matrix mycelium has been produced at different conditions and characterized by Fourier Transform Infrared (FTIR) spectroscopy, Thermogravimetry/Differential Thermal Analysis (TG/DTA), Differential Scanning Calorimetry (DSC), and flexural strength test. The FTIR spectra indicate that the chitin content in the mycelium increases with prolonged inoculation time. The TGA and DSC thermograms reveal that the thermal stability and glass transition (T_g) temperature of chitin in mycelium also improve with prolonged inoculation time. Finally, the effect of pressing time – from 20 min to 40 min - has shown improvement from 1.82 MPa to 3.91 MPa in the flexural strength of mycelium bio-composite board.*

Keywords: *Mycelium, Thermal, Bio-matrix, Bio-composite*

Introduction

Mycelium is a natural composite polymer composed of chitin, cellulose, protein and etc. Currently, mycelium has become unprocessed bio-matrix that able to glue wood material together to form a wood composite product. Mycelium as bio-matrix able to provide advantages in wood composite properties such as low environmental impact, low cost, reduced energy consumption and biodegradability [1, 2]. Although, the application mycelium composite was limited due to its mechanical properties compare to conventional engineered composite, it's still capable to be used in semi-structural application such as furniture, decking and etc. [3-5].

A single fibrous mycelium is called as hyphae. The elongation of the mycelium occurs through the extension of the cell membrane and cell wall at the hyphal tip. The hyphae growth direction will stimulate toward the location of nutrient. This mechanism is responsible for firing thickening structure at the hyphae tip that contains enzyme chitin synthase and glucan synthase. Chitin is rigid polymers that synthesized at the cell wall of mycelium and provide mechanical properties to the mycelium. As the chitin and glucan are continuously assembled, it will form crosslink and make the hyphae become more rigid [6, 7].

Previous studies report that, the production of mycelium become higher when inoculate using glucose, starch, follow by maltose, and lactose [8]. Although starch is a polysaccharide, it still can be consume by the mycelium in short time by breaking down into simple sugar (glucose). Basically, the mycelium able to grow on long polysaccharide such as cellulose material by penetrate and secret digestive enzyme[9]. Studies report that the physical and mechanical properties of mycelium as binder in bio-composite was also depend on the time consuming during inoculation process [3, 10].

Mycelium has been used in production of bio-composite replacing synthetic composite board [11]. Mycelium is a natural binder that might have different properties during its inoculation process [12].

In the current study, Mycelium was successfully extracted from starch (corn grain), and used in production of bio-composite board. The bio-composites were formed by incubation of rubber wood sawdust (RWS) and mycelium. The inoculation time-consuming will not only affect the composite board performance, but also might involve in cost and time during the production process. Finally, the properties of mycelium as binder at different inoculation time, and board performance at different pressing time during production were studied.

Experimental

Materials

The mycelium tissue used in this study was inoculated and extracted from starch (corn grain). The corn grain was sterile and inoculated with mycelium. The grown mycelium after 14 and 45 days was carefully extracted for further analyses using sterile forcep.



Fig. 1 Mycelium tissue from starch (corn grain)

Fabrication of mycelium bio-composite

Mushroom substrate from rubber wood sawdust (RWS) was obtained and disintegrated. The RWS was measured and placed into a sterile plastic mould with dimension 21 cm × 15 cm × 0.6 cm at target density 0.80 g/cm³. Then, the RWS was allowed to incubate for 45 days to allow the mycelium to inoculate and bind the fibers together. The species of mycelium used in this study was *Pleurotus Ostreatus* (white rot fungi) [9]. Then, the fully grown mycelium bio-composite boards were withdrawn from the mould and the fabrication process was continued by using a hot pressed machine at temperature 130 °C with pressure of 5 MPa for 20 min and

40 min. The composite board was cut and tested according to Japanese Industrial Standard JIS A 5908:2003 [13].



Fig. 2 Bio-composite from RWS inoculated with mycelium before hot press.

Fourier transforms infrared spectroscopy (FTIR)

The functional groups in mycelium structure were analysed using Fourier Transform Infrared Spectroscopy (FTIR) analyser using Perkin Elmer 1600 Infrared spectrometer. Approximately 5 mg of mycelium of each type of sample with 95 mg of finely ground potassium bromide (KBr) were mixed together and pressed into pellets of about 1mm in thickness. The FTIR spectrum of each sample was then analyzed using the Nicole infrared spectrophotometer (Avatar 360 FTIR E.S.P) between wave numbers of 4000 cm^{-1} and 470 cm^{-1} with a resolution 4 cm^{-1} . The functional groups of the compound of each material were recorded. The significant transmittance peaks at particular wave numbers were measured using the “find peak tool” provided by Nicolet OMNIC 5.01 software

Thermogravimetry (TG)

The thermal stability of mycelium was conducted using a Perkin Elmer 7 thermogravimeter at the temperature range from $30\text{ }^{\circ}\text{C}$ to $800\text{ }^{\circ}\text{C}$, where, 5-10 mg of the samples were placed in an aluminium pan with a heating rate of $20\text{ }^{\circ}\text{C min}^{-1}$ under a nitrogen atmosphere.

Differential scanning calorimetry (DSC)

DSC thermogram of mycelium was characterized using Perkin-Elmer Pyris 7 under a nitrogen purge at a heating rate of $10\text{ }^{\circ}\text{C}$. Approximately 10 mg of a dry sample was transferred into a hermetic aluminium pan and sealed. The sample was placed inside the chamber and empty hermetic aluminium pan as a reference. The sample was heated at $10\text{ }^{\circ}\text{C/ min}$ in nitrogen flux from room temperature to $400\text{ }^{\circ}\text{C}$. The thermal profile of the sample was recorded continuously over the temperature and time intervals.

Flexural strength testing

Three replicated specimens for flexural strength test with the dimension of $5\text{ cm} \times 15\text{ cm}$ were prepared from mycelium bio-composite board for the evaluation of mechanical properties according to Japanese Industrial Standard JIS A 5908:2003 [13]. The mechanical analysis was

measured using an Instron Testing System Model UTM-5582 equipped with a load cell capacity of 1000kg.

Results and Discussion

Reaction profile

In each spectrum in Figure 3, remarkable peaks for all samples were observed at 1650 cm^{-1} and 1550 cm^{-1} , which are due to the amide I and II, respectively, in protein. In this peak, the intensities of mycelium after inoculation for 45 days is stronger than mycelium after 14 days. The strong intensities can be due to the larger production of protein in mycelium during inoculation process[14]. In addition, a sharp peak at around 1374 cm^{-1} can be assigned to the absorption of chitin. The intensity of this absorption band is higher for the mycelium inoculated for 45 days as compared to the sample inoculated for 14 days. From the spectra, it is obvious that the mycelium with longer inoculation time has a smaller peak at $3000\text{--}2800\text{ cm}^{-1}$ as compared to the mycelium inoculated for 14 days. This phenomenon can be attributed to the formation of lipid [11]. Moreover, the peaks at $1200\text{--}900\text{ cm}^{-1}$ can be due to the polysaccharide formation in mycelium [11]. A broad band at 3400 cm^{-1} are attributed to hydroxyl groups of carbohydrates, the main component of cell walls in mycelium [11].

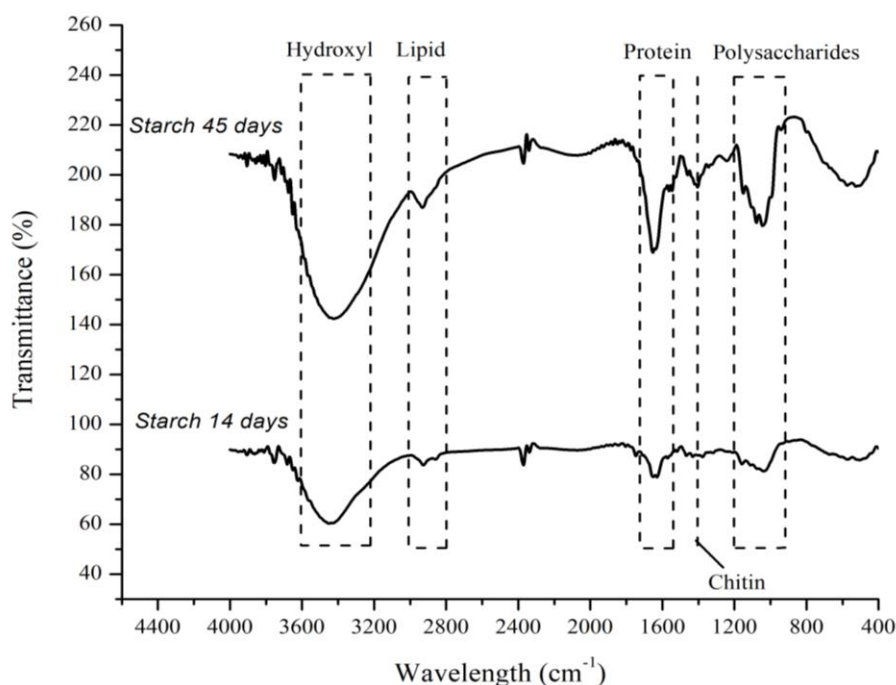


Fig. 3 FTIR spectra of mycelium at different inoculation days

Thermal Stability

The TGA-DTA thermograms of the samples were investigated in order to study the thermal decomposition profile. Figure 4 shows the thermal profile of mycelium, where shows three consecutive thermal decomposition steps. However, there are few differences between thermal degradation characteristic of mycelium inoculated for 14 days and 45 days. The mycelium inoculated for 45 days possessed a higher thermal degradation ($255\text{ }^{\circ}\text{C}$) as compared to mycelium inoculated for 14 days ($237\text{ }^{\circ}\text{C}$). Thus, it reveals that the mycelium inoculated for 45 days possess better thermal stability.

In the TGA-DTA thermograms, the first degradation stage at $30\text{--}200\text{ }^{\circ}\text{C}$ is attributed to the evaporation of free and chemically bonded water. The second stage at thermal degradation

230- 350 °C can be presumably due to the organic constituents such as chitin, amino acid, lipid, and polysaccharide including polymerization, decomposition of acetylated and de-acetylated unit of chitin in mycelium. These large molecules began to degrade by breakdown into low molecular weight volatiles [5, 10, 15]. The high degradation temperature shows that the mycelium as bio-matrix is thermally stable to be used in production and processing of bio-composite. While at stage three (400 °C to 800 °C), a small mass releasing occurs for the samples, which is due to the formation of CO, CO₂, and H₂O as a result of char formation [16].

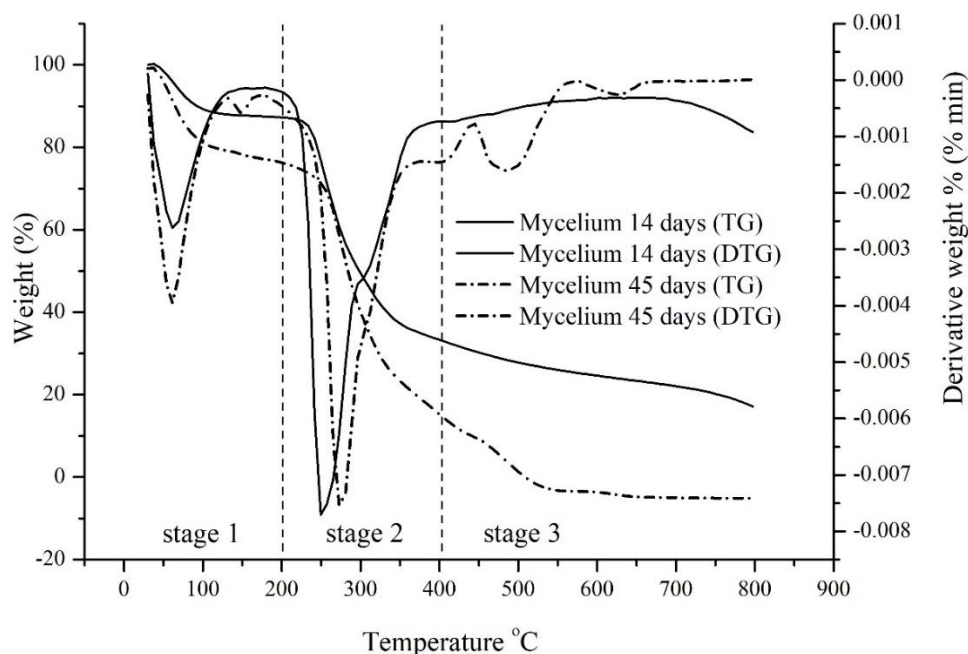


Fig. 4 TGA-DTA thermograms of mycelium inoculated with starch at different inoculation time.

Thermal Transitions

Thermal transitions of the samples were investigated using differential scanning calorimeter DSC (Figure 5). It could detect differences in fungal cell walls of mycelium at different inoculation days, which might have similar chemical compositions, but in different amount that led to increasing the mycelium density. Higher density of mycelium can be attributed to the appearance of a peak in the higher temperature range [14]. From the result, it can be observed that 45 days treated mycelium has higher density as compared to the mycelium after 14 days treatment. This variation in density is most probably due to the production of chitin. As mentioned before (FTIR and TG thermogram), the chitin content is higher in the mycelium inoculated for 45 days, where thermally more stable; chitin can impart rigidity into the mycelium, which can further affect density [11, 17]. It can be deduced that a multi-steps thermal differentiation in mycelium cell wall.

Figure 5 shows the DSC curves of mycelium inoculated with starch substrate at 14 days and 45 days. The DSC profiles of mycelium samples were exhibited a sharp endothermic peak in the temperature range from 95 °C to 126 °C (14 days) and 149 °C to 166 °C (45 days), which could be due to the glass transition (T_g) of chitin in mycelium [18]. The transition was due to the local relaxation of the backbone chain of chitin [19]. Both myceliums show the exothermic peaks; a weak transition peak (293 °C to 328 °C) for the sample after 14 days inoculation, and a broad transition peak (257 °C to 311 °C) for the sample with 45 days inoculation. The

exothermic peaks can be related to the decomposition of acetyl-glucosamine unit of chitin [20]. Chitin is thermally stable due to the high density and hydrogen bonding. The exothermic peak emerging from 388°C can be presumable from the evaporation of volatile low molecular product during depolymerisation [16].

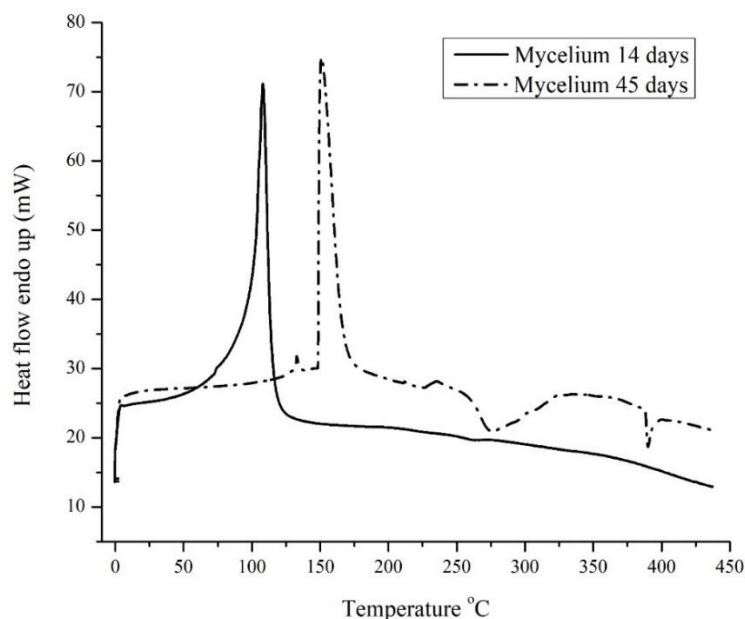


Fig. 5 DSC thermograms of mycelium after 14 and 45 days inoculation.

Flexural strength

The average value of the flexural strength of mycelium bio-composite board using RWS at pressing time 20 min and 40 min at 130 °C is shown in Figure 6. It can be clearly observed that (Figure 6) upon increasing the pressing time from 20 min to 40 min, the flexural strength increases.

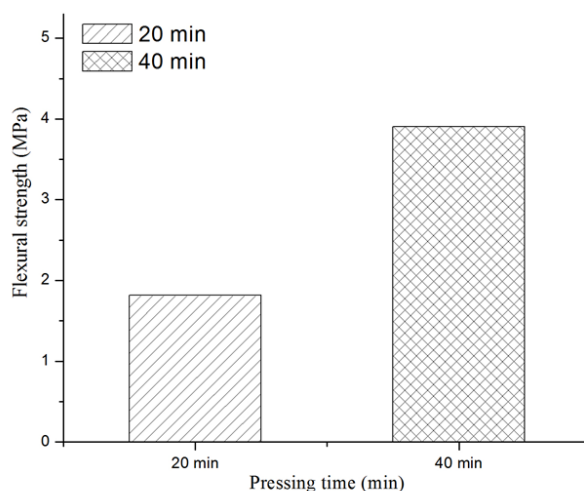


Fig. 6 Flexural strength versus pressing time.

The flexural strength of the composite board was measured to be around 1.82 MPa and 3.91 MPa for 20 min and 40 min hot pressing, respectively. The modulus of rupture for the composite board does not meet the requirement of the Japanese Industrial Standard. Both

composite boards had low bending strength because the mechanical properties of the board was attributed by self-hydrogen bonding between RWS fiber particles only [21].

The texture of RWS is highly pack, therefore the air cannot flow through the gaps between the particle, which further prevent the penetration of oxygen into the composite board [22]. The mycelium is unable to inoculate and bind the fibre together. Therefore, the bonding between fibres during board formation can be due to the pre-curing of lignin during pressing. The former can improve the interfacial bonding between fibres particles [23]. It was observed that upon increasing pressing time from 20 to 40 min, the pre-cure of lignin, and the flexural strength of the composite board were improved. An insufficient pressing time is one of the factors that can reduce the mechanical performance of the board due to high moisture content in the board as a result of incomplete hydrogen bonding between fiber particles [21].



Fig. 7 Internal part of the board

Conclusions

Primarily, mycelium was successfully extracted from starch (corn grain), and used in production of bio-composite board. The mycelium has been proved to be used as natural binder in production of mycelium bio-composite with different processing conditioned [11]. From the results, it can be concluded that the inoculation time affect the thermal, and mechanical properties of mycelium. Longer time inoculation was increased the chitin content, which can directly affect the properties of the product. Chitin is a polymer/binder, which can impart improved mechanical properties to the mycelium bio-composite. Since the effect on inoculation time was found to be able to improve the thermal properties of mycelium and increase the suitability of the mycelium to be used in production of bio-composite in extreme temperature. However, the effect of mycelium as binder in mycelium bio-composite board was found to be contradicted in this study. Based on the modulus of rupture and board image, it was found that the mycelium was unable to inoculate inside of the board, therefore fails to bind fibres. This is because the particle size of RWS use in this study was too fine which restricted the penetration of oxygen in the board as it is a crucial factor needed to the mycelium to inoculate. The production of bio-composite using mycelium as binder can be improved using an optimum particle size during inoculation and forming process.

Acknowledgement

This research was funded by the Universiti Sains Malaysia, under Graduate Research Assistant Scheme, 308/AIPS/415401.

References

- [1] Abhijith, R., A. Ashok, and C. Rejeesh, *Sustainable packaging applications from mycelium to substitute polystyrene: a review*. Materials Today: Proceedings, 2018. 5(1): p. 2139-2145.

- [2] Arifin, Y.H. and Y. Yusuf, *Mycelium fibers as new resource for environmental sustainability*. Procedia Engineering, 2013. **53**: p. 504-508.
- [3] Islam, M.R., et al., *Morphology and mechanics of fungal mycelium*. Scientific Reports, 2017. **7**(1): p. 13070.
- [4] Jiang, L., et al., *Manufacturing of biocomposite sandwich structures using mycelium-bound cores and preforms*. Journal of Manufacturing Processes, 2017. **28**: p. 50-59.
- [5] Jones, M., et al., *Thermal Degradation and Fire Properties of Fungal Mycelium and Mycelium-Biomass Composite Materials*. Scientific reports, 2018. **8**(1): p. 17583.
- [6] Reynaga-Peña, C.G., G. Gierz, and S. Bartnicki-Garcia, *Analysis of the role of the Spitzenkörper in fungal morphogenesis by computer simulation of apical branching in Aspergillus niger*. Proceedings of the National Academy of Sciences, 1997. **94**(17): p. 9096-9101.
- [7] Delpech, R., *Microecology: Using fast-growing filamentous fungi to study the effects of environmental gradients on the growth patterns of hyphae*. Journal of Biological Education, 2004. **38**(2): p. 90-95.
- [8] Barakat, O.S. and M. Sadik, *Mycelial growth and bioactive substance production of Pleurotus ostreatus in submerged culture*. Int. J. Curr. Microbiol. App. Sci, 2014. **3**(4): p. 1073-1085.
- [9] Taurachand, D. and K. Choi, *Mushroom growers' handbook 1. Oyster mushroom cultivation*. 2004.
- [10] Paulino, A.T., et al., *Characterization of chitosan and chitin produced from silkworm crysalides*. Carbohydrate Polymers, 2006. **64**(1): p. 98-103.
- [11] Haneef, M., et al., *Advanced Materials From Fungal Mycelium: Fabrication and Tuning of Physical Properties*. Scientific Reports, 2017. **7**: p. 41292.
- [12] Elsacker, E., et al., *Mechanical, physical and chemical characterisation of mycelium-based composites with different types of lignocellulosic substrates*. BioRxiv, 2019: p. 569749.
- [13] JIS, A., 5908: *Particleboards*. Japanese Industrial Standards. Tokyo, 2003.
- [14] Tsujiyama, S.-i., *Differential scanning calorimetric analysis of fruit-body and mycelium of Lentinula edodes*. Mycoscience, 1999. **40**(1): p. 69-72.
- [15] Jones, M., et al., *Waste-derived low-cost mycelium composite construction materials with improved fire safety*. Fire and Materials, 2018. **42**(7): p. 816-825.
- [16] Iqbal, M.S., et al., *Thermal studies of plant carbohydrate polymer hydrogels*. Carbohydrate polymers, 2011. **86**(4): p. 1775-1783.
- [17] Islam, M., et al., *Mechanical behavior of mycelium-based particulate composites*. Journal of materials science, 2018. **53**(24): p. 16371-16382.
- [18] Wang, Y., et al., *Crystalline structure and thermal property characterization of chitin from Antarctic krill (Euphausia superba)*. Carbohydrate polymers, 2013. **92**(1): p. 90-97.
- [19] Pizzoli, M., G. Ceccorulli, and M. Scandola, *Molecular motions of chitosan in the solid state*. Carbohydrate research, 1991. **222**: p. 205-213.
- [20] Nam, Y.S., et al., *Effect of the degree of deacetylation on the thermal decomposition of chitin and chitosan nanofibers*. Carbohydrate Polymers, 2010. **80**(1): p. 291-295.
- [21] Baskaran, M., et al., *Properties of binderless particleboard and particleboard with addition of urea formaldehyde made from oil palm trunk waste*. Journal of Physical Science, 2017. **28**(3): p. 151-159.
- [22] Pelletier, M., et al., *An evaluation study of mycelium based acoustic absorbers grown on agricultural by-product substrates*. Industrial Crops and Products, 2013. **51**: p. 480-485.



- [23] Boon, J.G., et al., *Physical and Mechanical Properties of Binderless Particleboard Made from Steam-Pretreated Oil Palm Trunk Particles*. Journal of Composites Science, 2019. 3(2): p. 46.
- [24] Appels, F.V.W., et al., *Fabrication factors influencing mechanical, moisture- and water-related properties of mycelium-based composites*. Materials & Design, 2019. **161**: p. 64-71.

EVALUATION OF THERMAL PERFORMANCE OF HYBRID NANOFLUIDS IN PULSATING HEAT PIPE

Muhammad Zufar¹

P. Gunnasegaran²

K.C. Ng³

Hemantkumar.B. Mehta⁴

¹Institute of Power Engineering, Universiti Tenaga Nasional, Putrajaya Campus, Jalan IKRAM-UNITEN, 43000, Kajang Malaysia (Email: zufarm@ymail.com)

²Institute of Power Engineering, Universiti Tenaga Nasional, Putrajaya Campus, Jalan IKRAM-UNITEN, 43000, Kajang Malaysia

³School of Engineering, Taylor's University, Lakeside Campus, No. 1 Jalan Taylors, Subang Jaya, Selangor, Malaysia

⁴Department of Mechanical Engineering, Sardar Vallabhbhai National Institute of Technology, Surat, Gujarat, India

Abstract: *The present article performed experimental and numerical study to determine the thermal performance of hybrid nanofluids in PHP under various heat input and filling ratio and to compare with water. Hybrid nanofluids possess greater thermal conductivity than water which leads to better heat transfer. Therefore, Al₂O₃-CuO hybrid nanofluid and SiO₂-CuO hybrid nanofluid with particle mass concentration of 0.1 % are used to evaluate the thermal performance of PHP with respect to start-up mechanism, steady state evaporator temperature, thermal resistance and formation of two phase flow. It is shown that the SiO₂-CuO hybrid nanofluid enhanced the thermal performance of PHP the most, consecutively Al₂O₃-CuO hybrid nanofluid and water. The optimal filling ratio of PHP is found to be 60 % at all the heat input and working fluid tested. Incorporating hybrid nanofluids in PHP have seen to expedite the start-up mechanism and also showed lower steady state evaporator temperature and thermal resistance values.*

Keywords: *Hybrid nanofluid, PHP, Thermal performance*

Introduction

Consumers demand for a high performance electrical device to cater for their daily life needs. These high performance devices produce high heat flux which demands an efficient heat transfer device. These past several decades, the advancement of the manufacturing and semiconductor industry have made it possible to produce millions transistors on a microchip. This leads to high heat flux dissipation out of the microchip when being used in an electrical device. Heat pipes is one of famous alternatives that have been used tremendously in removing high heat flux out of electrical devices. Pulsating heat pipes (PHPs) is a unique kind of heat pipes owing to its wickless structure. PHP is very different than the conventional heat pipes. This is because the transportation mechanism in PHP relies solely on the temperature and pressure indifference which are powered thermally. Therefore, a mechanical pump is not required in PHP. It consists of three sections namely the condenser, adiabatic and evaporator. Groeneveld et al. [1] conduct experimental study on orientation of a six turns PHP filled with methanol as the working fluid. The experiment is evaluated with constant filling ratio of 40 % for all the study parameters. It is shown that PHP presented best thermal resistance of 0.48 K/W at 100 W heating power when in a vertical orientation. The horizontal orientation exhibited the worst thermal resistance for PHP. On the other hand, Nazari et al. [2] perform an investigation using

graphene oxide nanofluid in PHP. The experimental study is conducted with different nanofluid mass concentrations of 0.25 % to 1.5 %. From the results, PHP charged with lower nanofluid concentration exhibited the best thermal resistance value. It is also observed that addition of nanoparticles in base fluid enhance the heat transfer by up to 40 % in relative to water as the working fluid. Nanofluids have been used vastly by researchers due to high thermal conductivity of the nanoparticles which help to improve the thermal performance of a heat transfer device. However, nanofluid characteristics are reported does not meet all the desired traits of an efficient working fluid [3]. Recently, hybrid nanofluids have grabbed attention due to the ideal thermal properties that can meet for specific heat transfer application [4]. Hybrid nanofluid is defined by a base fluid that are dispersed with two or more kind of nanoparticles. Sundar et al. [5] performed investigations on the enhancement of a circular tube using MWCNTs-Fe₃O₄/water hybrid nanofluid. The author reported the hybrid nanofluids improved in thermal performance in comparison to the single particle nanofluids. Study on hybrid nanofluid in heat pipes are still very few in the literature [6–9], hence the present research conducted an experimental and numerical investigation on PHP charged with water based Al₂O₃-CuO hybrid nanofluid and SiO₂-CuO hybrid nanofluid with mass concentrations of 0.1 % and to compare with water.

Methodology

Schematic diagram of PHP

The schematic diagram of experimental setup is depicted in Fig. 1. It consists of computer, data logger, vacuum pump, control panel, and power supply. These equipment are used in order to conduct the experimental investigations on a four turns PHP which is made up of copper material. The inner and outer diameter of PHP are given by 2 mm and 3 mm respectively. The evaporator region is equipped with five heating coils for heating purposes. The adiabatic section is insulated with glass wool to ensure no heat loss, while the condenser section is kept exposed to room temperature with natural convection. Seven K-type thermocouples with accuracy of ± 0.5 °C are installed at each sections in order for determination of temperature. Vacuum pump is initially used to create vacuum pressure in PHP up to 0.01 mbar. Then the charging valve is utilized to inject the working fluid into PHP. Control panel is adjusted in order to manipulate the heating power supply to the evaporator region. The data logger is used to collect data from the thermocouples at each sections and send it to the computer every 2 seconds. The heating power is increased once the temperatures given at each sections reaches steady state. The present experimental investigations of PHP is conducted using Al₂O₃-CuO hybrid nanofluid, SiO₂-CuO hybrid nanofluid and water under heat input of 10 W to 100 W at 50 % to 60 % filling ratio of PHP.

Preparation of hybrid nanofluids is firstly conducted using ultrasonic bath with duration of 360 minutes. Firstly, a magnetic stirrer is utilized by preparing individual single particle nanofluid with duration of 30 minutes. Then both single particle nanofluid is combined together and stirred again in the magnetic stirrer for another 30 minutes before going through the ultrasonication process. For example, to prepare Al₂O₃-CuO hybrid nanofluid, Al₂O₃ nanofluid and CuO nanofluid is prepared separately before mixed it together to form the hybrid nanofluid. Equation (1) is used to prepare the hybrid nanofluids at particle mass concentration of 0.1 %. Sedimentation analysis is conducted to evaluate the stability of the hybrid nanofluids and it is found that the hybrid nanofluids remain homogenous for up to 7 days without sedimentation.

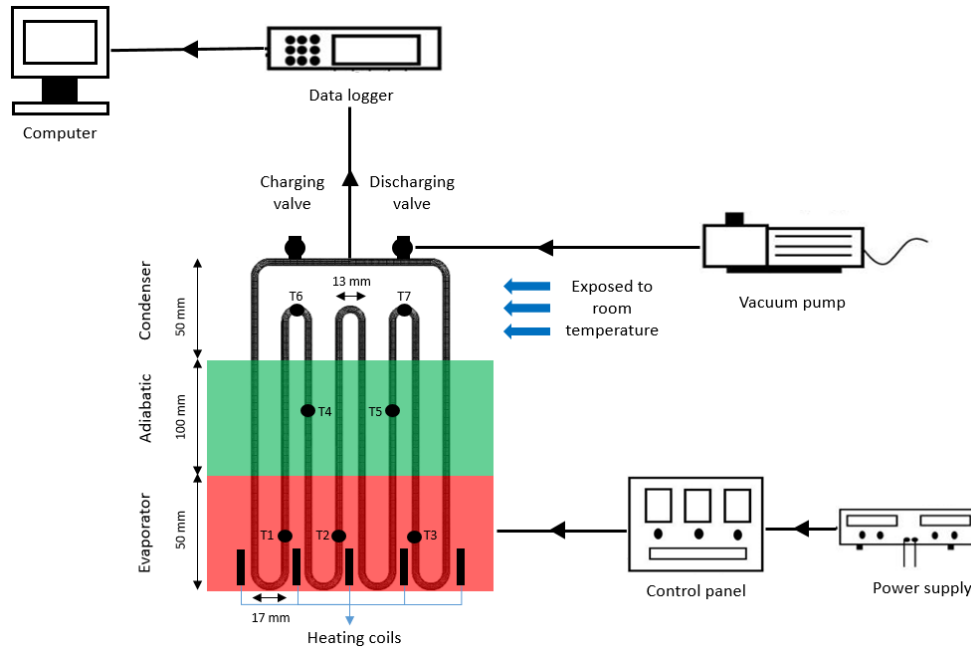


Fig. 1. Schematic diagram of experimental setup

$$\%mass \cdot concentration = \frac{W_{np}}{W_{bf} + W_{np}} \times 100 \quad (1)$$

Numerical methodology

The present numerical study is conducted using ANSYS Fluent software. A 3D four turns PHP is modelled according to the dimensions of the experimental rig. K-epsilon turbulence model is incorporated and the simulations is treated as transient. PISO is used for pressure velocity coupling because of it suitability for transient flow problem. Lee's model is utilized in consideration of the evaporation and condensation phenomena in PHP. VOF model is employed since PHP is a two-phase flow problems and it is able to track the interfaces of two or more immiscible phases. Equations (2) and (3) are solved in the simulations to ensure the interfaces can be successfully traced. Hexahedral meshes in generated throughout the domain owing to accuracy of the end results as compared to tetrahedral meshes. Consideration of thermal conductivity and viscosity of hybrid nanofluids are taken in the present numerical study. The thermal conductivity and viscosity values of hybrid nanofluids are evaluated experimentally in the present study at temperature ranged of 50 °C to 80 °C as shown in Table 1. Time step of 10^{-4} seconds is used which gives a total simulation time of 6 seconds for all the study parameters.

$$\frac{1}{\rho_q} \left[\frac{\partial}{\partial t} (\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q v) \right] = S_{\alpha q} + \sum_{p=1}^n (\dot{m}_{pq} - \dot{m}_{qp}) \quad (2)$$

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

Temperature (°C)	Al ₂ O ₃ -CuO/water		SiO ₂ -CuO/water		Water	
	Thermal conductivity (W/m.k)	Viscosity (cP)	Thermal conductivity (W/m.k)	Viscosity (cP)	Thermal conductivity (W/m.k)	Viscosity (cP)
50	0.76	0.69	0.70	0.64	0.64	0.59
60	0.78	0.65	0.72	0.60	0.66	0.45
70	0.80	0.62	0.74	0.55	0.68	0.38
80	0.81	0.58	0.76	0.49	0.69	0.36

Results

Start-up mechanism

Start-up mechanism is an essential analysis conducted in determination of pulsations in PHP. The success heat transfer mechanism of PHP depends heavily on the onset pulsations. Start-up mechanism is defined as the start of two-phase liquid vapour slugs flow in PHP. When start-up has been achieved, the fluid will be able to flow passively in PHP due to the temperature and pressure difference in the condenser and evaporator section. Fig. 2 illustrates the start-up mechanism of PHP charged with water under heat input of 40 W at filling ratio of 50 % and 60 %. From the results, PHP filled with water have achieved pulsations under heating power of 40 W. The pulsations shown by PHP at 60 % filling ratio is more vigorous in relative to PHP at 50 % filling ratio. Furthermore, the temperature difference between the evaporator and condenser section for PHP at 60 % filling ratio exhibited much smaller gap as compared to PHP at 50 % filling ratio. Smaller temperature difference will lead to a better thermal resistance value of PHP. On the other hand, the steady state evaporator temperature of PHP at 50 % filling showed higher value at 151 °C, whereas PHP at 60 % filling ratio obtained lower steady state evaporator temperature of 135 °C.

Fig. 3 depicted the start-up of PHP charged with Al₂O₃-CuO hybrid nanofluid under heat input of 20 W at filling ratio of 50 % and 60 %. It is observed that the start-up period can be attained earlier when PHP is charged with Al₂O₃-CuO hybrid nanofluid as the working fluid in comparison to water. Moreover, the heating power required in order to start the pulsations in PHP charged with Al₂O₃-CuO hybrid nanofluid is lower at 20 W. The steady state evaporator temperature shown value of 117 °C and 115 °C for PHP at 50 % and 60 % filling ratio respectively. It is calculated, the steady state evaporator temperature is enhanced by 22 % when PHP charged with Al₂O₃-CuO hybrid nanofluid compared to water. This shows that the suspension of hybrid nanoparticles in the base fluid water have seen to result in the improvement of thermal performance of PHP. In addition, PHP with Al₂O₃-CuO hybrid nanofluid presented smaller gap of temperature difference between evaporator and condenser section in comparison to PHP filled with water.

Likewise, PHP charged with SiO₂-CuO hybrid nanofluid at filling ratio of 50 % and 60 % required lower heating power in order to start-up pulsations at 20 W as shown in Fig. 4. This is due to the SiO₂-CuO hybrid nanoparticles dispersion which augmented the thermal conductivity of the base fluid water. Due to this reason, the heat transfer can be greatly enhanced and more heat can be absorb and dissipated much efficiently. Furthermore, using hybrid nanofluids in PHP have promote more nucleation sites which are essential for bubbles growth and development in PHP. Therefore, the passive pulsations of liquid vapor slugs flow in PHP charged with hybrid nanofluids can be obtained quicker in comparison to water as the working fluid in PHP. The steady state evaporator temperature of PHP filled with SiO₂-CuO hybrid nanofluid is obtained at 105 °C which is lower by 10 °C and 30 °C in relative to Al₂O₃-CuO hybrid nanofluid and water respectively at 60 % filling ratio. Additionally, the

temperature difference between the evaporator and condenser section presented smaller gap for PHP incorporated with $\text{SiO}_2\text{-CuO}$ hybrid nanofluid in comparison to $\text{Al}_2\text{O}_3\text{-CuO}$ hybrid nanofluid and water. Based on the results shown from Figs. 2, Figs. 3 and Figs. 4, the temperature difference in between the evaporator and condenser region is smaller for hybrid nanofluids and exhibited lower evaporator temperature in comparison to water.

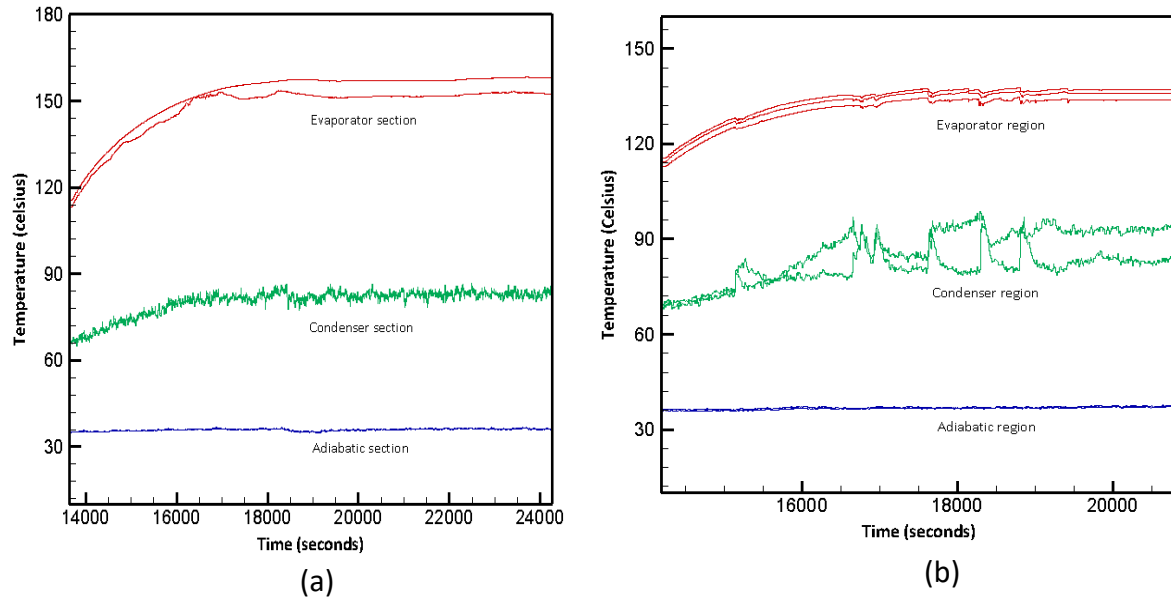


Fig. 2. Start-up mechanism of PHP charged with water under heat input of 40 W at filling ratio of (a) 50 % (b) 60 %

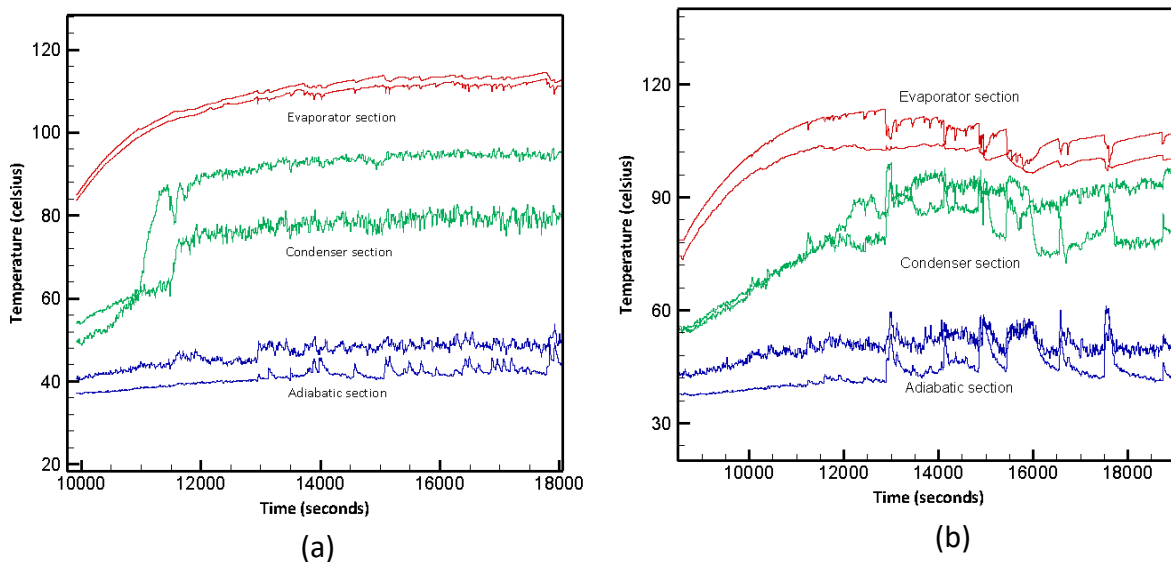


Fig. 3. Start-up mechanism of PHP charged with $\text{Al}_2\text{O}_3\text{-CuO}$ hybrid nanofluid under heat input of 20 W at filling ratio of (a) 50 % (b) 60 %

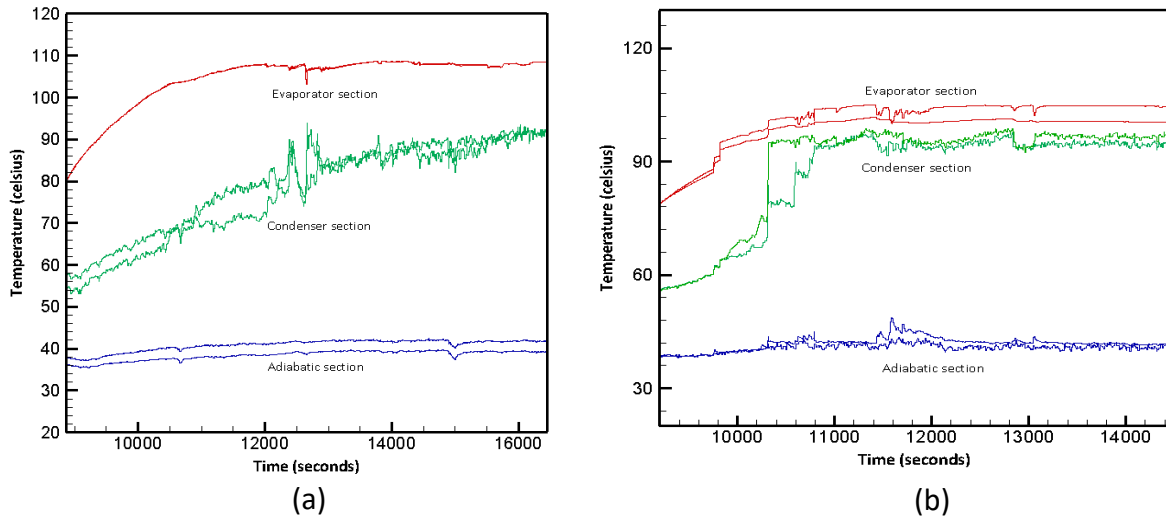


Fig. 4. Start-up mechanism of PHP charged with SiO₂-CuO hybrid nanofluid under heat input of 20 W at filling ratio of (a) 50 % (b) 60 %

Thermal resistance

The thermal performance of PHP in the present experimental investigation of PHP is determined by using thermal resistance. Equation (4) illustrates the thermal resistance equation, whereby T_e and T_c are the average steady state temperature of the evaporator and condenser sections respectively. The heat input supply to the evaporator section is denoted by Q .

$$R = (T_e - T_c) / Q \quad (4)$$

Fig. 5 depicts the thermal resistance of PHP charged with hybrid nanofluids and water under heating power of 10 W to 100 W at filling ratio of 50 % and 60 %. It is presented that the SiO₂-CuO hybrid nanofluid exhibited the lowest thermal resistance under all heat input and filling ratio tested compared to Al₂O₃-CuO hybrid nanofluid and water. The graph indicates that SiO₂-CuO hybrid nanofluid is the best working fluid for enhancement of thermal performance of PHP subsequently Al₂O₃-CuO hybrid nanofluid and water. Furthermore, it is also shown 60 % is the optimal filling ratio of PHP owing to the lower thermal resistance value in relative to PHP at 50 % filling ratio. The thermal resistance of PHP charged SiO₂-CuO hybrid nanofluid improved by 55 % as compared to water under heating power of 100 W at 60 % filling ratio. The lowest thermal resistance value is obtained at value of 0.27 °C/W for PHP charged with SiO₂-CuO hybrid nanofluid at heat input of 50 W at 60 % filling ratio. Al₂O₃-CuO hybrid nanofluid presented higher viscosity than SiO₂-CuO hybrid nanofluid, hence contributes to higher thermal resistance value. This is because the fluid flow in PHP is resisted by the shear forces existed in the walls of PHP, therefore heat cannot be efficiently absorbed and dissipated [10].

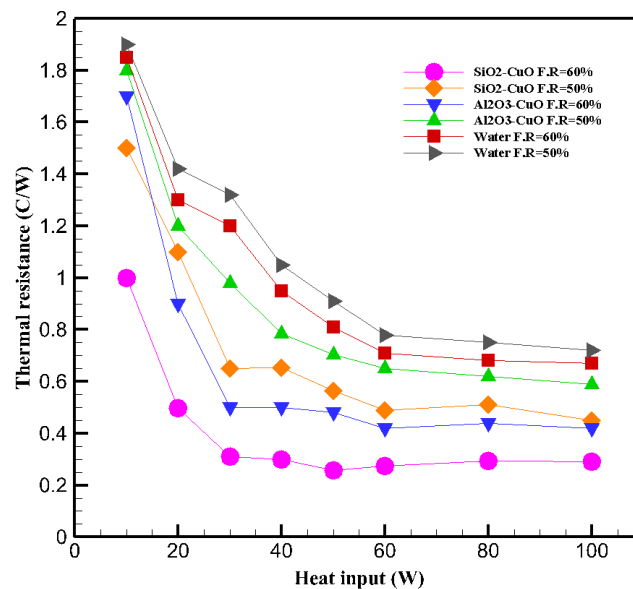


Fig. 5. Thermal resistance of PHP charged with hybrid nanofluids and water under heat input of 10 W to 100 W for 50 % and 60 % filling ratio

Numerical results on formation of bubbles in PHP

Numerical investigations is conducted for PHP charged with hybrid nanofluids and water under heat input of 100 W at 60 % filling ratio as shown in Figs. 6, 7, and 8. The volume fraction contour can help to give an insight on formation and development of vapour bubbles into vapour slugs in PHP. The red and blue colour in the volume fraction contour represent the vapour and liquid phase respectively. The simulation was treated as transient case with total simulation time of 6 s. The boundary condition for the evaporator section is supplied with constant temperature correspond to the steady state evaporator temperature obtained from the experimental study. Adiabatic and condenser section are supplied with zero heat flux and natural free convection with value of 25 W/m²k respectively. At simulation time of 0.1 s, it can be observed only tiny vapour bubbles appeared in the PHP charged with hybrid nanofluids and water. As time progressed, the vapour bubbles is seen to coalescence among each other which formed vapour slugs. It is shown that PHP charged with hybrid nanofluids attained faster vapour slugs formation at 0.5 s as compared to water at 1 s. This shows that consideration of augmentation of thermal conductivity of the hybrid nanofluids in the present numerical study have expedite the process of nucleate boiling in PHP. The evaporation of the hybrid nanofluids took place much quicker due to better heat transfer rate as compared to water. It is also shown that the vapour slugs formed into long vapour slugs at simulation time of 2 s, 3 s, and 4 s for working fluid of water, Al₂O₃-CuO hybrid nanofluid and SiO₂-CuO hybrid nanofluid respectively. A dry out phenomenon is seen to occur for PHP filled with water, while this event did not occur in PHP charged with hybrid nanofluids. This indicates that hybrid nanofluids is suitable for high heating power application, since it can dissipates heat at much greater rate than water. Whereby, PHP filled with water went through dry out phenomena because of accumulation of heat that are not being removed efficiently from the condenser section. The results and phenomenon observed in the present numerical investigations are also reported by previous researchers in the study of two phase flow in heat pipes [10–14]

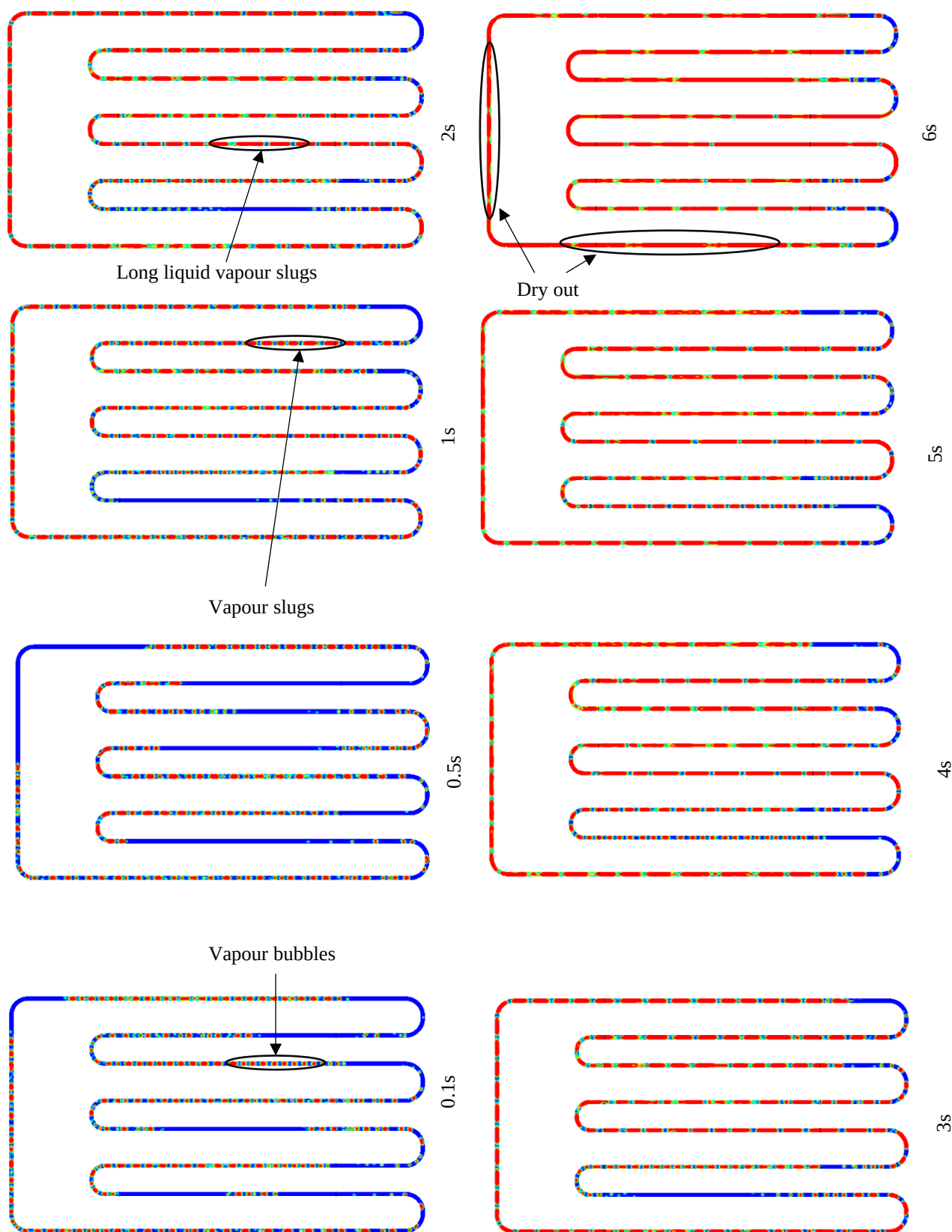


Fig. 6. Volume fraction contour of PHP charged with water under 100 W heat input at 60 % filling ratio

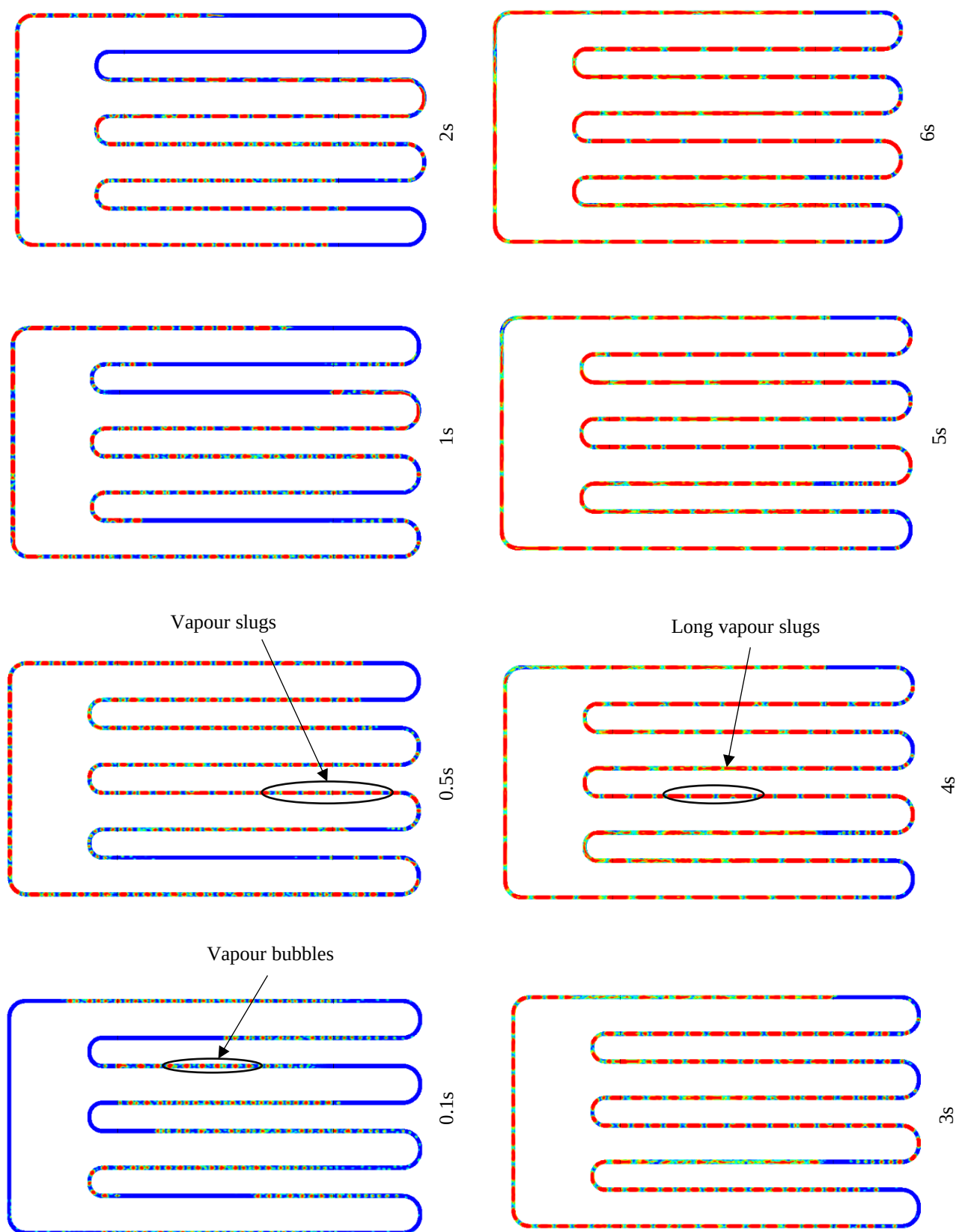


Fig. 7. Volume fraction contour of PHP charged with SiO₂-CuO hybrid nanofluid under 100 W heat input at 60 % filling ratio

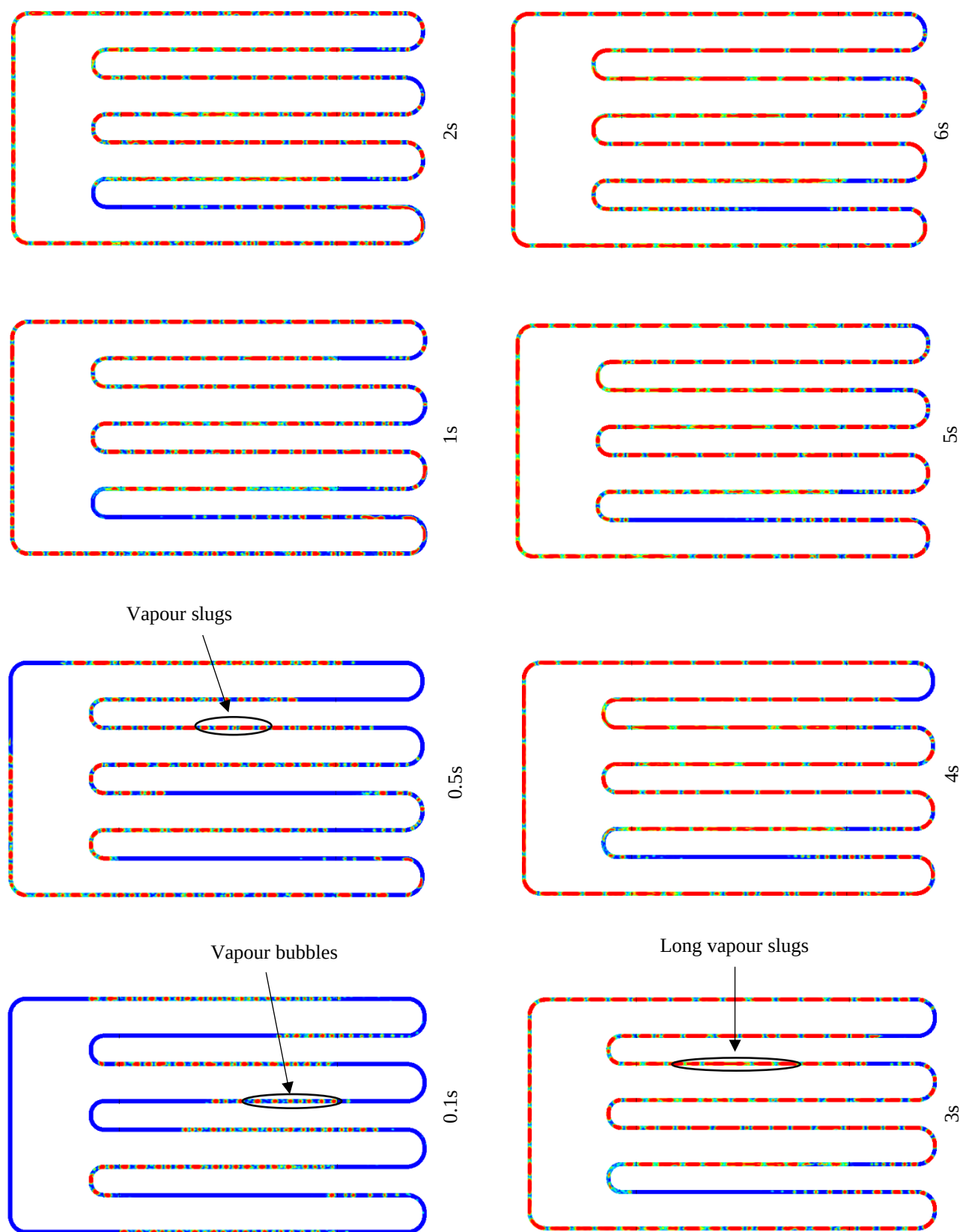


Fig. 8. Volume fraction contour of PHP charged with $\text{Al}_2\text{O}_3\text{-CuO}$ hybrid nanofluid under 100 W heat input at 60 % filling ratio

Conclusions

The present experimental and numerical investigations on PHP charged with hybrid nanofluids and water is evaluated in terms of the start-up mechanism, steady state evaporator temperature, thermal resistance, and two phase flow formation in PHP in which can be deduced as follows:

- The start-up mechanism can be achieved faster when hybrid nanoparticles is suspended in the base fluid water. Moreover, the steady state evaporator temperature is shown much lower for PHP charged with hybrid nanofluids than water.
- The temperature difference between the evaporator and condenser sections exhibited smaller gap when PHP is filled with hybrid nanofluids. The SiO₂-CuO hybrid nanofluid showed the smallest temperature gap in comparison to other working fluid.
- SiO₂-CuO hybrid nanofluid has the lowest thermal resistance value of 0.27 °C/W and presented an enhancement of 55 % in relative to water.
- It is shown that the Al₂O₃-CuO hybrid nanofluid has the highest thermal conductivity value, however the thermal performance of PHP is deteriorated due to its large viscosity value.
- 60 % is the optimal filling ratio of PHP for all the working fluid tested at all the study parameters. The temperature difference of evaporator and condenser sections exhibited smaller gap for PHP at 60 % filling ratio, hence leads to better thermal performance compared to PHP with 50 % filling ratio.
- In the numerical investigations, the dry out phenomena is observed when PHP is incorporated with water, whereas this phenomenon did not occur when PHP is filled with hybrid nanofluids. This designates that the hybrid nanofluids suitable for high heating power applications.

Acknowledgement

This study has been supported by Universiti Tenaga Nasional (UNITEN) internal grant: UNIIG 2018 (J510050887).

References

- [1] G. Groeneveld, H. J. Van Gerner, and W. W. Wits, "An experimental study towards the practical application of closed-loop flat-plate pulsating heat pipes," *THERMINIC 2017 - 23rd Int. Work. Therm. Investig. ICs Syst.*, vol. 2017-Janua, no. September, pp. 1–6, 2017.
- [2] M. A. Nazari, R. Ghasempour, M. H. Ahmadi, G. Heydarian, and M. B. Shafii, "Experimental investigation of graphene oxide nanofluid on heat transfer enhancement of pulsating heat pipe," *Int. Commun. Heat Mass Transf.*, vol. 91, pp. 90–94, 2018.
- [3] A. Selim, Ö. Aç, and B. Onur, "Experimental investigation on the viscosity characteristics of water based SiO₂ -graphite hybrid nano fl uids," vol. 97, pp. 30–38, 2018.
- [4] A. A. Minea, K. V. Sharma, S. Akilu, M. A. M.Said, and A. T. Baheta, "Properties of glycerol and ethylene glycol mixture based SiO₂ -CuO/C hybrid nanofluid for enhanced solar energy transport," *Sol. Energy Mater. Sol. Cells*, vol. 179, no. December 2017, pp. 118–128, 2017.
- [5] L. S. Sundar, M. K. Singh, and A. C. M. Sousa, "Enhanced heat transfer and friction factor of MWCNT – Fe₃O₄ / water hybrid nano fl uids ☆," *Int. Commun. Heat Mass Transf.*, vol. 52, pp. 73–83, 2014.
- [6] M. F. Nabil, W. H. Azmi, K. Abdul Hamid, R. Mamat, and F. Y. Hagos, "An experimental study on the thermal conductivity and dynamic viscosity of TiO₂-SiO₂

- nanofluids in water: Ethylene glycol mixture,” *Int. Commun. Heat Mass Transf.*, vol. 86, pp. 181–189, 2017.
- [7] K. A. Hamid, W. H. Azmi, M. F. Nabil, R. Mamat, and K. V. Sharma, “Experimental investigation of thermal conductivity and dynamic viscosity on nanoparticle mixture ratios of TiO₂-SiO₂ nanofluids,” *Int. J. Heat Mass Transf.*, vol. 116, pp. 1143–1152, 2018.
 - [8] A. Chaudhari and S. Y. Bhosale, “Performance Evaluation of Circular Heat Pipe using (CuO+BN)/H₂O Hybrid Nanofluid,” *Int. J. Eng. Manag. Res.*, vol. 6, no. 4, pp. 315–321, 2016.
 - [9] K. Swapnil, P. Dhananjay, Z. Gaurav, R. Anand, and D. P. Kamble, “Heat Transfer Enhancement of Circular Heat Pipe with Al₂O₃ BN/Water Hybrid Nanofluid,” *Int. J. Curr. Eng. Technol.*, vol. 4, no. 4, pp. 138–143, 2016.
 - [10] J. Qu and H. Wu, “Thermal performance comparison of oscillating heat pipes with SiO₂/water and Al₂O₃/water nanofluids,” *Int. J. Therm. Sci.*, vol. 50, no. 10, pp. 1954–1962, 2011.
 - [11] H. Goudmi, R. Benderradji, A. Beghidja, and T. Tayebi, “Numerical Study of Upward Vertical Two-Phase Flow Through an Annulus Concentric Pipe,” vol. 2, no. 2, pp. 187–206, 2019.
 - [12] S. Rudresha and V. Kumar, “CFD Analysis and Experimental Investigation on Thermal Performance of Closed loop Pulsating Heat pipe using different Nanofluids Experiments Apparatus and procedure .,” *Int. J. Adv. Res.*, vol. 2, no. 8, pp. 753–760, 2014.
 - [13] J. Venkata Suresh and P. Bhramara, “CFD Analysis of Multi turn Pulsating Heat pipe,” *Mater. Today Proc.*, vol. 4, no. 2, pp. 2701–2710, 2017.
 - [14] S. M. Pouryoussefi and Y. Zhang, “Analysis of chaotic flow in a 2D multi-turn closed-loop pulsating heat pipe,” *Appl. Therm. Eng.*, vol. 126, pp. 1069–1076, 2017.
 - [15] S. M. Pouryoussefi and Y. Zhang, “Analysis of chaotic flow in a 2D multi-turn closed-loop pulsating heat pipe,” *Appl. Therm. Eng.*, vol. 126, pp. 1069–1076, 2017.

COMPUTATIONAL METHODS FOR PREDICTING A PICO-HYDRO CROSS-FLOW TURBINE PERFORMANCE

Warjito¹
Budiarso²
Celine Kevin³
Dendy Adanta⁴
Aji Putro Prakoso⁵

¹Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Indonesia

²Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Indonesia

³Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Indonesia (Email: celinekevin.ck@gmail.com)

⁴Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Indonesia

⁵Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Indonesia

Abstract: Usually, to find out the maximum performance of a turbine, an experimental test is carried out using loading (preload) variations method. The high proportion of torque (τ) and runner rotation velocity (U) is an indication of maximum turbine performance. However, if there is mistaken in the shape of geometry it takes more time and funds. The correct solution is a prediction using the computational fluid dynamics (CFD) method. This study proposes the method to predict the maximum conditions of pico-hydro cross-flow turbines by comparing the preload acting on runners by CFD method. So, if something goes wrong in the design can be corrected immediately. The preloads variation in this study consists of 0 N·m, 30 N·m, 45 N·m, and 60 N·m with the head condition of 1 meter and mass flow of 10.5 kg/s. The CFD method with six-degree of freedom (6-DoF) was selected because the rotational turbine is one of the computational results not as boundary conditions. The model turbulence k - ϵ standard has been used to predict the turbulent flow. Based on results, transient data (torque and runner rotational velocity) obtained from computing is similar to testing, which fluctuates and becomes steady. Furthermore, used 45 N·m preload has more stable and had higher efficiency than the other. Turbine with 45 N·m preload produced 60.07% efficiency. This indicates that the turbine that is designed will work optimally at 45 N·m preload. Thus, CFD method with 6-DoF feature proposed to predict a pico-hydro cross-flow turbine performance can be an alternative before the turbine is manufactured to be tested or implemented.

Keywords: Computational, pico-hydro, crossflow turbine, six-dof, preload

Introduction

An estimated that 14% of the world population do not have access to electricity which is mostly in the developing country [1]. About 84% of those who do not have access to electricity are in rural areas and more than 95% of those are in developing countries [1]. The matter of energy resources is one of the problems faced in the procurement of electricity. Hence, the use of renewable energy is the solution to this problem [2]. There is a wide variety of renewable energy such as hydro, wind, solar, and other natural forces. Today, the most widely used renewable energy is hydropower [2].

Pico hydro is a hydropower plant that generates electricity under 5 kW [3]. This scale is suitable for the small-rural area that requires only a small amount of electricity daily. Compared to fossil fuel power plant, the pico-hydro gives us a low-cost electricity generation with the least pollution [4].

A pico-hydro cross-flow turbine is suitable to use in remote areas because of stable efficiency with a high deviation of discharge conditions, simple shape, and low investment cost compared than other types [5]. Subsequently, the cross-flow turbine has another advantage, such as less maintenance requirement, long life span, simple or less infrastructure, and economically friendly [6]. Thus, the selection of a cross-flow turbine for electrification in rural areas is considered appropriate because it has a simple design, modest efficiency, and it does not require high-head [7].

Cross-flow turbines are turbines that are used to convert kinetic energy of water to mechanical energy to rotate a shaft [4]. In contrast to other types of turbines, cross-flow turbines do not have radial or axial flow. In cross-flow turbines, the flow of water will cross the turbine blades.

To maximize the performance of cross-flow turbines, several methods can be used such as analytical, computational, and experimental method. Many experts have selected the computational fluid dynamics (CFD) method because it can represent a turbine's flow pattern in more detail than other methods [8]. To get the precision of predictions using CFD method, the model was used must similar to the real conditions [9]. In order for the model to be similar against real conditions, the density of material (moment of inertia) and loading (preload) of the runner should be the boundary conditions of the simulation. However, there has been no attention that discusses the moment of inertia and preload in the simulation of a pico-hydro cross-flow turbine. This study will focus on reviewing preload.

Preload is a load received by the runner to find out the maximum performance of a turbine. The greater the load, the greater the torque (τ) needed to turn the runner or generator. Usually, preload or loading is only done by experiment. Using the CFD method, this study will proposed methods to predict the maximum conditions of pico-hydro crossflow turbines by comparing the preload acting on runners. The CFD method was chosen because if something goes wrong in the design it can be corrected immediately.

Methodology

A solid-3D model was created in computer-aided design (CAD) software according to the available designs. The specific designs parameters shown in Table 1. Figure 1 shows the 3D model of the cross-flow turbine.

Table 1: Design Parameters

Design Parameter	Value
Number of blades, z	20
Outer diameter, D_o	180 mm
Inner diameter, D_i	117 mm
Blade's length, L	142 mm
Angle of attack, α	22°
Blade's inlet angle, β	42°

The computational method was obtained by using ANSYS® FLUENT 18.1™. In this study, a 2D transient domain was used to perform the simulation. The 2D domain was created in Ansys Design Modeler, with the thin surface feature. Multiphase model volume with implicit volume fraction parameters and body force equation was selected based on the condition that has water phase and air phase. The standard k- ϵ turbulent model also used in this simulation because of its reasonable accuracy for industry flow simulations [14]. The simulation also uses

a dynamic mesh setting with six degrees of freedom (6-DOF) user-defined function. The 6-DOF was chosen because it can represent the fluid dynamics and physics phenomenon on the cross-flow turbine impeller [15]. The equations are discretized using second-order upwind interpolation schemes in selected CFD packages and SIMPLE algorithms. The governing equations for k- ϵ are available in Eq. (1) for k and (2) for ϵ [16] :

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_K \quad (1)$$

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon \quad (2)$$

In this simulation, the main fluid used was water. So, in the inlet, the water volume fraction (α_w) value was 1 and the air volume fraction (α_a) value was 0, which mean there was no air in the inlet. On the contrary, the air volume fraction (α_a) in the outlet was 1. This simulation is carried out using a pressure inlet and pressure outlet as the boundary condition. The pressure at the inlet is set at 9810 Pa with the assumption the head height is 1 m. Simulations were carried out 4 times, with variations of preload of 0 Nm, 30 Nm, 45 Nm, and 60 Nm.



Fig. 1. 3D model of the cross-flow turbine

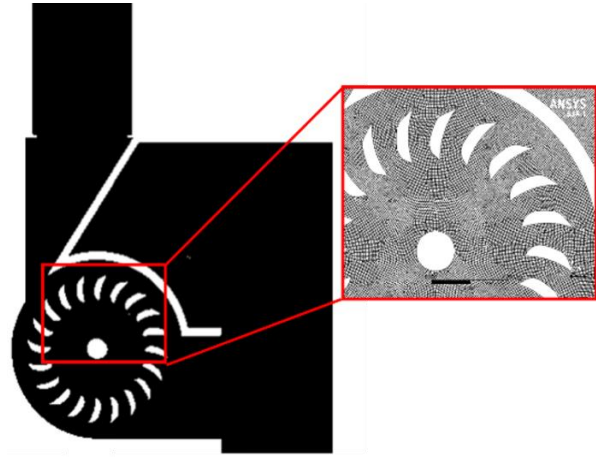


Fig. 2. 2D Domain mesh visualization – 122408 elements

A mesh independence study was executed using the single-phase – steady method. Mesh is an area where calculations will be made. Generally, the number of mesh is described as the number of elements. On the mesh independency test, variations in the number of mesh were used, namely 37589, 66079, 122408, and 230162 elements. The numbers of mesh were chosen to get a ratio value of 2.

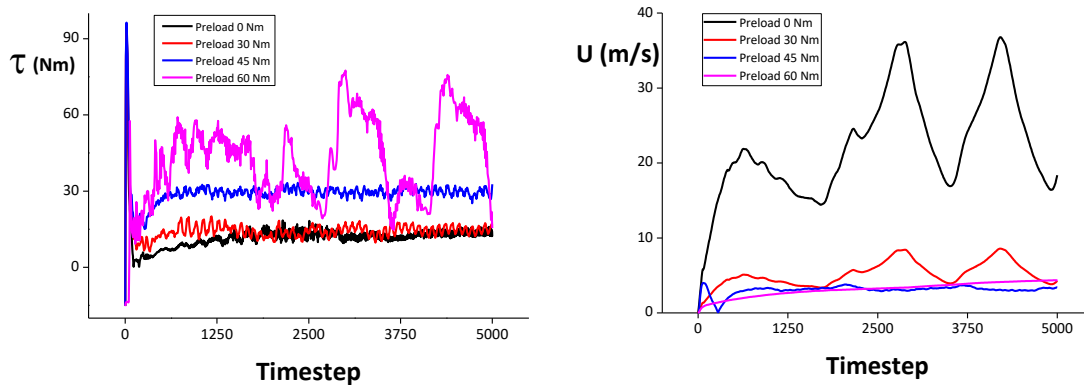
In the mesh independency test, it can be seen that the larger the number of mesh, the smaller the torque deviation produced. Using the Richardson extrapolation method [17], the estimated exact value of torque is 84.44925382 Nm. It can be concluded that the number of mesh 122408 (84.5082 Nm) has the least torque error (0.0698%) value compared to the other number of mesh variations. Thus, the number of mesh 122408 is good enough to use for the simulation. Figure 2 Show the mesh visualization with 122408 number of elements. Each simulation will run for 5 s with 5000 timesteps. Table 2 shows the mesh independency test result.

Table 2: Mesh Independency Test Result

Number of Elements	Torque, τ	Grid Refinement Ratio	GCI
37589	93.17 N·m	6	-
66079	84.94 N·m	4	2.224 %
122408	84.51 N·m	2	0.094 %
230162	84.45 N·m	1	0.012 %
~	84.45 N·m	0	-

Results

From Fig. 3, shows that the torque (τ) produced is inversely proportional to the runner's rotation velocity (U). Furthermore, the greater the preload given, the resulting τ increases but fluctuates, while the resulting U decreases but more stable. These results are similar to experimental results where there are transient and stable data. Based on Fig. 3-a and 3-b, the stable data occur in timestep 3750 and above, at this time the data obtained can be processed. Table 3 is the data that has been processed.



a. τ simulation results

b. U simulations results

Fig. 3. The computational results

Table 3 shows that with the head condition of 1 m, the efficiency of a cross-flow turbine with preload 45 Nm produced a higher efficiency than others. This is expected at 45 Nm preload, the resulting τ and U is more stable than the others. Furthermore, this shows that the maximum condition of the designed crossflow turbine is at 45 Nm preload with a U/W ratio of 0.75. In real conditions, the efficiency of impulse turbine (crossflow, Turgo and Pelton turbine) occurs at U/W of 0.5 [12]. The computation results obtained do not take into account losses due to friction so that the optimum U/W ratio will be obtained at 0.5 (if tested in real conditions).

Table 3: Computational results with variations in preload

Parameter	Preload			
	0 N·m	30 N·m	45 N·m	60 N·m
U/W	0.99	0.95	0.75	0.35
Wheel velocity stable, U	4.26 m/s	4.10 m/s	3.21 m/s	1.51 m/s
Average Torque, τ	1.51 N·m	14.77 N·m	29.73 N·m	44.89 N·m
Efficiency, η	6.39 %	19.29 %	60.07 %	7.47 %

Pressure contour and streamline are used to analyze whether the simulation results are similar to real conditions (qualitative analysis). In some cases of CFD, verification and validation of CFD results using qualitative methods are important because even though the setup is correct, but the physical phenomenon is not in accordance with applicable law, the results obtained are not unjustified. The water volume fraction and streamline are showed only the water phase.

Pressure contour can be seen in figure 4. At each difference in preload loading, it can be seen that pressure has a large value when the fluid first collides with the blade tip. When the preload is given 0 Nm, the turbine will spin very fast which causes the water not to pound the blade. The wheel spin very fast thought to be the cause of the low torque produced. The high-velocity area can be seen on the pressure contour where the pressure is low (at the blade area). Compared to when the turbine was given a preload of 45 Nm, the water pounded almost on all blades. This happens because the given preload acts like a brake which slows the turbine's rotation so that the water can hit the blade. With a large number of blades pounded by water, then the power produced will increase.

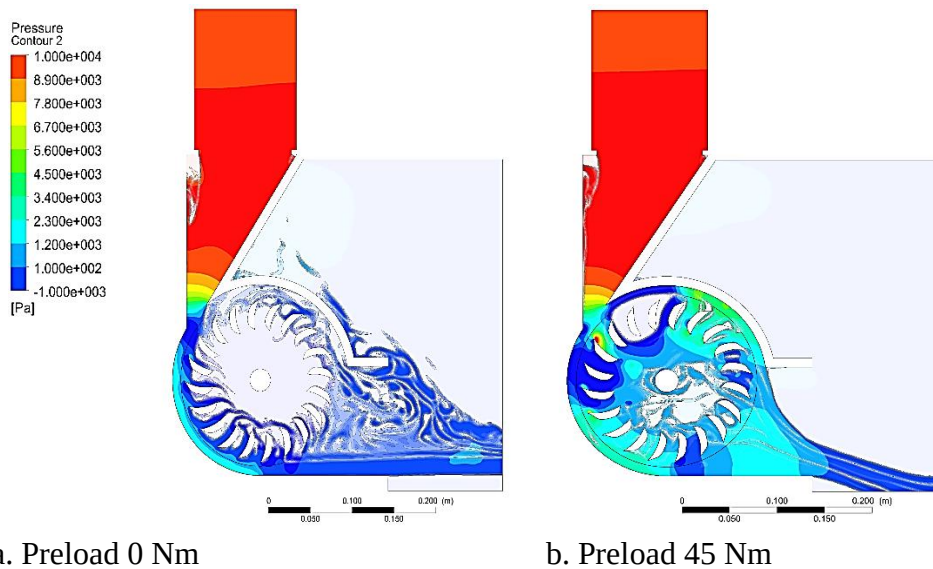
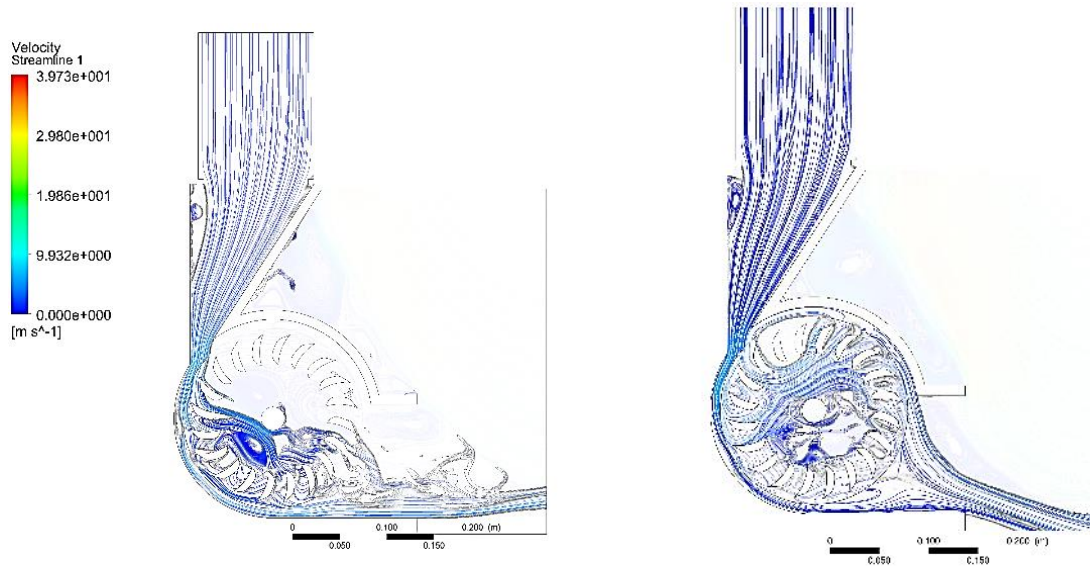


Fig. 4. Pressure contour on different preload acting on its blade

Streamline is used to study the process of converting energy from water to blades (see Fig. 5). In Fig. 5-a, on preload 0 Nm, streamline contour shows only a small portion of water mashing the blade. Also seen, in preload 0 Nm, the conversion energy process on the second stage di not nearly occur. Whereas when the preload is given, the turbine rotation is not so fast that it can be seen when the preload is given at 30 Nm, the water that hit the blade is increasing and so on. It can be concluded that when the turbine does not have preload, the energy conversion process in the second stage is almost no (so that performance is not optimum) due to the fast turning wheel so that the water cannot hit the blade.

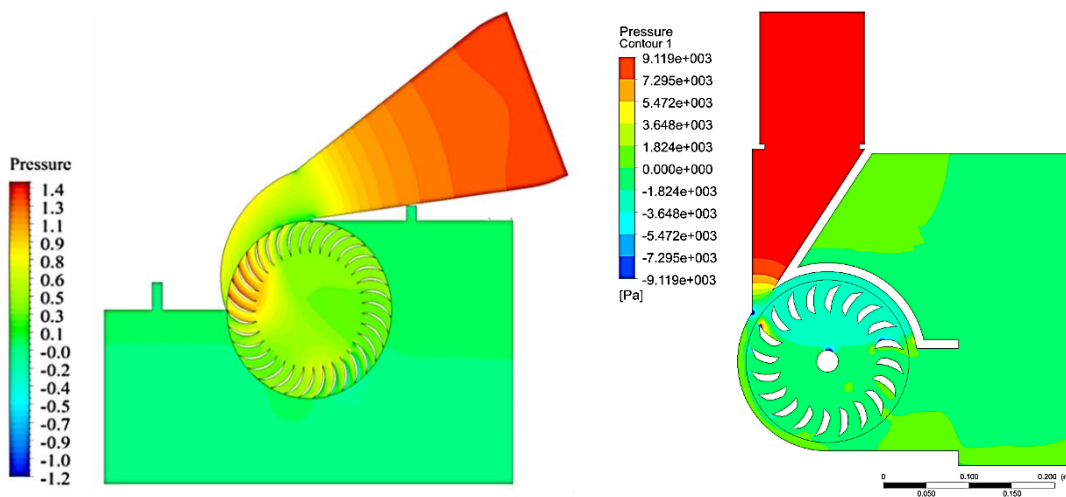


a. Preload 0 Nm

b. Preload 45 Nm

Fig. 5. Streamline on different preload acting on its blade

Fig. 6-a is a pressure contour from the Sammartano [18] simulation result and Fig. 6-b is a pressure contour from this simulation result. From the two figures above, it can be seen that both show similar pressure patterns. The pressure is initially high then starts to decline slowly before finally the pressure will rise again when the water hits the blade for the first time. The same phenomena also can be seen in the impeller area, which has the least pressure value.



a. Sammartano's Pressure contour [17]

b. Pressure contour Preload 30 Nm

Fig. 6. Pressure contour

Fig. 7-a is a pressure contour from the Zaffar [19] simulation result and Fig. 7-b is a streamline from this simulation result. From the two figures below, it can be seen that both show a similar pattern.

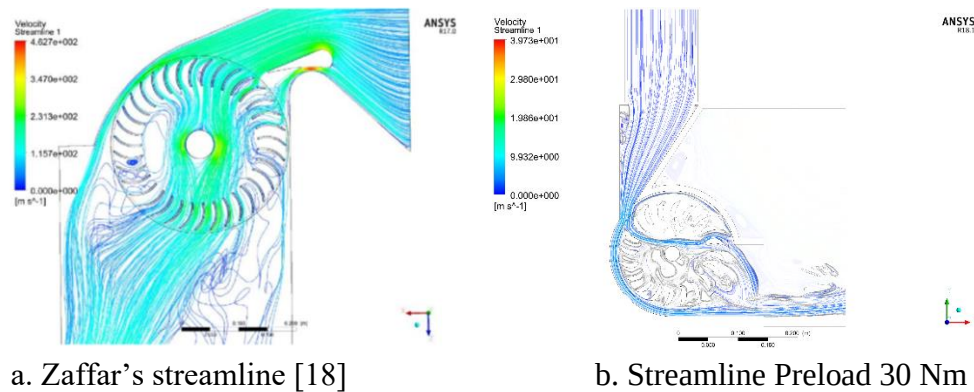


Fig. 7. Streamline

Conclusions

Computational results using the 6-DoF feature with preload variation obtain similar to testing. So this can be an alternative to predict the performance of a pico-hydro cross-flow turbines. The computational results showed that used 45 Nm preload has more stable and had higher efficiency than the other preload variation. The 45 Nm preload produced 60.07% efficiency with the head condition of 1 meter and mass flow of 10.5 kg/s which is the highest efficiency rather than the other variation. Therefore, the effect of preload on the computational results are the preload will produce a lower rotating speed value and a higher torque value when the given preload value increases.

Acknowledgment

This work supported by the Directorate of Research and Service Community (DRPM) Universitas Indonesia with grant No: 0050/UN2.R3.1/HKP.05.00/2019.

References

- [1] International Energy Database, Energy Access Database, (2019).
- [2] K. Kaygusuz, Renewable Energy: Power for a Sustainable Future, Energy Exploration & Exploitation. 19 (2010) 603–626. doi:10.1260/0144598011492723.
- [3] O. Paish, Small hydro power: technology and current status Paish, O. Renewable and Sustainable Energy Reviews, 2002, 6, (6), 537–556, Fuel and Energy Abstracts. 44 (2003) 242. doi:10.1016/S0140-6701(03)83053-9.
- [4] S. Pokhrel, Computational Modeling of A Williams Cross Flow Turbine, Wright State University, 2017.
- [5] D. Adanta, R. Hindami, Budiarto, Warjito, A.I. Siswantara, Blade Depth Investigation on Cross-flow Turbine by Numerical Method, in: 2018 4th International Conference on Science and Technology (ICST), IEEE, Yogyakarta, 2018: pp. 1–6.
- [6] S. Kumar Patro, T. kumar, A. kumar, S. sagar, K. kumar, G. Satya Prasad, Technical and economic feasibility of Pico hydro power plants integrated to Green buildings, International Research Journal of Engineering and Technology. (2016) 2395–56.
- [7] R. Montanari, Criteria for the economic planning of a low power hydroelectric plant, Renewable Energy. 28 (2003) 2129–2145.
- [8] Warjito, D. Adanta, Budiarto, A.P. Prakoso, The effect of bucketnumber on breastshot waterwheel performance, in: IOP Conference Series: Earth and Environmental Science, IOP Publishing, 2018: p. 12031.
- [9] J. Li, W. Du, L. Cheng, Numerical simulation and experiment of gas-solid two phase flow and ash deposition on a novel heat transfer surface, Applied Thermal Engineering.

- 113 (2017) 1033–1046. doi:10.1016/j.applthermaleng.2016.10.198.
- [10] V.R. Desai, A parametric study of the cross-flow turbine performance, 1993.
 - [11] Y.-D. Choi, J.-I. Lim, Y.-T. Kim, Y.-H. Lee, Performance and internal flow characteristics of a cross-flow hydro turbine by the shapes of nozzle and runner blade, *Journal of Fluid Science and Technology*. 3 (2008) 398–409.
 - [12] Harinaldi, Budiarto, *Sistem Fluida (Prinsip Dasar dan Penerapan Mesin Fluida, Sistem hidrolik dan Sistem Pnuematik)*, Erlangga, Jakarta, 2015.
 - [13] J.A. Butbul, The impact of inertia forces on a morphing wind turbine blade in a vertical axis configuration: A wind tunnel test (Order No. 1561514)., 2014.
 - [14] Inc. ANSYS, ANSYS FLUENT Theory Guide, Release 18.2. 15317 (2013) 373–464. doi:10.1016/0140-3664(87)90311-2.
 - [15] D. Adanta, Budiarto, Warjito, A.I. Siswantara, A.P. Prakoso, Performance Comparison of NACA 6509 and 6712 on Pico Hydro Type Cross-Flow Turbine by Numerical Method, *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*. 45 (2018) 116–127.
 - [16] A. Fluent, Ansys fluent theory guide, ANSYS Inc, USA. 15317 (2011) 724–746.
 - [17] T.S. Phillips, C.J. Roy, Richardson Extrapolation-Based Discretization Uncertainty Estimation for Computational Fluid Dynamics, *Journal of Fluids Engineering*. 136 (2014) 121401. doi:10.1115/1.4027353.
 - [18] V. Sammartano, C. Aricò, A. Carravetta, O. Fecarotta, T. Tucciarelli, Banki-Michell optimal design by computational fluid dynamics testing and hydrodynamic analysis, *Energies*. 6 (2013) 2362–2385.
 - [19] J.A. Chattha, A. Zaffar, B. Ibrahim, M. Asif, M.A. Sarwar, Optimisation of blade profiles of cross flow turbine, *International Journal of Power and Energy Conversion*. 9 (2018) 311. doi:10.1504/ijpec.2018.10011716.

EFFECT OF ANNULUS AND INLET HEIGHT RATIO IN STRAIGHT BLADE UNDERSHOT WATERWHEEL TURBINE BY COMPUTATIONAL METHOD

Warjito¹
Dendy Adanta²
Budiarso³
Sanjaya BS Nasution⁴
Muhamad Agil Fadhel Kurnianto⁵

¹Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Depok 16424, West Java, Indonesia

²Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Depok 16424, West Java, Indonesia

³Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Depok 16424, West Java, Indonesia

⁴Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Depok 16424, West Java, Indonesia

⁵Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Depok 16424, West Java, Indonesia (Email: agil.fadhel@gmail.com)

Abstract: *Potential of renewable energy of Indonesia is very abundant, especially from hydropower: 75 GW for a large scale; and 19 GW for mini, micro and pico scale. However, in 2018, there are 20 million peoples in Indonesia who do not yet have access to electricity. To overcome his problem, one type of pico hydro turbine technology is an undershot water wheel because turbine design and installation is believed to be easier (operational and maintenance) and cheaper (investment and operational costs) than other types. Although this turbine technology is quite old compare to other types, but there has been no attention or special discuss by researchers about the effect of annulus (h) and inlet height (L) on the power to be produced. So, this study will investigate the influence of h and L on the power to be produce using CFD tools with six-degrees of freedom (6-DoF) feature. Based on results, the h/L ratio has an influence on the performance of undershot waterwheel. The h/L ratio of 1 produces power higher than 0.5 and 1.5, which the h/L ratio of 1 resulting efficiency is 29.73%, 0.5 is 22.75%, and 1.5 is 26.55%. Thus, the h/L ratio of 1 recommended for designing undershot waterwheel turbines.*

Keywords: *Undershot, waterwheel, computational, blade, 6-DoF*

Introduction

Potential of renewable energy of Indonesia is very abundant, especially from hydropower: a large scale of 75 GW; and mini, micro and pico scale of 19 GW [1]. Although the number seems to be convincing, there are still regions which don't have access to electricity, particularly in remote areas. The latest report states that Indonesia's electrification ratio in 2018 reached 98%. The facts stand that the economic and improvement in people's living standards strongly depends on the utilization of electrical energy. To tackle this, pico-hydro turbines are thought to be appropriate types of turbine to be implemented, specifically for rural communities [2].

Pico hydro turbine is a hydropower plant with its ability to produce electricity below 5 kW [3]. Pico hydro also can be easily designed, developed and operating than solar PV and

wind turbine. Furthermore, pico hydro technology seem to be the most common technology known by most people in Indonesia [4]. One type of pico hydro technology that is familiar in Indonesia is undershot waterwheel because the design and installation of the turbine are believed to be easier (operational and maintenance) and cheaper (investment and operational cost) than other types [4]. The disadvantage of this turbine technology is its low efficiency [4].

To improve undershot waterwheel turbine performance, there are three methods that are often used such as analytical, computational and experimental methods. Computational methods, especially computational fluid dynamics (CFD) tools are currently widely used to solve fluid dynamics problems. This is because CFD tools can analyze a system by reducing the cost of the experiment and of course the long time in conducting the experiment or in the design stage of the engineering process that must be done becomes shorter [5]. Pujol et al. [6] characterizes undershot waterwheel using laboratory scale test data. Characterization results are used as a basis for improving turbine performance using the CFD tools. Nishi et al. [7] compared the performance of straight and curvature blade shapes of undershot waterwheel turbine using CFD tools. Based on results, the straight blade shape is better than the curve. Warjito et al. [4] proposed a number of blades equations for undershot waterwheel turbines. The proposed equation is tested using CFD tools and concludes verified to be used. Based on literature, blade shape has an important role in the undershot waterwheel turbine power generation. However, the influence of ratio of the outer (D_o) diameter with inner (D_i) or annulus (h) on the power to be generated has not yet been conducted in a comprehensive study. Denny [8] undershot water turbine has a ratio value (W / U) for the inlet speed (W) with a rotational speed (U) of 0.33. In his research [8], it was assumed that there was no loss of energy (due to, for example, splashing water) or wheel spokes so that it could reduce the results of analytical calculations. Rinaldi, et al. [9] an experimental study of the physical model of a water wheel as a generator. The researchers formulated the available power from high water fall and flat-water flow.

Based on previous study, the annulus (h) one of the parameters for the power to be produced because the turbine rotation speed (ω) is a function of the moment of inertia (I) of wheel, while the I is influenced by the h . Although this turbine technology is quite old compare to other types, but there has been no attention or special discuss by researchers about the effect of h and inlet height (L) on the power to be produced. So, this study will investigate the influence of h and L on the power to be produce using CFD tools with six-degrees of freedom (6-DoF) feature. The CFD tools with 6-DoF feature was used because the turbine rotation is a computational results not a boundary, which is as actual conditions [10].

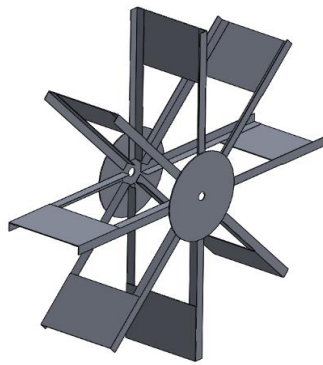
Methodology

Geometry

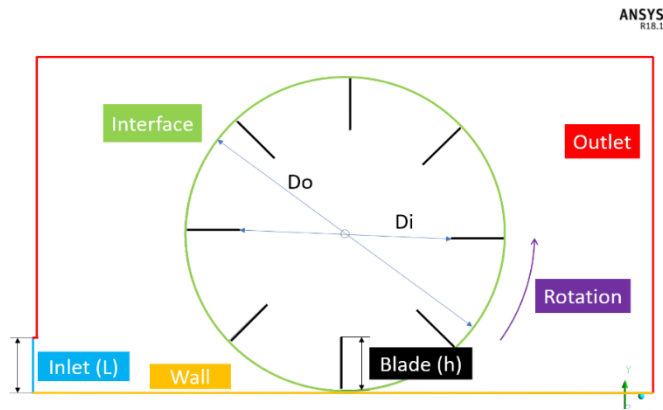
This study is a follow up of Warjito's study [4]. In this simulation, number of blades is constant, with varied value of ratio of annulus and head inlet (h/L) are 0.5, 1, and 1.5. The specific designs parameters shown in Table 1.

Table 5: Turbine Specific Design Parameters

Design Parameter	Dimension
Blade Height, h	0.164 m
Inner Diameter, D_i	0.820 m
Outer Diameter, D_o	0.984 m
Blade Width, W	0.240 m
Number of blades, z	8 blades
Blade Angle, θ	45°
Blade Thickness, t	0.002 m



a. Three-dimensional model of wheel



b. Boundary condition of simulation

Fig. 9. shows the model of the Undershot water wheel

Computational Method

The simulation of the undershot waterwheel was conducted using Ansys-FLUENT 18.1 in two-dimensional (2D) flow analysis. In the flow equation for Fluent ANSYS consists of a continuity equation and a Navier-Stokes equation. In this study, 2D transient domain was used to perform the simulation for fluent.

Boundary conditions: The simulation is for the straight blades waterwheel which consists of 8 blades and the inlet flow velocity (W) of the river is considered to be 1 m/s. The simulations were run using standard Volume of Fluid (VoF) multiphases modelling with constant interfacial surface tension, implicit volume fraction parameters and implicit body force was used. Volume of fluid (VoF) setting was enabled because there were two fluid phases, water and air, with a constant surface tension value of 0.0728 N/m between these fluids [11].

Dynamic mesh settings are also activated to enable the six degrees of freedom (6-DoF) feature, where 6-Dof used the object's forces and moments in order to calculate the translational and angular motion of the center of gravity of an object. In this case, 6-DoF were used to make it possible to investigate the phenomenon of fluid dynamics with the movement of domain blades that occur from interactions with fluids. The equation for the translational motion of the center of gravity is solved for in the inertial coordinate system [12]:

$$\vec{V}_G = \frac{1}{m} \sum_{k=0}^n \vec{F}_G \quad (1)$$

where V_G is the translational motion, f_G is the force because gravity, and m is the mass. The angular motion of the object (ω_B) is further computed using body coordinates (Eq. 2) [12]:

$$\vec{\omega}_B = L^{-1}(\sum \vec{M}_B - \vec{\omega}_B \times L\vec{\omega}) \quad (2)$$

where L is inertia, M_B is moment of the body, and ω_B is rigid body angular velocity.

The 6-DOF property option is activated by the moment of inertia of the undershot waterwheel. The moment of inertia is obtained from the results of mass-property simulation software design with the help of a computer or solidworks. The preload constraint option is also activated for 6-DOF simulations and is given as 3 N.m.

The k-epsilon ($k - \epsilon$) turbulence model is a model that is widely used in CFD to simulate and analyze turbulent flow conditions [13]. Standard ($k - \epsilon$) model approach is used to calculate the characteristic of the selected turbulence flow because its good accuracy for various turbulent streams which explains its popularity in industrial flow. This study uses a turbulence model of two equation results for the comparison results using two transport equations k and epsilon. This model is an improvement from the mixing length model because it has a good for turbulent flow of high Reynolds numbers [13]. The equations for setting the standard ($k - \epsilon$) are available in Eq. (3) for k and (4) for ϵ , respectively [12]:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial t}(\rho k u_i) = \frac{\delta}{\delta x_i} \left[\left(\mu + \frac{u_t}{\sigma_k} \right) \frac{\sigma_k}{\sigma_{xj}} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \quad (3)$$

and

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial t}(\rho \epsilon u_i) = \frac{\delta}{\delta x_j} \left[\left(\mu + \frac{u_t}{\sigma_\epsilon} \right) \frac{\sigma_\epsilon}{\sigma_{xj}} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + G_{3\epsilon} + G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon \quad (4)$$

Mesh and Timestep Independency Tests

The independence test serves to verify the number of mesh elements that will be used later. An accurate mesh is used to get the desired simulation quality. In the mesh independence test, the variation of mesh in this study was about 25196, 50969, and 101117 elements. The results of the independency mesh using Richardson extrapolation method [14] (see Table 2), selected grid consists of 50969 elements because the error value is 0.55%. Table 2 shows the mesh independency test result. Furthermore, for the timestep 0.001 is chosen because it has been available in the previous study [4].

Table 6: Mesh Independency Test Result

Number of Elements	Normalized Grid Spacing	Torque	GCI (%)
25196	2.00	32.11	3.62%
50969	1.41	30.99	0.55%
101117	1.00	30.82	0.10%

Fig. 2 show the mesh visualization with 50969 elements:

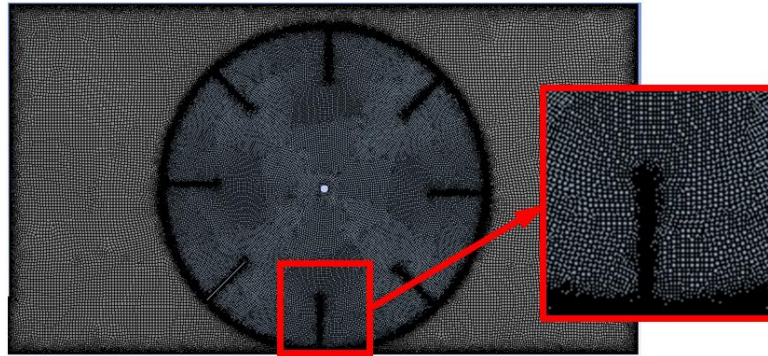


Fig. 10. 2D Domain mesh visualization – 50969 elements

Results and Discussions

Results

Figure 3 is the result of this study simulation. Furthermore, from table 4 is the result of interpretation of the figure 3. Figure 3a is a comparison graph between torque (τ) and timestep. Based on Fig. 3a, the torque (τ) obtained by the three blades fluctuates and until the timestep 4000 to 5000 the torque obtained is a steady state. The Figure 3b is a comparison graph between the wheel rotational velocity (U) and timestep. Based on Fig. 3b, the wheel rotation velocity (U) obtained becomes steady state in timestep 2000. The torque(τ) and (U) data at steady conditions are in Table 3. Based on Table 3, the undershot waterwheel with a ratio h/L 1 is the blade with the highest average efficiency value.

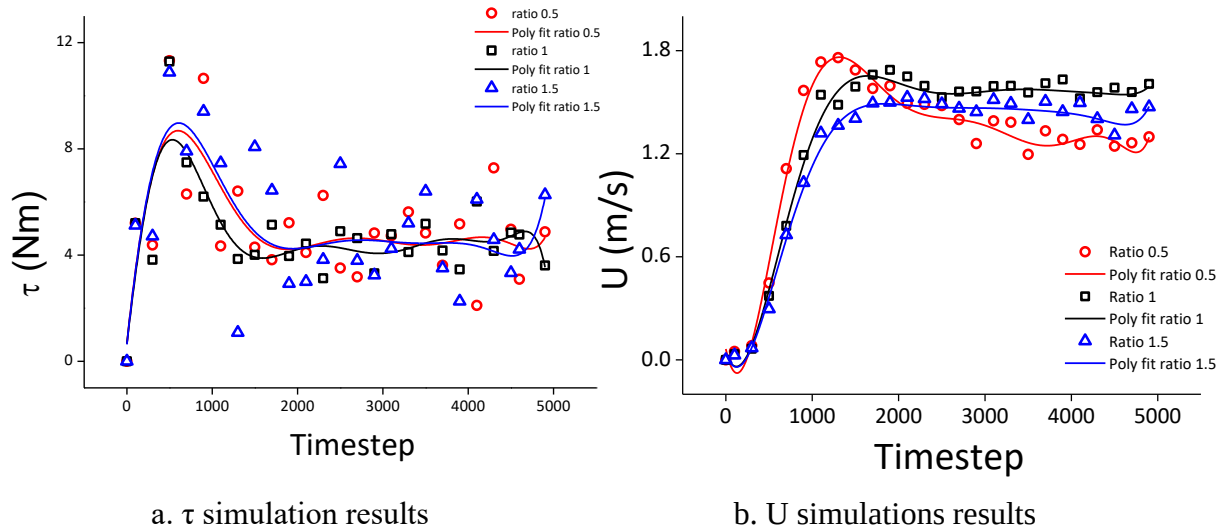


Fig. 11. The computational results

Table 7: Computational results with variations in h/L

Parameter	Ratio of h/L		
	0.5	1	1.5
W/U	0.64	0.83	0.71
Rotational wheel velocity stable, W -9.81	1.22 m/s	1.30 m/s	1.20 m/s
Torque stable, τ (N·m)	4.68	5.00	5.05
Power, P (Watt)	11.58	13.17	12.08
Efficiency, η (%)	22.75	29.73	26.55

Discussions

Table 3 is a summary of computational results for variations in the h/L ratio. Based on table 3, the height of the h/L ratio blade is 1 having an efficiency of 29.73% higher than the others. These results are not much different from previous studies [8]. Based on results, the optimum W/U ratio obtained in this case ranges from 0.64 to 0.83

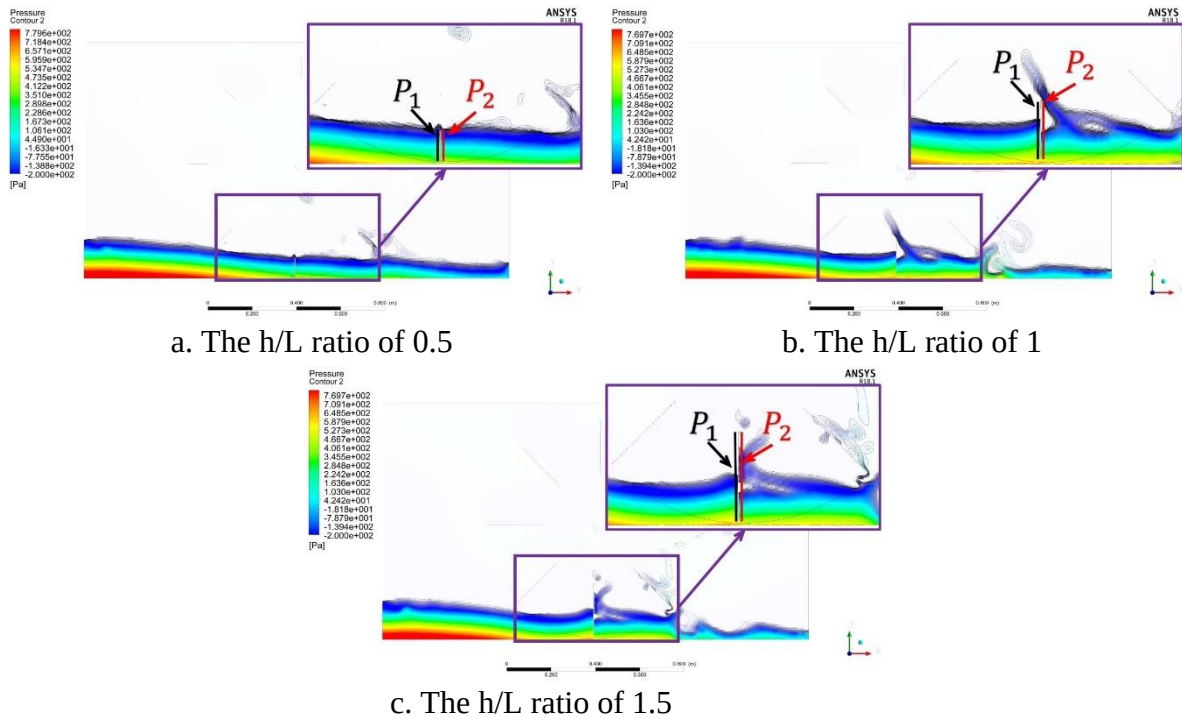


Fig. 12. Water pressure contour

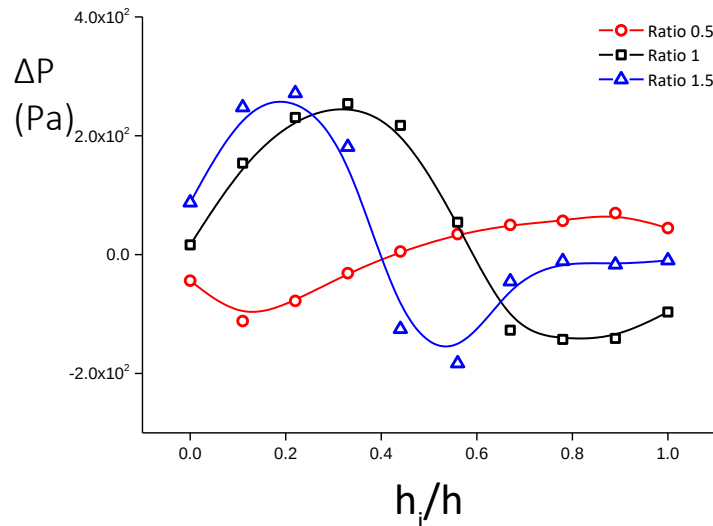


Fig. 13. Gradient pressure between upstream and downstream

The torque produced by the undershot turbine blade is caused by the hydrodynamic force between upstream and downstream. Based on table 3, it appears that h/L ratio of 0.5 produces torque less than 1 and 1.5. This is because the pressure gradient between the upstream and downstream of the active blade is small so the torque produced is also small (see Figure 4a and Figure 5). The low pressure gradient between upstream and downstream at h/L ratio of 0.5

is estimated to be caused by overflow of water that passes through the top of the active blade (see figure 4b). Whereas, h/L at 1 produces greater torque. This may be because the pressure gradient distribution between upstream and downstream of the active blade is higher than the others (see Figure 4b and Figure 5). In addition, at h/L ratio of 1 no water passes through the blade so the hydrodynamic force is absorbed more into the power (see figure 4b). On the h/L ratio of 1.5, the pressure gradient distribution between upstream and downstream is greater than h/L ratio of 0.5 and 1. However, the low torque is suspected because the higher the blade height, the greater the cross-sectional area, the greater the cross-sectional area causes the rotation of the blade to be driven by air also to increase. (see figure 6).

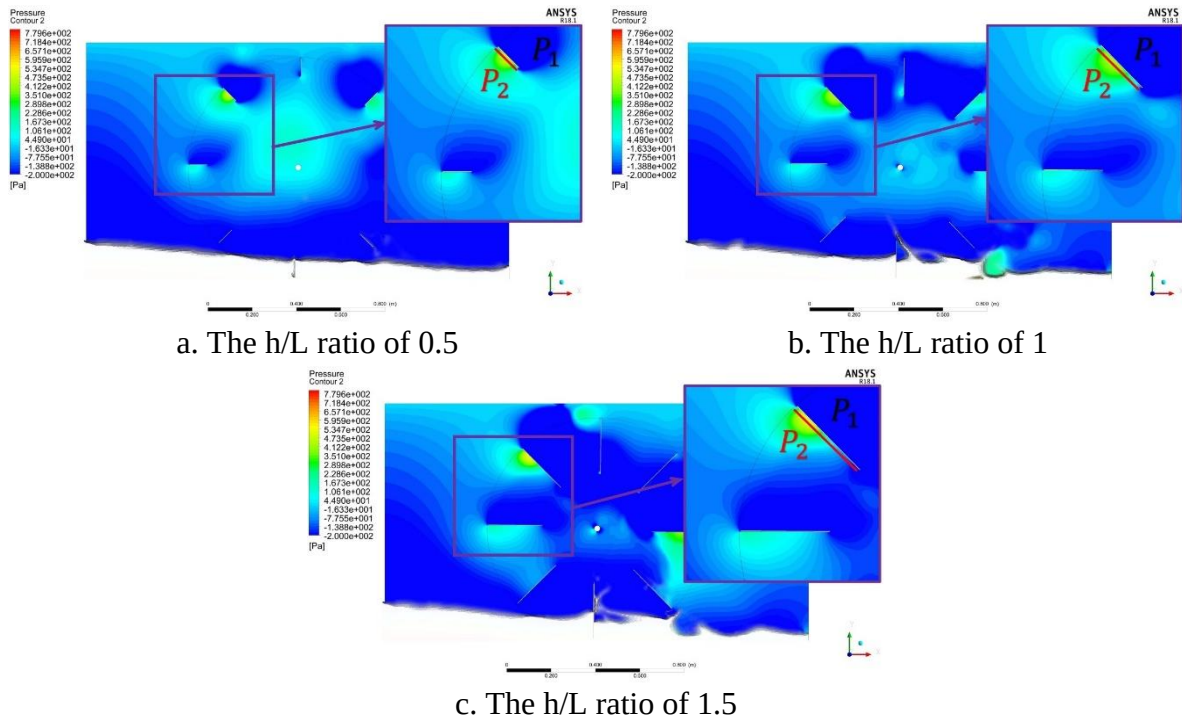


Fig. 14. Air pressure contour

The investigation results obtained verify the assumptions purposed by Nasution [], where the nasution recommends the optimum value h is 6 times Do . The results of the investigation also verified the simulation carried out by Yah. Yah simulates using the rotation periodicity method [15]. Yah [15] reports that the ratio of $h/Do \approx 0.2$ produces maximum efficiency. In this case the maximum efficiency is obtained at the ratio $h/Do \approx 0.16$, which is equivalent to $h/L = 1$.

Conclusions

Based on results, the h/L ratio has an influence on the performance of undershot waterwheel. The h/L ratio of 1 produces power higher than 0.5 and 1.5, which the h/L ratio of 1 resulting efficiency is 29.73%, 0.5 is 22.75%, and 1.5 is 26.55%. Thus, the ratio of h/L of 1 recommended for designing undershot waterwheel turbines. However, this propose must be proven by a comprehensive experimental study.

Acknowledgment

This work supported by the Directorate of Research and Service Community (DRPM) Universitas Indonesia with grant No: 0050/UN2.R3.1/HKP.05.00/2019.

References

- [1] Humas EBTKE, “Menuju Rasio Elektrifikasi 99 Persen pada 2019,” *Energi Baru Terbarukan dan Konservasi Energi (EBTKE)*, 2018. .
- [2] P. Maher, N. P. A. Smith, and A. A. Williams, “Assessment of pico hydro as an option for off-grid electrification in Kenya,” *Renewable Energy*, vol. 28, no. 9, pp. 1357–1369, 2003.
- [3] G. Müller, “Renewable Energy -Muller WATER WHEELS AS A POWER SOURCE,” pp. 1–9, 2007.
- [4] Warjito, S. A. Arifianto, Budiarmo, S. B. Nasution, and D. Adanta, “Effect of Blades Number on Undershot Waterwheel Performance with Variable Inlet Velocity,” in *2018 4th International Conference on Science and Technology (ICST)*, 2018, pp. 1–6.
- [5] E. Quaranta and R. Revelli, “Hydraulic behavior and performance of breastshot water wheels for different numbers of blades,” *Journal of Hydraulic Engineering*, vol. 143, no. 1, p. 4016072, 2016.
- [6] T. Pujol, A. K. Vashisht, J. Ricart, D. Culubret, and J. Velayos, “Hydraulic efficiency of horizontal waterwheels: Laboratory data and CFD study for upgrading a western Himalayan watermill,” *Renewable energy*, vol. 83, pp. 576–586, 2015.
- [7] Y. Nishi, T. Inagaki, Y. Li, R. Omiya, and J. Fukutomi, “Study on an undershot cross-flow water turbine,” *Journal of Thermal Science*, vol. 23, no. 3, pp. 239–245, 2014.
- [8] M. Denny, “The efficiency of overshot and undershot waterwheels,” vol. 25, pp. 193–202, 2004.
- [9] A. Junaidi, R. Rinaldi, and A. Hendri, “Model fisik kincir air sebagai pembangkit listrik,” *Jurnal Online Mahasiswa (JOM) Bidang Teknik dan Sains*, vol. 1, no. 2, pp. 1–9, 2015.
- [10] Warjito, D. Adanta, Budiarmo, and A. P. Prakoso, “The effect of bucketnumber on breastshot waterwheel performance,” in *IOP Conference Series: Earth and Environmental Science*, 2018, vol. 105, no. 1, p. 12031.
- [11] A. I. Siswantara and B. Budiarmo, “Assessment of Turbulence Model for Cross-Flow Pico Hydro Turbine Numerical Assessment of Turbulence Model for Cross-Flow Pico Hydro Turbine Numerical Simulation,” no. December, 2018.
- [12] A. Fluent, “ANSYS fluent theory guide 15.0,” ANSYS, Canonsburg, PA, 2013.
- [13] B. Launder, “Progress in the development of a Reynolds-stress turbulence closure,” no. October, 2014.
- [14] T. S. Phillips and C. J. Roy, “Richardson Extrapolation-Based Discretization Uncertainty Estimation for Computational Fluid Dynamics,” *Journal of Fluids Engineering*, vol. 136, no. 12, p. 121401, 2014.
- [15] N. F. Yah, M. S. Idris, and A. N. Oumer, “Numerical investigation on effect of immersed blade depth on the performance of undershot water turbines,” in *MATEC Web of Conferences*, 2016, vol. 74, p. 35.

EFFECT OF BLADE DEPTH ON THE ENERGY CONVERSION PROCESS IN CROSSFLOW TURBINE

Dendy Adanta¹
Warjito²
Budiarso³
Aji Putro Prakoso⁴
Elang Pramudya Wijaya⁵

¹Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Depok 16424, West Java, Indonesia

²Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Depok 16424, West Java, Indonesia

³Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Depok 16424, West Java, Indonesia

⁴Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Depok 16424, West Java, Indonesia

⁵Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Depok 16424, West Java, Indonesia (Email: elangpw@outlook.com)

Abstract: Indonesia as an archipelago country where almost 8% of the populations in the rural area has difficult access to electricity. On the other hand, Indonesia has many potential hydro energy sources until 19 GW for mini, micro and pico scale. Hydro power as one of the most environment friendly energy source is an answer of eco-friendly power plant in developing country. This study examines the effects of blade depth of the crossflow turbines in three variations of blade depth ratio. Using computational methods, this study compares three types of blade depth ratio, 1.5mm, 3mm, and 4.5mm blade depth. Computational result was compared to previous study. The boundary conditions are model turbulent using SST k- ω , 6-DoF, and two-dimensional transient domain. Multiphase volume of fluid (VoF) with implicit volume fraction parameters and implicit body force was used. From the result obtained, the maximum efficiency of crossflow turbine, shows that blade with higher blade depth ratio tends to have higher efficiency. However, the 3mm of blade depth has a maximum efficiency vulnerable to wider than 4.5mm and 0mm. Thus, the 3mm of blade depth is recommended for this condition.

Keywords: Picohydro, cross-flow turbine, blade depth, computation

Introduction

In 2015, more than 8% of villages in Indonesia did not yet have access to electricity grid [1]. Areas that are experiencing an electricity crisis due to the hilly topography and islands make it difficult to reach [2]. On the other hand, Indonesia has many potentials hydro energy sources until 19 GW for mini, micro and pico scale (< 5 kW) [2].

Pico hydropower is one of the most an energy source environmentally friendly. In developing countries, hydropower pico scale is being used as an independent power plant [3]. This is because hydro energy has several advantages include of the long-time predictability, large potential resource and has greater energy density which is about 832 times greater than that of wind and solar [4]. Among various types of pico hydropower, hydro crossflow turbine has several unique advantages [5], which made this study focus on studying this turbine.

Crossflow turbine is an impulse which converts kinetic energy of water to rotate runners [6], [7]. When water strike the runner blades, they give impact to the blades then

transferred their momentum to the blades [8]. Crossflow turbine can work in low head (<5 m) conditions and high discharge deviations [6]. These conditions are often found in rivers in Indonesia. Previous studies suspect that crossflow turbine is appropriate turbine for independent power plant to solve the electricity energy crisis in remote areas in Indonesia [2], [6], [7], [9].

The energy conversion of crossflow turbine divided into two stages [10]: 68.5% of energy was transfer in the 1st stage, then the 31.5% was transferred in the 2nd stage. Several design parameters such as number of blades, blade shapes, nozzle shape, and guide vanes, must be considered to obtain crossflow turbine's optimum performance. Crossflow turbine performance is predominantly influenced by blade shape [6]. This is because the blade is where energy is converted from water to the turbine. Based on previous study, the blade depth has affecting the performance of crossflow turbine, blade with 3mm blade depth had higher efficiency and more stable than others [6]. However, the results of the study obtained have not been validated. This study will re-investigation previous work where the results will be validated so that the conclusion obtained are are valid.

Methodology

Geometry turbine

The geometry of this crossflow turbine is using [11] model turbines with various blade depth. Detail of the geometry dimensions can be seen in Table 1.

Table 1: Turbine design parameters

Description	Dimension
Outer diameter, D_o	162mm
Inner diameter, D_i	104mm
Number of blades	35
Blade chord length, C	30.9633

Representational of Table 1 can be seen in Fig. 1.

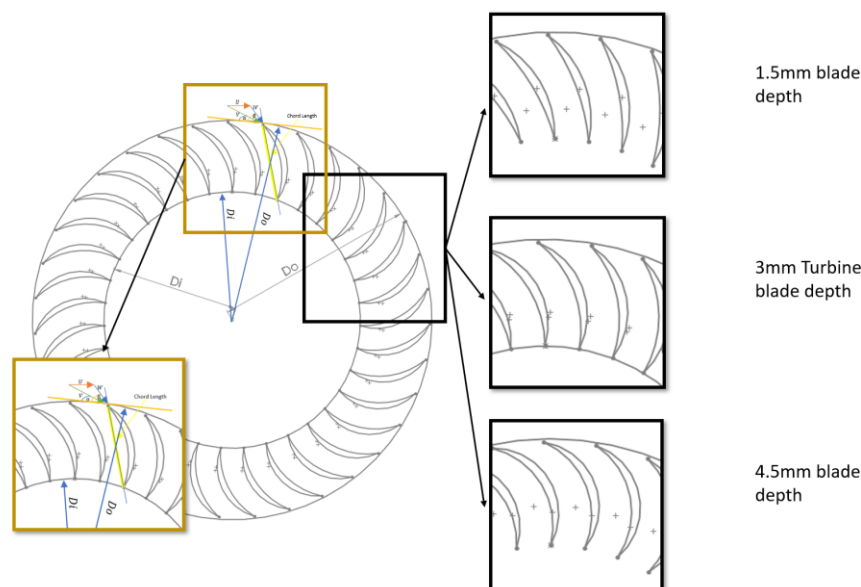


Fig. 1. 2D model of the blade

In this simulation, blade chord was constant, with varied blade depth multiple by 0.5 times and 1.5 times the [11] crossflow turbine model.

Simulation Setup

This simulation uses a pressure inlet with pressure based because the conditions is incompressible. The gravitational acceleration of 9.81 m/s^2 . Multiphase volume of fluid (VoF) with implicit volume fraction parameters and implicit body force was used. Water and air phase chosen based on the case with the condition. Volume of water is 1 water that went through inlet. Hence, the value of air in the inlet is 0 in the inlet section. The transport of VoF can be seen in Eq. 1.

$$\frac{1}{\rho_q} \left[\frac{\partial}{\partial t} (\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q) \right] = S_{a_q} + \sum_{p=1}^n (\dot{m}_{pq} - \dot{m}_{qp}) \quad (1)$$

Where, $\dot{m}_{pq}$ is the transfer of mass from both phase p to phase q [12]. Phase p is water and q is air. The Shear Stress Transport (SST) $k-\omega$ viscous model also used because based on assessment this model is optimally for this case [9].

This simulation is also use dynamic mesh setting with the six-degrees of freedom (6-DoF) feature for better view of fluid dynamics and flow in the crossflow turbine model [7]. The 6-DoF feature required some additional initial condition using moment of inertia and preload. This study used 1 N·m of preload, then the moment of inertia will be known by using computer-aided design (CAD) software.

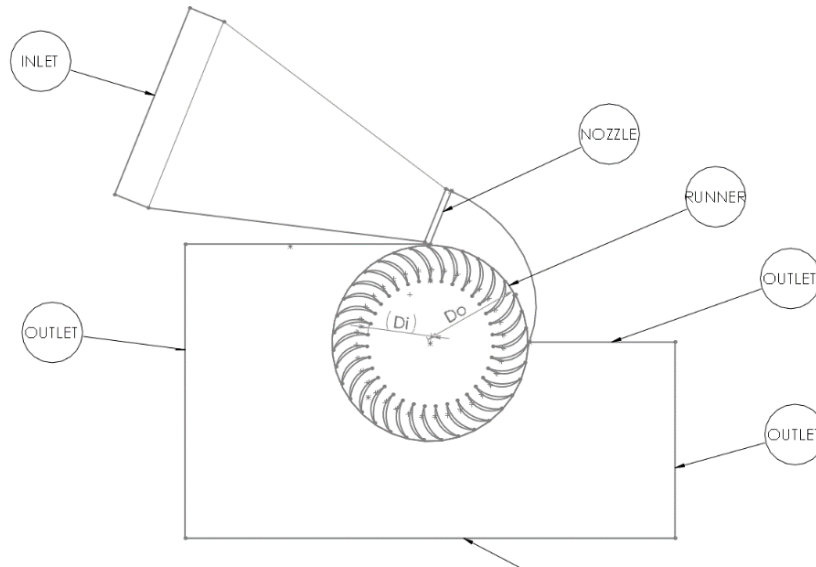


Fig. 2. Boundary Condition

In order to compute the translational motion of the turbine, by using this equation:

$$\vec{v}_g = \frac{1}{m} \sum \vec{f}_G \quad (2)$$

Where, to find $\vec{v}_g$ is determined by the equation above. To determine the angular motion of the object with the equation below;

$$\vec{\omega}_B = L^{-1} \left(\sum \vec{M}_B - \vec{\omega}_B \times L \vec{\omega}_B \right) \quad (3)$$

Where L is the inertia, $\vec{M}_B$ is the moment vector of the blade, then $\vec{\omega}_B$ is the rigid body angular velocity. Both of angular and translational velocities is also used to find the position of the rigid body.

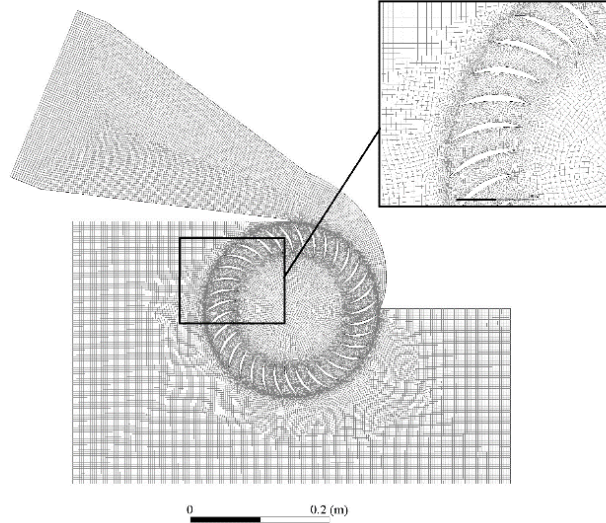


Fig. 3. 2D Domain mesh visualization – 96435 elements

Accurate mesh used to get more desirable quality of this simulation. To calculate error from Richardson extrapolation by using Grid Convergence Index (GCI), which calculated based on Equation below.

$$GCI_{ab} = H \left| \frac{1}{\tau_a} \frac{\tau_b - \tau_a}{r_{ab}^K - 1} \right| \times 100\% \quad (4)$$

Where H is safety factor of 1.25, f is a numerical solution of lower order discrete values, and r is the grid refinement ratio, which is represented to equation below:

$$r_{ab} = \left(\frac{M_a}{M_b} \right)^{0.5} \quad (5)$$

Further, determine the order of convergence observed (K) using Eq. 6 by numerical method: From the eq. 5, the extrapolated values can be varied by the order of p [13], and to estimate the accuracy can determined by using the equation below[14]:

$$K_{n+1} = \frac{\ln \left(\left(\frac{\tau_c - \tau_b}{\tau_b - \tau_a} (r_{ab}^K - 1) \right) + r_{ab}^K \right) \left(\frac{\varepsilon_{32}}{\varepsilon_{21}} \right)}{\ln (r_{ab} \cdot r_{bc})} \quad (6)$$

After all known, then determine the exact torque (τ_{exact}). Exact torque or exact value is an approach to find out the true value at the condition of the number of mesh is infinity [15]. τ_{exact} can be determined using Eq. 7:

$$\tau_{exact} = \tau_a - \left(\frac{\tau_b - \tau_a}{r_{ab}^K - 1} \right) \quad (6)$$

Several mesh independency tests conducted using steady state single phase method.

Results

Independency Test results

Four mesh size was estimated to find the suitable mesh. Several variations of mesh size start from 18k, 32k, 64k, 96k, and 128k, then translated to 1.99, 1.41, 1.15, and 1 (normalized grid spacing) (see Table 2). Based on GCI result, mesh number of 96,649 elements is optimal to use with the value of GCI is 1.47%. With the GCI concept, the timestep value is also analysed to get the optimal timestep called timestep convergency index (TCI). Based on analysis, this simulation was use 0.0005 s of time with 5000 timesteps, because the error is considered to be small at 1.35%.

Table 2: Mesh independency test result

Normalized Grid Spacing	Number of Elements	Torque [Nm]	GCI
1.99	32,484	255.06	-
1.41	64,450	266.59	1.54%
1.15	96,649	268.21	1.47%
1.00	128,848	269.84	0.99%
0.00	<i>Infinite</i>	271.98	-

Table 3: Timestep independency test result

Normalized Timestep Sizing	Timestep Size	Torque [Nm]	TCI
12.65	0.0032	109.82	-
2	0.0008	118.62	1.46%
1.58	0.0005	119.37	1.35%
1	0.0002	120.13	0.54%
0	0.0000	120.65	-

Crossflow Turbine Performance Results

From the computational result above in Fig. 4, which shows the turbine velocity characterized by ratio of rotation wheel velocity and absolute water velocity (U/V). Obtained from literature that when the U/V ratio value is between 0.42 and 0.50, impulse turbine has maximum efficiency [16]. In this Fig. 4, the maximum efficiency of crossflow turbine occurs in U/V ratio of 0.48 to 0.63 between 0.48-0.63 with blade depth ratio is varied from 0.5, and 1.5 times of the [11] model, shows that blade with higher blade depth ratio tends to has higher efficiency.

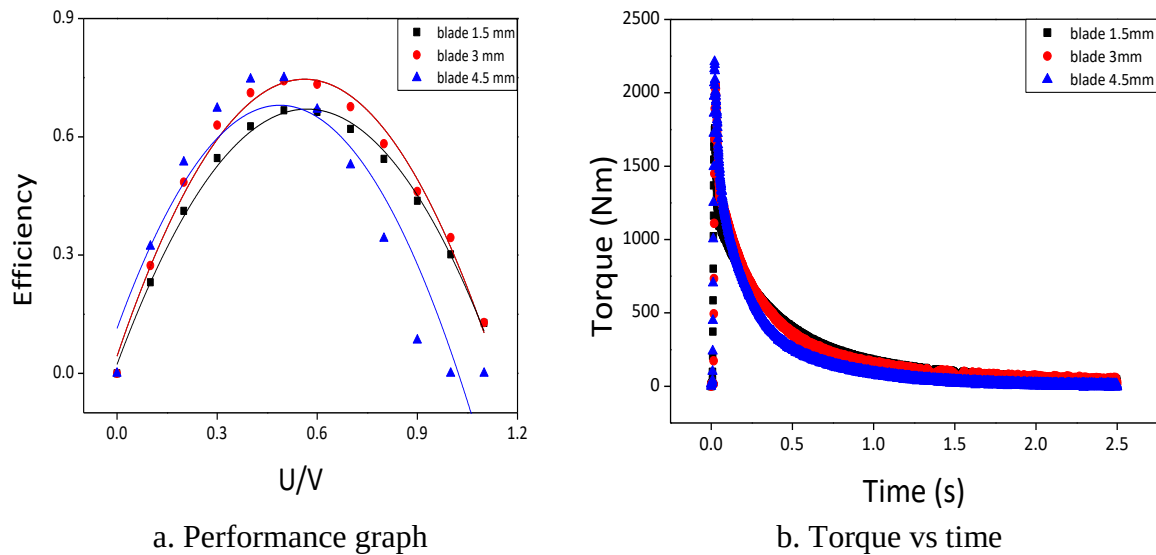


Fig. 4. The Computational Result

Discussion

The numerical simulation results in this study obtained that the 3mm-depth blade crossflow turbine has the highest efficiency than other tested cases. This results indeed convenient with theoretical approach of crossflow blade curvature that related with water velocity angle and runner's diameter ratio [17]. However, the higher impulse force impact in 1st-stage of energy transfer in this turbine by increasing the blade depth still couldn't increase this turbine's performance. The modification only disrupts the internal flow inside the crossflow turbine and reduce the energy transfer quality in 2nd-stage. The total pressure contour of 4.5mm-depth blade crossflow turbine in figure 5.C. is briefly explaining this situation.

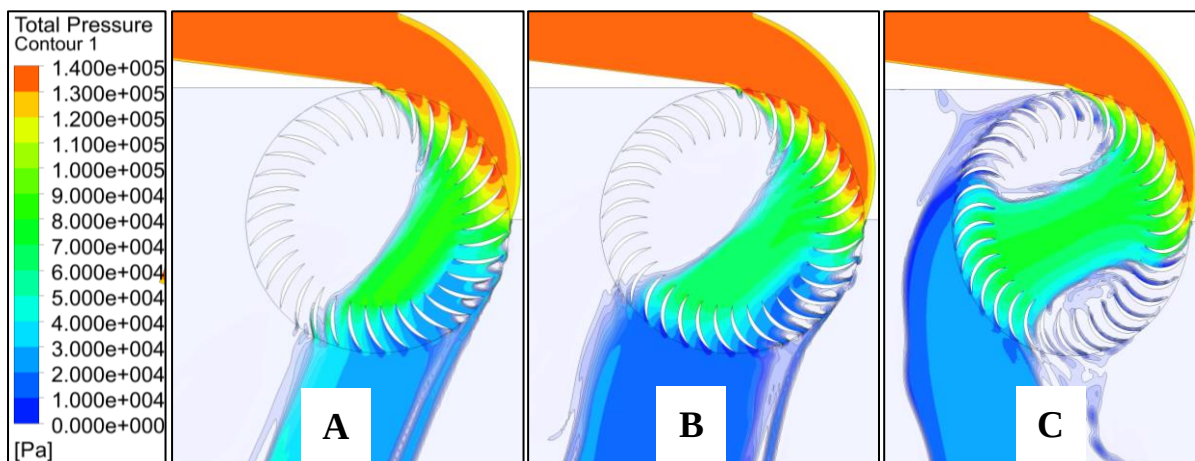


Fig. 5. Total Pressure Contour at Maximum Efficiency Condition in Different Blade Depth, A. 1.5 mm, B. 3 mm, C. 4.5 mm.

Total pressure of water is variable that can represent stored energy inside the water in hydro turbine analysis. From figure 5, it can be obtained that the energy transfer process in 1st-stage is quite similar which indicate that the increment of impulse force effect, which discussed before, indeed not escalate the energy transfer in 1st-stage. However, the total pressure drop in 2nd-stage is different at each blade depth results. It is indicated that the blade depth variation is

affecting the quality of 2nd-stage energy transfer. The total pressure drops in 2nd-stage at 1.5mm-depth blade crossflow turbine from figure 5.A. is from about 80kPa to about 40kPa or only about 40kPa. In the other hand, the 3mm-depth and 4.5mm-depth case has total pressure drop respectively about 65kPa and 60kPa. The obtained results in figure 5 is highly suitable with efficiency curve in figure 4.a. Moreover, it was also obtained that small amount of water is seems still stick to the turbine blades in 4.5mm-depth blade case which could make some friction loss.

The difference of blade depth in this study is also affecting the turbulence condition at the internal flow of crossflow turbine. Corresponding to prior study by Adanta [6],[7], and Warjito [5], the imperfect design of turbine blade could affect the internal flow condition. The effect of blade depth to the internal flow condition is represented by turbulence intensity contour in figure 6.

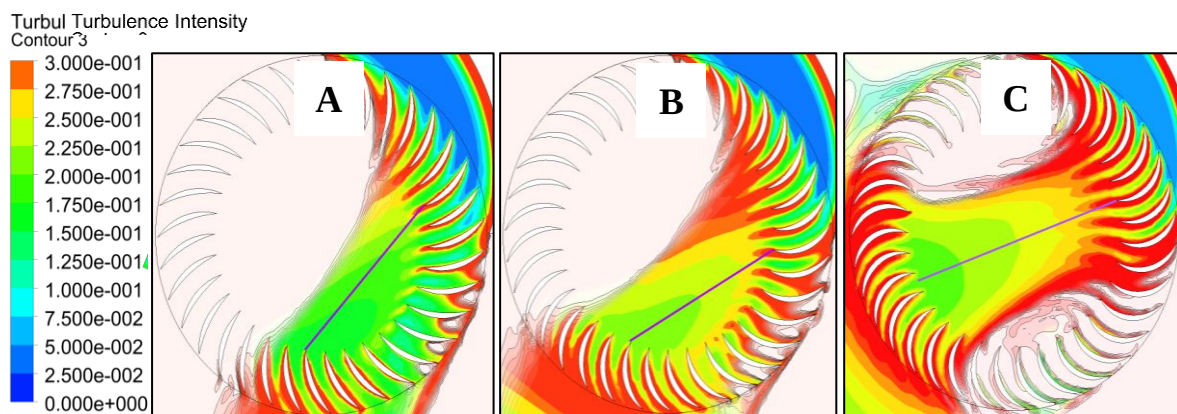


Fig. 6. Turbulence Intensity Contour on Internal Flow at Different Blade Depth, A. 1.5 mm, B. 3 mm, C. 4.5 mm.

It can be obtained simply that deeper blade curve resulting higher turbulence intensity on internal flow of crossflow turbine. It is also seen that 1.5mm-depth blade turbine has far lower than 3mm or 4.5mm case. However, the value of turbulence intensity at the purple line these for three cases are displayed in the graphic at figure 7.

It can be seen from figure 7 that the turbulence intensity of water near 1st-stage blade is very high especially at 4.5mm-dept case. In this case, the turbulence intensity is very high approaching 0.5, the high intensity turbulence then dissipated along the water flowing inside the turbine. The dissipated high intensity turbine could result energy loss inside the water which transformed to small amount of thermal energy. In the other side, the turbulence intensity at 1.5mm-depth crossflow turbine is low both on graphic and contour result which indicate smaller energy loss inside the water. However, it was discussed before that 1.5mm-depth case has low energy transfer quality.

The several discussions above resulting that 3mm-depth blade crossflow turbine provide more balance approach of energy transfer quality and the protection from energy loss inside the water caused by turbulence phenomenon. Although the turbulence loss effect is very low, it was still affecting the performance of cross-flow turbine like happened in 4.5mm-depth case than still should be minimized. The energy transfer process quality is highly affecting the crossflow turbine performance which is not good in 1.5mm-depth case.

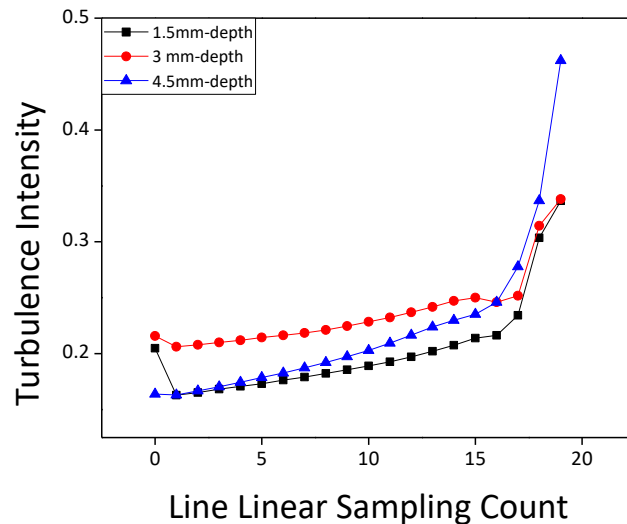


Fig. 7. Turbulence Intensity graph on purple line for each case.

Conclusions

The result of this study showed that the higher blade depth ratio tends to have higher efficiency. The 4.5mm of blade depth design produced the highest efficiency which valued 74.97%. However, the 3 mm of blade depth has a maximum efficiency vulnerable to wider than 4.5 mm. Thus, the 3 mm of blade depth is recommended for this condition.

Acknowledgement

This work supported by the Directorate of Research and Service Community (DRPM) Universitas Indonesia with grant No: 0050/UN2.R3.1/HKP.05.00/2019.

References

- [1] S. Indonesia, "Electricity Statistics 2011–2015."
- [2] D. A. B. W. A. I. Siswantara, "Assessment of Turbulence Modelling for Numerical Simulations into Pico Hydro Turbine," *J. Adv. Res. Fluid Mech. Therm. Sci.*, vol. 45, no. 1, pp. 21–31, 2018.
- [3] C. S. Kaunda, C. Z. Kimambo, and T. K. Nielsen, "Hydropower in the context of sustainable energy supply: a review of technologies and challenges," *ISRN Renew. Energy*, vol. 2012, 2012.
- [4] R. Sipahutar, S. M. Bernas, and M. S. Imanuddin, "Renewable energy and hydropower utilization tendency worldwide," *Renew. Sustain. Energy Rev.*, vol. 17, pp. 213–215, 2013.
- [5] Warjito, Ahmad Indra Siswantara, Dendy Adanta, Aji Putro Prakoso, Reza Dianofitra, "Comparison Between Airfoil Profiled Blade and Ordinary Blade in Cross-Flow Turbine Using Numerical Simulation," in *15th International Conference on Quality in Research*, 2017.
- [6] Dendy Adanta, Richiditya Hindami, Budiarmo, Warjito, Ahmad Indra Siswantara, "Blade Depth Investigation on Cross-flow Turbine by Numerical Method," in *2018 4th International Conference on Science and Technology (ICST)*, 2018, pp. 1–6.
- [7] Dendy Adanta, Budiarmo, Warjito, Ahmad Indra Siswantara, Aji Putro Prakoso, D. Adanta, Budiarmo, Warjito, A. I. Siswantara, and A. P. Prakoso, "Performance comparison of NACA 6509 and 6712 on pico hydro type cross-flow turbine by numerical method," *J. Adv. Res. Fluid Mech. Therm. Sci.*, vol. 45, no. 1, pp. 116–127, 2018.

- [8] O. Paish, "Small hydro power: technology and current status," *Renew. Sustain. energy Rev.*, vol. 6, no. 6, pp. 537–556, 2002.
- [9] Ahmad Indra Siswantara, Budiarto, Aji Putro Prakoso, Gun Gun R Gunadi, Warjito, Dendy Adanta, "Assessment of Turbulence Model for Cross-Flow Pico Hydro Turbine Numerical Simulation," *CFD Lett.*, vol. 10, pp. 38–48, 2018.
- [10] J. De Andrade, C. Curiel, F. Kenyery, O. Aguilón, A. Vásquez, and M. Asuaje, "Numerical investigation of the internal flow in a Banki turbine," *Int. J. Rotating Mach.*, vol. 2011, 2011.
- [11] V. Sammartano, G. Morreale, M. Sinagra, and T. Tucciarelli, "Numerical and experimental investigation of a cross-flow water turbine," *J. Hydraul. Res.*, vol. 54, no. 3, pp. 321–331, May 2016.
- [12] A. Fluent, "ANSYS fluent theory guide 15.0," ANSYS, Canonsburg, PA, 2013.
- [13] M. S. M. Ali, C. J. Doolan, and V. Wheatley, "Grid convergence study for a two-dimensional simulation of flow around a square cylinder at a low Reynolds number," in *Seventh International Conference on CFD in The Minerals and Process Industries (ed. PJ Witt & MP Schwarz)*, 2009, pp. 1–6.
- [14] T. Xing and F. Stern, "Factors of safety for Richardson extrapolation for industrial applications," IOWA UNIV IOWA CITY IIHR-HYDROSCIENCE AND ENGINEERING, 2008.
- [15] P. J. Roache, "Quantification of uncertainty in computational fluid dynamics," *Annu. Rev. Fluid Mech.*, vol. 29, no. 1, pp. 123–160, 1997.
- [16] Harinaldi, Budiarto, *Sistem Fluida (Prinsip Dasar dan Penerapan Mesin Fluida, Sistem hidrolis dan Sistem Pnuematik)*. Jakarta: Erlangga, 2015.
- [17] V. Sammartano, C. Aricò, A. Carravetta, O. Fecarotta, and T. Tucciarelli, "Banki-michell optimal design by computational fluid dynamics testing and hydrodynamic analysis," *Energies*, vol. 6, no. 5, pp. 2362–2385, 2013.

STAGNATION POINT FLOW OVER A STRETCHING/SHRINKING SHEET IN A CARBON NANOTUBES WITH SUCTION/INJECTION EFFECTS

Nur Hazirah Adilla Norzawary¹

Norfifah Bachok^{1,2}

Fadzilah Md Ali^{1,2}

¹Institute for Mathematical Research, Universiti Putra Malaysia, 43400, UPM Serdang, Selangor, Malaysia
(Email: nurhazirah0929@gmail.com)

²Department of Mathematics, Faculty of Science, Universiti Putra Malaysia, 43400, UPM Serdang, Malaysia

Abstract: This research is to explore the stagnation point flow problem in carbon nanotubes with suction/injection impacts over a stretching/shrinking sheet. The velocity of stretching/shrinking and the velocity of ambient fluid varies linearly with the range from the stagnation point. The governing partial differential equations are transformed into a system of nonlinear ordinary differential equations using a similarity transformation which is then solved numerically using a bvp4c solver in MATLAB. Two types of carbon nanotubes (CNTs), single- and multi-wall CNTs are used with water as the base fluids. The influence of the flow parameters on the dimensionless velocity, temperature, skin friction and Nusselts numbers are explored and presented in the form of graphs and interpreted physically. It is concluded that the effect of suction could widen the range of solutions exist and the effect of injection acted on the contrary.

Keywords: Stagnation point flow, stretching/shrinking sheet, carbon nanotubes, suction/injection effects, dual solutions

Introduction

Stagnation point flow represents a fluid flow near the stagnation region of solid surface at which fluid approaching the surface divides into different streams. Heimenz [1] is the first person who studied the 2D stagnation point flow towards a stationary semi-infinite wall where he reduced the Navier-Stokes equations to nonlinear ordinary differential equations by using a similarity transformation. After that, Homann [2] extended this problem [1] in consideration of axisymmetric stagnation point flow. Mahapatra and Gupta [3,4] investigated both stagnation-point flows and stretching surface effects. Crane [5] studied the steady 2D boundary layer flow due to the stretching of the flat elastic surface moving in the plane. But Khan and Pop [6] are the first who considered the problem on stretching sheet in nanofluids. After this seminal work, the research problem on the fluid flow field over a stagnation point towards a stretching/shrinking sheet has been examined by several researchers [7-12].

As many studies related to nanofluid, carbon nanotubes also show excellent due to its high warm, electrical and mechanical properties. Thus, Choi et al. [13] studied the thermal conductivity of oil-based carbon nanotubes. Carbon nanotubes (CNTs) is an allotrope of carbon, tube-shaped material, made of carbon, and having diameter measuring on the nanometre scale. They consist of single-walled (SWCNTs) and multi-walled (MWCNTs) carbon nanotubes. CNTs suspensions offer a higher thermal property contrasted with those different nanoparticles with a similar volume fraction [14,15]. In this way CNTs can improve both convective heat transfer and thermal conductivity of base fluids. Imtiaz et al. [16] studied convective flow of CNTs between rotating disks, while Hayat et al. [17] investigated on 3D

rotating flow of CNTs. Other than that, many other researchers also come across the advantages of CNTs and investigated various boundary layer problem on CNTs [18-20].

There have been a lot of research involving stagnation point flow over a permeable plate [21-24]. Najib et al. [25] studied stagnation point flow over an exponentially shrinking cylinder with suction/injection. While in this present paper, we study the stagnation point flow over a linearly stretching/shrinking sheet. We are also studying the boundary impacts of suction and injection. As a consequence, the suction or injection of a fluid through the bounding surface, such as in mass transfer cooling, can significantly change the flow field and thus impact the transfer rate of heat.

Methodology

Incompressible steady flow toward a stretching/shrinking sheet in carbon nanotubes with the presence of suction/injection was investigated. The stretching/shrinking velocity $U_w(x)$ and the ambient fluid velocity $U_\infty(x)$ are assumed to vary linearly from the stagnation point, i.e., $U_w(x) = ax$ and $U_\infty(x) = bx$, where a and b are constants. Two types of CNTs, single- and multi-wall were used with water as the base fluid. The boundary layer equations can be written as follows [26]:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0, \quad (1)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = U_\infty \frac{dU_\infty}{dx} + \frac{\mu_{nf}}{\rho_{nf}} \frac{\partial^2 u}{\partial y^2}, \quad (2)$$

$$u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \alpha_{nf} \frac{\partial^2 T}{\partial y^2} \quad (3)$$

subject to the boundary conditions

$$\begin{aligned} u &= U_w, v = V_w, T = T_w \text{ at } y = 0 \\ u &\rightarrow U_\infty, T \rightarrow T_\infty \text{ as } y \rightarrow \infty \end{aligned} \quad (4)$$

The velocity components in x and y directions are namely u and v respectively. It should be mention that $V_w = -(v_f b)^{1/2} S$, where $S > 0$ for suction and $S < 0$ for injection. Additionally, L denotes the slip length, μ_{nf} , α_{nf} , ρ_{nf} are the viscosity, thermal diffusivity and density of the nanofluid, respectively, which are given by Oztop and Abu-Nada [27]:

$$\begin{aligned} \alpha_{nf} &= \frac{k_{nf}}{(\rho C_p)_{nf}}, \mu_{nf} = \frac{\mu_f}{(1 - \varphi)^{2.5}}, \rho_{nf} = (1 - \varphi)\rho_f + \varphi\rho_{CNT}, \\ (\rho C_p)_{nf} &= (1 - \varphi)(\rho C_p)_f + \varphi(\rho C_p)_{CNT}, \quad \frac{k_{nf}}{k_f} = \frac{1 - \varphi + 2\varphi \frac{k_{CNT}}{k_{CNT} - k_f} \ln \frac{k_{CNT} + k_f}{2k_f}}{1 - \varphi + 2\varphi \frac{k_f}{k_{CNT} - k_f} \ln \frac{k_{CNT} + k_f}{2k_f}} \end{aligned} \quad (5)$$

where φ is the nanoparticle volume fraction, $(\rho C_p)_{nf}$ and $(\rho C_p)_{CNT}$ are the heat capacity of the nanofluid and carbon nanotubes, k_{nf} is the thermal conductivity of nanofluid, k_f and k_{CNT} are the thermal conductivities of the fluid and carbon nanotubes, respectively, and ρ_f and ρ_{CNT} are the densities of the fluid and carbon nanotubes, respectively. The use of the term for k_{nf}/k_f were taken from Xue [28] where Maxwell's theory model considers the thermal conductivity impact of space distribution of CNTs.

A similarity transformation for Eqs. (1) – (4) can be written as follows:

$$\eta = \left(\frac{b}{v_f}\right)^{1/2} y, \psi = (v_f b)^{1/2} x f(\eta), \theta(\eta) = \frac{T - T_\infty}{T_w - T_\infty} \quad (6)$$

where η is the similarity variable and ψ is the stream function defined as $u = \partial\psi/\partial y$ and $v = -\partial\psi/\partial x$, which identically satisfies Eq. (1). Employing the similarity variables (6), Eqs. (2) - (3) reduce to the following ordinary differential equations as:

$$\frac{1}{(1-\varphi)^{2.5}(1-\varphi+\varphi\rho_{CNT}/\rho_f)} f'''' + f f'' - f'^2 + 1 = 0 \quad (7)$$

$$\frac{1}{Pr} \left[\frac{k_{nf}/k_f}{1-\varphi+\varphi(\rho C_p)_{CNT}/(\rho C_p)_f} \right] \theta'' + f \theta' = 0 \quad (8)$$

subjected to the boundary conditions (4) which become

$$f(0) = S, f'(0) = \varepsilon, \theta(0) = 1 \\ f'(\eta) \rightarrow 1, \theta(\eta) \rightarrow 0 \text{ as } \eta \rightarrow \infty. \quad (9)$$

In the Eqs. (7) – (9), the prime notation is denoting differentiation with respect to η , Pr is the Prandtl number and ε is the velocity ratio parameter defined as

$$Pr = \frac{v_f}{\alpha_f}, \quad \varepsilon = \frac{a}{b} \quad (10)$$

where $\varepsilon > 0$ for stretching and $\varepsilon < 0$ for shrinking.

The physical quantities of interest, which are the skin friction coefficient C_f and the local Nusselt number Nu_x , are defined as

$$C_f = \frac{\tau_w}{\rho_f U_\infty^2}, \quad Nu_x = \frac{x q_w}{k_f (T_w - T_\infty)} \quad (11)$$

where the surface shear stress τ_w and the surface heat flux q_w are given as

$$\tau_w = \mu_{nf} \left(\frac{\partial u}{\partial y} \right)_{y=0}, \quad q_w = -k_{nf} \left(\frac{\partial T}{\partial y} \right)_{y=0} \quad (12)$$

with μ_{nf} and k_{nf} being the dynamic viscosity and thermal conductivity of the nanofluids, respectively. Using the similarity variables (6) into Eq. (11), we obtain

$$C_f Re_x^{1/2} = \frac{1}{(1-\varphi)^{2.5}} f''(0), \quad (13)$$

$$Nu_x / Re_x^{1/2} = -\frac{k_{nf}}{k_f} \theta'(0), \quad (14)$$

where $Re_x = U_\infty x / v_f$ is the local Reynolds number.

Results

The system of Eqs. (7) - (8) together with conditions in (9) are solved numerically using the bvp4c package in Matlab. Following to Oztop and Abu-Nada [27], we have considered the range of carbon nanotube volume fraction φ as $0 \leq \varphi \leq 0.2$, where $\varphi = 0$ is regular fluid with Prandtl number $Pr = 6.2$. The thermophysical properties of the base fluid and the carbon nanotubes are listed in Table 1.

Table 1: Thermophysical properties of CNTs (Khan et al. [29])

Physical properties	Base fluids	Nanoparticle	
	Water ($Pr = 6.2$)	SWCNT	MWCNT
ρ (kg/m^3)	997	2600	1600
c_p (J/kgK)	4179	425	796
k (W/mK)	0.613	6600	3000

Figures 1 and 2 illustrate the variation of $f''(0)$ and $-\theta'(0)$ with some values of the stretching/shrinking parameter ε , for three different values of carbon nanotube volume fraction ϕ , where $\phi = 0, 0.1$ and 0.2 for water-SWCNTs. It can be seen that there exist dual solutions when $\varepsilon_c < \varepsilon \leq -1$, while a unique solution exists when $\varepsilon > -1$ and no solutions are found to exist when $\varepsilon < \varepsilon_c < 0$, where ε_c is the critical value of ε .

Figures 3 and 4 show the variation of $f''(0)$ and $-\theta'(0)$ with some values of the stretching/shrinking parameter ε , for three different values of suction/injection parameter S , where $S = -0.5, 0$ and 0.5 for water base fluid when $\phi = 0.1$. When parameter S increases, the range of solutions exist becomes larger. When $\phi = 0$ and $S = 0.5$, the value of $\varepsilon_c = -1.5222$, which results in good agreement with Bhattacharyya and Layek [30]. Furthermore, as parameter S increases, the skin friction and heat loss from the surface increases. Hence, suction delays the boundary layer separation, while injection accelerates it.

Figures 5 and 6 illustrate the variations of the skin friction coefficient and the local Nusselt number, given by Eqs. (13) - (14) with the carbon nanotube volume fraction ϕ for three different values of S parameter which are $S = -0.5, 0$ and 0.5 with $\varepsilon = 0.5$. It is concluded that, when the S parameter increases, the skin friction coefficient increases as well as the local Nusselt number also increases. The skin friction coefficient and local Nusselt number of SWCNTs is found to be higher than MWCNTs. This is because density and thermal conductivity of SWCNTs are found to be higher than MWCNTs, refer to Table 1. Besides that, when the carbon nanotube volume fraction ϕ increases, the skin friction coefficient and local Nusselt number are also increases.

The velocity and temperature profiles for some values of S and carbon nanotubes parameters are presented in Figures 7-10. The term for both first and second solutions refers to the curves shown in Figures 1-4, where the first solution has larger value of $f''(0)$ and $-\theta'(0)$ compared to the second solution. These profiles satisfy the boundary conditions (9) asymptotically, which the supporting the existence of dual solutions shown in Figures 1-4.

Conclusions

We have studied theoretically and analysed the effects of carbon nanotubes volume fraction and suction/injection effect on the stagnation point flow over a stretching/shrinking sheet. The results indicate that

1. Solutions for a stretching sheet are unique and solutions for a shrinking sheet are non-unique.
2. The range of solutions widen with an increases of suction parameter, while injection parameter acted on contrary.
3. The skin friction and heat transfer increases with an increase of suction/injection parameter.
4. Single-walled CNTs are more efficient than multi-walled CNTs in both skin friction and local Nusselt number.

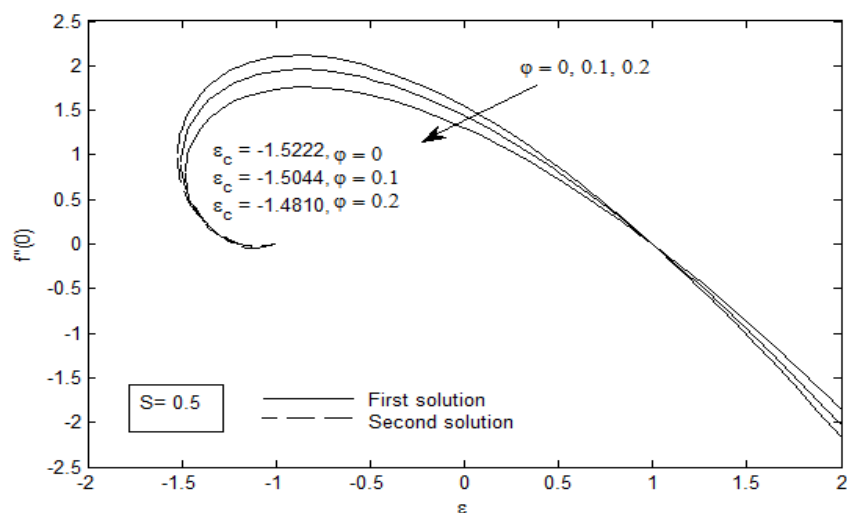


Fig. 1. Variation of $f''(0)$ with ε for some values of φ for water-SWCNT.

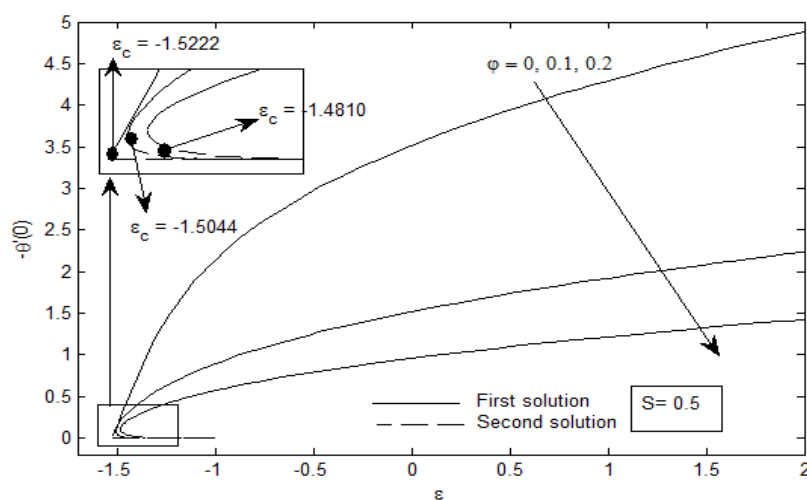


Fig. 2. Variation of $-\theta'(0)$ with ε for some values of φ for water-SWCNT when $Pr = 6.2$.

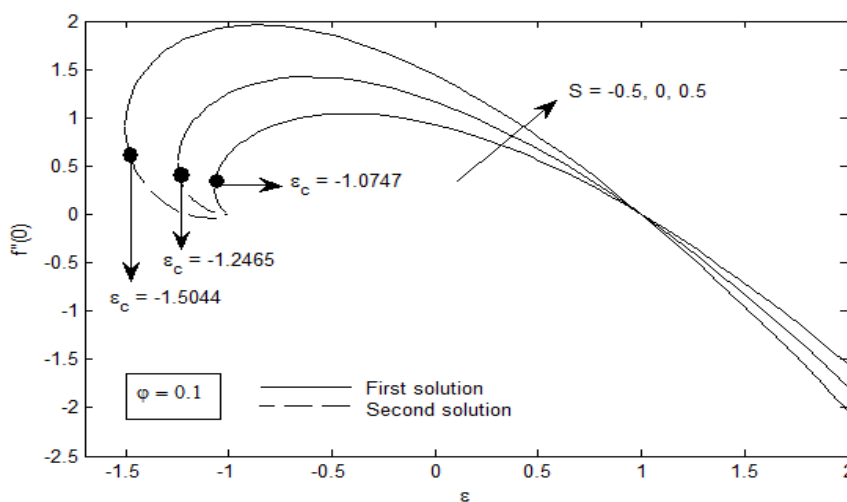


Fig. 3. Variation of $f''(0)$ with ε for some values of S for water-SWCNT.

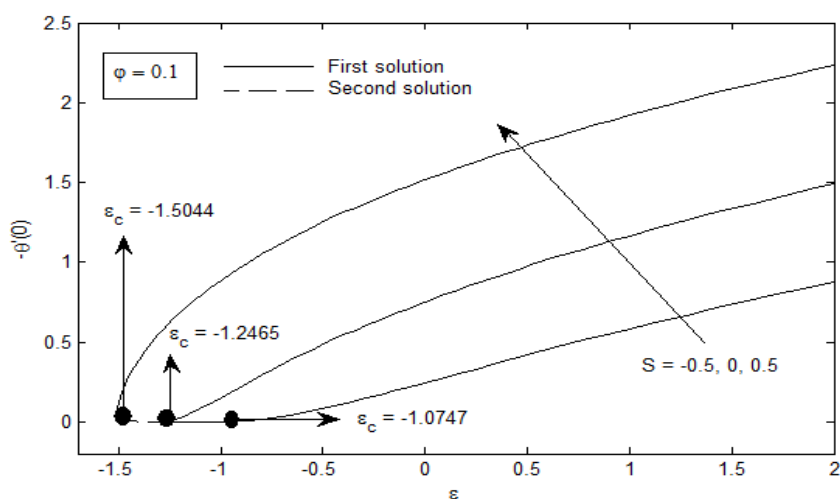


Fig. 4. Variation of $-\theta'(0)$ with ε for some values of S for water-SWCNT when $Pr = 6.2$.

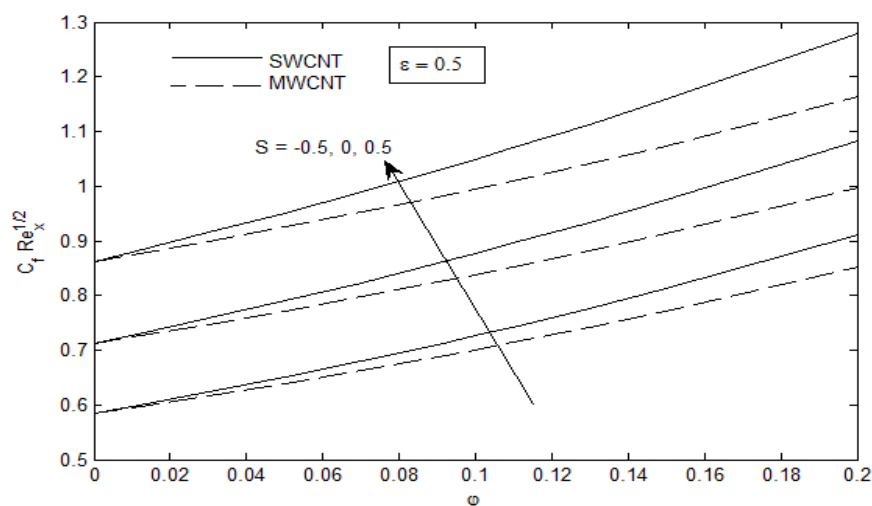


Fig. 5. Variation of skin friction coefficient with φ for different S and carbon nanotubes.

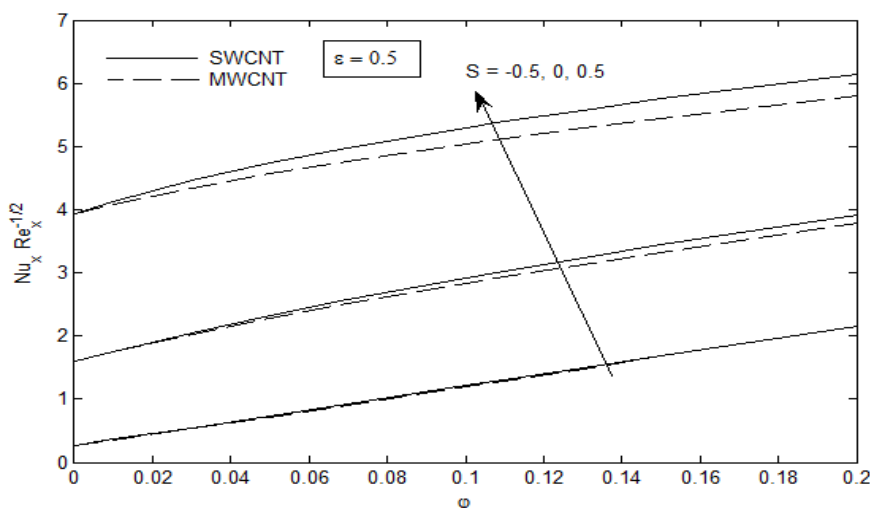


Fig. 6. Variation of local Nusselt number with φ for different S and carbon nanotubes.

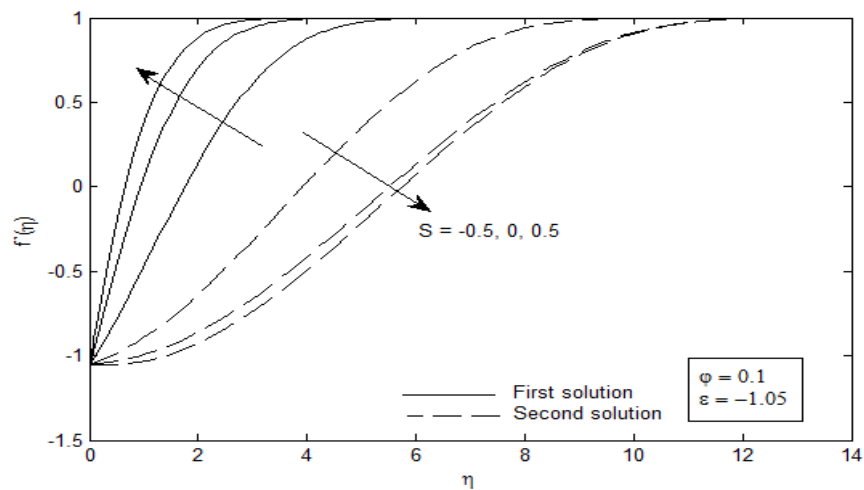


Fig. 7. Velocity profiles some values of S for water-SWCNT.

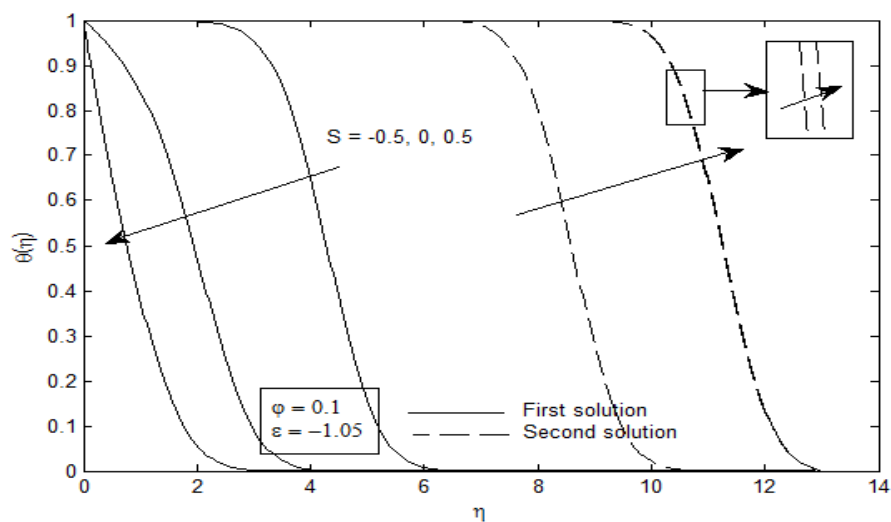


Fig. 8. Temperature profiles some values of S for water-SWCNT.

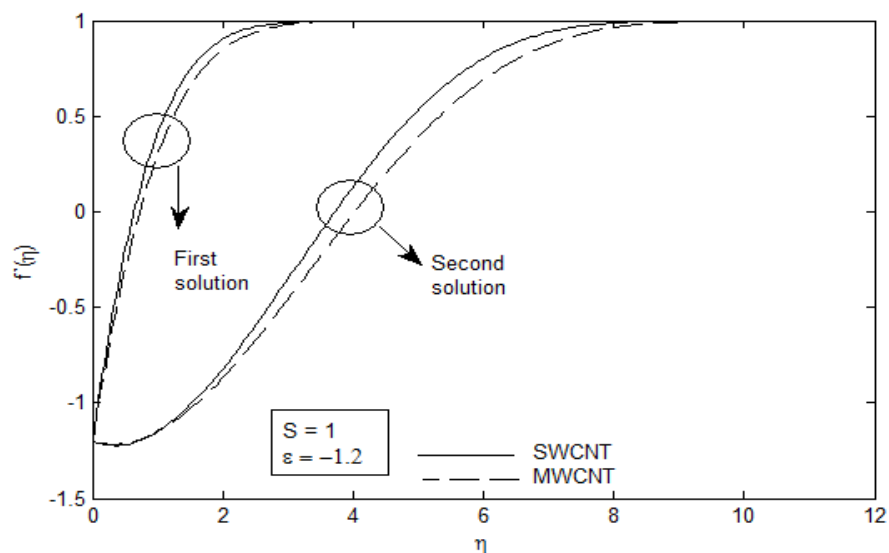


Fig. 9. Velocity profiles for different types of carbon nanotubes with water base fluid.

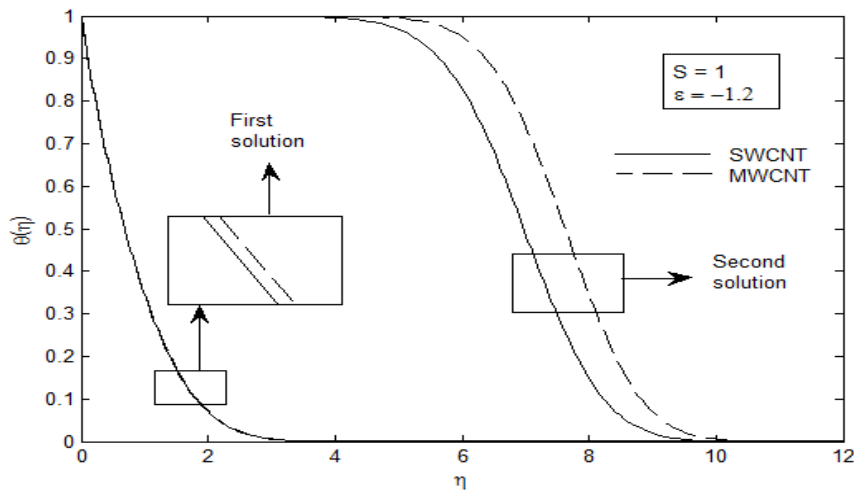


Fig. 10. Temperature profiles for different types of carbon nanotubes with water base fluid.

Acknowledgement

The authors gratefully acknowledge the financial support received in the form of a fundamental research grant scheme (FRGS/1/2018/STG06/UPM/02/4/5540155). Finally, a modest yet sincere gratitude for the honest feedback and suggestions from all reviewers.

References

- [1] Hiemenz, K. (1911). "Die Grenzschicht an einem in den gleichförmigen Flüssigkeitsstrom eingetauchten geraden Kreiszylinder." *Dingler's Polytech J.* 326:321-324.
- [2] Homann, F. (1936), "Der Einfluss grosser Zähigkeit bei der Stromung um den Zylinder und um die Kugel". *Z Angew Math Mech.* 16:153-164.
- [3] Mahapatra, TR., Gupta, AS. (2002). "Heat transfer in stagnation-point flow towards a stretching sheet." *Heat Mass Tran.* 38:517-521.
- [4] Mahapatra, TR., Gupta, AS. (2003). "Stagnation-point flow towards a stretching surface." *Can J Chem Eng.* 81:258-263.
- [5] Crane, L.J. (1970). "Flow Past a Stretching Plate." *Zeit. Angew. Math. Phys.* 21:645-647.
- [6] Khan, AV., Pop, I. (2010). "Boundary layer flow of a nanofluid past a stretching sheet." *Int J Heat Mass Tran.* 52:3187-3196.
- [7] Lok, YY., Ishak, A., Pop, I. (2011). "MHD stagnation-point flow towards a shrinking sheet." *Int J Numer Meth Heat Fluid Flow.* 21:61-72.
- [8] Ishak, A., Lok, YY., Pop, I. (2010). "Stagnation-point flow over a shrinking sheet in a micropolar fluid." *Chem Eng Comm.* 197:1417-1427.
- [9] Bachok, N., Ishak, A., Pop, I. (2010). "Melting heat transfer in boundary layer stagnation-point flow towards a stretching/shrinking sheet." *Phys Lett A.* 374:4075-4079.
- [10] Bachok, N., Ishak, A., Pop, I. (2011). "On the stagnation-point flow towards a stretching sheet with homogeneous-heterogeneous reactions effects." *Comm Nonlinear Sci Numer Simulat.* 16:4296-4302.
- [11] Othman, NA., Yacob, NA., Bachok, N., Ishak, A., Pop, I. (2017) "Mixed convection boundary-layer stagnation point flow past a vertical stretching/shrinking surface in a nanofluid." *Applied Thermal Engineering.* 115:1412-1417.
- [12] Bakar, NSA., Bachok, N., Arifin, NM., Pop, I. (2018). "Stability analysis on the flow and heat transfer of nanofluid past a stretching/shrinking cylinder with suction effect." *Results in Physics.* 9:1335-1344.

- [13] Choi, SUS., Zhang, ZG., Yu, W., Lockwood, FE., Grulke, EA. (2001). "Anomalous thermal conductivity enhancement in nanotube suspensions." *Appl. Phys. Lett.* 79: 2252–2254.
- [14] Maré, T., Halelfadl, S., Sow, O., Estellé, P., Duret, S., Bazantay, F. (2011). "Comparison of the thermal performances of two nanofluids at low temperature in a plate heat exchanger." *Exp. Therm. Fluid Sci.* 35:1535–1543.
- [15] Liu, MS., Lin, MCC., Huang, IT., Wang, CC. (2005). "Enhancement of thermal conductivity with carbon nanotube for nanofluids." *Int. Commun. Heat Mass Transf.* 32:1202–1210.
- [16] Imtiaz, M., Hayat, T., Alsaedi, A., Ahmad, B. (2016). "Convective flow of carbon nanotubes between rotating stretchable disks with thermal radiation effects." *Int. J. Heat Mass Transf.* 101:948–957.
- [17] Hayat, T., Haider, F., Muhammad, T., Alsaedi, A. (2017). "Three dimensional rotating flow of carbon nanotubes with Darcy-Forchheimer porous medium." *PLoS ONE.* 12:e0179576.
- [18] Hayat, T., Hussain, Z., Alsaedi, A., Ahmad, B. (2016). "Heterogeneous-homogeneous reactions and melting heat transfer effects in flow with carbon nanotubes." *J. Mol. Liq.* 220:200–207.
- [19] Sreedevi, P., Reddy, PS., Chamka, AJ. (2018). "Magneto-hydrodynamics heat and mass transfer analysis of single and multi-wall carbon nanotubes over vertical cone with convective boundary condition." *Int. J. Mech. Sci.* 135:646–655.
- [20] Anuar, NS., Bachok, N., Ariffin, MN, Rosali, H. (2019). "Stagnation point flow and heat transfer over an exponentially stretching/shrinking sheet in CNT with homogeneous-heterogeneous reaction:stability analysis." *Symmetry* 2019. 11:522.
- [21] Bachok, N., Ishak, A., Pop, I. (2010). "Mixed convection boundary layer flow over a permeable vertical plate embedded in an anisotropic porous medium." *Math. Prob. Eng.* 2010:659023.
- [22] Ishak, A., Nazar, R., Pop, I. (2007), "Boundary layer on a moving wall with suction and injection." *Chin. Phys. Lett.* 24:2274-2276.
- [23] Bachok, N., Ishak, A., Nazar, R., Senu, N. (2013). "Stagnation point flow over a permeable stretching/shrinking sheet in a copper-water nanofluid." *Boundary Value Prob.* 2012:1-10.
- [24] Al Sanea, SA. (2004). "Mixed convection heat transfer along a continuously moving heated vertical plate with suction or injection." *Int. J. Heat Mass Trans.* 47:1445-1465.
- [25] Najib, N., Bachok, N. Ariffin, MN. (2014). "Stagnation point flow and heat transfer over an exponentially shrinking cylinder with suction/injection". *Comps. and Maths. in Automation and Materials Sci.* 136-140.
- [26] Ahmad S, Rohni AM, Pop I. (2011) Blasius and Sakiadis problems in nanofluids. *Acta Mech.* 218:195-204.
- [27] Oztop HF, Abu-Nada E. (2008) Numerical study of natural convection in partially heated rectangular enclosures filled with nanofluids. *Int J Heat Fluid Flow.* 29:1326-1336.
- [28] Xue QZ. (2005) Model of thermal conductivity of carbon nanotube-based composites. *Phys. B Condens. Matter.* 368:302-307.
- [29] Khan WA, Khan ZH, Rahi M. (2014) Fluid flow and heat transfer of carbon nanotubes along a flat plate with Navier slip boundary. *Appl. Nanosci.* 4:633-641.
- [30] Bhattacharyya, K., Layek, GC. (2011). "Effects of suction/blowing on steady boundary layer stagnation-point flow and heat transfer towards a shrinking sheet with thermal radiation." *Int. Journal of Heat and Mass Trans.* 54:302-207.

GRAVITY-AIDED ULTRASONIC SEPARATION OF NANOPARTICLES FROM LIQUID USING MACRO-SCALE SEPARATOR

Khin Nwe Zin Tun¹

Thein Min Htike²

Khine Zin Mar³

¹Department of Mechanical Engineering, Yangon Technological University, Myanmar

²Department of Mechanical Engineering, Yangon Technological University, Myanmar

³Department of Mechanical Engineering, Yangon Technological University, Myanmar

Abstract: *Acoustophoresis is the technology to separate the microparticles and cells from suspending fluid. This research focuses on the separation of nanoparticles from water by using macro-scale fluidic separator which works based on gravity-aided ultrasonic standing wave technology. Titanium dioxide particles of 40 nm diameter were concentrated by the combination of ultrasonic standing wave field at 2.2 MHz and gravity-aided sedimentation. The purpose of this study is to investigate the performance of gravity-aided ultrasonic particle to concentrate nanoparticles. It was found that the separation efficiency is 83% at a flow rate of 0.1 mL/min. FEM simulations were also conducted to evaluate characteristics of variation of acoustic energy inside the fluidic channel. Results indicate that nanoparticles can be concentrated using gravity-aided ultrasonic standing wave field, however optimization of the design of the fluidic channel is required for increasing throughput of the separator.*

Keywords: *ultrasonic standing wave, acoustic radiation force, macro-scale fluidic separator, microparticles.*

Introduction

Particle separation methods have wide applications in engineering and life sciences technology. Using ultrasonic standing wave is one way of manipulating the particles in microfluidic systems. This ultrasound method is suitable for both biological and non-biological suspended micro-particles. The particle separation using ultrasonic standing wave relies on acoustic radiation force being directed on the particles either in a macro- or micro-channel both with flowing and stationary fluid volumes. The acoustic radiation force depends on the contrast between the mechanical parameters density and compressibility of the particles and the fluid.

Millimetre scale ultrasonic concentrator was first designed by Frank et al. in 1993 with the use of acoustic foil inside it. Later, this ultrasonic concentrator design was improved by Hill and Wood in which the use of acoustic foil was omitted. The Martyn Hill [4] designed the layer of resonators for ultrasonic particle manipulation by using one dimensional simulation model. This study observed different coupling layer materials (brass, aluminium, macor) of quarter wavelength resonator predicted the nodal position and device behaviour. Süleyman Büyükköçak et al. [10] discussed the particle trajectories of Y-shaped channel. Fluid flow inside the microchannel geometry operated at resonance frequency was modeled FEM based numerical simulation. The numerical prediction was validated with experimental result from the literature. Similar study the locations of particles prediction using a finite element model by Adrian Neild et al. [3]. They used simple model with resonance frequency to estimate the number of lines formed at this channel. Henrik Bruus et al. calculated the theory of acoustic radiation force on a second order, time average effect, micrometer sized particles in ultrasonic

field. Microchip-based devices for on-line particle separation were developed by Andreas Nilsson. [2] This on-line separation, 90% of the particles were recovered the acoustic separator at fundamental resonance frequency 2 MHz. Most of the models discussed earlier are based on one-dimensional analysis. The two-dimensional modelling of the separation of suspended particle as a function of time was reported by Francisco J. Trujillo et.al in 2013 by using finite element analysis.

Most studies literature focus on separation, trapping or concentration of microparticles. This study will focus on separation of nanoparticles using the ultrasonic standing wave field and gravity-aided sedimentation. Separation performance of macro-scale acoustic concentrator in flow field was investigated and two-dimensional acoustic simulation model was also constructed to analyze variation of sound field inside the fluidic channel.

Theory

The acoustic radiation force was calculated using Gor'kov's formulation [3]. The acoustic radiation force is defined as a function of a potential U.

$$F_a = -\nabla U \quad (1)$$

The force is the negative gradient of the force potential. Where the potential U is defined as,

$$U = 2\pi\rho_w r^3 \left[\frac{f_1}{3\rho_w^2 c_w^2} \langle p \rangle^2 - \frac{f_2}{2} \langle v \rangle^2 \right] \quad (2)$$

$$f_1 = 1 - \frac{\rho_w c_w^2}{\rho_p c_p^2}, \quad f_2 = \frac{2(\rho_p - \rho_w)}{2\rho_p - \rho_w} \quad (3)$$

So, r is the radius of particles, c is the speed of sound, p is the acoustic pressure and v is the fluid particles velocity. w and p refer to water and particle properties. And the acoustic number k is defined by $2\pi/\lambda$.

The notable aspect of the acoustic contrast factor is the possible sign change depending on the densities and compressibility of fluid and particles.

$$\emptyset = \frac{1}{3} \left(\frac{5\rho_p - 2\rho_w}{2\rho_p + \rho_w} - \frac{\rho_w c_w^2}{\rho_p c_p^2} \right) \quad (4)$$

Where, ρ_p = densities of particle

ρ_w = densities of water

c_p = sound speed of particle

c_w = sound speed of water

From the equation (4), when $\emptyset$ is positive, particles aggregate at the nodes of the sound pressure profile and other they aggregate at the anti-nodes.

The forces that affect the particle movement are not only acoustic radiation force, but also the drag forces on the particles due to the fluid flow. The stokes drag force F_{drag} can be expressed as:

$$F_{\text{drag}} = 6\pi\eta r(v_f - v_p) \quad (5)$$

For a spherical particle with radius r , moving at the fluid and particle velocity are v_f and v_p through a liquid with viscosity μ .

The relative densities of the particle and fluid control, the buoyancy force on a particle, where F_b is opposite to the gravitational force;

$$F_b = \frac{4}{3} \pi r^3 g (\rho_f - \rho_p) \quad (6)$$

where r is the radius of the particle, g is the gravitational force, ρ_f and ρ_p are the densities of the fluid and the particle. When referenced to the x and y coordinates of the particle, the x and y components of the buoyancy force depend on the orientation of the channel.

Experimental Setup

The experimental setup system involves four main parts, a function generator, an acoustic chamber, syringe pump and an oscilloscope as shown in Figure 1. The function generator (TEXIO, FGX-2005) is a power source to the piezoelectric transducer. The purpose of using a piezoelectric transducer is to induce waves in the ultrasonic frequency range to the aluminium channel. This creates acoustic standing waves on the channel that will result in acoustic radiation force on the particles. Oscilloscope (GDS-1102A-U) analyze the amplitude, frequency, and voltage of the observed waveform. Moreover, the resonance frequency of the transducer can be measured from oscilloscope. Syringe pump (FGX-2005) is used to pump the water-particle solution into the acoustic separator through the connector. The pumping flow rate can be adjusted in it.

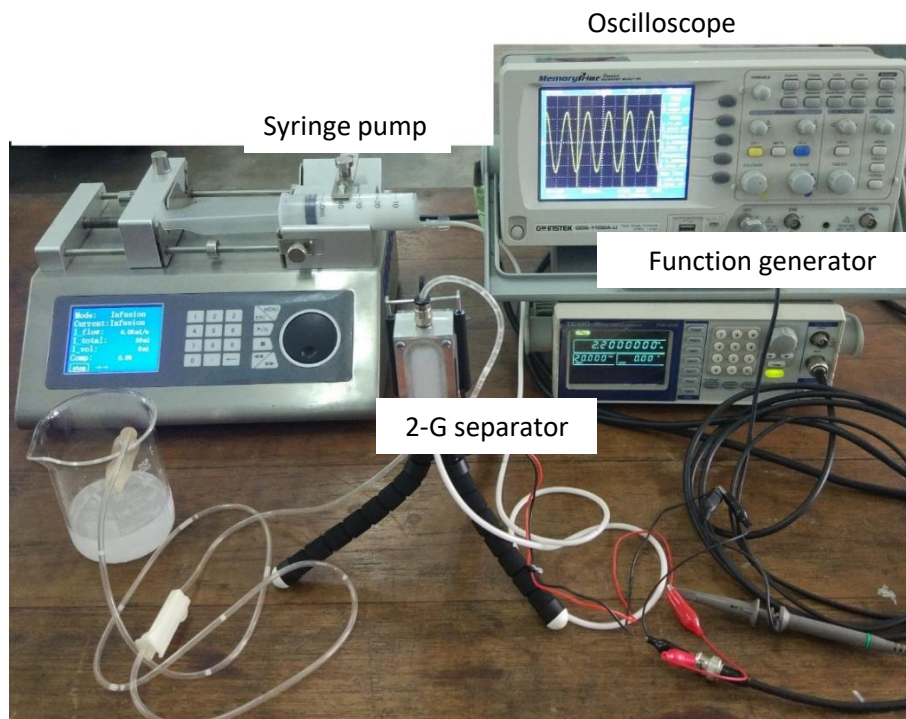


Fig. 1. Experimental Setup

Numerical Analysis

This simulation performed with COMSOL Multiphysics software in two dimensional finite element model of fluid layer was constructed using acoustic module. The rectangular model is 0.016m width and 0.04m length. For this simulation, maximum mesh size of the element is

around $\lambda/10$. The input parameters for the simulations are density of water $\rho_w = 1000 \text{ kg/m}^3$, viscosity $\mu = 0.9 \times 10^{-3} \text{ kg/ms}$, speed of sound of water $c = 1500 \text{ m/s}$, particle diameter = $10 \mu\text{m}$ and density of particle $\rho_p = 1050 \text{ kg/m}^3$.

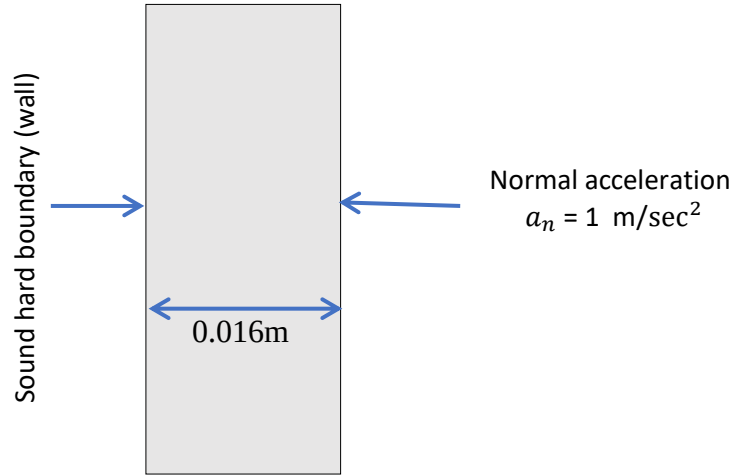


Fig. 2. Two-dimensional channel geometry

Fig. 4 shows the simplified model with boundary conditions. All the rigid walls of the channel were modeled as sound hard boundaries, except the wall where the piezo-actuator was attached. This side of the model was simplified by using an acceleration boundary:

$$-n \cdot \left(-\frac{1}{\rho_c} (\nabla p_t - q_d) \right) = a_n \quad (7)$$

The sound hard boundary was defined by:

$$-n \cdot \left(-\frac{1}{\rho_c} (\nabla p_t - q_d) \right) = 0 \quad (8)$$

Results and Discussion

Experimental Result

The fluid particle solution is inserted from the syringe pump into the separator through the inlet at the flow rate of 0.1 ml/min. The particles experience the acoustic radiation forces which are induced by the ultrasonic standing wave field and will move to the pressure node points. The separation lines or node points are obvious at the resonance frequency of 2.2 MHz as shown in Figure 3.

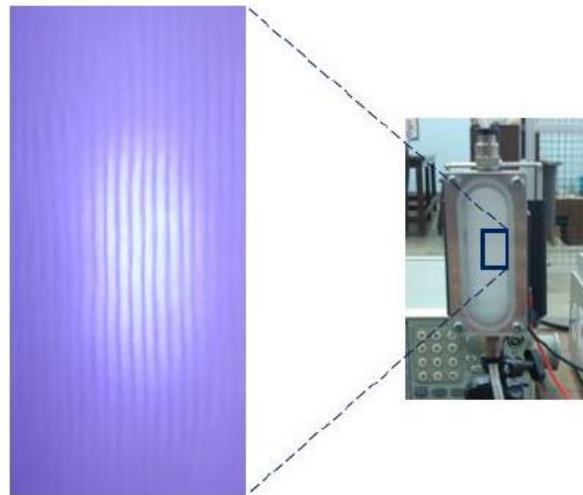


Fig. 3. Pressure Nodal Points in Fluid Layer

According to the ultrasonic standing wave principle, the trapped particles settle down when they reach critical volume. Therefore, the clean fluid is flowed through the outlet into the beaker and the settled particles will remain in the separator near inlet of the fluid flow. The particle concentration from the inlet and outlet of the separator is shown in Figure 4.

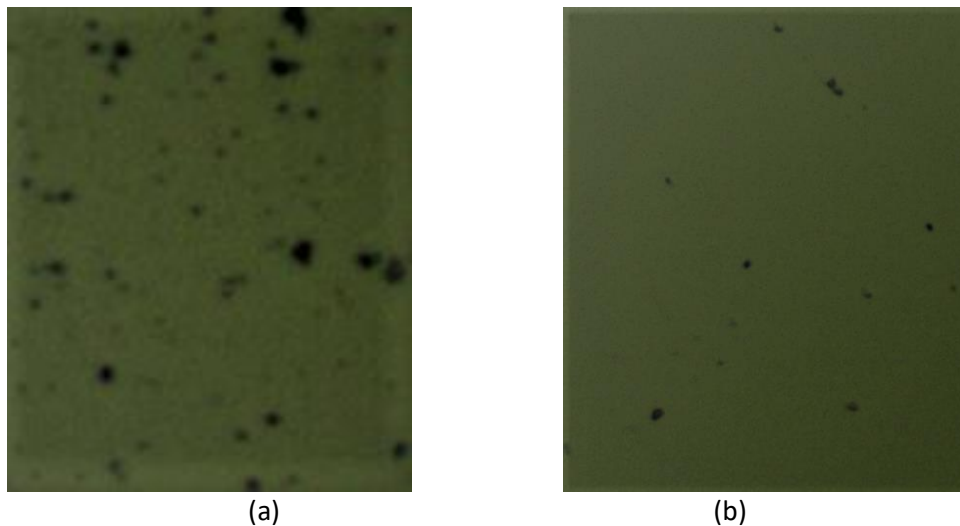


Fig. 4. Particle Concentration Image from CCD Camera with Microscope; (a) Inlet Concentration (b) Outlet Concentration

Figure 4 is the image of particle concentration in inlet and outlet solution in the area of 415×495 pixel square. According to Figure 4, the number of particles is 70 in inlet concentration and 12 in outlet concentration. Therefore, this separator can separate 82.85 % of the particles suspended in water.

Numerical Result

Figure 5(a) shows two-dimensional numerical simulation results for acoustic pressure distribution in the separator. The equally spaced lines along the separator show that there is standing wave across the separator width. According to Figure 5(a), the maximum acoustic pressure is 1.06×10^4 Pa and the minimum acoustic pressure is -1.06×10^4 Pa.

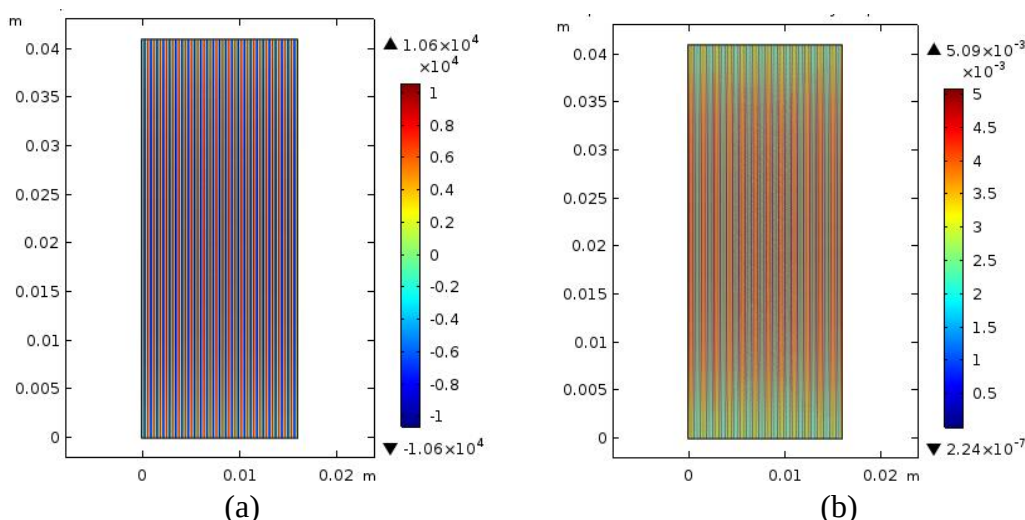


Fig. 5. Two-dimension model; (a) Acoustic Pressure and (b) Displacement Velocity Distribution

Absolute displacement velocity in the separator is plotted in Figure 5(b). From the comparison, it can be clearly seen that the velocity and the pressure plots have phase difference since the theoretical phase difference between the pressure and velocity is $\pi/2$.

Conclusions

In this paper, the separation of nanoparticles from suspending fluid was done using ultrasonic standing wave technology and gravity-aided sedimentation. Titanium dioxide nanoparticles were found to respond to primary acoustic radiation force at operating frequency 2.2 MHz and they formed agglomerates very quickly due to lateral radiation force as well as secondary radiation force. Once the agglomerates become large enough, they sediment at the bottom of the channel due to gravity. In this way, separation effect was achieved. The experimental results show that separation efficiency of 82.85 % can be achieved at a flow rate is 0.1 ml/min. However, the flow rate is still very low and the optimization of channel design is necessary for enhancing the flow rate. Other modes of operation such as sweeping mode could also assist to enhance flow rate. Simulation results indicated that there is lateral radiation force in the resonator which is responsible for formation particle agglomerates hence in good agreement with experimental observation. Future research should focus on optimization of the ultrasonic separation design with other modes of operation. Moreover, simulation model could be extended to 3D to investigate three-dimensional variation of the sound field inside the channel. Simulation of couples flow and acoustic field with particle trajectory model should also be done to identify separator's performance and it could be used for multiphysics optimization of the performance of the channel.

References

- [1] Jeremy J. Hawkes, 1998. "Microparticle manipulation in millimetre scale ultrasonic standing wave chambers", *Ultrasonics* 36, 925–931.
- [2] Andreas Nilsson, 2004. "Acoustic control of suspended particles in micro fluidic chips", *Lab on a Chip*, 2004-4, 131-135.
- [3] Adrian Neild, 2006. "Finite element modeling of a microparticle manipulator", *Ultrasonics* 44, e455–e460.
- [4] Martyn Hill et.al. 2008. "Modelling for the robust design of layered resonators for ultrasonic particle manipulation", *Ultrasonics* vol.48, pp. 521–528.

- [5] Cho, S. H, 2008. “Particle separation and control technology using ultrasound,” *Noise & Vibration* 18(5) : 14-19.
- [6] B. Lipkens, 2010. “Separation of bacterial spores from flowing water in macro-scale cavities by ultrasonic standing waves” *International Congress on Ultrasonics*.
- [7] Thein Min Htike and Kian-meng Lim, 2012. “Ultrasonic Concentration And Trapping of Microsized Particles in A Fluid Suspension”, *International journal of modern physics: conference series* Vol. 19.
- [8] Bruus H, 2012 “ Acoustofluidics 7: The acoustic radiation force on small particles,” *Lab On A chip* 12(6):1014–1021.
- [9] Francisco J. Trujillo, 2013. “Multiphysics modelling of the separation of suspended particles via frequency ramping of ultrasonic standing waves”, *Ultrasonics Sonochemistry* 20,655–666.
- [10] Süleyman Büyükkoçak et al. 2014. “Finite element modeling of micro-particle separation using ultrasonic standing waves”, *ICNMM2014*.

EFFECTS OF MOISTURE CONTENT AND TEMPERATURE ON THE DIELECTRIC STRENGTH OF TRANSFORMER INSULATING OIL

Sharin Ab Ghani¹
Norazhar Abu Bakar²
Imran Sutan Chairul³
Mohd Shahril Ahmad Khair⁴
Nur Hakimah Ab Aziz⁵

¹High Voltage Engineering Research Laboratory, Centre for Robotics and Industrial Automation, Faculty of Electrical Engineering, Universiti Teknikal Malaysia Melaka, Malaysia. (Email: sharinag@utem.edu.myutem)

²High Voltage Engineering Research Laboratory, Centre for Robotics and Industrial Automation, Faculty of Electrical Engineering, Universiti Teknikal Malaysia Melaka, Malaysia.

³High Voltage Engineering Research Laboratory, Centre for Robotics and Industrial Automation, Faculty of Electrical Engineering, Universiti Teknikal Malaysia Melaka, Malaysia.

⁴High Voltage Engineering Research Laboratory, Centre for Robotics and Industrial Automation, Faculty of Electrical Engineering, Universiti Teknikal Malaysia Melaka, Malaysia.

⁵High Voltage Engineering Research Laboratory, Centre for Robotics and Industrial Automation, Faculty of Electrical Engineering, Universiti Teknikal Malaysia Melaka, Malaysia.

Abstract: *The primary function of insulating oils is to dissipate heat and provide electrical insulation for turn-to-turn windings in power transformers. These oils also serve as a means to monitor the health status of power transformers. At present, most transformer failures are due to the presence of moisture as well as heat, oxidation, and electrical stresses. The main objective of this research was to study the effects of moisture content and temperature on the dielectric strength of transformer insulating oil, which are known to be the factors that lead to transformer failures. The effect of moisture content was investigated by varying the amount of distilled water added into the insulating oil samples from 1 mL to 5 mL with 1-mL increments. The effect of temperature was investigated by increasing the oil temperature from 40°C to 120°C with 20°C increments. The dielectric strength of each insulating oil sample was assessed by performing AC breakdown voltage tests according to the MS IEC 60156:2012 standard. Based on the results, it can be concluded that moisture and temperature have a significant effect on the dielectric strength of transformer insulating oil.*

Keywords: *Dielectric strength, Insulating oil, Transformer, Moisture, Temperature*

Introduction

The objective of this research is to study the electrical and chemical properties of transformer insulating oils with the aim to understand the basics of the power transformer function and insulating oil. Power transformer is essentially a device in static conditions. As the name implies, power transformers transform electrical energy from one circuit to another while maintaining the same frequency. This frequency is 50 Hz in Malaysia. In addition, power transformers are essential equipment in power networks to convert voltage levels and maintain power flow. The transformer steps up or steps down the voltage level, depending on the requirements. The transformer consists of many parts from which the main ones are primary windings, secondary windings, and magnetic core. Transformer failures are caused by various factors such as overloading, short circuit, switching surges and etc. According to the statistics of transformer failures given by the respondents from International Council on Large Electric

Systems (Cigre), the most common cause of transformer failures is insulation failure [1]. Figure 1 shows the statistics of the causes of transformer failures based on 964 major failures. The insulation system plays a significant role in influencing the lifetime of a power transformer [2]. The primary and secondary windings as well as the magnetic core of the transformer are insulated and cooled by solid and liquid insulations [3]. Solid insulation comprises pressboard, wood and unbleached softwood pulp [4]. “Kraft paper”, which is a widely used solid insulation, is made from unbleached softwood pulp, which flows into the sulfate process. Liquid insulation plays a dual role in high-voltage transformers [5]: (1) as a coolant for heat dissipation and (2) as an electrical insulator. In this paper, liquid insulation refers to petroleum-derived mineral oils. It is evident that insulation failure makes up the highest percentage of transformer failure causes. Insulation system plays a vital role in influencing the lifetime of the transformer because it insulates and cools the transformer. Similar to the other high-voltage systems, transformers require insulating materials (solid and liquid insulations) to dissipate excessive heat [6]. The main objective of this research is to investigate the electrical and chemical properties of a petroleum-derived mineral oil. These properties were assessed according to the MS IEC 60156:2012 and ASTM D2144 -07(2013) standard to ensure reliability of the experimental results [7]. This research is focused on the effects of moisture content and temperature on the properties of the mineral insulating oil.

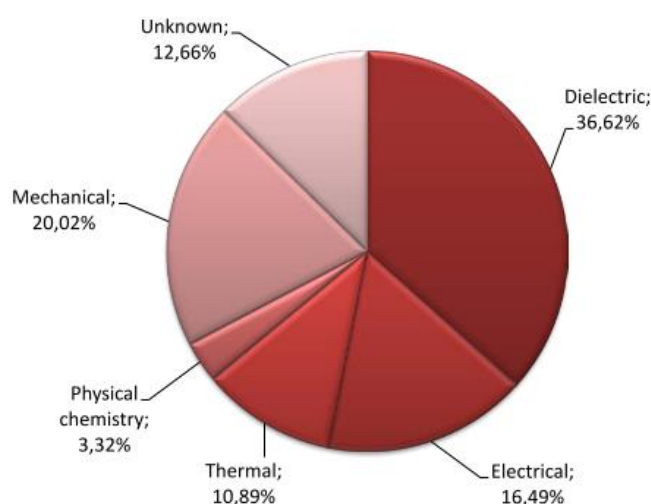


Figure 1: Causes of transformer failures based on 964 major failures according to the Cigre respondents [1]

As the name implies, liquid dielectric is a dielectric in liquid state. The main purpose of liquid dielectric (liquid insulation) is to prevent electrical discharge. In general, a good liquid dielectric is nonflammable and has high dielectric strength, good chemical stability, and good thermal properties [8]. In addition, a good liquid dielectric is capable of preventing breakdowns under electrical stresses. The electric capacitance per unit of volume of the liquid dielectric is determined by its dielectric constant, which is dependent on the operating temperature of the liquid dielectric and frequency or structure of its constituent molecules. Even though solid insulations are important in the electrical industry, liquid insulations are generally used in many electrical devices and equipment such as transformers, circuit breakers, capacitors, and cables. The properties of the liquid dielectric need to be assessed to determine whether the liquid dielectric is favorable to be used for transformer applications.

The distillation of crude petroleum produces a fraction of hydrocarbons, which are used to produce transformer insulating oils by refining. The boiling range of the collected fraction

and degree of refining are predefined prior to the process. The resultant product possesses properties that are suitable for transformer applications. Many international refining companies produce transformer insulating oils from crude petroleum. The physical, chemical, and electrical properties of the transformer insulating oils are determined to ensure that they conform with the specifications stipulated in transformer insulating oil standards [9][10]. Each property was compared with the specifications of the standard, which have been agreed between the insulating oil manufacturers and crude oil refiners. The amount of insulating oil contained in a transformer varies depending on the loading capacity and physical size of the transformer. For example, a 100-kVA rated distribution transformer contains ~140 liters of insulating oil. The viscosity of the insulating oil is particularly important for heat transfer by natural convection and it is a principal parameter in design calculations. When the transformer is in operation, the insulating oil is subjected to electrical and mechanical stresses. The interactions between the insulating oil and windings can lead to contamination, which will accelerate chemical reactions at high operating temperatures and alter the original properties of the insulating oil[11]. The electrical, physical, and chemical properties of the insulating oil need to be periodically tested to assess whether the oil is fit for service. In this research, the following electrical property (AC breakdown voltage or flashover point) was investigated whilst the chemical property examined was the functional groups. In general, it is crucial to determine the chemical compounds present in transformer insulating oils. The transformer insulating oil consists of various hydrocarbon compounds, each of which comes from different sources. The variations in chemical compounds reflect the behavior of different classes of hydrocarbons. The hydrocarbon compounds present in transformer insulating oil can be generally classified as naphthenes, aromatic hydrocarbons, and alkanes. The chemical formula of transformer oils is $C_{12}H_{10-x}Cl_x$, where x is polychlorinated biphenyl, with a value of 1–10 [5].

In this research, the effects on moisture content and temperature on the dielectric strength and chemical compounds of mineral insulating oil were analyzed. The dielectric strength (i.e., breakdown voltage) is the minimum voltage at which the liquid dielectric has lost some of its insulating property such that it becomes conductive. Breakdown occurs when the insulating oil reaches the stage where there is spark between the electrodes. The properties of the insulating oil at the atomic and molecular scale may play a role in influencing the breakdown voltage; however, in high-voltage breakdown tests, the electrode material, moisture content, gassing tendency, and temperature may also increase or decrease the breakdown voltage [12]. According to the MS IEC 60156:2012, the minimum breakdown voltage of transformer insulating oils is 30 kV [6]. In this research, the changes in the chemical compounds present in the insulating oils were analyzed using Fourier transform infrared (FTIR) spectroscopy, which is one of the most widely used analytical techniques. It is known that the presence of moisture has a significant effect on the insulating property of transformer insulating oils. Hence, ideally, the insulating oil should be free from moisture. In terms of the electrical properties, moisture can reduce the breakdown voltage of the insulating oil. The presence of moisture breaks down the molecular chains of the insulating oil [13], [14]. In terms of the chemical properties, the presence of moisture (H_2O) can be detected within a wavenumber range of 3600–3400 $1/cm$ in the FTIR spectra, such that the insulating oil becomes conductive [15], [16]. A higher insulating oil temperature can also affect the breakdown voltage and chemical makeup of the insulating oil. A higher temperature will increase the breakdown voltage of the insulating oil whereas the aliphatic (C–H) compound will decrease. The decrease in aliphatic compound appears as a strong absorption band within a wavenumber range of 3000–2850 $1/cm$, which is ascribed to C–H stretching [17].

Methodology

The methodology adopted in this research is described briefly in this section. A portable oil dielectric strength tester (Model: OTS60PB, Megger Ltd., UK) equipped with semi-spherical electrodes was used to conduct the breakdown voltage tests. The instrument can be used to test an insulating oil sample with a volume of 500 mL. An FTIR spectrometer (Model: FT/IR-6100, Jasco Inc., USA) was used to measure the FTIR spectra of the oil samples within a wavenumber range of 4000–400 1/cm. The instrument is equipped with an IR analyzer, which is a software package that enables one to search and access the reference IR spectra and analyze the chemical structures of the sample under investigation. The effects of moisture content were investigated by varying the amount of distilled water added into the oil sample from 1 mL to 5 mL with 1-mL increments whereas the effect of temperature was investigated by increasing the temperature of the oil sample from 40°C to 120°C with 20°C increments using a hot plate magnetic stirrer. Figure 2 shows the flow chart of the research methodology. The insulating oil used in the experiments was Hyrax Hypertrans uninhibited mineral oil, sourced from Hyrax Oil Sdn. Bhd.

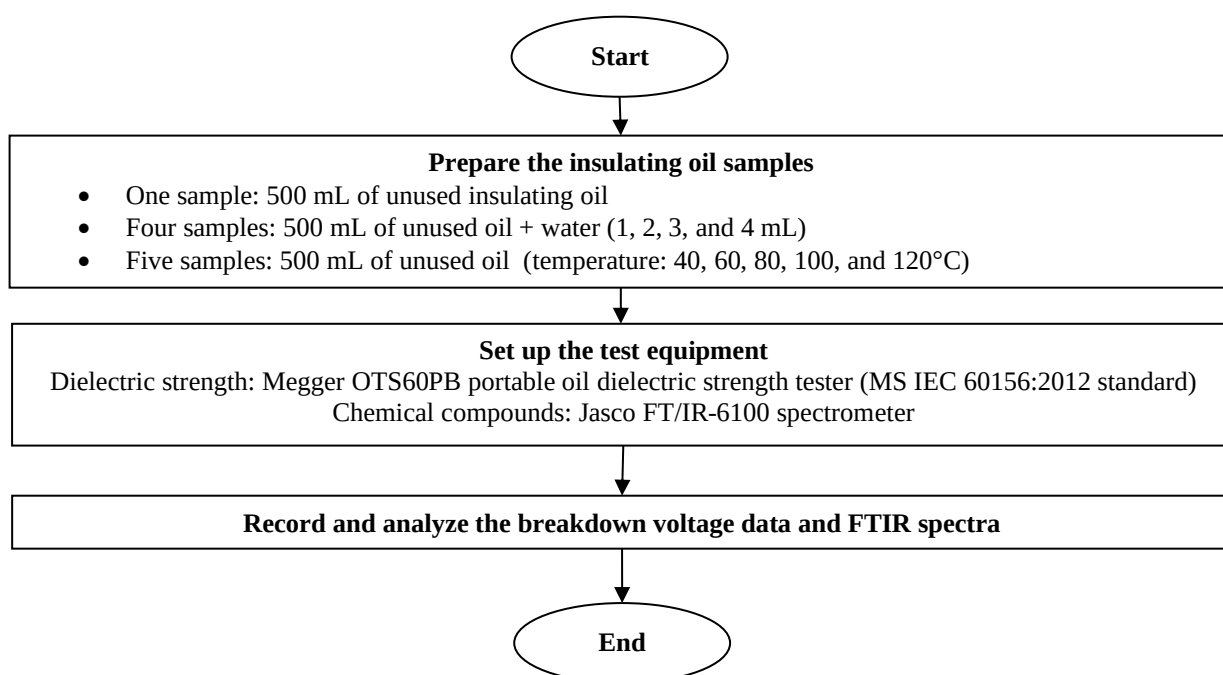


Figure 2: Flow Chart of the Research Methodology

Results and Discussion

Effect of moisture on the breakdown voltage of the insulating oil samples

Table 1 shows the results of the breakdown voltage tests of the fresh insulating oil samples with different moisture content. The volume of each sample was 500 mL. Each sample was stirred thoroughly using a hot plate magnetic stirrer upon the addition of water and then poured into the test cell of the portable oil dielectric strength tester. For each experiment, six readings were taken and the average breakdown voltage was determined, following the procedure outlined in the MS IEC 60156:2012 standard. It can be seen from Table 1 that the average breakdown voltage was 60 kV for the insulating oil with a moisture content of 0 mL. When 1 mL of water was added, the average breakdown voltage decreased to 18 kV, which does not fulfill the minimum breakdown voltage requirement for transformer insulating oils. The average breakdown voltage further decreased to 14 kV after 2 mL of water was added. The

average breakdown voltage further decreased to 10 kV after 3 and 4 mL of water was added into the insulating oil samples.

Table 1: Effect of Moisture Content on the Breakdown Voltage of the Insulating Oil Samples

Moisture content (mL)	Breakdown voltage (kV)						Average
	1	2	3	4	5	6	
0	60	60	60	60	60	60	60
1	20	10	16	13	12	34	18
2	11	11	15	14	11	19	14
3	10	10	10	9	10	11	10
4	9	10	10	12	9	9	10

Breakdown occurs when the insulating oil reaches a state where there is a spark between the electrodes. The properties of the insulating oil on the atomic and molecular scale play a role in influencing the breakdown voltage. It shall be noted that the Megger OTS60PB portable oil dielectric strength tester is fully automatic to record the data or implement the standard. It is expected that the fresh insulating oil will have the highest breakdown voltage value compared with those in which breakdown has occurred because the fresh insulating oil is free from contaminants. Based on the results, the addition of 1 mL of water (≈ 2000 ppm of water) reduced the breakdown voltage of the insulating oil with a maximum difference of 42 kV. This is expected because the presence of moisture delivers charge carriers, which will reduce the dielectric strength of the insulating oil. The addition of 2–4 mL of water into the insulating oil reduces the breakdown voltage to 10–14 kV. The results conform with the theory of breakdown in insulating oils (bubble theory), where the breakdown voltage decreases with an increase in the moisture content of the insulating oil [18].

Effect of moisture content on the chemical compounds of the insulating oil samples

The characteristics of chemical compounds reflect the behavior of the broad classes of hydrocarbons. The hydrocarbon compounds of insulating oils can be classified into three classes: (1) naphthenes, (2) aromatic hydrocarbons, and (3) alkanes. The chemical formula of transformer insulating oils is $C_{12}H_{10-x}Cl_x$, where x is polychlorinated biphenyl, with a value of 1–10. Figure 3 shows the FTIR spectra of the insulating oil samples with and without moisture. It is apparent that there are changes in the chemical compounds of the insulating oil owing to the presence of moisture.

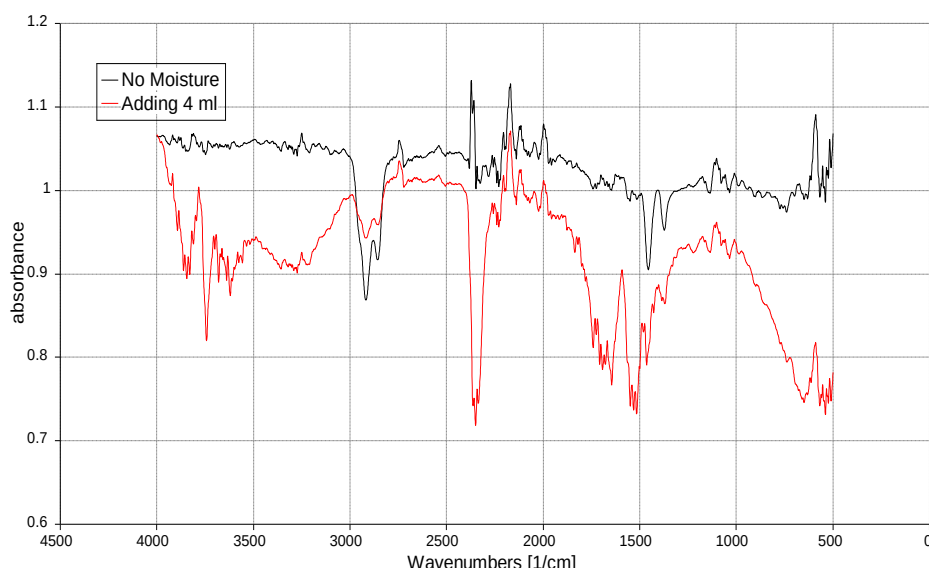


Figure 3: FTIR Spectra of the Insulating Oil Samples With and Without Moisture

It is crucial to monitor the presence of moisture (water) in insulating oils. It can be seen from Figure 3 that the absorption band at 3400 1/cm can be attributed to O–H stretching, which is a direct measure of the moisture concentration. The area of this absorption band is linearly correlated to the moisture concentration. It can be observed from Figure 4 that there is water vapor phase within a wavenumber range of 3600–3400 1/cm. The presence of moisture reduces the dielectric strength of the insulating oil because the moisture will break down the molecular chains of the insulating oil, making the oil more conductive.

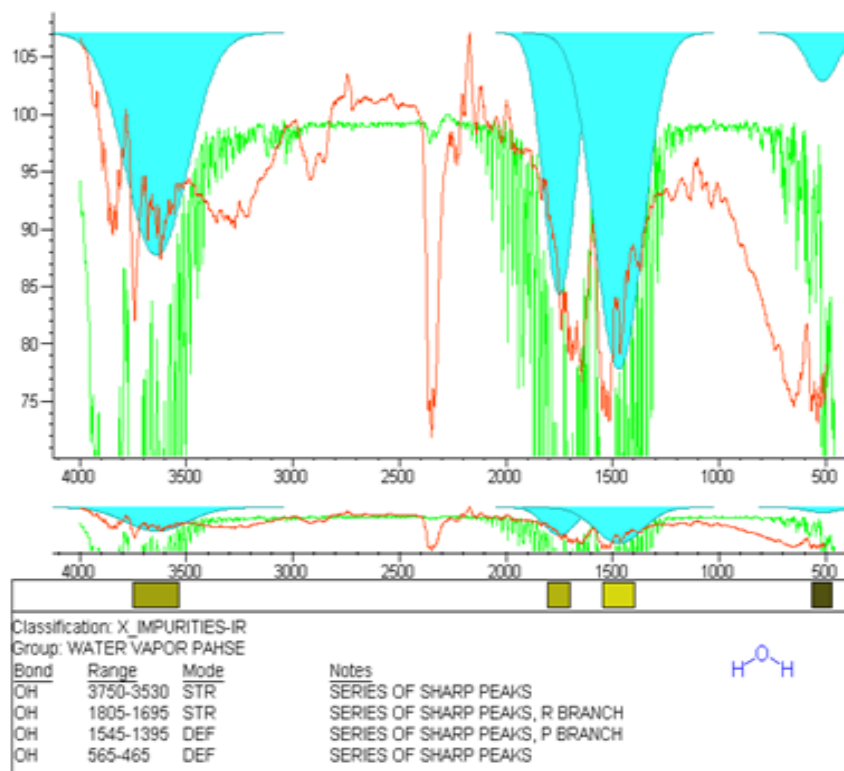


Figure 4: Analysis of Chemical Compounds of the Insulating Oil Sample with 4 mL of Water

Effect of temperature on the breakdown voltage of the insulating oil samples

Table 2 shows the effect of temperature on the breakdown voltage of the insulating oil samples. The average breakdown voltage was 60 kV for the insulating oil at ambient temperature (22°C). The average breakdown voltage decreased to 44 kV when the temperature reached 40°C and further decreased to 41 kV when the temperature was increased to 60°C. In contrast, the average breakdown voltage increased to 55, 60, and 59 kV when the temperature was increased to 80, 100, and 120°C, respectively.

Table 2: Effect of Temperature on the Breakdown Voltage of the Insulating Oil Samples

Temperature (°C)	Breakdown voltage (kV)						Average
	1	2	3	4	5	6	
22 (ambient temperature)	60	60	60	60	60	60	60
40	43	50	54	39	36	43	44
60	51	42	43	40	34	34	41
80	60	53	53	51	55	58	55
100	59	60	60	60	60	60	60
120	57	59	60	60	60	60	59

It is indeed expected that the fresh insulating oil sample will have the highest breakdown voltage compared with those in which breakdown has occurred because the sample is free from contaminants. When the temperature was increased to 40°C, the average breakdown voltage decreased to 44 kV. The average breakdown voltage slightly decreased to 41 kV when the temperature was increased to 60°C. However, the result is still considered acceptable. According to the MS IEC 60156:2012 standard, the minimum passing breakdown voltage is 30 kV. The results can be attributed to the presence of moisture in the test cell. In this research, the insulating oil was first heated in the beaker at a preset temperature and then poured into the test cell for the breakdown voltage tests. When the insulating oil sample was heated at a temperature of 80°C, the water began to vaporize and the breakdown voltage increased to 55 kV. The breakdown voltage increased to 60 kV when the insulating oil was heated at 100°C. The results conform well with the expectation that the breakdown voltage will increase with an increase in temperature.

Effect of temperature on the chemical compounds of the insulating oil samples

Figure 5 shows the FTIR spectra (absorbance versus the wavenumber) of the insulating oil samples at different temperatures. The black, red, green, blue, and light blue lines represent the FTIR spectra of the insulating oil samples preheated at 40, 80, 120, 160, and 180°C, respectively. The FTIR spectra within a wavenumber range of 3000–2800 1/cm were analyzed and it can be seen that there were significant differences in the carbon region. The characteristics of the chemical substances reflect the behavior of the broad classes of hydrocarbons. In general, the hydrocarbon compounds of insulating oils can be classified as naphthenes, aromatic hydrocarbons, and alkanes. The chemical formula of transformer insulating oils is $C_{12}H_{10-x}Cl_x$, where x is polychlorinated biphenyl, with a value of 1–10. The results indicate that temperature indeed plays a significant role in influencing the molecular bonds of the insulating oil. Figure 6 shows the FTIR spectra of the insulating oils at different temperatures (22 and 180°C). There was a strong absorption peak within a wavenumber range of 3000–2850 1/cm, which can be ascribed to C–H stretching. In general, a higher temperature can alter the molecular structure of the insulating oil. Figure 7 shows that the aforementioned

region is indicative of alkanes and aliphatic compound. The high transmittance in this region indicates a significant reduction in the C–H content. The breakdown of the C–H bond results in the appearance of combustible gases, particularly ethylene (C_2H_4) and acetylene (C_2H_2). This region is full of carbon compounds, which can lead to insulation failure.

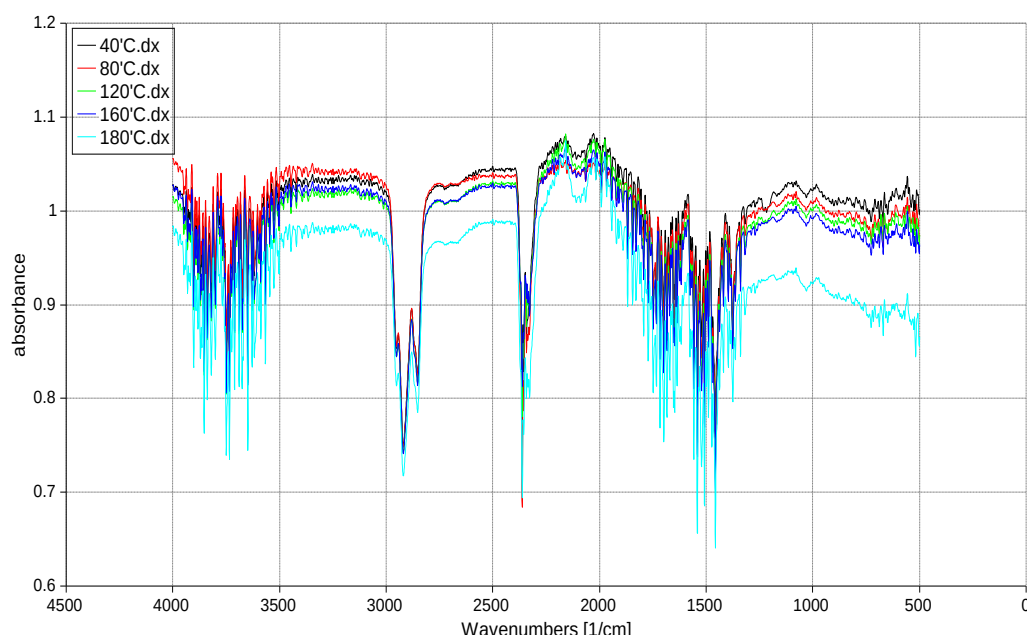


Figure 5: FTIR Spectra of the Insulating Oil Samples Preheated at Different Temperatures

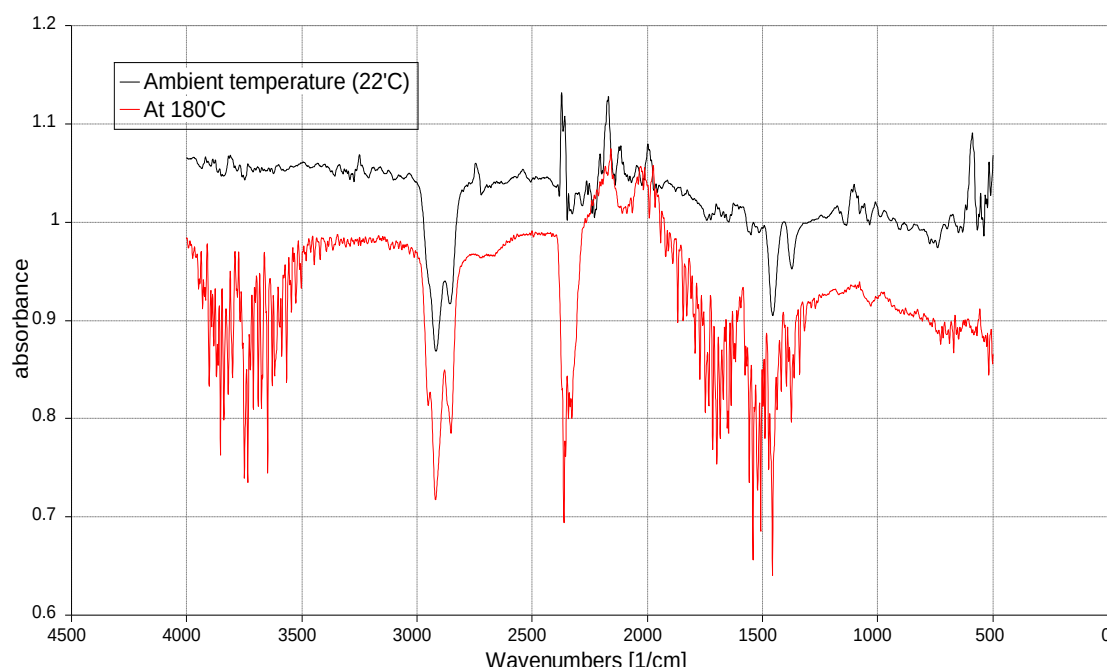


Figure 6: FTIR Spectra of the Insulating Oil Samples at 22 and 180°C

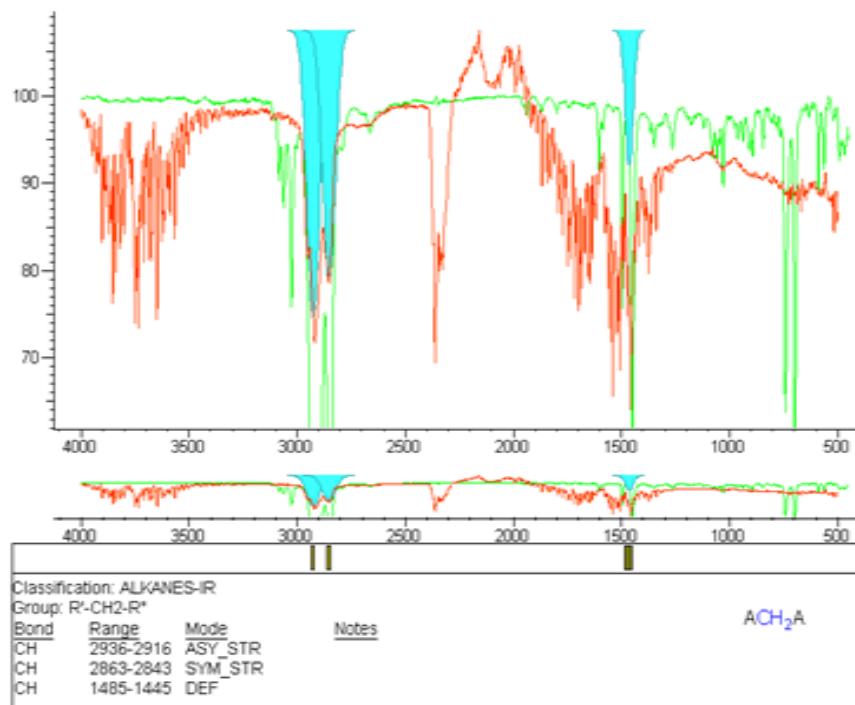


Figure 7: FTIR Spectra of the Insulating Oil Samples at 180°C

Conclusions

In this research, the electrical and chemical properties of mineral insulating oils with different moisture content and preheated at different temperatures were investigated by experiments. Based on the results, it can be concluded that the moisture content and temperature have a significant effect on the breakdown voltage and chemical compounds of the insulating oil. The FTIR spectra revealed that the moisture content of the insulating oil affects the absorbance and wavenumber at which the peak of the absorption band occurs, indicating that there are changes in the chemical compounds of the insulating oil. This is indeed expected because the presence of moisture breaks down the molecular bonds in the insulating oil. Likewise, temperature affects the FTIR spectra because as the oil is heated, the molecules absorb the energy and become much more active, as indicated by the fluctuations in the waveforms.

Acknowledgement

The authors gratefully acknowledge the financial support provided by the Universiti Teknikal Malaysia Melaka (UTeM) under the following grant: UTeM (PJP/2019/FKE(5A)/S01670).

References

- [1] CIGRE Working Group A2.37, "Transformer Reliability Survey," pp. 1–122, 2015.
- [2] Y. Z. Yang Ghazali, M. A. Talib, and A. Maria Soosai, "TNB Approach on Managing Asset Retirement for Distribution Transformers," in *23rd International Conference on Electricity Distribution*, 2015, no. June, pp. 1–5.
- [3] I. S. Chairul, S. A. Ghani, M. S. A. Khair, Y. H. M. Thayoob, and Y. Z. Y. Ghazali, "Kraft Paper Insulation's Life Assessment and Effects of Oxygen and Moisture to Paper Insulation's Deterioration Rate," in *IEEE International Conference on Power and Energy (PECon)*, 2012, pp. 728–731.

- [4] C. Krause, "Power Transformer Insulation - History, Technology and Design," *IEEE Transactions on Dielectrics and Electrical Insulation*, vol. 19, no. 6, pp. 1941–1947, 2012.
- [5] Nynas, "Base oil handbook," 2001.
- [6] I. Fofana, "50 Years in the Development of Insulating Liquids," *IEEE Electrical Insulation Magazine*, vol. 29, no. 5, pp. 13–25, Sep. 2013.
- [7] S. A. Ghani, N. A. Muhamad, I. S. Chairul, and N. Jamri, "A Study of Moisture Effects on the Breakdown Voltage and Spectral Characteristics of Mineral and Palm Oil-Based Insulation Oils," *ARPN Journal of Engineering and Applied Sciences*, vol. 11, no. 8, pp. 5012–5020, 2016.
- [8] J. Wada, G. Ueta, S. Okabe, and T. Amimoto, "Method to evaluate the degradation condition of transformer insulating oil - establishment of the evaluation method and application to field transformer oil," *IEEE Transactions on Dielectrics and Electrical Insulation*, vol. 22, no. 2, pp. 1266–1274, Apr. 2015.
- [9] "BS EN 60296:2012: Fluids for electrotechnical applications — Unused mineral insulating oils for transformers and switchgear," pp. 1–28, 2012.
- [10] ASTM, *D3487-09 Standard Specification for Mineral Insulating Oil Used in Electrical Apparatus*. 2009.
- [11] Y. Hadjadj, I. Fofana, J. Sabau, I. Canada, and P. Nw, "Assessing Insulating Oil Degradation by Means of Turbidity and UV / Vis Spectrophotometry Measurements," *IEEE Transactions on Dielectrics and Electrical Insulation*, vol. 22, no. 5, pp. 2653–2660, 2015.
- [12] M. Kohtoh, G. Ueta, S. Okabe, and T. Amimoto, "Transformer insulating oil characteristic changes observed using accelerated degradation in consideration of field transformer conditions," *IEEE Transactions on Dielectrics and Electrical Insulation*, vol. 17, no. 3, pp. 808–818, Jun. 2010.
- [13] F. Vahidi, S. Tenbohlen, M. Rösner, C. Perrier, and T. Stirl, "The influence of moisture during the electrical conductivity measurement on the high density impregnated pressboard," in *19th International Symposium on High Voltage Engineering, Pilsen, Czech Republic*, 2015, pp. 1–5.
- [14] I. Khelfane, A. Debche, A. Nacer, A. Beldjilali, T. Toudja, and H. Moulai, "Moisture and electrical discharges effect on naphthenic mineral oil properties," *IET Science, Measurement & Technology*, vol. 8, no. 6, pp. 588–594, 2014.
- [15] A. A. Suleiman, N. A. Muhamad, N. Bashir, N. S. Murad, Y. Z. Arief, and B. T. Phung, "Effect of moisture on breakdown voltage and structure of palm based insulation oils," *IEEE Transactions on Dielectrics and Electrical Insulation*, vol. 21, no. 5, pp. 2119–2126, 2014.
- [16] Y. Hadjadj, I. Fofana, and J. Jalbert, "Insulating oil decaying assessment by FTIR and UV-Vis spectrophotometry measurements," in *2013 Annual Report Conference on Electrical Insulation and Dielectric Phenomena*, 2013, pp. 1310–1313.
- [17] A. Munajad, C. Subroto, and Suwarno, "Study on the effects of thermal aging on insulating paper for high voltage transformer composite with natural ester from palm oil using fourier transform infrared spectroscopy (ftir) and energy dispersive x-ray spectroscopy (EDS)," *Energies*, vol. 10, no. 11, 2017.
- [18] R. Bartnikas, *Engineering Dielectrics: Volume III Electrical Insulating Liquids*. 100 Barr Harbor Drive, PO Box C700, West Conshohocken, PA 19428-2959: ASTM International, 1994.

A REVIEW ON DEVELOPMENT OF LIQUID COOLING SYSTEM FOR CENTRAL PROCESSING UNIT (CPU)

Muhammad Arif Bin Harun¹
Nor Azwadi Che Sidik²

¹Malaysia – Japan International Institute of Technology, Universiti Teknologi Malaysia, Jalan Sultan Yahya Petra, 54100 Kuala Lumpur, Malaysia

²Malaysia – Japan International Institute of Technology, Universiti Teknologi Malaysia, Jalan Sultan Yahya Petra, 54100 Kuala Lumpur, Malaysia

Abstract: *Electronic devices are becoming more efficient while getting smaller size and compact design thus increase heat generation significantly. High heat generation from high technology electronic devices are needed to be cool down or control it temperature to prevent overheating problems. Due to the high cooling performance of liquid cooling, electronic cooling system are shifting from air cooling system to liquid cooling system. In the past few decades, numerous methods proposed by researchers for central process unit (CPU) cooling using liquid system either active cooling or passive cooling system. Other than physical configuration such as heat sink design, different configurations of working fluids are widely been studied by most of the researchers. Different working fluids has different heat transfer performance. Furthermore, recent study has come out more interesting finding using nanofluid which can enhance heat transfer performance of liquid cooling. Nanofluid is a working fluid which has nanoparticles disperse in base fluid. In this paper, a comprehensive literature on the type of working fluid used in the respective system and methods of liquid cooling system for CPU including its cooling performance. Moreover, factors that affect the cooling performance also have been discussed. Finally, the conclusions and important summaries were presented according to the data collected.*

Keywords: *Liquid cooling, Central Processing Unit, Nanofluid.*

Introduction

Recent developments in electronic devices have heightened the need for higher efficient cooling system to counter the heat dissipated from the electronic device during operation. Nowadays, to remove heat from chip with surface area of few square centimetres is a big challenge that are need to be study. Electronic devices are becoming more efficient while getting smaller size and compact design. High heat dissipated from high technology electronic devices are needed to be cool down or control it temperature. Without proper method of cooling, abundant heat produces could damage the devices and disturb the whole system. Furthermore, high heat produce will increase the power needed by cooling devices such as fan for cooling process. Table 1 shows the heat thermal design power (TDP) from recent production of Central Processing Unit (CPU). This data was taken from intel website [1]. TDP is the maximum amount of power used by the CPU in watts. Therefore, CPU heat generation can be approximately estimating the same as the TDP value. Electronic component such as CPU should operate under 85°C to work properly and provide good reliability. Usually, most computers have thermal sensors in safety system to prevent overheating. As the temperature approaching temperature limit, the computer will slow down the core and lower the voltage. Proper cooling method could decrease the max temperature and enhance the CPU performance. Most efficient cooling method are high heat transfer, less consume spaces and low noise during operation.

Heat generation of CPU follows Moore's Law where number of transistors increase doubling-up every 18 months so does heat generation increase. Approximately, the total heat dissipation resulting from a high-end CPU is 110–140 W and could achieve 100°C which can damage the CPU. Since it was reported in 1970, transistor count on integrated circuit chips are only 1000 units. Surprisingly, in 2018, the transistor count on integrated circuit chips are 50 billion units [2]. Therefore, this abundant heat generation from the transistor units need highly efficient cooling system such as liquid cooling system.

Conventional air-cooling system are slowly approaching cooling limit. Although, air cooling system still capable for high heat transfer application but it involves with many drawbacks. For this issue, more fans are need to overcome high heat generation which lead high power need and occupy more spaces. Contrary, liquid cooling has better heat conductivity and heat capacity which can increase the cooling efficiency. Recently, CPU cooling system shifted from air cooling system to liquid cooling system due to their capability in heat transfer enhancement for desktop systems and workstations.

There are many researches on liquid cooling CPU using various method of cooling. Most of the study use air cooling as comparison for their proposed methods and working fluid. Many methods are suggested to increase heat transfer such as liquid jet impingement, thermosyphon, heat sink and direct liquid cooling using liquid block. Moreover, different configuration of heat sinks also had been study by many researchers. Other than liquid cooling methods, many types of working fluid had been study for liquid cooling CPU. In recent years, there has been an increasing interest on nanofluid as heat transfer working fluid. Nanofluid is a working fluid which has nanoparticles disperse in base fluid. Nanofluid can significantly enhance heat transfer performance due to the nanoparticle suspension on the base fluid. Various parameters of nanofluid had been used for heat transfer enhancement in liquid cooling CPU.

Due to many types of liquid cooling method and working fluids had been proposed by many researchers, this paper will provide comprehensive review specifically about liquid cooling system in CPU. Furthermore, recent study about water cooling in CPU will be discussed. This paper will provide information about the several methods of liquid cooling system for CPU in recent applications. To date, only a limited number of reviews on methods and working fluids have been identified. It is hoped that this research will contribute to a deeper understanding of liquid cooling system for CPU and offers some important insights for steps to increase efficiency of liquid system.

Table 8 Heat dissipated from CPU [1]

Processors	Thermal design power (TDP)
Intel® CORE™ i7-8550U PROCESSOR (8M Cache, 4.00 GHz)	25W
Intel® CORE™ i7-8750H PROCESSOR (9M Cache, 4.10 GHz)	45W
Intel® CORE™ i7-9700K PROCESSOR (12M Cache, 4.90 GHz)	95W
Intel® Core™ i7-8086K Processor (12M Cache, 5.00GHz)	95W

Working Fluid

The development of working fluid is actively study to increase the thermal performance of cooling system in electronic device. Air- and water-cooling system are favourably used today for CPU cooling. However, the heat dissipation capacity of conventional air-cooled heatsinks

is limited because of non-uniform temperature distribution in the base of heatsinks [3]. The most common working fluid used for liquid cooling is deionized water. It is cheap and better heat capacity than air cooling system. Moreover, ethylene glycol is one of the base fluids can be used. Ethylene glycol has structure similar to water but has lower freezing point and higher boiling point. Azmi, W.H., et al. [4] had done the review about heat transfer augmentation of ethylene glycol. The properties of water, ethylene glycol and air are present in Table 2. The combination of two base fluid is called binary fluid. During late 90s, application of nanoparticle provides effective ways to increase heat transfer characteristic of fluid [5]. Recent based fluid used are deionized water and ethylene glycol [6]. Further development of working fluid has come out with the nanoparticle in based fluid which is called nanofluid. Further development of enhance heat transfer working fluid is important to decrease high portion of energy for cooling and could increase the efficiency of heat removal [7]. Different nanoparticles have been developed and had been applied for many applications in heat transfer.

Table 9 Thermal properties of based fluids [8]

	Specific heat kJ/kgK	Thermal conductivity W/mK	Density kg/m ³	Boiling temperature °C	Freezing Temperature °C
Water	4.187	0.611	1000	99.974	0
Ethylene Glycol	0.769	0.258	1113.2	111.1	-12.9
Air	1.006	26.62	-	-	-

Recent application of nanofluid on electronics cooling

Replacing conventional working fluid with nanofluid is one of the beneficial developments. Nanofluid with high thermal conductivity will increase the cooling efficiency of cooling devices. The nanoparticle exist in fluid will decrease the thermal resistance for heat transfer. Table 3 shows the application of nanofluid or hybrid nanofluid in recent electronic devices. The thermal performance is compared with based fluid. From the literature review, nanofluids parameters that have effect on cooling device thermal performance are type of nanofluids, volume concentration, flow rate and temperature.

Table 3 A summary of application of nanofluid or hybrid nanofluid in liquid cooling system for CPU

Authors	Nanofluids	Findings
A.A.A.A. Al-Rashed, et al. [9]	Carboxymethyl Cellulose/CuO-water	With increase in Reynolds number and nanoparticle concentration to more than 1.5%, the thermal resistance reduces.
N. Zhao et al. [10]	TiO ₂ -water	The most appropriate nanoparticle mass fraction (0.3%) and groove depth (2 mm) for the lowest CPU temperature.
N. Zhao et al. [11]	TiO ₂ -water	Nanofluids with mass fraction of 0.3% and 0.4% show the best cooling performance for rectangular grooves structure and cylindrical bulges structure respectively. More than the critical mass fraction show decrease on Nu number
H. A. Hassan et al. [12]	Al ₂ O ₃ -water, CuO-water, SiO ₂ -water, ZnO-water	The SiO ₂ nanofluid has the highest Nusselt number value, followed by Al ₂ O ₃ , ZnO and CuO. The Nusselt number is improved with the increase of nanoparticle concentration. The Nusselt number increased with the decrease of

P. Gunnasegran et al. [13]	Fe ₂ NiO ₄ - water		nanoparticles diameter. The local Nusselt number considerably increased with increasing Reynolds number Average decrease of 5.75°C (14%) was achieved in core temperatures of desktop PC CPU charged with Fe ₂ NiO ₄ -H ₂ O
M.M.Sarafraz et al. [14]	Liquid gallium CuO-water		Results demonstrated that both coolants present the higher thermal performance in comparison with water. Gallium showed a superior thermal performance compared to nanofluid.
M. Bahiraei et al. [15]	Graphene/Ag-water		Nanofluid has much better cooling than pure water, reduction of surface temperature due to adding the nanoparticles is more prominent at lower Reynolds numbers, and applying the nanofluid leads to more uniform distribution of temperature.
B. Sun et al. [16]	Cu-water Al ₂ O ₃ -water		The convective heat transfer coefficient of nanofluids with a mass fraction of 0.1–0.4% was significantly higher than deionized water, and the convective heat transfer coefficient of Cu-water nanofluids was about 1.1–2 times the heat transfer coefficient of the DI water.
M. H. Al-Rasheed et al [17]	CuO-water		The results show 7.7% heat transfer improvement in thermal conductance is observed in case of nanofluids in comparison to water.
P. Gunnasegaran [18]	SiO ₂ -water		The optimized nanoparticle mass concentration and heat input are 0.48% and 59.97 W, respectively, the minimum R _{th} being 2.66 (C/W)
A. Turgut et al. [19]	Al ₂ O ₃ -water		With 1% volume concentration, nanofluid decreases the maximum temperature of the system, almost 2.7°C compared to water
M. Nazari et al. [20]	Al ₂ O ₃ -water CNT-water		An increase of 6% is also reported by using 0.5% volume fraction of Alumina nanofluid. The best heat transfer enhancement (about 13%) is related to CNT nanofluids with the volume fraction of 0.25% for the flow rate of 21 mL/s.
S. A. Jajja et al. [21]	MWCNT-water		The lowest value of heat sinks base temperature recorded was 49.7°C at a heater power of 255W by using a heat sink of 0.2mm fin spacing and MWCNT nanofluid as a coolant.
T. Yousefi et al. [22]	Al ₂ O ₃ -water		Introduction of 0.5 wt% Al ₂ O ₃ nanoparticles to the water coolant of heat pipe has led to a decrease in thermal resistance. It is shown that at 10 W, the presence of nanofluid has reduced the thermal resistance by 15%, while at 25 W, the thermal resistance has dropped by 22%.
N. Putra et al. [23]	Al ₂ O ₃ -water		Application of Al ₂ O ₃ nanofluid as working fluid would enhance thermal performance of vapor chamber by 9% compare with based fluid.

Table 4 Applications of nanofluid or hybrid nanofluid with heat transfer coefficient

Cooling device	Nanofluid	Vol. concentration (%)	Flow rate/Reynold's number	Heat transfer coefficient (maximum enhancement)	Ref
Microchannel Heat Sink	Carboxymethyl Cellulose/CuO-water	0-3	100 - 700	386.01W/m ² K	[9]
Heat Sink	Gallium CuO-water	0.1–0.3	0.35 and 1 gr/s	Gallium = 600W/m ² K CuO = 400W/m ² K	[14]
Heat Sink	Cu-water Al ₂ O ₃ -water	0.1–0.4	400–2000	Cu-water = 11100W/m ² K Al ₂ O ₃ -water = 8400W/m ² K	[16]
Heat Sink	Al ₂ O ₃ -water CNT-water	0.1- 0.25	5-20 ml/s	Al ₂ O ₃ -water = 340W/m ² K CNT - water = 365W/m ² K	[20]
Heat Sink	SiO ₂ -water, TiO ₂ -water, Al ₂ O ₃ -water	SiO ₂ : 0.5, 1.0, 1.5 TiO ₂ : 0.1, 0.25, 0.5 Al ₂ O ₃ : 0.5, 0.75, 1.0	0.5, 0.75 and 1.0 l/m	SiO ₂ -water = 1000W/m ² K TiO ₂ -water = 920W/m ² K Al ₂ O ₃ -water = 1250W/m ² K	[24]
Heat Sink	CuO-water	0.1 and 0.2	0.55-2.44l/m	1900 W/m ² K	[25]

Table 4 shows list of nanofluids that had been used in liquid cooling system for CPU with heat transfer coefficient analysis. Type of nanofluid had been used is CuO [9, 12, 14, 17], Al₂O₃ [12, 16] [23, 26, 27], TiO₂ [10, 24, 28], SiO₂ [18, 29], ZnO [12], graphene [15, 30], CNT [20] and MWCNT [21]. It is difficult to compare the best nanofluids as there are various concentration and heat generation is used. There is also other study on the maldistribution of nanofluid at heat sink with different design [30, 31]. Different type of nanofluids has different rate of heat transfer enhancement. One of the properties for heat transfer enhancement by nanofluids is thermal conductivity. Table 5 shows several thermal conductivities of nanoparticles. Supposedly, higher thermal conductivity of nanofluid has better thermal enhancement. B. Sun, H. Li [16] study confirm that higher thermal conductivity of Cu nanofluid has better thermal performance compare to Al₂O₃. But some study has contrary results [14, 26]. Based on the literature review on the study of thermal conductivity of nanofluid, thermophysical properties of nanofluid are strongly depend on concentration of the concentration on nanoparticle in base fluid and temperature. Therefore, it's hard to consider thermal conductivity specifically for heat transfer enhancement in liquid cooling

Table 5 Thermal conductivity of nanofluids

Nanoparticles	Thermal Conductivity(W/mK)	Ref
Water	0.6	
Al ₂ O ₃	36	[26]
SiO ₂	1.2	
ZnO	13	
CuO	440	[14]
Gallium	29.4	
Cu	401	[16]

Other than thermal conductivity of nanofluids, Brownian motion also being reported provide thermal enhancement using nanofluid. P. Selvakumar, S. Suresh [25] and C. Qi et al [32] strongly agree that the effects of thermal conductivity and Brownian motion may play a major role in the heat transfer enhancement when the nanoparticle mass fraction is less than the critical value. Other than that, reported that Brownian motion could only happen for heat transfer applications of flowing fluids [24, 27]. At stationary condition, Brownian motion could be neglected [33]. Unfortunately, there is no details analysis for Brownian motion.

Thermal enhancement strongly depends on concentration of nanofluids. From the literature reviews, it is confirmed that higher concentration of nanofluid has better heat transfer performance. Unfortunately, most of the studies just use small percentage of nanofluid approximately <5% concentration in liquid system [12, 13, 17]. One of the considerations of the selected concentration is the stability of nanofluid. Although higher concentration has better heat transfer enhancement, but higher concentration will decrease the stability of nanofluid. This is supported by Mehrali et al [34] where the stability of prepared graphene nanoplatelets drops as the concentration of nanofluid increase. Furthermore, higher concentration of nanofluid need more complicated and longer period of preparation which is inconvenient. Other than that, channel size of heat sink or liquid block also one of the factors of choosing nanofluid's concentration. Most of the channel heat sink are in micro or milli size. Higher concentration of nanofluid could cause pressure drop [14] or disturb the cooling system due to clogging at the channel of heat sink. Assume that can be made from the literature reviews are the concentration used in respective study are at the limit of heat sink's size. This is because the maximum of concentration did not state in each study. Only optimization study had shown the optimum concentration used in liquid cooling system [18].

Liquid block/heat sink

Liquid cooling has been widely used and have been commercialise. Water block with cold plate will have physical contact with the CPU and heat transfer will happen between CPU and working fluid [35]. Liquid cooling system used less space and noise-free operation because less fans are used. Parallel channel is used for conventional liquid block. There are many researchers study using this type of heat sink design. Summary on research of liquid block and heat sink are shown in Table 6.

Prem Gunnasegaran [13] had study loop heat pipe using Fe₂NiO₄/H₂O nanofluid. The experiment is also design with a commercial liquid cooling kit similar as used in real desktop PC CPU cooling. Few thermocouples are been used to collect data of temperature around the loop heat pipe. Copper block dimension is 50 x 50 x 4 mm. Heat input range are between 20W to 60W by adjusting the variable transformer. They found out that the optimum concentration for this nanofluid is 1% only as the thermal resistance is the lowest at this mass concentration.

Other than that, Atik, Ö. and H. Ertürk [36] had investigate the liquid cooling with two experimental setups. First one is with flow control and another one is with commercial liquid cooling system. A commercial liquid cooling system (Zalman LQ310) is used for second setup. Working fluid used for this study is hexagonal boron nitride nanofluids. Volume fraction of nanoparticle used are 0.1%, 0.5%, 1% and 2% and prepared by two-step method. The results show maximum enhancement using hBN–water nanofluid with 2% volume fraction for all setups.

Furthermore, Khaleduzzaman, S.S., et al. [37] had carried out experimental analysis using XSPC RASA to study the energy and friction factor for titanium dioxide nanofluid in a conventional water block heat sink. The size of the copper base water block is 94 x 94 x 20 mm. The special about this water block is it consists of 55 thin channels and the cross-sectional area of the channel is 33 x 33 mm. Furthermore, the maximum heaters capacity for this system is 80°C. Volume fraction of the nanofluid is 0.1% while various flowrate had been used which are 1.00, 1.25, and 1.50 L/min. The results show that the base temperature was decrease up to 6.40°C at 1.0 L/min flow rate. Thermal resistance was also to be decreased for nanofluid compare to water. Both parameters are found to be decreased as the flow rate increase. The highest improvement of heat transfer coefficient was 20.82% for the nanofluid and the maximum heat transfer effectiveness for the nanofluid was observed at 1 L/min flow rate.

Other commercial liquid cooling that used for investigation of nanofluid is been done by M. Rafati et al [24] with 3D Galaxy II from Gigabyte. Three nanoparticles of SiO₂, Al₂O₃ and TiO₂ were used with 0.5% and 1% volume concentration. As previous research nanofluid and higher flow rate shows better thermal enhancement for the system. Commercial liquid cooling system were already proven as an efficient cooling device. However, the previous study has not yet study using more than 100W heat input which is the current problem. Furthermore, the flow rate is not higher than 2 L/min due to the pressure limit from the piping.

Beside the study using commercial liquid block, some research had been done for the effect of parallel channel design on heat transfer. Kim [38] had experimentally study using two different design heat sinks on heat transfer and pressure drop characteristics of electronics cooling. The different between two heat sinks are the machined depth of the cover was 5.0mm and 8.0 mm. The working fluid used was FC-72. Electric heating from 100 to 300 W was supplied by cartridge heaters. The results show that the cooling capacity with FC-72 flow boiling was up to 330 % higher than that with liquid water. The pressure drops for both heat sinks is similar with each other. The effect of the machined depth configuration does not give significant effect on pressure drop. Most of the pressure drop caused by the change of flow paths such as sudden expansion and contraction, change of flow direction, and flow distribution and merging, rather than by the flows in small rectangular channels between the fins. Pressure drop with FC-72 was greater than liquid water. Therefore, in this research proved that the significant effect of pressure drop is caused by working fluid. Good stability of nanofluid is very important for the efficiency of cooling performance.

Other than that, comprehensive study about the effect of fin spacing on minichannel heat sinks for microprocessor cooling had been done by [39]. Five different heat sinks with fin spacings of 0.2 mm, 0.5 mm, 1.0 mm, and 1.5 mm along with a flat plate heat sink and 0.5 LPM, 0.75 LPM and 1.0 LPM flow rate were investigated. They found that decreasing the fin spacing of the heat sinks increased overall heat transfer coefficient while thermal resistance decreased. Significant finding, they found that 0.2 mm fin spacing at a flow rate of 1.0 LPM shows about 9% lower than the best reported base temperature of 40 °C using a nanofluid with 3D Galaxy II from Gigabyte with thermal design power of 125 W [24]. Other than that, for a flat plate heat sink, the maximum thermal resistance was 0.216 K/W that was reduced to as little as 0.03 K/W by using a heat sink of 0.2 mm fin spacing.

Next, the configuration on inlet and outlet arrangement had been study. In conventional liquid block inlet and outlet arrangement, working fluid enters to the liquid block in vertically upward and getting out vertically downward from the outlet. Therefore, Ming-Chang [40] had initially study effect of the inlet location on the performance of parallel-channel cold-plate. A total of five inlet configurations have been set up which are I-, Z-, J-, L-, and T-arrangement. Conventional liquid block is similar to I-arrangement. The results show that the I- and T-arrangement give the best heat transfer performance due to their impingement configuration. Maldistribution from I-arrangement could be solved by increasing the number of channels on cold-plate. On the other hand, one of the researchers S. Kumar, P.K. Singh [41] come out with novel inlet and outlet arrangement with different flow inlet angle. This is the further work of the Z-arrangement. The inlet angle used are $\theta = 90^\circ$, $\theta = 105^\circ$, and $\theta = 120^\circ$. Configuration of inlet and outlet arrangement are to ensure better flow distribution and heat transfer performance. In this design, working fluid enter the liquid block horizontally with different flow inlet angle. The results show that proposed inlet/outlet arrangement with flow inlet angle $\theta = 105^\circ$ has more uniform flow distribution in heat sink. Conventional arrangement has highest value of maldistribution. Furthermore, temperature drop of 2°C has been noticed with the proposed flow arrangement as compared to conventional flow arrangement for same volumetric flow rate. The inlet and outlet arrangement do not significantly affect the cooling performance. Therefore, inlet and outlet arrangement can be used for the arrangement of cooling system in the desktop.

Heat sink is a heat transfer device that transfer heat generated by electronic devices to fluid medium. Usually working fluid used are air and liquid. Heat sink are design to maximise surface area contact between the heat source and the working fluid. The performance of liquid cooling system is affected by the design of the heat sink and type of working fluid used. Uniform cooling distribution is the most important motivation of the further development of heat sink design in water block. Better uniform cooling distribution will prevent hotspot problem. Hotspot is where heat is concentrated at one spot where heat is failed to be dissipated.

Therefore, one of the research by Zhao and his co-workers [10] had experimentally study on thermo-hydraulic performance of nanofluids in CPU heat sink with different design which are rectangular grooves and cylindrical bugles. The working fluid used are TiO_2 -water nanofluids. The effect of rectangular grooves depth (1mm, 2mm) and cylindrical bugles arrangement which are aligned and staggered using nanofluid are analysed. The mass fraction and Reynold number thermal performance effect are also study. It is found that large depth rectangular groove and staggered arrangement cylindrical bulges are enhance heat transfer performance. Furthermore, mass fraction with 0.3% and 0.4% are the best heat transfer for cooling for both heat sink design.

Similar design also had been study by [42] where three types of heat sink design plate pin in line, strip pin in line and circular pin-fin. Heat flux applied for all models is 900W/m^2 . The inlet velocity varied from 0.05, 0.10, 0.15, 0.20 and 0.25 m/s. They found out trip pin fin design model has higher pressure drop than circular pin fin because the obstruction of design model for strip pin fin resulting in the air flow unable to flow through effectively. As the inlet velocity increase, the thermal enhancement also increases. The result show that plate fin heat sink has better thermal performance. Mehdi [30] had study cylindrical bugles mini pin fin heat sink with eco-friendly graphene nanoplatelets nanofluid. Four designs have been used in this study. Fin height used are 0.2, 0.4, 0.6 and 0.8 mm, and fin diameter used are 0.6, 0.9 and 1.2 mm. As fin height, fin diameter and mass concentration of nanofluid increase, the temperature of bottom surface decreases. Unfortunately, the pumping power increases. Another finding is the temperature distribution becomes more uniform with increasing the concentration, fin

height and fin diameter, which can decrease the possibility of hot spot formation. Furthermore, the fin height has more significant effect on the temperature compared with the fin diameter.

Next, new type of heat sink water block design had been proposed by Bahiraei, M. and S. Heshmatian [15] which are serpentine, parallel and distributor heat sink. At constant Reynold number, serpentine liquid block has better cooling but need higher pumping power. But for constant pumping power, distributor liquid block shows better results cooling. The uniform flow in distribution liquid block has decreased the possibility of formation of hot spots. Therefore, surface cooling is much more uniform. There are many other channel designs had been proposed by researchers, but only research that has been presented in this paper has been used for CPU cooling application.

In recent research, working fluid for cooling system are being research aggressively. Application of nanofluid in cooling system are popular being used by researchers. Mostly the application of nanofluids shows significant amount of heat transfer enhancement to the cooling system except [36]. Furthermore proper selection of nanofluid in particular cooling system are very important [43]. As the mass concentration increase, the thermal enhancement also increases. This prove nanofluid is the best alternative of working fluid to increase the cooling performance. However, there is a limit of nanoparticle concentration for cooling device as higher concentration will increase the pressure drop and disrupt the system. As from the literature review, the maximum concentration researchers had used is up to 3%. Optimum concentration and another parameter need to be analysed properly. One of the method of optimization can refer to [18] where design expert is been used. Optimized mass concentration, heat input and thermal resistance are study in the research. Stability of nanofluid are very important to sustain the efficiency of cooling device. Therefore, synergetic effect from hybrid nanofluid could solve this problem. Unfortunately, there are only few studies had done investigation using hybrid nanofluid.

Table 6 A summary of studies of heat sink/liquid block for CPU cooling

Ref	Heat sink/Liquid block	Working fluid	Findings
[10]	Cylindrical grooves heat sink	TiO ₂ -water	For aligned and staggered arrangement, Nusselt number increases by 31.3% and 30.9% respectively. CPU temperature is reduced by about 3.3 °C (8.2%) for nanofluid compared with deionized water.
[14]	Liquid block	Liquid gallium, CuO-water	Results demonstrated that both coolants present the higher thermal performance in comparison with water. Gallium showed a superior thermal performance compared to nanofluid.
[15]	Serpentine Parallel flow Distributor	Graphene-Ag-water	Novel distributor liquid block has a superior efficacy based on both thermal performance and irreversibility rates. The merit of using the nanofluid in the liquid blocks is greater than pure water
[19]	All in one (AIO) liquid cooler	Al ₂ O ₃ -water	With low volume concentration 1% of Al ₂ O ₃ particles decreases the maximum temperature of the system, almost 2.7°C compared to water.
[30]	Cylindrical pin fin heat sink	Graphene nanoplatelet	The fin height has more significant effect on the temperature compared with the fin diameter which is 21.27% enhancement.
[36]	Zalman LQ310	hBN-water Al ₂ O ₃ -water	Maximum enhancement in reduction of case-to-ambient thermal resistance is obtained by using hBN–water nanofluid with 2% volume fraction as 4.3% for

				commercial liquid cooling system, 7.8% and 11.2% for experimental cooling systems with finned and flat cold plates, respectively
[37]	XSPC Liquid block	RASA	TiO ₂ -water	The heat transfer coefficient of nanofluid was improved by 20.82% compared to water. The interface temperature of the water block was reduced up to 6.40°C by using the nanofluid
[38]	Rectangular heat sink	fin	FC-72	Compared with the water heat transfer coefficient, the improvement of FC-72 was up to 330 % and 180 % in the Type I and II heat sink, respectively. The pressure drop in the Type I and II heat sink showed a similar pressure drop
[39]	Flat channel sink	heat	Water	For a flat plate heat sink, the maximum thermal resistance was 0.216 K/W that was reduced to as little as 0.03 K/W by using a heat sink of 0.2 mm fin spacing.
[40]	Flat channel sink with I-, Z-, J-, L-, and T- inlet arrangement	heat	Water	I- and T-arrangement give the best heat transfer performance due to their impingement configuration
[41]	Mini-channel sink	heat	Water	Inlet/outlet arrangement with flow inlet angle $\theta = 105^\circ$ has more uniform flow distribution in heat sink. Temperature drop of 2 °C has been noticed with the proposed flow arrangement as compared to conventional flow arrangement for same volumetric flow rate.
[42]	Plate pin fin heat sink, Strip pin fin heat sink, Circular pin fin heat sink	heat	Water	The result show that plate fin heat sink has lower temperature reading compare to other design followed by circular pin fin and strip pin fin for all velocity rate.

To study the thermal performance of the working fluid in liquid cooling system, most of the researcher study on the maximum temperature readings, thermal resistance and heat transfer coefficient. For maximum temperature reading, liquid cooling system with nanofluid has lower maximum temperature reading compare to only base fluid. Turgut and Elbasan [19] focused their study on the maximum temperature reading of CPU using Al₂O₃-water nanofluid. They achieve 6.7% decrease maximum temperature compare to based fluid. The maximum temperature of CPU is taken after 100% workload is applied to the CPU in a duration of time. This analysis is the main and basic findings to analyse the heat transfer performance of liquid cooling system for CPU. For more details of heat transfer performance, some researchers study on thermal resistance analysis, heat transfer coefficient and Nusselt number. Thermal resistance analysis study by [13, 17, 18] found out that nanofluid has lower thermal resistance compare to based fluid. Study by [13] and [18] has more accurate value because the total thermal resistance value is taken from all parts of the cooling system compare to other that analyse the thermal resistance at water block or heat sink only.

Heat transfer coefficient is important to study the rate of nanofluid's heat transfer. The amount of heat transfer between the wall (surface of the heat sink) and the working fluid can be written as Equation 1.

$$q = hA\Delta T_b \quad (1)$$

Where q is the produced heat flux from heat source, A is the effective cross-section in contact with the surface of heat sink and ΔT_b is bulk temperature which is the difference between the wall temperature and the fluid temperature. Some researchers [20, 24] used LMTD method to estimate ΔT_b as stated in Equation 2.

$$\Delta T_b = \ln \frac{T_{cpu} - T_{in}}{T_{cpu} - T_{out}} \quad (2)$$

On the other hand, M.M. Sarafriz et al [14] calculate ΔT_b with arithmetic average of inlet and outlet temperatures readings. This is because liquid cooling system involve transient heat transfer coefficient.

Thermoelectric coolers (TECs)

Thermoelectric consist of two categories which are thermoelectric generators (TEGs) and thermoelectric coolers (TECs) [44]. TEGs transform heat energy to electrical energy. Seebeck effect is a phenomenon occur where voltage is generated which a temperature difference between two different electrical conductors or semiconductors. Meanwhile, TECs dissipated heat by using electric through the Peltier effect. The Peltier effect is described as a phenomenon whereby heat is dissipated or absorbed when an electric current flow across a junction between two materials. There are many commercialize thermoelectric Peltier available on the market. Usually TEC is used for cooling electronic device. Figure 2 shows the mechanism of Peltier effect in TEC.

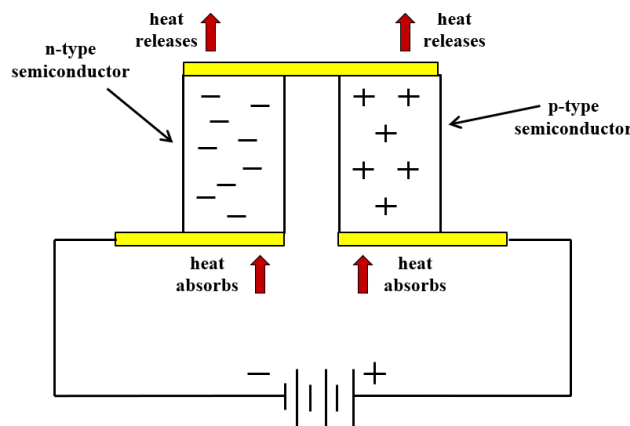


Fig 2. Peltier effect

Among conventional TEC that are usually being used, there are three types of TEC modelling available which are single-stage thermoelectric coolers, multi-stage thermoelectric coolers, and trapezoid and annular thermoelectric coolers. TEC are widely being used for cooling CPU to increase the CPU cooling performance while decreasing the noise of cooling system. The summary of TEC for CPU cooling are shown in Table 7. Application of TEC could reduce the noise to less than 40dB with a power consumption of 130 W to cool the CPU in [45]. As the TEC is a very simple yet high cooling performance device, researchers are interested to combine with different cooling application. This will increase the cooling performance and more heat can be absorb from the electronic device.

S.O. Tan, H. Demirel [45] had carried out study with a heat sink, water cooling system and thermoelectric cooler comparatively to examine the temperature and performance of the CPU and motherboard. The Peltier or thermoelectric cooler (TEC) module was proposed for

cooling the CPU to increase the CPU cooling power and decrease the noise of the cooling system. ATEC1-12706 Peltier module with 290 W power and 44/40.5 cm size was used under 12 V DC voltage to cool the CPU and Northbridge chipset. TEC has the lowest average CPU and motherboard temperature compare to water cooling system and air-cooling using heat sink. The temperature difference of 5°C reveal that the thermoelectric cooler is more effective than the water-cooling system.

A. R. Mohammad Siddique et al [46] carried out an experimental analysis with a liquid based thermoelectric electronic cooling system, using two developed aluminium heat exchangers. An Ethylene-Glycol based fluid was used as the coolant in the cooling system. The results show that large aluminium heat exchanger shows better performance compared to the small one. The commercially found cooling system performed the worst performance. The temperature variation was between 29-30°C, which is nearly 2°C less than the commercial cooling system and showed steady performance.

H.-S. Huang et al [47] investigates the thermal performance of a thermoelectric water-cooling device for electronic equipment. The influences of heat load and the thermoelectric cooler's current on the cooling performance of the thermoelectric device are experimentally and theoretically determined. Besides, this study verifies that the thermal performance of the conventional water-cooling device can be effectively enhanced by integrating it with the thermoelectric cooler when the heat load is below 57W. Higher heat loads decrease the efficiency of the TEC as Peltier effect cannot overcome a high heat load. Low temperature different from TEC cold side and hot side decrease the efficiency of TEC in cooling system. Further cooling system need to be study to cool down heat load more than 57W.

N. Ahammed et al. [48] studied on the performance of thermoelectric cooling of electronic devices with nanofluid in a multiport minichannel heat exchanger. The Bismuth Telluride (BiTe₃) thermoelectric cooler (TEC) with a $\Delta T_{\max}$ of 67 °C is used to extract heat from the electronic devices. Al₂O₃-water with volume concentrations of 0.1% and 0.2% is used as the coolant to remove the heat from the hot side of the TEC. The result showed 40% enhancement in the coefficient of performance (COP) of thermoelectric module for 0.2% of nanoparticle volume concentration. A 9.15% decrement in thermoelectric temperature difference between the hot and cold side has also been observed for nanofluids 0.2% concentration, which enhanced the module cooling capacity. These results show that nanofluid is very suitable to cool down the hot side of TEC. More comprehensive study needs to be done on effect of higher concentration of nanofluid in TEC.

P. Naphon, S. Wiriyasart [49] study on the liquid cooling in the mini-rectangular fin heat sink with and without thermoelectric for CPU. The de-ionized water is used as coolant. The de-ionized water was pumped into the mini-rectangular fin heat sink which installed on the hot side of thermoelectric. Water cooling without thermoelectric gives a larger CPU temperature drop while the energy consumption slightly increases. Thermoelectric has a significant effect on the CPU cooling of PC. However, energy consumption is also increased. Further study needs to be done to manage the energy consumption by TEC.

H.M. Hu et al. [50] had study on water-cooled TEC for CPU with two temperature controls for TEC are proposed and compared to prevent condensation on CPU and save energy. The CPU is directly connected with the cold side of TEC and the hot side of TEC is directly connected with water block. Two kinds of control strategies, including short circuit and open circuit, are investigated under different operating conditions and working environments. Temperature for CPU of short circuit is lower than open circuit. It indicates energy consumption using short circuit is reduced compared with that using open circuit. This study is one of the proposed methods to decrease the drawback of high energy consumption by TEC.

O. Khonsue [51] had carried out experimental investigation on liquid cooling in the micro channel fin heat sink with and without thermoelectric for central processor unit (CPU) of personal computer. The de-ionized water is used as coolant. The experiments are performed at no load and full load conditions within 60 min after steady state. Cooling system with thermoelectric gives the highest efficiency where 77% enhancement on no load and 22.21% enhancement on full load compare to air cooling system.

Table 7 A summary of studies of Thermoelectric cooler (TEC) for CPU cooling

Authors	Working Fluid	Findings
A. R. Mohammad Siddique et al [46]	Ethylene-Glycol	The temperature variation was between 29-30°C, which is nearly 2oC less than the commercial cooling system and showed steady performance.
H.-S. Huang et al [47]	Water	The thermal performance of the conventional water-cooling device can be effectively enhanced by integrating it with the thermoelectric cooler when the heat load is below 57W.
N. Ahammed et al. [48]	Al ₂ O ₃ -water	The result showed 40% enhancement in the coefficient of performance (COP) of thermoelectric module for 0.2% of nanoparticle volume concentration. A 9.15% decrement in thermoelectric temperature difference between the hot and cold side has also been observed for nanofluids 0.2% concentration
P. Naphon, S. Wiriyaart [49]	Water	Thermoelectric has a significant effect on the CPU cooling of PC. However, energy consumption is also increased.
H.M. Hu et al. [50]	Water	Energy consumption using short circuit is reduced compared with that using open circuit.
O. Khonsue [51]	Water	Cooling system with thermoelectric gives the highest efficiency where 77% enhancement on no load and 22.21% enhancement on full load compare to air cooling system.

Heat pipe

Heat pipe work by transferring energy from one point to another using working fluid or coolant. Heat pipe is a device with highly effective thermal conductors due to their high heat transfer for condensation and boiling. Heat pipe is consider as one of the best cooling devices due to their high thermal conductivity, reliability and cost effectiveness [52]. There are many types of heat pipe which are pulsating heat pipe, rotating heat pipe, oscillating heat pipe, wick heat pipe and thermosyphon and many more [53]. Usually heat pipe is divided into three parts which are the evaporator section, adiabatic (transport) section and condenser section. Heat dissipated at the external heat pipe parts will transfer to working fluid in evaporator parts. The heated working fluid will flow through adiabatic section to condenser. Vaporize working fluid will condense in condenser part where latent heat of vaporization is released to the heat sink. The capillary pressure/action is responsible to transport working fluid to the evaporator. Not enough capillary pressure to supply working fluid to evaporator could disrupt the heat pipe system. The summary of application of heat pipe for CPU cooling are shown in Table 8.

Wicked heat pipe is defined as heat pipe with working fluid flow by using capillary wick. Capillary-driven heat pipe, loop heat pipe (LHP) and capillary pumped loop heat pipe (CPLHP). Heat pipe for electronic cooling is usually in mini/micro size. Therefore, this type of heat pipe is very suitable apply in laptop and notebook which has limited space.

G. Zhou et al. [54] study an active air-cooling module based on a 1-mm-thick ultrathin miniature loop heat pipe with a flat evaporator for high-end ultra-slim laptop computers. Under natural convection, the proposed miniature loop heat pipe could efficiently transfer a maximum heat load of 15 W with a junction temperature below 96.6 °C for all test orientations. Under forced air cooling, the miniature loop heat pipe had the lower junction temperatures. A minimum total thermal resistance of 2.17 °C/W was achieved at 30 W.

Z.R. Lin et al. [55] developed and designed a novel miniature flat loop heat pipes (MFLHPs) with rectangular shaped evaporator for laptop cooling. Length of evaporator and connecting pipeline are the main factor that effect the performance of mLHP. Longer the length of evaporator and pipeline, better the heat transfer of LHP. But there is space limitation of mLHP. The mLHP can cope with the higher power dissipation and keep heater temperature stable with the increase of heat loads, which met the laptop cooling requirements. Unfortunately, small scale heat pipe prefers use air as heat transfer medium as liquid cooling system need space.

Heat pipe with liquid cooling system suitable in CPU cooling. Putra et al. [56] proposed a design of heat pipe of CPU cooling using coral biomaterial wick structure. Heat pipe was made from copper pipe and the wick structure was made from tabulate coral with a mean pore diameter of 52.95 µm. This design is compared with wick structure fabricated from sintered Cu-powder with a mean pore diameter of 58.57 µm. This wick structure will transport the working fluid naturally during heat transfer. The used of coral as wick structured can improved the performance of heat transfer while decrease the CPU temperature for 38.6 % at the maximum heat load compared to a conventional copper heat sink. This method also decreased the temperature of the simulator plate by as much as 44.25 °C compared to a heat pipe composed of a sintered Cu-powder wick.

Uddin et al. [57] had study on the heat transfer performance of a miniature heat pipe system (MHPS) used for CPU cooling. The MHPS consists of 6 parallel cylindrical miniature heat pipes (MHPs) which are connected to a copper block at the evaporator section and which are provided with 15 parallel perpendicular copper sheets at the condenser section, used as external cooling fins. Acetone and ethanol are used as working fluids. The results show that acetone has lower maximum and steady state temperature of the processor compare with ethanol. Additional use of a fan results in a much lower processor temperature for both working fluids. But current study had been introduced with nanofluid for working fluid.

T. Yousefi et al. [22] had study on the effects of inclination angle and nanofluids on heat transfer performance of a CPU cooling heat pipe. It is shown that inclination angle of the unit has a significant effect on the cooling process, since it directly influences the operation of the evaporator. It is observed that as the CPU temperature increases, the threshold angle decreases from 60° to 30°. At 10 W, the presence of nanofluid has reduced the thermal resistance by 15%, while at 25 W, the thermal resistance has dropped by 22%. It is shown that nanofluid and the inclination of heat pipe help significantly the heat transfer performance of heat pipe. Only low concentration of nanofluid could be used to prevent clogging at wick part.

E.N. Stephen et al. [58] had study on heat transfer performance of compact loop heat pipe (CLHP) with Al₂O₃-water and Ag-water nanofluid in 0.09% and 0.12% volume concentrations. The heat pipe was kept in vertical orientation. The experimental results showed that the evaporator thermal resistance is reduced by 34.70% and 20.21%, respectively, for 0.12 vol% of Ag, Al₂O₃ nanoparticles when compared with that of the distilled water. For the same volume concentrations of Ag, Al₂O₃ nanoparticles, an enhancement of 34.52%, 23.7%, 39.27% and 30.8%, respectively, observed for the convective heat transfer coefficients at the evaporator and condenser.

Thermosyphon is one of the heat pipes that been used in electronics cooling. The mechanisms of thermosyphon is similar to typical heat pipe as mention above and does not requiring any mechanical devices such as pump [59]. In this system, condenser is placed above the evaporator. The liquid is flow from the condenser to the evaporator by using gravity force which mean this type of heat pipe is wickless. After that, the vapor will flow to the condenser upwards due to lower density and pressure force. Therefore, lesser cost is needed fabricate this type of heat pipe than wick heat pipe. There are also thermosyphon with wick but the major force flow is gravity force.

Stable circulation of working fluid in thermosyphon are very important for better efficiency of cooling. A. Chauhan, S.G. Kandlikar [60] had proposed dual taper thermosiphon to increase the stability of fluid flow in thermosyphon. Tapered section in flow domain helps in pressure recovery. So, stable fluid could be established. The dual taper design has a smaller flow length and results in less expansion. This characteristic helps in establishing a more efficient vapor removal mechanism with stable fluid flow. The 2° dual taper angle performed the best among all taper angles tested and was able to dissipate a maximum of 280 W without reaching critical heat flux. The surface temperature was maintained at 45.5 °C with a heat transfer coefficient of 22.5 kW/m² °C. This type of heat pipe is suitable to apply in CPU cooling as long as the arrangement of condenser is above evaporator so gravity will make sure the working fluid flow properly. Unfortunately, this type of cooling is not yet available in commercial due to the limit of specific arrangement of the components.

D. Liu et al. [61] had proposed a thermoelectric mini cooler coupling with a micro thermosiphon cooling system for CPU cooling. When DC current was applied to the thermoelectric drying system, the Peltier cells absorbed heat from the metallic block outside CPU and dissipated heat to the heat sink, thus creating a heat flux between the junctions of two different types of materials. On the hot side, heat was rejected to the adjacent air. The heated air from the hot side of the thermoelectric unit was then passed through the evaporate section of the looped thermosiphon to further transport heat to the condensing section, which could result in two-stage heat transfer process. Experimental results indicated that the cooling production increases with promotion of thermoelectric operating voltage. The maximum surface temperature of CPU reached 70 °C as the prototype CPU power input was equivalent to 84 W. Although thermosyphon does not involve power consumption, but TEC still need power consumption.

Table 8 A summary of studies of heat pipe for CPU cooling

Authors	Working Fluid	Findings
G. Zhou et al. [54]	Air	Under natural convection, the proposed miniature loop heat pipe could efficiently transfer a maximum heat load of 15 W with a junction temperature below 96.6 °C for all test orientations. A minimum total thermal resistance of 2.17 °C/W was achieved at 30 W.
Z.R. Lin et al. [55]	Air	Longer the length of evaporator and pipeline, better the heat transfer of LHP.
Putra et al. [56]	Water	The used of coral as wick structured can improved the performance of heat transfer while decrease the CPU temperature for 38.6 % at the maximum heat load compared to a conventional copper heat sink.
Uddin et al. [57]	Acetone Ethanol	The results show that acetone has lower maximum and steady state temperature of the processor compare with ethanol.
T. Yousefi et al. [22]	Water	It is observed that as the CPU temperature increases, the threshold angle decreases from 60° to 30°. At 10 W, the presence

		of nanofluid has reduced the thermal resistance by 15%, while at 25 W, the thermal resistance has dropped by 22%.
E.N. Stephen et al. [58]	Al ₂ O ₃ -water Ag-water	The experimental results showed that the evaporator thermal resistance is reduced by 34.70% and 20.21%, respectively, for 0.12 vol% of Ag, Al ₂ O ₃ nanoparticles when compared with that of the distilled water.
A. Chauhan, S.G. Kandlikar [60]	Water	The 2° dual taper angle performed the best among all taper angles tested and was able to dissipate a maximum of 280 W without reaching critical heat flux. The surface temperature was maintained at 45.5 °C with a heat transfer coefficient of 22.5 kW/m ² °C.
D. Liu et al. [61]	Water	Experimental results indicated that the cooling production increases with promotion of thermoelectric operating voltage. The maximum surface temperature of CPU reached 70 °C as the prototype CPU power input was equivalent to 84 W.

Jet impingement

Jet impingement cooling method is one of the promising developments in heat transfer application. As its name, this method involve high velocity forced convection by ejecting coolant directly onto a heated surface through convergent nozzle [62]. Due to its high heat transfer rate, this method could benefit in electronic cooling application. To the current development, several parameters that had been study by researchers that effect the cooling performance which are jet to target distance, flow region (Reynold's number), surrounding (confined, semi confined), surface shape (rough or smooth, incline or flat, etc.) and fluid type. To support this, P. Singh [63] had comprehensively study the heat transfer of jet impingement with novel surface roughness. Variable involve in the research are jet to jet spacing, jet to target distance and Reynold's number. The results show that concentric surface roughness has higher heat transfer and fin effectiveness compared to traditional cubic and cylindrical roughness. Furthermore, as the as the jet to jet spacing, jet to target distance and Reynold's number increase, the Nusselt's number also increase. Therefore, heat transfer is increase. Summary of application of jet impingement for CPU cooling are shown in Table 9.

For electronic cooling such as CPU, the best configuration to apply this method is by combining heat sink and jet impingement. P. Naphon, S. Wongwise [64] had study on jet liquid impingement heat transfer characteristics in the mini-rectangular fin heat sink for CPU cooling. A larger velocity in the heat sink results in higher heat transfer coefficient. It is found that the CPU temperatures obtained from the jet liquid impingement cooling system are lower than those from the conventional liquid system. However, this technique requires higher energy consumption.

J.S. Bintoro et al. [65] had design and tested closed-loop cooling system for a CPU inside the desktop. Deionized water is used for working fluid. Surprisingly, this system managed to maintained the chip's surface temperature to be less than 95 °C at the maximum heat dissipation (200 W). Despite of this development, recent development is applying nanofluid in the system to increase cooling performance. Therefore, a combined three heat transfer enhancement techniques could increase significantly heat transfer performance.

P. Naphon et al. [66] had study on nanofluids jet impingement heat transfer and flow in a micro-channel heat sink. TiO₂-water nanofluids is used as working fluid. Convective heat transfer is found remarkably increase for 18.56% at 0.015% concentration. The obtained experimental results show that the suspending of nanoparticles in the base fluid remarkably increases the convective heat transfer in 18.56% at 0.015% nanofluids concentration.

S. Wiriyasart, P. Naphon [67] had study on the cooling performance of the liquid jet impingement of a cold plate heat sink with different fin geometries. Water and TiO₂-water are used as working fluids. Using nanofluids as working fluid gives the thermal resistance lower than de-ionized water as the working fluid. The cold plate heat sink with liquid jet impingement is capable for cooling high power electronic applications over 500 W at the small heating area of 40 mm × 40 mm. Jet-plate spacing to jet diameter ratios $H/D = 0.8$ has the best heat transfer enhancement. Study in various concentration of nanofluid are not yet study.

P.A. de Oliveira, J.R. Barbosa Jr [68] had proposed novel two-phase jet impingement heat sink for active cooling of electronic devices. A single orifice was used to generate the high-speed two-phase impinging jet on the heated surface and R-134a is used as working fluid. The maximum system cooling capacity with a single jet was approximately 160 W, which was achieved with an orifice diameter of 500 μ m. This proposed design achieves a jet impingement heat transfer coefficient above 15 kW/m²K and a heater surface temperature of approximately 30 °C. Next, further study by P.A. de Oliveira, J.R. Barbosa Jr [69] were on the effect of jet length and ambient temperature on the performance of a two-phase jet impingement heat sink refrigeration system. A reduction of the jet length promoted a more vigorous splattering of the jet on the heated surface, enhancing the droplet breakup, which in turn reduced significantly the critical heat flux. An increase of the hot reservoir temperature increased the jet impingement heat transfer coefficient.

B.P. Whelan et al. [70] had develop a miniature jet array water block and a tube bundle remote heat exchanger which can be manufactured simply and cost effectively. The liquid-based cooling system concept successfully dissipated the required 200 W with a chip temperature of 65°C and a water-block base temperature of 53°C. The overall system thermal resistance was determined to decrease from 0.25 K/W at 50W heat input to 0.18 K/W at 200 W loading. A similar drop in the system thermal resistance was obtained for increasing the liquid volumetric flow rate between 2 L/min and 5.5 L/min.

P. Naphon, S. Wongwise [71] had carried out an experimental study of jet nanofluids impingement system for cooling computer processing unit. TiO₂-water nanofluids is used as working fluid. Coolant flow rate varies from 0.008 to 0.020kg/s, the nozzle diameter is set to 1.00, 1.40, 1.80 mm, the distance nozzle-to-fins tip is 2.00 mm, the channel width of the mini-channel heat sink is 1.00 mm were used in the experiment. The results show that average CPU temperatures obtained from the jet nanofluids impingement cooling system are 3.0%, 6.25% lower than those from the jet liquid impingement and from the conventional liquid cooling systems, respectively.

Table 9 A summary of studies of jet impingement for CPU cooling

Authors	Working Fluid	Findings
P. Naphon, S. Wongwise [64]	Water	It is found that the CPU temperatures obtained from the jet liquid impingement cooling system are lower than those from the conventional liquid system.
J.S. Bintoro et al. [65]	Water	This system managed to maintained the chip's surface temperature to be less than 95°C at the maximum heat dissipation (200 W).
[66]	TiO ₂ -water	The obtained experimental results show that the suspending of nanoparticles in the base fluid remarkably increases the convective heat transfer in 18.56% at 0.015% nanofluids concentration.
S. Wiriyasart, P. Naphon [67]	Water TiO ₂ -water	Using nanofluids as working fluid gives the thermal resistance lower than de-ionized water as the working fluid. Jet-plate

		spacing to jet diameter ratios $H/D = 0.8$ has the best heat transfer enhancement.
P.A. de Oliveira, J.R. Barbosa Jr [68]	R-134a	The maximum system cooling capacity with a single jet was approximately 160 W, which was achieved with an orifice diameter of 500 μm . This proposed design achieves a jet impingement heat transfer coefficient above 15 $\text{kW/m}^2\text{K}$ and a heater surface temperature of approximately 30 $^{\circ}\text{C}$.
P.A. de Oliveira, J.R. Barbosa Jr [69]	R-134a	A reduction of the jet length reduced significantly the critical heat flux. An increase of the hot reservoir temperature increased the jet impingement heat transfer coefficient.
B.P. Whelan et al. [69]	Water	The overall system thermal resistance was determined to decrease from 0.25 K/W at 50W heat input to 0.18 K/W at 200 W loading.
P. Naphon, S. Wongwise [71]	TiO_2 -water	The results show that average CPU temperatures obtained from the jet nanofluids impingement cooling system are 3.0%, 6.25% lower than those from the jet liquid impingement and from the conventional liquid cooling systems, respectively.

Conclusion

This paper presented an inclusive review on development of liquid cooling system for CPU. Various methods and working fluid had been reviewed for CPU cooling. Furthermore, various aspects and parameter of each methods were considered. Liquid cooling system are proven to enhance the heat transfer performance for CPU cooling. Furthermore, various working fluid had been study including nanofluids. The presence of nanoparticles in liquid cooling system will also improve the performance of heat transfer. Unfortunately, only low concentration of nanofluids had been used due to the stability of the nanofluid and the limit of cooling system such as size of heat sink. Limited study had been done on hybrid nanofluid for CPU cooling. More synthesis of working fluid with higher heat transfer enhancement are needed in the future.

Application of liquid base working fluid in all liquid cooling methods (heat sink, thermoelectric cooler, heat pipe, jet impingement) enhance significantly the heat transfer performance of CPU cooling system. There are many types of heat sink and liquid block had been study. Current research actively study using heat sink design and different type of working fluid including nanofluid. More study on higher heat transfer enhancement of heat sink design and different type of working fluid need to be proposed in the future. Next, TEC method where the CPU is directly connected with the cold side of TEC and the hot side of TEC is directly connected with water block also one of the efficient ways for CPU cooling. But more study is needed to overcome high-power consumption. Other than that, heat pipe consists of wicked heat pipe and thermosiphon. The study of natural convection using capillary wick needs to be study more to increase the efficiency of cooling device Thermosiphon are limited to gravity orientation. Jet impingement with high velocity profile of working fluid can enhance the cooling performance of cooling device. Jet impingement method has many parameters can be study such as distance jet-to-surface, distance nozzle-to-fins, nozzle diameter and many more. But more study is needed for high power consumption by jet impingement method.

References

- [1] INTEL® CORE™ PROCESSOR FAMILY.
<https://www.intel.com/content/www/us/en/products/processors/core.html>
- [2] Roser, M. and H. Ritchie. *Technological Progress*. [cited 2019 21 August 2019]; Available from: <https://ourworldindata.org/technological-progress>.

- [3] Yabo, W., et al., *Energy saving potential of using heat pipes for CPU cooling*. Applied Thermal Energy, 2018: p. 630-638.
- [4] Azmi, W.H., et al., *Heat transfer augmentation of ethylene glycol: water nanofluids and applications — A review*. International Communications in Heat and Mass Transfer, 2016. **75**: p. 13-23.
- [5] Eastman, J., et al., *Enhanced Thermal Conductivity Through the Development of Nanofluids*. Vol. 457. 1996.
- [6] Ahmadi, M.H., et al., *A proposed model to predict thermal conductivity ratio of Al₂O₃/EG nanofluid by applying least squares support vector machine (LSSVM) and genetic algorithm as a connectionist approach*. Journal of Thermal Analysis and Calorimetry, 2019. **135**(1): p. 271-281.
- [7] Khalaj, A.H. and S.K. Halgamuge, *A Review on efficient thermal management of air- and liquid-cooled data centers: From chip to the cooling system*. Applied Energy, 2017: p. 1165-1188.
- [8] Sidik, N.A.C., M.N.A.W.M. Yazid, and R. Mamat, *A review on the application of nanofluids in vehicle engine cooling system*. International Communications in Heat and Mass Transfer, 2015. **68**: p. 85-90.
- [9] Al-Rashed, A.A.A.A., et al., *Numerical investigation of non-Newtonian water-CMC/CuO nanofluid flow in an offset strip-fin microchannel heat sink: Thermal performance and thermodynamic considerations*. Applied Thermal Engineering, 2019: p. 247-258.
- [10] Zhao, N., et al., *Experimental study on influences of cylindrical grooves on thermal efficiency, exergy efficiency and entropy generation of CPU cooled by nanofluids*. International Journal of Heat and Mass Transfer, 2019: p. 16-32.
- [11] Zhao, N., et al., *Experimental study on thermo-hydraulic performance of nanofluids in CPU heat sink with rectangular grooves and cylindrical bugles based on exergy efficiency*. Energy Conversion and Management, 2019. **181**: p. 235-246.
- [12] Hasan, H.A., Z. Alquziweeni, and K. Sopian, *Heat transfer enhancement using nanofluids for cooling a Central Processing Unit (CPU) system*. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, 2018. **51**(2): p. 145-157.
- [13] Gunnasegaran, P., M.Z. Abdullah, and M.Z. Yusoff, *Heat Transfer in a Loop Heat Pipe Using Fe₂NiO₄-H₂O Nanofluid*. MATEC Web Conf., 2017. **109**.
- [14] Sarafraz, M., et al., *On the convective thermal performance of a CPU cooler working with liquid gallium and CuO/water nanofluid: A comparative study*. Applied Thermal Engineering, 2017. **112**: p. 1373-1381.
- [15] Bahiraei, M. and S. Heshmatian, *Efficacy of a novel liquid block working with a nanofluid containing graphene nanoplatelets decorated with silver nanoparticles compared with conventional CPU coolers*. Applied Thermal Engineering, 2017. **127**: p. 1233-1245.
- [16] Sun, B. and H. Liu, *Flow and heat transfer characteristics of nanofluids in a liquid-cooled CPU heat radiator*. Applied Thermal Engineering, 2017. **115**: p. 435-443.
- [17] Al-Rashed, M.H., et al., *Investigation on the CPU nanofluid cooling*. Microelectronics Reliability, 2016. **63**: p. 159-165.
- [18] Gunnasegaran, P., et al., *Optimization of SiO₂ nanoparticle mass concentration and heat input on a loop heat pipe*. Case Studies in Thermal Engineering, 2015. **6**: p. 238-250.
- [19] Turgut, A. and E. Elbasan. *Nanofluids for electronics cooling*. in *2014 IEEE 20th International Symposium for Design and Technology in Electronic Packaging, SIITME 2014*. 2014.

- [20] Nazari, M., M. Karami, and M. Ashouri, *Comparing the thermal performance of water, Ethylene Glycol, Alumina and CNT nanofluids in CPU cooling: Experimental study*. Experimental Thermal and Fluid Science, 2014. **57**: p. 371-377.
- [21] Jajja, S.A., W. Ali, and H.M. Ali, *Multiwalled carbon nanotube nanofluid for thermal management of high heat generating computer processor*. Heat Transfer - Asian Research, 2014. **43**(7): p. 653-666.
- [22] Yousefi, T., et al., *Experimental investigation on the performance of CPU coolers: Effect of heat pipe inclination angle and the use of nanofluids*. Microelectronics Reliability, 2013. **53**(12): p. 1954-1961.
- [23] Putra, N., et al., *Application of Al₂O₃ nanofluid on sintered copper-powder vapor chamber for electronic cooling*, in *Advanced Materials Research*. 2013. p. 423-428.
- [24] Rafati, M., A.A. Hamidi, and M. Shariati Niaser, *Application of nanofluids in computer cooling systems (heat transfer performance of nanofluids)*. Applied Thermal Engineering, 2012. **45-46**: p. 9-14.
- [25] Selvakumar, P. and S. Suresh, *Convective performance of CuO/water nanofluid in an electronic heat sink*. Experimental Thermal and Fluid Science, 2012. **40**: p. 57-63.
- [26] Husam, H.A., A. Zainab, and S. Kamaruzzaman, *Heat Transfer Enhancement Using Nanofluids For Cooling A Central Processing Unit (CPU) System*. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, 2018: p. 145-157.
- [27] Roberts, N.A. and D.G. Walker, *Convective Performance of Nanofluids in Commercial Electronics Cooling Systems*. Applied Thermal Engineering, 2010. **30**(16): p. 2499-2504.
- [28] Siddique, A.B. and K.N. Prabhu. *Replacement of heat sink fan by nanocoolants for enhancement of CPU efficiency*. in *IET Conference Publications*. 2013.
- [29] !!! INVALID CITATION !!! [12, 24].
- [30] Bahiraei, M., S. Heshmatian, and M. Keshavarzi, *Multi-criterion optimization of thermohydraulic performance of a mini pin fin heat sink operated with ecofriendly graphene nanoplatelets nanofluid considering geometrical characteristics*. Journal of Molecular Liquids, 2019. **276**: p. 653-666.
- [31] Bahiraei, M., S. Heshmatian, and M. Keshavarzi, *Multi-attribute optimization of a novel micro liquid block working with green graphene nanofluid regarding preferences of decision maker*. Applied Thermal Engineering, 2018. **143**: p. 11-21.
- [32] Qi, C., et al., *Experimental study on thermo-hydraulic performances of CPU cooled by nanofluids*. Energy Conversion and Management, 2017. **153**: p. 557-565.
- [33] Eastman, J.A., et al., *Thermal transport in nanofluids*. Annu. Rev. Mater. Res., 2004. **34**: p. 219-246.
- [34] Mehrali, M., et al., *Investigation of thermal conductivity and rheological properties of nanofluids containing graphene nanoplatelets*. Nanoscale Research Letters, 2014. **9**(1): p. 15.
- [35] Kang, S. and J. Cennamo. *Closed Loop Liquid Cooling for High Performance Computer Systems*. in *ASME 2007 InterPACK Conference collocated with the ASME/JSME 2007 Thermal Engineering Heat Transfer Summer Conference*. 2007. Vancouver, British Columbia, CANADA.
- [36] Atik, Ö. and H. Ertürk, *Enhancement of CPU Liquid Cooling Performance using hBN Nanofluids*. Journal of Thermal Science and Engineering Applications, 2019: p. 1-5.
- [37] Khaleduzzaman, S.S., et al., *Experimental analysis of energy and friction factor for titanium dioxide nanofluid in a water block heat sink*. International Journal of Heat and Mass Transfer, 2017: p. 77-85.

- [38] Kim, C.-H., M.-J. Lee, and C.Y. Park, *An experimental study on the heat transfer and pressure drop characteristics of electronics cooling heat sinks with FC-72 flow boiling*. Journal of Mechanical Science and Technology 2018; p. 1449-1462.
- [39] Jajja, S.A., et al., *Water cooled minichannel heat sinks for microprocessor cooling: Effect of fin spacing*. Applied Thermal Engineering, 2014. **64**(1): p. 76-82.
- [40] Ming-Chang, L. and W. Chi-Chuan, *Effect of the inlet location on the performance of parallel-channel cold-plate*. IEEE Transactions on Components and Packaging Technologies, 2006: p. 30-38.
- [41] Kumar, S. and P.K. Singh, *Effects of flow inlet angle on flow maldistribution and thermal performance of water cooled mini-channel heat sink*. International Journal of Thermal Sciences, 2019. **138**: p. 504-511.
- [42] Wong, C.M., et al. *Variation in heat sink shape for thermal analysis*. in *AIP Conference Proceedings*. 2017. American Institute of Physics.
- [43] Atik, Ö. and H. Ertürk, *Enhancement of Central Processing Unit Liquid Cooling Performance Using Hexagonal Boron Nitride Nanofluids*. Journal of Thermal Science and Engineering Applications, 2019. **11**(3): p. 034501-034501-5.
- [44] Guclu, T. and E. Cuce, *Thermoelectric Coolers (TECs): From Theory to Practice*. Journal of Electronic Materials, 2019. **48**(1): p. 211-230.
- [45] Tan, S.O. and H. Demirel, *Performance and cooling efficiency of thermoelectric modules on server central processing unit and Northbridge*. Computers & Electrical Engineering, 2015. **46**: p. 46-55.
- [46] Mohammad Siddique, A.R., et al. *Experimental Analysis for Liquid Based Thermoelectric Electronic Cooling (LTEEC) System*. in *2nd International Conference on Electrical and Electronic Engineering, ICEEE 2017*. 2018.
- [47] Huang, H.S., et al., *Thermoelectric water-cooling device applied to electronic equipment*. International Communications in Heat and Mass Transfer, 2010. **37**(2): p. 140-146.
- [48] Ahammed, N., L.G. Asirvatham, and S. Wongwises, *Thermoelectric cooling of electronic devices with nanofluid in a multiport minichannel heat exchanger*. Experimental Thermal and Fluid Science, 2016. **74**: p. 81-90.
- [49] Naphon, P. and S. Wiriyaart, *Liquid cooling in the mini-rectangular fin heat sink with and without thermoelectric for CPU*. International Communications in Heat and Mass Transfer, 2009. **36**(2): p. 166-171.
- [50] Hu, H.M., et al., *Experimental study on water-cooled thermoelectric cooler for CPU under severe environment*. International Journal of Refrigeration, 2016. **62**: p. 30-38.
- [51] Khonsue, O., *Experimental on the liquid cooling system with thermoelectric for personal computer*. Heat and Mass Transfer, 2012. **48**(10): p. 1767-1771.
- [52] K, Z., et al. *Experimental study of flat heat pipe for cooling TV Boxes*. in *2018 IEEE International Telecommunications Energy Conference (INTELEC)*. 2018.
- [53] Faghri, A., *Review and Advances in Heat Pipe Science and Technology*. Journal of Heat Transfer, 2012. **134**(12): p. 1-18.
- [54] Zhou, G., J. Li, and Z. Jia, *Power-saving exploration for high-end ultra-slim laptop computers with miniature loop heat pipe cooling module*. Applied Energy, 2019. **239**: p. 859-875.
- [55] Lin, Z.R., et al. *An experimental study on applying miniature loop heat pipes for laptop PC cooling*. in *29th IEEE Semiconductor Thermal Measurement and Management Symposium*. 2013.
- [56] Putra, N., et al., *Thermal performance of biomaterial wick loop heat pipes with water-base Al₂O₃ nanofluids*. International Journal of Thermal Sciences, 2014. **76**: p. 128-136.

- [57] Uddin, A.I. and C.M. Feroz, *Effect of working fluid on the performance of a miniature heat pipe system for cooling desktop processor*. Heat and mass transfer, 2009. **46**(1): p. 113.
- [58] Stephen, E.N., et al., *Heat transfer performance of a compact loop heat pipe with alumina and silver nanofluid*. Journal of Thermal Analysis and Calorimetry, 2019. **136**(1): p. 211-222.
- [59] Ramezanizadeh, M., et al., *Application of nanofluids in thermosyphons: A review*. Journal of Molecular Liquids, 2018. **272**: p. 395-402.
- [60] Chauhan, A. and S.G. Kandlikar, *Characterization of a dual taper thermosiphon loop for CPU cooling in data centers*. Applied Thermal Engineering, 2019. **146**: p. 450-458.
- [61] Liu, D., et al., *Thermoelectric mini cooler coupled with micro thermosiphon for CPU cooling system*. Energy, 2015. **83**: p. 29-36.
- [62] Kheirabadi, A.C. and D. Groulx, *Cooling of server electronics: A design review of existing technology*. Applied Thermal Engineering, 2016. **105**: p. 622-638.
- [63] Singh, P., et al., *Effect of micro-roughness shapes on jet impingement heat transfer and fin-effectiveness*. International Journal of Heat and Mass Transfer, 2019: p. 80-95.
- [64] Naphon, P. and S. Wongwises, *Investigation on the jet liquid impingement heat transfer for the central processing unit of personal computers*. International Communications in Heat and Mass Transfer, 2010. **37**(7): p. 822-826.
- [65] Bintoro, J.S., A. Akbarzadeh, and M. Mochizuki, *A closed-loop electronics cooling by implementing single phase impinging jet and mini channels heat exchanger*. Applied Thermal Engineering, 2005. **25**(17): p. 2740-2753.
- [66] Naphon, P., L. Nakharintr, and S. Wiriyaart, *Continuous nanofluids jet impingement heat transfer and flow in a micro-channel heat sink*. International Journal of Heat and Mass Transfer, 2018. **126**: p. 924-932.
- [67] Wiriyaart, S. and P. Naphon, *Liquid impingement cooling of cold plate heat sink with different fin configurations: High heat flux applications*. International Journal of Heat and Mass Transfer, 2019. **140**: p. 281-292.
- [68] de Oliveira, P.A. and J.R. Barbosa, *Novel two-phase jet impingement heat sink for active cooling of electronic devices*. Applied Thermal Engineering, 2017. **112**: p. 952-964.
- [69] de Oliveira, P.A. and J.R. Barbosa, *Effect of jet length and ambient temperature on the performance of a two-phase jet impingement heat sink refrigeration system*. International Journal of Refrigeration, 2017. **75**: p. 331-342.
- [70] Whelan, B.P., R. Kempers, and A.J. Robinson, *A liquid-based system for CPU cooling implementing a jet array impingement waterblock and a tube array remote heat exchanger*. Applied Thermal Engineering, 2012. **39**: p. 86-94.
- [71] Naphon, P. and S. Wongwises, *Experimental study of jet nanofluids impingement system for cooling computer processing unit*. Journal of Electronics Cooling and Thermal Control, 2011. **1**(03): p. 38.

NUMERICAL INVESTIGATION OF TAPERED DL-MCHS WITH ALTERNATING FLOW CONFIGURATION

Jia-Wei Zuo¹
Kok-Cheong Wong²
Hoon Kiat Ng³

¹Department of Mechanical, Materials and Manufacturing Engineering, University of Nottingham Malaysia Campus, Jalan Broga, Semenyih, Selangor, Malaysia (Email: kedx5zja@email.nottingham.edu.my)

²Department of Mechanical, Materials and Manufacturing Engineering, University of Nottingham Malaysia Campus, Jalan Broga, Semenyih, Selangor, Malaysia

³Department of Mechanical, Materials and Manufacturing Engineering, University of Nottingham Malaysia Campus, Jalan Broga, Semenyih, Selangor, Malaysia

Abstract: The present study numerically investigated the thermal and hydraulic performances of width-tapered Double-layered microchannel heat sink (DL-MCHS) with alternating flow configuration. Three-dimensional conjugate heat transfer model has been used to study the performances of different geometries of tapered microchannel within the DL-MCHS by varying the inlet width and outlet width of channels. The DL-MCHS with alternating flow configuration shows an increase in hydraulic performance and a decrease in thermal performance with the increase in inlet channel width or outlet channel width. The maximum temperature difference and temperature distribution on the bottom surface of the heat sink along the flow direction has been studied to find the effects of channel inlet width and channel outlet width on temperature uniformity. Better temperature uniformity was observed for DL-MCHS with small inlet channel width or outlet channel width under alternating flow. FOM (Figure of merit) is used to evaluate the thermo-hydraulic performance among MCHS with different geometries of tapered channel designs under alternating flow. As compared to the benchmark of DL-MCHS with straight channel design with inlet and outlet channel size of 50 μ m, the optimum DL-MCHS with tapered channel design shows 76% improvement in thermo-hydraulic performance. The alternating flow configuration has also been compared to both parallel and counter flow configurations. The DL-MCHS with alternating flow shows better thermal performance for the case of higher channel inlet size or smaller channel outlet size as compared with parallel flow configuration. However, the alternating flow configuration and counter flow configurations show large similar thermal performance.

Keywords: Double-layer MCHS, Alternating flow, Thermo-hydraulic performance, Temperature uniformity

Introduction

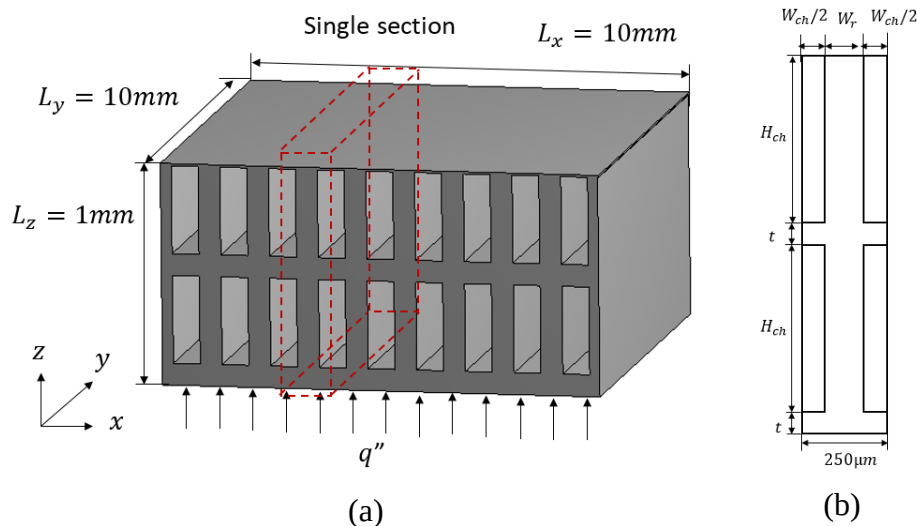
Nowadays, the problem of heat dissipation at high heat flux has received much attention due to the fast development of electronic components. As an attempt to solve this issue, Tuckerman and Pease [1] proposed the design of silicon-based single-layered microchannel heat sink (SL-MCHS), based on their experimental result, they found that by using water as coolant, the SL-MCHS can achieve maximum thermal resistance, R_t of 0.09°C/cm² when subject to the heat flux q'' of 790 W/cm². From there onwards, scientists [2-7] have studied the performance for MCHSs by utilizing porous material as part of the substrate material while some researchers [8-11] studied the performance for SL-MCHS by using Nano-fluid as coolant. Their findings concluded that the performance could be improved through different channel design and different coolant.

Vafai and Zhu [12] proposed the concept of DL-MCHS which is a substantial improvement over a conventional single-layer MCHS by reducing the temperature gradient along axial-direction. Following their work, other researches [12-17] have studied the performance for DL-MCHS. Wu et al. [13] numerically investigated the parametric effects of channel number, aspect ratio and velocity ratio on the overall thermal performance. Lin et al. [14] used a simplified conjugate-gradient method to optimize the flow and heat transfer for silicon-based DL-MCHS based on six different design variables. Wong and Muezzin [15] numerically studied the thermal performance for DL-MCHS with parallel and counter flow configuration. Wong and Ang [16] numerically studied the effects of vertically tapered and converging channel of a DL-MCHS on its thermal and hydraulic performance. Xie et al. [17] numerically investigated the laminar heat transfer and pressure of DL-MCHS. Wei et al. [18] experimental and numerical studied the thermal performance for stacked DL-MCHS by using silicon as substrate material and water as coolant. Wong et al. [19] numerically investigated the thermal performance for DL-MCHS with tapered channel profile, they found that for most of the DL-MCHS designs, DL-MCHS with counter flow is found to have better thermal performance than those of parallel flow but the performance become reverse for those DL-MCHS designs having highly converging channel with small channel outlet size. These studies all confirmed that the DL-MCHS has its own advantages when compared to the SL-MCHS.

Although DL-MCHS with parallel and counter flow configuration has been investigated by many researchers [12-19], DL-MCHS with tapered channel profile and with alternating flow configuration has seldom being investigated. Thus, the present study numerically investigated the thermal, hydraulic and thermo-hydraulic performance for DL-MCHS with alternating flow configuration for different W_i and W_o .

Methodology

The physical model of the DL-MCHS is illustrated in Fig 1a with an overall width, L_x length, L_y and height, L_z of 10mm, 10mm and 1mm, respectively. The heat sink consists of 40 repeated sections with each section width W of 250 μ m. The geometrical parameters for a single unit is shown in Fig.1b. Each unit has identical top and bottom layer dimensions and each unit can be divided into left and right part. Each layer has channel height, H_{ch} of 375 μ m and horizontal rib thickness, t of 125 μ m. For the left part and right part, the fluid flows along y-direction and negative y-direction, respectively. The computational domain of the single section is illustrated in Fig. 1c.



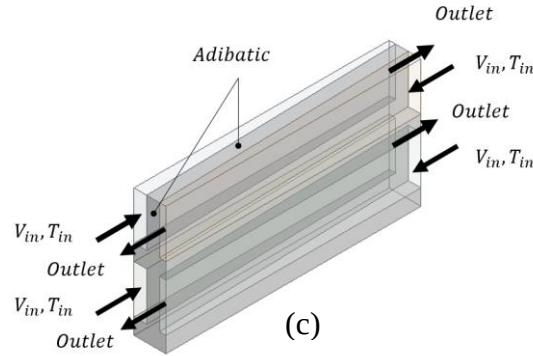


Figure 1 (a) Schematics of double-layer MCHS, (b) Repeating section of the DL-MCHS with alternating flow (c) Schematics of computational domain

Table 10 Geometrical Parameters used for DL-MCHS with alternating flow

Parameter	Minimum Value	Maximum Value
$W_i(\mu\text{m})$	50	200
$W_o(\mu\text{m})$	50	200
$W_r(\mu\text{m})$	50	200

Parametric studies were carried out to understand the effects of geometrical parameters channel outlet width, W_i and channel inlet width, W_o . In present study, W_o and W_i are both varied from $50\mu\text{m}$ to $200\mu\text{m}$ with step size of $10\mu\text{m}$. By varying W_i and W_o , the channel vertical rib thickness also varied from $50\mu\text{m}$ to $200\mu\text{m}$ with constant W of $500\mu\text{m}$. Based on different value of W_i and W_o , total 256 geometries were generated with the combination of parameters specified in Table 1. By varying the value of the inlet channel width W_i and outlet channel width, W_o , the MCHS designs with converging channel ($W_i > W_o$), straight channel ($W_i = W_o$) and then diverging channel ($W_i < W_o$) are created for investigation. To solve the problem, the following assumptions are made:

1. The effects of gravity and other forms of body forces are negligible.
2. The fluid flow and heat transfer are in steady-state.
3. The flow is incompressible and laminar
4. The properties of fluid and solid are constant
5. Heat losses of the MCHS to the ambient are ignored.

Based on the assumptions above, the governing equations are as follows:

- Continuity equation:

$$\nabla \cdot \vec{V} = 0 \quad (1)$$

where $\vec{V}$ is the velocity vector

- Momentum equation:

$$\rho_f(\vec{V} \cdot \nabla)\vec{V} = -\nabla p + \mu_f \nabla^2 \vec{V} \quad (2)$$

where ρ_f , μ_f and p are the coolant density, viscosity and pressure, respectively.

- Energy equation for fluid:

$$\rho_f c_{p,f}(\vec{V} \cdot \nabla)T_f = k_f \nabla^2 T_f \quad (3)$$

where T_f , $c_{p,f}$ and k_f are the coolant temperature, specific heat and thermal conductivity, respectively.

Energy equation for solid:

$$k_s \nabla^2 T_s = 0 \quad (4)$$

where k_s and T_s are the solid thermal conductivity and temperature, respectively.

All the cases are subjected to the same boundary conditions of a uniform base heat flux q'' of 100 W/cm^2 at the bottom surface of the heat sink, inlet temperature T_{in} of 300K , and an overall volumetric flow rate, $\dot{Q}$ of 200 ml/min . All the outlet is set to be pressure outlet P_{out} with zero pressure. Top, front, and back solid surfaces of the domain are assumed to be adiabatic. Left and right surfaces of the domain are set to be symmetry.

The boundary conditions are shown in Fig. 1(c) for DL-MCHS with alternating flow configuration. The inlet boundary conditions are stated as follows:

$$\text{At inlets: } v = V_{in}, \quad u = w = 0, \quad T_f = T_{in} \quad (5)$$

The outlet boundary conditions are stated as follows:

$$\text{At outlets: } P = P_{out} \quad (6)$$

The boundary conditions at the fluid-solid interface are as follows:

$$\vec{V} = 0, \quad T_s = T_f, \quad k_s \nabla T_s = k_f \nabla T_f \quad (7)$$

The model were solved by using SIMPLE algorithm for pressure-velocity coupling. The convection–diffusion formulation of the momentum and energy equation of the fluid was solved by using second-order upwind scheme. The convergence criteria of residuals for resolved continuity, velocity, and energy equation is set to below 10^{-5} .

Other parameters used to evaluate the performance of MCHSs are described as follows:

Total pumping power:

$$\Omega = \dot{Q}_{bottom} \Delta P_{bottom} + \dot{Q}_{top} \Delta P_{top} = v_{in,bottom} W_{ch} H_{ch} \Delta P_{bottom} + v_{in,top} W_{ch} H_{ch} \Delta P_{top} \quad (8)$$

Average heat transfer coefficient, h_m :

$$h_m = \frac{q''}{(\bar{T}_b - T_{in})} \quad (9)$$

Thermal resistance, R_t , which assesses the heat dissipation capability of MCHSs:

$$R_t = \frac{T_{b,max} - T_{in}}{q'' L_x L_y} \quad (10)$$

Maximum temperature difference on the bottom surface of the heat sink, ΔT_b , which represents the cooling uniformity of MCHSs:

$$\Delta T_b = T_{b,max} - T_{b,min} \quad (11)$$

where $\bar{T}_b$ is the average wall temperature wall temperature along the centreline of the bottom surface of the MCHS. $T_{b,max}$ and $T_{b,min}$ are the maximum and minimum bottom wall temperature on the bottom surface of the heat sink.

To compare the thermal-hydraulic performance between different MCHS designs, Figure of merit (FOM) is applied to determine the improvement of one design in terms of heat transfer and pumping power at the same time:

$$\text{FOM} = \frac{(h_{m,new}/h_{m,base})}{(\Omega_{new}/\Omega_{base})^{1/3}} \quad (12)$$

Results

Thermal and hydraulic performance

The code has been validated in the previous work [19]. The thermal performance in terms of thermal resistance, R_t and hydraulic performance in terms of pressure drop, ΔP plotted in $\log_{10}$ scale for DL-MCHS with different value of W_i and W_o under alternating flow is shown in Fig

2 and Fig 3, respectively. As can be seen from the Fig 2 that, the design with small W_i or W_o gives lower R_t and R_t increases with either W_i or W_o increases, where the lowest value of R_t appears at design with $W_i = 50\mu\text{m}$ $W_o = 50\mu\text{m}$ and highest value of R_t appears at design with $W_i = 200\mu\text{m}$ $W_o = 200\mu\text{m}$. As for hydraulic performance as shown in Fig. 3, opposite trend can be found from R_t that, the design with small W_i or W_o gives higher ΔP and ΔP decreases with the increase of either W_i or W_o increase. When the W_i or W_o is small, the decrement of ΔP is dramatic and the decrement becomes less significant with increase of either W_i or W_o . The lowest value of ΔP appears at design with $W_i = 200\mu\text{m}$ $W_o = 200\mu\text{m}$ and highest value of ΔP appears at design with $W_i = 50\mu\text{m}$ $W_o = 50\mu\text{m}$. This is as expected, since small channel width gives better thermal performance but in sacrifice of hydraulic performance [19].

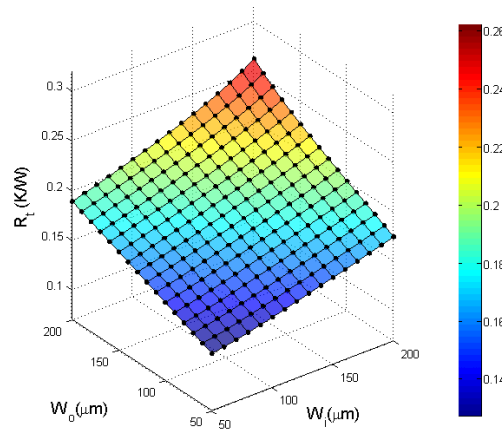


Figure 2 The thermal resistance for DL-MCHS with alternating flow configuration.

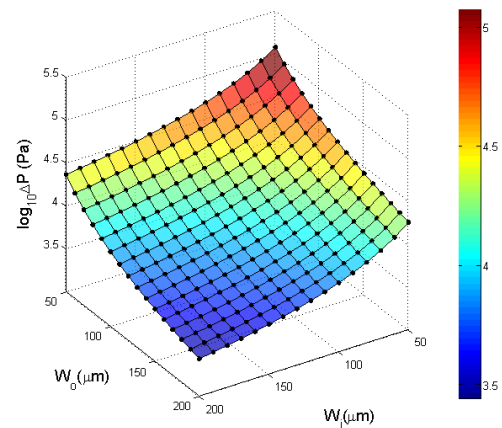


Figure 3 The pressure drop for DL-MCHS with alternating flow configuration

To see the effect of different flow configurations on thermal performance, the results of R_t for DL-MCHS with alternating flow configuration is compared to our previous study's results [19]. Totally 32 designs with 4 different values of W_i of and 4 different values of W_o for DL-MCHS with parallel and counter flow configuration is used for comparison as shown in Fig 4 and 5, respectively. As can be seen from Fig 4 that, the DL-MCHS with alternating flow configuration shows lower value of R_t when W_i at high values or W_o at small values as compared to the DL-MCHS with parallel flow configuration. For all designs investigated, the alternating flow configuration with design of $W_i = 50\mu\text{m}$ $W_o = 200\mu\text{m}$ shows 0.065K/W lower values of R_t as compared to the DL-MCHS with parallel flow configuration. As for DL-

MCHS with counter flow configuration shown in Fig 5, the DL-MCHS with alternating flow configuration shows largely similar results as compared to the DL-MCHS with counter flow configuration.

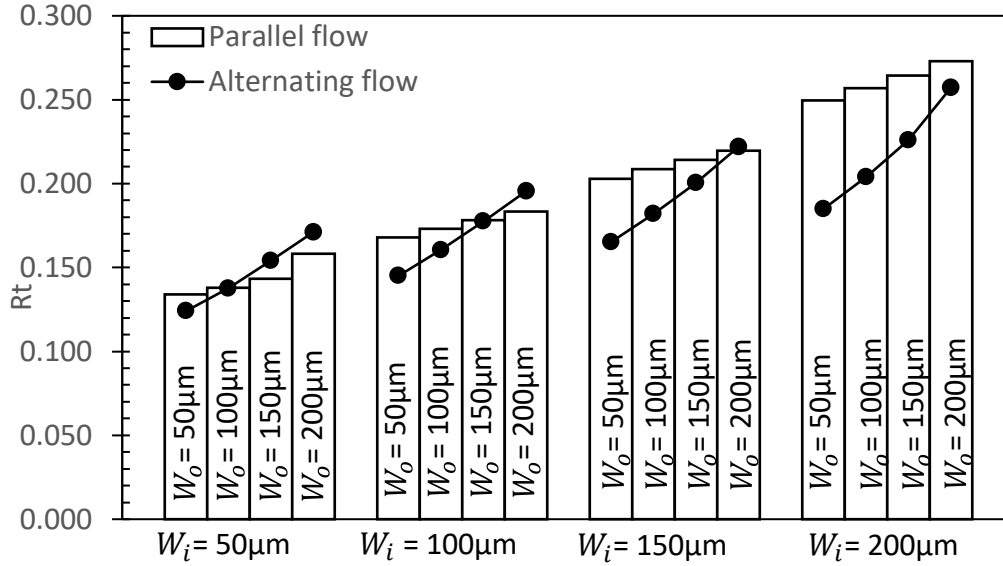


Figure 4 The comparison of R_t between DL-MCHS with alternating and parallel flow configuration.

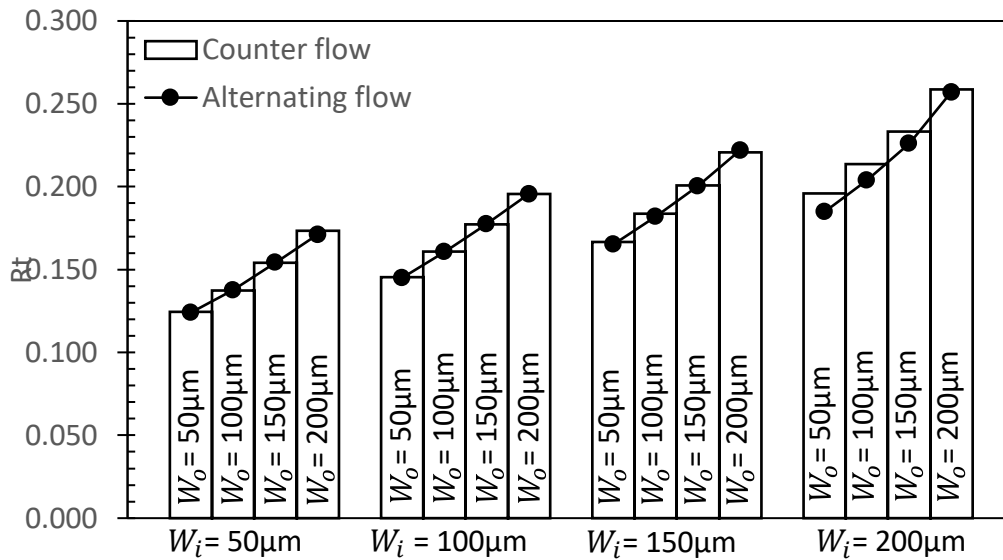


Figure 5 The comparison of R_t between DL-MCHS with alternating and counter flow configuration.

Temperature uniformity

Besides of thermal resistance, ΔT_b is another parameter to evaluate the thermal performance in terms of temperature uniformity. Fig. 6 shows the ΔT_b for DL-MCHS with alternating flow configuration and with different value of W_i and W_o . First looking into the effect of W_o , for any given value of W_i , ΔT_b increases with the increase of W_o , which $W_o = 50\mu\text{m}$ has the lowest value of ΔT_b and $W_o = 200\mu\text{m}$ has the highest value of ΔT_b for all values of W_i investigated. This indicate that the smaller W_o gives better temperature uniformity. Then looking into the effect of W_i , for any given value of W_o , ΔT_b decreases with the increase of W_i

until it reaches the lowest value of ΔT_b and then increases. The lowest value of ΔT_b for different W_o varies from $W_i = 80\mu\text{m}$ to $W_i = 100\mu\text{m}$. This indicate that the design with small channel inlet size has better temperature uniformity. For all cases investigated, the DL-MCHS design with $W_i = 80\mu\text{m}$ $W_o = 50\mu\text{m}$ under alternating flow has the lowest ΔT_b of 3.297°C .

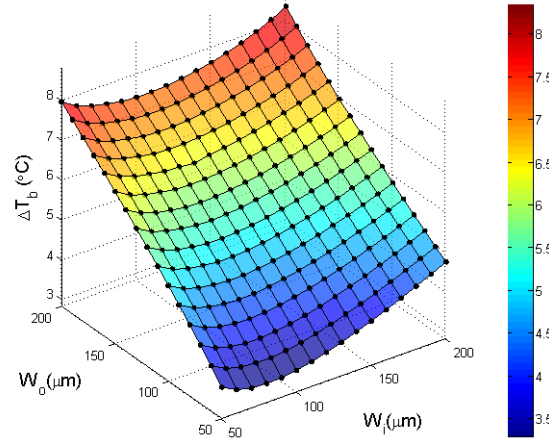


Figure 6 The ΔT_b for DL-MCHS with alternating flow configuration

To explain the above results, the temperature distribution on the bottom surface of the heat sink along the flow direction, $T_{y,\max}$ for DL-MCHS with alternating flow and different W_i and W_o is shown in Fig 7. As can be seen from Fig 7 that, for all designs investigated, the $T_{y,\max}$ increases with the increase of y/L until $T_{y,\max}$ reaches the symmetry position at $y/L=0.5$ and then decreases. This is as expected, since each DL-MCHS with alternating flow design has exact channel profile but different flow direction in each section's left and right part of the computational domain as mentioned in Fig. 1. For DL-MCHS with alternating flow configuration and straight channel design ($W_i = W_o$) as shown in Fig 7(a), the ΔT_b increases with the increase of W_{ch} . The reason is that the increase of temperature at $y/L=0.5$ ($T_{b,\max}$) is larger than the increase of temperature at $y/L=0$ ($T_{b,\min}$) with the increase of W_{ch} , which indicated that $T_{b,\max}$ is more sensitive to the changes of W_{ch} as compared to $T_{b,\min}$. Comparing the DL-MCHS designs with alternating flow configuration and same W_i of $50\mu\text{m}$ for different W_o of $50\mu\text{m}$, $100\mu\text{m}$, $150\mu\text{m}$, $200\mu\text{m}$ as shown in Fig 7(b), ΔT_b increases with the increase of W_o . This happens is because coolant's velocity decreases with the increase of W_o along the flow direction, which weakens the heat removal ability, greatly increases the $T_{b,\max}$ and increases the ΔT_b . While comparing the DL-MCHS with alternating flow configuration and with same W_o for different W_i of $50\mu\text{m}$, $100\mu\text{m}$, $150\mu\text{m}$, $200\mu\text{m}$ as shown in Fig 7(c), ΔT_b first decreases and then increases with the increase of W_i . The reason is that, with the increase of W_i , the channel width change at $y/L=0.5$ is only half as compared to $y/L=0$, which gives lower temperature increase at $y/L=0.5$. However, $T_{b,\max}$ is more sensitive to the channel width change as compared to $T_{b,\min}$. As a combination of this two effects, ΔT_b shows a decrease with the increase of W_i when W_i is small and shows an increase with the increase of W_i after W_i reaches a certain value.

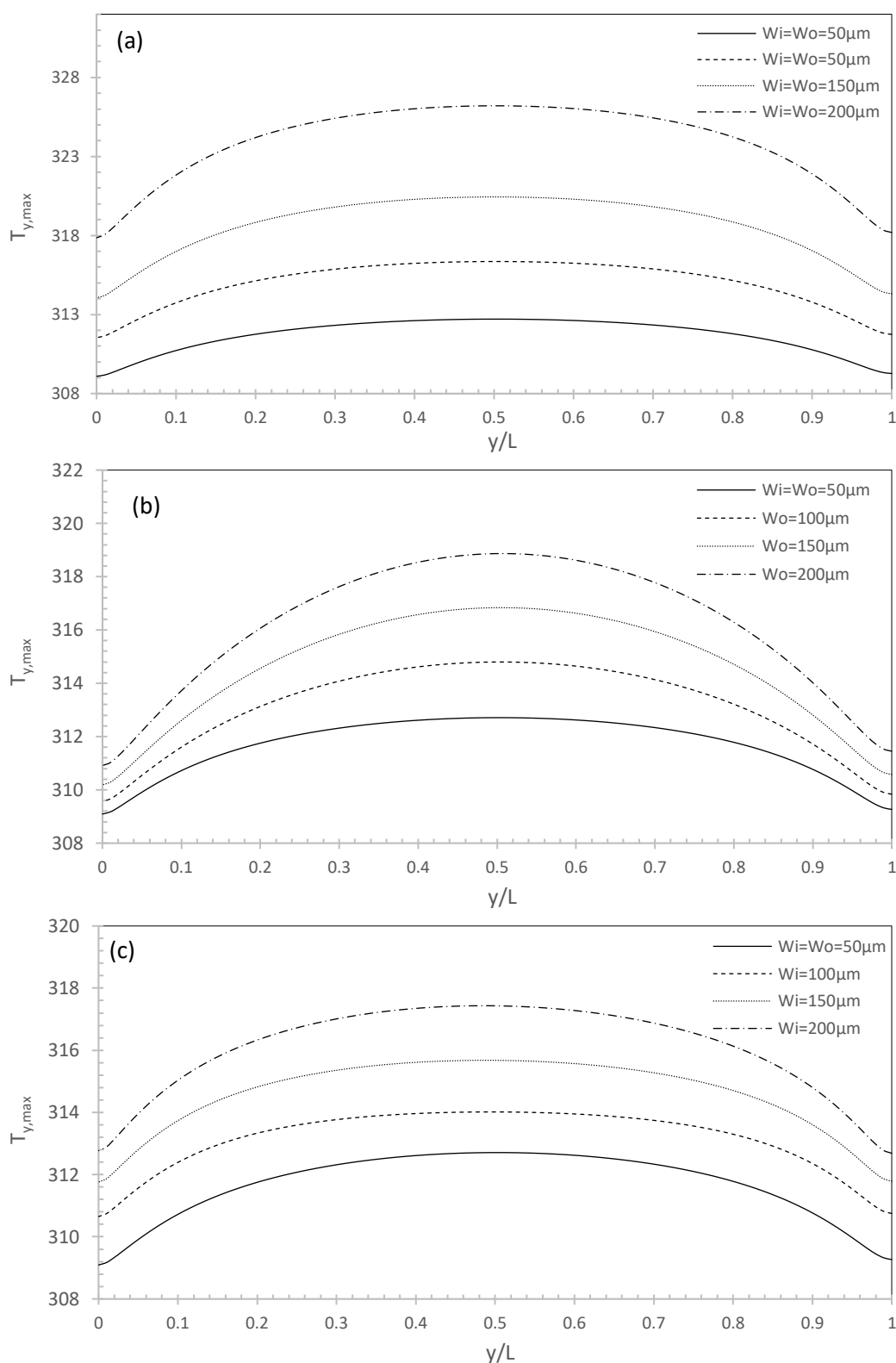


Figure 7 Temperature distribution for DL-MCHS with alternating flow configuration with (a) Straight channel design (b) tapered channel design with fixed W_i of $50\mu m$ (c) tapered channel design with fixed W_o of $50\mu m$

Thermo-hydraulic performance

To take into account of both thermal and hydraulic performance at the same time, Fig 8 shows the FOM for DL-MCHS with alternating flow configuration and different channel profile. For all designs investigated, the DL-MCHS with alternating flow and straight channel design of $W_i = 50\mu\text{m}$ $W_o = 50\mu\text{m}$ is used for comparison. First looking into the effect of W_i , for any given value of W_o , increasing the W_i will increase the FOM until an optimum point is reached. After this point, increasing W_i results in a decrease of FOM. Then looking into the effect of W_o , at low value of W_i ($W_i < 140\mu\text{m}$), The FOM increases with the increase of W_o . For higher value of W_i ($W_i \geq 140\mu\text{m}$), the FOM of the design with $W_o = 180\mu\text{m}$, $190\mu\text{m}$, and $200\mu\text{m}$ decrease rapidly with the increase of W_i . This indicated that the DL-MCHS with alternating flow configuration with designs of larger W_o have better thermos-hydraulic performance when W_i is small. This happens is because when W_i and W_o is small, increase W_i or W_o only slightly increases the R_t , but dramatically decreases the ΔP as shown in Fig 2 and 3. While keep increasing W_i or W_o , the decrements of ΔP becomes less significant as shown in Fig. 2. When W_i/W_o increased till a certain value, the increment of hydraulic performance can't cover the decrement of the thermal performance, which results in a decrease in the thermo-hydraulic performance. For all cases investigated, The value of FOM is larger than 1 and the design of $W_i = 160\mu\text{m}$ $W_o = 190\mu\text{m}$ gives the highest value of FOM of 1.7605 when compared to the design of $W_i = 50\mu\text{m}$ $W_o = 50\mu\text{m}$, which means that the design with $W_i = 160\mu\text{m}$ $W_o = 190\mu\text{m}$ has the best thermo-hydraulic performance among all cases investigated.

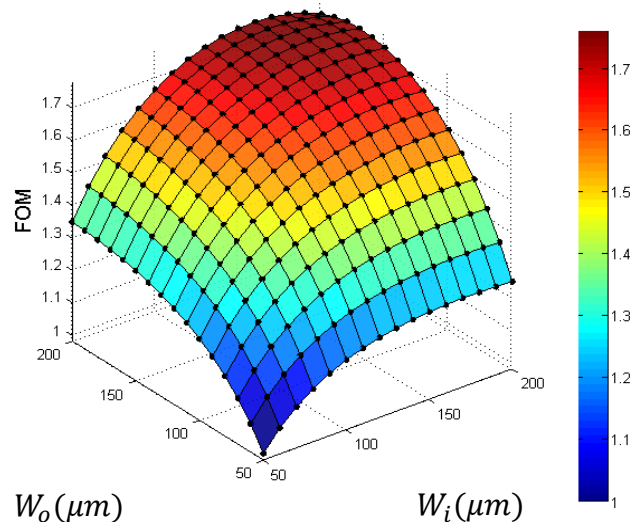


Figure 8 FOM for DL-MCHS with alternating flow configuration

Conclusions

The thermal and hydraulic performance have been investigated for DL-MCHS with alternating flow and different inlet width W_i , outlet width W_o through simulation. The thermal resistance, pressure drop, temperature uniformity and FOM have been analyzed. The conclusions can be drawn as follows

- The design with small inlet channel width or outlet channel width gives lower thermal resistance but results in high pressure drop.
- The DL-MCHS with alternating flow configuration shows better thermal performance (lower thermal resistance) for the case of higher channel inlet size or smaller channel outlet size as compared to the DL-MCHS with parallel flow configuration.

- The DL-MCHS with alternating flow configuration shows largely similar results of thermal resistance as compared to the DL-MCHS with counter flow configuration
- The DL-MCHS with alternating flow design with small inlet channel width or small outlet channel width gives better temperature uniformity. The optimum DL-MCHS with alternating flow with channel design has the lowest temperature difference of 3.297°C.
- The optimum DL-MCHS with alternating flow and tapered channel show about 76% better in thermo-hydraulic performance as compared with benchmark of DL-MCHS with straight channel design with inlet and outlet channel size of 50µm

Acknowledgement

The authors would like to thank MOSTI (now known as MESTECC) to fund this project through the Sciencefund Grant of Project No. 03-02-12-SF0260.

References (Chicago style) Minimum 15 references

- [1] D.B. Tuckerman, R.F.W. Pease, High-performance heat sinking for VLSI, IEEE Electron Device Lett. 2 (1981) 126–129.
- [2] A. Ghahremannezhad, K. Vafai, Thermal and hydraulic performance enhancement of microchannel heat sinks utilizing porous substrates, Int. J. Heat Mass Transf. 122 (2018) 1313–1326.
- [3] X.Y. Li, S.L. Wang, X.D. Wang, T.H. Wang, Selected porous-ribs design for performance improvement in double-layered microchannel heat sinks, Int. J. Therm. Sci. 137 (2019) 616–626.
- [4] G. Lu, J. Zhao, L. Lin, X. Wang, W. Yan, A new scheme for reducing pressure drop and thermal resistance simultaneously in microchannel heat sinks with wavy porous fins, Int. J. Heat Mass Transf. 111 (2017) 1071–1078.
- [5] A. Ghahremannezhad, H. Xu, M. Alhuyi Nazari, M. Hossein Ahmadi, K. Vafai, Effect of porous substrates on thermohydraulic performance enhancement of double layer microchannel heat sinks, Int. J. Heat Mass Transf. 131 (2019) 52–63.
- [6] S.L. Wang, X.Y. Li, X.D. Wang, G. Lu, Flow and heat transfer characteristics in double-layered microchannel heat sinks with porous fins, Int. Commun. Heat Mass Transf. 93 (2018) 41–47.
- [7] G. Hetsroni, M. Gurevich, R. Rozenblit, Sintered porous medium heat sink for cooling of high-power mini-devices, Int. J. Heat Fluid Flow. 27 (2006) 259–266.
- [8] A.A. Razali, A. Sadikin, A CFD Simulation Study on Pressure Drop and Velocity across Single Flow Microchannel Heat Sink, J. Adv. Res. Design. 8 (2015) 12-21.
- [9] A.A. Alfaryjat, H.A. Mohammed, N. Mariah, D. Stanciu, A. Dobrovicescu, Numerical investigation of heat transfer enhancement using various nano fluids in hexagonal microchannel heat sink, Therm. Sci. Eng. Prog. 5 (2018) 252–262.
- [10] Y. Zhang, J. Wang, W. Liu, Z. Liu, Heat transfer and pressure drop characteristics of R134a flow boiling in the parallel / tandem microchannel heat sinks, Energy Convers. Manag. 148 (2017) 1082–1095.
- [11] S.B. Abubakar, N.A. Che Sidik, A.S. Ahmad, The use of Fe₃O₄-H₂O₄ Nanofluid for Heat Transfer Enhancement in Rectangular Microchannel Heatsink, J. Adv. Res. in Material Sci. 23(2016) 15-24.
- [12] K. Vafai, L. Zhu, Analysis of two-layered micro-channel heat sink concept in electronic cooling, Int. J. Heat Mass Transf. 42 (1999) 2287–2297.

- [13] J.M. Wu, J.Y. Zhao, K.J. Tseng, Parametric study on the performance of double-layered microchannels heat sink, *Energy Convers. Manag.* 80 (2014) 550–560.
- [14] L. Lin, Y.Y. Chen, X.X. Zhang, X.D. Wang, Optimization of geometry and flow rate distribution for double-layer microchannel heat sink, *Int. J. Therm. Sci.* 78 (2014) 158–168.
- [15] K.-C. Wong, F.N.A. Muezzin, Heat transfer of a parallel flow two-layered microchannel heat sink, *Int. Commun. Heat Mass Transf.* 49 (2013) 136–140.
- [16] K. Wong, M. Ang, Thermal hydraulic performance of a double-layer microchannel heat sink with channel contraction, *Int. Commun. Heat Mass Transf.* 81 (2017) 269–275.
- [17] G. Xie, Y. Liu, B. Sunden, W. Zhang, Computational Study and Optimization of Laminar Heat Transfer and Pressure Loss of Double-Layer Microchannels for Chip Liquid Cooling, *J. Therm. Sci. Eng. Appl.* 5 (2013) 011004.
- [18] X. Wei, Y. Joshi, M.K. Patterson, Experimental and Numerical Study of a Stacked Microchannel Heat Sink for Liquid Cooling of Microelectronic Devices, *J. Heat Transfer.* 129 (2007) 1432.
- [19] K.-C. Wong, J.W. Zuo, H.K. Ng, The thermal performance of Double Layer Microchannel Heat Sink with tapered channel profile In: ICENS-Summer August 23-25, 2017, Sapporo, Japan

TEMPERATURE MINIMIZATION ON THE SUBSTRATE OF A HEAT SINK BY RIB-GROOVE MICROCHANNEL HEAT SINK WITH EFFECTIVE ENERGY CONSUMPTION: GROOVE GEOMETRY PARAMETER EFFECTS

Wan Mohd. Arif Aziz Japar¹

Nor Azwadi Che Sidik²

Yutaka Asako³

Mohd Irwan Mohd Azmi⁴

¹Malaysia – Japan International Institute of Technology (MJIIT), Universiti Teknologi Malaysia Kuala Lumpur, Jalan Sultan Yahya Petra, 54100 Kuala Lumpur, Malaysia

²Malaysia – Japan International Institute of Technology (MJIIT), Universiti Teknologi Malaysia Kuala Lumpur, Jalan Sultan Yahya Petra, 54100 Kuala Lumpur, Malaysia (Email: azwadi@utm.my)

³Malaysia – Japan International Institute of Technology (MJIIT), Universiti Teknologi Malaysia Kuala Lumpur, Jalan Sultan Yahya Petra, 54100 Kuala Lumpur, Malaysia

⁴Malaysia – Japan International Institute of Technology (MJIIT), Universiti Teknologi Malaysia Kuala Lumpur, Jalan Sultan Yahya Petra, 54100 Kuala Lumpur, Malaysia

Abstract: *With the unpredicted increment of power density in Microelectromechanical systems (MEMS) device, an effective cooling technique like micro-cooling method is required to ensure temperature of the device not exceeds or near to the allowable temperature for semiconductor component (Allowable temperature < 358.15 K). Microchannel heat sink is one of the best candidate among micro-cooling method which can meet a cooling demand for current electronic devices. The temperature of a substrate of microchannel heat sink is very important because the substrate directly attaches to the electronic components. By increasing a mass flowrate in a conventional microchannel heat sink (straight channel design) in order to reduce the substrate temperature is not an effective method because it reduces the effective energy consumption. So in this paper, Rib-groove microchannel heat sink design is proposed in order to the minimize the substrate temperature with effective energy consumption. In the present study, the effect of triangular groove geometry parameter was analysed numerically for the Re number ranging between 100 to 800. The result showed that, Rib-groove microchannel heat sink (TC-RR MCHS) TC-RR MCHS with triangular cavities pitch location of 140 μm (CV3) obtained the lowest average substrate temperature, $T_{b,ave}$ for all Re number. Beside that, the proposed design only required 1.84 mW of pumping power consumption, in order to achieve the lowest average substrate temperature of the straight channel microchannel heat sink with the pumping power consumption of 14.94 mW. Means that, the proposed design saved the energy consumption up to 87.8%.*

Keywords: *Rib-groove microchannel heat sink; substrate temperature; pumping power consumption; energy consumption*

Introduction

Nowadays, fast development in electronic industry witnesses many advanced technologies with unpredicted increment power density in their system especially in MEMS devices. As consequence, high heat flux is generated by those devices and thus reduces the performance of its system. Generally, the reliability of those devices might be reduced up to 50% of the original value if the temperature of the devices increase 10° sharply [1]. So, in order to maintain the device performance, high performance cooling techniques are required to improve its cooling

system. Microchannel heat sink is one of the cooling techniques that can meet the current cooling demand whereby the optimum junction temperature for electronic device should be controlled below than 358.15 K [2].

The first concept of microchannel heat sink was pioneered by Tuckerman and Pease [3] in 1981 from Stanford University, who performed the investigation of microchannel performance through experiment. In their study, the microchannel was fabricated by silicon and water was utilized as a fluid transportation. They demonstrated that, the straight channel microchannel heat can remove heat flux up to 790 W/m^2 with the temperature difference between the substrate and water inlet of 71°C . However, high mass flowrate was required to remove the heat flux due to thickened boundary layer in laminar flow. They suggested to used fin structure as a flow disruption in order to increase the thermal performance in the straight channel microchannel heat sink. Since then, many enhancement techniques were proposed in the open literature and can be grouped in two methods, active and passive method. Many scholars are more attracted to the passive method attributes to no additional spacing in cooling system, no external energy is required to improve heat transfer performance and no moving part whereby in contrast to air.

In recent year, many invention and innovation focus on single-layered microchannel heat sink [4-6], multi-layered microchannel heat sink [7, 8] and multi-layered microchannel heat with alternate structure [9]. Besides that, some former researchers are also interested on the utilization of nanofluid in microchannel heat sink application as reported in a comprehensive review paper published by Japar *et al.*, [10]. But, in current study, the investigation of passive method focused on single-layered microchannel heat sink. Flow disruption is one of the effective passive method that can intensively changes the flow structure of laminar by providing “blocking effect” either at the sidewalls or at the centre portion of channel. The “blocking effect” increases the degree of flow mixing between cold and hot water which contributes to the enhancement of heat transfer performance. In 2009, Hong and Cheng tried to explore the potential of offset strip-fin in microchannel design. They proved that, the existing of fins gradually increased the heat transfer performance in microchannel heat sink with excessive pressure drop penalty [11]. High degree of flow fixing also can be obtained by secondary channel geometry without any significant effect on pressure drop penalty. Lee *et al.*, [12] adopted this geometry in their novel microchannel heat sink. The secondary channel was created by copper oblique fins in the microchannel heat sink. They demonstrated that, heat transfer enhancement can be obtained by the secondary channel minimal pressure drop. Next two years, Lee and his team [13] published their other comprehensive study by optimizing the oblique fins geometry parameter which extended form their previous work [12]. The optimization outcome revealed that oblique fin with smaller angle and smaller fin pitch contributed to the higher heat transfer performance enhancement. However, from the view point of thermal management, this method is not very effective as a flow disruption method.

Above studies clearly show that, single passive method such as flow disruption can provide high heat transfer rate in an enhanced microchannel sinks. However, pressure drop issue becomes the bottleneck of further development of single passive method a thermal microchannel heat sink. Nowadays, many single-layered microchannel heat sinks are designed by more than one single passive method which called as hybrid passive method. In hybrid passive method, the balance between strength and weakness of single passive method are considered in microchannel design in order to obtain high heat transfer rate with minimal pressure drop. In 2013, Gong *et al.*, [14] analysed the performance of microchannel that structured by dimple geometry and wavy shape. In their design, the effect of dimple number in wavy channel has been study numerically. The highest Performance Factor of 1.42 was achieved at Re number of 200. They revealed that the presence of the dimple structure in wavy

channel enhanced the heat transfer performance and not apparently increase the flow resistance. Li *et al.*, [15] presented a numerical study to investigate the effect of ribs and cavities on fluid flow and heat transfer characteristic. The analysis revealed that combined effect of interruption, redevelopment of thermal boundary layer, the intensified mainstream disturbance and the chaotic mixing between hot and cold water contributed to enhancement of heat transfer performance. The Performance Factor of this design is higher than the dimple-wavy microchannel heat sink. The highest Performance Factor of this design was 1.62 and achieved at Re number of 500.

The hybrid designs presented in above literatures show that, the remarkable optimum overall performance of the enhanced microchannel heat sink can be obtained at high Re number which contributes to the high pumping power consumption. However, to the best of author's knowledge, only few studies focused on this characteristic. So in the present study, we extended Li *et al.*, [15]'work in order to enhance the heat transfer performance that contributes to the Performance Factor augmentation with effective pumping power consumption compared to conventional design microchannel heat sink (CR MCHS). In this study, three different shapes (CV1, CV2 and CV3) of triangular groove geometry in a hybrid design (TC-RR MCHS) are analysed numerically for the Re number ranging between 100 to 800. The shape of triangular groove geometry depends on the location of triangular groove corner along flow direction. Those three shapes performances are measured based on the average substrate temperature and pumping power consumption.

Geometry parameter of microchannel heat sink

The TC-RR MCHS is fabricated by copper and consist of ten microchannels on the heat sink. All the microchannel has a same geometry structure. In the current study, only one symmetrical part of microchannel was analysed due to computational cost and time. The single wall microchannel of TC-RR MCHS as shown in Figure 1(a) was selected as the symmetrical part and used in all analyses. Figure 1(b) illustrates the geometry parameter of the single wall microchannel that involves in this study. The details of all the parameter is showed in Table 1. The groove length is constant for all analyses. Only L1 and L2 parameters change based on shape design of CV1, CV2 and CV3.

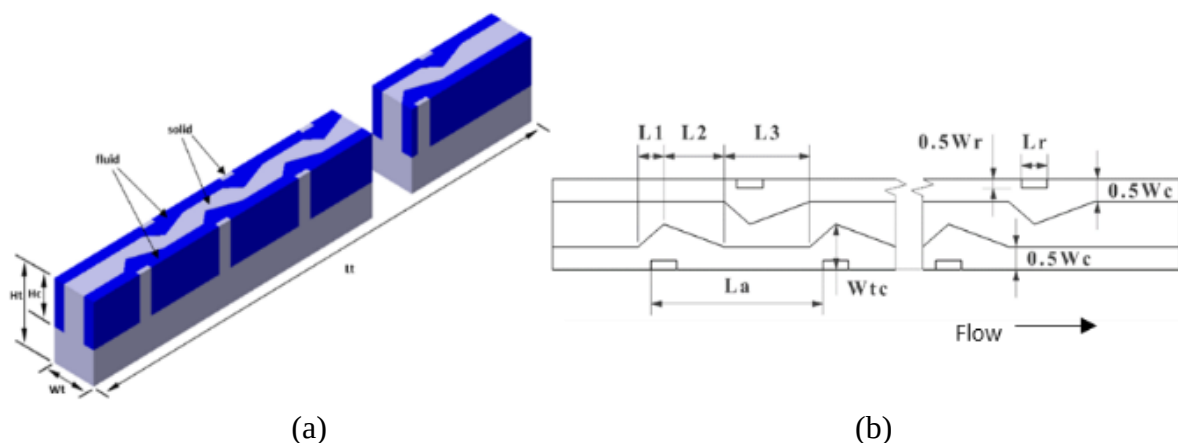


Fig. 1. TC-RR MCHS (a) One symmetrical part (b) Geometry parameter

Table 1: Geometry parameters of TC-RR MCHS

Parameter	Value
Lt	10,000 μm
Wt	200 μm
Ht	350 μm
Hc	200 μm
Wc	100 μm
Lr	60 μm
Wr	30 μm
La	400 μm
CV1:	
	- L1: 60 μm
	- L2: 140 μm
CV2:	
L1 & L2 (Three different shape)	- L1: 100 μm
	- L2: 100 μm
CV3:	
	- L1: 140 μm
	- L2: 60 μm
L3	200 μm
Wtc	112 μm

The shape of those designs are illustrated in Figure 2. The figure shows that, the shape of triangular groove changes with L1 and L2 parameters. The rib geometry is fixed at original location. Figure 2 is a top view of a triangular groove.

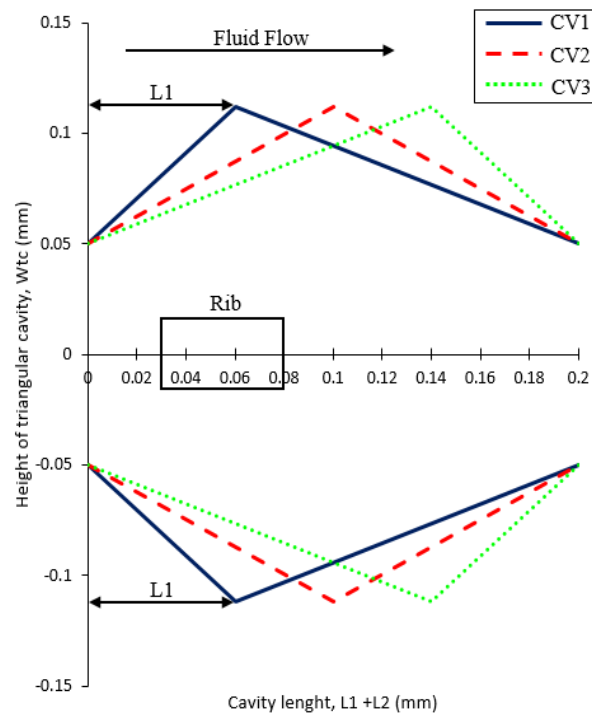


Fig. 2. Triangular cavities with three different shape

Numerical method approach

A Computational Fluid Dynamic (CFD) software such as ANSYS FLUENT 17.0 is used to solve the governing equations in a three dimensional designs according to the following assumptions.

- In the present study, fluid flow in all simulated geometry designs are continuum due to Knudsen number, (Kn) for the fluid flow is less than (10^{-3}) [16].
- For the continuum flow, Navier-Stokes equation and non-slip boundary condition are applicable
- Fluid is Newtonian and incompressible
- Due to microscopic size of channel and to prevent a high pressure drop, laminar flow is applied for all geometry.
- Fluid flow and heat transfer are simulated in steady-state.
- Constant thermophysical properties.
- Viscous dissipation is neglected due to the highest pressure drop in present study is less than 1 MPa with considering the first law of thermodynamics.
- Gravitational force and radiation heat transfer are neglected.

Governing equations

Based on the assumptions that made in the present study, governing equation for conservation of mass, momentum and energy equations can be written as:

Continuity equation:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (1)$$

where u , v and w are the velocity components in x , y and z -directions respectively.

Momentum equation:

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = -\frac{1}{\rho_f} \frac{\partial p}{\partial x} + \frac{\mu_f}{\rho_f} \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) \quad (2)$$

$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} = -\frac{1}{\rho_f} \frac{\partial p}{\partial y} + \frac{\mu_f}{\rho_f} \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) \quad (3)$$

$$u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} = -\frac{1}{\rho_f} \frac{\partial p}{\partial z} + \frac{\mu_f}{\rho_f} \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) \quad (4)$$

where ρ_f and μ_f are the density and dynamic viscosity of the working fluid(water), respectively, and p is the fluid(water) pressure.

There have two energy equations that related to the present study, namely, energy equation for fluid region (Eq. (5)) and energy equation for solid region (Eq. (6)):

$$u \frac{\partial T_f}{\partial x} + v \frac{\partial T_f}{\partial y} + w \frac{\partial T_f}{\partial z} = \frac{k_f}{\rho_f c_{pf}} \left(\frac{\partial^2 T_f}{\partial x^2} + \frac{\partial^2 T_f}{\partial y^2} + \frac{\partial^2 T_f}{\partial z^2} \right) \quad (5)$$

$$0 = k_s \left(\frac{\partial^2 T_s}{\partial x^2} + \frac{\partial^2 T_s}{\partial y^2} + \frac{\partial^2 T_s}{\partial z^2} \right) \quad (6)$$

where T_f , T_s , k_f , k_s and C_{pf} are the fluid's temperature, solid's temperature, fluid thermal conductivity, solid thermal conductivity and fluid specific heat, respectively.

Boundary Condition

Boundary condition is a condition for hydrodynamic and thermal that we applied on the simulated geometries in the present study. Uniform velocity with the temperature of 300 K is applied on the inlet channel for all cases. While at the outlet, we set as atmospheric pressure. A uniform heat flux of 100W/cm² is applied on the substrate of the heat sink. At the fluid-solid interface, no-slip and no penetration are assumed. Table 2 shows the details for other boundary conditions.

Table 2: Boundary condition

Boundary	Location	Condition
Hydrodynamic		No-slip and no penetration $u = v = w = 0$
	At the fluid-solid interface	$-k_s \left(\frac{\partial T_s}{\partial n} \right) = -k_f \left(\frac{\partial T_f}{\partial n} \right)$ where n is the coordinate normal to the wall
	At inlet, $x = 0$	$u_f = u_{in}$ $v = w = 0$
	At outlet, $x = L_t = 10 \text{ mm}$	$p_f = p_{out} = 1 \text{ atm}$
	At inlet, $x = 0$	$T_f = T_{in} = 300 \text{ K}$ (for water) $-k_s \left(\frac{\partial T_s}{\partial x} \right) = 0$ (for solid)
	At outlet, $x = L_t = 10 \text{ mm}$	$-k_f \left(\frac{\partial T_f}{\partial x} \right) = 0$ (for water) $-k_s \left(\frac{\partial T_s}{\partial x} \right) = 0$ (for solid)
Thermal	At top wall, $z = Ht = 0.35 \text{ mm}$	$u = v = w = 0$ $-k_s \left(\frac{\partial T_s}{\partial z} \right) = 0$
	At bottom wall, $z = 0$	$-k_s \left(\frac{\partial T_s}{\partial z} \right) = q = 100 \text{ W / cm}^2$
	At side wall, $y = 0$	$\frac{\partial}{\partial y} = 0$ (symmetry)
	At side wall, $y = Wt = 0.2 \text{ mm}$	$\frac{\partial}{\partial y} = 0$ (symmetry)

Grid independency test and CFD simulation

Mesh quality is very important to the convergence of numerical solution and numerical computation stability. In this paper, ANSYS is used to generate hexahedral meshing structure which can provide a faster solution time. Figure 3 illustrates the hexahedral mesh structure in TC-RR MCHS design. Finite volume method is utilized to discretize the governing equation. The SIMPLE algorithm was adopted to accomplish the pressure-velocity coupling. Furthermore, convergence criteria are set to be less than 10⁻⁶ for continuity and less than 10⁻⁹ for energy.

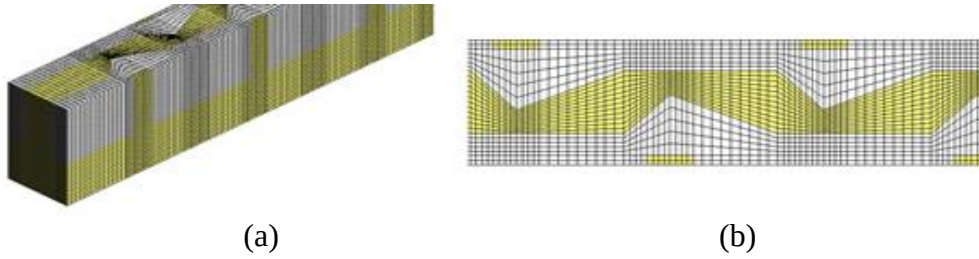


Fig. 3. Computational grid of TC-RR MCHS (a) Isometric view (b) Top view at x-z plane

Grid independency test is a process to find the optimum mesh structure that can be adopted in TC-RR MCHS design, so that the design can produce an accurate result with the lower time cost. Besides that, the test was conducted in order to ensure the simulation results are not affected by the grid size. The process is started with a fine mesh structure to a coarse mesh structure in a base design (CR MCHS). In the present study, the total number of element or grid for the finest mesh is 0.5 million for Reynolds number of 800. With the same Reynolds number, the number of element is reduced up to 0.15 million. By considering Nusselt number and pressure drop for each element number, the optimum mesh structure is obtained by calculating the relative error using following equation:

$$e\% = \left| \frac{J_2 - J_1}{J_1} \right| \times 100 \quad (7)$$

where J_1 represents the value of Nusselt number and pressure drop for the finest mesh structure while J_2 represents the value of Nusselt number and pressure drop for the coarser mesh. Based on the relative error presented in Table 3 for each element number, it clearly observed that element number of 0.3 million has a reasonable accuracy compared with other element numbers. Consequently, the number element of 0.3 million is adopted for all cases in current study.

Table 3: Grid independence test

Grid number (X10 ⁵)	Nu	e%	Pressure drop (Pa)	e%
5.0	8.07	-	64599.66	-
4.0	8.07	0	64574.90	0.038
3.5	8.07	0	64557.65	0.065
3.0	8.07	0	64535.46	0.099
2.5	8.07	0	64507.85	0.142
2.0	8.07	0	64480.42	0.185
1.5	8.07	0	64465.21	0.200

Data reduction

This section presents the relevant expressions that used to calculate the characteristics of heat transfer and fluid flow in microchannel heat sink.

The Reynolds number (Re) is expressed as follow:

$$Re = \frac{\rho u_m D_h}{\mu} \quad (8)$$

where D_h represents the hydraulic diameter and it is calculated as follow:

$$D_h = \frac{2H_c W_c}{H_c + W_c} \quad (9)$$

The average apparent friction factor and pumping power are given by:

$$f_{app,ave} = \frac{2D_h \Delta P}{L_t \rho u_m^2} \quad (10)$$

$$P_p = \dot{V} \Delta P \quad (11)$$

where L_t and ΔP are the total length of microchannel and pressure drop across microchannel respectively. The average heat transfer coefficient and the average Nusselt number are given by:

$$h_{ave} = \frac{q_w A_{film}}{A_{con.} (T_{w,ave} - T_{f,ave})} \quad (12)$$

$$Nu_{ave} = \frac{h_{ave} D_h}{k_f} \quad (13)$$

where q_w , A_{film} , A_{cond} , $T_{w,ave}$ and $T_{f,ave}$ are the heat flux per unit area, heated area, convection heat transfer area, average temperature of wall and average temperature of fluid, respectively.

Result and discussion

In order to verify the accuracy of the simulation model approach, data that obtained from simulation analysis, namely, friction factor and pressure drop for the base design (CR MCHS) are validated with previous correlations developed by former researchers. The friction factor is validated with Shah & London correlation [17] :

$$f_{app,ave} Re = \left[\left(\frac{3.2}{L_t / Re D_h} \right)^2 + (Po)^2 \right]^{\frac{1}{2}} \quad (14)$$

Poiseuille number (Po):

$$P_o = 96[1 - 1.3553 (AR) + 1.946 (AR)^2 - 1.7012 (AR)^3 + 0.9564 (AR)^4 - 0.2537 (AR)^5] \quad (15)$$

Pressure drop is validated with Steinke & Kandlikar [18]:

$$\Delta p = \frac{2(f Re) \mu_f u_m L_t}{D_h^2} + \frac{K \rho_f u_m^2}{2} \quad (16)$$

$$f Re = 24(1 - 1.3553 \alpha_c + 1.9467 \alpha_c^2 + 1.7012 \alpha_c^3 + 0.9564 \alpha_c^4 + 0.2537 \alpha_c^5) \quad (17)$$

$$K = 0.6797 + 1.2197 \alpha_c + 3.3089 \alpha_c^2 - 9.5921 \alpha_c^3 + 8.9089 \alpha_c^4 - 2.9959 \alpha_c^5 \quad (18)$$

where α_c is channel aspect ratio of width to height and K is Hagenbach's factor.

Validation of the simulation model with the previous correlations are illustrated in Figure 4. It clearly shows the simulation result has a good agreement with both correlations. This agreement indicates that the simulation model approach can be referred to predict the fluid flow and heat transfer characteristic in present study for all designs.

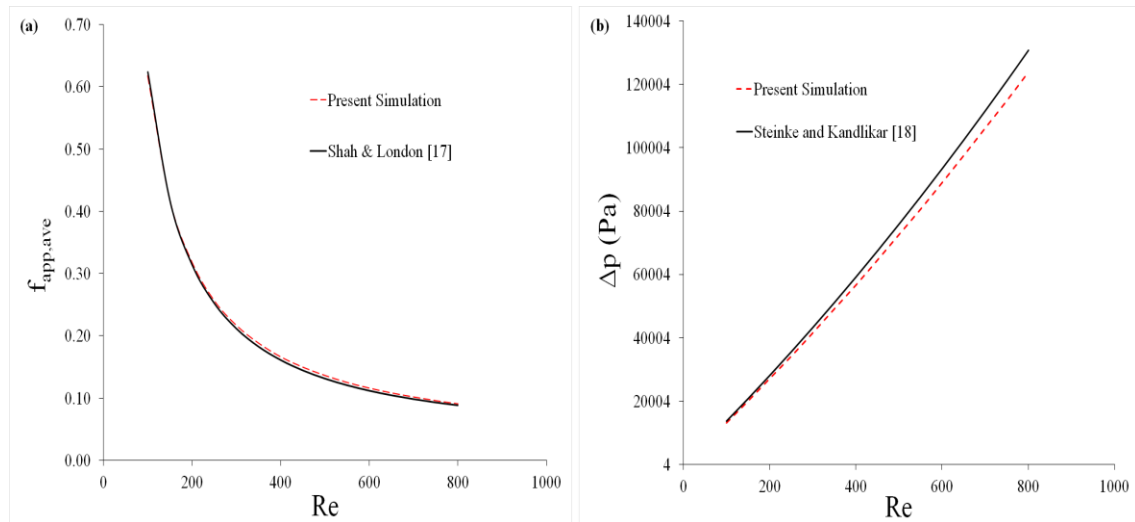


Fig. 4. Model validation (a) Friction factor according to Shah & London [17] (b) Pressure drop according to Steinke & Kandlikar [18].

As mentioned in above background study, the substrate temperature of microchannel heat sink is very important because it directly attaches to electronic components. So in this section, the performance of each design is evaluated based on the substrate temperature of microchannel heat sink. Figure 5 illustrates the substrate temperature of base design (CR MCHS) and enhanced designs (CV1, CV2 and CV3) from the bottom view with unshaded surface at Re number of 350. The figure shows CR MCHS obtain the highest substrate temperature distribution compared to the enhanced designs. The highest substrate temperature, $T_{b,max}$ of the CR MCHS is 326.22 K. It happens due to the boundary layer in laminar region is so thick and thus increases the thermal resistance in the fluid region. For the enhanced designs, TC-RR MCHS with CV3 design shows the lowest substrate temperature distribution than CV1 and CV2 designs. It is proven by the temperature distribution at the channel length, L_t of 5.1 mm. At that location, the substrate temperature distribution of CV3 is around 309.5 K compared to CV1 and CV2 with substrate temperature distribution of 310 K. Besides that, CV3 design reduces the maximum substrate temperate of CR MCHS by 3.8% compared to CV1 and CV2 that only by 3.6% for both designs. Means that, triangular groove geometry parameters have a significant effect on thermal performance.

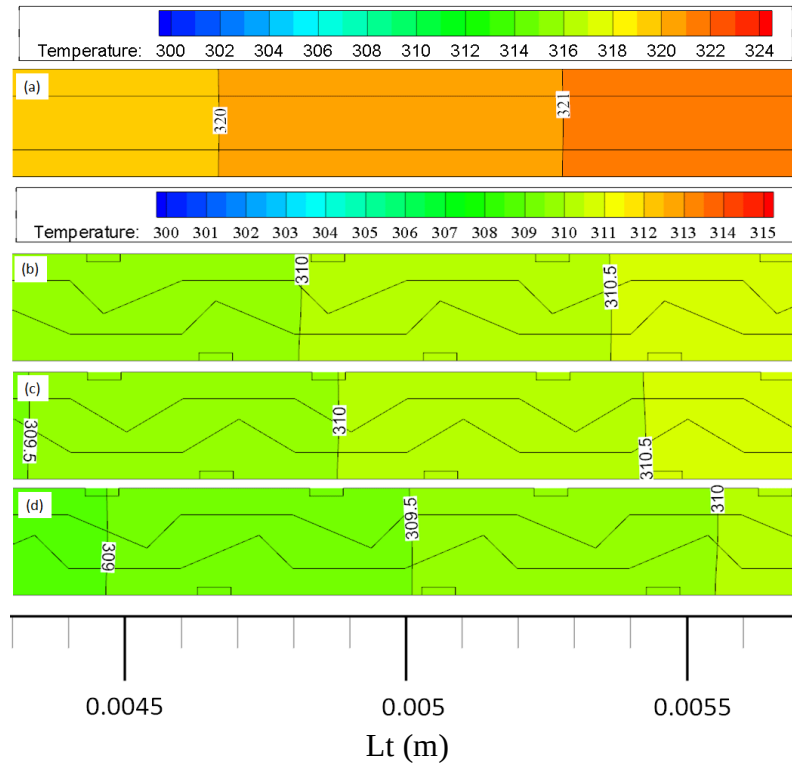


Fig. 5. Substrate temperature of microchannel sink from the bottom view with unshaded surface at $Re=350$ (a) CR MCHS with $T_{b,max}=326.22$ K (b) CV1 design with $T_{b,max}=314.60$ K (c) CV2 design with $T_{b,max}=314.47$ K (d) CV3 design with $T_{b,max}=313.81$ K

Figure 6 demonstrates the average substrate temperature, $T_{b,ave}$ of CR MCHS and TC-RR MCHS with different triangular groove shape (CV1, CV2 and CV3) for Re number of 100 to 800. The figure shows that, when the Re number increases, the average substrate temperature, $T_{b,ave}$ of all designs decreases sharply until a certain Re number and then decreases slowly until $Re = 800$. As expected, CR MCHS obtain the highest average substrate temperature, $T_{b,ave}$ compared to the enhanced designs due to thermal resistance created by laminar region. For enhanced designs, the average substrate temperature, $T_{b,ave}$ of CV1, CV2 and CV3 are quite similar at the lower Re number. But then, when the Re number increases further, the average substrate temperature, $T_{b,ave}$ of CV3, decreases significantly compared to CV1 and CV2 designs. However, from the viewpoint of energy saving, achieves high thermal performance with minimal pumping power consumption is good for enhanced microchannel heat sink in order to replace the CR MCHS as a cooling device.

To achieve the cooling requirement with effective energy consumption, the analysis between the average substrate temperature, $T_{b,ave}$ and pumping power consumption, P is conducted as shown in Figure 7. The figure presented that, when the mass flowrate of fluid increased, the average substrate temperature decreased while the pumping power consumption increased. As illustrated in the figure, CR MCHS requires 12.04×10^{-5} kg/s mass flowrate to achieve the lowest average substrate temperature of 314.1 K which consumes 14.94 mWatt pumping power. However, TC-RR MCHS with CV3 design only requires 3.01×10^{-5} kg/s mass flowrate to achieve the average substrate temperature of 314.1 K same as the lowest average substrate temperature of CR MCHS. Means that, the TC-RR MCHS with CV3 design saves

the energy consumption up to 87.8%, because only consumes 1.82 mWatt pumping power in order to pump the mass flowrate of 3.01×10^{-5} kg/s.

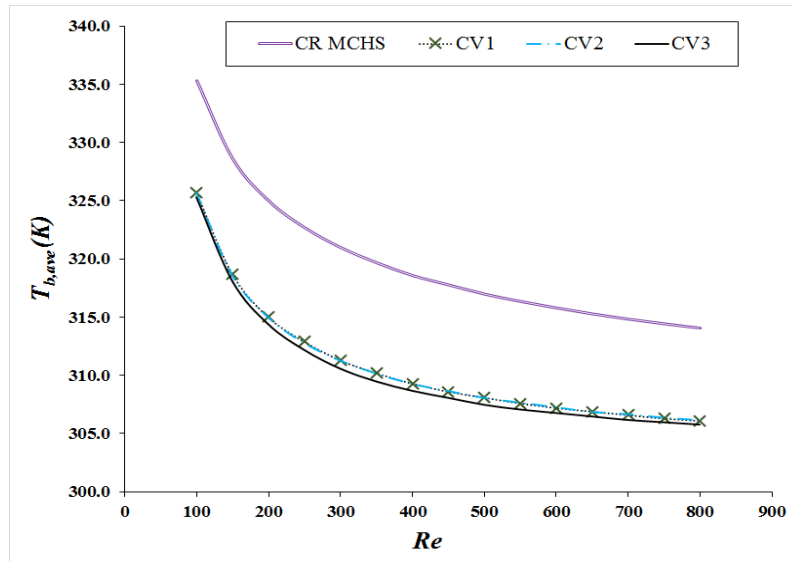


Fig. 6. Average substrate temperature, $T_{b,ave}$ of CR MCHS and enhanced MCHS

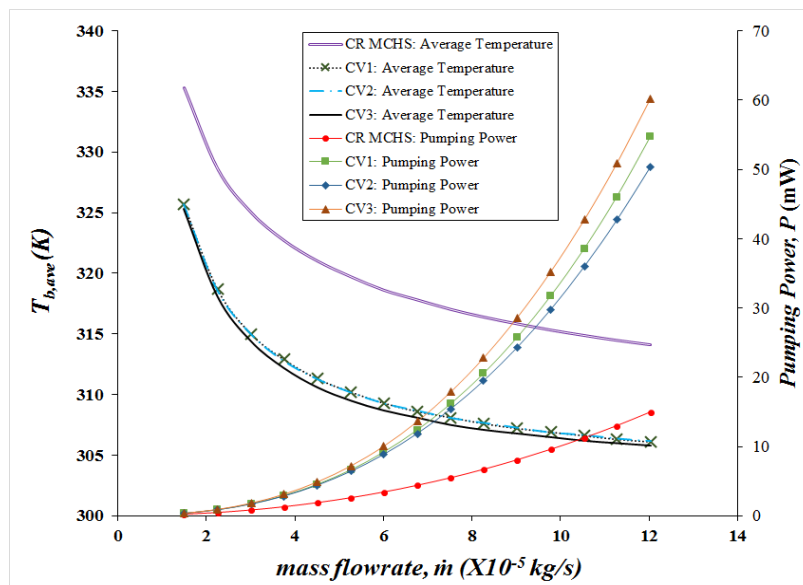


Fig. 7. Average substrate temperature, $T_{b,ave}$ and pumping power of CR MCHS and enhanced MCHS

Conclusion

In this study, TC-RR MCHS with three different triangular grooves (CV1, CV2 and CV3) were investigated numerically in order to optimize the heat transfer performance with minimal pressure drop compared to the base design or conventional microchannel heat sink (CR MCHS) for Re number of 100 to 800. The study showed that TC-RR MCHS with CV3 design provided the lowest average substrate temperature and saved energy consumption up to 87.8% compared to the conventional design (CR MCHS). Means that, this design can be used in an application which not requires a high energy consumption for their cooling system.

Acknowledgment and Funding

Authors wish to thanks Universiti Teknologi Malaysia for supporting this research activity by the Takasago grant [R.K130000.7343.4B365]

Reference

- [1] Li, Yanlong, Fengli Zhang, Bengt Sunden and Gongnan Xie. "Laminar thermal performance of microchannel heat sinks with constructal vertical Y-shaped bifurcation plates." *Applied Thermal Engineering* 73, no. 1 (2014): 185-195.
- [2] Yan, Yunfei, Hongyu Yan, Siyou Yin, Li Zhang and Lixian Li. "Single/multi-objective optimizations on hydraulic and thermal management in micro-channel heat sink with bionic Y-shaped fractal network by genetic algorithm coupled with numerical simulation." *International Journal of Heat and Mass Transfer* 129 (2019): 468-479.
- [3] Tuckerman, David B and Roger Fabian W Pease. "High-performance heat sinking for VLSI." *IEEE Electron device letters* 2, no. 5 (1981): 126-129.
- [4] Japar, Wan Mohd Arif Aziz, Nor Azwadi Che Sidik and Shabudin Mat. "A comprehensive study on heat transfer enhancement in microchannel heat sink with secondary channel." *International Communications in Heat and Mass Transfer* 99 (2018): 62-81.
- [5] Japar, Wmaa and N. A. C. Sidik. *The effectiveness of secondary channel on the performance of hybrid microchannel heat sink at low pumping power*. in *Materials Science and Engineering*. 2019. IOP Publishing.
- [6] Beng, Soo Weng and Wan Mohd Arif Aziz Japar. "Numerical analysis of heat and fluid flow in microchannel heat sink with triangular cavities." *Journal of Advanced research in fluid mechanics and thermal sciences* 34 (2017): 1-8.
- [7] Chamoli, Sunil, Ruixin Lu, Hao Chen, Yongpan Cheng and Peng Yu. "Numerical optimization of design parameters for a modified double-layer microchannel heat sink." *International Journal of Heat and Mass Transfer* 138 (2019): 373-389.
- [8] Zuo, J. W., K. C. Wong and H. K. Ng. "The thermal performance of three-layered microchannel heat sink with tapered channel profile." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 56, no. 1 (2019): 147-156.
- [9] Shen, Han, Gongnan Xie and Chi-Chuan Wang. "Heat transfer and thermodynamic analysis by introducing multiple alternation structures into double-layer microchannel heat sinks." *International Journal of Thermal Sciences* 145 (2019): 105975.
- [10] Japar, Wan Mohd Arif Aziz, Nor Azwadi Che Sidik, Siti Rahmah Aid, Yutaka Asako and Tan Lit Ken. "A Comprehensive Review on Numerical and Experimental Study of Nanofluid Performance in Microchannel Heatsink (MCHS)." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 45, no. 1 (2018): 165 – 176.
- [11] Hong, Fangjun and Ping Cheng. "Three dimensional numerical analyses and optimization of offset strip-fin microchannel heat sinks." *International Communications in Heat and Mass Transfer* 36, no. 7 (2009): 651-656.
- [12] Lee, YJ, PS Lee and SK Chou. "Enhanced thermal transport in microchannel using oblique fins." *Journal of Heat Transfer* 134, no. 10 (2012): 101901.
- [13] Lee, Yong Jiun, Pawan K Singh and Poh Seng Lee. "Fluid flow and heat transfer investigations on enhanced microchannel heat sink using oblique fins with parametric study." *International Journal of Heat and Mass Transfer* 81 (2015): 325-336.
- [14] Gong, Liang and Bo Wei. *The Characteristics of Fluid Flow and Heat Transfer in Wavy, Dimple and Wavy-Dimple Microchannels*. in *Applied Mechanics and Materials*. 2013. Trans Tech Publ.

- [15] Li, YF, GD Xia, DD Ma, YT Jia and J Wang. "Characteristics of laminar flow and heat transfer in microchannel heat sink with triangular cavities and rectangular ribs." *International Journal of Heat and Mass Transfer* 98 (2016): 17-28.
- [16] Service, Robert F. "Coming soon: the pocket DNA sequencer." *Science (New York, NY)* 282, no. 5388 (1998): 399.
- [17] Shah, Ramesh K and Alexander Louis London, *Laminar flow forced convection heat transfer and flow friction in straight and curved ducts-A summary of analytical solutions*. 1971, Stanford Univ CA Dept of Mechanical Engineering.
- [18] Steinke, Mark E and Satish G Kandlikar. "Single-phase liquid friction factors in microchannels." *International Journal of Thermal Sciences* 11, no. 45 (2006): 1073-1083.

A STUDY ON 2-BUTANOL DROPLET: ANALYSIS OF INTRODUCED CONSTANT IN THE NEW APPROACH ON FILM STAGNANT MODEL

E. A. Kosasih¹
H. Tandjono²
D. Prasetyo³

¹Department of Mechanical Engineering, Mechanical Engineering, Faculty of Engineering, University of Indonesia, 16424 Depok, Jawa Barat, Indonesia (Email: kossri@eng.ui.ac.id)

²Department of Mechanical Engineering, Mechanical Engineering, Faculty of Engineering, University of Indonesia, 16424 Depok, Jawa Barat, Indonesia

³Department of Mechanical Engineering, Mechanical Engineering, Faculty of Engineering, University of Indonesia, 16424 Depok, Jawa Barat, Indonesia

Abstract: In many different application, non-stationary liquid droplet evaporation phenomenon works as a fundamental. Therefore, an accurate prediction of heat and mass transfer on non-static droplet is required. Ranz-Marshall analogy combined with Stagnant film Model is one of the most used method on predicting single non-stationary high mass transfer rate droplet evaporation, but there are several problems regarding to the method such as Lewis number more than one, static film width, and unsimultaneously calculate heat and mass transfer. In this paper, A new approach to Stagnant Film Model is briefly explained that the stagnant film concentration isn't constant. Due to that reason, a new constant of C1 and C2 appeared as a result of integration. The equation is then simulated and compared to experiment and the standard Stagnant Film Model by using 2-butanol substance. The new Approach deviated less when predicting radius while both still failed to predict change in temperature. The appropriate value of C1 on predicting the right radius varied between -0.010 to -0.035 while the value of C2 is -4×10^{-8} . Since it is different for every set-up variables, we can sum up that the value of C1 is a function and which used to predict temperature might also be a different number since it is still failed to predict temperature and C2 is affected by C1.

Keywords: Droplet, Non-stationary, Heat and mass transfer model, 2-Butanol, A new approach to Stagnant Film Model.

Introduction

The study of droplet heat and mass transfer is beneficial to the development of technology as it is widely used in many variety of application. One of the most widely researched branch of droplet is the evaporation of fuel in many liquid fueled propulsion system [1-2]. Not only fuel, scholars also applied droplet study on cooling such as evaporative cooling for microprocessors [3] and cooling tower [4]. Droplet evaporation were also discussed in sprays for many application outside of fuel such as vacuum spray cooling [5] and spray drying for food powder [6]. Droplet science also play a huge part on daily life technology for instance, inkjet printing technology [7], surface coating [8], oil lubrication [9], and even medical means such as blood evaporation rate [10].

Droplet evaporation phenomenon is caused by heat and mass transfer. Heat and mass transfer coefficient used both for design and simulation is calculated by using the Ranz-Marshall equation [11]. This analogy required a Lewis (Le) number of one, which guarantees the similarity of temperature profiles and concentration profiles [12]. Kosasih et. al. [13-14] performed a study regarding this matter and found out that Lewis number on various droplet of

substances is more than one. The analogy model becomes less appropriate to use. The results of Chen's as well as M. I. Alhamid and E. A. Kosasih's study [15] also show that the value of heat and mass transfer coefficient deviant from the results of conventional calculations (Ranz-Marshall analogies). Besides that, there are six conditions for the analogy which one of them is low mass transfer rate, therefore the analogy is combined with another analytical model for heat and mass transfer [16].

Ranz-Marshall equation above is used in three analytical model for high rate of mass transfer which is stagnant film model, penetration model, and flat plate model. Kosasih [15] shows that stagnant film model inaccurately predict the heat and mass transfer of a droplet due to static film width and unsimultaneously calculate heat and mass transfer.

Due to those problems, a new approach to Stagnant Film Model was created by Kosasih and the correlation of Sherwood and Nusselt Number is indeed better [17]. Multiple constant were added as a new figure of correction due to integration factor. Study shows that the value of constant differs for each substances [18]. In this paper, validation of the new approach to Stagnant Film Model and its constants for 2-butanol will be discussed throughoutly to investigate the value of constant related.

The aim of this study are:

1. To validate the new approach to Stagnant Film Model on 2-butanol droplet heat and mass transfer.
2. To validate the value of constant (C_1 and C_2) by comparing numerical simulation to experiment.

Methodology

Mathematical Formulation

The mathematical formulation of the new approach to Stagnant Film Model by Kosasih [14] will be discussed in this chapter. As the air flows through the droplet, heat and mass transfer phenomenon occur. Reynold Number (Re) defines the characteristic of the flow which can be expressed as,

$$Re = \frac{\rho v d}{\mu} \quad (1)$$

where ρ , v , d , and μ are density, speed, contact diameter, and viscosity respectively. Ranz-Marshall analogy were used as a baseline non-dimensional number for heat and mass transfer carried by the flow.

$$Nu = 2 + 0.6Re^{1/2}Pr^{1/3} \quad (2)$$

$$Pr = \frac{\mu C_p}{K} \quad (3)$$

For heat transfer applied as Nusselt Number (Nu). Prantzel number (Pr) is expressed by eq. (3) where C_p and K are specific heat capacity and thermal conductivity respectively.

$$Sh = 2 + 0.6Re^{1/2}Sc^{1/3} \quad (4)$$

$$Sc = \frac{\mu \rho}{d} \quad (5)$$

Analogically, for mass transfer applied Sherwood Number (Sh). Schmidt Number (Sc) is expressed by equation (4). These formulas are valid for Reynold numbers below 200. The coefficient rate of heat and mass transfer is next determined by calculating k_L (mass transfer coefficient) and H_L (heat transfer coefficient),

$$k_{cL} = \frac{NuK}{d} \quad (6)$$

$$h_L = \frac{ShX_{AB}}{d} \quad (7)$$

where X is mole fraction. Stagnant film is an imaginary film without potential gradient outside of the film. The total mass transfer rate of material A, N_{A0} , defined as,

$$\exp\left(\frac{N_{A0}\delta_c}{C.D_{AB}}\right) = 1 + \frac{X_{A0} - X_{A\infty}}{1 - X_{A0}} \quad (8)$$

where δ_c thickness of concentration film, C as total concentration of A and B, and D_{AB} as mass diffusivity.

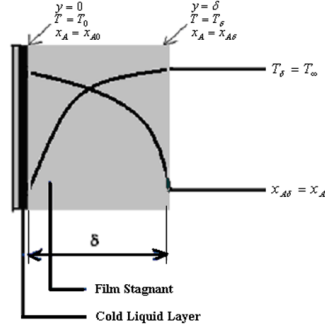


Fig 1. Illustration of Film Stagnant Model [14].

N_{A0} can also be defined as,

$$N_{A0} = k_c(X_{A0} - X_{A\infty})/(1 - X_{A0}) \quad (9)$$

For $X_\delta = X_\infty$ applied,

$$\exp\left(\frac{N_{A0}\delta_c}{C.D_{AB}}\right) = 1 + \frac{N_{A0}}{k_c} \quad (10)$$

and when N_{A0} approaching 0 on k_c , k_c approach k_{cL} (k_c for low mass transfer),

$$k_{cL} = \lim_{N_{A0} \rightarrow 0} k = \lim_{N_{A0} \rightarrow 0} \frac{N_{A0}}{\exp\left(\frac{N_{A0}\delta_c}{C.D_{AB}}\right) - 1} \quad (11)$$

and thus, without setting the thickness of concentration film (δ_c) as constant, the limit of eq. (9) can be solved and integrated as a solution which resulted as,

$$N_{A0} = k_{cL} \left[\ln\left(\frac{1 - X_{A\infty}}{1 - X_{A0}}\right) - C1 \right] \quad (12)$$

which in the original stagnant film model $C1$ is zero as a result of constant δ_c . Inside the film, the total heat transfer can be defined as q_0 .

$$q_0 = \frac{N_{A0} \bar{C}_{pA} (T_\delta - T_0)}{\exp\left(-N_{A0} \bar{C}_{pA} \delta_T / k\right) - 1} \quad (13)$$

$$q_0 = h(T_0 - T_\infty) \quad (14)$$

Since outside of the stagnant film doesn't applied potential gradient, eq. (11) and (12) can also be defined as,

$$\exp\left(-\frac{N_{A0} \bar{C}_{pA} \delta_T}{k}\right) = 1 - \frac{N_{A0} \bar{C}_{pA}}{h} \quad (15)$$

when N_{A0} approaching 0 on h , h approach h_L (h for low mass transfer),

$$h_L = \lim_{N_{A0} \rightarrow 0} h = \lim_{N_{A0} \rightarrow 0} \frac{N_{A0} \bar{C}_{pA}}{-\exp\left(-\frac{N_{A0} \bar{C}_{pA} \delta_T}{k}\right) + 1} \quad (16)$$

and thus, without thickness of concentration film (δ_c) as constant, the limit of eq. (14) can be solved and integrated as a solution which resulted as the total heat transfer rate (q_0),

$$q_0 + q_{cond} = \frac{N_0 C_P (T_\infty - T_0)}{\exp\left(-\frac{N_0 C_P}{h_L} - \frac{C_2 C_P}{k_L}\right) - 1} \quad (17)$$

where T is the temperature and q_{cond} as heat conducted by the thermocouple which is equal to zero on natural phenomenon. Since we have the value of N_0 and q_0 , variation of droplet radius and temperature can be expressed as,

$$N_0 = -\left(\frac{\rho}{M}\right) \frac{dr}{dt} \quad (18)$$

$$\frac{dT}{dt} = 3 \left(\frac{q_0 - N_0 h_{fg}}{N_0 r C p_A} \right) \quad (19)$$

with M as Mole of droplet and h_{fg} as saturated pressure. $C1$ and $C2$ then will be varied to overlook the best value of constant for the modified stagnant film model.

Experiment apparatus

Experiment data were also obtained to validate the compatibility between the modified stagnant film model and the constants.



Fig. 2: Data retrieval scheme.

With the same apparatus as schemed in Fig. 2, 2-butanol radius and temperature were recorded by picture and thermocouple respectively every 5 seconds through total four different combination of circumstances of varying air flow speed and air temperature. Thermocouple directly gave the temperature information to the user, while the radius is calculated by calibrating droplet pictures to a measuring instrument and then size will be counted by pixels. To note with, the droplet starting temperature is affected by the starting temperature value which caused the surrounding temperature to change.

Table 1. Temperature and Speed variation on airflow.

Temperature Variation	Speed Variation
35°C	1 m/s
55°C	7 m/s

Camera captured droplet's photo and then the size of droplet's diameter were calculated by pixels. The air also set to only affect the droplet by insulating thermocouple outside the droplet.

Results

Both simulation and experiment were then executed and plotted into Fig. 3 to 10. Fig. 3 to 6 shows the change of square radius by time while Fig 6. To 10 shows the change of temperature by time. Four series of plot can be found on the graphs, which are Experiment data, Modified Stagnant Film Model with the most appropriate $C1$ value ($C1$), Modified Stagnant Film Model with average $C1$ value of those 4 most appropriate value ($C1^*$), and the Standard Stagnant Film

Model (SFM). While the value of $C1$ is varied, $C2$ was set as constant value of -4×10^{-8} as the other value causes the simulation to corrupt.

It is shows that by Fig. 3 to 6, that the predicted value of radius square calculated by the Standard Film Stagnant Model deviate unconsistently from the experiment data, instead the Modified Stagnant Film Model deviate less to the experiment data. Although, the value of $C1$ is not constant and therefore is a function of something shown by having an average $C1$ of the most appropriate value of them four ($C1^*$) isn't constantly remove the error of the Stagnant Film Model. The value of $C1$ and $C1^*$ is stated at Table. 2.

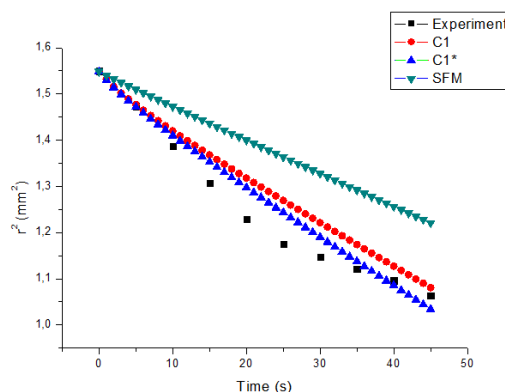


Fig. 3: 2-butanol change in radius square by time (35°C 1 m/s airflow).

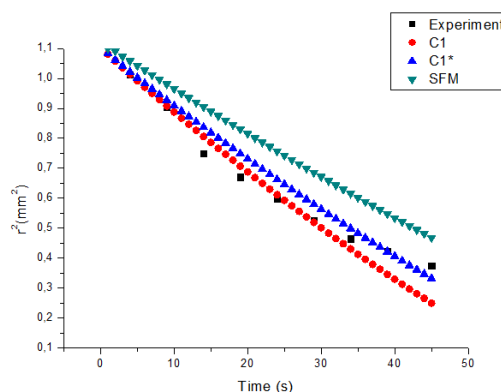


Fig. 4: 2-butanol change in radius square by time (35°C 7 m/s airflow).

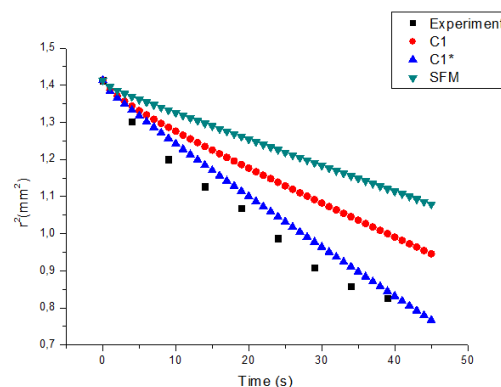


Fig. 5: 2-butanol change in radius square by time (35°C 7 m/s airflow).

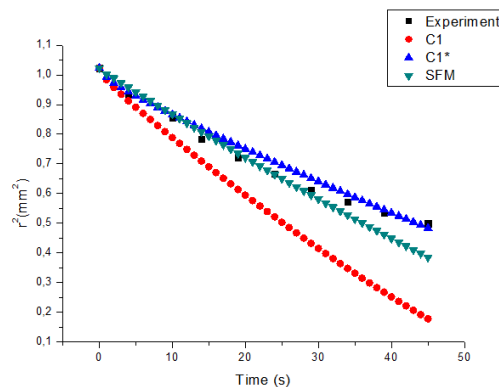


Fig. 6: 2-butanol change in radius square by time (55°C 7 m/s airflow).

Table 2. Aproprite Value of C1 Constant for each experiment and the average of C1 (C1*).

C1	0.1 m/s	0.7 m/s
35°C	-0.025	-0.013
55°C	-0.032	-0.02
C1*	0.225	

On the other hand, the result on temperature prediction on 2-butanol isnt satisfying enough. There are big deviation on both simulation on modified and original Stagnant Film Model which is shows in Fig. 7 to 10. The deviation on temperature for both simulation is unacceptable which is also discussed by Wang et. al. [18] on multiple substances. Heat transfer is affected by q_0 which is a function of the constants therefore, the model still needs to be revise regarding the value of C1 and C2.

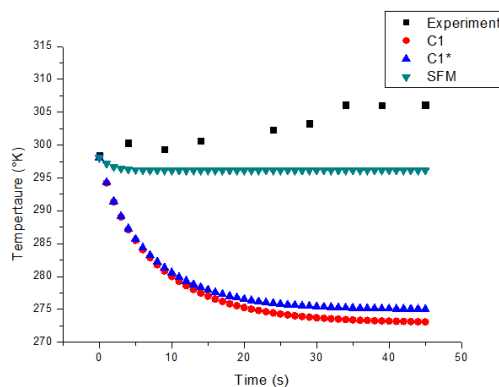


Fig. 7: 2-butanol change in temperature by time (35°C 1 m/s airflow).

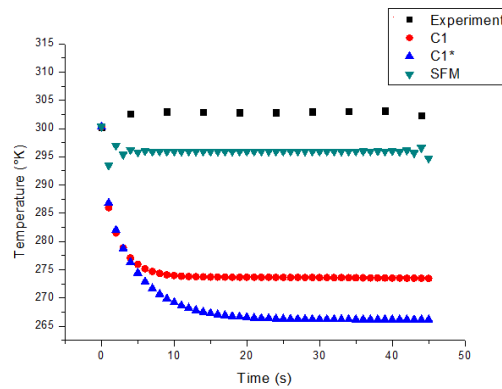


Fig. 8: 2-butanol change in temperature by time (35°C 7 m/s airflow).

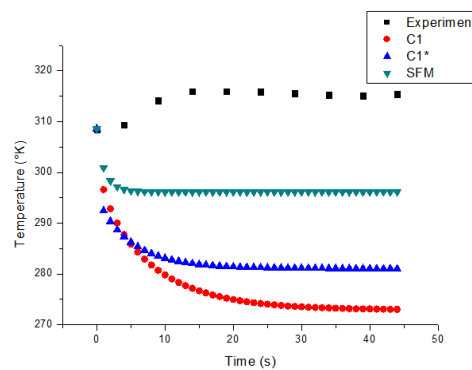


Fig. 5: 2-butanol change in radius square by time (55°C 1 m/s airflow).

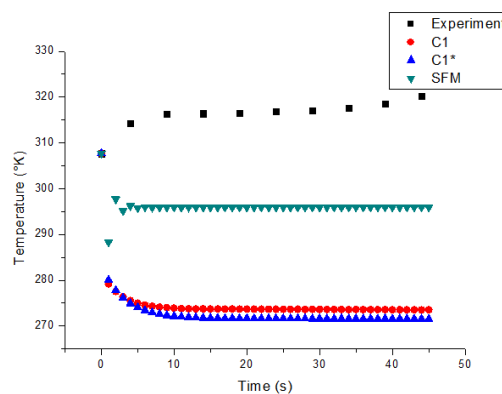


Fig. 10: 2-butanol change in temperature by time (55°C 7 m/s airflow).

Conclusions

The study concluded three point out of this paper:

1. The value of $C1$ and $C2$ which is a value of integration in the Stagnant Film Model is not zero, therefore the width of stagnant film concentration, δ_c , isn't constant.
2. The value of $C1$ in the New Approach to Stagnant Film Model caused less deviation on predicting the change in radius but since $C1$ isn't constant for different variables, then it is a function of something that need to be researched.
3. Both simulations failed to predict the change in temperature. On that case, more research on the value of $C2$ which affect heat transfer rate might correct the deviation.

Acknowledgement

This work was supported by The Ministry of Research, Technology, and Higher Education of The Republic of Indonesia (No. NKB-1634/UN2.R3.1/HKP.05.00/2019)

Nomenclature

Re	Reynold number (–)
v	Speed (m s^{-1})
d	Diameter (m)
T	Temperature ($^{\circ}\text{K}$)
t	Time (s)
Nu	Nusselt number (–)
Pr	Prandtl number (–)
K	Thermal conductivity ($\text{W m}^{-1} \text{K}^{-1}$)
C_p	Specific heat capacity ($\text{J kg}^{-1} \text{K}^{-1}$)
Sh	Sherwood number (–)
Sc	Schmidt number (–)
k	Mass transfer coefficient (m s^{-1})
h	Heat transfer coefficient ($\text{W m}^{-2} \text{K}^{-1}$)
X	Mole fraction (–)
D	Mass diffusivity ($\text{m}^2 \text{s}^{-1}$)
N	Total mass transfer rate ($\text{Mole s}^{-1} \text{m}^{-2}$)
C_1	Coefficient 1 (–)
q	Total heat transfer rate (W m^{-2})
C_2	Coefficient 2 ($\text{Mole s}^{-1} \text{m}^{-1}$)

Greek symbols

μ	Friction coefficient (N s m^{-2})
ρ	Density (Kg m^{-3})
δ	Thickness of concentration film (m)

Subscripts

L	Low heat transfer rate
A	Material (2-butanol)
B	Air
∞	Surrounding
C	Constant
0	Approaching 0

References

- [1] Xiao, G., Luo, K. H., Ma, X. Shuai, S. (2018). A molecular dynamics study of fuel droplet evaporation in sub- and supercritical conditions. *Proceedings of the Combustion Institute*, 37, 3219-3227.
- [2] Wang, X., Dai, M., Wang, J., Xie, Y., Ren, g. Jiang, G. (2019). Effect of ceria concentration on the evaporation characteristics of diesel fuel droplets. *Fuel*, 236, 1577-1585.
- [3] Chakraborty, S., Rosen, M. A., MacDonald, B. D. (2017). Analysis and feasibility of an evaporative cooling system with diffusion-based sessile droplet evaporation for cooling microprocessors. *Applied Thermal Engineering*, 125, 104-110.

- [4] Fisenko, S. P., Brin, A. A., Petruchik, A. I. (2004). Evaporative Cooling of Water in a Mechanical Draft Cooling Tower. *International Journal of Heat and Mass transfer*, 47, 165-177.
- [5] Wang, C., Xu, R., Song, Y., Jiang, P. (2017). Study on water flash evaporation in vacuum spray cooling. *International Journal of Heat and Mass Transfer*, 112, 279-288.
- [6] Woo, M. W. (2013). *Handbook of Food Powders: Process and Properties*. Sawston, Cambridge: Woodhead Publishing Series in Food Science, Technology, and Nutrition Vol 255.
- [7] Derby, B. (2010). Inkjet Printing of Functional and Structural Materials: Fluid Property Requirements, Feature Stability, and Resolution. *Annual Review of Materials Research*, 40, 395-414.
- [8] Hu, J., Xu, K., Lan, B., Jiang, X., Shu, L. (2014). The freezing process of continuously sprayed water droplets on the superhydrophobic silicone acrylate resin coating surface. *Applied Surface Science*, 317, 534-544.
- [9] Qian, Y., Zhao, P., Tao, C., Meng, S., Wei, J., Cheng, X. (2019). Experimental study on evaporation characteristics of lubricating oil/ gasoline blended droplet. *Experimental Thermal and Fluid Science*, 103, 99-107.
- [10] Brutin, D., Sobac, B., Nicloux, C. (2012). Influence of Substrate Nature on the Evaporation of Sessile Drop of Blood. *Journal of Heat Transfer*, 134, 61101 (7 pages).
- [11] J. P. Holman. (2010). *Heat Transfer: Tenth Edition*. McGraw-Hill.
- [12] Bird R. B., Stewart, W. E., Lightfoot, E. N. (2003). *Transport Phenomena*, second ed. New York: John Wiley & Sons.
- [13] Kosasih, E. A., Prasetyo, D., Siahaan, A., T., Yatim, A., Pamitran, A. S. (2019). Heat and mass transfer droplets of n-Hexane and n-Heptane towards temperature and rate of air flow rate variations. *AIP Conference Proceedings*, 2062, 020062.
- [14] Kosasih, E. A., Agung, I., Siahaan, A. T., Prasetyo, D. (2019). Heat and mass transfer of ethanol and methanol droplets with temperature and air flow rates variations. *AIP Conference Proceedings*, 2062, 020056.
- [15] Kosasih, E. A. (2006). *Perpindahan Panas dan Massa Dalam Proses Penguapan Tetesan: Suatu Pendekatan Baru Pada Model Film Stagnan (Doctoral Dissertation)*. The 9th Quality in Research (QIR) International Conference, Indonesia (2006).
- [16] Bird, R. B., et al. (2003). *Transport Phenomena: second edition*. John Wiley & Sons.
- [17] Kosasih, E. A., Hakim, A. A. R., Hussein, M. (2015). Analysis of butanol droplet evaporation using modified stagnant film model. *ICOME AIP Conference Proceeding*, **1983**, 020009.
- [18] Kosasih, E. A., Hakim, A. A. R., Hussein, M. (2018). Analysis of butanol droplet using modified stagnant film model. *AIP Conference Proceedings*, **1983**, 020009.
- [19] Wang, F., Yao, J., Yang, S., Liu, R., Jin, J. (2017). A new stationary droplet evaporation model and its validation. *Chinese Journal of Aeronautics*, Vol. 3 Issue 4, 1407-1416.
- [20] Walton, D. E. (2004). The evaporation of water droplets. A single droplet drying experiment. *Drying technology*, **22(3)**, 431-456 (2004).
- [21] Poiling, B. E., Prausnitz, J. M. O'connel, J. P. (2011). *The Properties of Gases and Liquids: fifth Edition*. McGraw-hill.

MIXED CONVECTIVE STAGNATION POINT FLOW OF HYBRID NANOFLUID OVER A PERMEABLE STRETCHING/SHRINKING CYLINDER WITH THERMAL STRATIFICATION EFFECT

Najiyah Safwa Khashi'ie^{1,2}
Ezad Hafidz Hafidzuddin³
Norihan Md Arifin^{1,4}
Nadiah Wahi⁴

¹Institute for Mathematical Research, Universiti Putra Malaysia, 43400 UPM Serdang, Selangor, Malaysia

²Fakulti Teknologi Kejuruteraan Mekanikal dan Pembuatan, Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100, Durian Tunggal, Melaka, Malaysia (Email: najiyah@utem.edu.my)

³Centre of Foundation Studies for Agricultural Science, Universiti Putra Malaysia, 43400 UPM Serdang, Selangor, Malaysia

⁴Department of Mathematics, Faculty of Science, Universiti Putra Malaysia, 43400 UPM Serdang, Selangor, Malaysia

Abstract: *The present study highlights the numerical and stability solutions of stagnation point flow using hybrid Cu-Al₂O₃/water nanofluid over a permeable stretching/shrinking cylinder. Mixed effects of the thermal convection and thermal stratification are also contemplated in the present work. The governing model is developed using the existing Tiwari and Das model of nanofluid with the thermophysical properties of hybrid nanoparticles. A set of transformation is adopted to reduce the complexity of the governing model (partial differential equations) and then, transformed them into a set of ordinary differential equations. The built-in-bvp4c function in Matlab software is used to compute the transformed model. The numerical results are examined and plotted within the specific range of suction, stretching/shrinking, copper nanoparticles volume fraction and thermal stratification parameters. The application of wall mass suction parameter is important to generate the flow due to the shrinking cylinder. It is noticeable that the first and second solutions of the shrinking flow only arise with the imposition of suction. The stability of solutions is also examined due to the presence of non-unique solutions and it is validated that the first solution is the physical/real solution.*

Keywords: *Hybrid nanofluid, stagnation point flow, stretching/shrinking cylinder, thermal stratification, dual solutions*

Introduction

Hybrid nanofluid is a new procreation of heat transfer fluid which may provide better heat transfer performance as compared to the traditional cooling fluid (water, ethylene glycol) and nanofluid (single nanoparticle). There are many review papers (see Sidik et al. [1], Shah and Ali [2], Xian et al. [3], Kumar and Arasu [4], Esfe et al. [5], Sajid and Ali [6], Gupta et al. [7], and Huminic and Huminic [8]) which discussed the significance of hybrid nanofluids in industrial and technological sectors, the preparation and stability of the hybrid nanoparticles. Among of the factors to create a stable hybrid nanofluid is by selecting a good combination of nanoparticles. The commonly used nanoparticles were categorized into these groups; metals (Cu, Ag), metal oxides (Al₂O₃, CuO, Fe₂O₃), carbon materials (graphite, MWCNTs, CNTs), metal carbide and metal nitride. Besides, the size, shape and solid volume fractions of the nanoparticles are also the important factors to optimize the thermal conductivity of the hybrid

nanofluid. Devi and Devi [9] introduced a new special form of thermophysical properties to analyze the boundary layer equations of the hybrid nanofluid, and then adopted it into the existing Tiwari and Das [10] model.

Stagnation point flow is generally referred to the flows which contact the solid surface and consequently, separate into two different region. This kind of flow has extensively been applied in the industrial and technological sectors since it has optimum heat transfer, fluid pressure and mass deposition rate around the stagnation point region. Hiemenz [11] and Homann [12] were among the earliest researchers who involved in the classical works of the stagnation point flow. The stagnation point flow due to a stretching flat sheet was first considered by Chiam [13] while Wang [14] extended the numerical study to a shrinking sheet and obtained two solutions. The duality of solutions identified by Wang [14] was validated by Awaludin et al. [15] and they showed that the first solution was real while the second solution was virtual using the stability analysis. However, due to the advancement in engineering and industrial sectors, fluid flows towards various types of surfaces such as cylinder, disc, wedge and cone was investigated. Early work on exterior fluid flow towards a stretching cylinder was conducted by Wang [16] while Ishak and Nazar [17] obtained a similarity solution for laminar boundary layer flow of a viscous fluid along a stretching cylinder. Mixed convective boundary layer flow with prescribed surface heat flux over a vertical cylinder was studied by Ishak [18] for impermeable cylinder and by Bachok and Ishak [19] for permeable cylinder. Ishak [18] found dual solutions for both buoyancy assisting and opposing flows and showed that the application of cylindrical surface could delay the boundary layer separation as compared to a flat sheet or plate. The stagnation point flow towards a stretching/shrinking cylinder was considered by Najib et al. [20] for viscous fluid and Omar et al. [21] for Cu-water nanofluid. Recently, Nadeem et al. [22] studied three dimensional stagnation point flow over a static cylinder and dual solutions were found. They also proved that the heat transfer rate of Cu-Al₂O₃/water hybrid nanofluid was greater as compared to the Cu-water nanofluid.

Inspired by all the previous literatures, the main idea of the present work is to numerically study the simultaneous effects of mixed convection and thermal stratification on the stagnation point flow of hybrid nanofluid towards a stretching/shrinking cylinder. The thermal stratification phenomenon is assumed with the usage of variable (non-constant) ambient temperature while the surface is permeable to enhance the flow due to the shrinking cylinder. In the study, the combination of copper (Cu) and alumina (Al₂O₃) is used. The governing model (partial differential equations) are converted into nonlinear ordinary differential equation using a set of similarity transformations and then, numerically computed using the boundary value problem solver (bvp4c) in Matlab software. Since all the previous literatures showed the existence of dual similarity solutions for the shrinking case, the present work also proposes the stability analysis to prove the real solution. All the authors are confident that there are no other literatures reported the present work.

Methodology

The present work deals with the stagnation point flow of a hybrid nanofluid towards a vertical permeable stretching/shrinking cylinder as portrayed in Figure 1. The cylinder with radius R is assumed to move with a linear velocity, $u_w(x) = ax/L$ while the free stream velocity is $u_e(x) = bx/L$ where L is the characteristic length of the cylinder and b is a positive constant. The permeable surface is important to maintain and enhance the flow due to shrinking cylinder. The fluid velocities are denoted by u and v which are measured in the x – and r – axes, respectively.

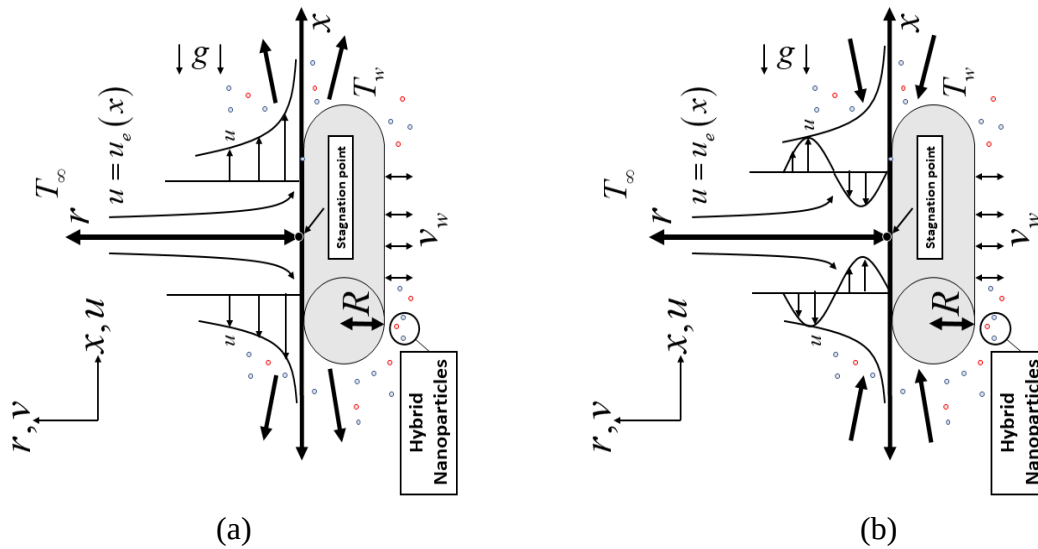


Fig. 1. The physical model with coordinate system for (a) stretching cylinder (b) shrinking cylinder.

Since the mixed effects of convective flow and thermal stratification are incorporated in the study, the variable wall temperature, $T_w(x) = T_0 + Ax/L$ with linear stratified ambient temperature, $T_\infty(x) = T_0 + Bx/L$ are used where T_0 is the initial ambient temperature of the hybrid nanofluid, A is the characteristic temperature of the cylinder and $B > 0$.

Under all these assumptions, the mathematical model for the governing problem is (see Najib et al. [20] and Omar et al. [21]):

$$\frac{\partial(ru)}{\partial x} + \frac{\partial(rv)}{\partial r} = 0, \quad (1)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial r} = u_e \frac{du_e}{dx} + \frac{\mu_{hnf}}{\rho_{hnf}} \left(\frac{\partial^2 u}{\partial r^2} + \frac{1}{r} \frac{\partial u}{\partial r} \right) + \frac{(\rho\beta_T)_{hnf} g (T - T_\infty)}{\rho_{hnf}}, \quad (2)$$

$$u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial r} = \left(\frac{k}{\rho C_p} \right)_{hnf} \left(\frac{\partial^2 T}{\partial r^2} + \frac{1}{r} \frac{\partial T}{\partial r} \right), \quad (3)$$

along with the initial and boundary conditions

$$u(x, r) = u_w(x), \quad v(x, r) = v_w, \quad T(x, r) = T_w(x) \quad \text{when } r = R, \quad (4)$$

$$u(x, r) \rightarrow u_e(x), \quad T(x, r) = T_\infty(x) \quad \text{when } r \rightarrow \infty. \quad (5)$$

T is the hybrid nanofluid temperature, g represents the gravitational acceleration while the dynamic viscosity, density, thermal expansion, thermal conductivity and specific heat of hybrid nanofluid are symbolized by μ_{hnf} , ρ_{hnf} , $(\beta_T)_{hnf}$, k_{hnf} and $(\rho C_p)_{hnf}$, respectively.

For the numerical computation, ρ_{hnf} , $(\rho C_p)_{hnf}$, μ_{hnf} , k_{hnf} and $(\rho\beta_T)_{hnf}$ will be substituted with the expressions as presented in Table 1. Table 1 also specifies the comparison of the thermophysical properties between hybrid nanofluid and nanofluid (single nanoparticle). Further, the stretching/shrinking permeable cylinder is imposed with wall mass suction if $v_w < 0$ while $v_w > 0$ stands for wall mass injection.

A set of similarity transformations is used to reduce the governing partial differential equations (1) – (3) to a set of ordinary differential equations. The similarity equations are

$$\eta = \frac{r^2 - R^2}{2R} \sqrt{\frac{b}{\nu_f L}}, \quad \psi = \sqrt{\frac{b\nu_f}{L}} x R f(\eta), \quad T = (T_w(x) - T_0) \theta(\eta) + T_\infty(x), \quad (6)$$

where $u = \frac{1}{r} \frac{\partial \psi}{\partial r}$ and $v = -\frac{1}{r} \frac{\partial \psi}{\partial x}$ which obeys Eq. (1). Therefore, the new transformed similarity differential equations with the coupled conditions are given by

$$\frac{\mu_{hnf}/\mu_f}{\rho_{hnf}/\rho_f} [(1 + 2\gamma\eta) f''' + 2\gamma f''] = f'^2 - ff'' - 1 - A_1 \lambda \theta, \quad (7)$$

$$\frac{1}{Pr(\rho C_p)_{hnf}} \frac{k_{hnf}/k_f}{(\rho C_p)_f} [(1 + 2\gamma\eta) \theta'' + 2\gamma \theta'] = (\theta + \delta) f' - f \theta', \quad (8)$$

$$f(\eta) = S, \quad f'(\eta) = \varepsilon, \quad \theta(\eta) = 1 - \delta \quad \text{at} \quad \eta = 0, \quad (9)$$

$$f'(\eta) \rightarrow 1, \quad \theta(\eta) \rightarrow 0 \quad \text{as} \quad \eta \rightarrow \infty,$$

and

$$A_1 = \frac{(1 - \phi_2) \left[(1 - \phi_1) \rho_f + \phi_1 \frac{(\rho \beta_T)_{s1}}{(\beta_T)_f} \right] + \phi_2 \frac{(\rho \beta_T)_{s2}}{(\beta_T)_f}}{(1 - \phi_2) \left[(1 - \phi_1) \rho_f + \phi_1 \rho_{s1} \right] + \phi_2 \rho_{s2}}. \quad (10)$$

The curvature of the cylindrical surface, thermal convection and thermal stratification parameters are in the form of

$$\gamma = \sqrt{L\nu_f/bR^2}, \quad \lambda = \frac{Gr_x}{Re_x^2} = \frac{g(\beta_T)_f (T_w - T_0) x^3 / \nu_f^2}{(bx^2/L\nu_f)^2}, \quad \delta = B/A, \quad (11)$$

accordingly. Further, s_1 and s_2 represent the first and second type of the nanoparticles while ϕ_1 and ϕ_2 are the first and second nanoparticles solid volume fractions, proportionately. $\varepsilon = a/b$ stands for the velocity ratio parameter and for case $\gamma > 0$ (cylinder), $\varepsilon > 0$ corresponds to the stretching flow case whereas $\varepsilon < 0$ corresponds to the shrinking flow case. The present problem is reduced to the stretching/shrinking flat surface if the value of γ is zero. In addition, $\lambda < 0$, $\lambda = 0$ and $\lambda > 0$ imply the opposing, forced convective and assisting flow, respectively.

Table 2 elucidates the values of the thermophysical properties (ρ, C_p, k, β_T) between the nanoparticles (copper, alumina) and regular fluid (water).

Table 1. The thermophysical properties between hybrid and traditional nanofluids (see Devi and Devi [9], Rostami et al. [23] and Das and Jana [24]).

Properties	Hybrid Nanofluid	Traditional Nanofluid
Density	$\rho_{hnf} = (1 - \phi_2) \left[(1 - \phi_1) \rho_f + \phi_1 \rho_{s1} \right] + \phi_2 \rho_{s2}$	$\rho_{nf} = (1 - \phi) \rho_f + \phi \rho_s$
Heat Capacity	$(\rho C_p)_{hnf} = (1 - \phi_2) \left[(1 - \phi_1) (\rho C_p)_f + \phi_1 (\rho C_p)_{s1} \right] + \phi_2 (\rho C_p)_{s2}$	$(\rho C_p)_{nf} = (1 - \phi) (\rho C_p)_f + \phi (\rho C_p)_s$
Dynamic Viscosity	$\frac{\mu_{hnf}}{\mu_f} = \frac{1}{(1 - \phi_1)^{2.5} (1 - \phi_2)^{2.5}}$	$\frac{\mu_{nf}}{\mu_f} = \frac{1}{(1 - \phi_1)^{2.5}}$
Thermal Conductivity	$k_{hnf} = \left[\frac{k_{s2} + 2k_{bf} - 2\phi_2 (k_{bf} - k_{s2})}{k_{s2} + 2k_{bf} + \phi_2 (k_{bf} - k_{s2})} \right] k_{bf}$ where $k_{bf} = \left[\frac{k_{s1} + 2k_f - 2\phi_1 (k_f - k_{s1})}{k_{s1} + 2k_f + \phi_1 (k_f - k_{s1})} \right] k_f$	$k_{nf} = \left[\frac{k_s + 2k_f - 2\phi (k_f - k_s)}{k_s + 2k_f + \phi (k_f - k_s)} \right] k_f$
Thermal Expansion	$(\rho \beta_T)_{hnf} = (1 - \phi_2) \left[(1 - \phi_1) (\rho \beta_T)_f + \phi_1 (\rho \beta_T)_{s1} \right] + \phi_2 (\rho \beta_T)_{s2}$	$(\rho \beta_T)_{nf} = (1 - \phi) (\rho \beta_T)_f + \phi (\rho \beta_T)_s$

Table 2. Thermophysical properties of the alumina, copper and water (see Rostami et al. [23], Das and Jana [24] and Oztop and Abu-Nada [25]).

Physical properties	ρ (kg/m ³)	C_p (J/kgK)	k (W/mK)	β_T (K ⁻¹)
Alumina	3970	765	40	0.85 x 10 ⁻⁵
Copper	8933	385	400	1.67 x 10 ⁻⁵
Water	997.1	4179	0.6130	21 x 10 ⁻⁵

The main physical interests of the study are the skin friction coefficient C_f and the local Nusselt number Nu_x which are generally in the form of

$$C_f = \frac{\tau_w}{\rho_f u_e^2}, \quad Nu_x = \frac{x q_w}{k_f (T_w(x) - T_0)}, \quad (12)$$

where τ_w is the surface shear stress and q_w is the surface heat flux:

$$\tau_w = \mu_{hnf} \left(\frac{\partial u}{\partial r} \right)_{r=R}, \quad q_w = -k_{hnf} \left(\frac{\partial T}{\partial r} \right)_{r=R}. \quad (13)$$

Using Equations (6), (12) and (13), the reduced skin friction coefficient and reduced local Nusselt number are given by

$$Re_x^{1/2} C_f = \frac{\mu_{hnf}}{\mu_f} f''(0), \quad Re_x^{-1/2} Nu_x = -\frac{k_{hnf}}{k_f} \theta'(0), \quad (14)$$

where $Re_x = u_e(x) x / \nu_f$ is the local Reynolds number.

Stability of Solution

Following Merkin [26], an unsteady form of Equations (2) and (3) is considered which are:

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial r} = u_e \frac{du_e}{dx} + \frac{\mu_{hnf}}{\rho_{hnf}} \left(\frac{\partial^2 u}{\partial r^2} + \frac{1}{r} \frac{\partial u}{\partial r} \right) + \frac{(\rho \beta_T)_{hnf} g (T - T_\infty)}{\rho_{hnf}}, \quad (15)$$

$$\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial r} = \left(\frac{k}{\rho C_p} \right)_{hnf} \left(\frac{\partial^2 T}{\partial r^2} + \frac{1}{r} \frac{\partial T}{\partial r} \right), \quad (16)$$

and the relevant similarity transformation is given by

$$\eta = \frac{r^2 - R^2}{2R} \sqrt{\frac{b}{v_f L}}, \quad \psi = \sqrt{\frac{b v_f}{L}} x R f(\eta, \tau), \quad T = (T_w(x) - T_0) \theta(\eta, \tau) + T_\infty(x), \quad (17)$$

where $\tau = \frac{b}{L} t$. Using Eq. (15), Eqs. (13) and (14) can be rewritten as

$$\frac{\mu_{hnf}/\mu_f}{\rho_{hnf}/\rho_f} \left[(1 + 2\gamma\eta) \frac{\partial^3 f}{\partial \eta^3} + 2\gamma \frac{\partial^2 f}{\partial \eta^2} \right] + 1 + f \frac{\partial^2 f}{\partial \eta^2} - \left(\frac{\partial f}{\partial \eta} \right)^2 + A_1 \lambda \theta - \frac{\partial^2 f}{\partial \eta \partial \tau} = 0, \quad (18)$$

$$\frac{1}{Pr} \frac{k_{hnf}/k_f}{(\rho C_p)_{hnf}/(\rho C_p)_f} \left[(1 + 2\gamma\eta) \frac{\partial^2 \theta}{\partial \eta^2} + 2\gamma \frac{\partial \theta}{\partial \eta} \right] + f \frac{\partial \theta}{\partial \eta} - \frac{\partial f}{\partial \eta} (\theta + \delta) - \frac{\partial \theta}{\partial \tau} = 0, \quad (19)$$

with the transformed conditions

$$f(0, \tau) = S, \quad \frac{\partial f}{\partial \eta}(0, \tau) = \varepsilon, \quad \theta(0, \tau) = 1 - \delta, \quad (20)$$

$$\frac{\partial f}{\partial \eta}(\eta, \tau) \rightarrow 1, \quad \theta(\eta, \tau) \rightarrow 0 \quad \text{as } \eta \rightarrow \infty.$$

According to Weidman et al. [27], the following perturbation equations (see Eq. (19)) are introduced to investigate the stability of the similarity solutions $f(\eta) = f_0(\eta)$ and $\theta(\eta) = \theta_0(\eta)$. An unknown eigenvalue σ is used in the formulation whereas $F(\eta)$ and $G(\eta)$ are a small relative to $f_0(\eta)$ and $\theta_0(\eta)$, correspondingly.

$$f(\eta, \tau) = f_0(\eta) + e^{-\sigma \tau} F(\eta), \quad (21)$$

$$\theta(\eta, \tau) = \theta_0(\eta) + e^{-\sigma \tau} G(\eta),$$

Hence, by employing Eq. (19) into Eqs. (16)-(18), the following linearized eigenvalue problem is attained such that

$$\frac{\mu_{hnf}/\mu_f}{\rho_{hnf}/\rho_f} \left[(1 + 2\gamma\eta) F''' + 2\gamma F'' \right] + f_0 F'' - (2f_0' - \sigma) F' + f_0'' F + A_1 \lambda G = 0, \quad (22)$$

$$\frac{1}{Pr} \frac{k_{hnf}/k_f}{(\rho C_p)_{hnf}/(\rho C_p)_f} \left[(1 + 2\gamma\eta) G'' + 2\gamma G' \right] + F \theta_0' + f_0 G' - (\theta_0 + \delta) F' - (f_0' - \sigma) G = 0, \quad (23)$$

$$F'(0) = F(0) = G(0) = 0, \quad (24)$$

$$F'(\eta) \rightarrow 0, \quad G(\eta) \rightarrow 0 \quad \text{as } \eta \rightarrow \infty.$$

The smallest eigenvalue σ_1 will determine the stability of the similarity solutions such that positive value of σ_1 implies that the solution is real or stable. In contrast, negative σ_1 indicates that the steady solution is unstable. Hence, to find the possible range of the smallest eigenvalue σ_1 , Eqs. (22)-(24) are computed using the bvp4c solver. However, a modification for Eq. (24) is needed so that the bvp4c code can be successfully executed. Following Harris et al. [28], $F'(\infty)$ in Eq. (24) is relaxed and substituted with a new condition $F''(0) = 1$.

Results and Discussion

The similarity solutions of the ordinary differential equations (see Eqs. (7) and (8)) coupled with the boundary conditions (9) are attained by using the bvp4c solver in the Matlab software. The combination of appropriate guess value, boundary layer thickness η_∞ and the value of the control parameters may lead to the existence of the unique or multiple solutions within the specified accuracy. The objectives of the present work are (a) to study the effect of parameters namely suction S , the surface curvature γ , thermal buoyancy parameter λ and copper nanoparticles volume fraction ϕ_2 when the stratified fluid is considered, and (b) to identify all the potential solutions arise from the governing model. In the present study, $\eta_\infty = 15$ and alumina nanoparticles volume fraction $\phi_2 = 0.1$ are fixed whereas $Pr = 6.2$ is used to represent the water (viscous fluid) as the base fluid (see Oztop and Abu-Nada [25]). A comparison between present data and previously published literatures (see Table 3) are conducted for the validation of the present model. Omar et al. [21] solved the stagnation point flow of Cu-water nanofluid over a stretching/shrinking cylinder whereas Bachok et al. [29] studied the stagnation point flow of nanofluid over a stretching/shrinking sheet. Both literatures adopted shooting method for the numerical computation. It is apparent from Table 3 that the comparison values of $f''(0)$ between present method and the existing literatures are in a good agreement, hence the authors are confident to use the model in the present work.

Table 3. The comparison values of $f''(0)$ for different stretching/shrinking parameter (ε) and curvature parameter (γ) when $\phi_2 = 0.1$, $\phi_1 = 0$, $Pr = 6.2$ and $\delta = \lambda = S = 0$.

ε	Present		Omar et al. [21] (shooting method)		Bachok et al. [29] (shooting method)	
	$\gamma = 0$	$\gamma = 0.4$	$\gamma = 0$	$\gamma = 0.4$	$\gamma = 0$	$\gamma = 0.4$
2	-2.217106	-2.396084	-2.217106	-2.396084	-2.217106	-
1	0	0	0	0	0	-
0.5	0.837940	0.935394	0.837940	0.935394	0.837940	-
0	1.447977	1.653867	1.447977	1.653867	1.447977	-
-0.5	1.757032	2.096118	1.757022	2.096118	1.757032	-
	1.271347	2.003737	1.271347	2.003737	1.271347	
-1.15	[0.137095]	[-0.131198]	[0.137095]	[-0.131198]	[0.137095]	-
	1.095419	1.939265	1.095419	1.939265	1.095419	
-1.2	[0.274479]	[-0.101872]	[0.274479]	[-0.101872]	[0.274479]	-
-1.2465	0.686382	1.865189	0.686380	1.865189	0.686379	-

The diversity of $Re_x^{1/2} C_f$ and $Re_x^{-1/2} Nu_x$ towards S with the enhancement of the curvature parameter γ and copper solid volume fractions ϕ_2 are manifested in Figures 2 and 3, respectively. The application of suitable wall mass suction parameter ($S > 0$) can generate the dual similarity solutions in the present work as interpreted in Figures 2 and 3. Nevertheless, if the cylinder is impermeable ($S = 0$) or injection parameter ($S < 0$) is applied, no similarity solutions (unique or dual) can be obtained. Figure 2 shows that less value of suction parameter S (less magnitude) is needed if the curvature parameter increases. However, in the present work, the authors only test the value of γ within the range of $0 \leq \gamma \leq 0.1$ and if $\gamma > 0.1$, the result may be different. For each value of S , both skin friction coefficient and heat transfer rate of shrinking cylinder ($\gamma = 0.1$) is greater as compared to the shrinking flat surface ($\gamma = 0$).

Figure 3 displays the $Re_x^{1/2} C_f$ and $Re_x^{-1/2} Nu_x$ when ϕ_2 is enhanced. It shows that both $Re_x^{1/2} C_f$ and $Re_x^{-1/2} Nu_x$ intensify with the increment of ϕ_2 . However, the result of heat transfer rate may differ if $S > 1$ is considered. It is clear in Figure 3(b) that as $S \rightarrow 1$, the heat transfer rate only slightly increases when ϕ_2 is increased. Figure 4 portrays the variations of $Re_x^{1/2} C_f$ and $Re_x^{-1/2} Nu_x$ towards the mixed convection parameter λ for different values of γ . The dual solutions are possible in the opposing flow region ($\lambda < 0$) and assisting flow region ($\lambda > 0$) while only unique solution is attained for the forced convective flow ($\lambda = 0$). However, it can be seen that in the assisting flow region, the second solution only can be achieved for a limited value of positive λ . Further investigation is required to study the parameters which can maintain the existence of the second solution in the assisting flow region. Additionally, the second solution for $Re_x^{-1/2} Nu_x$ becomes unbounded as $\lambda \rightarrow 0$ from left ($\lambda \rightarrow 0^-$) and right ($\lambda \rightarrow 0^+$). Figure 4 also reveals that the cylindrical surface ($\gamma > 0$) also can increase the existence range of the similarity solutions to Eqs. (7)-(9) and consequently, delays the boundary layer separation.

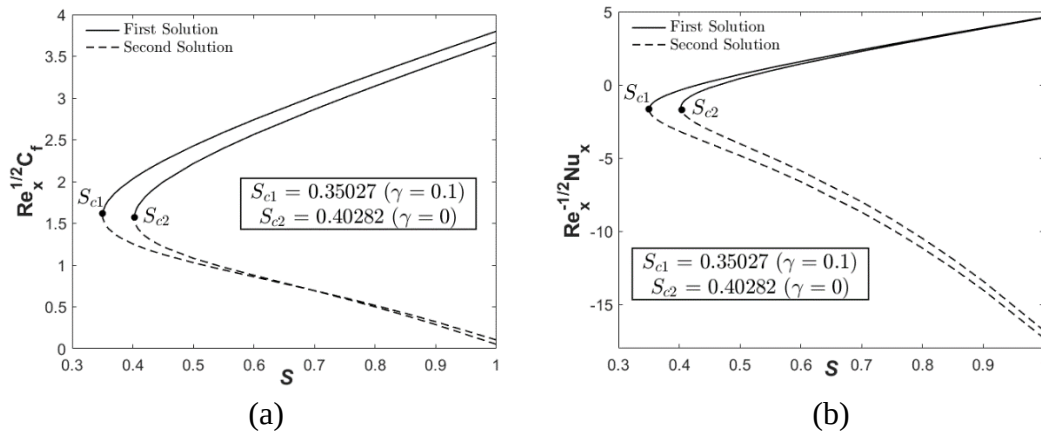


Fig. 2. (a) $Re_x^{1/2} C_f$ (b) $Re_x^{-1/2} Nu_x$ towards s for shrinking flat surface ($\gamma = 0$) and shrinking cylinder ($\gamma = 0.1$) when $\varepsilon = \lambda = -1$ and $\phi_2 = \delta = 0.005$.

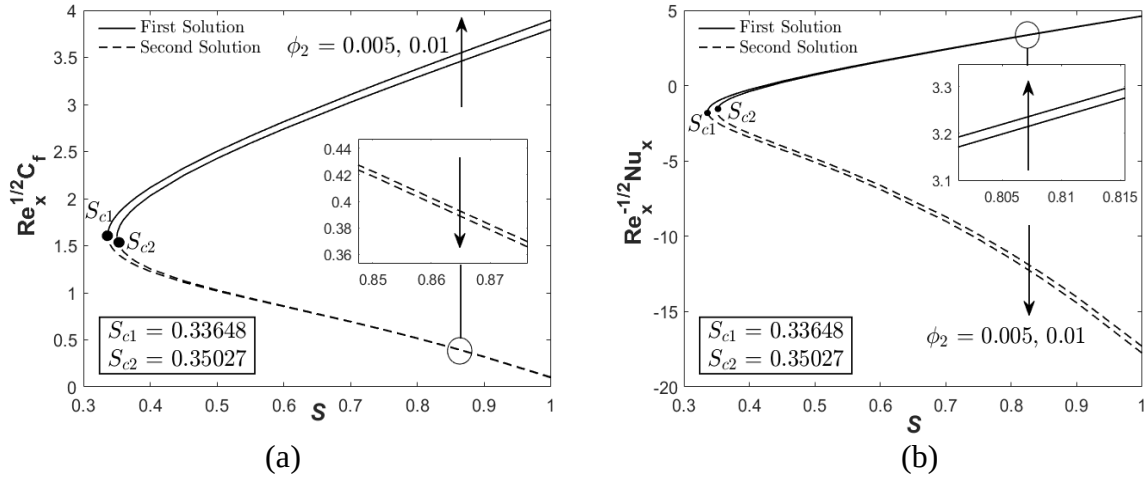


Fig. 3. (a) $Re_x^{1/2} C_f$ (b) $Re_x^{-1/2} Nu_x$ towards s for different ϕ_2 when $\gamma = 0.1$, $\varepsilon = \lambda = -1$ and $\delta = 0.005$.

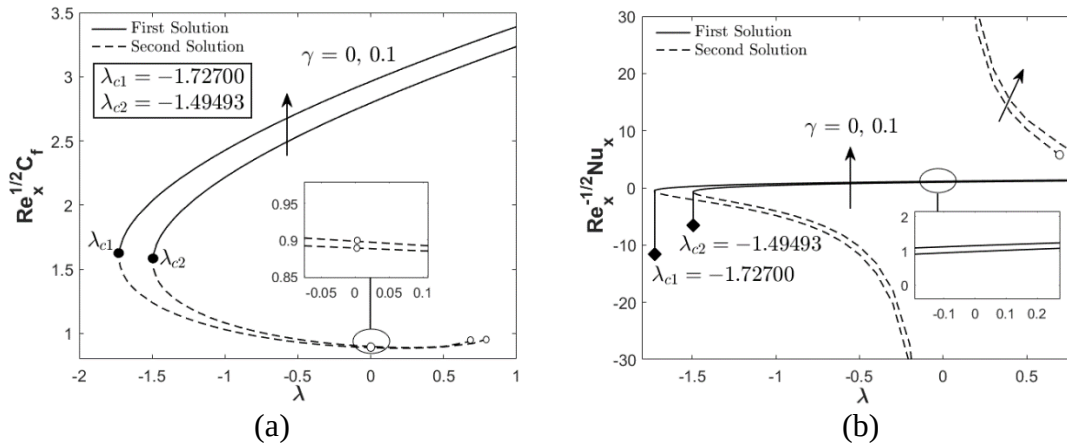


Fig. 4. (a) $Re_x^{1/2} C_f$ (b) $Re_x^{-1/2} Nu_x$ towards λ for various γ when $S = 0.5$, $\varepsilon = -1$ and $\phi_2 = \delta = 0.005$.

The velocity and temperature profiles for shrinking cylinder case ($\varepsilon < 0$), stretching cylinder case ($\varepsilon > 0$) and static cylinder case ($\varepsilon = 0$) are exhibited in Figure 5. The fluid velocity decreases when the velocity ratio parameter decrease whereas a contradictory result is obtained for the temperature profile. Otherwise, the second solution shows a different result as compared to the first solution. The stability of similarity solutions are determined by finding the smallest eigenvalue σ_1 . Thus, in the present work, Equations (22)-(24) with the new boundary condition are solved using the efficient bvp4c solver (code c and d) to find the σ_1 for both solutions. It is apparent from Figure 6 that the first solution has positive σ_1 while reverse results for the second solution. In addition, Figure 6 also depicts that σ_1 for both similarity solutions approach to 0 as $s \rightarrow s_c$ which justifies the fact that at $s = s_c$, the value

of σ_1 is zero. Hence, from this stability analysis result, the first solution is stable and real whereas the second solution is not real or virtual.

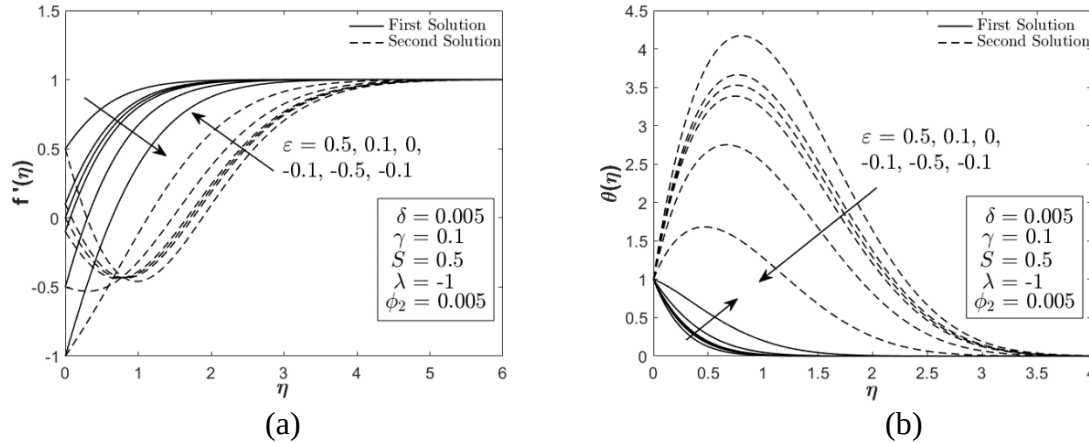


Fig. 5. (a) Velocity (b) Temperature profiles for various values of velocity ratio parameter.

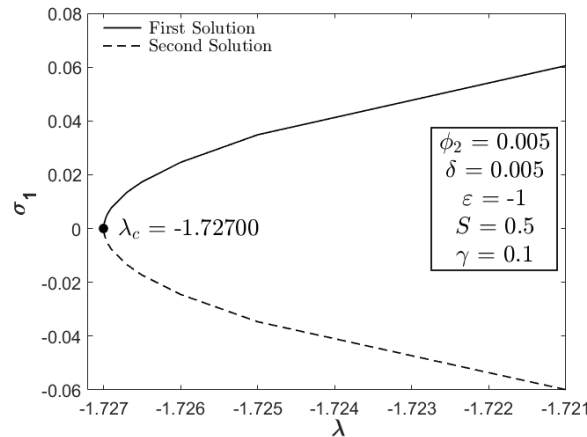


Fig. 6. The smallest eigenvalue σ_1 towards λ for both first and second solutions.

Conclusions

The present work deals with the numerical solution of stagnation point flow and heat transfer of a hybrid nanofluid over a vertical permeable stretching/shrinking cylinder. Thermal stratification process is considered as a result of a variable (linear stratified) ambient temperature while the wall temperature is also in a variable form to illustrate the mixed convection phenomenon. The present model in partial differential equations are converted to a nonlinear system of ordinary (similarity) differential equations using the similarity variable and further, computed using the boundary value problem solver (bvp4c) in the Matlab software. The conclusions are:

- the similarity solutions (first and second) are attained for the shrinking cylinder case with low magnitude if the wall mass suction parameter is imposed. No similarity solutions are generated if the cylinder is impermeable or injection parameter is applied.
- the higher values of curvature γ and copper nanoparticles volume fractions ϕ_2 parameters may inflate the reduced skin friction coefficient and local Nusselt number within the specific value of suction parameter.

- The first similarity solution exists for all values of the buoyancy parameter up to the critical value whereas the second similarity solution is impossible for the forced convective flow and higher magnitude of assisting flow.
- the execution of stability analysis proves the reliability of the first solution.
- the present work may contribute idea to the other researchers from various background (mathematics, mechanical, physics) on how to control (increase/decrease) the heat transfer rate by manipulating the parameters or the possibilities of gaining non-unique solutions in the computation.
- The present results only applicable to the combination of copper and alumina nanoparticles only. However, other researchers may extend the study using other hybrid nanofluids or physical parameters to achieve the desired outcome.

Acknowledgement

The authors gratefully acknowledge the financial support received in the form of Putra Grant 9619000 from Universiti Putra Malaysia. The support from Ministry of Education (Malaysia) and Universiti Teknikal Malaysia Melaka (UTEM-SLAB scholarship) also highly appreciated.

References

- [1] Sidik, Nor Azwadi Che, Isa Muhammad Adamu, Muhammad Mahmud Jamil, G. H. R. Kefayati, Rizalman Mamat, and G. Najafi. "Recent progress on hybrid nanofluids in heat transfer applications: a comprehensive review." *International Communications of Heat and Mass Transfer* 78, (2016): 68-79.
- [2] Shah, Tayyab Raza, and Hafiz Muhammad Ali. "Applications of hybrid nanofluids in solar energy, practical limitations and challenges: a critical review." *Solar Energy* 183, (2019): 173-203.
- [3] Xian, Hong Wei, Nor Azwadi Che Sidik, Siti Rahmah Aid, Tan Lit Ken, and Yutaka Asako. "Review on preparation techniques, properties and performance of hybrid nanofluid in recent engineering applications." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 45, no. 1 (2018): 1-13.
- [4] Kumar, D. Dhinesh, and A. Valan Arasu. "A comprehensive review of preparation, characterization, properties and stability of hybrid nanofluids." *Renewable and Sustainable Energy Reviews* 81, (2018): 1669-1689.
- [5] Esfe, Mohammad Hemmat, Mahmoud Kiannejad Amiri, and Ali Alirezaie. "Thermal conductivity of a hybrid nanofluid." *Journal of Thermal Analysis and Calorimetry* 134, no. 2 (2018): 1113-1122.
- [6] Sajid, Muhammad Usman, and Hafiz Muhammad Ali. "Thermal conductivity of hybrid nanofluids: a critical review." *International Journal of Heat and Mass Transfer* 126, (2018): 211-234.
- [7] Gupta, Munish, Vinay Singh, Satish Kumar, Sandeep Kumar, Neeraj Dilbaghi, and Zafar Said. "Up to date review on the synthesis and thermophysical properties of hybrid nanofluids." *Journal of cleaner production* 190, (2018): 169-192.
- [8] Huminic, Gabriela, and Angel Huminic. "Hybrid nanofluids for heat transfer applications—A state-of-the-art review." *International Journal of Heat and Mass Transfer* 125, (2018): 82-103.
- [9] Devi, SP Anjali, and S. Suriya Uma Devi. "Numerical investigation of hydromagnetic hybrid Cu–Al₂O₃/water nanofluid flow over a permeable stretching sheet with suction." *International Journal of Nonlinear Sciences and Numerical Simulation* 17, no. 5 (2016): 249-257.

- [10] Tiwari, Raj Kamal, and Manab Kumar Das. "Heat transfer augmentation in a two-sided lid-driven differentially heated square cavity utilizing nanofluids." *International Journal of heat and Mass transfer* 50, no. 9-10 (2007): 2002-2018.
- [11] Hiemenz, Karl. "Die Grenzschicht an einem in den gleichförmigen Flüssigkeitsstrom eingetauchten geraden Kreiszylinder." *Dinglers Polytech J* 326, (1911): 321-324.
- [12] Homann, Fritz. "Der Einfluss grosser Zähigkeit bei der Strömung um den Zylinder und um die Kugel." *ZAMM-Journal of Applied Mathematics and Mechanics/Zeitschrift für Angewandte Mathematik und Mechanik* 16, no. 3 (1936): 153-164.
- [13] TC, Chiam. "Stagnation-point flow towards a stretching plate." *Journal of the physical society of Japan* 63, no. 6 (1994): 2443-2444.
- [14] Wang, C.Y. "Stagnation flow towards a shrinking sheet." *International Journal of Non-Linear Mechanics* 43, no. 5 (2008): 377-382.
- [15] Awaludin, I. S., P. D. Weidman, and Anuar Ishak. "Stability analysis of stagnation-point flow over a stretching/shrinking sheet." *AIP Advances* 6, no. 4 (2016): p.045308.
- [16] Wang, C.Y. "Fluid flow due to a stretching cylinder." *The Physics of fluids* 31, no. 3 (1988): 466-468.
- [17] Ishak, Anuar Mohd, and Roslinda Mohd Nazar. "Laminar boundary layer flow along a stretching cylinder." *European Journal of Scientific Research* 36, no. 1 (2009): 22-29.
- [18] Ishak, A. "Mixed convection boundary layer flow over a vertical cylinder with prescribed surface heat flux." *Journal of Physics A: Mathematical and Theoretical* 42, no. 19 (2009): p.195501.
- [19] Bachok, Norfifah, and Anuar Mohd Ishak. "Mixed convection boundary layer flow over a permeable vertical cylinder with prescribed surface heat flux." *European Journal of Scientific Research* 34, no. 1 (2009): 46-54.
- [20] Najib, N., Bachok, N., Arifin, N.M. and Ishak, A. "Stagnation point flow and mass transfer with chemical reaction past a stretching/shrinking cylinder." *Scientific reports* 4, (2014): p.4178.
- [21] Omar, Noor Syamimi, Norfifah Bachok, and Norihan Md Arifin. "Stagnation point flow over a stretching or shrinking cylinder in a copper-water nanofluid." *Indian Journal of Science and Technology* 8, (2015): 1-7.
- [22] Nadeem, S., Nadeem Abbas, and A. U. Khan. "Characteristics of three dimensional stagnation point flow of Hybrid nanofluid past a circular cylinder." *Results in physics* 8, (2018): 829-835.
- [23] Rostami, Mohammadreza Nademi, Saeed Dinarvand, and Ioan Pop. "Dual solutions for mixed convective stagnation-point flow of an aqueous silica–alumina hybrid nanofluid." *Chinese Journal of Physics* 56, no. 5 (2018): 2465-2478.
- [24] Das, S., and R. N. Jana. "Natural convective magneto-nanofluid flow and radiative heat transfer past a moving vertical plate." *Alexandria Engineering Journal* 54, no. 1 (2015): 55-64.
- [25] Oztop, Hakan F., and Eiyad Abu-Nada. "Numerical study of natural convection in partially heated rectangular enclosures filled with nanofluids." *International journal of heat and fluid flow* 29, no. 5 (2008): 1326-1336.
- [26] Merkin, J.H. "On dual solutions occurring in mixed convection in a porous medium." *Journal of engineering Mathematics* 20, no. 2 (1986): 171-179.
- [27] Weidman, P. D., D. G. Kubitschek, and A. M. J. Davis. "The effect of transpiration on self-similar boundary layer flow over moving surfaces." *International journal of engineering science* 44, no. 11-12 (2006): 730-737.



- [28] Harris, S. D., D. B. Ingham, and I. Pop. "Mixed convection boundary-layer flow near the stagnation point on a vertical surface in a porous medium: Brinkman model with slip." *Transport in Porous Media* 77, no. 2 (2009): 267-285.
- [29] Bachok, Norfifah, Anuar Ishak, and Ioan Pop. "Stagnation-point flow over a stretching." *Nanoscale research letters* 6, no. 1 (2011): 1-10.

EFFECT OF BUOYANCY FORCE ON THE FLOW AND HEAT TRANSFER AROUND A THIN NEEDLE IN Al_2O_3 -Cu HYBRID NANOFLUID

Siti Nur Alwani Salleh¹

Norfifah Bachok²

Norihan Md Arifin³

Fadzilah Md Ali⁴

¹Institute for Mathematical Research, Universiti Putra Malaysia, 43400, UPM Serdang, Selangor, Malaysia (Email: alwani24salleh@gmail.com)

²Department of Mathematics, Faculty of Science, Universiti Putra Malaysia, 43400, UPM Serdang, Selangor, Malaysia

Abstract: An analysis has been performed to study the effect of buoyancy force in the boundary layer flow past a thin needle in a hybrid nanofluid. In this study, we have taken into consideration a combination of two types of nanoparticles which are alumina and copper in the base fluid. The coupled nonlinear partial differential equations representing momentum and heat equations are reduced into a set of nonlinear ordinary differential equations. The transformed equations are evaluated numerically by adopting bvp4c method with the help of MATLAB software. Effects of involved controlling physical parameters, namely buoyancy or mixed convection parameter, velocity ratio parameter, solid volume fraction parameters and needle thickness on the skin friction coefficient and heat transfer rate as well as velocity and temperature profiles are indicated through graphs and then discussed. It is noticed that the presence of hybrid nanoparticles has a significant impact on the skin friction coefficient and also the heat transfer rate.

Keywords: Thin needle, Buoyancy force, Hybrid nanofluid, Multiple solutions, Stability analysis

Introduction

A few decade ago, the existence of nanofluid as a heat transfer fluid becoming one of the subjects of interest in fluid mechanics. Nevertheless, this subject is no longer beneficial to some heat transfer processes. In this regard, a new competent working fluid known as hybrid nanofluid which consist of two or three nanoparticles has been proposed to replace a nanofluid with single component (material or nanoparticle). Similar to nanofluid with one component, hybrid or combined nanomaterials have a very small dimension that is 10-100 nm in diameter. In order to form the hybrid nanofluid, nanoparticles with different types: metallic (Ag, Al, Cu and Zn) and non-metallic or metal oxide (Al_2O_3 , CuO, MgO, and Fe_3O_4) must undergo a synthetization process. A detail and comprehensive study of the process has been analyzed by Sarkar et al. [1]. In their study, they found that the metallic nanomaterials possess high thermal conductivity, but weak in stability and reactivity. In contrast to that, metal oxide have low thermal conductivity, but good in others properties like stability and reactivity. The combination of the deficiencies and strengths of those materials is capable of making them complement each other.

Authentically, hybrid nanomaterials have very unusual features that a single component cannot possess. It may be lacking in some of the thermal or chemical properties. Unlike a nanofluid, hybrid nanofluid have many good properties such as better mechanical resistance,

chemical stability, thermal conductivity, physical strength and so on [1]. These properties make hybrid nanofluid be a main topic in experimental [2-5] and computational studies. Possible areas in which hybrid nanofluid is widely used include: air conditioning system, solar heating, microfluidics, space aircrafts and ships, transportations, manufacturing, medical and lubrication. The influence of alumina-copper hybrid nanofluid has been studied by Moghadassi et al. [6], and they noticed that hybrid nanofluid has shown better performance of heat transfer compared to nanofluid. The heat transfer analysis of copper-alumina/water hybrid nanofluid around a stretched surface is performed by Devi and Devi [7]. Later, Mehryan et al. [8] examined the flow of natural convection in a cavity filled with a porous medium immersed in a hybrid nanofluid. A comprehensive studies on the hybrid nanofluid topic are presented in Refs. [9-11].

The flow over a thin needle is a very important and significant process due to it has numerous applications especially in engineering and biomedical areas. It is typically used in circulatory problems, transportations, wire coating and in hot wire anemometer or protected thermocouple for computing the wind velocity. The term thin needle is defined as a rebellious body whose sizes are comparable to that of boundary layer or smaller. It is worth mentioning that, the motion of the needle disturbs the free-stream flow, and this is a main focus of the flow and heat transfer analysis in experimental studies for finding the velocity and temperature profiles. This extraordinary feature of the thin needle making such topic interesting for some researchers. Lee [12] started to investigate the boundary layer flow past a stretching surface in viscous fluid by discussing the asymptotic behaviors of the system. Narain and Uberoi [13] performed the force convection flow over an isothermal thin needle in viscous fluid. A year later, Narain and Uberoi [14] examined the mixed convection flow on the slender needle by taking into consideration both isoflux and isothermal wall situations. Subsequently, Chen and Smith [15] studied analytically the steady forced convection flow and heat transfer from nonisothermal thin needle. A more works on the thin needle in viscous fluid can be found in the current literatures [16-18].

The study of thin needle in a nanofluid is first considered by Grosan and Pop [19]. In their research, they solve the effect of variable wall temperature on the classical problem of forced convection heat transfer past a thin needle. The problem of stagnation point flow in water-carbon nanofluid with variable heat flux on the thin needle is investigated by Hayat et al. [20]. In recent years, many researchers include Krishna et al. [21], Ahmad et al. [22] and Soid et al. [23] had performed the numerical studies on the thin needle in nanofluid by using various physical effects. Very recently, Salleh et al. [24] carried out a stability analysis of nanofluid flow and heat transfer past a slender needle by considering the heat generation and chemical reaction effects. The consideration of stability analysis has been introduced by Merkin [25] after he got an idea how to identify the stable solution (or physically realizable) if there exist more than one solution in a system. Since the work of Merkin [25] has been well received, more publications are found in the existing literature [26-28].

Being motivated by the above referred studies, in this work we analyzed the steady behavior of incompressible flow and wall heat transfer around a continuously moving thin needle saturated by hybrid nanofluid in the presence of buoyancy effect. Two kinds of nanoparticles, namely alumina (Al_2O_3) and copper (Cu) are considered in order to investigate the behavior of hybrid nanofluid. The governing mathematical system is computed numerically via MATLAB function `bvp4c`. Graphical demonstrations have been made for the velocity, temperature, skin friction coefficient and local Nusselt number with respect to some pertinent parameters.

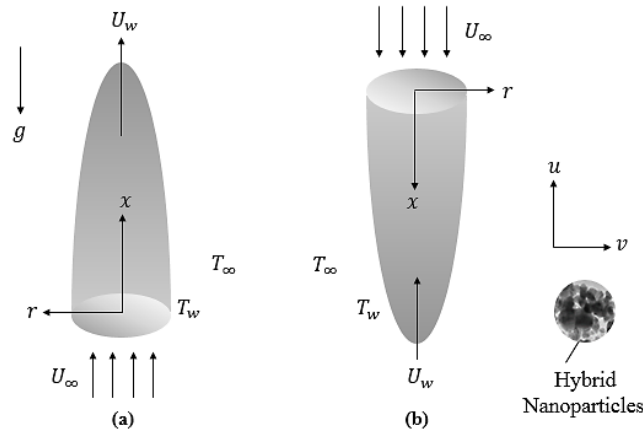


Fig. 1. Cylindrical coordinate system: (a) assisting flow and (b) opposing flow

Methodology

The flow of alumina-copper hybrid nanofluid of uniform ambient temperature T_∞ past a moving thin needle of uniform wall temperature T_w is examined. The flow system is assumed to be in the x direction, which is along the needle surface, and r axis is normal to it as shown in Fig. 1. In Fig. 1, U_w denotes a uniform velocity of the needle that moves in the same and opposite ways to the free stream flow of uniform velocity U_∞ , and $r = R(x)$ represents the needle radius. Taking into account the above considerations, equations for the continuity, momentum and energy can be written as:

$$(ru)_x + (rv)_r = 0, \quad (1)$$

$$uu_x + vv_r = \frac{\mu_{hnf}}{\rho_{hnf}} \frac{1}{r} (ru_r)_r + \frac{(\rho\beta)_{hnf}}{\rho_{hnf}} g(T - T_\infty), \quad (2)$$

$$uT_x + vT_r = \frac{1}{r} \frac{k_{hnf}}{(\rho C_p)_{hnf}} (rT_r)_r. \quad (3)$$

The corresponding boundary conditions for Eqs. (1) – (3) are

$$u = U_w, \quad v = 0, \quad T = T_w \quad \text{at} \quad r = R(x), \quad (4)$$

$$u \rightarrow U_\infty, \quad T \rightarrow T_\infty \quad \text{as} \quad r \rightarrow \infty,$$

where the velocities components along x and r axis are u and v , respectively. Besides that, T is the temperature of nanofluid, μ is the dynamic viscosity, ρ is the density, β is the volumetric expansion coefficient, k is the thermal conductivity, g is the acceleration due to gravity and C_p is the specific heat at uniform pressure. The subscripts ‘ hnf ’, ‘ nf ’ and ‘ s ’ represent the terms for ‘hybrid nanofluid’, ‘nanofluid’ and ‘solid nanoparticle’, respectively. The solid nanoparticles volume fraction for alumina and copper are given by ϕ_1 and ϕ_2 , respectively.

The thermophysical properties of the base fluid and solid nanoparticles at 25 °C and the applied models for physical properties of hybrid nanofluid and nanofluid are tabulated in Table 1 and Table 2, respectively [7].

Table 1: Thermophysical properties of the base fluid and solid nanoparticles at 25°C.

Properties	Fluid phase (water)	Al ₂ O ₃	Cu
ρ (kg / m ³)	997.0	3970	8933
C_p (J/kg K)	4180	765	385
k (W / mK)	0.6071	40	400
$\beta \times 10^5$ (1 / K)	21	0.85	1.67

Table 2: Applied models for physical properties of the nanofluid and hybrid nanofluid

Properties	Nanofluid	Hybrid Nanofluid
Density	$\rho_{nf} = (1 - \phi)\rho_f + \phi\rho_s$	$\rho_{hnf} = (1 - \phi_2)[(1 - \phi_1)\rho_f + \phi_1\rho_{s1}] + \phi_2\rho_{s2}$
Heat capacity	$(\rho C_p)_{nf} = (1 - \phi)(\rho C_p)_f + \phi(\rho C_p)_s$	$(\rho C_p)_{hnf} = (1 - \phi_2)[(1 - \phi_1)(\rho C_p)_f + \phi_1(\rho C_p)_{s1}] + \phi_2(\rho C_p)_{s2}$
Dynamic viscosity	$\mu_{nf} = \frac{\mu_f}{(1 - \phi)^{2.5}}$	$\mu_{hnf} = \frac{\mu_f}{(1 - \phi_1)^{2.5}(1 - \phi_2)^{2.5}}$
Thermal conductivity	$\frac{k_{nf}}{k_f} = \frac{k_s + 2k_f - 2\phi(k_f - k_s)}{k_s + 2k_f + \phi(k_f - k_s)}$	$\frac{k_{hnf}}{k_{bf}} = \frac{k_{s2} + 2k_{bf} - 2\phi_2(k_{bf} - k_{s2})}{k_{s2} + 2k_{bf} + \phi_2(k_{bf} - k_{s2})}$, where $\frac{k_{bf}}{k_f} = \frac{k_{s1} + 2k_f - 2\phi_1(k_f - k_{s1})}{k_{s1} + 2k_f + \phi_1(k_f - k_{s1})}$
Buoyancy coefficient	$(\rho\beta)_{nf} = (1 - \phi)(\rho\beta)_f + \phi(\rho\beta)_s$	$(\rho\beta)_{hnf} = (1 - \phi_2)[(1 - \phi_1)(\rho\beta)_f + \phi_1(\rho\beta)_{s1}] + \phi_2(\rho\beta)_{s2}$

The stream function $\psi(x, r)$ defined as $u = r^{-1}\psi_r$ and $v = -r^{-1}\psi_x$ which satisfies the continuity Eq. (1). Next, we introduce the following dimensionless and similarity variables into Eqs. (1)-(4)

$$\psi = \nu x f(\eta), \quad \eta = \frac{Ur^2}{\nu x}, \quad \theta(\eta) = \frac{T - T_\infty}{T_w - T_\infty}, \quad (5)$$

in which $\theta(\eta)$ is the dimensionless temperature with η as the similarity variable. Introducing the above transformations (5) into Eqs. (2)-(4), yields the following non-linear equations:

$$\frac{1}{B_1 B_2} (2\eta f''' + 2f'') + ff'' + \frac{B_3}{B_2} \frac{\lambda}{4} \theta = 0, \quad (6)$$

$$\frac{2C_1}{Pr C_2} (\eta \theta'' + \theta') + f \theta' = 0. \quad (7)$$

such that

$$B_1 = (1 - \phi_1)^{2.5} (1 - \phi_2)^{2.5},$$

$$B_2 = (1 - \phi_2) \left[(1 - \phi_1) + \phi_1 \frac{\rho_{s1}}{\rho_f} \right] + \phi_2 \frac{\rho_{s2}}{\rho_f},$$

$$B_3 = (1 - \phi_2) \left[(1 - \phi_1) + \phi_1 \frac{(\rho\beta)_{s1}}{(\rho\beta)_f} \right] + \phi_2 \frac{(\rho\beta)_{s2}}{(\rho\beta)_f},$$

and

$$C_1 = \frac{k_{hnf}}{k_f} = \frac{k_{s2} + 2B_1k_f - 2\phi_2(B_1k_f - k_{s2})}{k_{s2} + 2B_1k_f + \phi_2(B_1k_f - k_{s2})} C_0 \quad \text{where} \quad C_0 = \frac{k_{s1} + 2k_f - 2\phi_1(k_f - k_{s1})}{k_{s1} + 2k_f + \phi_1(k_f - k_{s1})}, \quad (8)$$

$$C_2 = (1 - \phi_2) \left[(1 - \phi_1) + \phi_1 \frac{(\rho C_p)_{s1}}{(\rho C_p)_f} \right] + \phi_2 \frac{(\rho C_p)_{s2}}{(\rho C_p)_f}.$$

The transformed boundary conditions are now becoming

$$f(c) = \frac{\varepsilon}{2}c, \quad f'(c) = \frac{\varepsilon}{2}, \quad \theta(c) = 1, \quad (9)$$

$$f'(\eta) = \frac{1}{2}(1 - \varepsilon), \quad \theta(\eta) \rightarrow 0 \quad \text{as} \quad \eta \rightarrow \infty.$$

Here, primes refer to differentiation with respect to η where $\eta = c$ is the needle thickness, λ is the mixed convection parameter, ε is the velocity ratio or moving parameter between the surface and the flow with the composite velocity of $U = U_w + U_\infty$ and Pr is the Prandtl number. These parameters and dimensionless numbers are defined as follows:

$$\lambda = \frac{Gr_x}{Re_x^2}, \quad \varepsilon = \frac{U_w}{U}, \quad Pr = \frac{\nu}{\alpha}, \quad (10)$$

where the local Grashof number is $Gr_x = (1 - C_\infty)(T_w - T_\infty)g\rho_{f,\infty}\beta x^3 / \nu^2\rho_f$ and the local Reynolds number is $Re_x = Ux / \nu$. Noteworthy when $\lambda > 0$, the flow is assisting and when $\lambda < 0$, the flow is opposing.

The practical quantities in this study are the shear stress coefficient or skin friction C_f and the local Nusselt number Nu_x which is defined by

$$C_f = \frac{\mu_{hnf}}{\rho U^2} (u_r)_{r=c} = \frac{4c^{1/2}}{(1 - \phi_1 - \phi_2)^{2.5}} (Re_x)^{-1/2} f''(c), \quad (11)$$

$$Nu_x = -\frac{x}{(T_w - T_\infty)} (T_r)_{r=c} = -\frac{2k_{hnf}}{k_f} c^{1/2} (Re_x)^{1/2} \theta'(c). \quad (12)$$

Stability analysis

To perform a stability analysis, first we consider the unsteady form of Eqs. (2) and (3) as follows:

$$u_t + uu_x + vv_r = \frac{\mu_{hnf}}{\rho_{hnf}} \frac{1}{r} (ru_r)_r + \frac{(\rho\beta)_{hnf}}{\rho_{hnf}} g(T - T_\infty), \quad (13)$$

$$T_t + uT_x + vT_r = \frac{1}{r} \frac{k_{hnf}}{(\rho C_p)_{hnf}} (rT_r)_r. \quad (14)$$

By initiating the dimensionless time variable $\tau = 2Ut / x$, the following new similarity variables are

$$\psi(x, r) = \nu x f(\eta, \tau), \quad \eta = \frac{Ur^2}{\nu x}, \quad \theta(\eta, \tau) = \frac{T - T_\infty}{T_w - T_\infty}, \quad \tau = \frac{2Ut}{x}, \quad (15)$$

It is worth knowing that the function of τ in this analysis is well related to an initial value problem that is consistent with the solution that will be yielded in practice. Using Eq. (15) into Eqs. (13) and (14), we obtain

$$\frac{1}{B_1 B_2} \left[2\eta \frac{\partial^3 f}{\partial \eta^3} + 2 \frac{\partial^2 f}{\partial \eta^2} \right] + \frac{B_3}{B_4} \frac{\lambda}{4} \theta - \frac{\partial^2 f}{\partial \eta \partial \tau} + \tau \frac{\partial f}{\partial \eta} \frac{\partial^2 f}{\partial \eta \partial \tau} + f \frac{\partial^2 f}{\partial \eta^2} - \tau \frac{\partial^2 f}{\partial \eta^2} \frac{\partial f}{\partial \tau} = 0, \quad (16)$$

$$\frac{C_1}{Pr C_2} \left[2\eta \frac{\partial^2 \theta}{\partial \eta^2} + 2 \frac{\partial \theta}{\partial \eta} \right] - \frac{\partial \theta}{\partial \tau} + \tau \frac{\partial f}{\partial \eta} \frac{\partial \theta}{\partial \tau} + f \frac{\partial \theta}{\partial \eta} - \tau \frac{\partial f}{\partial \tau} \frac{\partial \theta}{\partial \eta} = 0, \quad (17)$$

together with the conditions below:

$$f(c, \tau) = \tau \frac{\partial f}{\partial \tau}(c, \tau) + \frac{\varepsilon}{2} c, \quad \frac{\partial f}{\partial \eta}(c, \tau) = \frac{\varepsilon}{2}, \quad \theta(c, \tau) = 1, \quad (18)$$

$$\frac{\partial f}{\partial \eta}(\eta, \tau) \rightarrow \frac{1-\varepsilon}{2}, \quad \theta(\eta, \tau) \rightarrow 0 \quad \text{as} \quad \eta \rightarrow \infty.$$

In the work of Weidman et al. [29], they introduced the following two equations to identify the stability of the solution $f = f_0(\eta)$ and $\theta = \theta_0(\eta)$ for Eqs. (16)-(18).

$$f(\eta, \tau) = f_0(\eta) + e^{-\gamma \tau} F(\eta, \tau), \quad (19)$$

$$\theta(\eta, \tau) = \theta_0(\eta) + e^{-\gamma \tau} G(\eta, \tau),$$

Here, γ is the eigenvalue parameter, $F(\eta, \tau)$ is the small relative to $f_0(\eta)$ and $G(\eta, \tau)$ is the small relative to $\theta_0(\eta)$.

Replacing Eq. (19) into Eqs. (16)-(18), the following linear eigenvalue problems can be written as:

$$\frac{2\eta}{B_1 B_2} F_0''' + \frac{2}{B_1 B_2} F_0'' + \frac{B_3}{B_2} \frac{\lambda}{4} G_0 + f_0 F_0'' + F_0 f_0'' + \gamma F_0' = 0, \quad (20)$$

$$\frac{2\eta C_1}{Pr C_2} G_0'' + \frac{2C_1}{Pr C_2} G_0' + f_0 G_0' + F_0 \theta_0' + \gamma G_0 = 0. \quad (21)$$

Then, the boundary conditions become

$$F_0(c) = 0, \quad F_0'(c) = 0, \quad G_0(c) = 0, \quad (22)$$

$$F_0'(\eta) \rightarrow 0, \quad G_0(\eta) \rightarrow 0 \quad \text{as} \quad \eta \rightarrow \infty.$$

The next step to perform this stability is to assume $\tau = 0$ which corresponds to an initial growth or decay of the solution (19). Hence, in the meantime functions $F(\eta, \tau)$ and $G(\eta, \tau)$ can be rewritten as $F_0(\eta)$ and $G_0(\eta)$, respectively. There are two possibilities to relax the boundary conditions whether on $F_0'(\eta)$ or on $G_0(\eta)$ (see Harris et al. [30]). For the current work, we choose to relax the condition on $F_0'(\eta) \rightarrow 0$ as $\eta \rightarrow \infty$. Thus, we solve the system of Eqs. (20)-(22) together with the condition $F_0''(c) = 1$.

Results and Discussion

The aim of this section is to analyze the variations of solid nanoparticles volume fraction (ϕ_1 and ϕ_2), mixed convection parameter (λ), velocity ratio parameter (ε), thickness of the needle (c) on the dimensionless velocity and temperature profiles, skin friction coefficient and the local Nusselt number or heat transfer rate. To get a clear understanding into a present study, these variations are presented with the help of graphical illustrations as shown in Figs. 2-11. In certain parameter domain, it is reveal that the dual solutions are likely to exist in such domain. The way we differentiate the dual solutions is by considering the upper branch solution as the

solution with higher values of reduced skin friction $f''(c)$ and reduced local Nusselt number $-\theta'(c)$, and lower branch solution as the solution with lesser values of $f''(c)$ and $-\theta'(c)$.

The variation trends of reduced skin friction coefficient and the reduced local Nusselt number as well as the dimensionless velocity and temperature profiles with mixed convection parameter λ are shown in Figs. 2-5 for several types of fluid, namely base fluid ($\phi_1 = \phi_2 = 0$), Al_2O_3 nanofluid ($\phi_1 = 0.1, \phi_2 = 0$) and Al_2O_3 -Cu hybrid nanofluid ($\phi_1 = \phi_2 = 0.1$). As we can see from Figs. 2 and 3, the reduced skin friction coefficient and the local Nusselt number are higher for hybrid nanofluid followed by nanofluid and the base fluid in a certain domain of the λ . This situation occurs due to the decrement of both velocity and thermal boundary layer thicknesses at the needle wall as can be seen in Figs. 4 and 5. The truth is by combining two different kinds of nanoparticles, its physical properties will automatically combine each other, and consequently, provides a good performance on both $f''(c)$ and $-\theta'(c)$. An interesting behavior is found in Figs. 2 and 3 where the dual solutions appear in a certain region of the mixed convection parameter, say $\lambda_c < \lambda \leq 0$. λ_c is the critical value of the parameter λ by which the upper and lower branch solutions connect. In addition, only unique solutions exist for $\lambda > 0$ and no solutions for $\lambda < \lambda_c$.

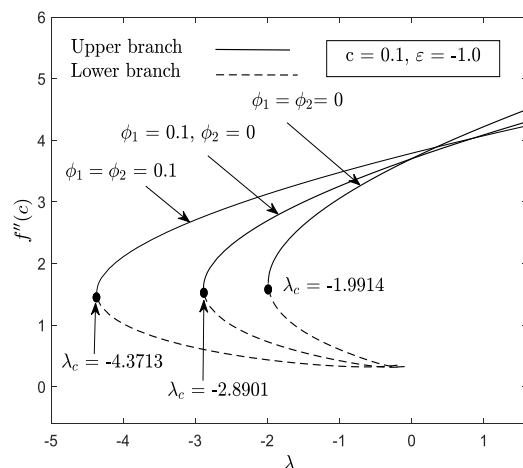


Fig. 2. Variation of ϕ_1 and ϕ_2 on reduced skin friction coefficient $f''(c)$

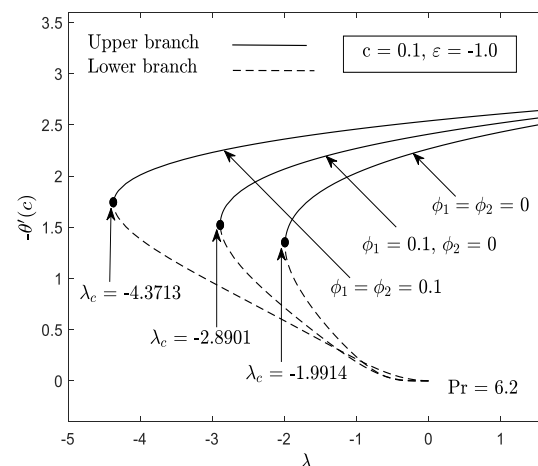


Fig. 3. Variation of ϕ_1 and ϕ_2 on reduced local Nusselt number $-\theta'(c)$

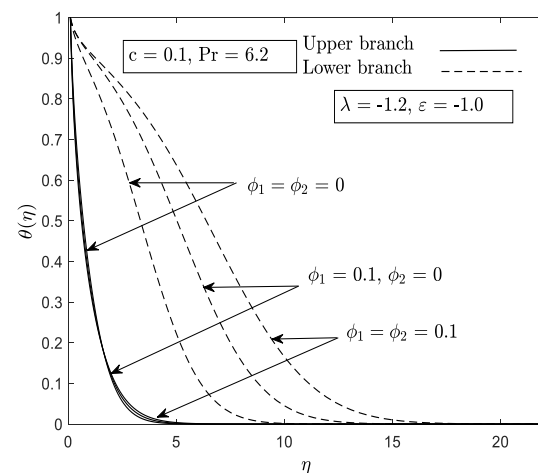
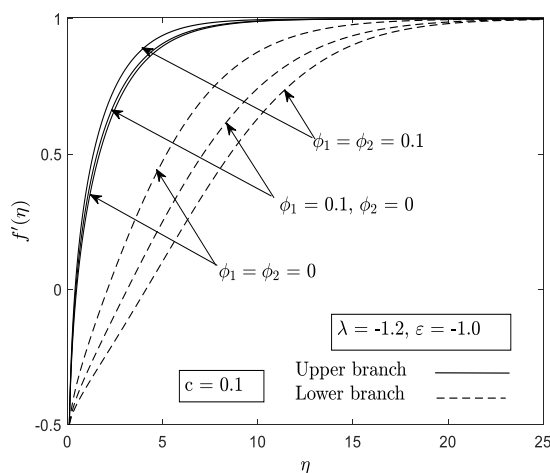


Fig. 4. Variation of ϕ_1 and ϕ_2 on dimensionless velocity profiles $f'(\eta)$

Fig. 5. Variation of ϕ_1 and ϕ_2 on dimensionless temperature profiles $\theta(\eta)$

Figures 6-7 illustrated the influence of the Cu nanoparticle volume fraction ϕ_2 on the reduced skin friction coefficient and the reduced local Nusselt number versus velocity ratio parameter ε when $\phi_1 = 0.1$. It is observed from Figs. 6 and 7 that the reduced skin friction coefficient and local Nusselt number increase with the increase in the parameter ϕ_2 . However, the reduced local Nusselt number only increases in a certain range of $\varepsilon_c < \varepsilon \leq -0.4$. The increment in the Fig. 6 is usually due to the movement of the needle that opposes the direction of free-stream flow ($\varepsilon < 0$) which then causes the velocity ratio to diminish, and as a result delay the movement of the needle and induces more friction on the surface. The addition of more copper nanoparticles in the system gives space to drag force to occur on the needle wall. Accordingly, enhances the magnitude of the reduced skin friction coefficient because of the collision between those nanoparticles. The velocity and temperature distributions for varying copper nanoparticle volume fraction ϕ_2 are presented in Figs. 8-9. In these profiles, we noted that the monotonic behavior of the lines had satisfied the far field boundary conditions (9). Thus, the numerical result obtained for the current work is confirmed.

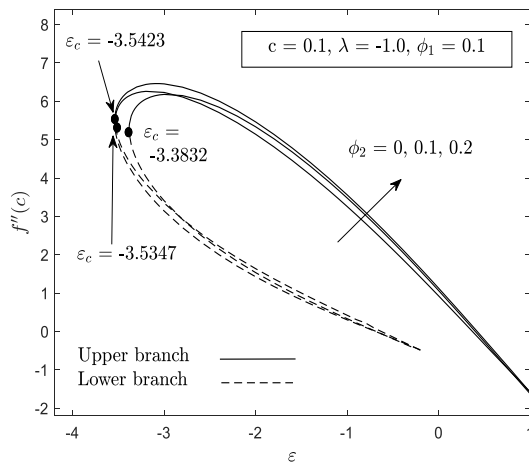


Fig. 6. Variation of ϕ_2 on reduced skin friction coefficient $f''(c)$

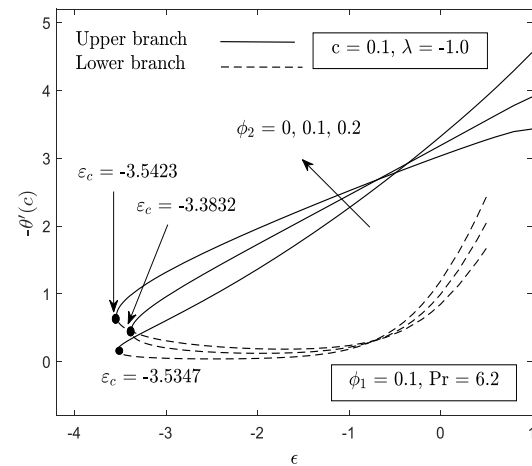


Fig. 7. Variation of ϕ_2 on reduced local Nusselt number $-\theta'(c)$

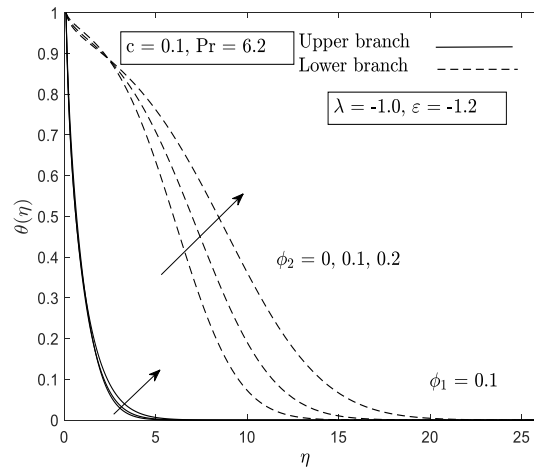
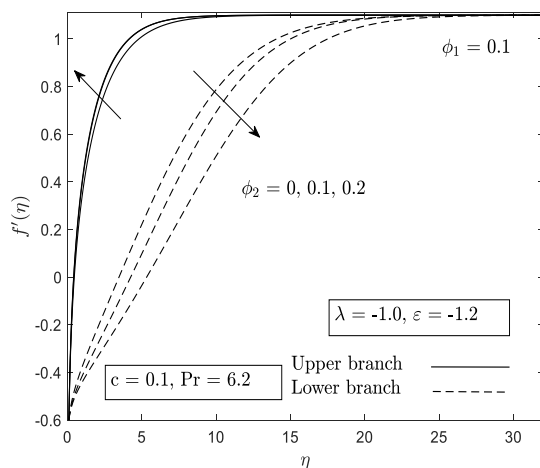


Fig. 8. Variation of ϕ_2 on dimensionless velocity profiles $f'(\eta)$

Fig. 9. Variation of ϕ_2 on dimensionless temperature profiles $\theta(\eta)$

Table 3 is prepared to show the effect of the embedded parameters of interest, namely, ϕ_1 , ϕ_2 , C , λ and ε on the numerical values of skin friction coefficient and the rate of heat transfer (or local Nusselt number) when $Pr = 6.2$. From this table one can see that the skin friction coefficient $C_f(Re_x)^{1/2}$ and local Nusselt number $Nu_x(Re_x)^{-1/2}$ are noticed to increase as the nanoparticle volume fraction of both alumina (ϕ_1) and copper (ϕ_2) increase. Besides that, as expected the values of skin friction and heat transfer rate seem to decrease as the needle thicknesses enlarge. This is due to the increment in the momentum and thermal boundary layer thicknesses as the values of c increase as illustrated in Figs. 10 and 11. In heat transfer process, the thin wall of the needle reduces the time required for the heat to transfer from the wall to the fluid. This consequently increases the heat transfer rate on the wall. An increase in the values of mixed convection parameter λ results in an increase in the skin friction coefficient and heat transfer rate. Meanwhile, the results are invertible as the velocity ratio parameter ε increase.

Table 3: Computational values of skin friction coefficient $C_f(Re_x)^{1/2}$ and local Nusselt number $Nu_x(Re_x)^{-1/2}$ for some values of ϕ_1 , ϕ_2 , C , λ and ε when $Pr = 6.2$

ϕ_1	ϕ_2	c	λ	ε	$f''(c)$	$C_f(Re_x)^{1/2}$	$-\theta'(c)$	$Nu_x(Re_x)^{-1/2}$
0.1	0.1	0.1	-1.4	-1.6	4.523646	9.9959399	1.984125	2.1863580
					4.590731	11.920341	2.070002	2.6012594
					4.631926	14.291457	2.132963	3.0555809
0.1	0				4.212214	6.9336856	1.633697	1.3608258
	0.15				4.613473	11.979393	2.091203	2.6327284
	0.2				4.678829	14.436173	2.170023	3.1207317
	0.1	0.05			8.908554	13.919616	4.329385	3.3733659
		0.15			2.876124	7.7837389	1.124287	1.5173126
		0.2			1.931310	6.0353438	0.651044	1.0145596
		0.1	-1.2		4.601803	10.168644	2.004244	2.2085276
			-1.0		4.677727	10.336414	2.023470	2.2297132
			-0.8		4.751631	10.49972	2.041894	2.2500151
			-1.4	-1.4	4.157435	9.1867203	2.130768	2.3479476
				-1.2	3.763593	8.3164442	2.277014	2.5090998
				-1.0	3.344662	7.3907287	2.423059	2.6700306

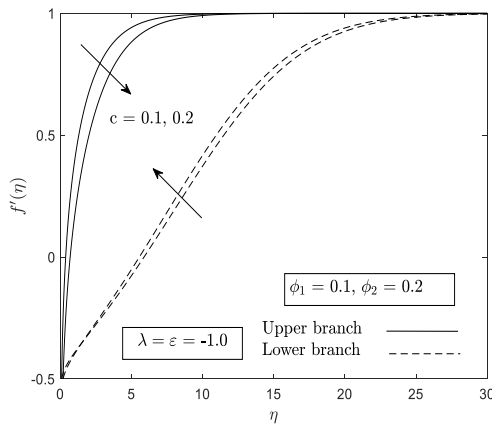


Fig. 10. Variation of c on dimensionless velocity profiles $f'(\eta)$

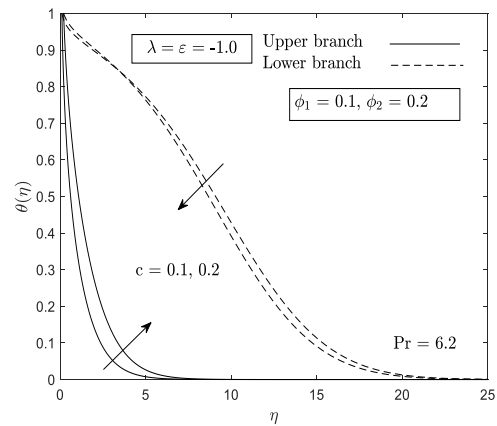


Fig. 11. Variation of c on dimensionless temperature profiles $\theta(\eta)$

For the stability analysis results, we present the minimum eigenvalues γ for different values of solid nanoparticle volume fractions (ϕ_1 and ϕ_2) and mixed convection parameter λ as tabulated in Table 4. It is seen from the table that, the upper branch solutions give the positive values, whereas the lower branch solutions give the negative values. This implies that the upper branch solution is stable and physically realistic, while the lower branch solution is not. Physically, it makes sense because the upper branch solution always takes the highest values which is near to the needle wall. It is worth knowing that the solutions that near to the surface always provides a good physical interpretation.

Table 4: Minimum eigenvalues γ when $Pr = 6.2$, $c = 0.1$ and $\epsilon = -1.0$ for different values of λ , ϕ_1 and ϕ_2

ϕ_1 and ϕ_2	λ	Upper branch solution	Lower branch solution
$\phi_1 = \phi_2 = 0$	-1.92	0.0730	-0.0672
	-1.924	0.0708	-0.0654
	-1.9248	0.0704	-0.0650
$\phi_1 = 0.1, \phi_2 = 0$	-2.84	0.0528	-0.0497
	-2.846	0.0494	-0.0467
	-2.8462	0.0493	-0.0466
$\phi_1 = \phi_2 = 0.1$	-4.33	0.0400	-0.0381
	-4.336	0.0369	-0.0353
	-4.336	0.0364	-0.0348

Conclusions

The consideration of hybrid nanofluid in the steady flow of a moving thin needle with the presence of buoyancy effect has been performed in this work. The followings are the effect of the physical parameters of interest on the characteristics of the fluid flow:

- Hybrid nanofluid gives higher magnitudes of the reduced skin friction coefficient and local Nusselt number than nanofluid and base fluid.
- The presence of high copper nanoparticle volume fraction ϕ_2 increases the reduced skin friction and local Nusselt number when ϕ_1 is constant.

- The existence of the dual solutions is found when the needle opposes the free-stream direction ($\varepsilon < 0$) and when the flow is opposing ($\lambda < 0$).
- The skin friction coefficient and the rate of heat transfer occur between the needle and fluid flow increases when the needle wall become thinner.
- It has been proved by stability analysis that the lower branch solution represents an unstable solution, while upper branch solution represents a stable solution.

Acknowledgement

The authors would like to acknowledge the financial support received from Universiti Putra Malaysia through Putra Grant (GP-IPS/2018/9667900) and from Ministry of Higher Education Malaysia through Fundamental Research Grant Scheme (FRGS/1/2018/STG06/UPM/02/4/55401 55).

References

- [1] J. Sarkar, P. Ghosh, A. Adil, 2015. Sustain. Energy Rev. 43. 164-177.
- [2] S. Suresh, K. Venkitaraj, P. Selvakumar, M. Chandrasekar, 2011. Colloids Surf. A. 388. 41-48.
- [3] D. Madhesh, R. Parameshwaran, S. Kalaiselvam, 2014. Exp. Therm. Fluid Sci. 52. 104-115.
- [4] M. Afrand, D. Toghraie, N. Sina, 2016. International Commun. Heat Mass Transfer. 75. 262-269.
- [5] M.H. Esfe, S.H. Rostamian, A. Alirezaie, 2017. Applied Thermal Engineering. 111. 1202-1210.
- [6] A. Moghadassi, E. Ghomi, F. Parvizian, 2015. International Journal of Thermal Sciences. 92. 50-57.
- [7] S.S.U. Devi, S.A. Devi, 2016. Canadian Journal of Physics. 94. 490-496.
- [8] S.A.M. Mehryan, F.M. Kashkooli, M. Ghalambaz, A.J. Chamkha, 2017. Advanced Powder Tech. 28. 2295-2305.
- [9] M.A. Mansour, S. Siddiqua, R.S.R. Gorla, A.M. Rashad, 2018. Thermal Science and Engineering Progress. 6. 57-71.
- [10] I. Waini, A. Ishak, I. Pop, 2019. International Journal of Heat and Mass Transfer. 136. 288-297.
- [11] M. Sheikholeslami, S.A.M. Mehryan, A. Shafee, M.A. Sheremet, 2019. Journal of Molecular Liquids. 277. 388-396.
- [12] L.L. Lee, 1967. The Physics of Fluids. 10. 1820-1822.
- [13] J.P. Narain, S.M. Uberoi, 1972. ASME Journal of Heat Transfer. 94. 240-242.
- [14] J.P. Narain, S.M. Uberoi, 1973. International Journal of Heat and Mass Transfer. 16. 1505-1512.
- [15] J.L.S. Chen, T.N. Smith, 1978. ASME Journal of Heat Transfer. 100. 358-362.
- [16] A. Ishak, R. Nazar, I. Pop, 2007. Chinese Physics Letters. 24. 2895-2897.
- [17] S. Ahmad, N.M. Arifin, R. Nazar, I. Pop, 2008. International Communications in Heat Mass Transfer. 35. 157-162.
- [18] M.I. Afridi, M. Qasim, 2018. International Journal of Thermal Science. 123. 117-128.
- [19] T. Grosan, I. Pop, 2011. Journal of Heat Transfer. 133. 054503.
- [20] T. Hayat, M.I. Khan, M. Farooq, T. Yasmeen, A. Alsaedi, 2016. Journal of Molecular Liquids. 224. 786-791.
- [21] P.M. Krishna, R.P. Sharma, N. Sandeep, 2017. Nuclear Engineering and Technology. 49. 1654-1659.

- [22] R. Ahmad, M. Mustafa, S. Hina, 2017. Chinese Journal of Physics. 55. 1264-1274.
- [23] S.K. Soid, A. Ishak, I. Pop, 2017. Applied Thermal Engineering. 114. 58-64.
- [24] S.N.A. Salleh, N. Bachok, N.M. Arifin, F.M. Ali, 2019. Symmetry. 11. 543.
- [25] J.H. Merkin, 1985. Journal of Engineering Mathematics. 20. 171-179.
- [26] S.N.A Salleh, N. Bachok, N.M. Arifin, 2019. Malaysian Journal of Science Special Issue. 1. 19-32.
- [27] I. Waini, A. Ishak, I. Pop, 2019. Chinese Journal of Physics. 60. 651-658.
- [28] N.S. Khashi'ie, N.M. Arifin, E.H. Hafidzuddin, N. Wahi, M.R. Ilias, 2019. Universal Journal of Mech. Eng. 7. 206-214.
- [29] P.D. Weidman, D.G. Kubitschek, A.M.J. Davis, 2006. International Journal of Engineering Science. 44. 730-737.
- [30] S.D. Harris, D.B. Ingham, I. Pop, 2009. Transport of Porous Media. 77. 267-285.

STABILITY ANALYSIS OF MHD FLOW AND HEAT TRANSFER PASSING A PERMEABLE EXPONENTIALLY SHRINKING SHEET WITH PARTIAL SLIP AND THERMAL RADIATION

Nurul Shahirah Mohd Adnan¹

Norihan Md Arifin^{1,2}

Norfifah Bachok^{1,2}

Fadzilah Md Ali^{1,2}

¹Institute for Mathematical Research, Universiti Putra Malaysia, 43400 UPM Serdang, Selangor, Malaysia

²Department of Mathematics, Faculty of Science, Universiti Putra Malaysia, 43400 UPM Serdang

Abstract: *We have investigated the steady magnetohydrodynamic (MHD) flow and heat transfer induced by an exponentially shrinking sheet with partial slip, thermal radiation and suction. A third order ordinary differential equation corresponding to the momentum equation is derived using similarity transformation method. A second order ordinary differential equation corresponding to the energy equation also is derived using similar method. Then, we used Matlab software to obtain the numerical results. The effects of magnetic parameter and mass suction parameter are analyzed and presented. From the results, we notice that first and second solutions exist in certain range of suction parameter. Hence, we continue further in performing a stability analysis*

Keywords: *Boundary layer; Magnetic Field; Slip; Thermal radiation*

Introduction

A few decades ago, several authors have studied about heat transfer and fluid flow passing stretching or shrinking sheet. Wang [1] has investigated the problem of a shrinking sheet with the stagnation flow. Saleh et al. [2] investigated steady mixed convection stagnation flow towards a vertical shrinking sheet. Hafidzuddin et al. [3] analyzed the unsteady problem for the three-dimensional flow of the permeable stretching/shrinking surface. In recent years, the studies of magnetohydrodynamic (MHD) have received a great of attention due to its important in many field. Some studies about MHD can be found in Ishak [4], Khan et al. [5], Abbas et al. [6], Ali et al. [7] and Kumar and Varma [8].

Recently, Ahmad and Pop [9], Zaimi and Ishak [10], Naramgari and Sulochana [11] and Sharma et al. [12] found dual solutions in their study. The stability analysis of dual solutions have been performed by many reseachers. According to Merkin [13], positive eigenvalues mean the solution is stable and negative eigenvalue imply otherwise. Ishak [14] and Yasin et al. [15] study the case of shrinking sheet and they concluded that second solution is unstable while first solution is stable. Nazar et al. [16] investigated the stability analysis of three-dimensional flow induced with shrinking sheet in a Cu-water nanofluid. They also found that second solution is unstable and vice versa.

Based on previous study, the present investigation extends the problem by Sharma et al [12] to the case of magnetohydrodynamic (MHD) flow over an exponentially shrinking sheet. Stability analysis is conducted to identify the stability of dual solutions obtained.

Problem Formulation

A steady two-dimensional case of MHD flow passing an exponentially permeable shrinking sheet is considered. The governing equations are written in usual notation as:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (1)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \nu \frac{\partial^2 u}{\partial y^2} - \frac{\sigma \beta_0^2}{\rho} u \quad (2)$$

$$u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \alpha \frac{\partial^2 T}{\partial y^2} - \frac{1}{\rho C_p} \frac{\partial q_r}{\partial y} \quad (3)$$

where u is the velocity component along x axes while v is the velocity component along y axes. α , T , ν , ρ and C_p are thermal diffusivity, the temperature of fluid, kinematic viscosity, density and specific heat at constant pressure and the conditions at the boundary are

$$v = v_w(x), u = U + Nv \frac{\partial u}{\partial y}, T = T_w + D \frac{\partial T}{\partial y} \text{ at } y = 0 \quad (4)$$

$$u \rightarrow 0, T \rightarrow T_\infty \text{ as } y \rightarrow \infty$$

where $v_w(x) = V_0 \exp(x/2L)$, $U = -U_0 \exp(x/L)$ and $T_w = T_0 \exp(x/2L)v_0$. v_0 is the mass flux velocity with $v_0 < 0$ (corresponds to the suction) and $v_0 > 0$ (corresponds to the injection). U represents the velocity of the shrinking, T_w represents the temperature of variable at the sheet, L represents the length, U_0 represents the velocity and T_0 represents the temperature. Here $N = N_1 \exp(-x/2L)$ represents velocity slip factor while $D = D_1 \exp(-x/2L)$ is the thermal slip factor. Both N_1 and D_1 are initial value for velocity and thermal slip factor, respectively. $N = D = 0$ is the no-slip case. For the radiation flux q_r , we used Rosseland approximation and we can write

$$q_r = -\frac{4\sigma^*}{3k^*} \frac{\partial T^4}{\partial y} \quad (5)$$

where k^* is the mean absorption coefficient and σ^* is Stefan-Boltzman constant. We further assumed that the term T^4 can be expressed as a linear function of temperature itself. Thus, T^4 can be expanded using Taylor series. Next, the terms of higher-order are ignored and yields

$$T^4 \cong 4T_\infty^3 T - 3T_\infty^4 \quad (6)$$

Introducing stream function ψ which always be denoted as $u = \partial \psi / \partial y$ and $v = -\partial \psi / \partial x$. Next, we introduce the following similarity variables

$$\psi = \sqrt{2\nu L U_0} f(\eta) \exp(x/2L), T = T_0 \exp(x/2L) \theta(\eta),$$

$$\eta = y \sqrt{\frac{U_0}{2\nu L}} \exp(x/2L).$$

(7)

Using equation (7), we get

$$u = U_0 \exp(x/L) f'(\eta), v = -\sqrt{\frac{\nu U_0}{2L}} \exp(x/2L) [f(\eta) + \eta f'(\eta)]$$

(8)

Primes here denote differentiation with respect to η . We are used equations (5) and (7). Thus, equations (2) and (3) become

$$f''' + ff'' - 2f'^2 - Mf' = 0 \quad (9)$$

$$\frac{1}{Pr} \left(1 + \frac{4}{3} R \right) \theta'' + f \theta' - f' \theta = 0 \quad (10)$$

where $M = \frac{2L\sigma\beta_0^2}{u_0\rho e^{x/L}}$ is parameter for magnetic, $Pr = \frac{\nu}{\alpha}$ is the Prandtl number and $R = \frac{4\sigma^* T_\infty^3}{k_m k^*}$

is the parameter for radiation. The transformed boundary conditions are:

$$f(0) = S, f'(0) = -1 + \lambda f''(0), \theta(0) = 1 + \delta \theta'(0) \quad (11)$$

$$f'(\eta) \rightarrow 0, \theta(\eta) \rightarrow 0 \text{ as } \eta \rightarrow \infty$$

where $\lambda = N_1 \sqrt{U_0 \nu / 2L} (> 0)$, $\delta = D_1 \sqrt{U_0 / 2\nu L} (> 0)$ and $S = -V_0 \sqrt{U_0 / 2\nu L}$. λ is velocity slip parameter, δ for thermal slip parameter and S is the parameter for the constant mass transfer. $S > 0$ corresponds to the suction while $S < 0$ corresponds to the injection.

The main physical quantities in this study are the skin friction coefficient, C_f and local Nusselt number, Nu_x . C_f and Nu_x are defined as follows

$$C_f = \frac{\mu}{\rho [U_0 \exp(x/L)]^2} \left(\frac{\partial u}{\partial y} \right)_{y=0}, Nu_x = \frac{L}{T_0 \exp(x/2L)} \left(-\frac{\partial T}{\partial y} + q_r \right)_{y=0} \quad (12)$$

Substituting equations (5) and (7) into equation (12) and we obtain

$$(2Re_x)^{1/2} C_f = f''(0), (2/Re_x)^{1/2} Nu_x = -(1+R)\theta'(0) \quad (13)$$

where $Re_x = (U_0 L / \nu) \exp(x/L)$ is the local Reynolds number.

Stability Analysis

In this study, we are using the stability analysis developed by Weidman et al. [17]. According to them, a variable τ has to be introduced. To perform stability analysis, unsteady cases are considered. Hence, equations (2) and (3) will be replaced by

$$\frac{\partial u}{\partial t} u \frac{\partial u}{\partial x} + \nu \frac{\partial u}{\partial y} = \nu \frac{\partial^2 u}{\partial y^2} - \frac{\sigma \beta_0^2}{\rho} u \quad (14)$$

$$\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + \nu \frac{\partial T}{\partial y} = \alpha \frac{\partial^2 T}{\partial y^2} - \frac{1}{\rho C_p} \frac{\partial q_r}{\partial y} \quad (15)$$

Here, τ refers to the time. Next, we introduced the new dimensionless variables for the unsteady problem based on the variables (7) previously

$$\psi = \sqrt{2\nu L U_0} f(\eta, \tau) \exp(x/2L), T = T_0 \exp(x/2L) \theta(\eta, \tau), \quad (16)$$

$$\eta = y \sqrt{\frac{U_0}{2\nu L}} \exp(x/2L), \tau = \frac{U_0 t}{2L} \exp(x/L)$$

So that, equations (14) and (15) change to the following equations

$$\frac{\partial^3 f}{\partial \eta^3} + f \frac{\partial^2 f}{\partial \eta^2} - 2 \left(\frac{\partial f}{\partial \eta} \right)^2 - \frac{\partial^2 f}{\partial \eta \partial \tau} - M \frac{\partial f}{\partial \eta} = 0 \quad (17)$$

$$\frac{1}{Pr} \left(1 + \frac{4}{3} R \right) \frac{\partial^2 \theta}{\partial \eta^2} + f \frac{\partial \theta}{\partial \eta} - \frac{\partial f}{\partial \eta} \theta - \frac{\partial \theta}{\partial \tau} = 0 \quad (18)$$

with the following boundary conditions

$$f(0, \tau) = s, \frac{\partial f}{\partial \eta}(0, \tau) = -1 + \lambda \frac{\partial^2 f}{\partial \eta^2}, \theta(0, \tau) = 1 + \delta \frac{\partial \theta}{\partial \eta} \quad (19)$$

$$\frac{\partial f}{\partial \eta}(\eta, \tau) \rightarrow 0, \theta(\eta, \tau) \rightarrow 0 \text{ as } \eta \rightarrow \infty$$

We can test the stability of the solution $f(\eta) = f_0(\eta)$ and $\theta(\eta) = \theta_0(\eta)$. It is satisfying the boundary-value problem (9)-(11). We can write

$$f(\eta, \tau) = f_0(\eta) + e^{-\gamma \tau} F(\eta, \tau), \theta(\eta, \tau) = \theta_0(\eta) + e^{-\gamma \tau} G(\eta, \tau), \quad (20)$$

Here, γ is an unknown eigenvalue. $F(\eta, \tau)$ and $G(\eta, \tau)$ are small relative to $f_0(\eta)$ and $\theta_0(\eta)$. After that, equation (20) is substituted into equations (17) and (18) and obtained as below

$$\frac{\partial^3 F}{\partial \eta^3} + f_0 \frac{\partial^2 F}{\partial \eta^2} + f_0'' F - (4f_0' + M - \gamma) \frac{\partial F}{\partial \eta} - \frac{\partial^2 F}{\partial \eta \partial \tau} = 0, \quad (21)$$

$$\frac{1}{\text{Pr}} \left(1 + \frac{4}{3} R \right) \frac{\partial^2 G}{\partial \eta^2} + f_0 \frac{\partial G}{\partial \eta} + \theta_0' F - f_0' G - \theta_0 \frac{\partial F}{\partial \eta} + \gamma G - \frac{\partial G}{\partial \tau} = 0 \quad (22)$$

together with the following boundary conditions

$$F(0, \tau) = 0, \frac{\partial F}{\partial \eta}(0, \tau) = \lambda \frac{\partial^2 F}{\partial \eta^2}(0, \tau), G(0, \tau) = \delta \frac{\partial G}{\partial \eta}(0, \tau)$$

$$\frac{\partial F}{\partial \eta}(\eta, \tau) \rightarrow 0, G(\eta, \tau) \rightarrow 0 \text{ as } \eta \rightarrow \infty \quad (23)$$

After that, we set $\tau = 0$, $F = F_0(\eta)$ and $G = G_0(\eta)$ to obtain the following:

$$F_0''' + f_0 F_0'' + f_0'' F_0 - (4f_0' + M - \gamma) F_0' = 0, \quad (24)$$

$$\frac{1}{\text{Pr}} \left(1 + \frac{4}{3} R \right) G_0'' + f_0 G_0' + \theta_0' F_0 - f_0' G_0 - \theta_0 F_0' + \gamma G_0 = 0 \quad (25)$$

alongside the boundary conditions:

$$F_0(0) = 0, F_0'(0) = \lambda F_0''(0), G_0(0) = \delta G_0'(0) \quad (26)$$

$$F_0'(\eta) \rightarrow 0, G_0(\eta) \rightarrow 0 \text{ as } \eta \rightarrow \infty$$

By relaxing a boundary condition on $G_0(\eta)$ or $F_0'(\eta)$ [17], we can determine the possible values of γ . The aim of this procedure is to determine the stability of the problem and for this present problem, we relax $F_0'(\infty) \rightarrow 0$. Then, we solved the system with the new condition at the boundary $F_0''(0) = 1$.

Methodology

By apply similarity transformation method, the three partial differential equations were transformed to become two ordinary differential equations. After that, we solved numerically equations (9) and (10) together with boundary conditions (11) by using bvp4c function in Matlab. In this study, we fixed the values of $R = \lambda = \delta = 0.1$ and $\text{Pr} = 0.7$. To ensure that our results are correct, a comparison has been made with the results obtained by Sharma et al. [18]. We found that the comparisons are in excellent agreement for $S = 2.20$ and $M = 0$. Figure 1

represents the skin friction coefficient at the surface for selected values of M while Figure 2 shows the heat transfer rate at the surface for selected values of M . From both figures, we noticed that there are admit dual solutions when $s > s_c$, where s_c is the critical value of suction. To delay or accelerate the separation of the boundary layer, we can set magnetic parameter M as controller parameter because when the value of M is increases, the range of the solution will be increasing. Thus, we can conclude that the changes of the rate of heat transfer and surface stress are influenced by magnetic parameter M .

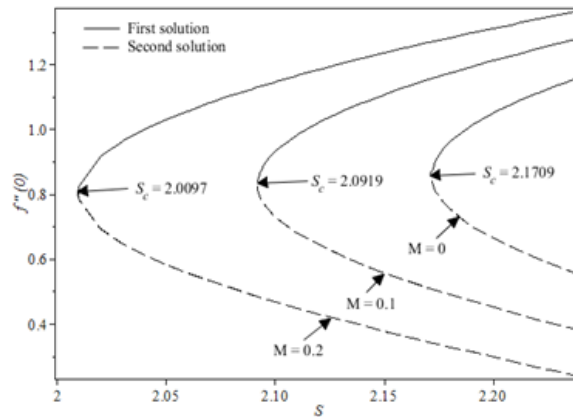


Figure 2. Variation of $f''(0)$ for several values of M with s

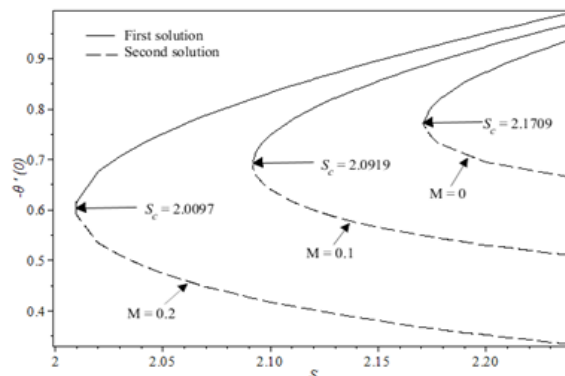


Figure 3. Variation of $-\theta'(0)$ for several values of M with s

Further, Figures 3-6 display profiles to support the dual solutions that exist in Figures 1 and 2. Figures 3 and 5 shows profiles for the velocity while both Figures 4 and 6 show the temperature profiles. All profiles show the thickness of the boundary layer for the second solution always presents greater compared to the first solution. There are display first and second solutions. Hence, a stability analysis is conducted to know which one is unstable and stable. Table 1 presents the smallest eigenvalues γ for selected values of M and s . From Table 1, it is clearly shown that the smallest eigenvalue γ is positive for the first solution and it is in opposite value for the second solution. Thus, the first solution is linearly stable while second solution is linearly unstable.

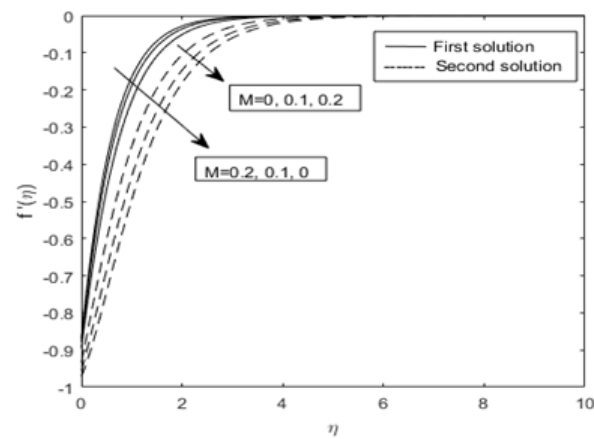


Figure 4. Velocity profile for different values of M with $S = 2.20$

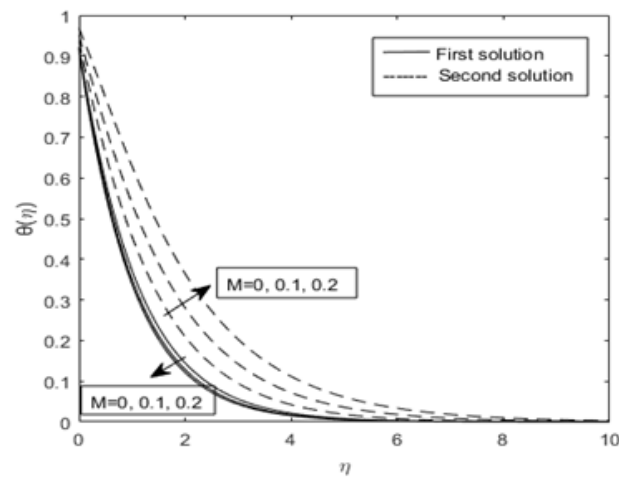


Figure 5. Temperature profile for different values of M with $S = 2.20$

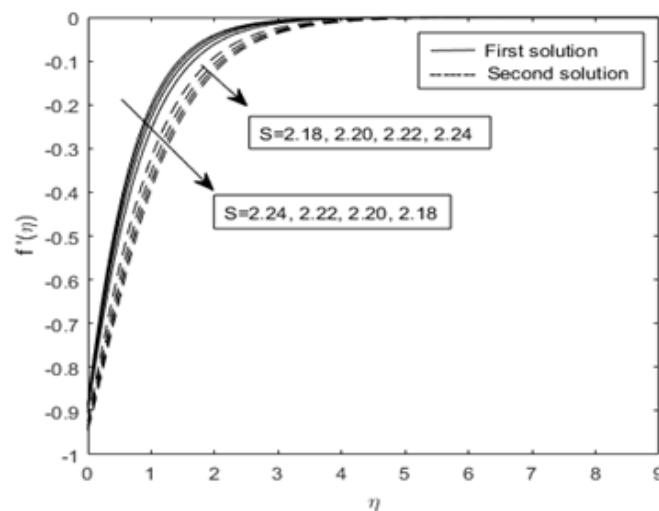


Figure 6. Velocity profile for different values of S with $M = 0$

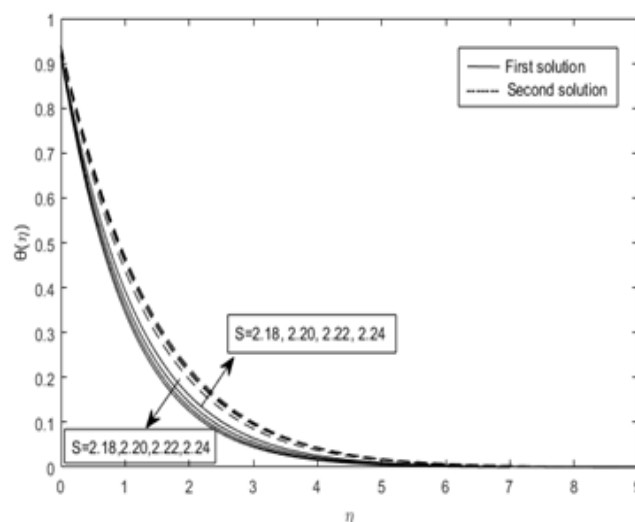


Figure 7. Temperature profile for different values of S with $M = 0$

Table 1. Smallest eigenvalues γ for some values of M and S

M	S	First Solution	Second Solution
0	2.18	0.3015	-0.2985
	2.20	0.3952	-0.3903
	2.22	0.4758	-0.4671
	2.24	0.5427	-0.5316
0.1	2.18	0.8358	-0.5289
	2.20	0.8774	-0.6394
	2.22	0.8987	-0.6886
	2.24	0.9293	-0.7349
0.2	2.18	1.1906	-0.7827
	2.20	1.2384	-0.8243
	2.22	1.3251	-0.8643
	2.24	1.3749	-0.9028

Conclusion

A study on magnetohydrodynamic (MHD) flow induced by shrinking sheet with partial slip, thermal radiation and suction was carried. Similarity transformations were introduced to reduce the partial differential equations into ordinary differential equations. The bvp4c solver was then used to solve the ordinary differential equations and boundary conditions. The results showed that increasing the magnetic parameter, it contributes to increase the surface stress and heat transfer rate. Then, also we found that there exists dual solution. The stability analysis was done to determine the stability of dual solutions. Based on the results, the first solution is stable and has physical meaning.

Acknowledgements

The authors gratefully acknowledge the financial support received in the form of the Putra Grant from Universiti Putra Malaysia Vot number 9624700

References

- [1] Wang, C.Y., "Stagnation flow towards a shrinking sheet." *International Journal of Non-Linear Mechanics*, 43, (2008): 377-382.
- [2] Saleh S.H.M., N.M. Arifin, R. Nazar, F.M. Ali, and I. Pop, "Mixed convection stagnation flow towards a vertical shrinking sheet." *International Journal of Heat and Mass Transfer*, 73, (1985): 839-848.
- [3] Hafidzuddin, E.H., R. Nazar, N.M. Arfin, and I. Pop, "Stability analysis of unsteady three-dimensional viscous flow over permeable stretching/shrinking surface.", *Journal of Quality Measurement and Analysis*, 11 (2015): 19-31.
- [4] Ishak, A., "MHD boundary layer flow due to an exponentially stretching sheet with radiation effect." *Sains Malaysiana*, 40 (2011): 391-395.
- [5] Khan, M.S, I. Karim, L.E. Ali, and A. Islam, "Unsteady MHD free convection boundary-layer flow of a nanofluid along a stretching sheet with thermal radiation and viscous dissipation effects." *International Nano Letters*, 2 (2012): 1-9
- [6] Abbas, Z., M. Sajid, and T. Hayat, "MHD boundary-layer flow of an upper-convected Maxwell fluid in a porous channel." *Theor. Computational Fluid Dynamics*, 20 (2006): 229-238.
- [7] F.M. Ali, R. Nazar, N.M. Arifin, and I. Pop, MHD stagnation-point flow and heat transfer towards stretching sheet with induced magnetic field, *Applied Mathematics and Mechanics*, 32, (2011): 409-418.
- [8] Kumar R. V. M. S. S. K., and S. V. K. Varma, "MHD Boundary Layer Flow of Nanofluid Through a Porous Medium over a Stretching Sheet with Variable Wall Thickness: Using Cattaneo-Chritov heat flux model." *J. Theoretical Appl. Mech.*, 48 (2018): 72-92.
- [8] Ahmad, S., and I. Pop, "Melting effect on mixed convection boundary layer flow about a vertical surface embedded in a porous medium: Opposing flows case," *Transp. Porous Med.*, 102 (2014): 317-323.
- [9] Zaimi, K., and Ishak, A., "Boundary layer flow and heat transfer over a permeable stretching/shrinking sheet with a convective boundary condition." *Journal of Applied Fluid Mechanics*, 8 (2015): 499-505.
- [10] Naramgari, S., Sulochana, C., "Dual solutions for MHD stagnation-point flow of a nanofluid over a stretching surface with induced magnetic field." *Int. J. Sci. Eng.*, 9 (2015): 1-8.
- [11] Merkin, J.H., "On dual solutions occurring in mixed convection in a porous medium," *J. Eng. Math.*, 20 (1985): 171-179.
- [12] Sharma, R. A., Ishak, R. Nazar, and I. Pop, "Boundary layer flow and heat transfer over a permeable exponentially shrinking sheet in the presence of thermal radiation and partial slip." *J. Applied Fluid Mechanics*, 7 (2014): 1735-3645.
- [13] Ishak, A., "Flow and heat transfer over a shrinking sheet: a stability analysis." *Int. J. Mech. Aeros. Ind. Mech. Eng.*, 8 (2014) : 902-906.
- [14] Yasin, M.H.M, A. Ishak, and I. Pop, "Boundary layer flow and heat transfer past a permeable shrinking surface embedded in a porous medium with a second-order slip: a stability analysis." *Appl. Thermal Eng.*, 115 (2016): 1407-1411.
- [15] Nazar, R. A., Noor, K. Jafar, and I. Pop, "Stability analysis of three-dimensional flow and heat transfer over a permeable shrinking surface in a Cu-water nanofluid." *Int. J. Math. Phy. Elec. Comp. Eng.*, 8 (2014): 782-788.
- [16] Weidman, P.D., D.G. Kubitschek, and A.M.J. Davis, "The effect of transpiration on self-similar boundary layer flow over moving surfaces." *Int. J. Eng. Sci.*, 44 (2006) : 730-737.



- [17] Harris, S.D., D.B. Ingham, and I. Pop, "Mixed convection boundary-layer flow near the stagnation point on a vertical surface in a porous medium: brinkman model with slip." *Transport Porous Media*, 77 (1985):267-285.

MAGNETOHYDRODYNAMIC (MHD) SLIP DARCY FLOW OF VISCOELASTIC FLUID OVER A STRETCHING SHEET AND HEAT TRANSFER WITH THERMAL RADIATION AND VISCOUS DISSIPATION

Nur Syahirah Wahid¹
Mohd Ezad Hafidz Hafidzuddin²
Norihan Md Arifin^{1,3}
Mustafa Turkylmazoglu⁴
Nor Aliza Abd Rahmin¹

¹Department of Mathematics, Universiti Putra Malaysia, Serdang Selangor 43400 UPM, Malaysia

²Centre of Foundation Studies For Agricultural Science, Universiti Putra Malaysia, Serdang Selangor 43400 UPM, Malaysia (Email: ezadhafidz@upm.edu.my)

³Institute for Mathematical Research, Universiti Putra Malaysia, Serdang Selangor 43400 UPM, Malaysia

⁴Mathematics Department, University of Hacettepe, Beytepe, Ankara, Turkey

Abstract: *The magnetohydrodynamic (MHD) slip Darcy flow of viscoelastic fluid over a stretching surface in a porous medium with the presence of thermal radiation and viscous dissipation is examined. The results for two viscoelastic fluids which is elastico-viscous fluid and second grade fluid are obtained and compared. The governing partial differential equations are reduced to non-linear ordinary differential equations with the aid of similarity transformation, which are then solved analytically by using exact analytical method. The effects of the physical parameters on the velocity and temperature fields are presented through graphs and are discussed. Skin friction and heat transfer coefficients are computed and analyzed.*

Keywords: *Viscoelastic fluid, Velocity slip, Magnetohydrodynamic, Porous medium, Viscous dissipation, Thermal radiation*

Introduction

The study towards the flow on boundary layer of non-Newtonian fluids has been the subject of great interest to investigators and researchers. Viscoelastic fluid is a common form of non-Newtonian fluid, which this fluid having the characteristic that shows both viscous and elastic properties under some circumstances. This viscoelastic reaction and the fluid flow past a stretching sheet are so significant in some real life applications especially in the engineering field, for instance, in the paper production, extrusion of plastic sheets, glass blowing, and metal spinning.

The flow of viscoelastic fluid past a stretching sheet has been studied by Rajagopal *et al.* [1]. Later, Andersson [2] has added the magnetic effects towards the flow of a viscoelastic fluid past a stretching sheet and he obtained the exact analytical solution of the governing non-linear boundary layer equation. Consequently, Ariel [3] extended the problem by taking into account the flow with the existence of suction and found the exact solution. The investigation on the steady magnetohydrodynamic (MHD) viscoelastic fluid flow over a semi-infinite, impermeable stretching sheet with the presence of thermal radiation and internal heat generation or absorption is investigated by Datti *et al.* [4]. Moreover, Liu [5] presented the exact analytical solutions for the flow and heat transfer of second grade fluid over a stretching sheet with the presence of magnetic and viscous dissipation. Khan and Sanjayanand [6] has

examined the viscoelastic boundary layer flow and heat transfer over an exponential stretching sheet. Viscous dissipation has been considered in the heat transfer and the obtained exact solution which then being compared with the numerical solution.

Later, Cortell [7] performed an analysis on the flow and heat transfer of viscoelastic fluid by considering the magnetic field parameter, viscous dissipation and suction parameters. Moreover, partial slip has been taken into account into the flow of viscoelastic fluid past a stretching sheet, where this analysis has been carried out by Ariel *et al.* [8]. Next, closed form of analytical solution has been obtained by Khan [9] on the heat transfer in a viscoelastic fluid flow over a stretching surface with the consideration of heat source, suction and radiation. Hayat *et al.* [10] then studied the magnetohydrodynamic (MHD) flow of a second grade fluid with thermal radiation effect by the implementation of homotopy analysis method (HAM). Abel *et al.* [11] studied the heat transfer in viscoelastic fluid over a stretching sheet by considering the effects of viscous dissipation and non-uniform heat source in both types of heating process.

Further, Abel and Manesha [12] have added the thermal radiation parameter as the extension of the analysis. The analytic solution of magnetohydrodynamic (MHD) flow and heat transfer over a stretching sheet for both second grade and Walters' liquid B fluids has been solved analytically by Chen [13]. Turkyilmazoglu [14] studied the slip flow of viscoelastic fluid past a stretching surface in magnetic field and have found multiple analytical solutions. The analytical solution of magnetohydrodynamic (MHD) viscoelastic fluid flow and heat transfer in a parallel plate channel with a stretching wall is obtained by employing homotopy analysis method [15]. Then, Turkyilmazoglu [16] has extended his work [14] by analyzing both types of viscoelastic fluids. Later, Nayak *et al.* [17] have investigated the heat and mass transfer of the viscoelastic fluid bounded by a stretching sheet with consideration of various parameters and conditions. Furthermore, Nadeem *et al.* [18] have considered Cattaneo Christov heat flux in the flow of viscoelastic fluid over a stretching surface with the presence of Newtonian heating and porosity parameter. Optimal homotopy analysis method has been used to solve the problem. Spectral homotopy analysis approach is being implemented by Fagbade *et al.* [19] in their investigation on natural convection flow of viscoelastic fluid with thermal and heat source or sink. Recently, the study about viscoelastic fluid is being investigated by Chen *et al.* [20], which they have developed a double Maxwell model to examine the flow of magnetohydrodynamic viscoelastic fluid over a stretching sheet by using infinite difference method. So, through the review on these literatures, we want to extend the studies of the viscoelastic fluid (non-Newtonian fluid) with the consideration of various parameters and conditions. In order to solve the problem, we have implemented the exact analytical method.

Thus, the major purpose of this investigation is to formulate a mathematical model for magneto-hydrodynamic (MHD) slip Darcy boundary layer flow of viscoelastic fluid past a linear stretching surface with the presence of thermal radiation and viscous dissipation in porous medium. The analytical solutions for the velocity and the temperature distributions are obtained using the exact method. The profiles are plotted and discussed for variations of pertinent parameters such as viscoelastic, porosity, magnetic, and thermal radiation parameters, as well as the Prandtl and Eckert numbers. The skin friction and heat transfer coefficients have been computed and analyzed through numerical data, and then being compared to other previous results to verify the technique of the method that we used.

Modeling

We consider two-dimensional (2D) flow of viscoelastic fluid bounded by a linear stretching surface. The incompressible viscoelastic fluid saturates the porous space characterizing Darcy model. Here, x - axis is parallel to stretchable surface and we suppose $u_w(x) = cx$ defines

the stretching velocity along the x – direction. Meanwhile, y – axis is perpendicular to x – axis. An applied uniform magnetic field of strength B_0 is encountered parallel to the y – axis. It should be mentioned that we have modified the model by Nayak *et al.* [17], by considering the presence of velocity slip parameter. With all of these assumptions, the boundary layer equations governing the flow of viscoelastic fluid and heat transfer in the presence of magnetohydrodynamic, porous medium, viscous dissipation and thermal radiation can be written as follows [17]:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0, \quad (1)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \nu \frac{\partial^2 u}{\partial y^2} + \frac{k_0}{\rho} \left(v \frac{\partial^3 u}{\partial y^3} + u \frac{\partial^3 u}{\partial x \partial y^2} - \frac{\partial u}{\partial y} \frac{\partial^2 u}{\partial x \partial y} + \frac{\partial u}{\partial x} \frac{\partial^2 u}{\partial y^2} \right) - \frac{\nu}{K^*} u - \frac{\sigma B_0^2 u}{\rho}, \quad (2)$$

$$u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \alpha \left(\frac{\partial^2 T}{\partial y^2} \right) + \frac{\mu}{\rho C_p} \left(\frac{\partial u}{\partial y} \right)^2 + \frac{k_0}{\rho C_p} \left(\frac{\partial u}{\partial y} \frac{\partial}{\partial y} \left(u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) \right) - \frac{1}{\rho C_p} \frac{\partial q_r}{\partial y} + \frac{\sigma B_0^2 u^2}{\rho C_p}, \quad (3)$$

subject to the associated boundary conditions:

$$u = u_w(x) = cx + l \frac{\partial u}{\partial y}, \quad v = 0, \quad T = T_w(x) = T_\infty + A \left(\frac{x}{x_L} \right)^2 \quad \text{at } y = 0, \quad (4)$$

$$u \rightarrow 0, \quad \frac{\partial u}{\partial y} \rightarrow 0, \quad T \rightarrow T_\infty, \quad \text{as } y \rightarrow \infty,$$

where u and v are the velocity components in x – and y – directions respectively, while $\nu = \frac{\mu}{\rho}$ is the kinematic viscosity, where μ is the coefficient of fluid viscosity and ρ is the fluid density. Then, σ is for fluid electrical conductivity, K^* for the permeability of porous media, B_0 for uniform magnetic field, T for temperature, α stands for thermal diffusivity and C_p is the specific heat at constant pressure. T_w and T_∞ stands for constant surface temperature and ambient fluid temperature respectively, c for positive stretching rate constant with T^{-1} as the dimension, A is a constant, l is the slip parameter and x_L is the characteristic length. Also, by applying Rosseland approximation for radiation we may expressed the radiative heat flux q_r as follows [21]:

$$q_r = - \frac{4\sigma^*}{3k^*} \frac{\partial T^4}{\partial y}, \quad (5)$$

where σ^* and k^* are the Stefan–Boltzman constant and the absorption coefficient, respectively. We assume that the temperature difference within the flow is such that the term T^4 , which can be expanded in a Taylor series about T_∞ can be presented as a linear function of temperature as shown below

$$T^4 \cong 4T_\infty^3 T - 3T_\infty^4. \quad (6)$$

By using the approximation as above, we have

$$q_r = -\frac{16\sigma^* T_\infty^3}{3k^*} \frac{\partial T}{\partial y}. \quad (7)$$

By using the appropriate similarity transformation

$$u = cx f'(\eta), \quad v = -(cv)^{\frac{1}{2}} f(\eta), \quad \eta = \left(\frac{c}{v}\right)^{\frac{1}{2}} y, \quad \theta(\eta) = \frac{T - T_\infty}{T_w - T_\infty}, \quad (8)$$

the continuity equation (1) is proven and the momentum (2) and energy (3) equations are reduced into ordinary differential equations as below respectively

$$f''' + ff'' + k_1^*(2f'f''' - f''^2 - ff^{iv}) - Kf' - f'^2 - Mf' = 0, \quad (9)$$

$$\left(1 + \frac{4}{3}R_d\right)\theta'' + Prf\theta' - 2Prf'\theta + PrEc\left(Mf'^2 + f''^2 + k_1^*f''(f'f'' - ff''')\right) = 0, \quad (10)$$

subject to the following transformed boundary conditions

$$f = 0, \quad f' = 1 + Lf''(0), \quad \theta = 1, \quad \text{at} \quad \eta = 0, \quad (11)$$

$$f' \rightarrow 0, \quad f'' \rightarrow 0, \quad \theta \rightarrow 0, \quad \text{as} \quad \eta \rightarrow \infty,$$

where f and θ is the dimensionless stream function and temperature function respectively, and the prime denotes differentiation with respect to η . Here, k_1^* is the viscoelastic parameter, K for porosity parameter, M for the magnetic parameter, R_d for the thermal radiation parameter, Ec for Eckert number, Pr is for the Prandtl number and L is the velocity slip parameter. Here $k_1^* < 0$ represents elastic-viscous fluid, $k_1^* > 0$ is for second grade fluid and $k_1^* = 0$ for Newtonian fluid. The definitions below will define the parameters specifically:

$$k_1^* = \frac{k_0 c}{\mu}, \quad K = \frac{\nu}{cK^*}, \quad M = \frac{\sigma B_0^2}{\rho c}, \quad R_d = \frac{4\sigma^* T_\infty^3}{k^* k}, \quad Ec = \frac{c^2 x_L^2}{AC_p}, \quad Pr = \frac{\nu}{\alpha}, \quad L = l \sqrt{\frac{c}{\nu}}. \quad (12)$$

Following [17], the skin friction coefficient is

$$Re_x^{\frac{1}{2}} C_f = -f''(0), \quad (13)$$

And the local Nusselt number is

$$Re_x^{\frac{1}{2}} Nu_x = -\theta'(0), \quad (14)$$

in which $Re_x = u_w x / \nu$ represents the local Reynolds number.

Exact analytical method

Velocity field

We can assume that (9) possesses a solution of exponential type [22]

$$f(\eta) = \frac{1}{c} (1 - e^{-z\eta}), \quad (13)$$

where c is the slip condition produces the relation,

$$c = z(1 + Lz). \quad (14)$$

Consequently, by substituting (13) and (14) into equation (9), it gives another relation in the form of a cubic algebraic equation

$$Lz^3 + (1 + k_1^*)z^2 - (M + K)Lz - M - K - 1 = 0. \quad (15)$$

Then, solving the above algebraic equation, we obtain the positive value of root z and the velocity profile is as shown below

$$f'(\eta) = \frac{1}{1 + Lz} e^{-z\eta}. \quad (16)$$

Temperature field

By referring to [23] and [24], the exact solution for temperature field can be obtained. We suppose the following transformation, where $t = e^{-z\eta}$ and the following relations between the derivatives with respect to η and the derivatives with respect to t is introduced:

$$\frac{d}{d\eta} \theta = -zt \frac{d}{dt} \theta, \quad \frac{d^2}{d\eta^2} \theta = z^2 \left[t^2 \frac{d^2}{dt^2} \theta + t \frac{d}{dt} \theta \right]. \quad (17)$$

By applying the transformation, we obtain

$$t\theta''(t) + (n - mt)\theta'(t) + 2m\theta(t) + P \operatorname{Ec} \left(\frac{z^2 t}{c^2} + \frac{k_1^* z^3 t}{c^3} + \frac{Mt}{c^2} \right) = 0, \quad (18)$$

with the following set of boundary conditions

$$\theta(0) = 0, \quad \theta(1) = 1. \quad (19)$$

Therefore, solving the ordinary differential equation (18), we obtain the exact solution for temperature field,

$$\begin{aligned} \theta(\eta) = & -\frac{1}{2} \frac{m^2(e^{-z\eta})^2 - 2m e^{-z\eta}(n+1) + n^2 + n}{c^3 m^2(n+1)} \operatorname{Ec} P (k_1^* z^3 + c z^2 + M c) \\ & + \frac{1}{2} H \frac{(e^{-z\eta})^{-n+1} (\operatorname{Ec} P k_1^* z^3 + \operatorname{Ec} P c z^2 + \operatorname{Ec} M P c + 2c^3 n + 2c^3)}{c^3(n+1)} \\ & - \frac{((z^2 + M)c + k_1^* z^3) P \left(m e^{-z\eta} - \frac{1}{2} n \right) \operatorname{Ec}}{c^3 m^2}, \end{aligned} \quad (20)$$

where $H = \frac{\text{hypergeometric}([-1-n], [-n+2], m e^{-z\eta})}{\text{hypergeometric}([-1-n], [-n+2], m)}$, $n = 1 - \frac{P}{zc}$, $m = -\frac{P}{zc}$, and $P = \frac{\operatorname{Pr}}{(1 + \frac{4}{3} R_d)}$.

Result and Discussion

In order to solve and analyze this analysis, we have used Maple software to facilitate the process. So, in this section, we will discuss and analyze the results that we obtained. The effects of various influential parameters on the velocity $f'(\eta)$ and temperature $\theta(\eta)$ distributions will be explored and discussed. Both elastico-viscous ($k_1^* < 0$) and second grade fluids ($k_1^* > 0$) are considered.

Impacts of porosity parameter K , magnetic parameter M and velocity slip parameter L on the non-dimensional velocity field $f'(\eta)$ are plotted in Figures 1, 2 and 3 respectively. Figure 1 presents the impact of porosity parameter K on velocity field $f'(\eta)$ for both elastico-viscous and second grade fluids which it can be concluded that the increment of porosity parameter K leads to decrement in the velocity field $f'(\eta)$ for both fluids. Physically,

the existence of porous media is to increase the resistance to the fluid flow which causes decay in fluid velocity and related momentum layer thickness. Next, Figure 2 displays the change in velocity field $f'(\eta)$ for varying magnetic parameter M for both elastico-viscous and second grade fluids. It is found that the influence of magnetic parameter M slows the velocity field $f'(\eta)$. The higher the value of magnetic parameter M , the slower the velocity field $f'(\eta)$. Meanwhile, Figure 3 shows the impact of velocity slip parameter L towards the velocity field $f'(\eta)$, which it depicts that the presence of slip parameter decays the velocity field $f'(\eta)$. This is because, the presence of slip parameter L allows more fluid to slip past the sheet causing the flow to slow down. We also found out that the second grade fluid increases the velocity field $f'(\eta)$ as being compared to the elastic-viscous fluid.

Furthermore, Figures 4-9 are plotted to explore the impacts of porosity parameter K , magnetic parameter M , thermal radiation parameter R_d , Eckert number Ec , velocity slip parameter L and Prandtl number Pr on the non-dimensional temperature field $\theta(\eta)$. Figure 4 depicts the change in temperature field $\theta(\eta)$ for varying porosity parameter K . It is observed that the higher value of porosity parameter K leads to an increment in temperature field $\theta(\eta)$ as well as more thermal layer thickness for both fluids. Then, Figure 5 reveals the effect of magnetic parameter M on the temperature field $\theta(\eta)$. It is noticed that the higher the magnetic parameter, the stronger the temperature field $\theta(\eta)$. Physically, the Lorentz force that is produced by the transverse magnetic field is opposed to the motion of the flow, which produces the resistance that enhances the temperature. The effect of thermal radiation parameter R_d on temperature field $\theta(\eta)$ is shown in Figure 6. The increasing values of thermal radiation parameter R_d leads to stronger temperature field $\theta(\eta)$ as well as the thermal layer thickness. This means that, the radiation should be minimum in order to initiate the cooling process. Same goes to Eckert number Ec which is the viscous dissipation parameter, where the increment of Eckert number enhances the temperature field $\theta(\eta)$ as shown in Figure 7. Also, as displays in Figure 8, the increases of velocity slip parameter L causing the increment in temperature field $\theta(\eta)$. Again, we also found out that the second grade fluid decreases the temperature field $\theta(\eta)$ as being compared to the elastic-viscous fluid. Meanwhile, Figure 9 illustrates the impact of Prandtl number Pr on the temperature field $\theta(\eta)$. It is noted that the temperature field $\theta(\eta)$ is lower for increasing values of Prandtl number Pr for both fluids, elastico-viscous and second grade fluids respectively.

Moreover, Table 1 is arranged to analyze the skin friction coefficient $-f''(0)$ for varying porosity parameter K , magnetic parameter M , and velocity slip parameter L . It is shown that the skin friction coefficient $-f''(0)$ value is increasing as the value of magnetic parameter M and porosity parameter K increasing, in the presence of both elastico-viscous and second grade fluids. Meanwhile differ for velocity slip parameter L , where the value of skin friction coefficient $-f''(0)$ is decreasing when the value of velocity slip parameter L increasing. We also have compared the value of skin friction coefficients $-f''(0)$ with the analytical results provide by Nayak *et al.* [17] and it shows a good agreement.

Furthermore, Table 2 depicts the numerical data for local Nusselt number $-\theta'(0)$ of different values of thermal radiation parameter R_d , Eckert number Ec , and velocity slip parameter L for both elastico-viscous and second grade fluids. It is noted that the local Nusselt number $-\theta'(0)$ is reducing as thermal radiation parameter R_d , Eckert number Ec , and velocity slip parameter L increasing for both fluids. Also, in order to convince the validity of the method we used, we also have compared the value of Nusselt number $-\theta'(0)$ with the previous work done by Nayak *et al.* [17], Liu [5] and Datti *et al.* [4], and it can be seen that the values are almost the same as can be seen in Table 3 and 4. Table 3 displays the numerical data for local Nusselt number $-\theta'(0)$ for different values of magnetic parameter M and Prandtl number Pr

for second grade fluid as to be compared with previous work by Nayak *et al.* [17] and Liu [5]. The values of Nusselt number $-\theta'(0)$ are decreasing as the magnetic parameter M increasing, while the opposite trend occurs for Prandtl number Pr . Then, Table 4 shows the numerical data for local Nusselt number $-\theta'(0)$ for different values of magnetic parameter M and porosity parameter K for elastico-viscous fluid as to be compared with previous work by Nayak *et al.* [17] and Datti *et al.* [4]. The increment in porosity parameter K causes decrement in heat transfer coefficient which is the Nusselt number $-\theta'(0)$.

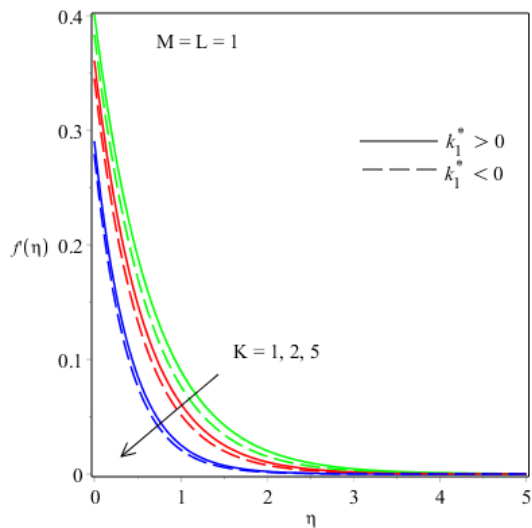


Fig. 8 Velocity profile for different values of K .

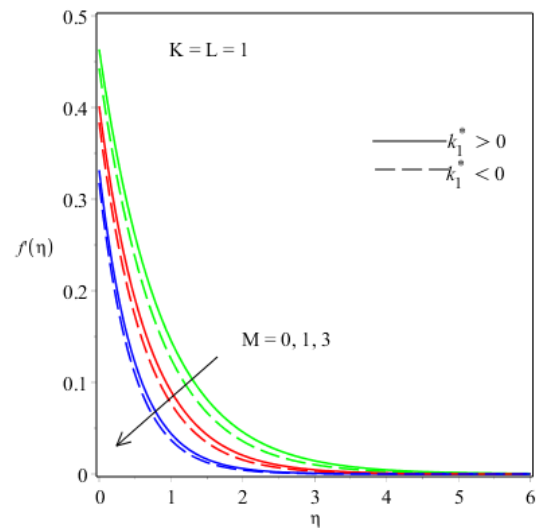


Fig. 2 Velocity profile for different values of M .

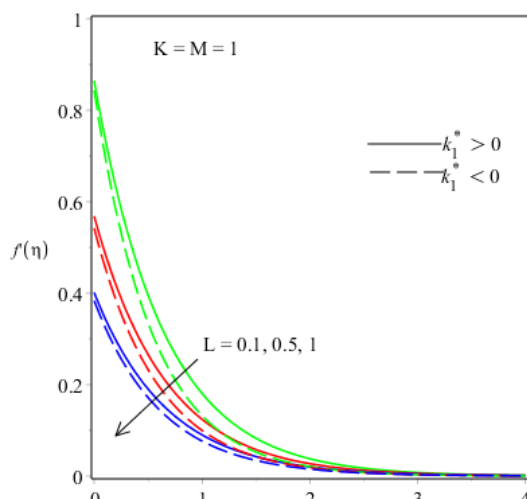


Fig. 3 Velocity profile for different values of L .

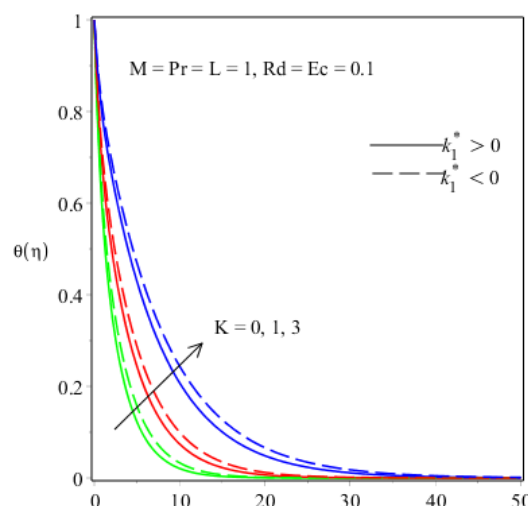


Fig. 4 Temperature profile for different values of K .

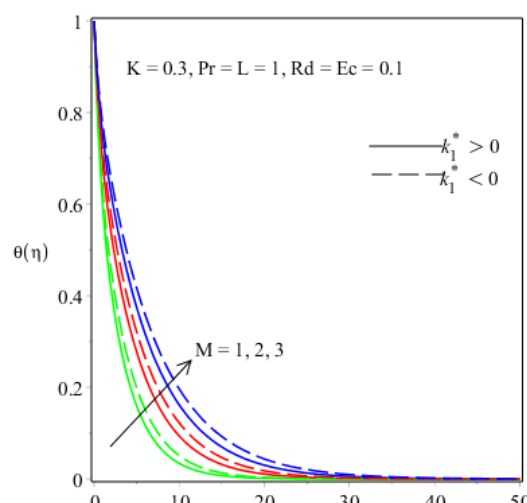


Fig. 5 Temperature profile for different values of M .

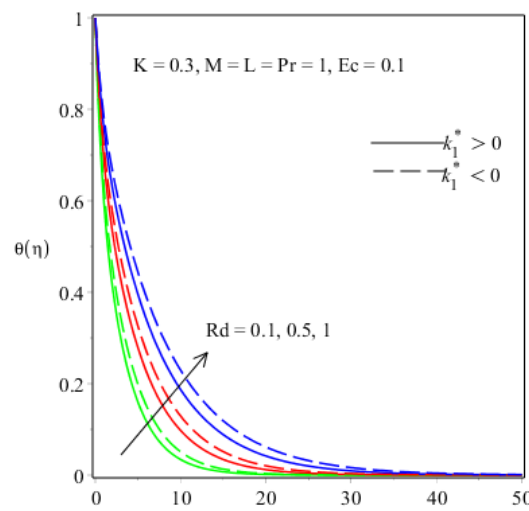


Fig. 6 Temperature profile for different values of R_d .

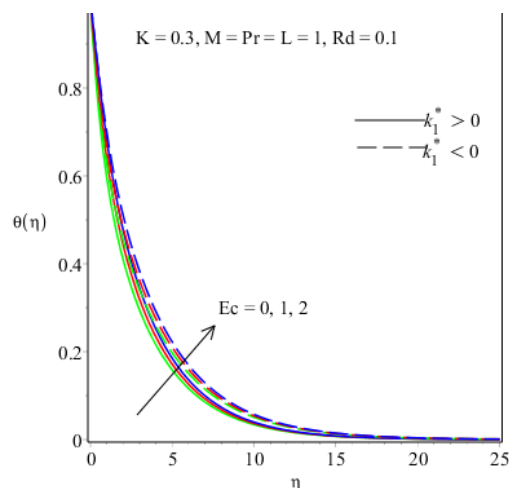


Fig. 7 Temperature profile for different values of Ec .

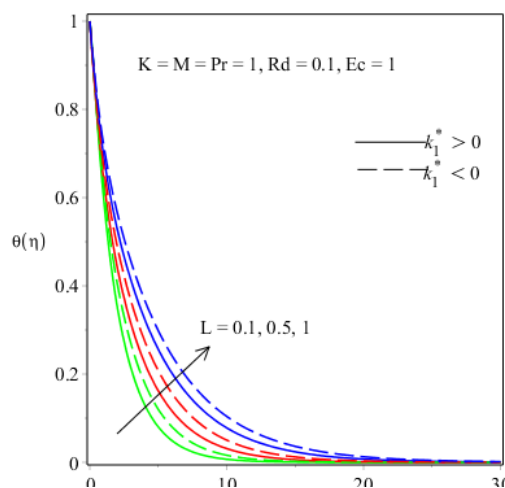


Fig. 8 Temperature profile for different values of L .

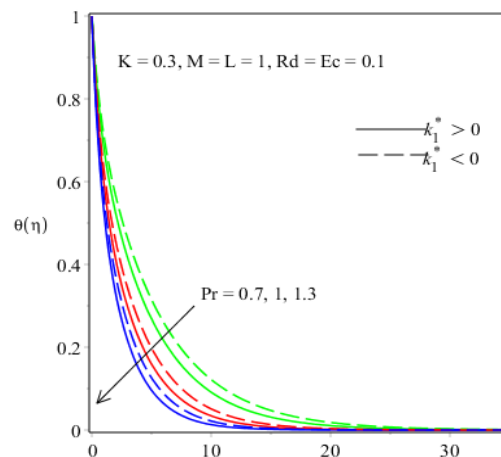


Fig. 9 Temperature profile for different values of Pr.

Table 11 : Comparison of skin friction coefficients.

M	K	k_1^*	L	$-f''(0)$	
				Nayak <i>et al.</i> [17]	Present
1	0.01	-0.5	0	2.00499	2.00499
	2			2.82843	2.82843
0.5	0.01			1.73781	1.73781
	2			2.64575	2.64575
			1	-	0.65017
			2	-	0.38889
		0.2	0.5	-	0.90628
			1	-	0.62063
1	0.01	0.5	0	1.15758	1.15758
	2			1.63299	1.63299
0.5	0.01			1.00333	1.00333
	2			1.52753	1.52753
1	0.01	1		1.00250	1.00250
	2			1.41421	1.41421
0.5	0.01			0.86891	0.86891
	2			1.32288	1.32288

Table 12: Nusselt numbers for elastico-viscous and second grade fluids for $M = 1$, $Pr = 1$ and $K = 0.01$.

k_1^*	R_d	Ec	L	$-\theta'(0)$
1	0.1	0.1	1	0.67753
				0.57966
	0.3	0.2		0.56643
				0.55320
			2	0.43419
-0.1	0.1	0.1	1	0.65394
				0.55788
	0.3	0.2		0.54566
				0.53344
			2	0.42072

Table 13: Comparison of Nusselt numbers for a second grade fluid for $k_1^* = 1.0$, $Ec = 0.2$, $R_d = 0$, $L = 0$ and $K = 0.01$.

M	Pr	$-\theta'(0)$		
		Liu [5]	Nayak <i>et al.</i> [17]	Present
0	1	1.337265	1.331574	1.332375
	10	4.48696	4.478859	4.482695
1	1	1.13333	1.131589	1.132255
	10	3.74805	3.741224	3.744717

Table 14: Comparison of Nusselt numbers for elastico-viscous fluid for $Ec = 0$, $R_d = 0$, $Pr = 1$ and $L = 0$.

k_1^*	M	K	$-\theta'(0)$		
			Datti <i>et al.</i> [4]	Nayak <i>et al.</i> [17]	Present
0	0	0.01	1.3333	1.331932	1.331932
		2		1.126898	1.126898
	1	0.01	1.2158	1.214771	1.214771
		2		1.055361	1.055361
-0.1	0.1	0.01	1.3035	1.302129	1.302129
		2		1.093343	1.093343

Conclusion

The slip darcy boundary layer flow of viscoelastic fluid and heat transfer over a linear stretchable surface with the presence of magnetohydrodynamic (MHD), porous medium, viscous dissipation and thermal radiation have been discussed. The governing partial differential equations are reduced to non-linear ordinary differential equations facilitate by appropriate similarity transformation. These non-linear ordinary differential equations were then solved analytically by using exact analytical method. The conclusion can be listed as follows:

- Increment in the value of porosity parameter K , magnetic parameter M and velocity slip parameter L lead to decrement in velocity field $f'(\eta)$.
- Increment in the value of porosity parameter K , magnetic parameter M , velocity slip parameter L , thermal radiation parameter R_d and Eckert number Ec (viscous dissipation parameter) lead to increment in temperature field $\theta(\eta)$.
- Increment in the value of Prandtl number Pr , decays the temperature field $\theta(\eta)$.
- Both velocity $f'(\eta)$ and temperature $\theta(\eta)$ fields display opposite behavior for velocity slip parameter L .
- The second grade fluid ($k_1^* > 0$) enhances the velocity field $f'(\eta)$ and reduces the temperature field $\theta(\eta)$ compared to elastico-viscous fluid ($k_1^* < 0$).
- Elastico-viscous fluid ($k_1^* < 0$) enhances the skin friction coefficient $-f''(0)$ and reduces the heat transfer coefficient $-\theta'(0)$ compared to second grade fluid.
- Skin friction coefficient $-f''(0)$ is increasing for larger magnetic parameter M and porosity parameter K , while decreasing for larger value of velocity slip parameter L for both fluids.
- Local Nusselt number $-\theta'(0)$ for both fluids is higher for smaller porosity parameter K , magnetic parameter M , velocity slip parameter L , thermal radiation parameter R_d and Eckert number Ec , while the opposite trend for Prandtl number Pr .

Acknowledgement

The financial support received in the form of Putra Grant 9619000 from Universiti Putra Malaysia are gratefully acknowledged and appreciated by the authors.

References

- [1] Rajagopal, Kumbakonam R., T. Y. Na, and A. S. Gupta. "Flow of a viscoelastic fluid over a stretching sheet." *Rheologica Acta* 23, no. 2 (1984): 213-215.
- [2] Andersson, H. I. "MHD flow of a viscoelastic fluid past a stretching surface." *Acta Mechanica* 95, no. 1-4 (1992): 227-230.
- [3] Ariel, P. D. "MHD flow of a viscoelastic fluid past a stretching sheet with suction." *Acta mechanica* 105, no. 1-4 (1994): 49-56.
- [4] Datti, P. S., K. V. Prasad, M. Subhas Abel, and Ambuja Joshi. "MHD visco-elastic fluid flow over a non-isothermal stretching sheet." *International Journal of Engineering Science* 42, no. 8-9 (2004): 935-946.
- [5] Liu, I-Chung. "Flow and heat transfer of an electrically conducting fluid of second grade in a porous medium over a stretching sheet subject to a transverse magnetic field." *International Journal of Non-Linear Mechanics* 40, no. 4 (2005): 465-474.
- [6] Khan, Sujit Kumar, and Emmanuel Sanjayanand. "Viscoelastic boundary layer flow and heat transfer over an exponential stretching sheet." *International Journal of Heat and Mass Transfer* 48, no. 8 (2005): 1534-1542.
- [7] Cortell, Rafael. "A note on flow and heat transfer of a viscoelastic fluid over a stretching sheet." *International Journal of Non-Linear Mechanics* 41, no. 1 (2006): 78-85.
- [8] Ariel, P. D., T. Hayat, and S. Asghar. "The flow of an elastico-viscous fluid past a stretching sheet with partial slip." *Acta Mechanica* 187, no. 1-4 (2006): 29-35.
- [9] Khan, Sujit Kumar. "Heat transfer in a viscoelastic fluid flow over a stretching surface with heat source/sink, suction/blowing and radiation." *International Journal of Heat and Mass Transfer* 49, no. 3-4 (2006): 628-639.
- [10] Hayat, T., Z. Abbas, M. Sajid, and S. Asghar. "The influence of thermal radiation on MHD flow of a second grade fluid." *International Journal of Heat and Mass Transfer* 50, no. 5-6 (2007): 931-941.
- [11] Abel, M. Subhas, P. G. Siddheshwar, and Mahantesh M. Nandeppanavar. "Heat transfer in a viscoelastic boundary layer flow over a stretching sheet with viscous dissipation and non-uniform heat source." *International Journal of Heat and Mass Transfer* 50, no. 5-6 (2007): 960-966..
- [12] Abel, M. Subhas, and N. Mahesha. "Heat transfer in MHD viscoelastic fluid flow over a stretching sheet with variable thermal conductivity, non-uniform heat source and radiation." *Applied Mathematical Modelling* 32, no. 10 (2008): 1965-1983.
- [13] Chen, Chien-Hsin. "On the analytic solution of MHD flow and heat transfer for two types of viscoelastic fluid over a stretching sheet with energy dissipation, internal heat source and thermal radiation." *International Journal of Heat and Mass Transfer* 53, no. 19-20 (2010): 4264-4273.
- [14] Turkyilmazoglu, M. "Multiple solutions of heat and mass transfer of MHD slip flow for the viscoelastic fluid over a stretching sheet." *International Journal of Thermal Sciences* 50, no. 11 (2011): 2264-2276.
- [15] Raftari, Behrouz, and Kuppapalle Vajravelu. "Homotopy analysis method for MHD viscoelastic fluid flow and heat transfer in a channel with a stretching wall." *Communications in nonlinear science and numerical simulation* 17, no. 11 (2012): 4149-4162.

- [16] Turkyilmazoglu, Mustafa. "Multiple analytic solutions of heat and mass transfer of magnetohydrodynamic slip flow for two types of viscoelastic fluids over a stretching surface." *Journal of Heat Transfer* 134, no. 7 (2012): 071701.
- [17] Nayak, Manoj Kumar, Gauranga Charan Dash, and Lambodar Prasad Singh. "Heat and mass transfer effects on MHD viscoelastic fluid over a stretching sheet through porous medium in presence of chemical reaction." *Propulsion and Power Research* 5, no. 1 (2016): 70-80.
- [18] Nadeem, Sohail, Shafiq Ahmad, and Noor Muhammad. "Cattaneo-Christov flux in the flow of a viscoelastic fluid in the presence of Newtonian heating." *Journal of Molecular Liquids* 237 (2017): 180-184.
- [19] Fagbade, A. I., B. O. Falodun, and A. J. Omowaye. "MHD natural convection flow of viscoelastic fluid over an accelerating permeable surface with thermal radiation and heat source or sink: Spectral Homotopy Analysis Approach." *Ain Shams Engineering Journal* 9, no. 4 (2016): 1029-1041.
- [20] Chen, Xuehui, Weidong Yang, Xinru Zhang, and Fawang Liu. "Unsteady boundary layer flow of viscoelastic MHD fluid with a double fractional Maxwell model." *Applied Mathematics Letters* 95 (2019): 143-149.
- [21] Brewster, M. Quinn. *Thermal radiative transfer and properties*. John Wiley & Sons, 1992.
- [22] Crane, Lawrence J. "Flow past a stretching plate." *Zeitschrift für angewandte Mathematik und Physik ZAMP* 21, no. 4 (1970): 645-647.
- [23] Ebaid, Abdelhalim, and Hibah S. Alhawiti. "New application for the generalized incomplete gamma function in the heat transfer of nanofluids via two transformations." *Journal of Computational Engineering* 2015 (2015).
- [24] Aly, Emad H., and Abdelhalim Ebaid. "Exact analysis for the effect of heat transfer on MHD and radiation Marangoni boundary layer nanofluid flow past a surface embedded in a porous medium." *Journal of Molecular Liquids* 215 (2016): 625-639.

NUMERICAL EXAMINATION ON TWO-EQUATIONS TURBULENCE MODELS FOR FLOW ACROSS NACA 0012 AIRFOIL WITH DIFFERENT ANGLE OF ATTACK

Yu Han Ng¹
Wah Yen Tey^{1,2}
Lit Ken Tan²
Gerald P. Arada³
M.W. Muhiedeen¹

¹Department of Mechanical Engineering, Faculty of Engineering, UCSI University, Kuala Lumpur, Malaysia

²Takasago i-Kohza, Malaysia-Japan International Institute of Technology, Universiti Teknologi Malaysia, Kuala Lumpur, Malaysia

³Gokongwei College of Engineering, De La Salle University, Manila, Philippines

*(Corresponding Author's Email: teywy@ucsiuniversity.edu.my)

Abstract: Selection of an appropriate and efficient turbulence models is important for fast and accurate computation in fluid dynamics. In order to investigate the computational efficiency of turbulence models, numerical examination based on two-equations turbulence models for flow across NACA 0012 airfoil was carried out by using ANSYS Fluent at various angle of attack (-12° to 20°) and at a Reynold number of 3×10^6 . The case study is chosen as its transition from viscid to inviscid flow region which would put a strain on computational performance of turbulence models. The two-equation models being investigated are Standard $k-\epsilon$ model, RNG $k-\epsilon$ model, $k-\epsilon$ Realizable model, Standard $k-\omega$ model, $k-\omega$ BSL model and $k-\omega$ SST model. The drag, lift and pressure coefficient between simulation and experimental results are compared. The convergence rate of these turbulence models is collated as well. The contours of static pressure and velocity magnitude was simulated, and boundary layer separation was noticed from 10° angle of attack. In general, the predicted data have good agreement with experimental data. Amongst the investigated models, $k-\omega$ SST model showed the best agreement with experimental result meanwhile RNG $k-\epsilon$ model showed the slowest convergence rate among all the turbulence models.

Keywords: Two-equations models, Reynolds-Averaged Navier-Stokes Equations (RANS), Ansys Fluent, Flow across Airfoil, NACA 0012

Introduction

Turbulent flow is three-dimensional, unpredictable fluid flow phenomenon which is highly diffusive and dissipative, in which the eddies are prevalent across the flow [1]. Indeed, there are many natural phenomenon and scientific case studies which involve flow turbulence, such as the turbulence in ocean and wind turbine [2,3], engineering piping [4,5], flow-induced vibration [6,7] and operation of bioreactor [8,9]. Transition from a smooth laminar flow to a chaotic turbulent flow would give rise to complex fluid dynamics phenomenon, which would involve the mixing of fluids, dissipation of energy and increment of drag.

With the advancement of Computational Fluid Dynamics (CFD), various methods have been introduced and widely applied to predict the nature of turbulence. Basically, turbulent flow can be investigated using three main methods: Direct Eddy Simulation (DNS), Large Eddy Simulation (LES) and turbulence modelling based on Reynolds-Averaged Navier-Stokes (RANS) equation. DNS solves the all the spatial and temporal scale of turbulence down to

Kolmogorov scale, and therefore it would render a result with superior accuracy. However, it calls upon a very large computational memory and most of the DNS studies are still limited to low Re flow [10–13]. In LES, the analysis on small eddies is replaced with other models such as wall adapting local eddy viscosity model [14], mixed time scale model [15] and parabolised stability equations [16]. Compared with the DNS computation cost of $Re^{2.64}$, it is only $Re^{1.85}$ for LES [17]. This has enabled the simulation on a flow with higher Re and been evinced in the recent works of Kim et al. [16], Badoe et al. [18], Arora et al. [19] and Toubiana et al. [20]. Meanwhile, the turbulence modelling based on RANS would have the lowest computation cost with the sacrifice of results' accuracy. Mathematical details of RANS and its turbulence models will be explained in the next section. The review on these three methods can be referred to the literature by Argyropoulos and Markatos [21].

Two-equations turbulence modelling approach is one of the most popular methods being applied for turbulence prediction as it can greatly simplify the complex flow in practical [1]. Since many turbulent models have been introduced, selection of a robust model is critical for CFD simulation. In this study, six two-equations turbulent models (Standard k- ϵ , RNG k- ϵ , k- ϵ Realisable, standard k- ω , k- ω BSL and k- ω SST) are considered to simulate the flow across NACA 0012 airfoil. The co-existence of viscous and inviscid flow regions makes the case study an excellent platform to assess their computational performance. Finite volume based commercial software ANSYS Fluent was applied in current study. Pressure coefficient, drag coefficient and lift coefficient are analysed at different angle of attack at $Re = 3 \times 10^6$ and compared with experiment data. The pressure-velocity contours are computed while convergence performance of these models is compared too.

Methodology

Reynold-Averaged Navier-Stokes equations (RANS)

Consider the conventional Continuity and momentum equations which govern the flow of viscous incompressible fluid as:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (1)$$

$$\rho \left(\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} \right) = - \frac{\partial P}{\partial x_i} + \frac{\partial (2\mu S_{ij})}{\partial x_i} \quad (2)$$

where u_i and u_j is flow velocity while ρ is fluid density. The strain-rate tensor, S_{ij} for Newtonian fluid can be defined as in Eq. (3). By inserting Eq. (3) into (1), Eq. (4) can be formed.

$$S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (3)$$

$$\rho \left(\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} \right) = - \frac{\partial P}{\partial x_i} + \frac{\partial^2 u_i}{\partial x_j^2} \quad (4)$$

In turbulent flow, the fields become random functions of space and time. Thus, the variables of u_i and P can be divided into mean part and fluctuating part as:

$$u_i = \bar{u}_i + u_i', \quad P = \bar{P} + P_i' \quad (5)$$

Since it is not necessary to concern all the detail of turbulence fluctuations, the RANS equations can be formed by averaging the equations:

$$u_i' = 0 \rightarrow u_i = \bar{u}_i \quad (6)$$

$$P_i' = 0 \rightarrow P_i = \bar{P}_i \quad (7)$$

By substituting (5) into (1) and (2), Continuity and momentum equations become:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (8)$$

$$\rho \left(\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial (\bar{u}_i \bar{u}_j)}{\partial x_j} \right) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} (2\mu \bar{S}_{ij} - \rho \bar{u}_i' u_j') \quad (9)$$

and mean strain-rate tensor S_{ij} becomes:

$$S_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad (10)$$

By combining Eqs. (8)-(10), the momentum equations can be expressed as:

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial (\bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \nu \left(\frac{\partial^2 \bar{u}_i}{\partial x_i \partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} \right) \quad (11)$$

where τ_{ij} is the Reynold's stress tensor. By devising the instantaneous properties into mean and fluctuating parts, 3 unknown quantities were appeared. Unfortunately, that are no any additional equation to gain. Hence, this system is not a closed system. In order to solve the closure problem, the various types of turbulence models were applied to close the RANS equation by predicting the turbulent viscosity. RANS equations require closure for Reynolds stress and the effect of turbulence can be presented as an eddy viscosity.

$$\tau_{ij} = -\overline{u_i u_j} = 2\mu_t S_{ij} - \frac{2}{3} k \delta_{ij} \quad (12)$$

where μ_t is the kinetic eddy viscosity, δ_{ij} is Kronecker delta and k is the turbulence kinetic energy which can be defined as:

$$k = \frac{1}{2} \overline{u_i u_i} \quad (13)$$

In order to close the equations, turbulence models are required. In the following section, the formulation of Standard k - ϵ , RNG k - ϵ , k - ϵ Realisable, standard k - ω , k - ω BSL and k - ω SST will be explained.

Turbulence models

Standard k - ϵ model

The original Standard k - ϵ model was proposed by Launder and Spalding [22]. In Standard k - ϵ model, turbulent kinetic energy k and dissipation rate ϵ can be defined as in Eq. (14) and (15) respectively.

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \quad (14)$$

$$\frac{\partial (\rho \epsilon)}{\partial t} + \frac{\partial (\rho \epsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_i} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon \quad (15)$$

The turbulent viscosity μ_t can calculated from formula as:

$$\mu_t = \rho c_\mu \frac{k^2}{\epsilon} \quad (16)$$

while G_k is the generation of turbulent kinetic energy due to the mean velocity gradient, which can be defined as:

$$G_k = -\rho u_i u_j \frac{\partial u_j}{\partial x_i} \quad (17)$$

G_b is the generation of turbulent kinetic energy due to buoyancy and Y_M is the contribution of the fluctuating dilation in compressible turbulence to overall dissipation rate, and they can be defined in Eq. (18) and (19) respectively.

$$G_k = \beta g_i \frac{\partial \mu_t}{\partial x_i} \quad (18)$$

$$Y_M = 2\rho \varepsilon M_t^2 \quad (19)$$

S_k and S_ε are user sources term. σ_ε is the turbulent Prandtl number of the dissipation rate and σ_k is the turbulent Prandtl number of the turbulent kinetic energy. The standard values of the constant which obtained experimentally are: $C_\mu = 0.09$, $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$ while $\sigma_\varepsilon = 1.3$.

RNG k-ε model

RNG k - ε model is quite similar to standard k - ε model. The only different is that RNG k - ε model uses a technique renormalization group theory [23] but Standard k - ε model does not. The new terms were added into to the transport equation for dissipation rate and turbulent kinetic energy. This helped the RNG model to improve the accuracy of swirling flow. The transport equation for turbulent kinetic energy k and dissipation rate ε is:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\alpha_k \mu_{eff} \frac{\partial k}{\partial x_i} \right) + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (20)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon \quad (21)$$

where α_ε is the inverse effective turbulent Prandtl number of the dissipation rate and α_k is the inverse effective turbulent Prandtl number of the turbulent kinetic energy. In RNG model, the turbulent viscosity ν_t was obtained effectively by a scale elimination procedure as:

$$d \left(\frac{\rho^2 k}{\sqrt{\varepsilon \mu}} \right) = 1.72 \frac{\hat{\nu}}{\sqrt{\nu^3 - 1 + C_\nu}} d\hat{\nu} \quad (22)$$

This equation is used to determine the accurate description of how the effective turbulent transport varies with effective Reynold number to determine better accuracy result in near-wall flow and low-Re number flow. Thus, RNG model should give a more accurate result. However, the formula of turbulence viscosity in high-Re number is same as standard model as in Eq. (16). Although the formula is same as standard model, C_μ of RNG model has a different value with standard model. The C_μ in RNG model is 0.0845 which is very close to standard model (0.09). The standard values of the constant which obtained experimentally are: $C_\mu = 0.0845$, $C_{1\varepsilon} = 1.42$ while $C_{2\varepsilon} = 1.68$.

k-ε Realisable model

k - ε Realisable model is the newest model in k - ε variation. It was developed by Shih et al.[24]. This model is different with Standard k - ε model in terms of turbulent viscosity and transport equation for dissipation rate ε was. This new transport equation for dissipation rate was derived from the equation for the transport of the mean-square vorticity fluctuation. Thus, k - ε Realisable model is more suitable to use in predicting separate flows and flows with complex

secondary flow feature. The turbulent kinetic energy k of k - ε Realisable model is same with standard k - ε model as in Eq. (14) while its dissipation rate ε equation is in Eq. (23).

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho\varepsilon u_j)}{\partial x_j} = \frac{\partial}{\partial x_i} \left[\mu + \left(\frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_{1\varepsilon} S \varepsilon - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_{1\varepsilon} \rho \frac{\varepsilon}{k} C_{3\varepsilon} G_b + S_\varepsilon \quad (23)$$

The turbulent viscosity of realizable model is same with the other k - ε models as in Eq. (16). However, C_μ is not a constant in realizable model and can be calculated from formula:

$$C_\mu = \frac{1}{A_0 + A_s \frac{k U^*}{\varepsilon}} \quad (24)$$

$$U^* = \sqrt{S_{ij} S_{ij} + \Omega_{ij} \Omega_{ij}} \quad (25)$$

$$\Omega_{ij} = \overline{\Omega_{ij}} - 3\varepsilon_{ijk} \omega_k \quad (26)$$

where $\overline{\Omega_{ij}}$ is the mean rate of rotation tensor viewed in a moving reference frame with angular velocity ω_k . The constants A_0 and A_s are:

$$A_0 = 4.04 \quad \text{and} \quad A_s = \sqrt{6} \cos \phi$$

where:

$$\phi = \frac{1}{3} \cos^{-1} \left(\sqrt{6} \frac{S_{ij} S_{jk} S_{ik}}{S^3} \right) \quad S = \sqrt{S_{ij} S_{ij}}$$

0.9 is found for the standard value of C_μ in the solution of equation of turbulent viscosity for an inertial sub layer in the equilibrium boundary layer. The model constant in this model are $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.9$, $\sigma_k = 1.0$ while $\sigma_\varepsilon = 1.2$ and C_1 can be calculated by:

$$C_1 = \max \left[0.43, \frac{\eta}{\eta + 5} \right] \bigg|_{\eta = \sqrt{2 S_{ij} S_{ij}} \frac{k}{\varepsilon}} \quad (27)$$

Standard k - ω model

A Standard k - ω model was developed by Wilcox [25] in 1988. The Standard k - ω model was developed for better calculation of low Reynolds number effects, compressibility and shear flow spreading. The free shear flow prediction in this model has a very good agreement in calculation of mixing layer, far wake and round, radial and plane jet. Hence, it is suitable to apply on wall-bounded flows and free shear flow. The free shear flow prediction in this model has a very good agreement in calculation of mixing layer, far wake and round, radial and plane jet. This Standard k - ω model governs the transport equations for turbulent kinetic energy and specific dissipation rate as shown as:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - Y_k + S_k \quad (28)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho \omega u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + S_\omega \quad (29)$$

where G_k is the generation of turbulent kinetic energy from the mean velocity gradient, G_ω is the generation of specific dissipation rate, Y_k is the dissipation of turbulent kinetic energy due to turbulence, Y_ω is the dissipation of k due to turbulence, S_k and S_ω are user source terms. σ_ω is

the turbulent Prandtl number of specific dissipation rate and σ_k is the turbulent Prandtl number of the turbulent kinetic energy. The turbulent viscosity of Standard k - ω model is:

$$\mu_t = \alpha^* \frac{\rho k}{\omega} \quad (30)$$

A low Reynold number correction is caused from turbulent viscosity that damped by coefficient α^* . The coefficient α^* can be calculated by:

$$\alpha^* = \alpha_\infty^* \left(\frac{\alpha_0^* + \text{Re}_t / \text{Re}_k}{1 + \text{Re}_t / \text{Re}_k} \right) \quad (31)$$

where:

$$\text{Re}_t = \frac{\rho k}{\omega}, \text{Re}_k = 6, \alpha_0^* = \frac{\beta_1}{3}, \beta_1 = 0.072$$

For high Reynold number of this model, $\alpha^* = \alpha_\infty^* = 1$. The dissipation of k and ω , Y_k and Y_ω can be defined respectively as in Eq. (32) and (33).

$$Y_k = \rho \beta^* f_\beta k \omega \quad (32)$$

$$Y_\omega = \rho \beta^* f_\beta \omega^2 \quad (33)$$

***k*- ω BSL model**

k - ω BSL model was proposed by Menter [26] in 1994. This model combines the advantages of the k - ε model and k - ω model. This model effectively blends the robust and accurate formation of the k - ω model in the near wall region with the free-stream independence of the k - ε model in the far field. However, this model still fails to proper predict the onset and amount of flow separation from the smooth surfaces. Due to the both models do not account for transport of the turbulent shear stress, the overprediction of the eddy may occur. This k - ω BSL model governs the transport equations via the turbulent kinetic energy k and specific dissipation rate ω as shown in Eqs. (34) and (35).

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right] + P + \beta^* \rho \omega k \quad (34)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho \omega u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right] + \frac{\gamma}{\nu_t} P + \beta^* \rho \omega^2 + 2(1 - F_1) \frac{\rho \sigma_{\omega 2}}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (35)$$

where:

$$P = \tau_{ij} \frac{\partial u_i}{\partial x_j} \quad (36)$$

$$F_1 = \tan \left\{ \left[\min \left[\max \left(\frac{2\sqrt{k}}{0.09\omega y'}, \frac{500}{y^2 \omega} \right), \frac{4\rho\sigma_{\omega 2}}{CD_{k\omega} y^2} \right] \right]^4 \right\} \quad (37)$$

where

$$\mu_t = \frac{\rho k}{\omega} \quad (37)$$

$$CD_{k\omega} = \max \left(2\rho\sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-20} \right) \quad (38)$$

The coefficient of β, γ, σ_k and σ_ω are computed as:

$$\left. \begin{aligned} \beta &= F_1 \beta_1 + (1 + F_1) \beta_2 \\ \gamma &= F_1 \gamma_1 + (1 + F_1) \gamma_2 \\ \sigma_k &= F_1 \sigma_{k1} + (1 + F_1) \sigma_{k2} \\ \sigma_\omega &= F_1 \sigma_{\omega 1} + (1 + F_1) \sigma_{\omega 2} \end{aligned} \right\} \quad (39)$$

the model constants are: $\beta^* = 0.09$, $\beta_1 = 0.075$, $\beta_2 = 0.0828$, $\gamma_1 = \beta_1 / \beta^*$, $\gamma_2 = \beta_2 / \beta^*$, $\sigma_{k1} = 0.5$, $\sigma_{k2} = 1.5$, $\sigma_{\omega 1} = 0.5$ and $\sigma_{\omega 2} = 0.856$.

k- ω SST model

k- ω SST model is modified model from standard *k- ω* model. The *k- ω SST* model was developed by Menter [27] too in 1993. This model effectively blends the robust and accurate formation of the *k- ω* model in the near wall region with the free-stream independence of the *k- ε* model in the far field. By these features, the *k- ω SST* model is more reliable and accurate for a wider class of flows. This model also can be perceived as a combination of *k- ε* model and *k- ω* model. This model was developed to overcome the problem of overprediction in *k- ω BSL* model. This *k- ω SST* model governs the transport equations via the turbulent kinetic energy and specific dissipation rate as show below.

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - Y_k + S_k \quad (40)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho \omega u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + D_\omega + S_\omega \quad (41)$$

where G_k is the generation of turbulent kinetic energy from the mean velocity gradient, G_ω is the generation of specific dissipation rate, Y_k is the dissipation of turbulent kinetic energy due to turbulence, Y_ω is the dissipation of k due to turbulence, S_k and S_ω are user sources term. σ_ω is the turbulent Prandtl number of specific dissipation rate and σ_k is the turbulent Prandtl number of the turbulent kinetic energy. The turbulent viscosity is:

$$\mu_t = \frac{\rho k}{\omega A} \quad A = \max \left(\frac{1}{\alpha'}, \frac{S F_2}{a_1 \omega} \right) \quad (42)$$

The σ_k and σ_ω are given by:

$$\sigma_k = \frac{1}{F_1 / \sigma_{k,1} + (1 - F_1) / \sigma_{k,2}} \quad (43)$$

$$\sigma_\omega = \frac{1}{F_1 / \sigma_{\omega,1} + (1 - F_1) / \sigma_{\omega,2}} \quad (44)$$

The blending function F_1 and F_2 are:

$$F_1 = \tanh(\Phi_1^4) \quad (45)$$

$$F_2 = \tanh(\Phi_2^2) \quad (46)$$

where:

$$\Phi_1 = \min \left[\max \left(\frac{\sqrt{k}}{0.09 \omega y}, \frac{500 \mu}{\rho y^2 \omega} \right), \frac{4 \rho k}{\sigma_{\omega,2} D_\omega^+ y^2} \right] \quad (47)$$

$$D_{\omega}^{+} = \max \left(2\rho \frac{1}{\sigma_{\omega,2}} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-10} \right) \quad (48)$$

$$\Phi_2 = \max \left(\frac{2\sqrt{k}}{0.09\omega y}, \frac{500\mu}{\rho y^2 \omega} \right) \quad (49)$$

The Y_k and Y_{ω} are defined as:

$$\gamma_k = \rho \beta * k \omega \quad (50)$$

$$\gamma_{\omega} = \rho \beta * \omega^2 \quad (51)$$

The model constants are: $\beta_{i,1} = 0.075$, $\beta_{i,2} = 0.0828$, $\sigma_{k,1} = 1.176$, $\sigma_{k,2} = 1.0$, $\sigma_{\omega,1} = 2.0$ while $\sigma_{\omega,2} = 1.168$.

Computational method

In this study, 3×10^6 Reynolds number flow across two-dimensional NACA 0012 airfoil is simulated by using ANSYS Fluent version 17.0 at various angle of attack (-12° to 20°). C-type topology was decided to use in simulation because the structured quadrilateral element had the advantages of a higher degree of control and accuracy, lower memory consumption and a faster convergence rate. Thus, it can minimize the skewness of a near wall mesh. The residual setting is set to 0.0001. The domain height was set to 20 chord length and the domain length was set to 30 chord length. The airfoil was located at the centre of the semicircle. According to Yao et al. [28], the grid cells were sufficient when the number of grids is above 80000. Thus, total 104000 quadrilateral cells were applied in the mesh generation. The Figure 1 showed the mesh of the computational domain around NACA 0012.

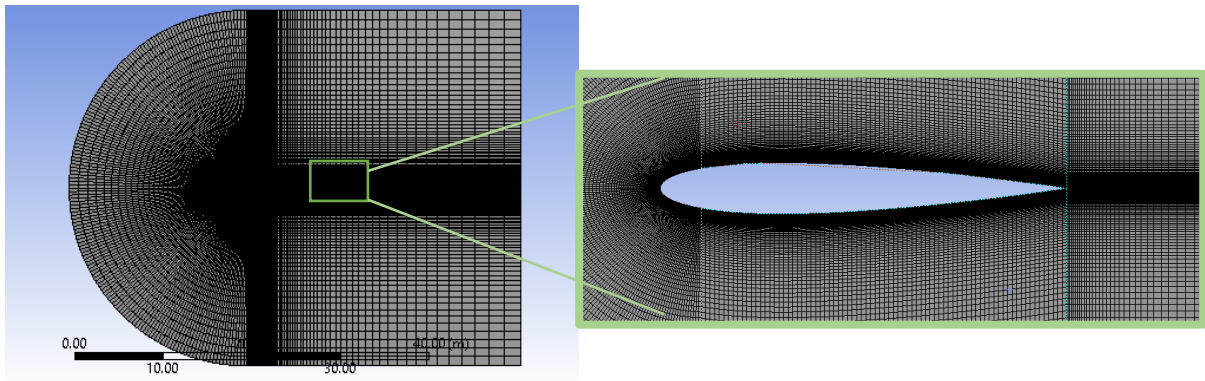


Fig. 1. Mesh of the computational domain around NACA 0012

The inlet boundary condition is set to 5% turbulence intensity of velocity inlet. However, the outlet boundary condition is set to 5% turbulence intensity of pressure outlet. No slip wall boundary condition is set on the airfoil surface. SIMPLEC algorithm is used to solve the pressure-velocity coupling in momentum equation and momentum, turbulence kinetic energy and dissipation ratio. Second-order upwind model is applied for discretisation of convection term due to its accuracy in calculation [29]. The numerical detail that used in simulation is summarised as Table 1.

Table 1 Numerical detail for simulation on flow across NACA 0012 airfoil

Input	Value
Reynold number	3×10^6
Operating temperature	300 K
Density of fluids	1.225 kg/m^3
Viscosity of fluids	$1.7894 \times 10^{-5} \text{ kg/ms}$
Flow velocity	44 m/s
Chord length	1 m
Angle of attack	-12° to 20°

Results and Discussion

Lift and Drag Coefficient

A graph of lift coefficient versus angle of attack was plotted as shown as Figure 2 while a graph of drag coefficient versus angle of attack was plotted as shown as Figure 3. By observing the graph in Figure 2, the experimental curve can be noticed that the lift coefficient increases linearly while increasing the angle of attack at low angle of attack which is in the range of -12° to 15° . In -9° to 9° of angle of attack, all turbulence models show a very good agreement with the experimental data for lift coefficient [30]. All the turbulence models also increased linearly from -10° to 10° which similar to experimental data. After 12° angle of attack, all the turbulence model started to have some different with experimental data. However, the standard $k-\Omega$ model had the lowest deviation with experimental data compared to other turbulence models while the flow starts to separate. At higher angle of attack, the upper surface flow of the airfoil began to separate. The flow no longer follows along the airfoil surface. This is a region above the upper surface and near the airfoil trailing edge. In this region, the velocity is low and the flow reverses in place in a turbulence motion. This separation flow is known as trailing edge separation [28].

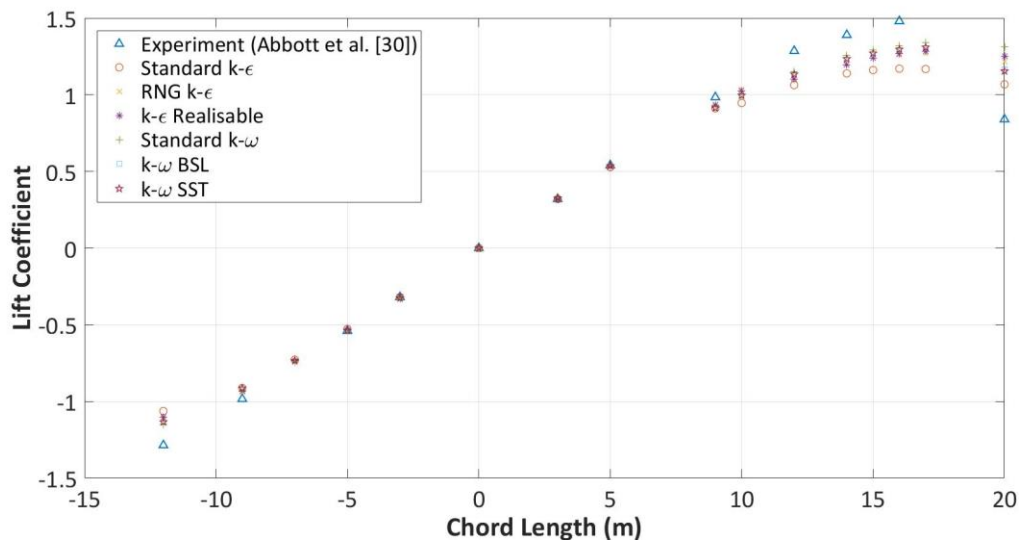


Fig. 2. Comparison graph of lift coefficient versus angle of attack between experimental data from [30] and simulation result of turbulence models for NACA 0012 airfoil.

As the angle of attack is increased further, the separation flow region continuous moves toward the leading edge. At certain critical angle of attack, the lift coefficient will drop rapidly and drag coefficient will increase rapidly. This condition was called stall and the angle of attack where the stall begin was known as stall angle. This condition is called stall and the critical

angle of attack is called stall angle. In the stall angle, the airfoil will have a maximum lift coefficient [31]. The four turbulence models were noticed that the stall angle is around 16° to 17° by observing both Figure 2 and 3. Around the stall angle, the standard $k-\omega$ is closer to the experimental data.

For drag coefficient, all the turbulence models overpredict the drag coefficient by observing Figure 3. However, all the turbulence models still get the same shape of curve with the experimental data. By comparing the turbulence models with the experimental data, the standard $k-\omega$ show the best agreement with experimental data in drag coefficient curve. The overprediction of the drag coefficient may be caused by the sensitivity of drag coefficient to the roughness of surface [32]. Indeed, the real airfoil has laminar flow over the forward half. The turbulence models are unable to calculate the transition point between laminar and turbulent. The models just consider that the boundary layer is turbulent along the chord length. However, in theory, the turbulence boundary layer will carry more energy. Thus, the drag coefficient is much higher than the viscous boundary layer. By observing and comparing the curves in drag coefficient, the $k-\omega$ SST model is found to have a better agreement with experimental data.

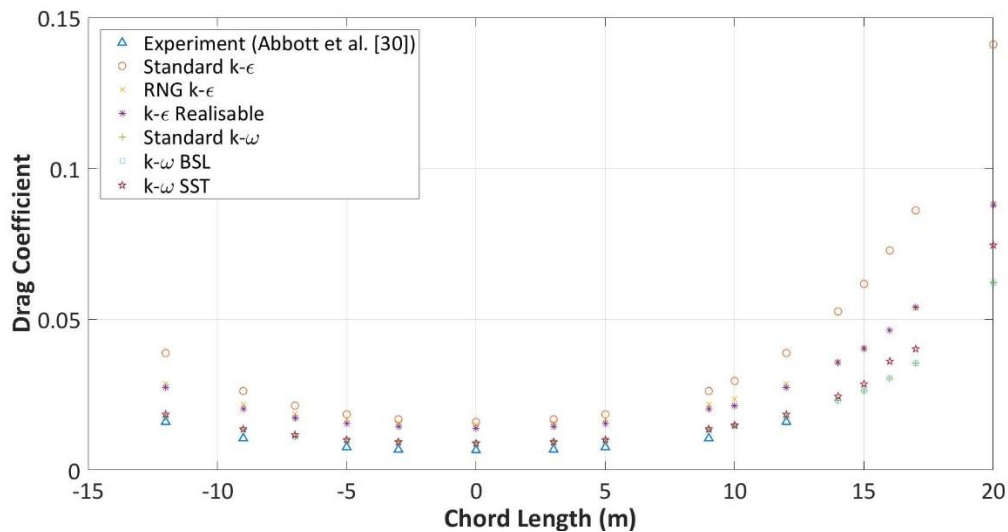


Fig. 3. Comparison graph of drag coefficient versus angle of attack between experimental data from [30] and simulation result of turbulence models for NACA 0012 airfoil.

A chart of Root Mean Square Error of drag coefficient and lift coefficient in four different turbulence models for NACA 0012 was plotted as shown as Figure 4. By comparing the root mean square error in lift coefficient, $k-\omega$ SST model is noticed that have the lowest root mean square error among the models. However, $k-\omega$ BSL model is found that have the lowest root mean square error in drag coefficient model. In overall comparison in lift and drag coefficient, $k-\omega$ SST model should be the best model among the four model. The $k-\omega$ SST model have a lowest root mean square error in lift coefficient and second lowest root mean square error in drag coefficient. Thus, the $k-\omega$ SST model can predict the most accurate lift coefficient and acceptable drag coefficient among the four turbulence models in NACA 0012.

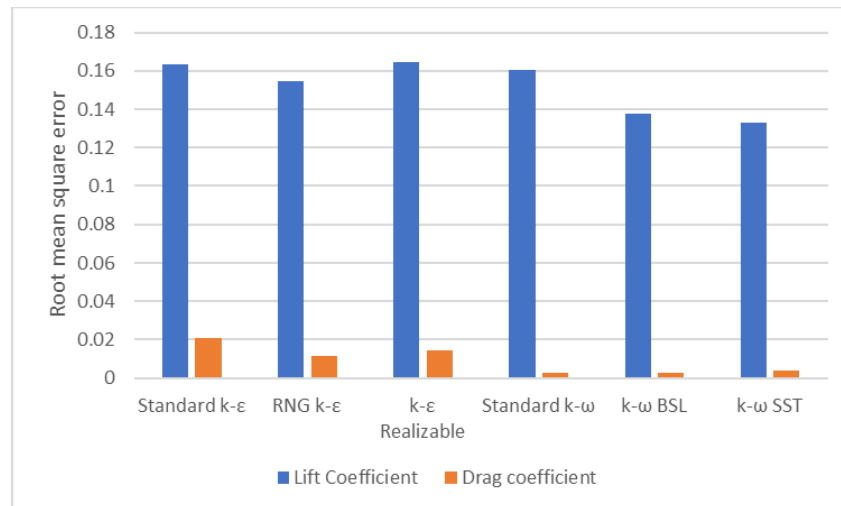


Fig. 4. Chart of Root Mean Square Error of drag coefficient and lift coefficient in different turbulence models for NACA 0012.

Pressure Coefficient

The distribution of pressure coefficient on NACA 0012 airfoil at different angle of attack in different turbulence models are shown as Figs 5-11. However, the experimental data [32] of distribution of pressure coefficient on NACA 0012 airfoil at 0°, 10° and 15° angle of attack in four turbulence models was shown as Fig. 7, 9 and 10. By comparing the different turbulence model result, it showed that all the turbulence models had almost same curve with each other but standard $k-\epsilon$ shown small deviation with other turbulence models at lower and higher angle of attack (-12°, 12° and 20°). Moreover, the $k-\epsilon$ Realizable model also shown small deviation with other turbulence models at 20° angle of attack. By comparing the simulation result with the experimental data, the turbulence models showed a very good agreement with the experimental data.

While the angle of attack was less than zero, the pressure coefficient of upper surface was positive and lower surface was negative. This showed that lift force pointed below while the angle of attack was negative or less than zero. However, the pressure coefficient of upper surface was negative and lower surface was positive when the angle is more than zero. This showed that lift force pointed upward while the angle of attack was positive or more than zero. By observing the result, it can be noticed that the leading edge had a larger curvature. This showed that the flow on the surface had a large acceleration and lower static pressure on the surface. It also can be seen that the angle of attack between -10° to 10° had obvious inclination at the trailing edge. While the angle of attack was closing to zero, the inclination was increased. This was an anti-curvature section on the trailing edge. In this section, the velocity was lower and the pressure was increased. From the result, it also can be noticed that while the angle of attack was increased from 0° to 20° or decreased from 0° to -12°, the difference of pressure coefficient between upper and lower surface at leading edge was increased. This showed that when the difference of pressure coefficient between upper and lower surface at leading edge was higher, the lift force mostly come from leading edge.

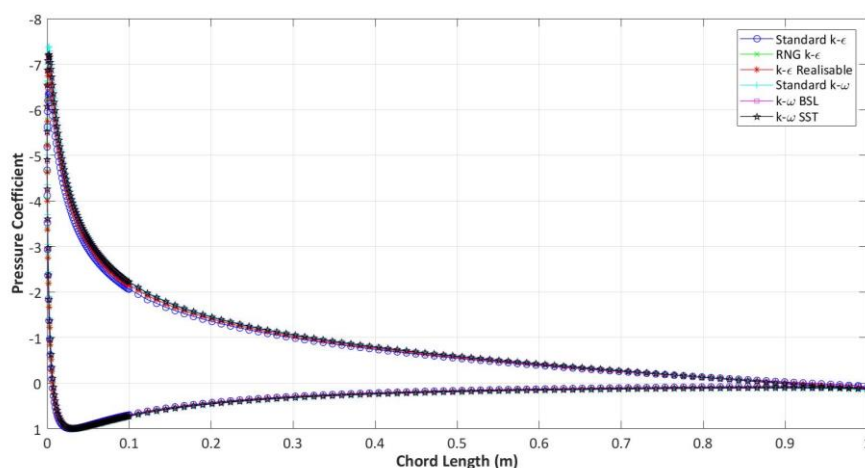


Fig. 5. Graph of pressure coefficient against airfoil position for four different turbulence models at -12° angle of attack on NACA 0012.

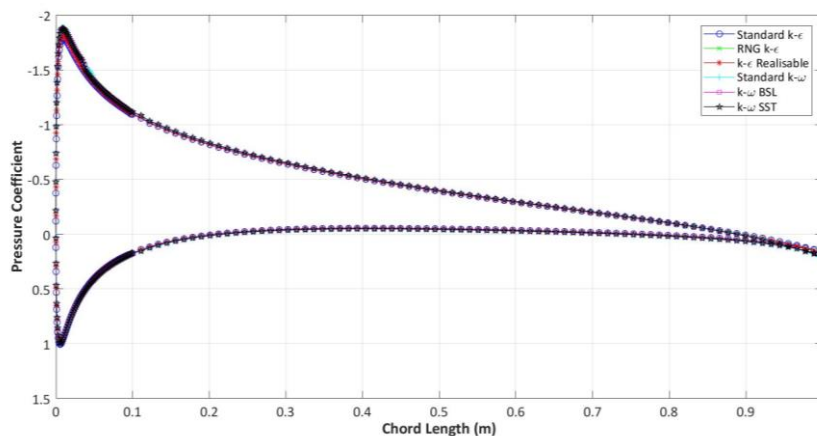


Fig. 6. Graph of pressure coefficient against airfoil position for four different turbulence models at -5° angle of attack on NACA 0012.

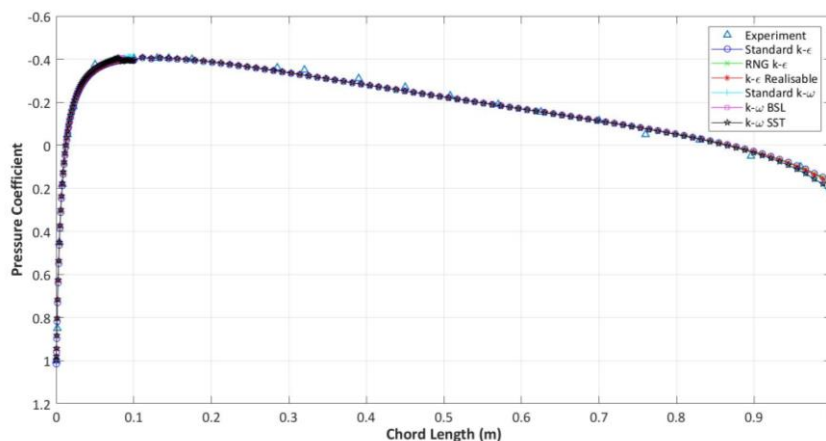


Fig. 7. Graph of pressure coefficient against airfoil position for four different turbulence models at 0° angle of attack on NACA 0012.

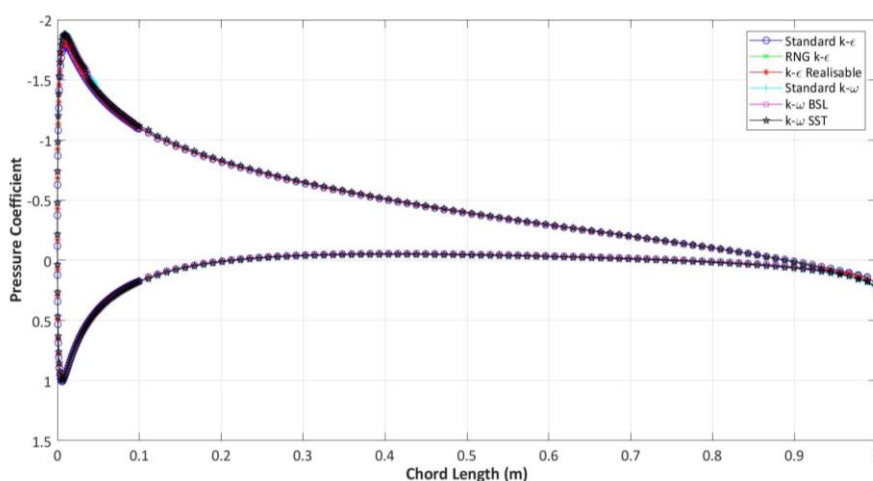


Fig. 8. Graph of pressure coefficient against airfoil position for four different turbulence models at 5° angle of attack on NACA 0012.

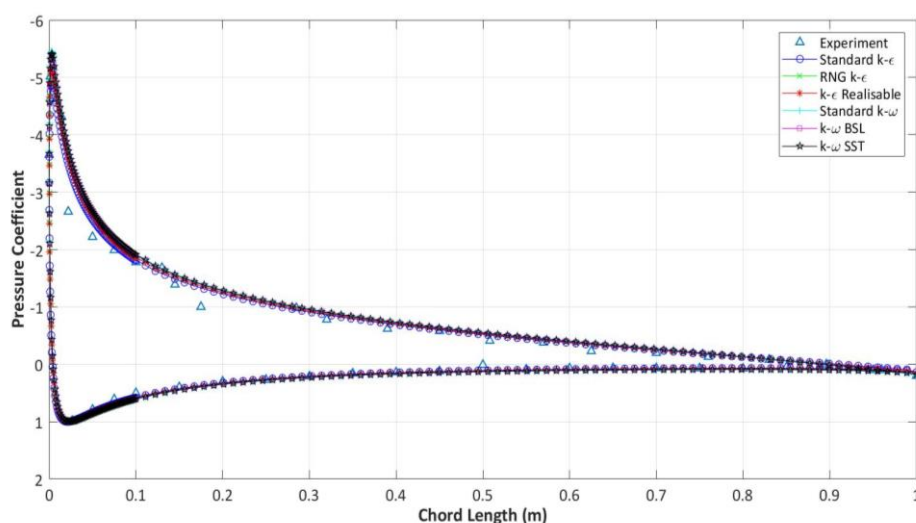


Fig. 9. Graph of pressure coefficient against airfoil position for four different turbulence models at 10° angle of attack on NACA 0012.

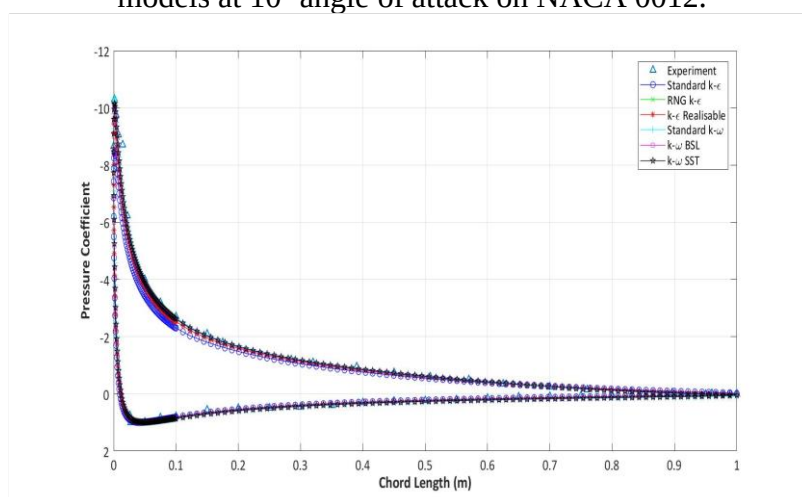


Fig. 10. Graph of pressure coefficient against airfoil position for four different turbulence models at 15° angle of attack on NACA 0012.

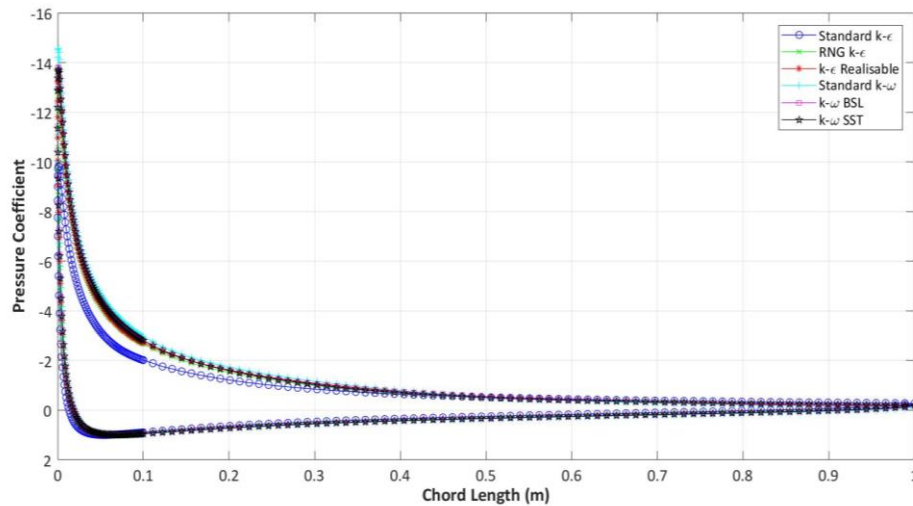


Fig. 11. Graph of pressure coefficient against airfoil position for four different turbulence models at 20° angle of attack on NACA 0012.

Contours of static pressure and velocity magnitude

The general flow pattern of the flow across NACA 0012 is discussed here by taking the simulation results by RNG $k-\epsilon$ model as example. Figure 12 showed contours of static pressure across NACA 0012 at -12° to 20° angle of attack. In addition, Figure 13 showed contours of velocity magnitude across NACA 0012 at -12° to 20° angle of attack.

At 0° angle of attack, the static pressure across NACA 0012 were symmetric. Thus, the lift force was zero in this angle of attack. The stagnation point also was noticed that located exactly at the nose of the airfoil. The maximum pressure and zero velocity occurred at the stagnation point. While the angle of attack increase in positive or negative, the static pressure across airfoil became non-symmetric. As the angle of attack increase from 0° to 20°, the pressure on lower surface became higher than the pressure on upper surface according to Bernoulli's principle. The imbalance of pressure caused the lift force in upward direction on airfoil. However, while the angle of attack increase in negative, the lift force was negative due to the higher static pressure on upper surface. By the contours, low velocity can be observed in high pressure region. Moreover, when the angle of attack increases positively from zero angle, the stagnation point is moved toward the trailing edge on the bottom surface. In opposite, the stagnation point is moved toward the trailing edge on the upper surface as the angle of attack increase in negative from zero angle. Thus, the lift coefficient will continue increase with angle of attack until maximum in stall angle then start to decrease. As the angle of attack up to about 12°, the boundary layer still fully attached the airfoil. However, if the angle of attack further increases from 10°, the boundary layer is thickened from the trailing edge and move toward leading edge direction. Thus, the boundary layer separation is started to occur. In the end, lift force will decrease obviously and drag force increase rapidly. This condition also known as stall.

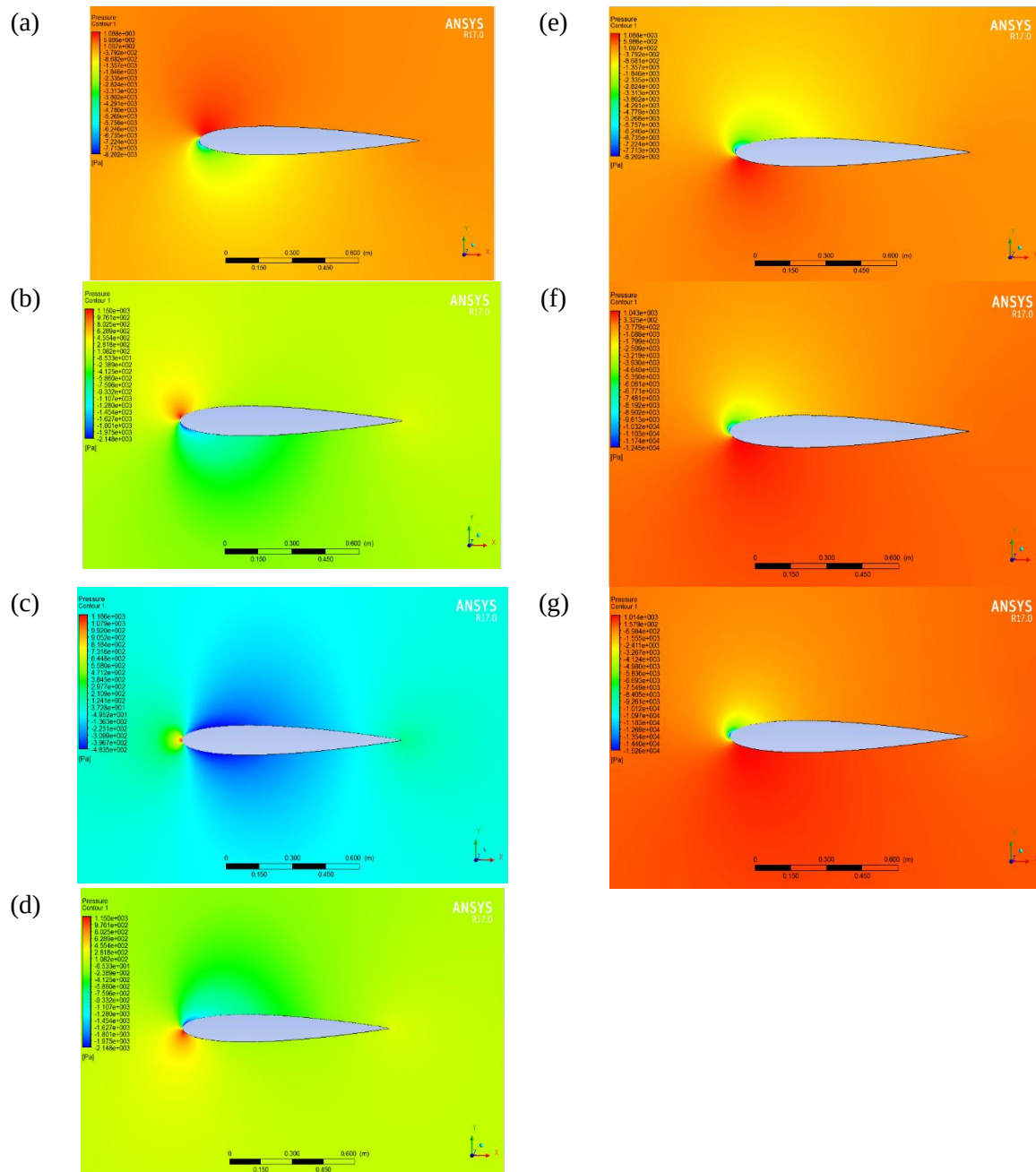


Fig. 12. Contours of static pressure across NACA 0012 at (a) -12° , (b) -5° , (c) 0° , (d) 5° , (e) 12° and (f) 16° and (g) 12° .

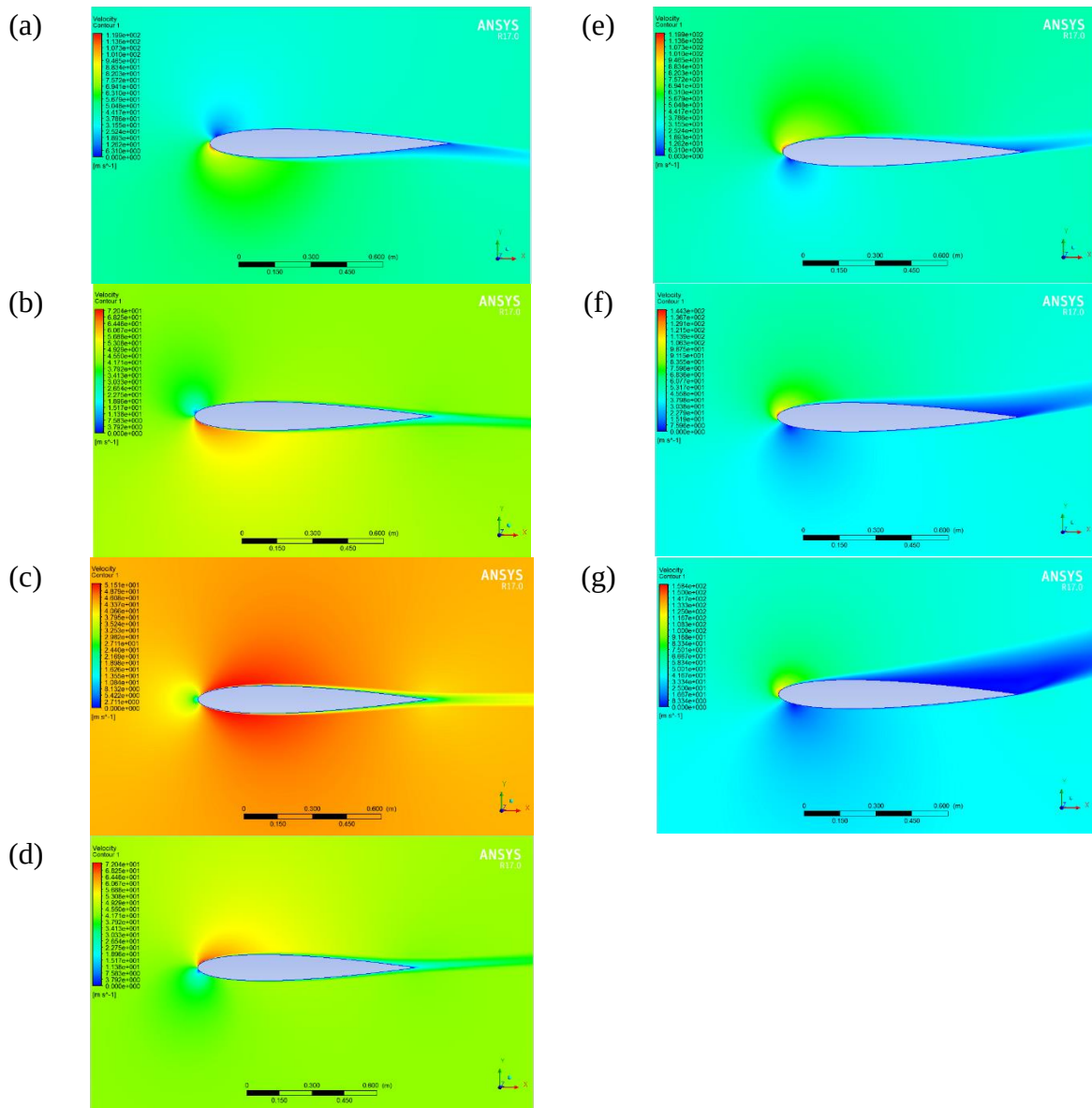


Fig. 13. Contours of velocity across NACA 0012 at (a) -12° , (b) -5° , (c) 0° , (d) 5° , (e) 12° and (f) 16° and (g) 12° .

Convergence rate

Table 2 was tabulated to show number of iterations until 0.0001 for the turbulence model at 0° angle of attack. By the result, RNG k- ϵ had the lowest convergence rate among all turbulence model.

Tab. 2. Convergence rate for four different turbulence models

Turbulence model	Number of iteration
Standard k- ϵ	847
RNG k- ϵ	953
k- ϵ realizable	869
Standard k- ω	390
k- ω BSL	616
k- ω SST	568

Conclusions

In this study, the flow across NACA 0012 airfoil is studied in various angle of attack which is -12° to 20° by using four different two-equation turbulence models in ANSYS Fluent R17. The two-equation turbulence models that used in the study are standard $k-\varepsilon$ model, RNG $k-\varepsilon$ model, $k-\varepsilon$ Realizable model, standard $k-\omega$ model, $k-\omega$ BSL model and $k-\omega$ SST model. It is aiming to analyse the velocity and pressure field around NACA 0012 airfoil in different angle of attack. Moreover, lift, drag and pressure coefficient are also analysed in different angle of attack and compared to experimental data. The convergence rate of two-equation turbulence models is compared in this study.

For lift coefficient result, $k-\omega$ SST model showed the best agreement with the experimental result among other turbulence models. However, the $k-\omega$ BSL model showed the best agreement with experimental result in drag coefficient among other turbulence models. However, the $k-\omega$ SST model has a best result in overall. It can get the best result in lift coefficient and acceptable result in drag coefficient. The predicted drag coefficients of the turbulence models are higher than the experimental data. This overprediction is occurred because the actual airfoil has laminar flow over forward half. In overall, RNG $k-\varepsilon$ model is noticed that is a most suitable turbulence model to study flow across NACA 0012 as it gets a best result in lift coefficient and second upper good result in drag coefficient.

In pressure coefficient result, it showed that all the turbulence models had almost same curve with each other but standard $k-\varepsilon$ shown small deviation with other turbulence models at lower and higher angle of attack (-12° , 12° and 20°). By comparing the simulation result with the experimental data, the turbulence models showed a very good agreement with the experimental data.

In comparison of convergence rate, RNG $k-\varepsilon$ model showed the lower convergence rate. Thus, RNG $k-\varepsilon$ model should be the best turbulence model in this study. The drag and lift coefficient result also showed the RNG $k-\varepsilon$ model is the most suitable turbulence model in overall.

Acknowledgement

The authors would like to thank the support from Takasago Research Fund (Vote No: 4B314) from Malaysia-Japan International Institute of Technology, Universiti Teknologi Malaysia and the Pioneer Scientist Incentive Fund (PSIF) with vote number Proj-In-FETBE-038.

References

- [1] Roland Schiestel. *Modeling and Simulation of Turbulent Flows* (2010).
- [2] Pyakurel, P., J. H. VanZwieten, M. Dhanak, N. I. Xiros. "Numerical modeling of turbulence and its effect on ocean current turbines." *International Journal of Marine Energy* 17 (2017): 84–97.
- [3] Anup, K., J. Whale, S. P. Evans, P. D. Clausen. "An investigation of the impact of wind speed and turbulence on small wind turbine operation and fatigue loads." *Renewable Energy* 146 (2020): 87–98.
- [4] Zhang, T., Y. O. Zhang, H. Ouyang. "Structural vibration and fluid-borne noise induced by turbulent flow through a 90° piping elbow with/without a guide vane." *International Journal of Pressure Vessels and Piping* 125 (2015): 66–77.
- [5] Bryant, D. B., E. M. Sparrow, J. M. Gorman. "Turbulent pipe flow in the presence of centerline velocity overshoot and wall-shear undershoot." *International Journal of Thermal Sciences* 125 (2018): 218–230.

- [6] Liu, H., D. Chen, L. Hu, D. Yuan, H. Gao. "Numerical investigations on flow-induced vibration of fuel rods with spacer grids subjected to turbulent flow." *Nuclear Engineering and Design* 325 (2017): 68–77.
- [7] Quen, L. K., A. Abu, N. Kato, P. Muhamad, L. K. Tan, H. S. Kang. "Performance of two- and three-start helical strakes in suppressing the vortex-induced vibration of a low mass ratio flexible cylinder." *Ocean Engineering* 166 (2018): 253–261.
- [8] Zambra, C. E., C. Rosales, N. O. Moraga, M. Ragazzi. "Self-heating in a bioreactor: Coupling of heat and mass transfer with turbulent convection." *International Journal of Heat and Mass Transfer* 54(23–24) (2011): 5077–5086.
- [9] Liu, Y., Y. Zhang, L. Zhou. "Numerical study on bubble-liquid two-phase turbulent hydrodynamics in extremely narrow shape bioreactor." *International Communications in Heat and Mass Transfer* 108 (2019): 104286.
- [10] Gui, N., J. Yan, Z. Li, J. Fan. "Direct numerical simulation of confined swirling jets." *International Journal of Computational Fluid Dynamics* 28(1–2) (2014): 76–88.
- [11] Nagata, T., T. Nonomura, S. Takahashi, Y. Mizuno, K. Fukuda. "Direct numerical simulation of flow around a heated/cooled isolated sphere up to a Reynolds number of 300 under subsonic to supersonic conditions." *International Journal of Heat and Mass Transfer* 120 (2018): 284–299.
- [12] Orazzo, A., S. Tanguy. "Direct numerical simulations of droplet condensation." *International Journal of Heat and Mass Transfer* 129 (2019): 432–448.
- [13] Chu, X., G. Yang, S. Pandey, B. Weigand. "Direct numerical simulation of convective heat transfer in porous media." *International Journal of Heat and Mass Transfer* (2019): 11–20.
- [14] Nicoud, F., F. Ducros. "Subgrid-scale stress modelling based on the square of the velocity gradient tensor." *Flow, Turbulence and Combustion* 62(3) (1999): 183–200.
- [15] Inagaki, M., T. Kondoh, Y. Nagano. "A mixed-time-scale SGS model with fixed model-parameters for practical LES." *Journal of Fluids Engineering, Transactions of the ASME* 127(1) (2005): 1–13.
- [16] Kim, M., J. Lim, S. Kim, S. Jee, J. Park, D. Park. "Large-eddy simulation with parabolized stability equations for turbulent transition using OpenFOAM." *Computers and Fluids* 189 (2019): 108–117.
- [17] Choi, H., P. Moin. "Grid-point requirements for large eddy simulation: Chapman's estimates revisited." *Physics of Fluids* 24(1) (2012).
- [18] Badoe, C. E., Z. T. Xie, N. D. Sandham. "Large Eddy simulation of a heaving wing on the Cusp of transition to turbulence." *Computers and Fluids* 184 (2019): 64–77.
- [19] Arora, K., K. Chakravarthy, D. Chakraborty. "Large eddy simulation of supersonic, compressible, turbulent mixing layers." *Aerospace Science and Technology* 86 (2019): 592–598.
- [20] Toubiana, E., R. Gautier, D. Bougeard, S. Russeil. "Large Eddy Simulation of transitional flows in an elliptical finned-tube heat exchanger." *International Journal of Thermal Sciences* 144 (2019): 158–172.
- [21] Argyropoulos, C. D., N. C. Markatos. "Recent advances on the numerical modelling of turbulent flows." *Applied Mathematical Modelling* 39(2) (2015): 693–732.
- [22] Launder, B. E., D. B. Spalding. "The numerical computation of turbulent flows." *Computer Methods in Applied Mechanics and Engineering* 3(2) (1974): 269–289.
- [23] Yakhot, V., S. A. Orszag, S. Thangam, T. B. Gatski, C. G. Speziale. "Development of turbulence models for shear flows by a double expansion technique." *Physics of Fluids A* 4(7) (1992): 1510–1520.
- [24] Shih, T. H., W. W. Liou, A. Shabbir, Z. Yang, J. Zhu. "A new $k-\varepsilon$ eddy viscosity

- model for high reynolds number turbulent flows.” *Computers and Fluids* 24(3) (1995): 227–238.
- [25] Wilcox, D. C. “Reassessment of the scale-determining equation for advanced turbulence models.” *AIAA Journal* 26(11) (1988): 1299–1310.
 - [26] Menter, F. R. “Two-equation eddy-viscosity turbulence models for engineering applications.” *AIAA Journal* 32(8) (1994): 1598–1605.
 - [27] Menter, F. R. “Zonal two equation κ - ω turbulence models for aerodynamic flows.” *AIAA 23rd Fluid Dynamics, Plasmadynamics, and Lasers Conference, 1993* (1993).
 - [28] Yao, J., W. Yuan, J. Wang, J. Xie, H. Zhou, M. Peng, Y. Sun. “Numerical simulation of aerodynamic performance for two dimensional wind turbine airfoils.” *Procedia Engineering* 31 (2012): 80–86.
 - [29] Tey, W. Y., R. Y. W. Hong, Y. Asako, H. S. Kang, K. C. Ng. “Analysis on computational efficiency of convection discretisation schemes in SIMPLE algorithm.” *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 58(1) (2019): 100–117.
 - [30] Albert Abbot, Ira Doenhoff. *THEORY OF Including a Summary of Airfoil Data* (1959).
 - [31] Jeppe Johansen and Jens N. Sørensen. *Prediction of laminar/turbulent transition in airfoil flows*, 36(4) (1999).
 - [32] Ahmed, T., T. Md. Amin, S. M. R. Islam, S. Ahmed. “Computational study of flow around a NACA 0012 wing flapped at different flap angles with varying Mach numbers.” *The Global Journal of Researches in Engineering* 13(4-J) (2013).

EVALUATING THERMAL PERFORMANCE OF POUCH TYPE LITHIUM POLYMER BATTERY CELL

Adam Maxwell Doumin¹

Akhmal Fahrudin¹

Zul Hilmi Che Daud^{2*}

Zainab Asus²

Izhari Izmi Mazali¹

Mohd Farid Muhamad Said²

Fitri Yakub³

¹School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia

²Automotive Development Centre (ADC), School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia

³Malaysia-Japan International Institute of Technology, Universiti Teknologi Malaysia, 54100, Kuala Lumpur, Malaysia

*(Corresponding Author's Email: hilmi@mail.fkm.utm.my)

Abstract: *This article presents an experimental study of battery temperature under constant discharge rate. This study consider several constant discharge rates, constant air velocities, gap between battery cells in the battery module and the placement of thermocouple sensors in order to study the thermal behaviour of the battery cell. The relationship of temperature and discharge rates at different point on battery cell surface is compared. The heat generation on the battery surface as a function of discharge time are captured by RTD thermocouple and linked to the LabVIEW software. An axial fan creates constant air velocities that help in removing heat away from battery module during discharge process. Variables that are considered in this experiment are the discharging rate, air velocities and the thermal behaviour of the Lithium-ion battery cell on various points across the battery cell surface.*

Keywords: *Lithium-Ion Battery, Thermal Behavior, Hybrid Electric Vehicle, Experimental Study*

Introduction

In recent years, many car manufacturers leaning towards producing alternative powertrain vehicle instead of conventional internal combustion vehicle. The production of Hybrid Electric Vehicle (HEV) and Electric Vehicle (EV) as alternative powertrain vehicle are growing fast. The main purpose of using these alternative ways is to reduce the usage of petrol and diesel and also to reduce exhaust emission which instigating greenhouse effect [1,2]. Recently, some HEVs and EVs available in the market are using NiMH batteries as their power source, but the usage of Lithium-ion batteries (LiB) are expected to grow fast in HEV and EV markets. The LiB a more preferable power storage medium for HEV and EV since it has high energy density, low self-discharge rate, low maintenance and lightweight [3, 4, 5, 6]. Although LiB possess good characteristic for equipping in HEVs and EVs, it has a downside of temperature sensitivity [3]. If LiB is used at temperature higher than the allowable operating temperature, it can significantly affect the lifecycle and the capacity of the battery. Operating LiB below the minimum temperature range may result in low efficiency of power delivery, thus lower performance [7, 8, 9, 10, 11]. Optimum operating temperature of LiB is within the range of 20°C to 65°C.

In this experimental study focused on the thermal behavior of LiB at different discharge rates and different cell locations through cooling system. The Lithium-ion battery module is placed inside an airflow tunnel while thermocouple sensors are placed on the battery cell surface and anemometer is used to measure the air velocity in determining the distribution of temperature at various points on the battery cell surface during the discharging process with different discharging rates.

Experimental Set Up

Alignment and installation of the Lithium-ion battery cells

Three battery cells of 40Ah each are packed into a battery module. The cells are link in series connection making a module of 12 Volts, 40Ah. The gap between each battery cell is 3mm. Figure 1 shows the arrangement of gap divider (from acrylic sheet) on the battery surface and with three RTD thermocouples. The placement of the three RTDs is close to positive and negative electrode connector. Previous research shows that this location is expected to have the most significant temperature. The completed battery module is then placed inside an airflow tunnel as shown in figure 2.

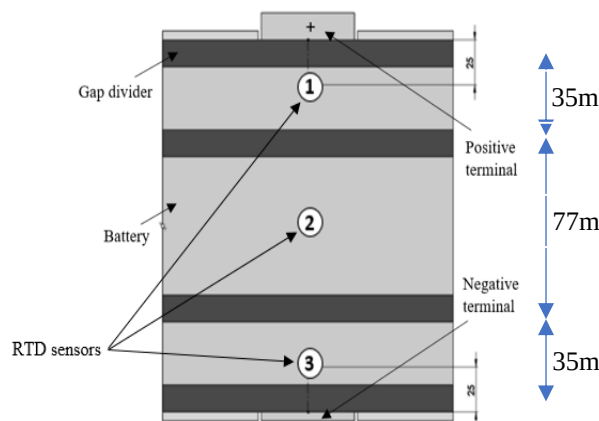


Fig. 1. Gap divider arrangement and placement of RTDs on battery cell surface



Fig. 2. Completed battery module and air tunnel

Experimental Procedure

After alignment of battery module is done, the battery module is then connected for discharging process. An electronic load machine is used to discharge the battery. The positive terminal from the battery module is connected to positive terminal on the machine, labeled in red while negative terminal from the battery module is connected to negative terminal on the machine, labeled in black. Figure 3 shows the connection from battery to electronic load.

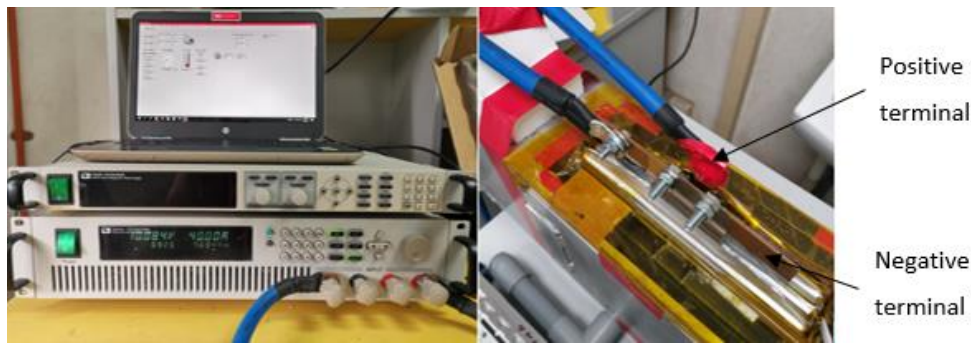


Fig. 3. Connection electronic load machine to battery module

Experiment procedure are as follow;

- i. The experiment is conducted only when the battery cells are full charge. It is considered that the battery cells are fully charged or 100 % SOC when the battery total voltage is equal to 12 Volts or 4 Volts per cell.
- ii. The experiment is started by setting the fan speed in order to obtain the desired constant cooling air velocity, 1.5 m/s.
- iii. At this stage the essential information is recorded; air velocity, air temperature, and battery cell voltage.
- iv. The battery cells are then discharged at a constant discharge rate, 1C discharge rate (40 Amps).
- v. The discharge process is continued until the battery SOC reaches a limit of 20% SOC
- vi. The cooling air velocity is maintained constant at 2 m/s from $t=0s$ until 100 seconds after the end of discharge process, in order to reduce the battery cell temperature rapidly after the end of discharge.
- vii. During discharge process, several parameters are monitored to ensure the validity of experiment result and for safety reasons. These include the constant cooling air velocity at 1.5 m/s, cooling air temperature, maximum cell surface temperature which has to stay below $60^{\circ}C$, and the battery voltage above 2.8 Volts
- viii. The experimental results including cell temperatures, battery voltage and current and cooling air velocity and temperature are recorded automatically through Labview software

The experimental procedure is than repeated for 3C and 5C discharge rate.

Results and Discussion

Graph 4-6 show the battery cell surface temperature for discharge rate of 1C, 3C and 5C respectively. T1, T2 and T3 are the notations for location 1, 2 and 3 respectively on the battery surface as shown in figure 1. In general, for all 3 discharges rate point 1 (near to positive electrode terminal) is the hottest point followed by point 3 (near to negative electrode terminal) and the lowest is point 2. During discharge process, all the current flows to the positive and negative terminal from the entire electrode plate [12]. Thus, the current densities and consequently the temperature of positive and negative terminal are higher than the other parts of the battery cell [12]. Furthermore, the temperature at the positive terminal is higher than the negative terminal due to lower electrical conductivity of the positive electrode, despite the fact that the current flow in both terminal are similarly high [13].

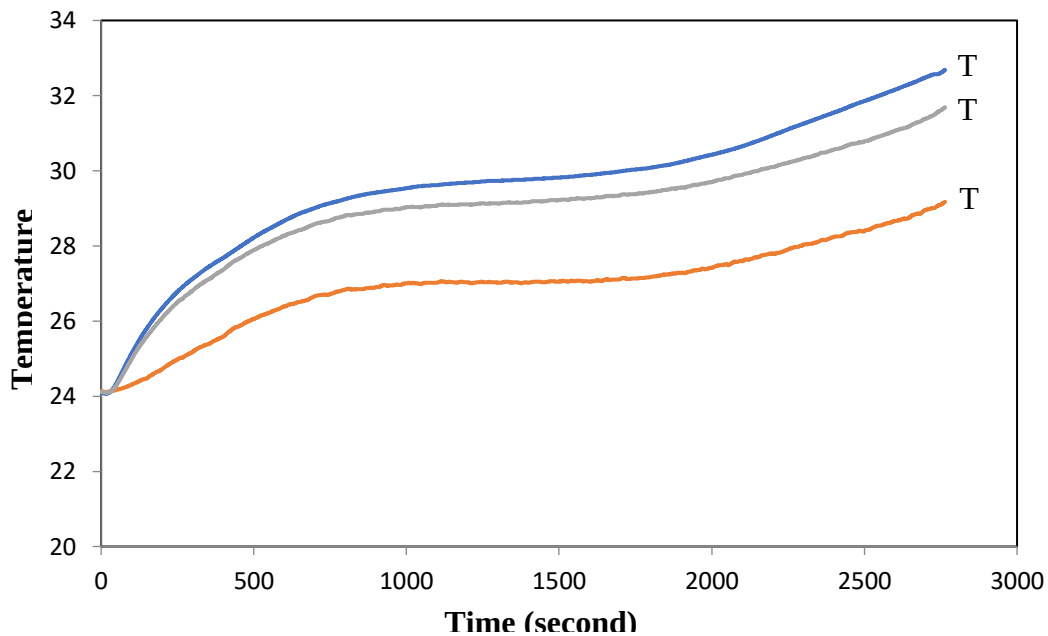


Fig. 4. Battery cell surface temperature for 1C discharge rate.

Figure 4 shows the battery cell surface temperature at 1C discharge rate for 3 different locations with constant air velocities of 1.5m/s. At the end of discharge process, the highest temperature is at T1 with 33°C followed by T2 with 31°C. The lowest temperature is at T2 with 29.0°C. In general, the temperature increase is relatively small. This is because only small amount of current is discharged from the battery. Heat generation inside the battery is proportional to the amount of discharge current [14, 15]. The higher the amount of current discharged, produced more heat thus resulting in higher temperature increase.

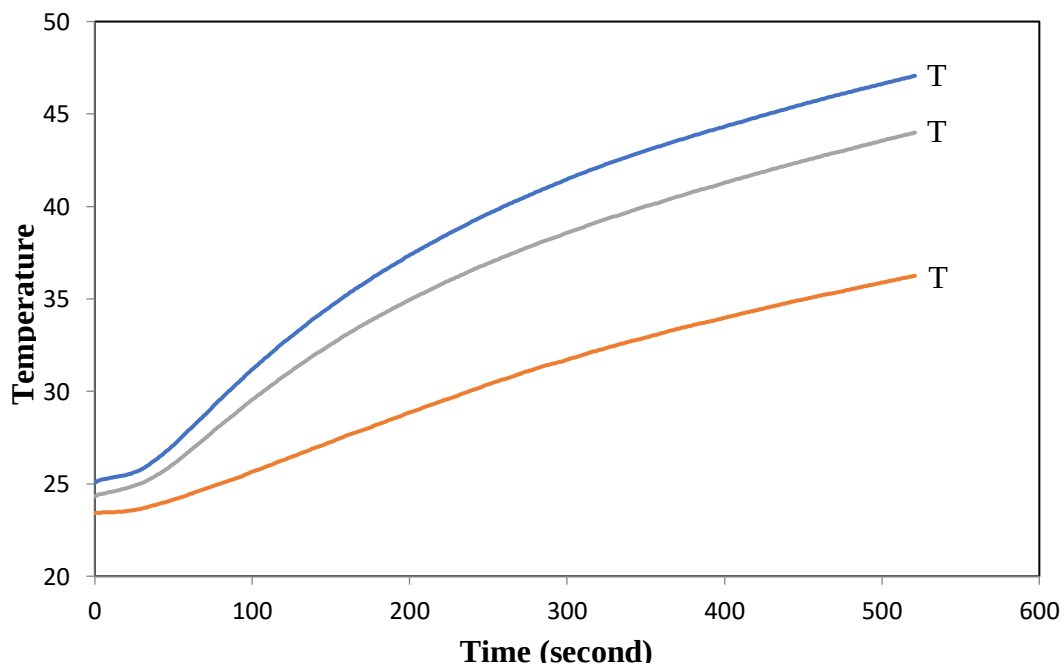


Fig. 5. Battery cell surface temperature for 3C discharge rate.

Figure 5 shows the battery cell surface temperature at 3C discharge rate for 3 different locations with constant air velocities of 1.5m/s. Result shows same trend as in figure 4, highest temperature T1, followed by T3, and T2. T1 recorded the end of discharge temperature of 47°C, 44°C for T3 and 36°C for T2. The temperature increase is quite significant as the end of discharge temperature approaching 50°C. It shows that for this type of battery a significant cooling system is needed if the battery is used under higher discharge rate. This condition should be put in consideration as allowable operating temperature is normally 60°C to avoid damaging the battery.

Figure 6 shows the battery cell surface temperature at 5C discharge rate for 3 different locations with constant air velocities of 1.5m/s. Experiment stop at 205 seconds when the battery reach the cut-off voltage of 8.4V. This is far from theoretical discharge duration which should last until 570 second. Experimental time is only about one-third of theoretical time. Figure 6 shows that temperature at T1 52°C, 45°C at T3 and 34°C at T2. Even though the temperature still not reaching 60°C, but discharge time is too short caused by the rapid decreased of voltage. From this result, it can be concluded that this battery do not support deep discharge (more than 3C).

Figure 7 shows the cell surface temperature behaviour at T1 with different discharge rates of 1C, 3C and 5C under 1.5 m/s cooling air velocities. This result is to compare the effect of amount of discharge current on the cell temperature. Results confirm the theory that higher discharge rate produced more heat, thus more temperature increase.

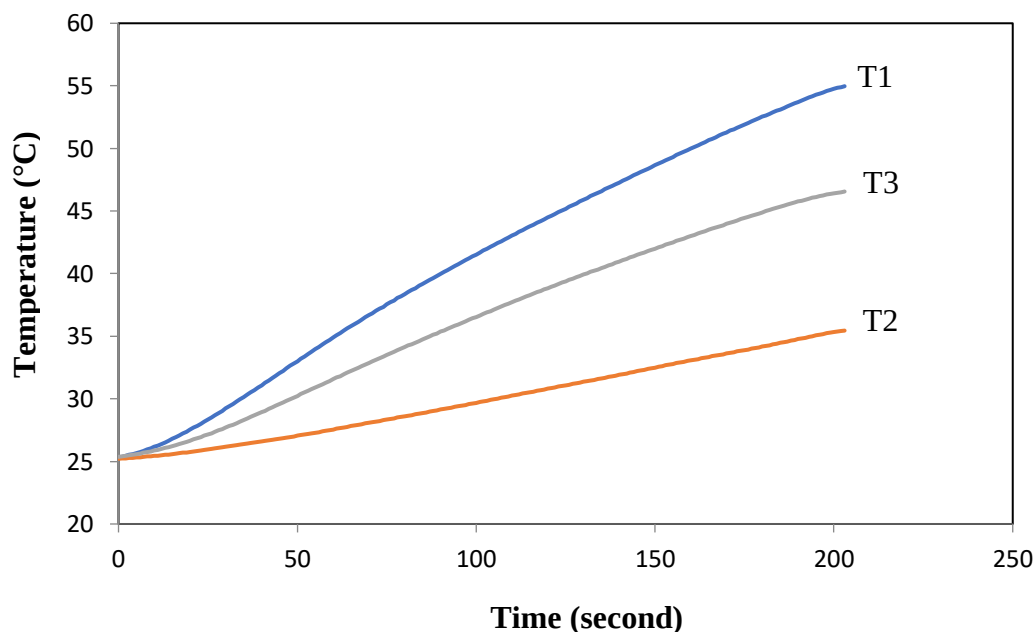


Fig. 6. Battery cell surface temperature for 5C discharge rate.

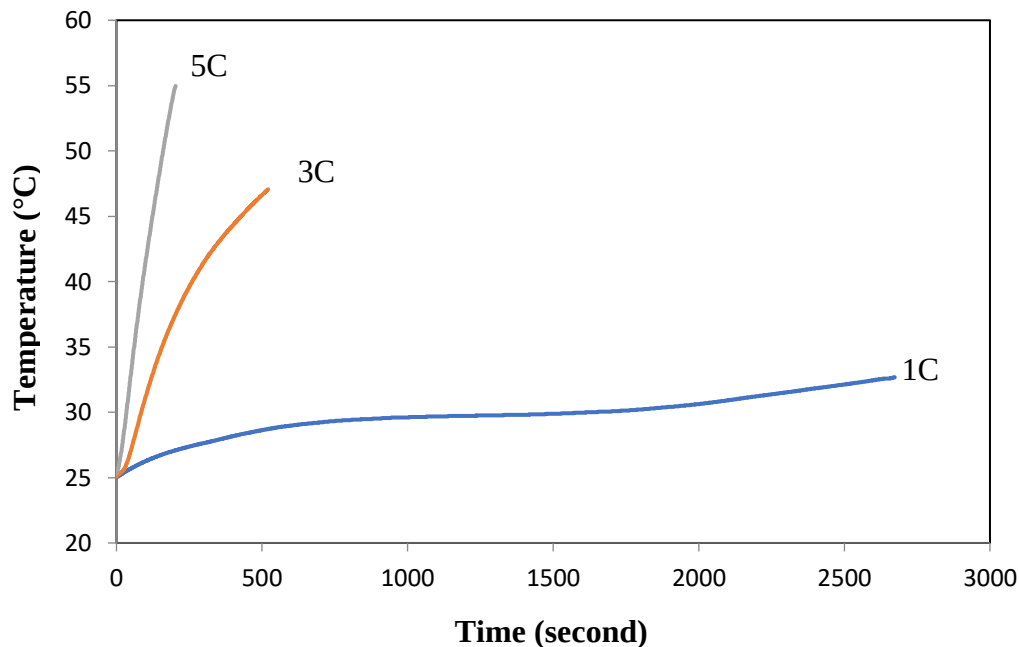


Fig. 7. Cell temperature at T1 for 3 different discharge rates

Conclusions

Throughout this experimental study, it is clear that the distribution of temperature across the surface of battery cell is not uniform. Temperature near positive and negative terminal is higher than other location because current is flowing through these small electrodes during discharge process. Current densities and consequently the temperature of positive and negative terminal are higher than the other parts of the battery cell. Furthermore, the temperature at the positive terminal is higher than the negative terminal due to lower electrical conductivity of the positive electrode, despite the fact that the current flow in both terminals is similarly high. The temperature increases with the increasing of discharge rate. The higher the discharge rate, the higher the heat produced by the battery. This is observed for from 1C to 3C and 5C discharged rate. Results from experiment show that this battery does only support up to 3C discharge rate. So, the battery is not suitable for deep discharging for discharge rate of 5C or above. Under 1C discharge rate, temperature increase is relatively small. However, for 3C discharge rate a more significant temperature increase is recorded thus need more attention for cooling system.

Acknowledgement

This research was funded by a grant from Research University Grant (UTM-TDR: Q.J130000.3551.06G06).

References

- [1] "Industry Week: Cars Pass Power Plants as Top Source of Us Carbon Emissions." Industry Week, 2017, accessed January, 2019, <https://www.industryweek.com/environment/cars-pass-power-plants-top-source-us-carbon-emissions>.
- [2] Lanzarotto, D, M Marchesoni, M Passalacqua, A Pini Prato, and M Repetto. "Overview of Different Hybrid Vehicle Architectures." IFAC-PapersOnLine 51, no. 9 (2018): 218-22.
- [3] Hannan, M.A., et al., "Review of energy storage systems for electric vehicle applications: Issues and challenges". Vol. 69. 2017. 771-789.

- [4] Ehsani, Mehrdad, Yimin Gao, Stefano Longo, and Kambiz Ebrahimi. *Modern Electric, Hybrid Electric, and Fuel Cell Vehicles*. CRC press, 2018.
- [5] Larminie, James, and John Lowry. *Electric Vehicle Technology Explained*. John Wiley & Sons, 2012.
- [6] Hannan, Mohammad A, MS Hossain Lipu, Aini Hussain, and Azah Mohamed. "A Review of Lithium-Ion Battery State of Charge Estimation and Management System in Electric Vehicle Applications: Challenges and Recommendations." *Renewable and Sustainable Energy Reviews* 78 (2017): 834-54.
- [7] Guo, M., G.-H. Kim, and R.E. White, "A three-dimensional multi-physics model for a Li-ion battery". *Journal of Power Sources*, 2013. 240: p. 80-94.
- [8] Wang, Qingsong, Ping Ping, Xuejuan Zhao, Guanquan Chu, Jinhua Sun, and Chunhua Chen. "Thermal Runaway Caused Fire and Explosion of Lithium Ion Battery." *Journal of power sources* 208 (2012): 210-24.
- [9] Mohammadian, Shahabeddin K, and Yuwen Zhang. "Thermal Management Optimization of an Air-Cooled Li-Ion Battery Module Using Pin-Fin Heat Sinks for Hybrid Electric Vehicles." *Journal of Power Sources* 273 (2015): 431-39.
- [10] Wang, Qian, Bin Jiang, Bo Li, and Yuying Yan. "A Critical Review of Thermal Management Models and Solutions of Lithium-Ion Batteries for the Development of Pure Electric Vehicles." *Renewable and Sustainable Energy Reviews* 64 (2016): 106-28.
- [11] Abdul-Quadir, Y., P. Heikkilä, T. Lehmuspelto, J. Karppinen, T. Laurila, and M. Paulasto-Kröckel. "Thermal Investigation of a Battery Module for Work Machines." Paper presented at the 2011 12th Intl. Conf. on Thermal, Mechanical & Multi-Physics Simulation and Experiments in Microelectronics and Microsystems, 18-20 April 2011.
- [12] Kim, U.S., Shin, C.B. and Kim, C., 2009. "Modeling for the scale-up of a lithium-ion polymer battery". *Journal of Power Sources*, 189(1): 841–846.
- [13] Chacko, S. and Chung, Y.M., 2012. "Thermal modelling of li-ion polymer battery for electric vehicle drive cycles". *Journal of Power Sources*, 213: 296–303.
- [14] Zul Hilmi Che Daud, Zainab Asus, Izhari Izmi Mazali, Mohd Kameil Abdul Hamid, Adam Maxwell Doumin, Fitri Yakub. " Experimental Study of Lithium-ion Battery Thermal Behaviour Under Abuse Discharge Condition." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*, 54, Issue 1 (2019) 102-111.
- [15] Zul Hilmi Che Daud, Daniela Chrenko, El-Hassane Aglzim, Alan Keromnes, and Luis Le Moyne. "Experimental Study of Lithium-Ion Battery Thermal Behaviour for Electric and Hybrid Electric Vehicles." In *Vehicle Power and Propulsion Conference (VPPC), 2014 IEEE*, pp. 1-6. IEEE, 2014.

EFFECT OF SUCTION/INJECTION ON STAGNATION POINT FLOW OF HYBRID NANOFLUID OVER AN EXPONENTIALLY SHRINKING SHEET WITH STABILITY ANALYSIS

Nur Syazana Anuar^{1*}

Norfifah Bachok^{1,2}

Norihan Md Arifin^{1,2}

Haliza Rosali¹

¹Department of Mathematics, Faculty of Science, Universiti Putra Malaysia, 43400 UPM Serdang, Selangor, Malaysia

²Institute for Mathematical Research, Universiti Putra Malaysia, 43400, UPM Serdang, Selangor, Malaysia

*(Corresponding Author's Email: nursyazana931@gmail.com)

Abstract: This research aims to investigate the effect of suction/injection on the stagnation point flow over an exponentially shrinking sheet in hybrid nanofluid. The governing boundary layer equations are transformed into an ordinary differential equation using a similarity variable. The features of the flow and heat transfer characteristic for various values of suction/injection parameter, nanoparticle volume fraction parameter and shrinking parameter are presented graphically and discussed. The results acquired are in good agreement with previously published results. Non-unique solutions are visible for the certain range of shrinking parameter. It is worth to mention that the heat transfer rate of hybrid nanofluid is greater compared to regular nanofluid. A stability analysis is performed to identify the stability of solution and it is confirmed that the first solution is stable whereas the second solution is unstable

Keywords: Hybrid Nanofluid, Suction/Injection, Dual Solutions, Stability Analysis

Introduction

Hybrid nanofluid has been streamlined as a new class of nanofluid, characterized by its thermal properties and potential utilities that serve the purpose of increasing the rate of heat transfer. Hybrid nanofluids are fluids that consist of two kinds of nanometer-sized particles, called nanoparticles dispersed in a base fluid. These hybrid nanofluids have numerous conceivable applications in all fields of heat transfer such as micro fluidics, manufacturing, transportation, defence, medical, naval structures, acoustics and many more. Since then, many experimental and numerical research articles have been published with concept of hybrid nanofluid. Momin [1] investigated experimentally the mixed convection flow in inclined tube for hybrid nanofluid. They found that hybrid nanofluids show higher friction factor and Nusselt number when compared to pure water. Heat transfer enhancement of Ag-CuO/water hybrid nanofluid in the presence of heat generation, radiation and chemical reaction had been studied by Hayat and Nadeem [2]. Further, Manjunatha et al. [3] analysed the effect of variable viscosity on heat transfer enhancement of hybrid nanofluid. Boundary layer flow and heat transfer over nonlinear stretching/shrinking surface in hybrid nanofluid was investigated by Waini et al. [4]. Most of the researcher concluded that hybrid nanofluid flow plays a more substantial role in the process of heat transfer than a regular nanofluid flow. In addition, the investigation of hybrid nanofluid has been reported by many researchers such as Tayebi and Chamkha [5], Ghadikolaei et al. [6], Ghalambaz et al. [7] and Waini et al. [8].

Much attention has been paid in the past decade to this new type of flow generated by a shrinking sheet due to its applications in industry. Miklavcic and Wang [9] were the first to investigate the viscous flow over a shrinking sheet. They observed that in order to maintain the flow of shrinking sheet, the mass suction is required. The stagnation point flow over a stretching/shrinking sheet in micropolar fluid with magnetohydrodynamic and slip effect has been investigated by Soid et al. [10]. Later, Kamal et al. [11] studied the effect of heat source on the stagnation point flow over stretching/shrinking sheet. While the MHD boundary layer flow over a stretching/shrinking wedge has been examined by Awaludin et al. [12]. On the other hand, the flow of heat transfer over an exponentially shrinking sheet was first studied by Magyari and Keller [13]. Very recently, Anuar et al. [14] investigated an exponentially vertical stretching/shrinking sheet of carbon nanotube with slip and suction effects. These authors have shown that there exists dual solution for the case of shrinking sheet. Afterwards, the study of boundary layer flow over an exponentially stretching/shrinking sheet have been investigated by Rehman and Sheikh [15], Alavi et al. [16], Jusoh et al. [17], Anuar et al. [18] and many more.

The present study is devoted to extend the work of Bachok et al. [19] to the case permeable plate in hybrid nanofluid. The implementation of suction and injection effects in this study can significantly change the flow field and eventually affect the heat transfer rate at the surface [20]. Using suitable similarity variables, ordinary differential equations are formulated and solved numerically using bvp4c package in Matlab software. The effect of governing parameters on heat transfer and fluid flow are illustrated graphically. The results obtained will then be examined to identify the stability of the solutions. We mention here the excellent papers on stability analysis by Merkin [21], Weidman et al. [22], Harris et al. [23], Najib et al. [24], Anuar et al. [25] and Naganthran et al. [26].

Methodology

Mathematical Formulation

Let us consider the steady two-dimensional boundary layer flow of a hybrid nanofluid over an exponentially stretching/shrinking sheet with suction/injection effects. Figure 1 illustrates the flow configuration and coordinate system. Here, x is the coordinate measured along the sheet in vertical direction and y is the coordinate measured in the direction normal to the sheet. The flow is assumed to be generated by stretching/shrinking sheet with velocity $U_w(x) = ae^{x/L}$ and inviscid flow velocity $U_\infty(x) = be^{x/L}$ where a, b are constants and L is a characteristic length of the sheet. It is also assumed that the temperature of the sheet is $T_w(x) = T_\infty + T_o e^{2x/L}$ where T_∞ is the temperature of the ambient fluid and T_o is the constant that measures the rate of temperature increase along the sheet. The velocity of the suction or injection is assumed to be $v_w(x) = -((\nu_f b)/(2L))^{1/2} e^{x/2L} s$ where ν_f is the kinematic viscosity. It should also be mentioned that $s > 0$ is for suction and $s < 0$ is for injection. Hence, the governing equations of momentum and energy equations can be written as [19]

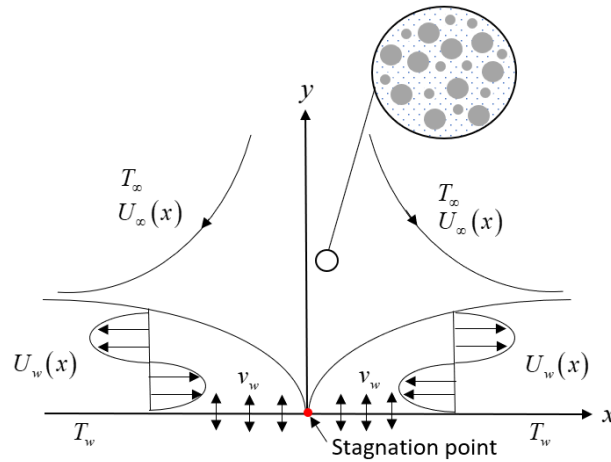


Fig. 1. Physical Model and Coordinate system

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (1)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = U_{\infty} \frac{dU_{\infty}}{dx} + \frac{\mu_{hnf}}{\rho_{hnf}} \frac{\partial^2 u}{\partial y^2} \quad (2)$$

$$u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \alpha_{hnf} \frac{\partial^2 T}{\partial y^2} \quad (3)$$

and the associated boundary conditions are

$$\begin{aligned} u = U_w(x), \quad v = v_w(x), \quad T = T_w(x) \quad \text{at } y = 0 \\ u \rightarrow U_{\infty}(x), \quad T \rightarrow T_{\infty} \quad \text{as } y \rightarrow \infty \end{aligned} \quad (4)$$

Here, μ_{hnf} is the dynamic viscosity of the hybrid nanofluid, ρ_{hnf} is the density of the hybrid nanofluid and α_{hnf} is the effective thermal diffusivity of hybrid nanofluid which are defined as [27]

$$\begin{aligned} \alpha_{hnf} &= \frac{k_{hnf}}{(\rho C_p)_{hnf}}, \mu_{hnf} = \frac{\mu_f}{(1-\varphi_1)^{2.5} (1-\varphi_2)^{2.5}}, \rho_{hnf} = (1-\varphi_2) [(1-\varphi_1)\rho_f + \varphi_1\rho_{s1}] + \varphi_2\rho_{s2}, \\ (\rho C_p)_{hnf} &= (1-\varphi_2) [(1-\varphi_1)(\rho C_p)_f + \varphi_1(\rho C_p)_{s1}] + \varphi_2(\rho C_p)_{s2}, \\ \frac{k_{hnf}}{k_{bf}} &= \frac{k_{s2} + 2k_{bf} - 2\varphi_2(k_{bf} - k_{s2})}{k_{s2} + 2k_{bf} + \varphi_2(k_{bf} - k_{s2})} \quad \text{where} \quad \frac{k_{bf}}{k_f} = \frac{k_{s1} + 2k_f - 2\varphi_1(k_f - k_{s1})}{k_{s1} + 2k_f + \varphi_1(k_f - k_{s1})} \end{aligned} \quad (5)$$

Where k is the thermal conductivity and ρC_p is the effective thermal conductivity where the subscript hnf , f , $s1$ and $s2$ represents hybrid nanofluid, fluid, CuO solid fraction and Ag solid fraction. Furthermore, φ_1 and φ_2 represents CuO and Ag nanoparticle volume fraction. It is worth to mention that hybrid nanofluid are formed by adding the mixture of Ag nanoparticles into a CuO/water nanofluid. The thermophysical properties of nanoparticles are illustrated as in Table 1.

Table 1: Thermophysical properties of nanoparticle (Hayat and Nadeem [2])

Physical Properties	Base fluid (Water)	CuO	Ag
$\rho(\text{kg/m}^3)$	997.1	6320	10500
$c_p(\text{J/kgK})$	4179	531.8	235
$k(\text{W/mK})$	0.613	76.5	429

The dimensionless variables are introduced as:

$$\eta = y \left(\frac{b}{2\nu_f L} \right)^{1/2} e^{x/2L}, \quad \psi = (2\nu_f Lb)^{1/2} f(\eta) e^{x/2L}, \quad \theta(\eta) = \frac{T - T_\infty}{T_w - T_\infty} \quad (6)$$

where η is the similarity variable and the stream function ψ is defined as $u = \partial\psi/\partial y$ and $v = -\partial\psi/\partial x$, hence the equation of continuity (1) is satisfied. By substituting Eq. (6) into governing equation Eqs. (2)-(4), the non-dimensional equations are obtained as follow:

$$\frac{\mu_{hnf}/\mu_f}{\rho_{hnf}/\rho_f} f''' + ff'' - 2f'^2 + 2 = 0 \quad (7)$$

$$\frac{1}{Pr} \frac{k_{hnf}/k_f}{(\rho C_p)_{hnf}/(\rho C_p)_f} \theta'' + f\theta' - f'\theta = 0 \quad (8)$$

the corresponding boundary conditions are:

$$f(0) = s, \quad f'(0) = \varepsilon, \quad \theta(0) = 1, \quad f'(\eta) \rightarrow 1, \quad \theta(\eta) \rightarrow 0 \quad \text{as } \eta \rightarrow \infty \quad (9)$$

Here, prime denote differentiation with respect to η , Pr is the Prandtl number and $\varepsilon = a/b$ is the shrinking parameter with $\varepsilon < 0$.

The physical quantities of interest are skin friction coefficient and local Nusselt number which are defined as

$$C_f = \frac{\tau_w}{\rho_f U_\infty^2}, \quad Nu_x = \frac{2Lq_w}{k_f (T_w - T_\infty)} \quad (10)$$

where the surface shear stress τ_w and the heat flux q_w are given by

$$\tau_w = \mu_{hnf} \left(\frac{\partial u}{\partial y} \right)_{y=0}, \quad q_w = -k_{hnf} \left(\frac{\partial T}{\partial y} \right)_{y=0} \quad (11)$$

using the similarity variables (6), we obtain:

$$\text{Re}_x^{1/2} C_f = \frac{\mu_{hnf}}{\mu_f} f''(0), \quad \text{Re}_x^{-1/2} Nu_x = -\frac{k_{hnf}}{k_f} \theta'(0) \quad (12)$$

where $\text{Re}_x = U_\infty x / \nu_f$ is the local Reynold number.

Flow Stability

We now investigate the stability of solutions by first considering the time-dependent problem (see Merkin [21]). Therefore, Eq. (1) holds, while Eqs. (2) and (3) are replaced by:

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = U_\infty \frac{dU_\infty}{dx} + \frac{\mu_{hnf}}{\rho_{hnf}} \frac{\partial^2 u}{\partial y^2} \quad (13)$$

$$\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \alpha_{hnf} \frac{\partial^2 T}{\partial y^2} \quad (14)$$

with the corresponding boundary conditions:

$$\begin{aligned} u = U_w(x), \quad v = v_w(x), \quad T = T_w(x) \quad \text{at } y = 0 \\ u \rightarrow U_\infty(x), \quad T \rightarrow T_\infty \quad \text{as } y \rightarrow \infty \end{aligned} \quad (15)$$

where t denotes the time. The new variable τ and new non-dimensionless variables are introduced:

$$\eta = y \left(\frac{b}{2\nu_f L} \right)^{1/2} e^{x/2L}, \quad \psi = (2\nu_f L b)^{1/2} f(\eta, \tau) e^{x/2L}, \quad \theta(\eta, \tau) = \frac{T - T_\infty}{T_w - T_\infty}, \quad \tau = \frac{bt}{2L} e^{x/2L} \quad (16)$$

using the similarity transformation (16), the time-dependent governing equations (13) and (14) are transformed as follows:

$$\frac{\mu_{hnf}/\mu_f}{\rho_{hnf}/\rho_f} \frac{\partial^3 f}{\partial \eta^3} - 2 \left(\frac{\partial f}{\partial \eta} \right)^2 + 2 + f \frac{\partial^2 f}{\partial \eta^2} - 2\tau \left(\frac{\partial f}{\partial \eta} \frac{\partial^2 f}{\partial \eta \partial \tau} - \frac{\partial f}{\partial \tau} \frac{\partial^2 f}{\partial \eta^2} \right) - \frac{\partial^2 f}{\partial \eta \partial \tau} = 0 \quad (17)$$

$$\frac{1}{\text{Pr}(\rho C_p)_{hnf}} \frac{k_{hnf}/k_f}{(\rho C_p)_f} \frac{\partial^2 \theta}{\partial \eta^2} + f \frac{\partial \theta}{\partial \eta} - \theta \frac{\partial f}{\partial \eta} - 2\tau \left(\frac{\partial f}{\partial \eta} \frac{\partial \theta}{\partial \tau} - \frac{\partial f}{\partial \tau} \frac{\partial \theta}{\partial \eta} \right) - \frac{\partial \theta}{\partial \tau} = 0 \quad (18)$$

subject to boundary conditions

$$\begin{aligned} f(0, \tau) = s, \quad \frac{\partial f}{\partial \eta}(0, \tau) = \varepsilon, \quad \theta(0, \tau) = 1, \\ \frac{\partial f}{\partial \eta}(\eta, \tau) \rightarrow 1, \quad \theta(\eta, \tau) \rightarrow 0 \quad \text{as } \eta \rightarrow \infty \end{aligned} \quad (19)$$

The following term [22] are introduced:

$$f(\eta, \tau) = f_o(\eta) + e^{-\gamma\tau} F(\eta, \tau), \quad \theta(\eta, \tau) = \theta_o(\eta) + e^{-\gamma\tau} G(\eta, \tau) \quad (20)$$

Therefore, the stability solution $f(\eta) = f_o(\eta)$ and $\theta(\eta) = \theta_o(\eta)$ fulfilling the boundary-value problem can be identified where $F(\eta, \tau)$ and $G(\eta, \tau)$ are small relative to $f_o(\eta)$ and $\theta_o(\eta)$, while γ is an unknown eigenvalue.

Upon substituting (20) into Eqs. (17) and (18) along with the boundary conditions (9), the new linearized problems are obtained. Setting $\tau = 0$ as suggested by Weidman et al. [22], eventually $F(\eta) = F_o(\eta)$ and $G(\eta) = G_o(\eta)$. Hence, we obtained the final equations as in the following form:

$$\frac{\mu_{hnf}/\mu_f}{\rho_{hnf}/\rho_f} F_o''' + f_o F_o'' + f_o'' F_o - (4f_o' - \gamma) F_o' = 0 \quad (21)$$

$$\frac{1}{\text{Pr}(\rho C_p)_{hnf}} \frac{k_{hnf}/k_f}{(\rho C_p)_f} G_o'' + f_o G_o' + F_o \theta_o' - F_o' \theta_o - G_o f_o' + \gamma G_o = 0, \quad (22)$$

the boundary conditions are now reduced to:

$$\begin{aligned} F_o(0) = 0, \quad F_o'(0) = 0, \quad G_o(0) = 0, \\ F_o'(\eta) \rightarrow 0, \quad G_o(\eta) \rightarrow 0, \quad \text{as } \eta \rightarrow \infty \end{aligned} \quad (23)$$

The smallest eigenvalues γ can be determined by relaxing the boundary condition on $F_o(\eta)$ as suggested by Harris et al. [23]. For the present research, the boundary conditions $F_o'(\eta) \rightarrow 0$ as $\eta \rightarrow \infty$ are relaxed and replaced by new boundary condition $F_o''(0) = 1$.

Results

The bvp4c solver in Matlab software is used to solve the system of ordinary differential equations (7) and (8) with the boundary condition (9). In order to validate the numerical results and the accuracy of applied technique in the present problem, the comparison values of skin friction coefficient $f''(0)$ and heat transfer $-\theta'(0)$ for viscous fluid and $s = 0$ are shown in Table 2 and 3. It is clear that the obtained outcomes are in good agreement with the previously reported results of Bachok et al. [19] and Subhashini et al. [28]. The values of $f''(0)$ and $-\theta'(0)$ for hybrid nanofluid ($\phi_1 = \phi_2 = 0.1$) are also included in these tables for future reference. Following Oztop and Abu-Nada [29], the values of nanoparticle volume fractions are varying from 0 to 0.2 ($0 < \phi < 0.2$) and Prandtl number is taken as 6.2 throughout the whole research. The effects of suction/injection parameter, nanoparticle volume fraction parameter and stretching/shrinking parameter on the dimensionless velocity, temperature, skin friction coefficient and local Nusselt number are demonstrated graphically and discussed in details. This research confirms the existence of non-unique solutions for certain range of shrinking sheet ($\varepsilon \leq -1$) and unique solution when $-1 < \varepsilon \leq 0$. However, there are no solution when $\varepsilon < \varepsilon_c$, where ε_c is a critical value of ε which depends on other parameters.

The effect of suction/injection parameter s on skin friction $f''(0)$ and velocity profile $f'(\eta)$ are shown in Figures 2 and 3. It is observed that in the case of hybrid nanofluid ($\phi_1 = \phi_2 = 0.1$), the presence of suction ($s = 0.2$) could increase the flow resistance, while injection ($s = -0.2$) decreases the flow resistance for the first solution. Additionally, we observed the contradictory behaviour for the second solution. Figures 4 and 5 present the performance of heat transfer $-\theta'(0)$ and temperature profile $\theta(\eta)$ for various values of suction/injection parameter s . Injection is discussed to inspect the rise in temperature, while suction depicts the reverse trends to cool the system for the first solution and vice versa for the second solution. It is further observed that when suction takes place in the boundary layer, the solution existence range is bound to increase and hence postponing the boundary layer separation. Also, it is seen that these profiles satisfied asymptotically the far field boundary conditions equation (9).

Figures 6 and 7 reflect the influence of Ag nanoparticle volume fraction ϕ_2 on skin friction and heat transfer in the presence of suction ($s = 0.2$). It is analysed that the skin friction and heat transfer increase by increasing values of Ag nanoparticle volume fraction. However, Figure 7 displayed the reverse trend when $\varepsilon > -0.8$. Moreover, it has been observed that addition of nanoparticle volume fraction in nanofluid for suction case causes the postponing of boundary layer separation. The effect of Ag nanoparticle volume fraction in the presence of injection ($s = -0.2$) on skin friction and heat transfer are shown in Figures 8 and 9. From these

figures, it is found that the skin friction and heat transfer increase with the increasing values of Ag nanoparticle volume fraction. It is noticed that for the case of injection, addition of Ag nanoparticle volume fraction in nanofluid tends to accelerate the boundary layer separation. It is important to note that hybrid nanofluid offers higher skin friction and heat transfer compared to nanofluid for both suction and injection case.

The effects of suction/injection parameter s and nanoparticle volume fraction are depicted in Figures 10 and 11 for skin friction coefficient $Re_x^{1/2} C_f$ and local Nusselt number $Re_x^{-1/2} Nu_x$. It is witnessed that for increasing values of Ag nanoparticle volume fraction ϕ_2 and suction/injection parameter s , the skin friction coefficient and local Nusselt number increases. Also, the values of these quantities increase almost linearly with increasing values of CuO nanoparticle volume fraction ϕ_2 . Figures 12 and 13 represent the effect of shrinking sheet on velocity $f'(\eta)$ and temperature profile $\theta(\eta)$ for the case of suction and hybrid nanofluid. It is found that with an increasing magnitude of shrinking sheet, the velocity profile decreases for the first solution and increases for the second solution. Meanwhile, temperature profile increase for the first solution and decrease for the second solution.

Since the obtained results is non-unique solutions, we are intended to do a stability analysis in order to identify which solution is stable and physically realizable. The system of linearized equations (21) and (22) along with the boundary condition (23) are solved using the bvp4c solver in Matlab software to obtain the values of smallest eigenvalues γ . Table 4 illustrates the smallest eigenvalues for selected value of shrinking sheet, suction/injection parameter and Ag nanoparticle volume fraction when CuO nanoparticle volume fraction fixed to 0.1. These values are approaching to zeros as the value of shrinking parameter close to its critical value. Moreover, we can observe that the first solution presents the positive values while the second solution shows the negative values. The positive value indicates that there exists only slight disturbance in the flow which does not interrupt the boundary layer separation and vice versa for the second solution. Hence, we can conclude that the first solution is stable meanwhile the second solution is unstable.

Table 2: Values of $f''(0)$ for some values of ε when $s = 0$ and $Pr = 6.2$

ϕ_1	ϕ_2	ε	Bachok et al. [19]	Subhashini et al. [28]	Present results
0	0	-0.5	2.1182	2.1176	2.1182
		0	1.6872	1.6863	1.6872
0.1	0.1	-0.5			2.5391
		0			2.0225

Table 3: Values of $-\theta'(0)$ for some values of ε when $s = 0$ and $Pr = 6.2$

ϕ_1	ϕ_2	ε	Bachok et al. [19]	Present results
0	0	-0.5	0.6870	0.6870
		0	1.7148	1.7148
0.1	0.1	-0.5		0.8603
		0		1.4585

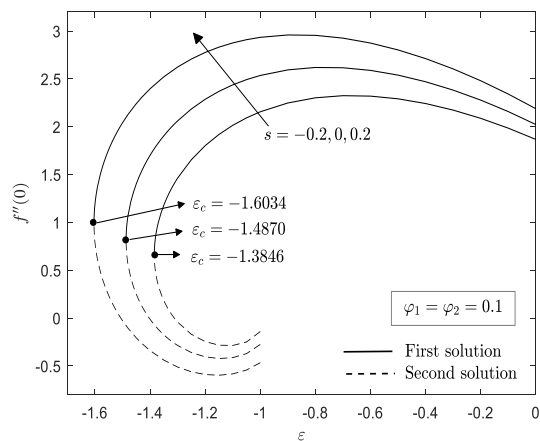


Fig. 2. Skin friction $f''(0)$ for different s

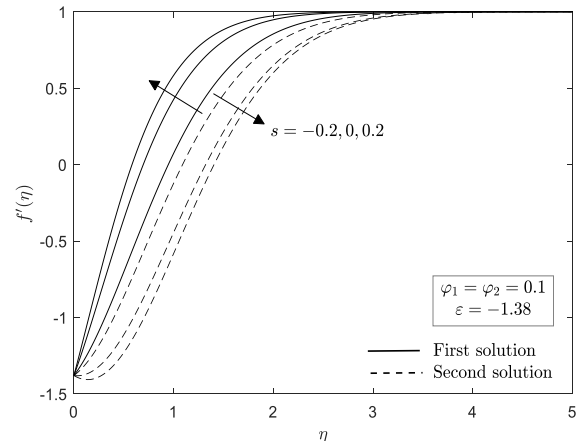


Fig. 3. Velocity profile $f'(\eta)$ for different s

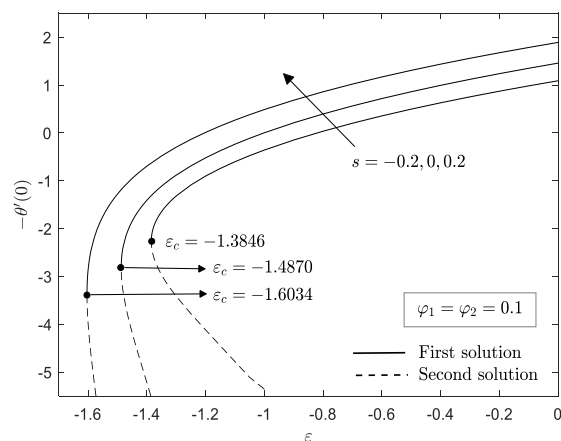


Fig. 4. Heat transfer $-\theta'(0)$ for different s

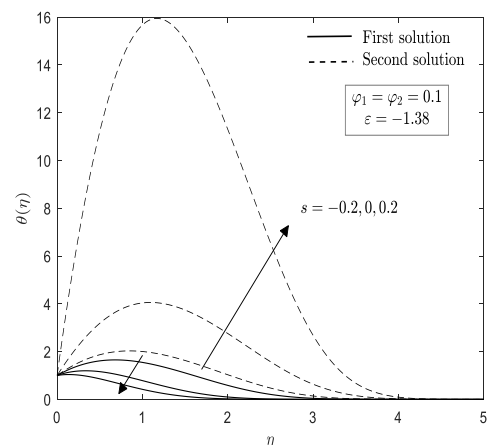


Fig. 5. Temperature profile $\theta(\eta)$ for different s

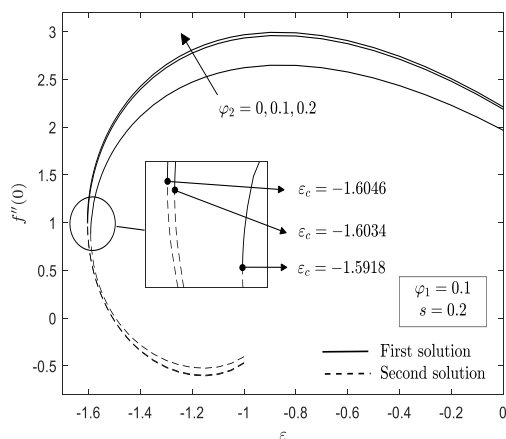


Fig. 6. Skin friction $f''(0)$ for different values of Ag nanoparticle volume fraction ϕ_2 when $s = 0.2$

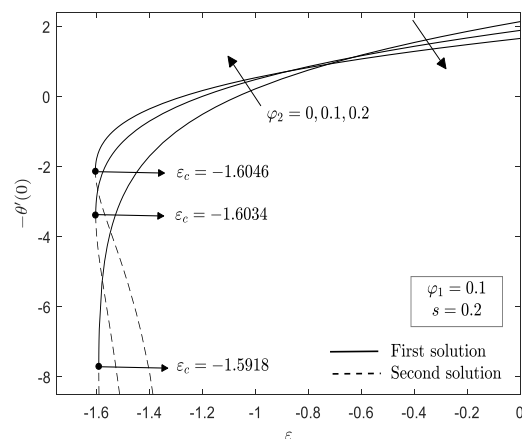


Fig. 7. Heat transfer $-\theta'(0)$ for different values of Ag nanoparticle volume fraction ϕ_2 when $s = 0.2$

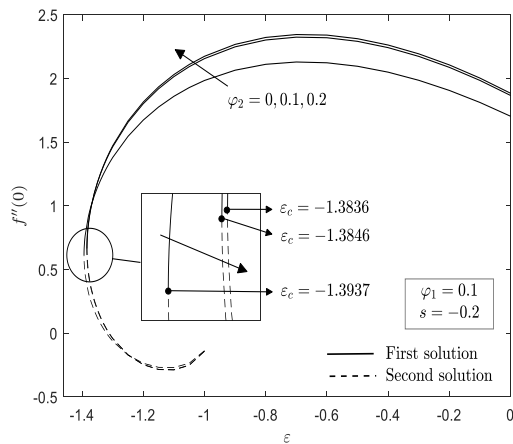


Fig. 8. Skin friction $f''(0)$ for different values of Ag nanoparticle volume fraction φ_2 when $s = -0.2$

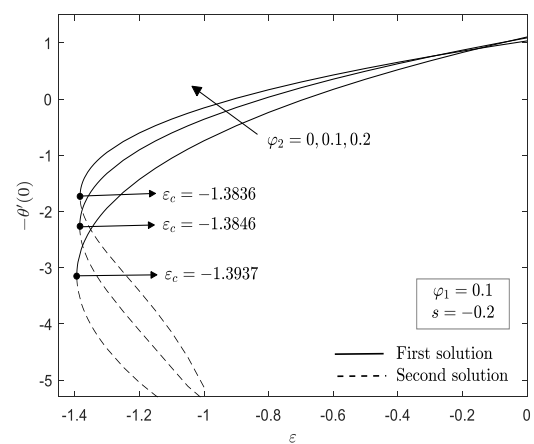


Fig. 9. Heat transfer $-\theta'(0)$ for different values of Ag nanoparticle volume fraction φ_2 when $s = -0.2$

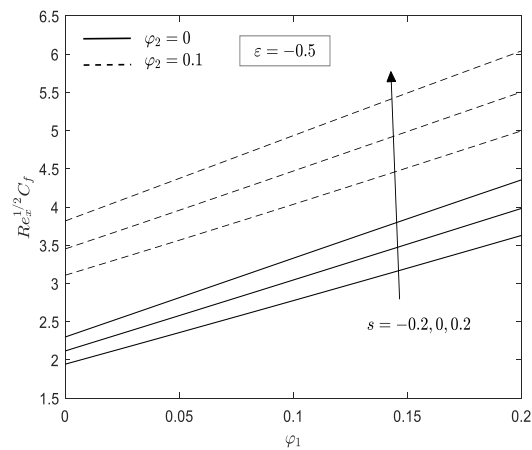


Fig. 10. Skin friction coefficient $Re_x^{1/2} C_f$ for different values of s

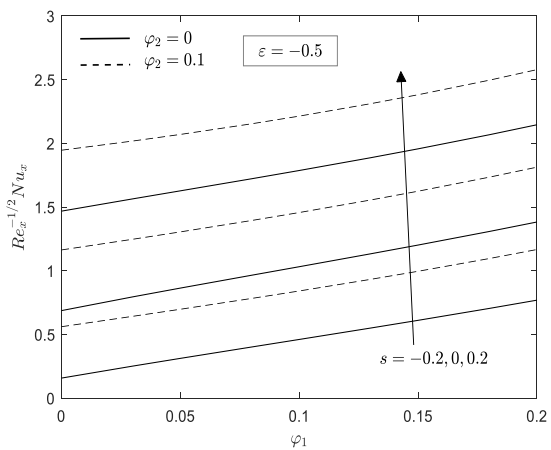


Fig. 11. Nusselt number $Re_x^{-1/2} Nu_x$ for different values of s

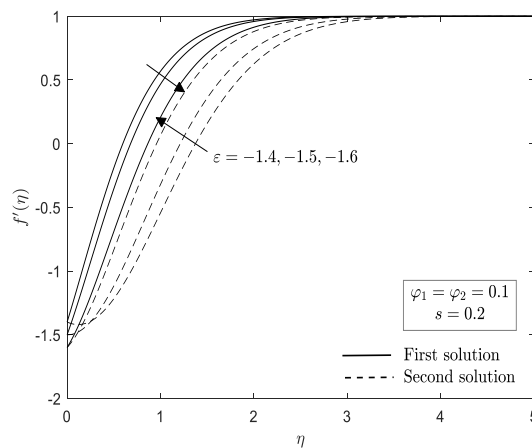


Fig. 12. Velocity profile $f'(\eta)$ for different values of shrinking parameter ε

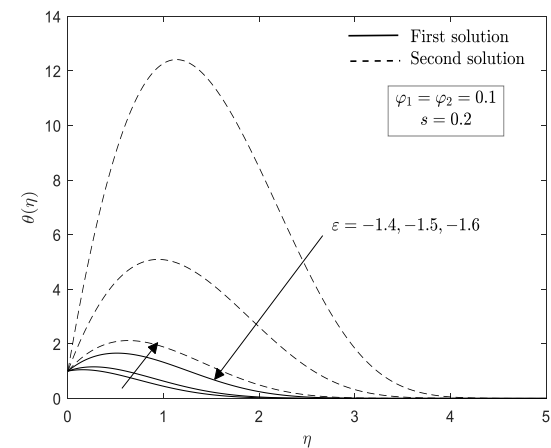


Fig. 13. Temperature profile $\theta(\eta)$ for different values of shrinking parameter ε

Table 4: Smallest eigenvalues γ for selected values of s , ϕ_2 and ε when $\phi_1 = 0.1$

s	ϕ_2	ε	First solution	Second solution
-0.2	0	-	0.05739	-0.05733
		1.3936		
		-1.393	0.13530	-0.13496
		-1.39	0.30584	-0.30414
	0.1	-	0.06184	-0.06177
		1.3845		
		-1.384	0.12784	-0.12755
		-1.38	0.34170	-0.33958
0.2	0	-	0.06220	-0.06212
		1.5917		
		-1.591	0.14597	-0.14556
		-1.59	0.21508	-0.21418
	0.1	-	0.05977	-0.05971
		1.6033		
		-1.603	0.10509	-0.10488
		-1.6	0.29319	-0.29152

Conclusions

In the present study, the effect of suction/injection on the stagnation point flow over an exponentially shrinking sheet of hybrid nanofluid was studied numerically. From the numerical results obtained, some important conclusions are summarized:

- The non-unique solutions exist for the certain range of shrinking parameter ($\varepsilon \leq -1$) and unique solutions when $\varepsilon > -1$.
- The suction parameter has high impact on skin friction and heat transfer compared to injection parameter.
- The existence of suction parameter expands the range of solutions to exists and consequently delays the boundary layer separation.
- The increasing value of Ag nanoparticle volume fraction on CuO/water nanofluid causes the increases in skin friction and heat transfer.
- For the case of suction, the range of solutions to exist increase with increasing value of Ag nanoparticle volume fraction and contradictory observation are made for the case of injection.
- Stability analysis ratifies the first solution as a stable solution and the second solution as an unstable solution.

Acknowledgement

This research was supported by fundamental research grant scheme (FRGS/1/2018/STG06/UPM/02/4/5540155) and MyBrainsc from the Ministry of Higher Education Malaysia.

References

- [1] Momin, G. G. "Experimental investigation of mixed convection with water-Al₂O₃ & hybrid nanofluid in inclined tube for laminar flow." *International Journal of Scientific & Technology Research* 2 (2013): 195-202.
- [2] Hayat, T. and Nadeem, S. "Heat transfer enhancement with Ag–CuO/water hybrid nanofluid." *Results in physics* 7 (2017): 2317-2324.
- [3] Manjunatha, S., Kuttan, A. B., Jayanthi, S., Chamkha A. and Giressha B. J. "Heat transfer enhancement in the boundary layer flow of hybrid nanofluids due to variable viscosity and natural convection." *Heliyon* 5, no. 4 (2019): e01469.
- [4] Waini, I., Ishak, A, and Pop, I. "Hybrid nanofluid flow and heat transfer over a nonlinear permeable stretching/shrinking surface." *International Journal of Numerical Methods for Heat & Fluid Flow* (2019).
- [5] Tayebi, T. and Chamkha, A. J. "Buoyancy-driven heat transfer enhancement in a sinusoidally heated enclosure utilizing hybrid nanofluid." *Computational Thermal Sciences: An International Journal* 9, no. 5 (2017).
- [6] Ghadikolaie, S. S., Yassari, M., Sadeghi, H., Hosseinzadeh, K. and Ganji, D. D. "Investigation on thermophysical properties of TiO₂–Cu/H₂O hybrid nanofluid transport dependent on shape factor in MHD stagnation point flow." *Powder technology* 322 (2017): 428-438.
- [7] Ghalambaz, M., Sheremet, M. A., Mehryan, S. A. M., Kashkooli, F. M. and Pop, I. "Local thermal non-equilibrium analysis of conjugate free convection within a porous enclosure occupied with Ag–MgO hybrid nanofluid." *Journal of Thermal Analysis and Calorimetry* 135, no. 2 (2019): 1381-1398.
- [8] Waini, I., Ishak, A. and Pop I. "Unsteady flow and heat transfer past a stretching/shrinking sheet in a hybrid nanofluid." *International Journal of Heat and Mass Transfer* 136 (2019): 288-297.
- [9] Miklavčič, M. and Wang. C. "Viscous flow due to a shrinking sheet." *Quarterly of Applied Mathematics* 64, no. 2 (2006): 283-290.
- [10] Soid, S. K., Ishak, A. and Pop, I. "MHD stagnation-point flow over a stretching/shrinking sheet in a micropolar fluid with a slip boundary." *Sains Malaysiana* 47, no. 11 (2018): 2907-2916.
- [11] Kamal, F., Zaimi, K., Ishak, A. and Pop I. "Stability analysis on the stagnation-point flow and heat transfer over a permeable stretching/shrinking sheet with heat source effect." *International Journal of Numerical Methods for Heat & Fluid Flow* 28, no. 11 (2018): 2650-2663.
- [12] Awaludin, I. S., Ishak, A. and Pop, I. "On the stability of MHD boundary layer flow over a stretching/shrinking wedge." *Scientific reports* 8, no. 1 (2018): 13622.
- [13] Magyari, E. and Keller, B. "Heat and mass transfer in the boundary layers on an exponentially stretching continuous surface." *Journal of Physics D: Applied Physics* 32, no. 5 (1999): 577.
- [14] Anuar, N.S., Bachok, N., Arifin, N.M., Rosali, H. "Mixed Convection Flow and Heat Transfer of Carbon Nanotubes Over an Exponentially Stretching/Shrinking Sheet with Suction and Slip Effect." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 59, no. 2 (2019): 232-242.
- [15] Rehman, A. and Sheikh, N. "Boundary Layer Stagnation-Point Flow of Micropolar Fluid over an Exponentially Stretching Sheet." *International Journal of Fluid Mechanics & Thermal Sciences* 3, no. 3 (2017): 25-31.
- [16] Alavi, S. Q., Hussanan, A., Kasim, A. R. M., Rosli, N. and Salleh, M. Z. "MHD Stagnation Point flow Towards an Exponentially Stretching Sheet with Prescribed wall

- Temperature and Heat Flux." *International Journal of Applied and Computational Mathematics* 3, no. 4 (2017): 3511-3523.
- [17] Jusoh, R., Nazar, R. and Pop, I. "Magnetohydrodynamic rotating flow and heat transfer of ferrofluid due to an exponentially permeable stretching/shrinking sheet." *Journal of Magnetism and Magnetic Materials* 465 (2018): 365-374.
 - [18] Anuar, N. S., Bachok, N., Arifin, N. M. and Rosali, H. "Stagnation Point Flow and Heat Transfer over an Exponentially Stretching/Shrinking Sheet in CNT with Homogeneous–Heterogeneous Reaction: Stability Analysis." *Symmetry* 11, no. 4 (2019): 522.
 - [19] Bachok, N., Ishak, A. and Pop, I. "Boundary layer stagnation-point flow and heat transfer over an exponentially stretching/shrinking sheet in a nanofluid." *International Journal of Heat and Mass Transfer* 55, no. 25-26 (2012): 8122-8128.
 - [20] Al-Sanea, S. A. "Mixed convection heat transfer along a continuously moving heated vertical plate with suction or injection." *International Journal of Heat and Mass Transfer* 47, no. 6-7 (2004): 1445-1465.
 - [21] Merkin, J. H. "On dual solutions occurring in mixed convection in a porous medium." *Journal of engineering Mathematics* 20, no. 2 (1986): 171-179.
 - [22] Weidman, P. D., Kubitschek, D. G. and Davis, A. M. J. "The effect of transpiration on self-similar boundary layer flow over moving surfaces." *International journal of engineering science* 44, no. 11-12 (2006): 730-737.
 - [23] Harris, S. D., Ingham, D. B. and Pop, I. "Mixed convection boundary-layer flow near the stagnation point on a vertical surface in a porous medium: Brinkman model with slip." *Transport in Porous Media* 77, no. 2 (2009): 267-285.
 - [24] Najib, N., Bachok, N., Arifin, N. M. and Ali, F. M. "Stability analysis of stagnation point flow in nanofluid over stretching/shrinking sheet with slip effect using buongiorno's model." *Numerical Algebra, Control & Optimization* (2019): 151-158.
 - [25] Anuar, N. S., Bachok, N. and Pop, I. "A stability analysis of solutions in boundary layer flow and heat transfer of carbon nanotubes over a moving plate with slip effect." *Energies* 11, no. 12 (2018): 3243.
 - [26] Naganthran, K., Nazar, R. and Pop, I. "Effects of thermal radiation on mixed convection flow over a permeable vertical shrinking flat plate in an Oldroyd-B fluid." *Sains Malaysiana* 47, no. 5 (2018): 1069-1076.
 - [27] Devi, S. S. U. and Devi, S. P. A. "Numerical investigation of three-dimensional hybrid Cu–Al₂O₃/water nanofluid flow over a stretching sheet with effecting Lorentz force subject to Newtonian heating." *Canadian Journal of Physics* 94, no. 5 (2016): 490-496.
 - [28] Subhashini, S. V., Sumathi, R. and Momoniat, E. "Dual solutions of a mixed convection flow near the stagnation point region over an exponentially stretching/shrinking sheet in nanofluids." *Meccanica* 49, no. 10 (2014): 2467-2478.
 - [29] Oztop, H. F. and Abu-Nada, E. "Numerical study of natural convection in partially heated rectangular enclosures filled with nanofluids." *International journal of heat and fluid flow* 29, no. 5 (2008): 1326-1336.

CHARACTERIZATION OF LIQUID OIL PRODUCT FROM HIGH DENSITY POLYETHYLENE WITH PYROLYSIS USING L-SHAPE HEAT PIPE HEAT EXCHANGER AS LIQUID COLLECTION SYSTEM

Imansyah Ibnu Hakim¹

Nandy Putra^{1*}

Robbih Rizky Yudianto¹

Nasrudin Abdullah²

¹Heat Transfer Laboratory, Department of Mechanical Engineering Universitas Indonesia, Kampus UI, Depok 16424, Indonesia

²Department of Mechanical Engineering University of Samudra, Aceh Indonesia

*(Corresponding Author's Email: ali@gmail.com)

Abstract: *The increasing world population causes an increase in plastic waste generation which can endanger the environment. Pyrolysis can be used to overcome this problem by converting plastic waste into liquid fuel which can be used for transportation and electric generator. In this study, HDPE (High Density Polyethylene) was used because this type of plastic is one of the most produced. In addition, in several studies it was known that HDPE plastic can produce liquid oil up to 80% of the total mass. In this study an LCS (Liquid Collection System) was used by integrating HPHE (Heat Pipe Heat Exchanger). A total of 29 HPHE in the LCS are used because of its nature which can passively reduce heat, thus saving cooling costs. In this experiment, 50 grams of pellet-shaped HDPE with a size of 3-5 mm was heated in a fixed-bed reactor with a temperature of 450-600 °C for 3 hours with an electric power of 3000 Watts. In this experiment it was found that the reactor temperature to produce the most liquid oil was 550 °C with the amount produced at 70.2 %wt. The liquid oil that was produce has a characteristic of kinematic viscosity at 40 °C is 2.045 c St, density at 0.83 gram/ml, heating value at 0.185 MJ/kg, Cetane Number at 32.6, RON at 40.5.*

Keywords: *liquid oil; high density polyethylene; heat pipe heat exchanger; pyrolysis*

Introduction

The uses of plastic in daily application has been growing in the last few decades due to its practicality, durability and its cost[1]. Currently the world production of plastic has risen exponentially which also causes the increase amount of plastic waste that may harm the environment and marine ecosystems[2] because thermoplastics may not be bio-degraded for a long time[3], this may be correlated with the rapidly increasing population growth. Indonesia as one of the most populated country in the world has produced 65.2 million tons of plastic waste in 2016[4]. An effective method that could process plastic waste are needed to balance its production. A conventional mechanical recycling techniques can only recycle 15-20% of all plastic waste types[5] and cost inefficient[6]. The uses of uncontrolled incineration may cause an air and waterborne pollution[7][8]. Pyrolysis however is a promising waste-to-energy method for sustainable plastic waste management along with production of liquid oil as a source of energy [9][10] and the char and gases can be sold or furtherly process to add value[11]. Also due to the absent of O₂ in the process, pyrolysis may reduce gaseous pollutants[12][8]. This experiment will use HDPE to be pyrolyzed because it is one of the commonly used plastics[13][14] and has a high liquid oil yield that may reach 80.88wt%[15].

An L shape heat pipe will be used because of its ability to passively remove heat from a system[16], meaning there will be no additional energy needed for cooling in order to condense the vapor produced from the reactor. A previous study [17] has carried out an experiment of pyrolysis using merbau wood while applying heat pipe as heat exchanger and successfully condensed the vapor produced with a liquid yield of 42.5wt%.

Materials and methods

The materials that will be used in this experiment are virgin HDPE in the form of pellet with the size of 3-5 mm as shown in figure 1 below. A virgin HDPE pellets were chosen in order to avoid contaminant which could influence the characterization of the product that is produced in this experiment such as colouring, coating, etc.



Fig 1 HDPE Pellet

Then this HDPE will be thermally pyrolyzed in a non-catalytic fixed bed non-sweep gas reactor because of its widely use in the process[18][19][20][21]. The reactor will be heated using an electric heater to raise the temperature of the reactor to 450-600 °C with a heat rate ± 30 °C/min. The experimental set-up can be seen in figure 2 below. This experimental set up consists of a reactor which will be used to contain the HDPE during pyrolysis that was made from stainless steel 316 L Sch80 with a height of 25 cm and an inner diameter of 76.66 cm, with maximum volume capacity of 1064,82 cm³. An elbow that is made from stainless steel 316 L Sch80 will be used to connect the reactor with the LCS. A LCS is a SS304 pipes, Sch10, with three-inch diameter, and 660 mm length equipped with two JIS 10K flanges of three-inch diameter. This pipe has been modified to be able to insert 32 HPHE so this LCS can be used to cool the vapor produced by the reactor and turn it into a liquid. According to [22], the number of effective HPHE needed to cool was only 18 thus the number of HPHE has been reduced to its effective number. The HPHE that is used in this experiment are L-Shape Finned HPHE as shown in figure 3. This HPHE are equipped with 52 fin with the size of 21 × 16.5 × 1 mm and an inter spacing of 1.52 mm. This shape was chosen because of its simplicity to be inserted and pull out for cleaning purposes. Around the hole that is used to insert the HPHE are covered with silicon to prevent the vapor produced from leaking. Then, there is also a reducer equipped with JIS 10K flanges that connects to the LCS. This reducer will separate the non-condensable gas that is produced during the pyrolysis. At the end of the reducer, a 500 ml of pyrex glass is installed to act as liquid oil container which will contain the liquid oil that is produced during the experiment.

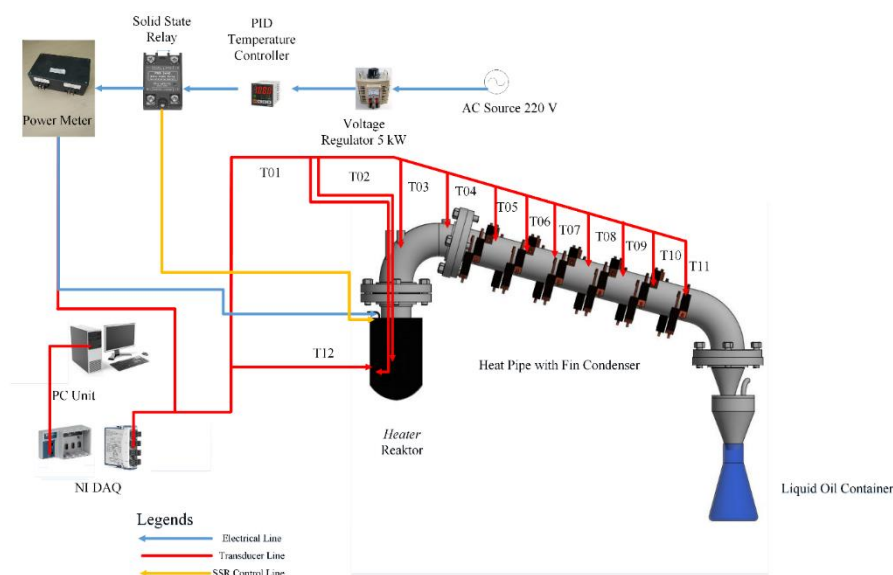


Fig 2 Experimental Set Up

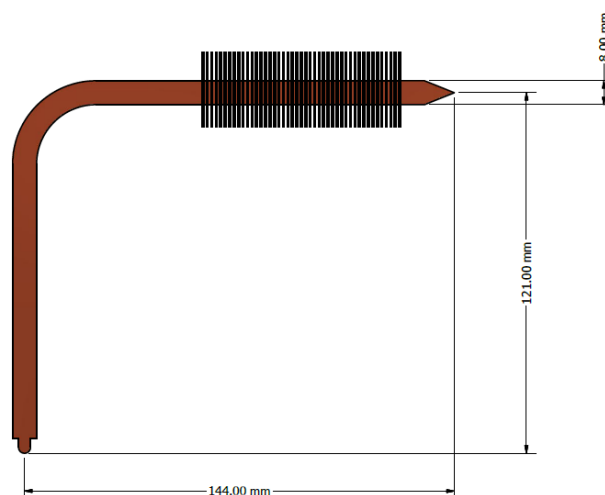


Fig. 3 L-Shape Finned Heat Pipe Heat Exchanger

Experimental

A 50 gram of virgin HDPE pellet with the size of 3-5 mm will be pyrolyzed in a non-catalytic fixed bed non sweep gas reactor. In this experiment the temperature that will be used to heat the HDPE are 450° to 600° C with the incremental temperature of 50° C. In order to make sure all of the material inside the reactor are thermally decomposed, this pyrolysis will be going on for 3 hours. The temperature inside the reactor will be controlled using a PID temperature controller. In order to control the temperature heating rate, a voltage regulator is used and set at 3000 Watt. During the experiment, a type K thermocouple that is connected to NI-DAQs 912 are used to record the temperature change that happens. These thermocouples are placed according to fig 2. The main purpose to use this thermocouple was to observe how well the HPHE can perform to cool the vapor that is produced. After 3 hours the liquid contained in the LOC will be weighed using a weighing scale. Then the remaining residue inside the reactor are collected as well and a mass balance will be calculated using equation 1. and a percentage of its product will be calculated using equation 2.

$$M_{\text{total}} = M_{\text{gas}} + M_{\text{Liquid}} + M_{\text{solid}} \quad 1)$$

$$\%Product = \frac{M_{\text{product}}}{M_{\text{total}}} \times 100\% \quad 2)$$

The aims to characterize the liquid oil product are to determine the similarity to current fuel characteristics and its quality. The characterization shall consist of densities, kinematic viscosities, GC-MS, heat value, RON, and cetane number to determine its quality.

Result and Discussion

Product

Using a non-catalytic fixed bed non-sweep gas reactor, pyrolysis experiment was performed using HDPE as its main material and HPHE to passively cool the vapor from the reactor. The main products that have been produced in this experiment are ash, liquid oil, and a non-condensable gas which can be seen in figure 4. The HPHE located in the LCS has successfully condensed the vapor produced in the reactor and turned it into liquid oil substance, while the residue from the thermal decomposition are ashes. From table 1 we can see the weight fraction of each product from this experiment, and 550 °C has the highest weight fraction of liquid product at 70.2wt%. Also, it appears that the higher the temperature used in this experiment will result in more production of gasses, this phenomena occurs due to the vapor that has been produce in the reactor are immediately gasified in the reactor because of its high temperature, thus, resulting in more non-condensable gas production. In this experiment, however, at 450 °C, the reactor temperature did not reach the thermal cracking point of HDPE and thus resulting as wax and non-condensable gasses as its main product. Unfortunately, the wax produced in this experiment was spread out inside the entire experimental set up leading to calculating the mass to be impossible.

Table 1 Mass Fraction of The Product

Reactor Temperature (°C)	Product Liquid (wt%)	Solid (wt%)	Residue	Gas(wt%)	Heat Rate (°C/min)
450		Wax			30,25
500	61,1	1,29		37,61	30,45
550	70,2	1,75		28,05	30,22
600	24,9	0,46		74,64	30,13



a) Ash



b) Non-Condensable Gas



c) Liquid Oil

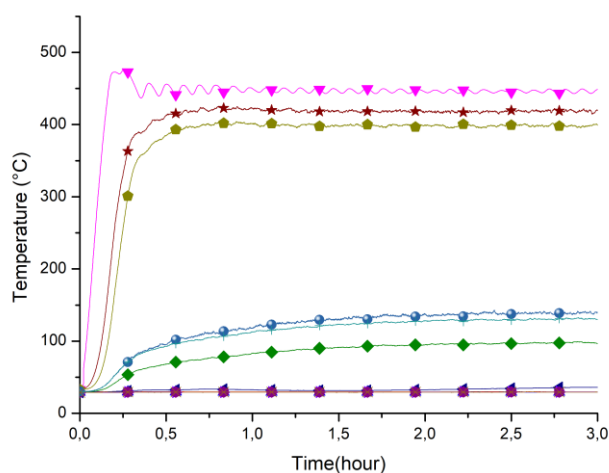
Fig. 4 Main Product from Pyrolysis

Temperature Profile

Using a reactor temperature at 450-600°C with heater power at 3000 Watt, a 50 gram of HDPE was pyrolyzed in a fixed bed non-sweep gas reactor. The temperature profile of this experiment was observed using a type K a thermocouple. The recorded temperature can be seen in figure 5. It can be seen that in figure 5 a) the temperature profile for T01 and T02 that did not produce any liquid oil, differs from the temperature profile of b),c),d).

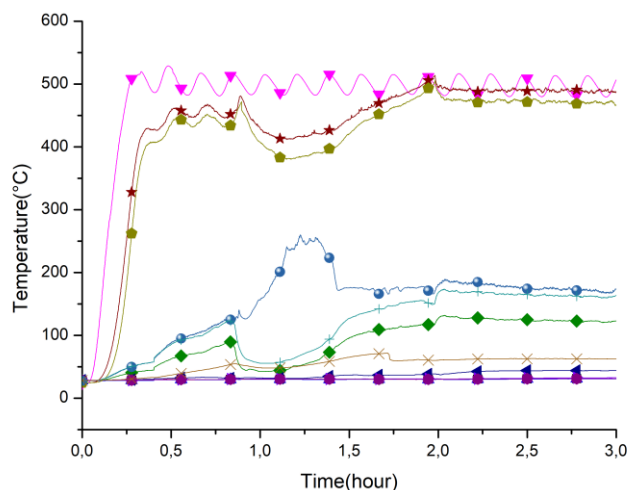
—★— T01 —●— T02 —●— T03 —+— T04 —◆— T05 —▲— T06
—▼— T07 —●— T08 —■— T09 —●— T10 —▲— T11 —▼— T12

3000 Watt 450 °C

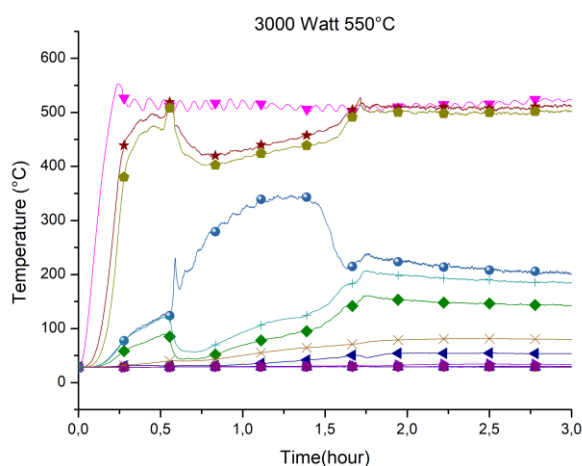


a) Experiment at 450 °C

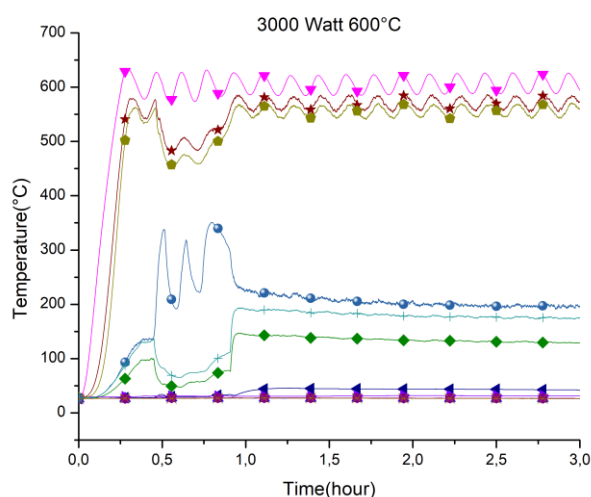
3000 Watt 500 °C



b) Experiment at 500 °C



c) Experiment at 550 °C



d) Experiment at 600 °C

Fig. 5 Temperature Profile from Experiment

Product Characterization

The densities and kinematic viscosity of the liquid oil produced is then characterized because it is a key parameter to determine its similarity to a fuel[23], also a GC-MS analysis is done to observe the compound contained in the liquid oil in table 2. Then, to evaluate its quality the heat value, RON and Cetane Number are also characterized. Then, these characteristics are compared to Indonesian fuel as in table 2 below.

Table 2 Characteristic of the Liquid Oil

Characteristic	Liquid Oil HDPE	Pertamina DEX[24]	Pertamax[25]
Density (g/ml)	0.83	0.82-0.86	0.715
Kinematic Viscosity (c St 40 °C)	2.045	2-4,5	-
Heat Value (MJ/kg)	0.185	-	-
RON	40.5	-	92
Cetane Number	32.60	53	-

It can be seen that the liquid oil produced in this experiment has low heating value, octane and cetane number compared to the research done by [15] due to ignition as it can be seen in the temperature spike at T01, T02 and T03 in the figure b), c), d).

Table 3 GC-MS Result

No.	Retention Time	Compound	Cas Number	Area %
1	4.520	1-methyl-cyclohexene	591-49-1	1,93
2	4.814	1,3-cycloheptadiene	4054-38-0	1,33
3	5.162	Cyclooctane	292-64-8	2,69
4	5.408	Octane	111-65-9	2,3
5	6.456	2,4-dimethyl-1-heptene	19549-87-2	2,48
6	7.708	1-nonene	124-11-8	2,22
7	7.917	Nonane	111-84-2	2,14
8	9.847	1-decene	872-05-9	2,84
9	10.024	Decane	124-18-5	2,02
10	11.848	Undecane	1120-21-4	3,34
11	13.190	1,11-dodecadiene	5876-87-9	2,19
12	13.345	1-dodecene	112-41-4	3,23
13	16.437	Dodecane	295-17-0	2,55
14	16.309	1-Tridecene	629-59-4	5,05
15	16.164	Cyclotetradecane	1120-36-1	1,41
16	15.319	1,13-tetradecadiene	21964-49-8	2,07
17	14.870	1-tetradecene	295-17-0	3,16
18	13.490	tetradecane	1000141-67-3	2,41
19	36.141	Cyclotetradecane	6765-39-5	1,63
20	30.622	1-pentadecene	544-76-3	2,87
21	27.038	pentadecane	629-73-2	2,45
22	26.289	cetane	629-62-9	3,5
23	23.706	hexadecane	2437-56-1	2,16
24	23.214	1-heptadecane	112-40-3	1,42
25	20.946	(Z)-3-heptadecene	629-62-9	4,54
26	20.684	Heptadecane	629-50-5	4,31
27	19.791	octadecane	74685-33-9	5,37
28	19.309	1-octadecane	629-92-5	2,66
29	19.127	nonadecane	112-88-9	9,01
30	17.844	(E)-3-eicosane	593-45-3	3,79
31	17.710	tridecane	629-78-7	7,59
32	16.752	Pentadecane	13360-61-7	3,34
Total:				100

Conclusion

From this experiment, a 50 gram HDPE with the size of 3-5 mm within a fixed bed non-sweep gas reactor under temperature of 450-600 °C was observed and showed that using L-Shape heat pipe heat exchanger as condenser can effectively cool the smoke that is produced during a

pyrolysis and turn it into liquid oil. The temperature which produced the highest amount of liquid oil is 550 °C with the amount of liquid produced are 70.2% wt. The liquid oil that is produced has a characteristic that has similarities with diesel fuel with densities at 0.83 g/ml, kinematic viscosity of 2.045 c St at 40 °C, RON number of 40.5, cetane number of 32.6.

Acknowledgement

The author would like to thank the Directorate of Research and Community Engagements Universitas Indonesia for funding this research through the PITTA B 2019 scheme with contract number NKB-0738/UN2.R3.1/HKP.05.00/2019.

Reference

- [1] N. H. H. Bashir, "Plastic problem in Africa," no. 02, p. 11, 2013.
- [2] J. R. Jambeck *et al.*, "the ocean," vol. 347, no. 6223, 2015.
- [3] M. N. Siddiqui, "Journal of Hazardous Materials," vol. 167, pp. 728–735, 2009.
- [4] F. V. W. Silpa Kaza, Lisa Yao, Perinaz Bhada-Tata, *What a Waste 2.0*. WORLD BANK GROUP, 2018.
- [5] M. Rehan *et al.*, "Effect of zeolite catalysts on pyrolysis liquid oil," *Int. Biodeterior. Biodegrad.*, vol. 119, pp. 162–175, 2017.
- [6] T. Karayildirim and S. Uc, "Liquefaction of municipal waste plastics in VGO over acidic and non-acidic catalysts q," vol. 82, pp. 415–423, 2003.
- [7] R. Miandad, M. Rehan, and A. Nizami, "The Energy and Value-Added Products from Pyrolysis of Waste Plastics," 2016.
- [8] F. Pinto, P. Costa, I. Gulyurtlu, and I. Cabrita, "Pyrolysis of plastic wastes . 1 . Effect of plastic waste composition on product yield," vol. 51, pp. 39–55, 1999.
- [9] M. Rehan *et al.*, "Environmental Effects Pyrolytic liquid fuel : A source of renewable electricity generation in Makkah Pyrolytic liquid fuel : A source of renewable electricity generation in Makkah," *Energy Sources, Part A Recover. Util. Environ. Eff.*, vol. 38, no. 17, pp. 2598–2603, 2016.
- [10] I. De Marco, B. Caballero, A. Torres, M. F. Laresgoiti, and M. A. Cabrero, "Recycling polymeric wastes by means of pyrolysis," vol. 824, no. October 2000, pp. 817–824, 2002.
- [11] J. Jamradloedluk and C. Lertsatitthanakorn, "Characterization and Utilization of Char Derived from Fast Pyrolysis of Plastic Wastes," *Procedia Eng.*, vol. 69, pp. 1437–1442, 2014.
- [12] W. Kaminsky, M. Predel, and A. Sadiki, "Feedstock recycling of polymers by pyrolysis in a fluidised bed *," vol. 85, pp. 1045–1050, 2004.
- [13] N. Kiran, E. Ekinici, and C. E. Snape, "Pyrolysis of virgin and waste polypropylene and its mixtures with waste polyethylene and polystyrene," vol. 24, pp. 173–181, 2004.
- [14] P. T. Williams and E. A. Williams, "Fluidised bed pyrolysis of low density polyethylene to produce petrochemical feedstock," vol. 51, pp. 107–126, 1999.
- [15] P. Taylor and W. Ahmad, "International Journal of Green Energy," no. December, pp. 37–41, 2014.
- [16] W. Yan, "ScienceDirect Preliminary Study for the Passive Containment Cooling System Analysis of the Advanced PWR," vol. 39, pp. 240–247, 2013.
- [17] N. Abdullah, T. Jordy, I. Hakim, N. Putra, and R. Koestoer, "Influence of Feedstock Particle Size from Merbau Wood (Intsia bijuga) on Bio-Oil Production Using a Heat Pipe Fin L- Shaped Condenser in a Pyrolysis Process," 2018.
- [18] J. Liu, J. Jiang, and W. Yang, "Preparation of High heating value gas , High quality bio-oil and Added Value Carbon Materials from Caragana Pyrolyzed Via Super-high

- Temperature Steam,” vol. 515, pp. 2152–2161, 2012.
- [19] A. Khasri, M. Ridzuan, M. Jamir, and A. R. Mohamed, “Pyrolysis of Empty Fruit Bunch (EFB) in a vertical fixed bed reactor Pyrolysis of Empty Fruit Bunch (EFB) in a vertical fixed bed reactor,” no. April 2015, 2014.
 - [20] N. Bordoloi, R. Narzari, D. Sut, R. Saikia, R. S. Chutia, and R. Kataki, “Characterization of bio-oil and its sub-fractions from pyrolysis of *Scenedesmus dimorphus*,” *Renew. Energy*, pp. 1–9, 2016.
 - [21] R. Garg, N. Anand, and D. Kumar, “Pyrolysis of babool seeds (*Acacia nilotica*) in a fixed bed reactor and bio-oil characterization,” *Renew. Energy*, vol. 96, pp. 167–171, 2016.
 - [22] N. Abdullah, I. I. Hakim, N. Putra, and R. A. Koestoer, “An Experimental Study of the Vapor Temperature in the Reaction Zone for Producing Liquid from Camphor Wood in a Non-sweeping Gas Fixed-bed Pyrolysis Reactor,” no. December, 2018.
 - [23] A. Demirbas, “Relationships derived from physical properties of vegetable oil and biodiesel fuels,” vol. 87, pp. 1743–1748, 2008.
 - [24] PT. PERTAMINA, “PERTAMINA DEX MATERIAL SAFETY DATA SHEET,” 2007.
 - [25] P. PERTAMINA, “PERTAMAX MATERIAL SAFETY DATA SHEET,” 2007.

MAGNETIC FIELD EFFECT ON MIXED CONVECTION HEAT TRANSFER IN A LID-DRIVEN RECTANGULAR CAVITY

N.A. Bakar¹
R. Roslan²
A. Karimipour³

¹Department of Science and Mathematics, Centre for Diploma Studies, Universiti Tun Hussein Onn Malaysia, 84600 Pagoh, Johor, Malaysia

²Centre for Research in Computational Mathematics, Faculty of Sciences and Technology, Universiti Tun Hussein Onn Malaysia, 84600 Pagoh, Johor, Malaysia (Email: rozaini@uthm.edu.my)

³Department of Mechanical Engineering, Faculty of Engineering, Najafabad Branch Islamic Azad University, 8514143131 Isfahan, Iran

Abstract: *This study aims to analyze numerically the fluid flow and heat transfer in two-dimensional rectangular cavity in the presence of magnetic field. The bottom and top walls of the cavity are kept at constant temperature, T_h and T_c respectively, where $T_h > T_c$, while the vertical walls are kept insulated perfectly. The top wall is moving at a constant speed in the positive horizontal direction. The dimensionless governing equations were solved using finite volume method and SIMPLE algorithm. Influence of Hartmann number (between 0 and 60) on the thermal characteristics and fluid flow were analyzed. The resulting numerical solutions are demonstrated graphically in terms of the streamlines and isotherm plots, and the variation of Nusselt numbers. It is found that the Hartmann number have significant effect on fluid flow structure and temperature field.*

Keywords: *Cavity, finite volume method, lid-driven, magnetic field, mixed convection.*

Introduction

In the system of most engineering applications such as heat exchangers, building design, solar collectors, insulation materials or heat pump systems, mixed convection, namely, natural and forced convection are occurred in the same time [1] (Hussain et al. 2017). In addition, magnetic field are good control parameters for heat and fluid flow inside the enclosure is the attraction of many recent researchers due its applications in many vital industries like astrophysics, geophysics, aeronautics, metallurgy, chemical and petroleum industries, crystal growth in liquid, cooling of nuclear reactor, microelectronic devices and solar technology [2](Oğlakkaya and Bozkaya 2018).

Chamkha [3] made a numerical work on hydromagnetic combined convection flow in a lid-driven square cavity with internal heat generation using finite volume approach. The presence of the internal heat generation effects was found to decrease the average Nusselt number significantly for aiding flow and to increase it for opposing flow. Rahman et al. [4] analyzed the effect of Reynolds and Prandtl number on MHD mixed convection in a lid-driven rectangular cavity along with a heat conducting circular block using finite element technique. The results indicated that both the streamlines and isotherms strongly depend on the Reynolds number and Prandtl number. Oztop, Al-Salem, and Pop [5] investigated laminar mixed convection flow in the presence of magnetic field in a top sided lid-driven square cavity heated by a corner heater.

Sivasankaran et al. [6] studied mixed convection of sinusoidal boundary temperatures at the sidewalls of a square cavity in the presence of magnetic field numerically. They solve

the governing equations by employing the finite volume method. They found that magnetic field strongly affected the flow behavior and heat transfer rate inside the cavity. Numerical study of laminar magnetohydrodynamic mixed convection in an inclined lid-driven square cavity with opposing temperature gradients was investigated by Ahmed, Mansour, and Mahdy [7]. The results indicate that the rate of heat transfer along the heated walls is enhanced on increasing either Hartmann number or inclination angle. Al-Salem et al. [8] and Kefayati et al. [9] analyzed the effects of moving lid-direction on MHD mixed convection in a square cavity with the bottom wall being linearly heated. It is found that direction of lid is more effective on heat transfer and fluid flow in the case of mixed convection.

For instance, Muthtamilselvan and Doh [10], Kefayati [11], Kefayati [12] and Mahalakshmi et al. [13] investigated the performance of nanofluid in a lid-driven cavity with the presence of magnetic field. The results indicate that heat transfer depended on the strength of magnetic field while the suspended nanoparticles strongly improve the heat transfer effect. The mixed convection flow of nanofluids in a lid-driven square cavity under the influence of an inclined cavity was investigated by Ismael et al. [14] and Selimefendigil and Oztop [15]. They found that both orientation of magnetic field and the strength strongly affected the heat transfer process. Oztop et al. [16] examined the heat transfer performance of MHD flow in a nanofluid-filled lid-driven cavity with partially heated wavy wall. Selimefendigil and Oztop [17] investigated the effect of inclined magnetic field on the mixed convection in a trapezoidal cavity.

Based on the literature review, most of the researches were laminar mixed convection in a cavity especially square in shape subjected to different types of heating with the presence of magnetic field. However, the mixed convection heat transfer fluid flow of water-filled in a rectangular cavity with adiabatic vertical walls and horizontal magnetic field have thus far attracted relatively little attention. Hence, the present study performs an investigation by numerical simulation into the effect of magnetic field on the mixed convection heat transfer in a rectangular cavity filled with water.

Methodology

Mathematical Modelling

A shallow rectangular cavity with moving top lid is shown in Figure 1. Furthermore, the top lid has a constant speed U_0 that moves from the left to the right direction of x -axis. The width is denoted as L and the height is as H . The cavity is filled with Newtonian fluid that is heated from the bottom lid, T_h while the top lid is kept at cold temperature, T_c such that $T_h > T_c$. Both of the vertical walls are adiabatic. A uniform magnetic field strength B is imposed to the cavity in horizontal direction parallel to the x -axis.

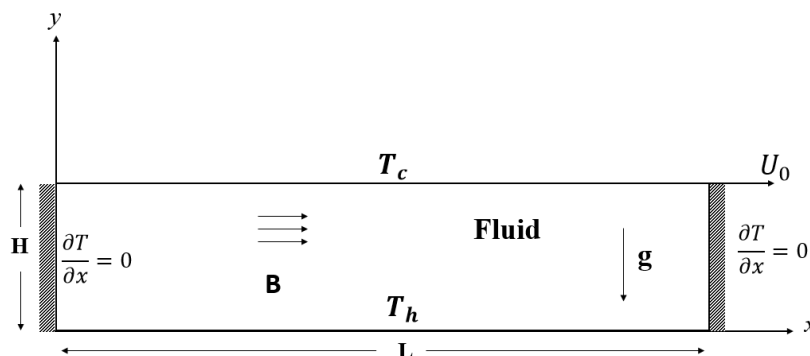


Fig. 1. Geometrical configuration

The induced magnetic field due to the motion of the electrically conducting fluid is neglected in comparison to the external magnetic field. Additionally, the imposed and induced electrical fields, the Joule heating of the fluid and the effect of viscous dissipation are negligible. The working fluid is incompressible, steady, two-dimensional and laminar flow. The gravitational force acts vertically downward. It is assumed that the thermophysical properties of the fluid are constant. The extent of the variation of density with temperature varies from one fluid to another. In order to represent the variation of density with temperature, well known Boussinesq approximation is used in present study. After the mathematical rearrangement, the governing equations are expressed as follows:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0, \quad (1)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right), \quad (2)$$

$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) + g\beta(T - T_c) - \frac{\sigma B^2}{\rho} v, \quad (3)$$

$$u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \alpha \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right), \quad (4)$$

where x and y are the coordinates directions. The variables u , v , p and T are the velocity components of the fluid in x - and y - directions, the pressure, and the temperature, respectively. The parameters β , g , σ , ν , and ρ are the fluid thermal expansion coefficient, the gravity, the electrical conductivity, the kinematic viscosity, and the density, respectively. The parameter B is the magnetic induction coefficient while $\alpha = k/\rho c$ is the thermal diffusivity, k is the thermal conductivity and c is the heat capacity. The early stage boundary conditions of the problem are given as:

$$\begin{aligned} \text{Top wall:} \quad & u = U_0, \quad v = 0, \quad T = T_c, \\ \text{Bottom wall:} \quad & u = v = 0, \quad T = T_h, \\ \text{Left and right walls:} \quad & u = v = 0, \quad \frac{\partial T}{\partial x} = 0. \end{aligned} \quad (5)$$

The formulated dimensional governing Eqs. (1) – (5) have been reduced to non-dimensional equations using the following variables,

$$\begin{aligned} X = x/H, \quad Y = y/H, \\ U = u/U_0, \quad V = v/U_0, \quad \theta = T - T_c/T_h - T_c, \\ Gr = g\beta(T_h - T_c)H^3/\nu^2, \quad Pr = \nu/\alpha, \quad P = p/\rho U_0^2, \quad Re = U_0 H/\nu, \\ Ha^2 = B^2 H^2 \sigma / \rho \nu, \end{aligned} \quad (6)$$

where parameters θ , P , Gr , Re , Pr , Ri and Ha^2 are the non-dimensional temperature, the non-dimensional pressure, the Grashof number, the Reynolds number, the Prandtl number, the Richardson number and the Hartmann number, respectively. Eqs. (1) – (5), on using Eq. (6), now become,

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0, \quad (7)$$

$$U \frac{\partial U}{\partial X} + V \frac{\partial U}{\partial Y} = -\frac{\partial P}{\partial X} + \frac{1}{Re} \left(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2} \right), \quad (8)$$

$$U \frac{\partial V}{\partial X} + V \frac{\partial V}{\partial Y} = -\frac{\partial P}{\partial Y} + \frac{1}{Re} \left(\frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2} \right) + \frac{Gr}{Re^2} \theta - \frac{Ha^2}{Re} V, \quad (9)$$

$$U \frac{\partial \theta}{\partial X} + V \frac{\partial \theta}{\partial Y} = \frac{1}{Re Pr} \left(\frac{\partial^2 \theta}{\partial X^2} + \frac{\partial^2 \theta}{\partial Y^2} \right). \quad (10)$$

where Gr/Re^2 is the Richardson number Ri . The governing equations are subject to the following dimensionless boundary conditions,

$$\begin{aligned} \text{Top wall:} & \quad U = 1, \quad V = 0, \quad \theta = 0, \\ \text{Bottom wall:} & \quad U = V = 0, \quad \theta = 1, \\ \text{Left and right walls:} & \quad U = V = 0, \quad \frac{\partial \theta}{\partial X} = 0. \end{aligned} \quad (11)$$

Numerical Method

The set of partial differential Eqs. (7) – (10) together with the boundary conditions (11) have been solved numerically by finite volume method. The set of algebraic equations produced were solved using the SIMPLE algorithm and the tridiagonal matrix algorithm (TDMA) iteratively. The numerical solutions are portrayed on a staggered grid system, such that the velocity components are stored halfway between the scalar storage locations. The calculations of U , V and θ are iteratively performed until the following convergence criterion is fulfilled:

$$\varepsilon = \frac{\sum_{j=1}^m \sum_{i=1}^n |\zeta_{i,j}^{k+1} - \zeta_{i,j}^k|}{\sum_{j=1}^m \sum_{i=1}^n |\zeta_{i,j}^{k+1}|} \leq 10^{-7}, \quad (12)$$

where ε is the tolerance, m and n are the number of grid points in x - and y -direction, respectively. k is the iteration number and ζ is any computed variables. Nusselt number is obtained to investigate the heat transfer rate as follows,

$$\overline{Nu} = \frac{1}{L/H} \int_0^{L/H} - \left(\frac{\partial \theta}{\partial Y} \right)_{Y=0,1} dX, \quad (13)$$

The computational procedure is incorporated into the programming language, FOTRAN90. Grid independence test was conducted in order to ensure grid-independency solutions. The test is performed for $Ri = 1.0$, $Pr = 7.0$, and $Ha = 10.0$ in a bottom heated, top cooled moving lid and vertically adiabatic walls. A 100×20 grid points with clustering towards the walls is chosen and the calculation were done for $Ri = 1.0$ and for different values of Ha . The comparisons of the average Nusselt number along the horizontal walls is done between the present computation with Sharif [18] and Iwatsu et al. [19] and Khanafer and Chamka [20]. The results showed insignificant differences and is considered as a good agreement.

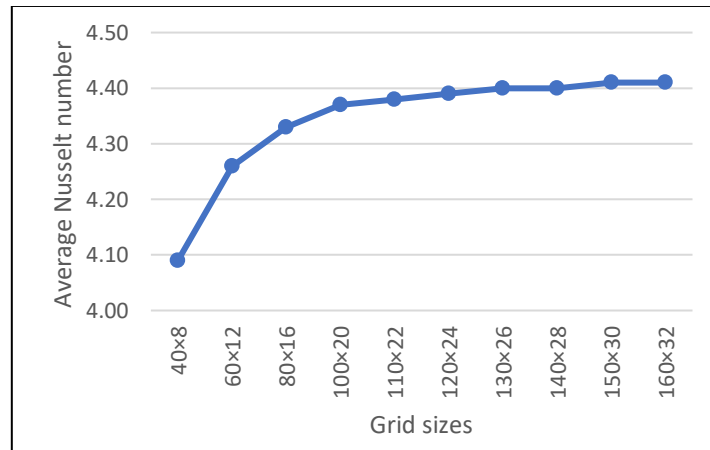


Fig. 2. Variation of mean Nusselt number with grid mesh size with $Ri = 1.0$, $Pr = 7.0$, and $Ha = 10.0$

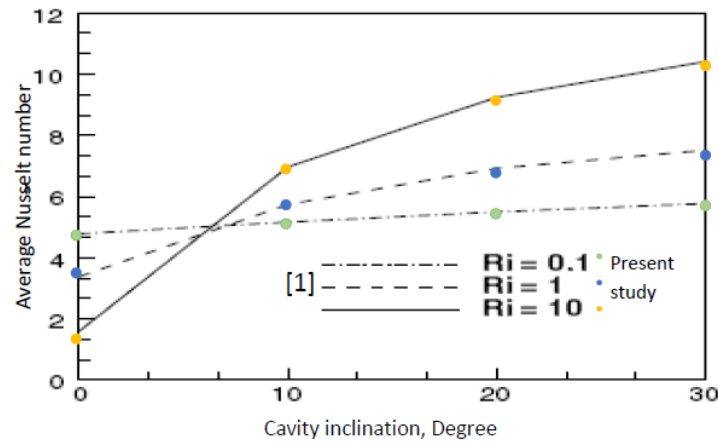


Fig. 3. Comparison of current study and Sharif [18]

Table 1: Comparisons of the maximum and minimum values of the horizontal and vertical velocities at the mid-section of the cavity between present solution and Iwatsu et al. [19] and Khanafer and Chamka [20]

Re = 400.0				
	Iwatsu et al. [19]	Khanafer and Chamka [20]	Present	
U_{min}	-0.3197	-0.3099	-0.3023	
U_{max}	1.0000	1.0000	1.0000	
V_{min}	-0.4459	-0.4363	-0.4219	
V_{max}	0.2955	0.2866	0.2802	
Re = 100.0				
U_{min}	-0.2122	-0.2037	-0.2049	
U_{max}	1.0000	1.0000	1.0000	
V_{min}	-0.2506	-0.2448	-0.2328	
V_{max}	0.1765	0.1699	0.1673	

Results and discussion

This section discusses the results obtained from the numerical simulations of mixed convection fluid flow and heat transfer in a lid-driven rectangular cavity filled with Newtonian fluid in the presence of magnetic field with heated motionless bottom wall, cooled moving top wall and other vertical walls are adiabatic. The following are the controlling parameters values used in the simulation: Reynold number ($Re = 100.0$), Prandtl number ($Pr = 6.2$), Grashof number ($Gr = 10^4$), Hartmann number ($Ha = 0 - 60$) and Richardson number ($Ri = 1.0$).

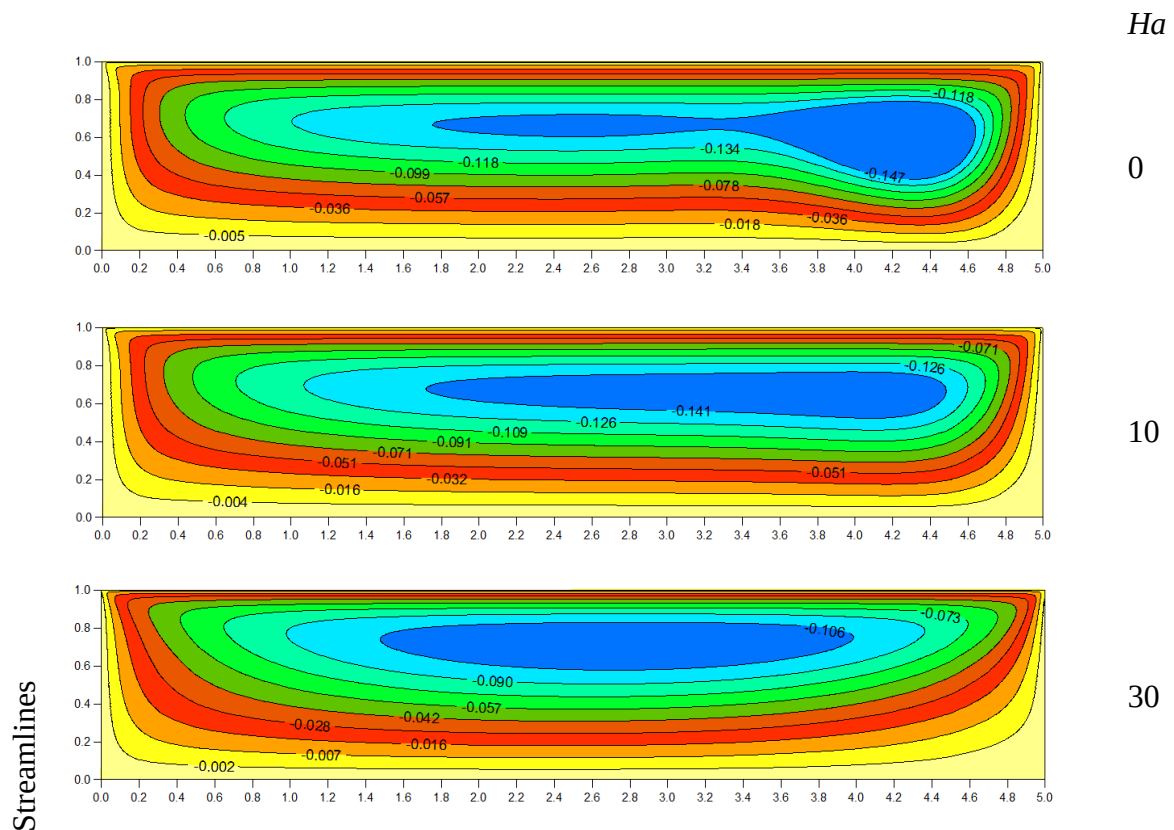
Figure 4 presents the streamlines for different values of Ha . The streamlines show that a primary clockwise recirculating vortex occupied the cavity. The core of the vortex is situated near the right wall and seem like that the streamlines are almost separated halfway. This shows that the fluid flow is dominated by the shear force created by the constant movement of the top lid. Imposing the magnetic field to the cavity with $Ha = 10$, the strength of the rotating vortex is reduced. It can be seen that the primary vortex is elliptic in shape with the increasing of the magnetic field. As the value of Ha increases to 30, the core vortex is pushed to the top wall of the cavity. This is due to the establishment of the conduction mode of heat transfer. As the value of magnetic field increases, the core vortex expands horizontally. It indicates the reduction of the flow strength inside the cavity.

The effect of Ha on the isotherms can be seen in Figure 5. When there is no magnetic field imposed to the cavity, high temperature gradient exists at the vicinity of right bottom wall indicating the effect of the top wall movement. When the magnetic field increases, the temperature gradient, flow intensity and velocity decreases. The isotherms become almost parallel to the adiabatic wall and evenly distributed especially near the bottom wall. This phenomenon indicates that heat transfers by conduction.

Conclusions

In the present study, mixed convection heat transfer in a rectangular cavity filled with water in the presence of horizontal magnetic field is studied numerically by finite volume method. The bottom and top walls of the cavity are kept at constant temperature, T_h and T_c respectively, while the vertical walls are kept insulated perfectly. The top wall is moving at a constant speed in the positive horizontal direction. The effect of mixed convection ($Ri = 1.0$) and Ha on the fluid flow and heat transfer mode are investigated. From the numerical results, the following are concluded:

- As Ha increases, the convective flows become weaker and therefore the heat transfer rate decreases.
- The recirculating vortex is pushed toward the top lid as the magnetic field strength increases due to the decreases of the convective flow hence, conduction is the mode of heat transfer.



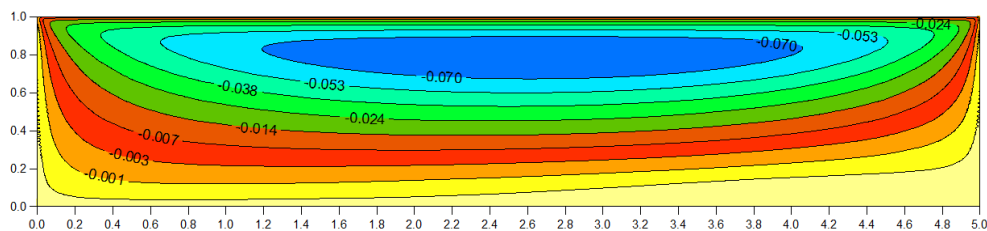


Fig. 4. Variation of streamlines for different Ha value while $Ri = 1.0$

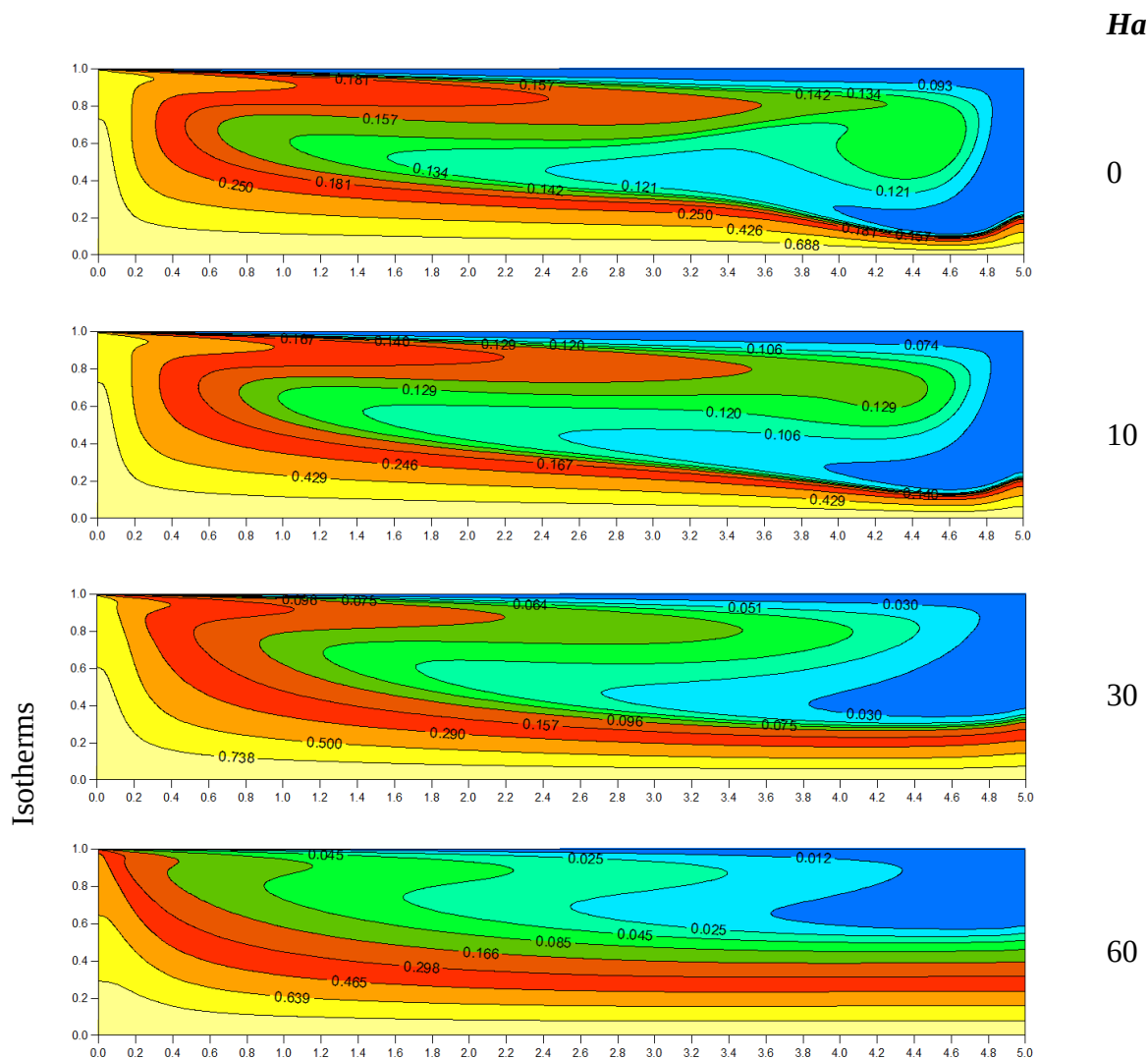


Fig. 5. Variation of isotherms for different Ha value while $Ri = 1.0$

Acknowledgement

The author would like to acknowledge the financial support received from Universiti Tun Hussein Onn Malaysia Grant No. TIER 1/H184 and Ministry of Higher Education of Malaysia (MyBrain15).

References

- [1] Hussain, S., S. Ahmad, K. Mehmood, and M. Sagheer. 2017. "Effects of Inclination Angle

- on Mixed Convective Nanofluid Flow in a Double Lid-Driven Cavity with Discrete Heat Sources.” *International Journal of Heat and Mass Transfer* 106: 847–60. <https://doi.org/10.1016/j.ijheatmasstransfer.2016.10.016>.
- [2] Oğlakkaya, F. S., and Canan Bozkaya. 2018. “Unsteady MHD Mixed Convection Flow in a Lid-Driven Cavity with a Heated Wavy Wall.” *International Journal of Mechanical Sciences* 148 (February): 231–45. <https://doi.org/10.1016/j.ijmecsci.2018.08.036>.
 - [3] Chamka, A. J. 2002. “Hydromagnetic combined convection flow in a vertical lid-driven cavity with internal heat generation or absorption.” *Numerical Heat Transfer, Part A* 41, 529–546.
 - [4] Rahman, M. M., M. M. Billah, M. A.H. Mamun, R. Saidur, and M. Hasanuzzaman. 2010. “Reynolds and Prandtl Numbers Effects on Mhd Mixed Convection in a Lid-Driven Cavity along with Joule Heating and a Centered Heat Conducting Circular Block.” *International Journal of Mechanical and Materials Engineering* 5 (2): 163–70.
 - [5] Oztop, Hakan F., Khaled Al-Salem, and Ioan Pop. 2011. “MHD Mixed Convection in a Lid-Driven Cavity with Corner Heater.” *International Journal of Heat and Mass Transfer* 54 (15–16): 3494–3504. <https://doi.org/10.1016/j.ijheatmasstransfer.2011.03.036>.
 - [6] Sivasankaran, S., A. Malleswaran, Jinho Lee, and Pon Sundar. 2011. “Hydro-Magnetic Combined Convection in a Lid-Driven Cavity with Sinusoidal Boundary Conditions on Both Sidewalls.” *International Journal of Heat and Mass Transfer* 54 (1–3): 512–25. <https://doi.org/10.1016/j.ijheatmasstransfer.2010.09.018>.
 - [7] Ahmed, Sameh E., M. A. Mansour, and A. Mahdy. 2013. “MHD Mixed Convection in an Inclined Lid-Driven Cavity with Opposing Thermal Buoyancy Force: Effect of Non-Uniform Heating on Both Side Walls.” *Nuclear Engineering and Design* 265: 938–48. <https://doi.org/10.1016/j.nucengdes.2013.06.023>.
 - [8] Al-Salem, Khaled, Hakan F. Öztop, Ioan Pop, and Yasin Varol. 2012. “Effects of Moving Lid Direction on MHD Mixed Convection in a Linearly Heated Cavity.” *International Journal of Heat and Mass Transfer* 55 (4): 1103–12. <https://doi.org/10.1016/j.ijheatmasstransfer.2011.09.062>.
 - [9] Kefayati, Gh R., M. Gorji-Bandpy, H. Sajjadi, and D. D. Ganji. 2012. “Lattice Boltzmann Simulation of MHD Mixed Convection in a Lid-Driven Square Cavity with Linearly Heated Wall.” *Scientia Iranica* 19 (4): 1053–65. <https://doi.org/10.1016/j.scient.2012.06.015>.
 - [10] Muthamilselvan, M., and Deog Hee Doh. 2014. “Magnetic Field Effect on Mixed Convection in a Lid-Driven Square Cavity Filled with Nanofluids.” *Journal of Mechanical Science and Technology* 28 (1): 137–43. <https://doi.org/10.1007/s12206-013-0953-6>.
 - [11] Sharif, M. A.R. 2007. “Laminar Mixed Convection in Shallow Inclined Driven Cavities with Hot Moving Lid on Top and Cooled from Bottom.” *Applied Thermal Engineering* 27 (5–6): 1036–42. <https://doi.org/10.1016/j.applthermaleng.2006.07.035>.
 - [12] Kefayati, G. H. R. 2013. “Lattice Boltzmann simulation of MHD natural convection in a nanofluid-filled cavity with sinusoidal temperature distribution,” *Powder Technology* 243. 171–183.
 - [13] Kefayati, G.H.R., Gorji-Bandpy, M., Sajjadi, H. & Ganji, D.D. 2012. Lattice Boltzmann simulation of MHD mixed convection in a lid-driven square cavity with linearly heated wall. *Scientia Iranica B* 19(4): 1053-1065.
 - [14] Khanafer, K.M. & Chamkha, A.J. 1999. Mixed convection flow in a lid-driven enclosure filled with a fluid-saturated porous medium. *Heat and Mass Transfer* 42: 2465-2481.
 - [15] Iwatsu, R., Hyun, J.M. & Kuwahara, K. 1993. Mixed convection in a driven cavity with a stable vertical temperature gradient. *International Journal of Heat and Mass Transfer* 36:

1601-1608.

Nomenclature

B	Magnetic field
c	Specific heat capacity, J/kg K
g	Acceleration due to gravity, m/s ²
Gr	Grashof number
H	Height width of the cavity, m
k	Thermal conductivity of fluid, W/m K
L	Width of the cavity, m
$\overline{Nu}$	Average Nusselt number
p	Pressure
P	Dimensionless pressure
Pr	Prandtl number
Re	Reynolds number
Ri	Richardson number
t	Time
T	Temperature, °C
T_c	Temperature at cold wall, °C
T_h	Temperature at hot wall, °C
u, v	Velocities in the x- and y-direction respectively
U, V	Dimensionless velocity in X- and Y-direction respectively
U_0	Lid velocity, m/s
ν	Kinematic viscosity, m ² /s
x, y	Cartesian coordinates
X, Y	Dimensionless Cartesian coordinates

Greek symbols

α	Thermal diffusivity, m ² /s
β	Coefficient of thermal expansion of fluid, K ⁻¹
θ	Dimensionless temperature
ρ	Fluid density, kg/m ³
φ	Inclination angle
τ	Dimensionless time
ψ	Stream function

BOUNDARY LAYER FLOW AND HEAT TRANSFER OF BLASIUS AND SAKIADIS PROBLEMS IN NANOFUIDS WITH PARTIAL SLIP AND THERMAL CONVECTION

Nor Hathirah Abd Rahman^{1*}

Norfifah Bachok^{1,2}

Haliza Rosali¹

¹Department of Mathematics, Faculty of Science, Universiti Putra Malaysia, UPM Serdang, Selangor 43400, Malaysia

²Institute for Mathematical Research, Universiti Putra Malaysia, UPM Serdang, Selangor 43400, Malaysia

*(Corresponding Author's Email: norhathirah95@gmail.com)

Abstract: This study aims to investigate the steady two-dimensional laminar boundary layer flow past a fixed (Blasius) or past a moving (Sakiadis) semi-infinite flat plate in water-based nanofluids with partial slip and thermal convective boundary condition. The similarity equations are solved numerically for three types of metallic or nonmetallic nanoparticles such as copper (Cu), alumina (Al₂O₃), and titania (TiO₂) in the base fluid of water with the Prandtl number $Pr = 6.2$ to investigate the effect of the solid volume fraction parameter ϕ of the nanofluids. The governing partial differential equations are transformed into a system nonlinear ordinary differential equations using a similarity transformation which is then solved numerically using a shooting method in Maple software. The numerical results are presented in tables and graphs for the skin friction coefficient C_f and local Nusselt number Nu which represents the heat transfer rate at the surface as well as the velocity and temperature profile for a range of various parameters such as nanoparticles volume fraction, slip parameter and Biot number. The results indicate that the solid volume fraction affects the fluid flow and heat transfer characteristics.

Keywords: Heat transfer, Blasius, Sakiadis, Partial Slip, Nanofluids

Introduction

In 1908, Blasius were among the first who studied the problems of boundary layer flow past a static semi-infinite flat plate, without considering the heat transfer aspect. Blasius himself gave the matching inner and outer series solutions and the Blasius equation was never yielded to exact analytical solution. While, in the 1961, Sakiadis had been investigated the boundary layer flow over a continuous solid surface moving with constant velocity. He found the exact equation as Blasius, but in different boundary conditions. Pantokratoras [1] had presented a theoretical study of the effect of variable fluid properties on the classical Blasius and Sakiadis flow. These investigation concerns about engine oil, water and air taking into account the variation of their physical properties with temperature. The result of numerical simulation of the governing equations and cover large temperature differences are obtained. Ishak et al. [2] have study the extended classical Blasius and Sakiadis equations, by considering a uniform free stream parallel to a fixed or moving flat plate, which has more practical significance. It is assumed that the plate is subjected to a constant heat flux, and moves in the same or opposite direction to the free stream.

However, the above problems have not dealt with the nanofluid. In this field, there are two types of model that have been found which are Buongiorno [3] model and Tiwari and Das [4] model. Tiwari and Das [4] model analyses the behavior of nanofluids into the nanoparticle

volume fraction ϕ parameter. Ahmad et al. [5] had investigated the classical problems of forced convection boundary layer flow and heat transfer past a semi-infinite static flat plate (Blasius problem) and past a moving semi-infinite flat plate (Sakiadis problem) using nanofluids. This problems solved by using the model proposed by Tiwari and Das [4] by the shooting method. It is found that the inclusion of nanoparticles into the base water fluid has produced the increasing of skin friction, heat transfer coefficients and volume fraction. Then, Bachok et al. [6] had extended the Blasius and Sakiadis problems in nanofluids, by considering a uniform free stream parallel to a fixed or moving flat plate, which has more practical significance. The plate was assumed moves in the same or opposite direction to the free stream. The effect of the nanoparticle volume fraction parameter of the nanofluids on the heat transfer characteristics is investigated. The results indicate that dual solutions exist when the plate and the free stream move in the opposite directions. Anuar and Bachok [7] have studied the classical problems of boundary layer flow and heat transfer characteristics past a semi-infinite static flat plate (Blasius problem) and past a moving semi-infinite flat plate (Sakiadis problem) in a water-based nanofluids with Prandtl number ($Pr = 6.2$) by using the effects of Brownian motion Nb , thermophoresis Nt and nanoparticle volume fraction ϕ parameters.

Noghrehabadi et al. [8] have analyzed the development of the slip effects on the boundary layer flow and heat transfer over a stretching surface in the presence of nanoparticle fractions. In the modeling of nanofluid the dynamic effects including the Brownian motion and thermophoresis are considered. A similarity solution is presented in the case of constant wall temperature. The solution depends on a Prandtl number, slip factor, Brownian motion number, Lewis number, and thermophoresis number. These five parameters is numerically investigated for the dependency of the local Nusselt and local Sherwood numbers. But, they not investigated the effects of slip boundary condition in the presence of dynamic effects of nanoparticles yet. The slip parameter had strongly influenced the flow velocity and the surface shear stress on the stretching sheet and also reduced Nusselt number and reduced Sherwood number. The stability of unsteady boundary layer flow and heat transfer over stretching/shrinking sheet immersed in Cu-water nanofluid with the presence of partial slip, Soret and Dufour effects have been studied by Dzulkifli et al. [9]. Najib et al. [10] investigated on boundary layer flow and mass transfer near stagnation point past a stretching or shrinking cylinder in copper water nanofluid under consideration of chemical reaction and slip effect. The numerical results indicate that with consideration of slip at the boundary causes to decrease the skin friction coefficient but increased the mass transfer rate.

Battaler [11] had analysed the effects of thermal radiation on the laminar boundary layer about a flat-plate in a uniform stream of fluid (Blasius flow), and about a moving plate in a quiescent ambient fluid (Sakiadis flow) both under a convective surface boundary condition. Ishak et al. [12] had investigated the steady laminar boundary layer flow over a moving plate in a moving fluid with convective surface boundary condition and in the presence of thermal radiation. They investigated under certain conditions, the present problem reduces to the classical Blasius and Sakiadis problems. They found that the heat transfer rate at the surface decreases in the presence of thermal radiation and convective boundary condition.

Therefore, in this paper, we extend the work of Ahmad et al. [5] by including the effect of partial slip and thermal convective boundary condition on the steady two-dimensional laminar boundary layer flow past a Blasius and Sakiadis semi-infinite plate in a nanofluid by using Tiwari and Das [4] model. The effects of the partial slip, thermal convective boundary condition, solid volume fraction and the types of nanoparticles on characteristics of energy flow will be studied numerically and discussed further. For some particular cases of the present study, the results are compared with Ahmad et al. [5] to support their validity.

Methodology

Consider the steady two-dimensional boundary layer flow past a fixed (Blasius) or past a moving (Sakiadis) semi-infinite flat plate in a water-based nanofluid containing different types of nanoparticles: Cu, Al₂O₃ and TiO₂. It is assumed that the nanofluid is incompressible, laminar flow and the viscous dissipation and radiation effects are neglected. Let u and v be the velocity components along x and y directions. The flow takes places at $y \geq 0$ where y is the coordinate measured normal to the surface. Further, we assume that the uniform temperature and the uniform nanofluid volume fraction at the surface of the plate are T_w and C_w , while the uniform temperature and the uniform nanofluid volume fraction far from the surface of the plate are T_∞ and C_∞ , respectively.

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (1)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \frac{\mu_{nf}}{\rho_{nf}} \frac{\partial^2 u}{\partial y^2} \quad (2)$$

$$u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \alpha_{nf} \frac{\partial^2 T}{\partial y^2} \quad (3)$$

We assume that these equations are subject to the boundary conditions:

i) Blasius problem

$$\begin{aligned} v &= 0, u = L \left(\frac{\partial u}{\partial y} \right) \text{ at } y = 0 \\ u &= U \text{ as } y \rightarrow \infty \end{aligned} \quad (4)$$

ii) Sakiadis problem

$$\begin{aligned} v &= 0, u = U + L \left(\frac{\partial u}{\partial y} \right) \text{ at } y = 0 \\ u &= 0 \text{ as } y \rightarrow \infty \end{aligned} \quad (5)$$

where u and v are the velocity components along the axes x and y . U is the constant velocity of the free stream or that of a moving flat plate and L denotes the slip length, T is the temperature of a nanofluid. The boundary conditions for the energy equation are

$$-k \left(\frac{\partial T}{\partial y} \right) = h_f (T_w - T) \text{ at } y = 0, \quad T = T_\infty \text{ as } y \rightarrow \infty \quad (6)$$

It is that ρ is the pressure of nanofluid, μ_{nf} is the dynamic viscosity of the nanofluid, ρ_{nf} is the density of the nanofluid and α_{nf} is the thermal diffusivity of the nanofluid, which are given by Oztop and Abu-Nada [13].

$$\mu_{nf} = \frac{\mu_f}{(1 - \varphi)^{2.5}}, \quad \rho_{nf} = (1 - \varphi)\rho_f + \varphi\rho_s, \quad \alpha_{nf} = \frac{k_{nf}}{(\rho C_p)_{nf}}$$

$$\frac{k_{nf}}{k_f} = \frac{(k_s + 2k_f) - 2\varphi(k_f - k_s)}{(k_s + 2k_f) + \varphi(k_f - k_s)}, \quad (\rho C_p)_{nf} = (1 - \varphi)(\rho C_p)_f + \varphi(\rho C_p)_s$$

Where φ is the nanoparticle volume fraction parameter of the nanofluid, k_{nf} is the thermal conductivity of the fluid fraction, k_s is the thermal conductivity of the nanoparticle volume fraction, ρ_f is the reference density of solid fraction, μ_f is the viscosity of the fluid fraction and $(\rho C_p)_{nf}$ is the heat capacitance of the nanofluids, where C_p is the specific heat at constant pressure. The viscosity μ_{nf} of the nanofluid given by Brinkman [14] can be approximated as viscosity of the base fluid μ_f containing dilute suspension of fine spherical particles.

To obtain a similarity solution for equation (1) – (6), the similarity transformation is introduced:

$$\psi = (Uv_f x)^{\frac{1}{2}} f(\eta), \quad \theta(\eta) = \frac{(T - T_\infty)}{(T_w - T_\infty)}, \quad \eta = (U/v_f x)^{\frac{1}{2}} y \quad (7)$$

where v_f is the kinematic viscosity of the fluid fraction and ψ is the stream function that is defined as $u = \frac{\partial \psi}{\partial y}$ and $v = -\frac{\partial \psi}{\partial x}$, which satisfy equation (1). By substituting variables (7) into equations (2) and (3), the transformed ordinary differential equations are obtained:

$$\frac{1}{(1 - \varphi)^{2.5}(1 - \varphi + \varphi \rho_s / \rho_f)} f''' + \frac{1}{2} f f'' = 0 \quad (8)$$

$$\frac{1}{\text{Pr} \left[\frac{k_{nf}/k_f}{(1 - \varphi) + \varphi (\rho C_p)_s / (\rho C_p)_f} \right]} \theta'' + \frac{1}{2} f \theta' = 0 \quad (9)$$

The corresponding initial and boundary conditions (4), (5) and (6) are

$$f(0) = 0, \quad f'(0) = \sigma f''(0), \quad f'(\infty) \rightarrow 1 \quad (10)$$

$$f(0) = 0, \quad f'(0) = 1 + \sigma f''(0), \quad f'(\infty) \rightarrow 0 \quad (11)$$

$$\theta'(0) = Bi(\theta(0) - 1), \quad \theta(\infty) \rightarrow 0 \quad (12)$$

where primes denote differentiation with respect to η . $\text{Pr} = \nu_f / \alpha_f$ is the Prandtl number, $\sigma = L \left(\frac{U}{v_f x} \right)^{1/2}$ is the slip parameter and $Bi = \frac{c}{k} \left(\frac{v_f}{U} \right)^{1/2}$ is the Biot number.

Quantities of practical interest which are the skin friction coefficient C_f and the local Nusselt number Nu are defined as

$$C_f = \frac{\tau_w}{\rho_f U^2}, \quad Nu = \frac{x q_w}{k_f (T_w - T_\infty)} \quad (13)$$

where τ_w is the surface shear stress and q_w is the surface heat flux which can be expressed as

$$\tau_w = \mu_{nf} \left(\frac{\partial u}{\partial y} \right)_{y=0}, \quad q_w = -k_{nf} \left(\frac{\partial T}{\partial y} \right)_{y=0} \quad (14)$$

with μ_{nf} is the dynamic viscosity of nanofluids and k_{nf} is thermal conductivity of nanofluids. Using the variables of similarity (7), we obtained

$$Re_x^{1/2} C_f = \frac{1}{(1 - \varphi)^{2.5}} f''(0), \quad Re_x^{-1/2} Nu = -\frac{k_{nf}}{k_f} \theta'(0) \quad (15)$$

Results

The nonlinear ordinary differential equations (8) and (9) subject to the boundary conditions (10) and (11) has been solved numerically using the shooting method. It is an iterative algorithm technique implemented in Maple program which attempts to identify the appropriate initial conditions for a related initial value problem (BVP). In this method, we are setting the different initial guesses for the values of $f''(0)$ and $-\theta'(0)$ in which all the profiles must satisfy the boundary conditions (10) and (11) asymptotically. The effect of the nanoparticle volume fraction φ , slip parameter σ and Biot number Bi are analysed for three different nanofluids: Cu, Al_2O_3 and TiO_2 as the working fluids and water as the base working fluid. The Prandtl number is taken to be $\text{Pr} = 6.2$ and nanoparticles value fraction is considered from 0 to 0.2 ($0 \leq \varphi \leq 0.2$), where $\varphi = 0$ is corresponding to the regular fluid. The thermophysical properties of the base fluid and nanoparticles are listed in Table 1.

Table 1: Thermophysical properties of the fluid phase (water) and nanoparticles Oztop and Abu-Nada [13]

Physical properties	Fluid phase (water)	Cu	Al ₂ O ₃	TiO ₂
C_p (J/kgK)	4179	385	765	686.2
ρ (kg/m ³)	997.1	8933	3970	4250
k (W/mK)	0.613	400	40	8.9538

Figures 1(a) until 2(b) illustrate the variations of the skin friction $f''(0)$ and local Nusselt number $-\theta'(0)$ for different nanoparticles, Biot number, nanoparticle volume fraction and also slip parameter for both Blasius and Sakiadis problem. These reduced skin friction and reduced Nusselt number are shown graphically for several values of slip parameter σ and Biot number Bi . Figures 1 (a) and (b) illustrate the variations of the skin friction coefficient given by equation (15) with parameter ϕ for three different nanoparticles: Copper Cu, Alumina Al₂O₃ and Titania TiO₂, respectively when $Pr = 6.2$. It is interesting to observe that the skin friction coefficient increase almost monotonically with increasing ϕ but the reverse effect for slip parameter for both Blasius and Sakiadis problem. These figures show that Al₂O₃ has the lowest skin friction coefficient and the difference between the values for TiO₂ and Al₂O₃ is very small, as can be seen from Tables 2 and 3 and Figures 1 (a) and (b).

However, this behaviour is the same with the reported by Ahmad et al. [5]. This is explained by looking at equation (11) where the ordinary differential equation shows that the nanoparticle volume fraction ϕ parameter and slip parameter σ give the effect to the result. Other than that, Tables 2 and 3 also shows the comparison of skin friction coefficient with Ahmad et al. [5], which show a favorable agreement, thus give confidence that the numerical results obtained are accurated.

However, we are more interested to know the influenced of the nanoparticle volume fraction ϕ parameter and slip σ parameter towards heat transfer rate. The variation of Nusselt number with ϕ for Blasius and Sakiadis problem are presented in Figures 2 (a) and (b) considering various value of σ parameter. It is seen that Nusselt number increase when parameter ϕ increase and it is also increase when parameter σ increasing for Blasius problem but not in Sakiadis problem. From the Figure 2 (a), Cu has the highest heat transfer rate, while TiO₂ has the lowest heat transfer rate compared to Cu due to domination of conduction mode of heat transfer. Table 1 clearly shows that TiO₂ has the lowest value of thermal conductivity compared with Cu and Al₂O₃. In contrast, for the Sakiadis problem, Al₂O₃ has the highest heat transfer rate. The thermal conductivity of Al₂O₃ is approximately one-tenth of Cu, as given in Table 1. However, has its own unique property which is low thermal diffusivity. Decrease in thermal diffusivity leads to higher temperature gradient and thus will increse an enhancement in heat transfers. However, Cu nanoparticle have high value of thermal diffusivity and therefore, this automatically will reduces the temperature gradient and will affect the performance of Cu-water working fluid. In summation, it is noted that the lowest heat transfer rate obtained to TiO₂ nanoparticles.

Further, Figures 3 - 6 presents the velocity and temperature for both Blasius and Sakiadis problems for some values of nanoparticle volume fraction ϕ parameter ($0 \leq \phi \leq 0.2$) when $Pr = 6.2$, $Bi = 0.1$ and Cu-water as working fluid. Figures 3 and 5 show that the momentum boundary layer increases with nanoparticle volume fraction ϕ for the Blasius problem, while decrease with nanoparticle volume fraction ϕ for the Sakiadis problem. However, Figures 4 and 6 show that the thermal boundary layer thickness increases with an increase in the parameter ϕ because of the increase in the local Nusselt number, as can be seen

from Figures 2. Therefore, nanofluids are capable to change the velocity and temperature profiles within the boundary layer.

However, for Figures 7 - 10 presents the velocity and temperature for both Blasius and Sakiadis problems for different values of slip parameter σ , ($\sigma = 0, 0.2, 0.4$) when $Pr = 6, 2, Bi = 0.1$, $\varphi = 0.1$ and Cu-water as working fluid. Figures 7 and 9 show that the momentum boundary layer increases with slip σ parameter for the Blasius problem but decreases for Sakiadis problem. It is same goes for the thermal boundary layer thickness in Figures 8 - 10 which is decreases for Blasius problem but increases for Sakiadis problems as the slip σ parameter increase. It can be seen that all these profiles are asymptotically satisfied all boundary conditions equations (10) - (12). Hence, the numerical results we obtained are valid.

Table 2: Values of $Re_x^{1/2} C_f$ for the Blasius problem

σ	φ	Ahmad et al. [5]			Present		
		Cu-water	Al ₂ O ₃ -water	TiO ₂ -water	Cu-water	Al ₂ O ₃ -water	TiO ₂ -water
0	0	0.3321	0.3321	0.3321	0.3321	0.3321	0.3321
	0.002	0.3355	0.3339	0.3340	0.3355	0.3339	0.3340
	0.004	0.3390	0.3357	0.3359	0.3390	0.3357	0.3359
	0.008	0.3459	0.3394	0.3398	0.3459	0.3394	0.3398
	0.01	0.3494	0.3412	0.3417	0.3494	0.3412	0.3417
	0.012	0.3528	0.3431	0.3436	0.3528	0.3431	0.3436
	0.014	0.3563	0.3449	0.3456	0.3563	0.3449	0.3456
	0.016	0.3597	0.3468	0.3476	0.3597	0.3468	0.3476
	0.018	0.3632	0.3487	0.3495	0.3632	0.3487	0.3495
	0.02	0.3667	0.3551	0.3515	0.3667	0.3551	0.3515
	0.1	0.5076	0.4316	0.4362	0.5076	0.4316	0.4362
	0.2	0.7066	0.5545	0.5642	0.7066	0.5545	0.5642
0.2	0				0.3298	0.3298	0.3298
	0.002				0.3332	0.3316	0.3317
	0.004				0.3367	0.3334	0.3336
	0.008				0.3435	0.3371	0.3375
	0.01				0.3469	0.3389	0.3394
	0.012				0.3503	0.3408	0.3413
	0.014				0.3537	0.3426	0.3432
	0.016				0.3571	0.3445	0.3452
	0.018				0.3605	0.3463	0.3471
	0.02				0.3639	0.3482	0.3491
	0.1				0.5030	0.4287	0.4332
	0.2				0.6996	0.5511	0.5606
0.4	0				0.3238	0.3238	0.3238
	0.002				0.3271	0.3256	0.3257
	0.004				0.3304	0.3274	0.3275
	0.008				0.3370	0.3309	0.3313
	0.01				0.3403	0.3327	0.3332
	0.012				0.3436	0.3345	0.3351
	0.014				0.3469	0.3364	0.3370
	0.016				0.3502	0.3382	0.3389
	0.018				0.3534	0.3400	0.3408

	0.02		0.3567	0.3418	0.3427
	0.1		0.4908	0.4209	0.4252
	0.2		0.6816	0.5419	0.5509

Table 3: Values of $-Re_x^{1/2}C_f$ for the Sakiadis problem

σ	φ	Ahmad et al. [5]			Present		
		Cu-water	Al ₂ O ₃ -water	TiO ₂ -water	Cu-water	Al ₂ O ₃ -water	TiO ₂ -water
0	0	0.4446	0.4446	0.4446	0.4438	0.4438	0.4438
	0.002	0.4492	0.4470	0.4471	0.4484	0.4454	0.4455
	0.004	0.4538	0.4494	0.4497	0.4530	0.4462	0.4463
	0.008	0.4630	0.4544	0.4548	0.4530	0.4486	0.4489
	0.01	0.4676	0.4568	0.4574	0.4623	0.4535	0.4540
	0.012	0.4722	0.4593	0.4600	0.4669	0.4560	0.4566
	0.014	0.4768	0.4618	0.4626	0.4715	0.4585	0.4592
	0.016	0.4814	0.4643	0.4653	0.4808	0.4635	0.4645
	0.018	0.4860	0.4668	0.4679	0.4854	0.4660	0.4671
	0.02	0.4906	0.4693	0.4705	0.4900	0.4685	0.4697
	0.1	0.6788	0.5778	0.5840	0.6784	0.5768	0.5830
	0.2	0.9446	0.7428	0.7556	0.9442	0.7410	0.7540
0.2	0				0.3925	0.3925	0.3925
	0.002				0.3964	0.3925	0.3948
	0.004				0.4003	0.3968	0.3970
	0.008				0.4080	0.4011	0.4015
	0.01				0.4118	0.4033	0.4038
	0.012				0.4156	0.4055	0.4060
	0.014				0.4195	0.4076	0.4083
	0.016				0.4233	0.4098	0.4106
	0.018				0.4272	0.4120	0.4129
	0.02				0.4310	0.4142	0.4152
	0.1				0.5885	0.5103	0.5151
	0.2				0.8152	0.6588	0.6690
0.4	0				0.3531	0.3531	0.3531
	0.002				0.3564	0.3550	0.3551
	0.004				0.3598	0.3570	0.3571
	0.008				0.3664	0.3608	0.3611
	0.01				0.3697	0.3627	0.3631
	0.012				0.3729	0.3647	0.3652
	0.014				0.3762	0.3666	0.3672
	0.016				0.3795	0.3686	0.3692
	0.018				0.3828	0.3706	0.3713
	0.02				0.3861	0.3725	0.3733
	0.1				0.5219	0.4591	0.4631
	0.2				0.7205	0.5949	0.6033

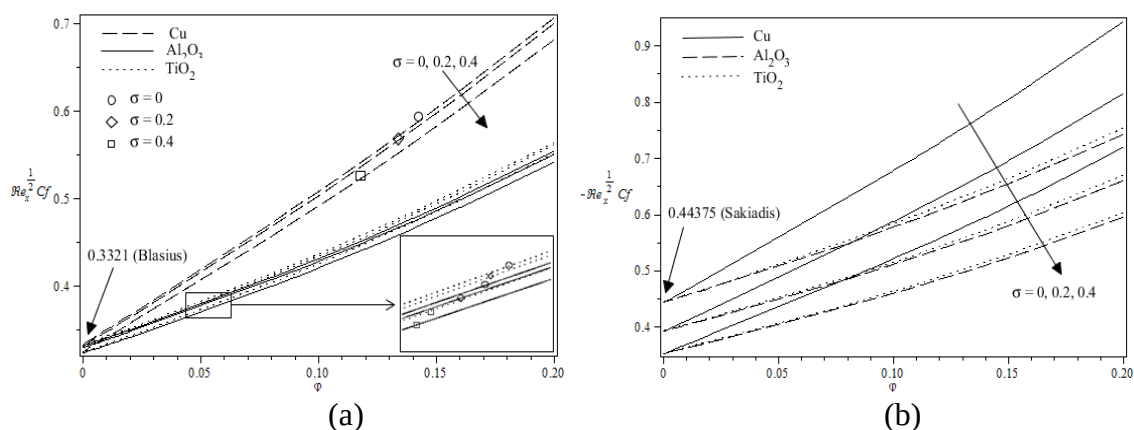


Fig. 1. Variation of the skin friction coefficient with ϕ for (a) Blasius problem and (b) Sakiadis problem

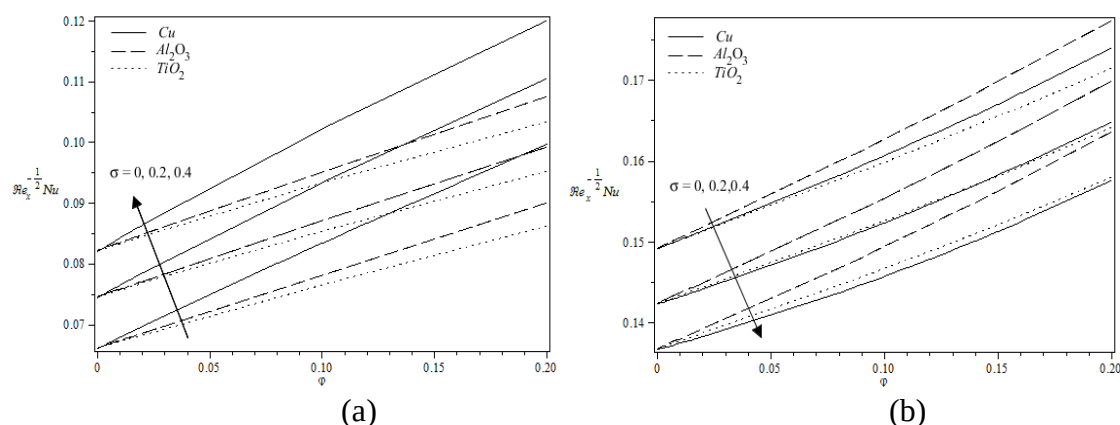


Fig. 2. Variation of the Nusselt number coefficient with ϕ for (a) Blasius problem and (b) Sakiadis problem

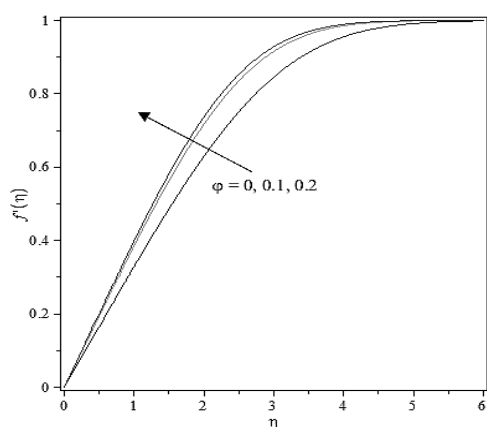


Fig. 3. Velocity profiles for various ϕ for the Blasius problem with Cu-water as working fluid with $\sigma = 0.2$ and $Pr = 6.2$.

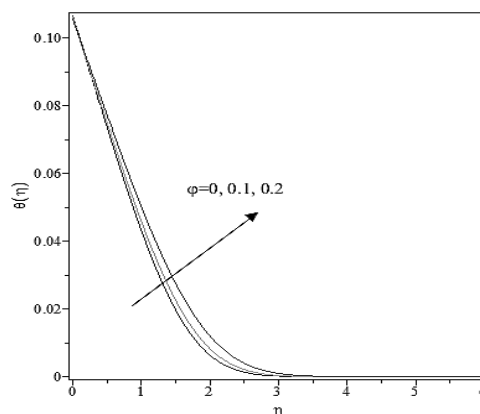


Fig. 4. Temperature profiles for various ϕ for the Blasius problem with Cu-water as working fluid with $\sigma = 0.2$, $Bi = 0.1$ and $Pr = 6.2$.

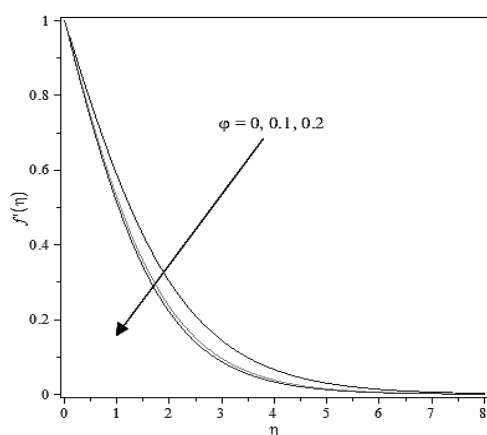


Fig. 5. Velocity profiles for various φ for the Sakiadis problem with Cu-water as working fluid with $\sigma = 0.2$ and $Pr = 6.2$.

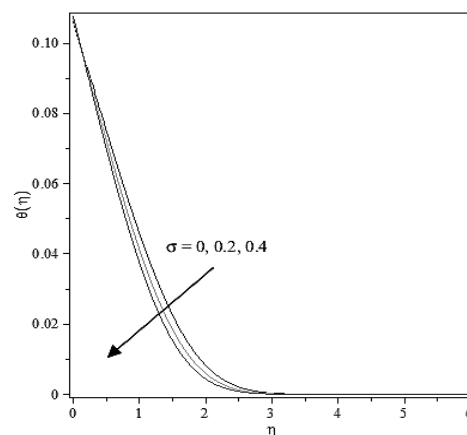


Fig.8. Temperature profiles for various σ for the Blasius problem with Cu-water as working fluid with $\varphi = 0.1$, $Bi = 0.1$ and $Pr = 6.2$.

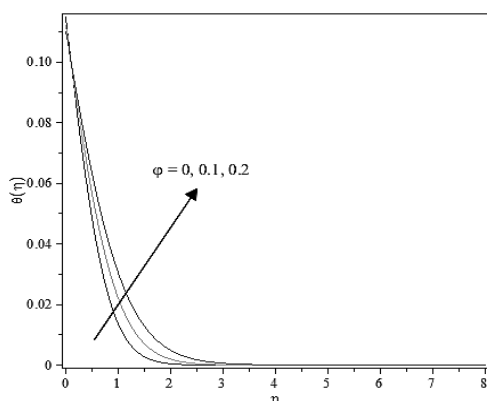


Fig. 6. Temperature profiles for various φ for the Sakiadis problem with Cu-water as working fluid with $\sigma = 0.2$, $Bi = 0.1$ and $Pr = 6.2$.

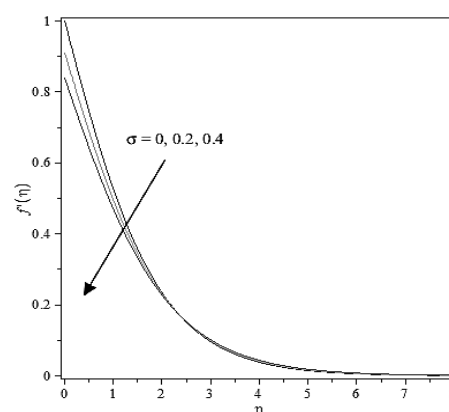


Fig. 9. Velocity profiles for various σ for the Sakiadis problem with Cu-water as working fluid with $\varphi = 0.1$ and $Pr = 6.2$.

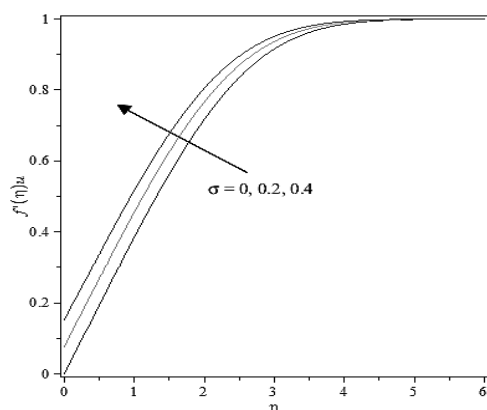


Fig.7. Velocity profiles for various σ for the Blasius problem with Cu-water as working fluid with $\varphi = 0.1$ and $Pr = 6.2$.

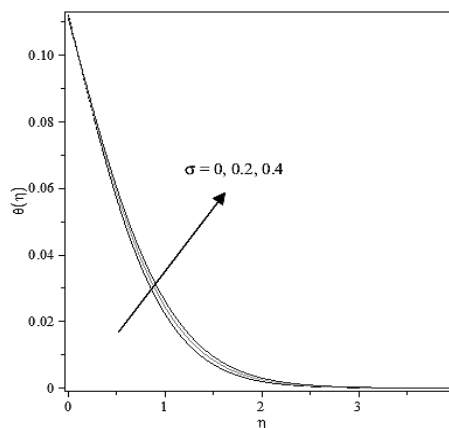


Fig. 10. Temperature profiles for various σ for the Sakiadis problem with Cu-water as working fluid with $\varphi = 0.1$, $Bi = 0.1$ and $Pr = 6.2$.

Conclusions

We have studied theoretically and analysed the effects of partial slip and thermal convective boundary condition on the flow and the heat transfer characteristics for both Blasius and Sakiadis problems in presence of nanofluids, and considered using the model proposed by Tiwari and Das [4]. The effects of nanoparticle volume fraction ϕ , slip parameter σ as well as Biot number on the skin friction coefficient and heat transfer rate at the surface were investigated and discussed. The problem was solved using a shooting method by Maple software. The result indicate that

- as the slip parameter σ increase, the skin friction will decrease for both Blasius and Sakiadis problems, while the Nusselt number increase for Blasius problem but decrease for Sakiadis problem.
- as the Biot number increase the thickness of thermal boundary layer also increase for both Blasius and Sakiadis problems.
- the presence of nanoparticles into the base water fluid has produced an increase in the skin friction and the heat transfer (Nusselt number) coefficients, which increases significantly with an increase in nanoparticle volume fraction. The addition of nanoparticles showed an improvement in the heat transfer rate.
- nanofluids are capable to change the velocity and temperature profiles in the boundary layer.
- Cu has the highest values for both skin friction coefficient and Nusselt number in the base fluid of water with the Prantl number $Pr = 6.2$ compared with TiO_2 and Al_2O_3 .

Acknowledgement

The authors gratefully acknowledge the financial support received in the form of a fundamental research grant scheme (FRGS/1/2018/STG06/UPM/02/4/5540155). Lastly, a humble yet sincere gratitude to all the reviewers for their honest feedback and suggestions.

References

- [1] Pantokratoras, A. "The Blasius and Sakiadis flow with variable fluid properties." *Heat Mass Transfer*, vol. 44 (2008): 1187–1198.
- [2] Ishak, A., Nazar, R., and Pop, I. "Flow and heat transfer characteristics on a moving flat plate in a parallel stream with constant surface heat flux." *Heat Mass Transfer*, vol. 45 (2009): 563–567.
- [3] Buongiorno, J. "Convective Transport in Nanofluids." *Journal of Heat Transfer*, vol. 128 (2006): 240–250.
- [4] Tiwari, R. and Das, M. "Heat Transfer Augmentation in a Two-Sided Lid-Driven Differentially Heated Square Cavity Utilizing Nanofluids." *International Journal of Heat and Mass Transfer*, vol. 50 (2007): 2002–2018.
- [5] Ahmad, S., Rohni, M. A., and Pop, I. "Blasius and Sakiadis Problems in Nanofluids." *Acta Mech*, vol. 218 (2010): 195–204.
- [6] Bachok, N., Ishak, A., and Pop, I. "Boundary layer flow over a moving surface in a nanofluid with suction or injection." *Acta Mech. Sin*, vol. 28, no. 1 (2012): 34–40.
- [7] Anuar, N. S. and Bachok, N. "Blasius and Sakiadis Problems in Nanofluids using Buongiorno Model and Thermophysical Properties of Nanofluids." *European International Journal of Science and Technology*, vol. 5 (2016): 65–81.
- [8] Noghrehabadi, A., Pourrajab, R., and Ghalambaz, M. "Flow and heat transfer of nanofluids over stretching sheet taking into account partial slip and thermal convective boundary conditions." *Heat Mass Transfer*, vol. 49 (2012): 1357–1366.

- [9] Dzulkifli, N. F., Bachok, N., Pop, I., Yacob, N. A., Arifin, N. M., and Rosali, H. "Stability of Partial slip, Soret and Dufour effects on unsteady boundary layer flow and heat transfer in Copper-water nanofluid over a stretching/shrinking sheet." *Journal of Physics: Conference Series*, vol. 890 (2017): 1–7.
- [10] Najib, N., Bachok, N., Arifin, N. M., Ali, F. M., and Pop, I. "Stability solutions on stagnation point flow in Cu-water nanofluid on stretching/shrinking cylinder with chemical reaction and slip effect." *Journal of Physics: Conference Series*, vol. 890 (2017): 12–30.
- [11] Bataller, R. "Radiation effects for the Blasius and Sakiadis flows with a convective surface boundary condition." *Applied Mathematics and Computation*, vol. 206 (2008): 832–840.
- [12] Ishak, A., Yacob, N. A., and Bachok, N. "Radiation effects on the thermal boundary layer flow over a moving plate with convective boundary condition." *Meccanica*, vol. 46 (2011): 795–801.
- [13] Oztop, H. and Abu-Nada, E. "Numerical study of natural convection in partially heated rectangular enclosures filled with nanofluids." *Int. J. Heat Fluid Flow*, vol. 29 (2008): 1326–1336.
- [14] Brinkman, H. C. "The viscosity of concentrated suspensions and solutions." *Journal of Chemical Physics*, vol. 20 (1952): 571–581.

CHARACTERIZATION OF U-BEND HEAT PIPE HEAT EXCHANGER TO APPLY AS DEHUMIDIFIER IN THE ELECTRONIC FACTORY

Imansyah Ibnu Hakim¹

Nandy Putra²

Dzikry Dalwatul Ielmi³

¹Heat Transfer Laboratory, Department of Mechanical Engineering Universitas Indonesia, Kampus UI, Depok 16424, Indonesia (Email: imansyah@eng.ui.ac.id)

²Heat Transfer Laboratory, Department of Mechanical Engineering Universitas Indonesia, Kampus UI, Depok 16424, Indonesia

³Heat Transfer Laboratory, Department of Mechanical Engineering Universitas Indonesia, Kampus UI, Depok 16424, Indonesia

Abstract: Building has high energy consumption and the highest energy demand comes from HVAC system. HVAC produces high amount of heat, most of them could not be used and considered as waste. Unfortunately, these heat wastes are currently only being used for cooling purposes with the presence of heat exchangers such as heat pipes. In actual, these heat pipes can give another benefit by controlling air moisture level in the implementation of HVAC system. This benefit can be very vital especially in the electronic industry where the components are very sensitive to humidity. When the air becomes too humid, the electronic component will fail due to electrostatic discharge. This experiment consists of some work path. First, it starts with designing heat pipe configuration from U-Bend Heat Pipe. Next step is by analyzing the performance of the heat pipe configuration. The output of the HVAC for electronic industry is specific for some fabrication which is about 30 °C and below 60 %RH. In different variable of intake speed and temperature, some of the testing configurations fulfill those standards. A configuration which uses eight finned U-Bend Heat Pipe in a row could fulfill the IPC/JEDEC J-STD-020D standards below 30 °C and 60 %RH for specific electronic part process. The best heat recovery process was obtained from air velocity of 1.0 m/s with 35 °C inlet temperatures, which equals to 175.20 W of heat transfer. Furthermore, the highest effectiveness was achieved at 26.09% with the air velocity of 0.5 m/s and 30 °C inlet temperature. Therefore, U-bend heat pipe can also be used as an alternative for dehumidification purposes and it meets the electronic industry needs.

Keywords: U-bend heat pipe, dehumidification, electronic industry

Introduction

Industry has a large energy consuming process. Industry also produce heat in that process where most of the heat becomes waste and recovering that heat waste is a challenge. Recovering heat from those processes could reduce the greenhouse emission and reduce production cost[1]. Energy with heat and electricity form is required in almost all kinds of industries. Almost 50% annual global energy consumption comes from building that uses HVAC system for 30% to 50% from their total energy demand[2]. Building with environmental comfort and concern are necessary for more energy efficient solutions, thus continual replacement of clean air is needed for conducive and comfortable internal condition. Higher air circulation means higher energy consumption. Conventional method for conditioning the air is using a chiller to get a low temperature with a cost of high humidity, then a heater is used to decrease the humidity[3].

Heat pipe is a passive cooling technology that is capable of conveying heat by evaporation and condensation process cycle. Heat pipe consists of three main parts. First, a tube shell that covers the inner component of a heat pipe; usually metal with high conductivity such as copper and aluminum are used for this part. Second, a wick that makes porosity inside the shell, which is used for fluid circulation inside the shell. The last part is fluid that has a specific working temperature, so the fluid can work properly. Heat pipe is divided to three sections. First is the evaporator where the fluid in this section evaporates and specifically takes heat for the process. Second section is adiabatic section that theoretically must be sealed through to keep the heat inside the heat pipe. The last section is condenser section that release its heat to the surrounding[4]. Heat pipe also has a limit due to working fluid, thermal resistance and capacity of its transfer[5]. Capillary pressure also has a limit, because of initial circulation of fluid due to capillarity of incompressible state of the fluid[6]. There are other kinds of limitation which also makes heat pipe not work such as entertainment limit, sonic limit and viscosity limit[7-8].

Heat that continues through the system can be measured by specific heat, mass flow and delta temperature[13]. Heat Pipe Performance could be measured from Effectiveness and Heat Recovery of System. Effectiveness means how the heat pipe system can actually perform compared to its maximum capability with temperature differences. Heat Recovery represents actual heat that is transferred through the system[14].

Most of heat pipes are used for cooling system, for example in the electronic device. Nowadays, electronic devices become more compact, so the cooling solution must be simple but reliable. Heat pipe that doesn't need initial power to work, makes most of electronics today use heat pipe for cooling solution that could carry 30 W of power[15]. Personal Computer also require improved thermal management to dissipate the heat for chipsets, because more performance means more heat produced. Even heat sink, which is connected to heat pipe from hot side of chipsets, also uses forced convections which are generated by a fan[16]. Electronic Manufacturing has limited condition of relative humidity, some storage, assembly and manufacturing process need specified temperature and humidity condition[9-10]. Electronic parts need to remain dry even from the moisture. Specific handling method and storage must be followed. Such conditions are needed to avoid electrostatic discharge, for example Non-Hermetic Solid State Surface Mount Devices needs an environment which is below 30 °C and 60 %RH[9-10]. Managing static electricity is an important issue in the process as well as in the electronics industry. Electrostatic Discharge is an electromagnetic pulse that occurs as an effect from electronic part failure, which could be caused by wet surface, bad grounding and even just humid air. Electronic related worker has to use Protective Personal Equipment[17]. Industry must have Electrostatic Protected Area to avoid unwanted Electrostatic Discharge[18-20]. Even if it is just an air gap inside the machinery, if the relative humidity is high the Electrostatic Discharge could occur[19].

Improvement of heat pipe could make the system work even in a little temperature difference of two heat pipe sides[21]. Also the use of nano-particles could improve heat pipe performances up to 40%[22-25]. The inclination of heat pipe also could burden fluid circulation; gravity could either support or inhibit fluid circulation[23]. Small gadgets use heat pipe for cooling solution. Due to compact design needs and excellent heat transfer performance. Heat pipe have been applied in several smart phone[24].

Methodology

In this experiment we follow several stages:

1. Design heat pipe configuration with available heat pipe.

2. Assemble heat pipe and another supporting parts, like ducting, chiller, air heater and blower.
3. Initial Test that ensures every component works.
4. Retrieve Data that would be used to analyze the performance of heat pipe.
5. Compare the temperature and relative humidity output with electronic industry standard.

Heat could be calculated by simple equation of heat transfer.

$$Q = m \cdot c \cdot \Delta T \quad (1)$$

where Q = Heat (Watt), m = mass Flow (kg), c = Specific Heat (J/kgK), ΔT = Temperature Differences (K)

The channel uses Polyurethane Ducting with cross section size of 250x160 mm. The heat pipe U has a size of 270x220 mm. The heat pipe diameter is 10 mm. The heat pipes are configured in one row, and the heat pipes are connected with fin for better heat dissipation area as seen in **Fig. 1**.

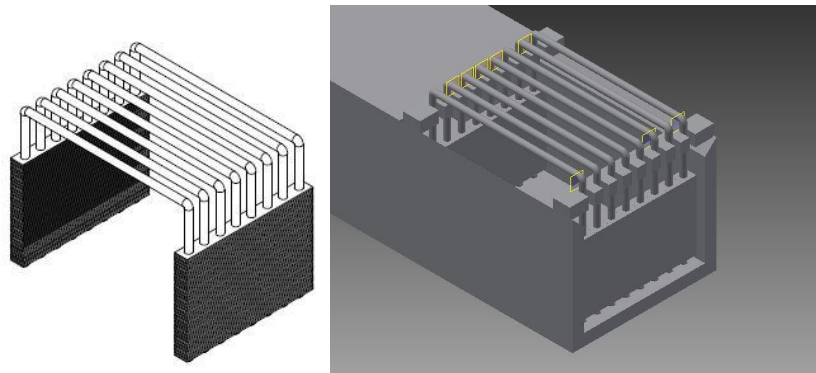


Fig. 1 Heat Pipe Configuration

Manufacture and assembly process are very important to meet the need of the experiment. First, constructing the ducting channel, then manufacturing the frame, last connecting electrical supply and control. Air heater and blower are installed in order to have air intake temperature variation and velocity variation respectively. **Fig. 2** shows the experimental set up.



Fig. 2 Experimental Set Up

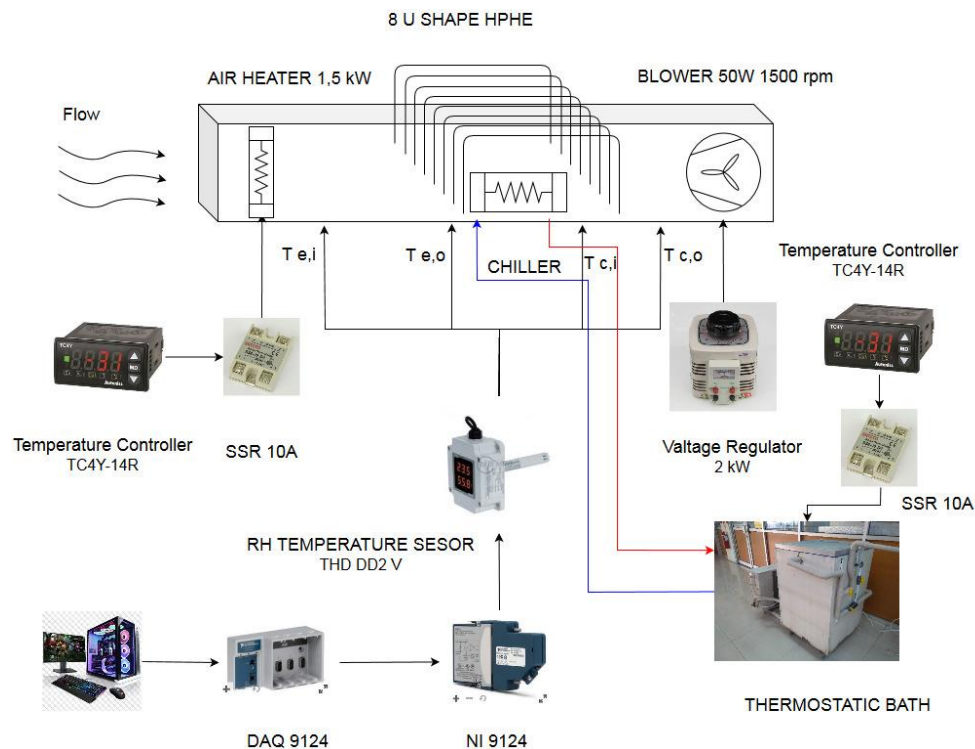


Fig. 3 Experimental Set Up Diagram

Experimental Component:

- Air Heater 1500 W with temperature control
- Blower 20 cm diameter with 1500 rpm
- Circulating Thermostatic Bath for cooled water cooling coil supply.
- DAQ NI and Personal Computer

Sensor:

- Type K thermocouple that measure $T_{e,i}$; $T_{e,o}$; $T_{c,i}$; $T_{c,o}$
- RH sensor Autonics THD that measure in the same point of temperature
- Hotwire Anemometer to measure air velocity
- Flow meter to measure water circulation speed

Dehumidifier System Testing

The experiment is done with different intake velocity and temperature variation. Intake air velocity varies from 0.5 m/s, 1.0 m/s and 1.5 m/s. Intake temperature varies from 30 °C, 35 °C, 40 °C and 45 °C. After that, output temperature and relative humidity is compared with the JEDEC J-STD-020 Standard [11]. Specific for non-hermetic solid-state devices, the standard environment must be below 30 °C and 60 %RH. Data acquisition happens when temperature and RH are steady for more than 20 minutes. Data taking is done in three repetitions to make sure of the characteristic of system and data error.

Results

Psychrometric Chart

Psychrometric chart describes actual process of air conditioning. What kind of heat that occurs in every process could be explained; it could be possible when we plot temperature and relative humidity condition before and after process.

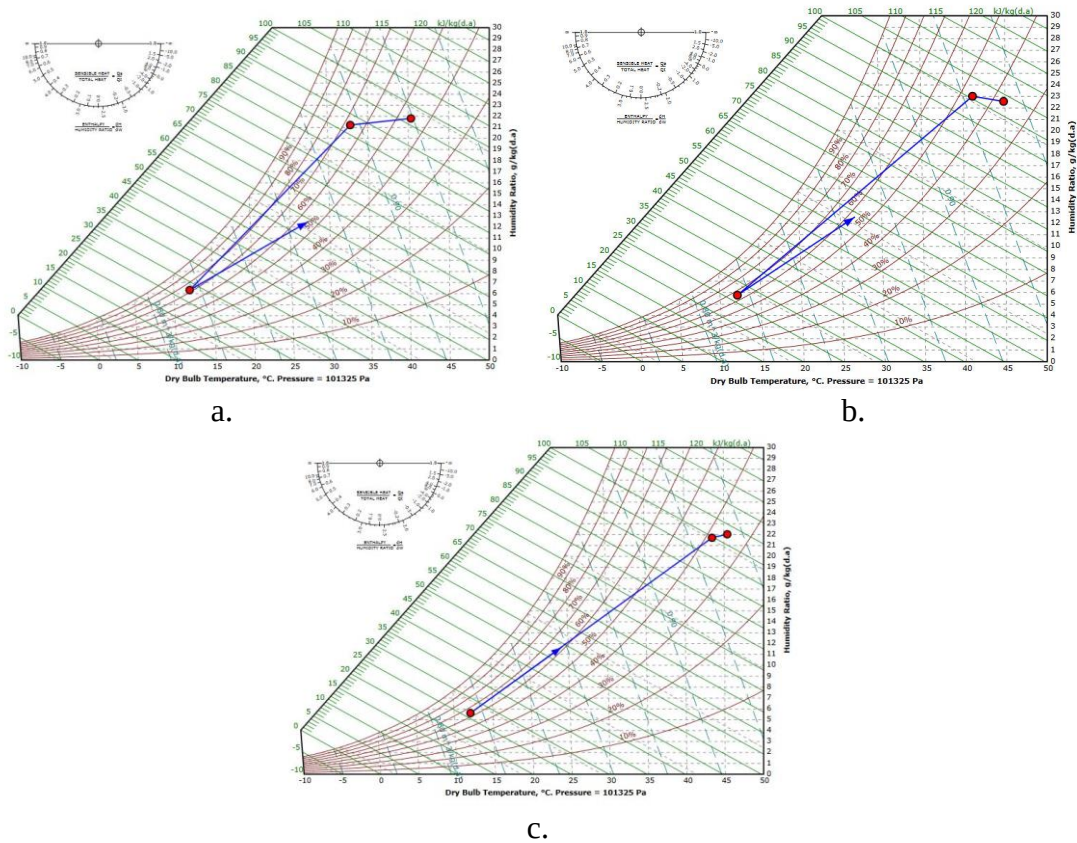


Fig. 4 Psychrometric chart in air conditioning process at velocity 0.5 m/s(a), 1.0 m/s(b), and 1.5 m/s(c)

In **Fig. 4**, it is described how air conditioning process occurs inside the system. The actual process that happened was close to the theoretical process. In theory, the process follows sensible cooling, saturation process with dehumidification and sensible reheating. But in the actual process, inclination happened. It happened because dehumidification that occurs before saturation. This Phenomenon occurs because air moisture which is normally dropped and becomes condensate instead taken by air, so humidity could increase or decrease. Every output condition like temperature and RH could meet the standard[11] for specific condition and process in electronic industry.

Average Temperature and Average Relative Humidity

Table 1 Air Average Temperature

v e,i m/s	T e,i °C	T e,i °C	T e,o °C	T c,i °C	T c,o °C	T amb °C
0.5	30	30.04	25.21	11.51	21.41	28.76
	35	34.95	28.93	11.87	24.32	28.55
	40	40.33	32.83	12.01	27.29	28.10
	45	45.21	36.60	12.55	30.01	28.19
1	30	30.26	26.33	10.63	19.94	27.72
	35	34.80	30.01	11.05	22.29	27.63
	40	40.35	35.79	11.71	25.00	27.64
	45	44.82	41.13	12.13	26.79	27.60

1.5	30	30.20	28.61	10.62	18.47	27.63
	35	35.41	33.25	11.41	20.31	27.63
	40	40.55	38.47	11.60	22.72	27.65
	45	45.43	43.51	12.10	24.21	28.72

Table 1 describes data tabulation of average temperature. Most of the output temperature condition T c,o, could meet the standard[11] specifically below 30 °C. Data was retrieved from three repetitions. After that, average temperature could be calculated. Even for some condition, it does not meet the standard.

Table 2 Average Relative Humidity

v e,i	T e,i	RH e,i	RH e,o	RH c,i	RH c,o	T amb
m/s	°C	%	%	%	%	°C
0.5	30	73.32	95.61	78.81	67.88	28.76
	35	57.13	80.19	75.60	61.14	28.55
	40	45.50	66.76	72.27	54.79	28.10
	45	37.15	55.96	67.85	49.20	28.19
1	30	69.81	85.36	74.61	70.31	27.72
	35	57.45	71.66	72.61	65.83	27.69
	40	45.19	56.43	68.85	60.49	27.64
	45	37.19	45.92	65.48	56.67	27.60
1.5	30	70.02	78.26	74.63	75.56	27.63
	35	52.03	58.59	71.62	68.95	27.63
	40	44.72	50.11	68.87	66.56	27.65
	45	35.22	38.34	63.97	61.60	28.72

Table 2 describes data tabulation of average relative humidity. Most of the output RH condition RH c,o, could meet the standard[11] specifically below 60% RH. Data was retrieved from three repetitions. After that, average temperature could be calculated. Even for some condition, it does not meet the standard.

Heat Recovery and Effectiveness

Performance of heat pipe configuration, heat recovery and effectiveness could be calculated from data. From equation 1 we could find heat recovery and effectiveness. Heat recovery is described as actual heat that happen in the system. Effectiveness is how the system performed compared to maximum heat that could be performed. The differences between maximum and actual come from the cold temperature, for maximum get the temperature from lowest temperature; after chiller. But for actual get the temperature from pre-cooling; after heat pipe evaporator.

$$e = \frac{Q_{act}}{Q_{max}} \quad (2)$$

where e is effectiveness, Q_{act} is Heat Recovery and Q_{max} is maximum heat that could perform In this tabulation some justification occurs. Specific heat at 30 °C - 45 °C are not too different, cp was assumed at 1008 J/Kg.K. and in mass flow we assumed in every velocity condition were same in every section.

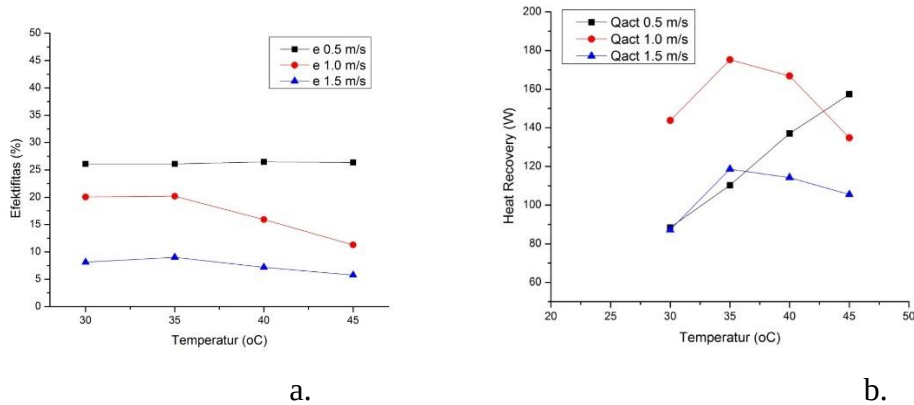


Fig 5 Effectiveness(a) and Heat Recovery(b) in every intake velocity variation

In **Fig. 5** section, the highest effectiveness occurred in highest temperature and lowest velocity. However, the profile is different with the condition where temperature is above 35 °C in 1.0 m/s and 1.5 m/s velocity. This could occur because characteristic of heat pipe configuration and also particular air properties. **Fig. 5** section b shows heat recovery that has different characteristic in 0.5 m/s compare to 1.0 m/s and 1.5 m/s intake velocity. This could occur because characteristic of heat pipe configuration and also particular air properties. Highest actual heat occurred in temperature 35 °C which is 179.20 Watt.

Table 3 Effectiveness and Heat Recovery

v e,i m/s	T e,i °C	Qact W	Effectiveness %
0.5	30	88.43	26.09
	35	110.23	26.10
	40	137.07	26.46
	45	157.39	26.35
1	30	143.83	20.02
	35	175.20	20.16
	40	166.83	15.92
	45	134.86	11.27
1.5	30	87.24	8.11
	35	118.63	9.00
	40	114.27	7.19
	45	105.51	5.76

Table 3 describes that highest effectiveness occurred at velocity 0.5 m/s and temperature 40 °C with 26.46 %. And lowest effectiveness occurred at velocity 1.5 m/s and temperature 45 °C. This happened because in high velocity air contact time with the heat pipe or fin is very short which makes heat absorption low and not effective. If the air contact time was enough, heat could be absorbed or dissipated more effectively. Effectiveness is also affected by temperature difference; the more temperature difference, the more heat could be passed.

Conclusions

Using heat pipe could conserve energy and save energy waste. Requirement for specific need in electrical industry [11], temperature must be set under 30 °C and relative humidity must be below 60% RH can be fulfilled by heat pipe configuration in HVAC system. Heat pipe can act for reheating or dehumidification process. Even for some condition could be above the air condition standard. Using heat pipe could decrease relative humidity and perform efficiency on energy usage. Some point of experiment:

- Heat pipe could be used to perform dehumidification. Furthermore, it could be improved for better efficiency
- Heat Recovery and effectiveness could describe the heat pipe performance. Heat recovery is actual heat, and effectiveness is comparing actual and maximum heat that could be performed. It depends on air velocity and air temperature.
- Best heat recovery occurs at velocity 1.0 m/s and temperature 35 °C with 175.20 Watt. And best effectiveness occurs at velocity 0.5 m/s and temperature 30 °C with 26.09 %.
- U-bend heat pipe can also be used as an alternative for dehumidification purposes and it meets the electronic industry needs.

Acknowledgement

The authors would like to thank Direktorat Riset dan Pengabdian Masyarakat(DRPM) Universitas Indonesia for funding this research project under the Hibah PITTA B Project, NKB-0738/UN2.R3.1/HKP.05.00/2019.

References

- [1] Sharma A., Jain S., Khausik S.C., Kakkar A. Potential of U-shaped Heat Pipe Heat Exchanger in Tropical Climates for Low Sensible Heat Ratio Applications. ASHRAE Transactions, Volume 123, Part 1. 2017.
- [2] Jouhara H., Almahmoud S., Chauhan A., Delpech B. Experimental investigation on a flat heat pipe heat exchanger for waste heat recovery in steel industry. International Conference on Sustainable Energy and Resource Use in Food Chains, ICSEF 2017.
- [3] Jouhara H., Economic Assessment of The Benefits of Wraparound Heat Pipes in Ventilation Processes for Hot and Humid Climates. International Journal of Low-Carbon Technologies, 2009.
- [4] Mahdavi M., Tiari S., Schamphleire S. D., Qiu S., Experimental Study of The /thermal Characteristic of A Heat Pipe. Experimental Thermal and Fluid Science 93, 2018.
- [5] Brouwers J.J.H., Geld V. D., Kamenade H.P., Zeegers J. C. H. Towards an Optimal Design of Heat Pipe Equipped Heat Exchangers. Klima. 2007.
- [6] Mozumder A. K., Akon A. F. Chowdhury M. S. H., BANIK S. C., Performance of Heat Pipe for Different Working Fluids and Fill Ratios. Journal of Mechanical Engineering, Vol ME 41, 2010.
- [7] Putra N., Septiadi N., Teknologi Pipa Kalor Teori, Desain dan Aplikasi. Jakarta. UI-Press
- [8] D. Reay and P. Kew. 2006. Heat Pipe Theory, Design and Applications. Oxford. Butterworth-Heinemann
- [9] Condair. Hummidification to Combat Static in Electronics Manufacturing
- [10] Condair. Eliminating Static with Humidification and Humidification Control
- [11] IPC/JEDEC J-STD-020D. Moisture/Reflow Sensitivity Classification for Non-hermetic Solid State Surface Mount Devices. Syfer Surface Mount Capacitor Test Result.
- [12] JESD625-A. Requirement fir Handling Electrostatic-Discharge-Sensitive Devices(ESDS).

- [13] J.P. Holman. Heat Transfer, Eight, SI Metric Edition col 10. USA. McGraw-Hill Inc. 2001.
- [14] Putra N. S. D., T. Anggoro, A. Winarta. Experimental Study of Heat Pipe Heat Exchanger in Hospital HVAC System for Energy Conservation. International Journal on Advance Science, Engineering and Information Technology. 2017.
- [15] Asirvatham G. L., Nimmagadda R., Wongwises S., Heat Transfer Performance of Screen Mesh Wick Heat Pipes using Silver-water Nanofluid. International Jurnal of Heat and Mass Transfer 60. 2013.
- [16] Wang Jung-Chang. U- and L-shaped Heat Pipes Heat Sinks for Cooling Electronic Components Employed a Least Square Smoothing Method. Microelectronics Reliability 54. 2014.
- [17] Paasi J., Nurmi S., Vuorinen R., Strengell S., Maijala P. Performance of ESD Protective Materials at Low Relative Humidity. Journal of Electrostatics 51-52. 2001.
- [18] Tamminen P., Viheriakoski T., Sydanheimo L., Ukkonen L., ESD Qualification Data Used as The Basis for Building Electrostatic Discharge Protected Areas. Journal of Electrtostatics 77. 2015.
- [19] Wang X., Taylor N., Edin H., Effect of Humidity on Partial Discharge in a Metal-dielectric Air Gap on Machine Insulation at Traoezoidal Testing Voltages. Journal ofElectristatics 83. 2016.
- [20] Takai T., Kaneko M., Honda M., One of The Methods of Observing ESD Around Electronic Equipment. Journal of Electrostatics 42. 1998.
- [21] Kempers R., Ewing D., Ching C.Y., Effect of Number of Mesh Layers and Fluid Loading on The Performance of Screen Mesh Wicked Hat Pipe. Applied Thermal Engineering 26. 2006.
- [22] Solomon D. A., Ramachandran K., Pillai B.C., Thermal Performance of a Heat Pipe with Nanoparticles Coated Wick. Applied Thermal Engineering 36. 2012.
- [23] Tharayil T., Asirvatham L. G., Cassie C. F. M., Wongwises S., Performance of Cylindrical and Flattened Heat Pipes at Various Inclination Including Repeatability in Anti-gravity – A Comparative Study. Applied Thermal Engineering 122. 2017.
- [24] Zhou W., Xie P., Li Y., Yan Y., Li B., Thermal Performance of Ultra-thin Flattened Heat Pipes. Applied Thermal Engineering 117. 2017.
- [25] Shentilkumar R., Vaidyanathan S., Sivaraman B., Effect of Inclination Angle in Heat Pipe Performance Using Copper Nanofluid. Procedia Engineering 38. 2012.

AVERAGE FRICTION FACTOR BETWEEN INLET AND OUTLET FOR LAMINAR FLOW OF GAS IN MICROCHANNELS

Chungpyo Hong¹
Yutaka Asako²
Mohammad Faghri³
Gian Luca Morini⁴

¹Department of Mechanical Engineering, Kagoshima University, Kagoshima 890-8580, Japan (Email: hong@mech.kagoshima-u.ac.jp)

²Department of Mechanical Precision Engineering, Malaysia-Japan International Institute of Technology, Universiti Teknologi Malaysia, Kuala Lumpur, 54100 Kuala Lumpur, Malaysia

³Department of Mechanical, Industrial and System Engineering, University of Rhode Island Kingston, RI 02881, USA

⁴Department of Industrial Engineering, University of Bologna, Via del Lazzaretto 15/5, 40131 Bologna, Italy

Abstract: *The average friction factor between inlet and outlet will help piping design where the estimation of pressure loss between relatively distant positions is done using average friction factor. Therefore, this paper presents $(f_d \cdot Re)_{ave}$, the product of average Darcy friction factor and Reynolds number between the inlet and the outlet for microchannel laminar gas flows obtained numerically for circular microtubes with 50, 100, 150 μm by varying the aspect ratio (i.e. length/diameter) from 100 to 400. The numerical methodology was based on the Arbitrary-Lagrangian-Eulerian (ALE) method. Axis-symmetric compressible momentum and energy equations of an ideal gas were solved for a wide range of Reynolds and Mach numbers with adiabatic wall boundary condition. The stagnation pressure was chosen in such a way that the outlet Mach number ranged from 0.1 to 1.0. The outlet pressure was fixed at the atmospheric condition. As a result of that, the average Darcy and Fanning friction factors between the inlet and outlet were obtained and compared with empirical correlations on Moody chart. The numerical results are in excellent agreement with the experimental ones. The $(f_d \cdot Re)_{ave}$ and $(f_f \cdot Re)_{ave}$ were also obtained and represented as a function of average Mach numbers with comparison with local $f \cdot Re$ correlations proposed in previous study.*

Keywords: *Average friction factor, Mach number, Gas flow, Microtube*

Introduction

Development of the MEMS technology has increased the need for an understanding of fluid flow in micro flow devices such as micro-valves, micro-pumps, micro-reactor and many other micro fluid systems. A microchannel passage is a basic and important element to design the micro flow devices and for this reason during the last decade a series of investigations have been made with the aim to clarify the main scale effects playing a role in microchannels. Since the early work by Wu and Little [1], numerous experimental and numerical investigations on microchannel gas flow have been undertaken and the literature is surveyed in the article by Kawashima and Asako [2].

The pressure loss determined by a friction factor between two point is one of significant factors to design microchannel lines. There are two friction factors based on the pressure loss and friction. The former one is called as Darcy friction factor and later is called as Fanning friction factor. The both friction factors for gases with large variations in the physical

properties flowing through channels were obtained under the assumption of isothermal flow by most of the researchers [1, 3~7] due to the measurement limitation of gas temperature flowing through a channel. Fortunately for gas flow through a micro-channel with thermally insulated wall (adiabatic wall), the estimation of gas static temperature at the micro-channel outlet can be done using a quadratic equation proposed by Kawashima and Asako [8]. A new data reduction methodology for the average Fanning friction factor calculation between the inlet and outlet considering the effect of a decrease in gas temperature has been developed by Hong et al. [9]. Rehman et al. [10] experimentally and numerically investigated the average friction factor along adiabatic microchannels with compressible gas flows including choking flow regime.

Attention will now be focused on $f \cdot Re$, the product of a friction factor and the Reynolds number for microchannel gas flow. Asako et al. [11 and 12] and Hong et al. [13–16] conducted numerical investigations on flow characteristics of gas flow in microchannels with both no-slip and slip boundary conditions. They proposed local $f \cdot Re$ correlations for both friction factors as a function of Mach number and Knudsen number corresponding to compressibility and rarefaction effect locally obtained for the quasi-fully developed region after the entrance region of microchannels within laminar flow regime. They also reported how to predict pressure variations and mass flow rate for microchannel gas flow with using the $f \cdot Re$ correlations. Their results are in excellent agreement with the experimental data [7 and 17]. There seems to be no study on average Darcy friction factor as a function of average Mach number between the inlet and outlet for adiabatic microchannels. This is the motivation of the present numerical study.

Formulation

Description of the problem

The schematic diagram of gas flow in a microchannel for numerical calculation is shown in Figure 1. The numerical calculations were performed for circular microtubes as microchannels. The numerical methodology to solve the governing equations is based on the Arbitrary-Lagrangian-Eulerian (ALE) method. A detail description of the numerical computation is well documented in the previous work [11] and will not be repeated here. Only the brief description is reported here. The flow is assumed to be steady, axisymmetric, laminar and the fluid is assumed to be an ideal gas. Compressible gas in a reservoir at the stagnation pressure, p_{stg} and the stagnation temperature, T_{stg} passes through an adiabatic microtube and discharges into the atmosphere ($p_{out}=10^5$ Pa) under an increasing inlet pressure. The physical properties assumed to be constant. Inlet values of velocity, density and temperature are obtained with isentropic process between the inlet and the stagnation area. It is also assumed that the velocity, pressure, temperature and density profiles at the inlet are uniform. The thermal boundary condition on the wall is adiabatic.

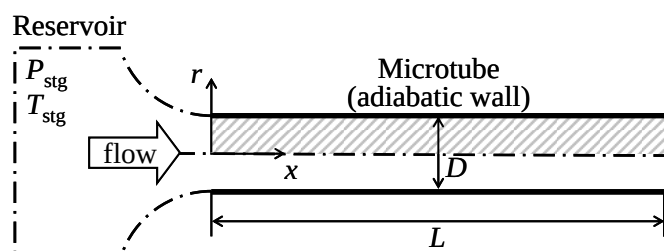


Fig. 1. Schematic diagram of a problem

Dimensionless variables

The Reynolds number and Mach number are defined as

$$Re = \frac{\bar{u} D}{\mu / \bar{\rho}} \quad (1)$$

$$Ma = \frac{\bar{u}}{\sqrt{\gamma(\gamma-1)\bar{i}}} = \frac{\bar{u}}{\sqrt{\gamma R \bar{T}}} \quad (2)$$

where D is a tube diameter. $\bar{u}$, $\bar{\rho}$ and $\bar{i}$ are the average velocity, density and specific internal energy at a cross-section as

$$\bar{u} = \frac{1}{A} \int u \, dA, \quad \bar{\rho} = \int \rho u \, dA / \int u \, dA, \quad \bar{p} = \frac{1}{A} \int p \, dA, \quad \bar{i} = \frac{1}{\gamma-1} \frac{\bar{p}}{\bar{\rho}}, \quad \bar{T} = \frac{\bar{p}}{R \bar{\rho}} \quad (3)$$

The average Mach number between the inlet and the outlet can be defined as

$$Ma_{ave} = \frac{\bar{u}_{ave}}{\sqrt{\gamma R \bar{T}_{ave}}} \quad (4)$$

where the average velocity, $\bar{u}_{ave}$ and the average temperature, $\bar{T}_{ave}$ can be obtained by integrating between the inlet (x_1) and the outlet (x_2) as

$$\bar{u}_{ave} = \frac{1}{x_2 - x_1} \int_{x_1}^{x_2} \bar{u} \, dx, \quad \bar{T}_{ave} = \frac{1}{x_2 - x_1} \int_{x_1}^{x_2} \bar{T} \, dx \quad (5)$$

In the previous study [9], a new data reduction methodology for the average Fanning friction factor calculation between the inlet and outlet through adiabatic microtubes has been developed. The developed average friction factor was employed in the present study. And the average Fanning and Darcy friction factors between the inlet (x_1) and the outlet (x_2) are

$$f_{f,ave} = \frac{1}{x_2 - x_1} \int_{x_1}^{x_2} f_f \, dx = \frac{D}{x_2 - x_1} \left\{ \int_{x_1}^{x_2} \left(\frac{2}{\bar{\rho}} \, d\bar{p} \right) - \int_{x_1}^{x_2} \left(\frac{2\bar{p}}{\bar{\rho}^2 \bar{u}^2 R \bar{T}} \, d\bar{p} \right) - \int_{x_1}^{x_2} \left(\frac{2}{\bar{T}} \, d\bar{T} \right) \right\} \\ = \frac{D}{x_2 - x_1} \left[-2 \ln \frac{\bar{p}_{in}}{\bar{p}_{out}} + 2 \ln \frac{\bar{T}_{in}}{\bar{T}_{out}} - \frac{1}{\left(\bar{\rho}_{in}^2 \bar{u}_{in}^2 R \times \left(\bar{T}_{in} + \frac{\bar{u}_{in}^2}{2c_p} \right) \right)} \right. \\ \left. \times \left\{ \frac{\bar{p}_{out}^2 - \bar{p}_{in}^2}{2} + \frac{B^2}{2} \ln \frac{\bar{p}_{out} + \sqrt{\bar{p}_{out}^2 + B^2}}{\bar{p}_{in} + \sqrt{\bar{p}_{in}^2 + B^2}} + \frac{1}{2} \left(\bar{p}_{out} \sqrt{\bar{p}_{out}^2 + B^2} - \bar{p}_{in} \sqrt{\bar{p}_{in}^2 + B^2} \right) \right\} \right] \quad (6)$$

$$f_{d,ave} = \frac{1}{x_2 - x_1} \int_{x_1}^{x_2} f_d \, dx = \frac{D}{x_2 - x_1} \left\{ - \int_{x_1}^{x_2} \left(\frac{2\bar{p}}{\bar{\rho}^2 \bar{u}^2 R \bar{T}} \, d\bar{p} \right) \right\} \\ = \frac{D}{x_2 - x_1} \left[- \frac{1}{\bar{\rho}_{in}^2 \bar{u}_{in}^2 R \times \left(\bar{T}_{in} + \frac{\bar{u}_{in}^2}{2c_p} \right)} \right. \\ \left. \times \left\{ \frac{\bar{p}_{out}^2 - \bar{p}_{in}^2}{2} + \frac{B^2}{2} \ln \frac{\bar{p}_{out} + \sqrt{\bar{p}_{out}^2 + B^2}}{\bar{p}_{in} + \sqrt{\bar{p}_{in}^2 + B^2}} + \frac{1}{2} \left(\bar{p}_{out} \sqrt{\bar{p}_{out}^2 + B^2} - \bar{p}_{in} \sqrt{\bar{p}_{in}^2 + B^2} \right) \right\} \right] \quad (7)$$

where

$$B^2 = 4 \times \alpha \frac{\bar{\rho}_{in}^2 \bar{u}_{in}^2 R^2}{2c_p} \times \left(\bar{T}_{in} + \frac{\bar{u}_{in}^2}{2c_p} \right) \quad (8)$$

Result and discussion

Numerical computations were conducted for microtubes of $D=50$, 100 and 150 μm whose aspect ratio is 100, 200 and 400. The tube diameter, the tube length, the stagnation pressure and corresponding Reynolds number and Mach number are listed in Table 1.

Table 1: Tube diameter, length, p_{stg} and corresponding Re and Ma

$D(\mu\text{m})$	$L(\text{mm})$	$p_{\text{stg}} (\text{kPa})$	Re	Ma_{in}	Ma_{out}
50	5	150 - 500	159 - 1514	0.096 - 0.287	0.141 - 1.105
	10	150 - 600	84 - 1454	0.051 - 0.225	0.075 - 1.072
	15	200 - 700	101 - 1245	0.046 - 0.163	0.090 - 0.95
100	10	150 - 375	537 - 2595	0.164 - 0.333	0.236 - 0.981
	20	150 - 450	316 - 2609	0.096 - 0.273	0.140 - 0.986
	40	150 - 500	168 - 2195	0.051 - 0.203	0.075 - 0.859
150	15	150 - 250	1019 - 2550	0.21 - 0.327	0.297 - 0.693
	30	150 - 400	657 - 2578	0.133 - 0.27	0.194 - 0.7
	60	150 - 400	368 - 2342	0.074 - 0.206	0.109 - 0.644

Mach number

The Mach numbers obtained for microtubes with the different diameter and almost the same Reynolds number were plotted in Fig. 2 as a function of x/L . The pressure distributions of them are also plotted in Fig. 3. The Mach number of the smaller tube is higher than that of the larger tube and steeply goes up near the outlet since the gas in the smaller tube expands even more near the outlet as shown in Fig. 3. And the lines of the Mach number and pressure are not straight near the outlet. This phenomenon is dominant for smaller diameter microtubes as one of compressibility effects. Therefore, as mentioned above, average Mach number between the inlet and outlet was obtained with integration from the inlet and outlet. And, for $D=100 \mu\text{m}$, both integral and arithmetic average Mach number between the inlet and outlet are plotted in Fig. 4 as a function of Reynolds number. The inlet Mach number, integral average Mach number and arithmetic average Mach number is higher for the shorter length. However, the outlet Mach number is independent of micro-tube length and it can be represented as a function of Reynolds number and diameter. Near future, a correlation among the outlet Mach number, Reynolds number and diameter will be studied, $Ma_{\text{out}}=f(Re, D)$. The values of integral average Mach numbers is lower than those of arithmetic ones since flow acceleration due to gas expansion occurs near the outlet. Qualitatively similar results for the microtube of $D=50$ and $150 \mu\text{m}$ are also obtained.

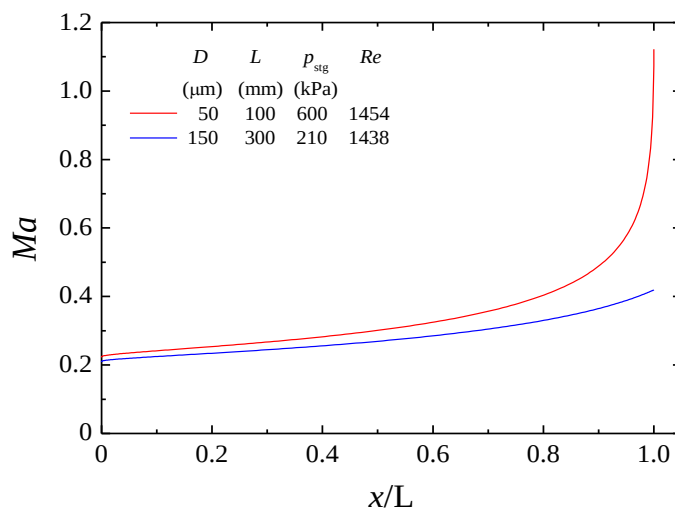


Fig. 2. Mach number as a function of x/L

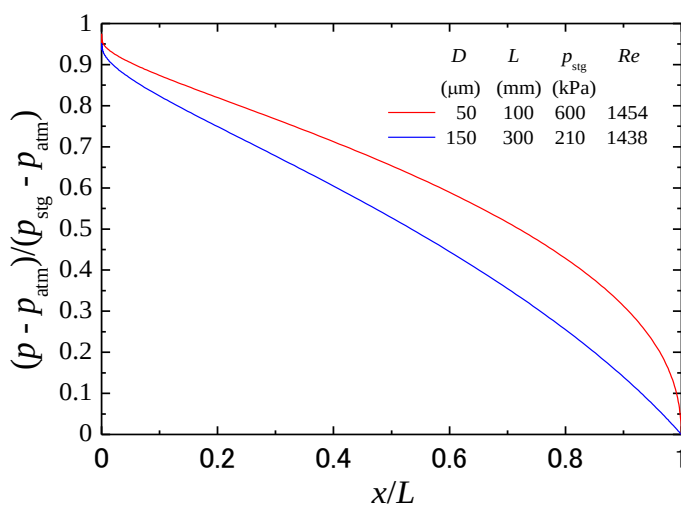


Fig. 3. Pressure distributions as a function of x/L

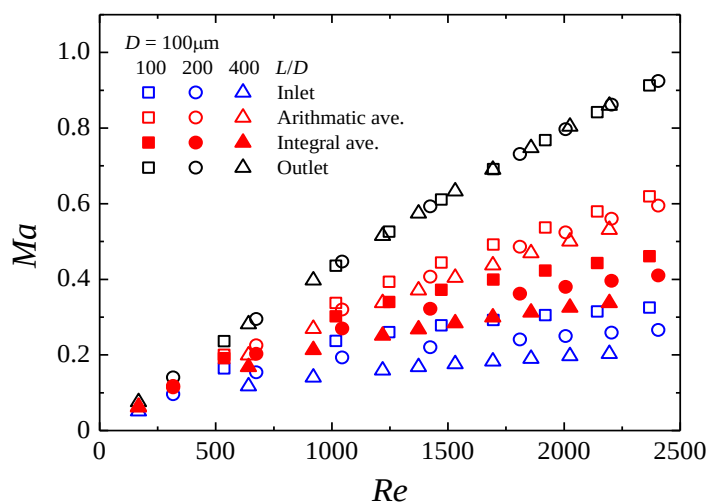


Fig. 4. Average Mach number as a function of Reynolds number

The average Darcy and Fanning friction factor between the inlet and outlet, $f_{d,ave}$ and $f_{f,ave}$ obtained for all tubes are plotted in Fig. 5 as function of Reynolds number with white and solid symbols, respectively. The solid line and the dotted line in the figure represent obtained by the theoretical formula ($f=64/Re$) and $f=0.3164/Re^{0.25}$ (Blasius equation) for the incompressible flow theory, respectively.

In the case of slow flow ($Re \leq 300$ and $Ma_{out} \leq 0.2$ as shown in Fig. 4), the both values of $f_{d,ave}$ and $f_{f,ave}$ are almost identical to the value of the incompressible theory. On the other hand, in the case of fast flow ($Re > 300$ and $Ma_{out} > 0.2$ as shown in Fig. 4) both values of $f_{d,ave}$ and $f_{f,ave}$ deviate more and more from those of incompressible theory ($f=64/Re$) with an increase in Re because of the entrance region and compressibility effects. The difference between $f_{d,ave}$ and $f_{f,ave}$ is larger than that of $Re < 300$ since flow accelerates with increase in Ma_{ave} and Re .

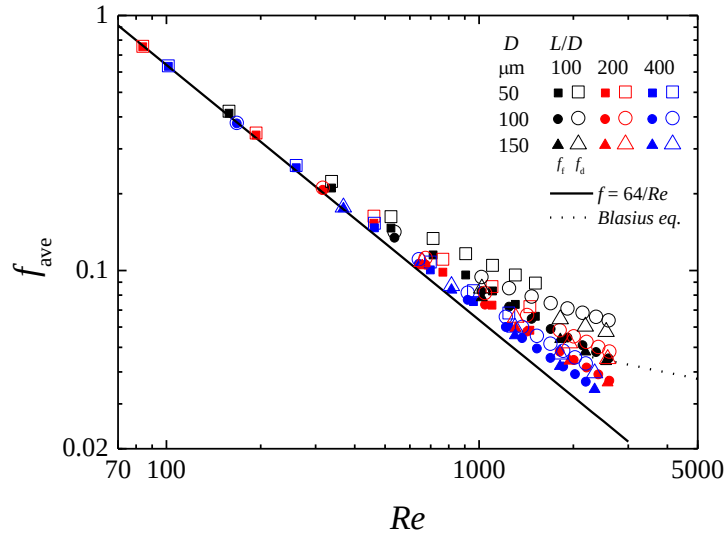


Fig. 5 Friction factors as function of Re

The products of average Darcy & Fanning friction factors and Reynolds number between the inlet and the outlet, $(f_d \cdot Re)_{ave}$ and $(f_f \cdot Re)_{ave}$ for all tubes were obtained. The both values of $(f_d \cdot Re)_{ave}$ and $(f_f \cdot Re)_{ave}$ are plotted in Figs. 6 and 7 as a function of the average Mach number, respectively. In order to estimate the loss of entrance region, we employed local $f_d \cdot Re$ and $f_f \cdot Re$ correlations proposed by the previous study [12]. In the previous study [12], the local $f_d \cdot Re$ and $f_f \cdot Re$ in quasi-fully developed region after the entrance region for a microtube as a quadratic function of Mach number were numerically obtained and their correlations were proposed as:

$$f_d \cdot Re = 64 - 11.99 Ma + 263.7 Ma^2 \quad (9)$$

$$f_f \cdot Re = 64 + 2.703 Ma + 93.89 Ma^2 \quad (10)$$

The proposed local $f_d \cdot Re$ and $f_f \cdot Re$ are also plotted in the figures with solid lines, respectively. Both values of $(f_d \cdot Re)_{ave}$ and $(f_f \cdot Re)_{ave}$ increase with an increase in the average Mach number due to the compressibility as local $f_d \cdot Re$ and $f_f \cdot Re$ correlations increase with an increase in the local Mach number. The difference between $(f_d \cdot Re)_{ave}$ and local $f_d \cdot Re$ can be considered as the loss of the entrance region since $(f_d \cdot Re)_{ave}$ has both losses of flow acceleration and entrance region.

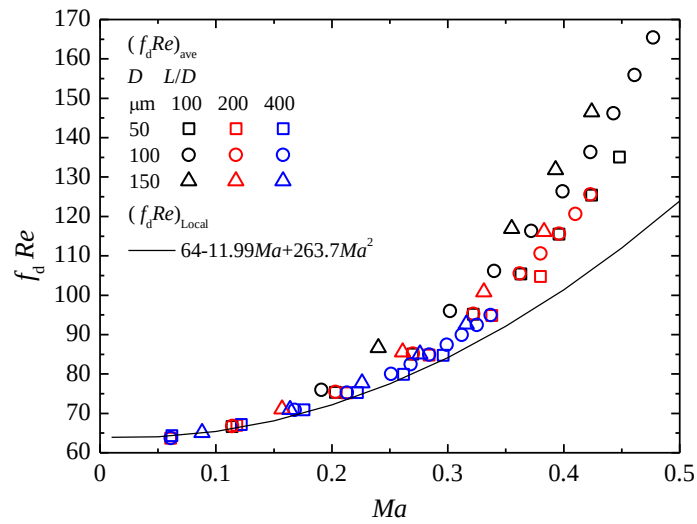


Fig. 6 $f_d \cdot Re$ as function of Ma

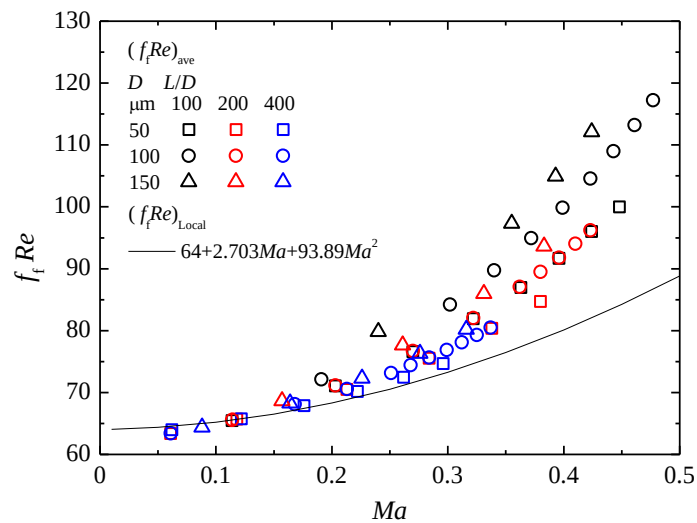


Fig. 7 $f_f \cdot Re$ as function of Ma

The difference between $(f_f \cdot Re)_{ave}$ and local $f_f \cdot Re$ can be also considered as the loss of the entrance region since $(f_f \cdot Re)_{ave}$ has both losses of friction on the wall and entrance region.

Conclusion

The numerical calculations were performed to obtain average Darcy and Fanning friction factors for circular microtubes with 50, 100, 150 μm by varying the aspect ratio (i.e. length/diameter) from 100 to 400. The following conclusions were reached.

1. The values of integral average Mach numbers is lower than those of arithmetic ones since flow acceleration due to gas expansion near the outlet.
2. The differences between $(f_d \cdot Re)_{ave}$ & $(f_f \cdot Re)_{ave}$ and local $f_d \cdot Re$ & $f_f \cdot Re$ are due to the loss of the entrance region.

References

- [1] Wu, P., and Little W. A., "Measurement of friction factors for the flow of gases in very fine channels used for micro miniature Joule-Thompson refrigerators." *Cryogenics* 23 (1983): 273–277.
- [2] Kawashima, D., and Y. Asako. "Measurement of quasi-local friction factor of gas flow in a micro-tube." *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science* 230, no. 5 (2016): 782– 792.
- [3] Yang, C., Chen, C., Lin, T., and Kandlikar, S. G., "Heat transfer and friction characteristics of air flow in microtubes." *Exp. Therm. Fluid Sci.* 37 (2012): 12–18.
- [4] Lorenzini, M., Morini, G. L., and Salvigni, S., "Laminar, transitional and turbulent friction factor for gas flows in smooth and rough microtubes." *Int. J. Therm. Sci.* 49 (2010): 248–255.
- [5] Turner, S. E., Lam, L. C., Faghri, M., and Gregory, O. T., "Experimental investigation of gas flow in microchannel." *J. Heat Transfer* 127 (2004): 753–763.
- [6] Tang, G. H., Li, Z., He, Y. L., and Tao, W. Q., "Experiment study of compressibility, roughness and rarefaction influences on microchannel flow." *Int. J. Heat Mass Transfer* 50 (2007): 2282–2295.
- [7] Hong, C., Yamada, T., Asako, Y., and Faghri, M., "Experimental investigations of laminar, transitional and turbulent gas flow in micro-channels. " *Int. J. Heat Mass Transfer* 55 (2012): 4397–4403.
- [8] Kawashima, D., and Asako, Y., "Data reduction of friction factor of compressible flow in micro-channels." *Int. J. Heat Mass Transfer* 77 (2014): 257–261.
- [9] Hong, C., Asako, Y., Morini, G. L., and Rehman, D., "Data reduction of average friction factor of gas flow through adiabatic micro-channels." *Int. J. Heat Mass Transfer* 129 (2019): 427–431.
- [10] Rehman, D., Morini, G. L., and Hong, C., "A Comparison of Data Reduction Methods for Average Friction Factor Calculation of Adiabatic Gas Flows in Microchannels." *Micromachines* 10 (2) (2019): p171.
- [11] Asako, Y., Pi, T., Turner, S. E., and Faghri, M., " Effect of compressibility on gaseous flows in micro-channels." *Int. J. Heat Mass Transfer* 46 (2003): 3041–3050.
- [12] Asako, Y., Nakayama, K., and Shinozaki, T., "Effect of compressibility on gaseous flows in a micro-tube." *Int. J. Heat Mass Transfer* 48 (2005): 4985–4994.
- [13] Hong, C., Asako, Y., Faghri, M., and Lee, J-H., "Poiseuille Number Correlations for Gas Slip Flow in Micro-Tubes." *Numer. Heat Transfer A.* 56 (2009): 785–806.
- [14] Hong, C. Asako, Y., Turner, S. E., and Faghri, M., "Friction Factor Correlations for Gas Flow in Slip Flow Regime." *J. of Fluids Eng.* 129 (2007): 1268–1276.
- [15] Hong, C., Asako, Y., and Lee, J-H., "Poiseuill Number Correlation for High Speed Micro-Flows." *J. Phys. D: Appl. Phys.* 41 (2008):, 105111–105121.
- [16] Hong, C., Asako, Y., and Faghri, M., "Friction Factor Correlations for Compressible Gaseous Flow in a Concentric Micro Annular Tube." *Numer. Heat Transfer A.* 56 (2012): 163–179.
- [17] Hong, C., Tanak, G., Asako, Y., and Katanoda, H., "Flow Characteristics of Gaseous Flow through a Microtube discharged into the Atmosphere." *Int. J. Heat Mass Transfer* 121 (2018): 187–195.

UNCERTAINTY OF TEMPERATURE MEASURED BY THERMOCOUPLE

Yoong Yen Kee¹
Yutaka Asako²
Tan Lit Ken³
Nor Azwadi Che Sidik⁴

¹Department of Mechanical Precision Engineering, Malaysia-Japan International Institute of Technology, Universiti Teknologi Malaysia Kuala Lumpur, 54100 Kuala Lumpur, Malaysia

²Department of Mechanical Precision Engineering, Malaysia-Japan International Institute of Technology, Universiti Teknologi Malaysia Kuala Lumpur, 54100 Kuala Lumpur, Malaysia

³Department of Mechanical Precision Engineering, Malaysia-Japan International Institute of Technology, Universiti Teknologi Malaysia Kuala Lumpur, 54100 Kuala Lumpur, Malaysia

⁴Department of Mechanical Precision Engineering, Malaysia-Japan International Institute of Technology, Universiti Teknologi Malaysia Kuala Lumpur, 54100 Kuala Lumpur, Malaysia

Abstract: Reliability of data is important for researchers to verify their research result. For temperature measurement involving thermocouple, uncertainty needs to be determined before deciding the reliability of data. In this research, four error sources were proposed to have contributed in the uncertainty of temperature measured by a thermocouple which were resolution limit of data acquisition device, error in temperature measurement based on voltage measurement, reference junction compensation error and data fluctuation. Experiments were carried out to obtain reference junction compensation error and data fluctuation using HIOKI data logger (LR8400-20). The uncertainty was obtained by combining all the error values with root-sum-square equation. The uncertainty for K thermocouple obtained from this research was 0.42 °C.

Keywords: Uncertainty, thermocouple, reference junction compensation

Introduction

A thermocouple is a widely used temperature sensor consists of two different electrical conducting wires which shares two connecting junctions. The connecting junctions are known as reference junction (cold junction) and measuring junction (hot junction). Typical connections of a thermocouple are shown in Fig. 1. The working principle of a thermocouple is based on Seebeck effect which is a phenomenon where the thermo-electromotive force (EMF) is induced when there is a temperature difference between two junctions [1]. The difference of the electrical potential produced by a thermocouple depends on the difference of the junction temperatures. Therefore, the reference junction temperature should be known to determine the measuring junction temperature. The reference junction used to be placed in an ice-water bath of 0 °C as shown in Fig. 1 (a). However, it is inconvenient to keep the temperature of the ice-water bath 0 °C for a long time. Therefore, a data logger has been developed in which the reference junction compensation (RJC) is automatically conducted measuring the terminal temperature in a data logger as shown in Fig. 1 (b).

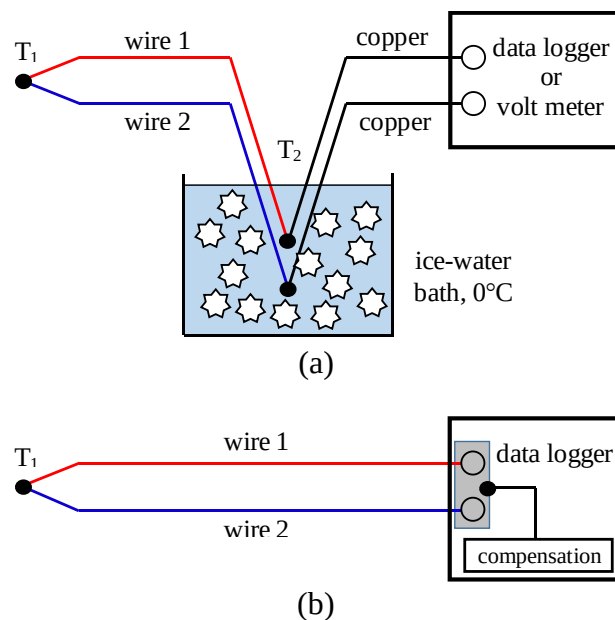


Fig. 1. Conceptual connection for thermocouple and data logger [2].

The accuracy of the temperature measurement by a thermocouple is provided in a product specification sheet of a data logger. Examples of manufacturers' reported accuracy of the temperature measurement with a type T thermocouple are tabulated in Table 1. The resolution is known as the least unit of data that can be obtained by a device and value smaller than that will not be able to record. The EMF of a thermocouple is measured by a voltage measurement unit and the measured voltage is interpreted to the temperature using the inverse polynomials. Therefore, the accuracy of the temperature measurement depends on the accuracy of the voltage measurement. When thermocouple wires are directly connected to the terminal of a data logger, the terminal becomes the reference junction. Therefore, it is necessary to measure the terminal temperature for the RJC. However, as shown in Table 1, the accuracy of the RJC is relatively poor.

Table 1: Examples of accuracy of the temperature measurement (T thermocouple)

	accuracy of voltage measurement	re- solution	accuracy of temperature measurement	accuracy of RJC	total accuracy
O	$\pm 20\mu\text{V}$	-	$\pm 0.1\%$ of rdg	$\pm 0.5^\circ\text{C}$	$\pm(0.1\%$ of rdg $+0.5^\circ\text{C})$
K	$\pm 10\mu\text{V}$	0.01°C	-	$\pm 0.5^\circ\text{C}$	-
H	$\pm 10\mu\text{V}$	0.01°C	$\pm 0.6^\circ\text{C}$	$\pm 0.5^\circ\text{C}$	$\pm 1.1^\circ\text{C}$
E	$\pm(0.02\%$ of rdg $+6\mu\text{V})$	0.1°C	$\pm(0.02\%$ of rdg $+0.2^\circ\text{C})$	$\pm 0.3^\circ\text{C}$	$\pm(0.02\%$ of rdg $+0.5^\circ\text{C})$

An uncertainty analysis of experimental measurements is necessary for results to be used to their fullest value. In the temperature measurement with a thermocouple, a few factors contribute in the uncertainty of the measured temperature. Four factors are considered in this research which are the resolution of a data logger, the accuracy of temperature measurement caused by the accuracy of the voltage measurement, the accuracy of the RJC and the fluctuation of data [2]. The uncertainty based on the resolution and the accuracy of the temperature

measurement caused by the voltage measurement is the instrument uncertainty. The uncertainty based on the RJC is the RJC uncertainty. The uncertainty based on the data fluctuation is the random uncertainty. As mentioned earlier, the accuracy of the RJC listed in Table 1 ranges from 0.3 to 0.5 °C and is poor. Therefore, measuring the terminal temperatures and the internal terminal temperature which is used for the RJC, the accuracy of the RJC is assessed. Also the uncertainty of the measured temperature are obtained in this research.

Experimental setup

In this research, a HIOKI data logger (LR8400-20) with a voltage/temperature unit (LR8500) is used to assess the RJC error. The LR8500 unit has terminals of 15 channels. Two type T thermocouples are used to measure the both temperatures of the (+) and (-) terminals of each channel. To avoid the RJC error, an ice-water bath is used. LR8400-20 measures an internal temperature, $T_{internal}$, in the LR8500 unit and it is used for the RJC. To know the $T_{internal}$, the (+) and (-) terminals of the channel 1 are shunted by a copper wire as shown in Fig. 2. In this case, the EMF is zero and the displayed temperature of this channel is the internal temperature. The connection of the thermocouples are also shown in Fig. 2. All the measurements are conducted after 30 minutes of warming-up.

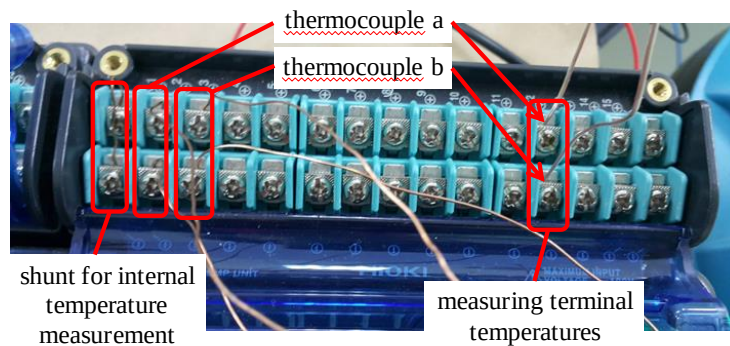


Fig. 2. Temperature measurement of (+) and (-) terminals

Results

Accuracy of Reference Junction Compensation

Before measuring the terminal temperatures, the internal temperatures, $T_{internal}$, are obtained by shunting all the (+) and (-) terminals of each channel. The results are shown in Fig. 3. The $T_{internal}$ of all channels except channels 14 and 15 are identical. The $T_{internal}$ at the channel 14 and 15 are 0.1 °C and 0.2 °C lower than that at the channel 13. LR8400-20 measures the temperature at somewhere in the LR8500 unit and make it the internal temperature, $T_{internal}$. The reason why the $T_{internal}$ at the channels 14 and 15 are lower than that at the channel 13 is to consider the heat loss from the LR8500 unit.

Measurements on temperatures at the (+) terminal and the (-) terminal, T_+ and T_- , and the internal temperature, $T_{internal}$, were repeated few times. The measured temperatures are plotted in Fig. 4. Temperature differences among the T_+ , T_- and the terminal temperatures, $T_{internal}$ are observed and also there is the temperature difference between the T_+ and T_- . However, the maximum difference between the T_- and T_+ is less than 0.3 °C. Note that data are obtained at different time. Therefore, the internal temperature at each channel is different.

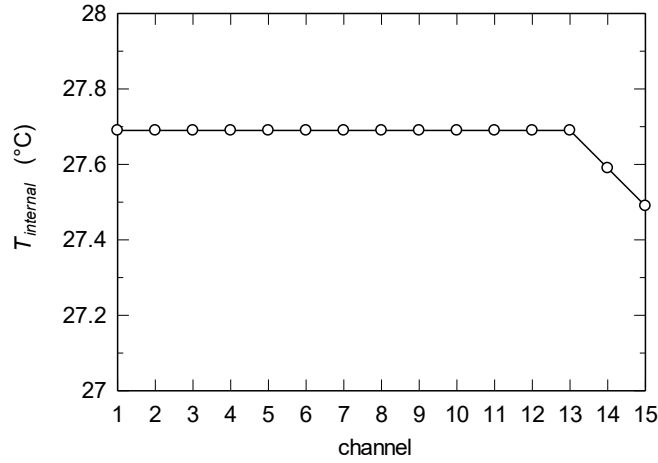


Fig. 3. Internal temperatures at each channel

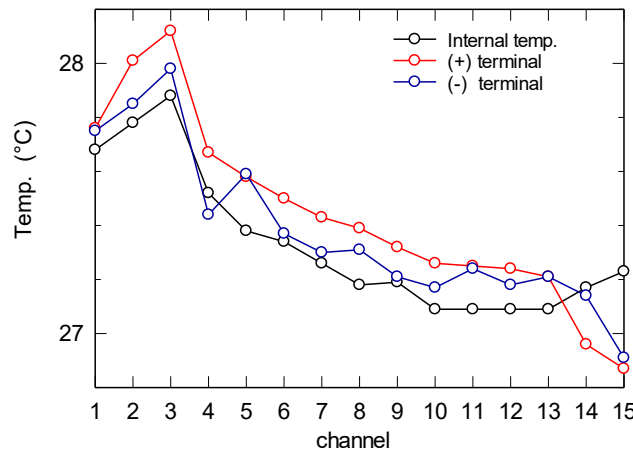


Fig. 4. Terminal temperatures and internal temperatures

Here we consider a case where the T_- is higher than the T_+ as shown in Fig. 5 (a). In such a case, the EMF is expressed as

$$E = \int_{T_-}^{T_H} S_{12} dT + \int_{T_+}^{T_-} S_1 dT \quad (1)$$

where S_{12} and S_1 are the relative Seebeck coefficient of the thermocouple and absolute Seebeck coefficient of the wire 1, respectively. Since the maximum difference between the T_- and T_+ is less than 0.3 °C, we assume that the S_1 between the T_- and T_+ is constant. Then, Eq. (1) can be rewritten as

$$E \approx \int_{T_-}^{T_H} S_{12} dT + S_1(T_- - T_+) \quad (2)$$

The T_r in Fig. 5 (b) is the reference temperature under a situation that the EMF is equal to the EMF calculated from Eq. (2). In such a case, the EMF of the thermocouple is expressed as

$$E = \int_{T_-}^{T_H} S_{12} dT + \int_{T_r}^{T_-} S_{12} dT \approx \int_{T_-}^{T_H} S_{12} dT + S_{12}(T_- - T_r) \quad (3)$$

Substituting Eq. (3) into (2), the following equation is obtained.

$$T_r = T_- - \frac{S_1}{S_{12}}(T_- - T_+) \quad (4)$$

Note that the values of S_{12} and S_1 depend on the materials of the thermocouple, therefore, the T_r also depends on the type of the thermocouple. Values of S_{12} and S_1 of the K and T thermocouples at a room temperature are listed in Table 2 [3].

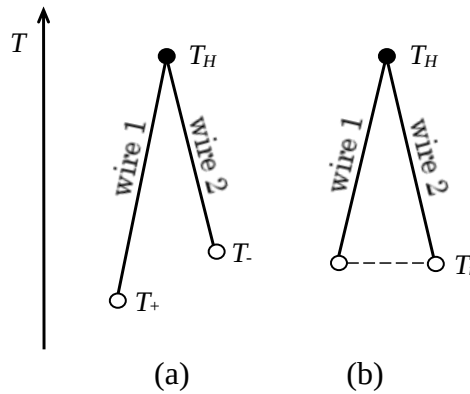


Fig. 5. Conceptual diagram to calculate EMF

Table 2: Relative and absolute Seebeck coefficients of thermocouple K and T

Thermocouple	S_{12}	S_1
K	40.5	22.2
T	40.2	1.9

The temperature difference, $T_r - T_{internal}$, expresses the error of the RJC. The temperature difference, $T_r - T_{internal}$, is plotted in Fig. 6. The $T_r - T_{internal}$ ranges from -0.31 to 0.21 for the T thermocouple and from -0.34 to 0.20 for the K thermocouple. Therefore, the accuracy of the RJC is ± 0.31 for the T thermocouple and ± 0.34 for the K thermocouple. The accuracy of the RJC provided by the manufacturer is ± 0.5 °C. The measured accuracy of the RJC are in the range of the accuracy provided by the manufacturer. Note that Eq. (4) is also available in the case that the T_+ is higher than the T_- .

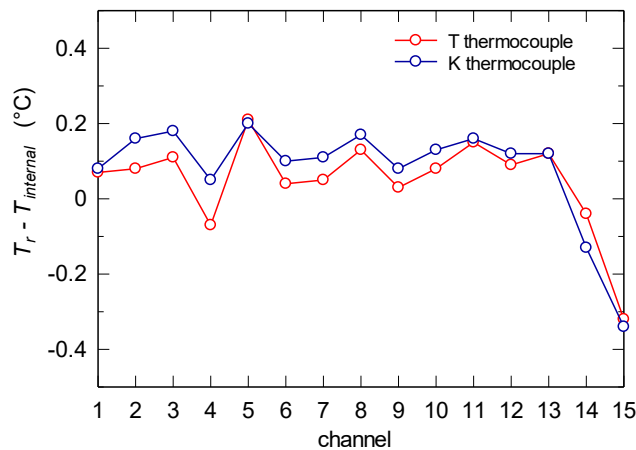


Fig. 6. Temperature difference between reference and internal temperatures

Uncertainty of measured temperature

Four factors are considered for the uncertainty of measured temperature which are the resolution of a data logger, the accuracy of temperature measurement caused by the accuracy of the voltage measurement, the accuracy of the RJC and the fluctuation of data.

Uncertainty of instrument

The uncertainty of the instrument consists of the resolution of a data logger, the accuracy of temperature measurement caused by the accuracy of the voltage measurement. Since the HIOKI's resolution is 0.01 °C based on the specification of the LR8400-20, the uncertainty of the resolution is ± 0.005 °C. The EMF produced by a thermocouple is measured by the voltage measuring unit and the measured voltage is interpreted to the temperature using the inverse polynomials. The accuracy of the voltage measurement is $\pm 10 \mu\text{V}$ based on the specification of the LR8400-20. The inverse polynomials is almost the linear function. Therefore the conversion coefficients are 0.0251 for the K thermocouple and 0.0259 for the T thermocouple [4]. The uncertainty of temperature measurement are tabulated in Table 3. The uncertainty of the resolution is much smaller than the uncertainty of temperature measurement. Therefore, the uncertainty of the instrument is equal to the uncertainty of temperature measurement.

Table 3: Uncertainty of temperature measurement caused by accuracy of voltage measurement.

Accuracy of voltage measurement	Thermocouple K	Thermocouple T
$\pm 10 \mu\text{V}$	± 0.25 °C	± 0.26 °C

Uncertainty of fluctuation

The uncertainty of fluctuation was obtained by measuring temperature of a metal block under controlled surrounding temperature. The 50/60 Hz cut-off filter setting is selected. The measured temperature was plotted in Fig. 7. Although the room temperature is controlled, the metal block temperature slightly increases or decreases with time like a temperature drift. Therefore, the temperature drift is expressed by a linear function (a red dashed line in Fig.7) and the temperature fluctuation, T' , is calculated from the following equation. The linear equation of the temperature drift is obtained by a curve fitting.

$$T' = T_{\text{measured}} - T_{\text{drift}} \quad (5)$$

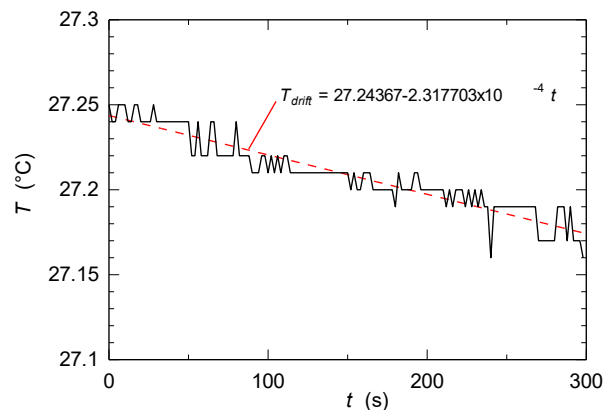


Fig. 7. Temperature of a metal block

The temperature fluctuation is plotted in Fig. 8 as a function of time. The range of the uncertainty of the resolution is also plotted in Fig. 8. The uncertainty of the resolution is smaller than the fluctuation. The temperature measurements were repeated on different days. The standard deviation of the fluctuation, σ , is calculated and $\pm 2\sigma$ is chosen for the uncertainty due to the fluctuation and the result is tabulated in Table 3. The uncertainty of random fluctuation is not constant. However, it is much smaller than the uncertainty of the instrument and the uncertainty of the reference junction compensation. This is because the 50/60 Hz cut-off filter setting is selected.

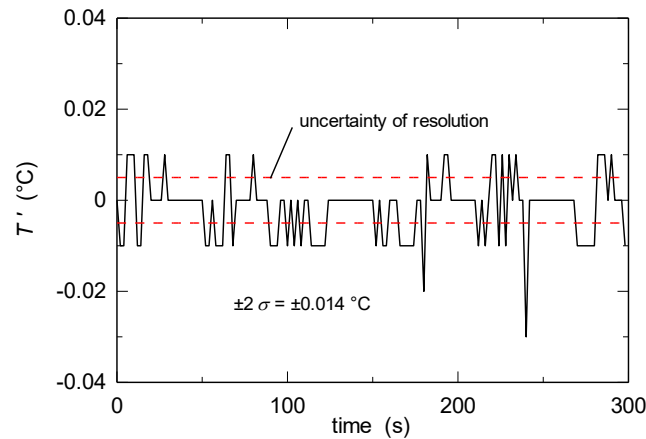


Fig. 6. Graph showing fluctuating data and temperature gap of thermocouple data

Table 4: Uncertainty of random fluctuation obtained on different day

Date	2σ (°C)	
	Thermocouple K	Thermocouple T
25/4/2019	0.0150	0.0286
6/5/2019	0.0137	0.0148
7/5/2019	0.0129	0.0149
9/5/2019	0.0122	0.0156

Overall uncertainty of measured temperature

Overall uncertainties obtained with root-sum-square, Eq. (6), are listed in Table 4. The overall uncertainties for the K thermocouple and the T thermocouple are ± 0.42 °C and ± 0.40 °C, respectively. Random error is too small and it does not affect overall uncertainty value. The uncertainty of the reference junction compensation is relatively large, the usage of the ice-water bath is recommended in an academic purpose.

$$u_{total} = \sqrt{u_{inst}^2 + u_{RJC}^2 + u_{rand}^2} \quad (6)$$

Table 5: Overall uncertainty

Thermocouple	u_{inst}	u_{RJC}	u_{rand}	u_{total}
k	± 0.25	± 0.34	$\pm 0.012 \sim \pm 0.015$	± 0.42
t	± 0.26	± 0.31	$\pm 0.015 \sim \pm 0.029$	± 0.40

Conclusion

1. Uncertainty of measured temperature by the HIOKI data logger (LR8400-20) is 0.42 °C for the thermocouple K and 0.40 °C for the thermocouple T when the RJC is used.

This value is much less than 1.1 °C which is reported by the manufacturer and data logger is reliable.

2. The uncertainty of the reference junction compensation is relatively large, the usage of the ice-water bath is recommended in an academic purpose.

Acknowledgement

The authors would like to express their appreciation to Takasago Thermal Engineering Co. Ltd., Japan, for providing financial support for this work through Takasago education and research grant (Vote No: 4B365).

References

- [1] Bajzek, T.J., *Thermocouples: a sensor for measuring temperature*. IEEE Instrumentation & Measurement Magazine, 2005. **8**(1): p. 35-40.
- [2] Recktenwald, G., Lecture notes of ME449 "*Uncertainty estimation and calculation*". 2012, Version 0.48, <http://web.cecs.pdx.edu/~gerry/class/ME449/lectures/pdf/uncertaintySlides.pdf>
- [3] Bentley, R.E., *Handbook of temperature measurement Vol. 3*, Springer-Verlag, Singapore, 1998.
- [4] ASTM, *A manual on the use of thermocouples in temperature measurement*, ASTM Philadelphia, New York, 4th ed., 1993.

NUMERICAL ANALYSIS OF CALCIUM CARBONATE (CaCO₃) SUSPENSION FLOW IN PENTAGON SPIRAL PIPE WITH PITCH VARIATION

Gunawan¹
Yanuar²
M. R. Setiawan³
A. Arianda⁴

¹Department of Mechanical Engineering, Universitas Indonesia, Jakarta, 16424, Indonesia (Email: gunawan_kapal@eng.ui.ac.id)

²Department of Mechanical Engineering, Universitas Indonesia, Jakarta, 16424, Indonesia

³Student of Department of Mechanical Engineering, Universitas Indonesia, Jakarta, 16424, Indonesia

⁴Student of Department of Mechanical Engineering, Universitas Indonesia, Jakarta, 16424, Indonesia

Abstract: Energy loss due to friction in fluid transportation was a serious energy efficiency problem. Then, calcium carbonate (CaCO₃) comes as a solution to solve this issue. The use of CaCO₃ in water flow through the pentagon spiral pipe was effective to create more efficient flow by promoting drag reduction (DR). This research aimed to optimize the pentagon spiral pipe geometry to promotes drag reduction by CaCO₃. The analysis has been done by using Computational Fluid Dynamics (CFD) with Ansys Fluent 19.2 in a steady-state flow. The flow model of K-E RNG (Renormalization Group) was adopted to analyze the pitch variation effect on the drag reduction in various Reynolds number. The results showed the strong dependency of the drag reduction on the pitch variation and Reynolds number. The Pentagon spiral pipe geometry can be optimized with modification in pitch according to the fluid application condition.

Keywords: Energy efficiency, Calcium carbonate suspension, Spiral pipe, drag reduction

Introduction

This present of work shows a comparison between the predictions from a computational fluid dynamics (CFD) model of flow in pentagon spiral pipe and experimental data of drag reduction (DR) by calcium carbonate (CaCO₃) that already reported in Sealtial Mau et al. [1]. Ghorai et al. have conducted a research that have aims to propose a global empirical correlation for the interfacial friction factor and create a local modeling of wavy stratified air–water flow in pipe [2]. The result shows things that could not be seen in the experimental method such as velocity profiles in liquid phase, velocity profiles in gas phase as well as interfacial roughness in wavy stratified flow.

In the early 2019, a research about new method of revealing the drag reduction mechanism was conducted [3]. The research has aim to establish a DNS framework of gas-liquid drag-reducing flow and use this framework to obtain numerical results and try to explain the mechanism of two-phase drag reduction from the results. Their new method used was Efficient interface tracking method—coupled volume-of-fluid and level set (VOSET) and typical polymer constitutive model Giesekus combined with the momentum equation of the two-phase turbulent flow. Results show that drag reduction mechanism can be done due to local enhancement in the core region with local suppression near the walls of turbulent fluctuations, and as same as single phase flow, gas-liquid drag-reducing flow can be the global suppression of turbulent fluctuations. These results can be found only if we used CFD model.

Drag reduction in internal flow also already conducted in 2013 [4]. They analyze the internal flow of tube covered by riblet structure using RANS (Reynolds-Averaged Navier-Stokes) and RNG (Renormalization Group) k-ε model to generate the governing equation. Result shows that riblet structure weakens the velocity fluctuation and strength of turbulence burst in boundary layer, macroscopically reducing the velocity gradient in near wall region, and consequently diminishing the skin-friction resistance. Another case of k-ε model already used to analyze the fluid dynamic behaviors of the two-phase pipe flow in 2012 [5]. Results show a verified conclusion to the actual process in the term of crude oil loss of an oil tanker caused by VOC emission. RNG k-ε model itself was firstly researched to generate the equation for turbulent flow [6].

In the terms of passive control, super hydrophobic media is one of the famous ways. It reveals its high relation to the drag reduction [7]. Another famous way is spiral pipe. Yanuar et al. investigated the effect of calcium carbonate nanoparticles on the flow in a pentagon spiral pipe. The result show that spiral pipe prevents the sedimentation of heavy particle. The geometry of spiral pipe also generates circumferential flow resulting in the working fluids twisting at a certain Reynolds and it enhance the phenomenon of drag reduction [8].

The aim of this study is to optimize the pentagon spiral pipe geometry to promote drag reduction by CaCO₃. Analysis has been conducted by using Computational Fluid Dynamics (CFD) with Ansys Fluent 19.2 in a steady-state flow and RNG k-ε was used for as the flow model to solve the governing equation.

Methodology

Mathematical Model

Mathematical model that will be used in this study is obtained from continuity and momentum equation that differentiated from Navier-stokes equation that shows below.

$$\frac{d}{dt}(\alpha_f \rho_f) + \nabla \cdot (\alpha_f \rho_f \mathbf{u}_f) = 0 \quad (1)$$

$$\frac{d}{dt}(\alpha_f \rho_f) + \nabla \cdot (\alpha_f \rho_f \mathbf{u}_f \mathbf{u}_f) = -\alpha_f \nabla p + \nabla \cdot (\alpha_f \tau_f) + \alpha_f \rho_f g - f_{drag} \quad (2)$$

As flow characteristics observed in this research represent a turbulent flow, the flow is visualized by applying the standard k-ε equations.

$$\frac{d}{dt}(\alpha_f \rho_f k) + \nabla \cdot (\alpha_f \rho_f \mathbf{u}_f k) = \nabla \cdot (\alpha_f \Gamma_k \nabla k) + \alpha_f G - \alpha_f \rho_f \varepsilon \quad (3)$$

$$\frac{d}{dt}(\alpha_f \rho_f \varepsilon) + \nabla \cdot (\alpha_f \rho_f \mathbf{u}_f \varepsilon) = \nabla \cdot (\alpha_f \Gamma_\varepsilon \nabla \varepsilon) + \alpha_f \frac{\varepsilon}{k} (c_1 G - c_2 \rho_f \varepsilon) \quad (4)$$

In this study, the equations are solved in ANSYS Fluent 19.2 to execute CFD simulations. Besides, the scientifically detailed flow issue is controlled exclusively by the Reynolds number.

Geometry Model

In this study, the model used is pentagon spiral pipe that modeled in Autodesk Inventor 2019. The pentagon spiral pipe characteristic is same as spiral pipe used in past study conducted by Sealtial Mau et al. [1]. Figure 1 shows the geometry of spiral pipe being modeled.

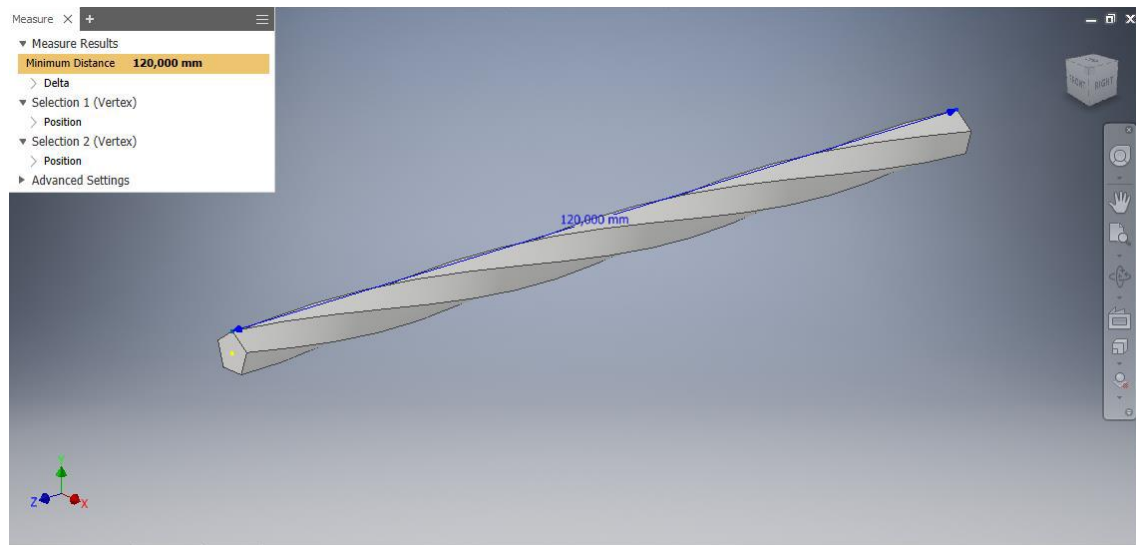


Fig. 1. Pentagon spiral pipe model

Since we will be focusing our research in the pitch variation effects for drag reduction, we use three different geometry models with diameter ratio (P/D) of 7.3, 9.05, and 10.8 respectively. The P/D of 10.8 has been used in the previous experimental research conducted by Sealtial et al. [1] for water flow drag reduction using CaCO₃ and P/D 7,3 . Next, P/D of 7.3 variation has been used because according to Yanuar et al. [9] it is the best P/D for drag reduction in a three-lobed spiral pipe.

Viscosity Model

Viscosity model that will be used is Power Law model due to the change of the fluid viscosity. Then the effective viscosity model is stated as:

$$\mu_{eff} = K \left(\frac{du}{dy} \right)^{n-1} \quad (5)$$

where K is a consistency index and shear stress, τ , is given by the power of n for the velocity gradient.

Verification and Validation

The verification has been done by using Mesh independency study to determine the perfect number of elements for accurate results with efficient use of time. From the conducted mesh independency study (Figure 2.), it can be concluded that from 600,000 of elements total we can obtain a reliable result.

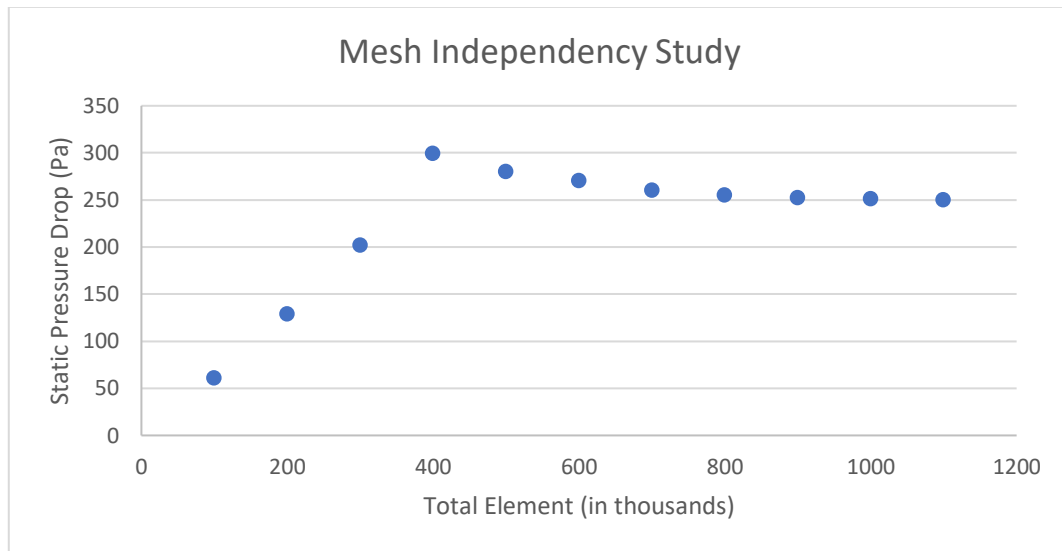


Fig. 2. Mesh independency study

In consideration of the mesh independency from 800,000 total elements onwards, we conduct the simulation with 600,000 elements in totals since according to Figure 3. the time consumed for analyzing time are increasing exponentially after 600,000 elements total.

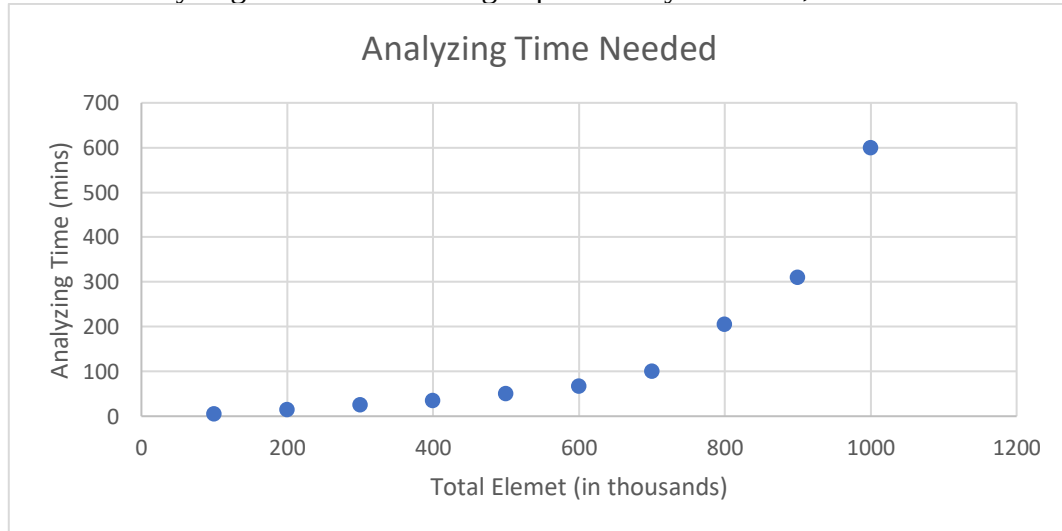


Fig. 3. Analyzing time

Validation of this research has been done by comparing the results with the previous experimental research conducted by Sealtial et. al. [1]. The simulation results are in good agreement with the previous experiment with less than 5 percent margin of error. Figure 4 shows the validation result of this work.

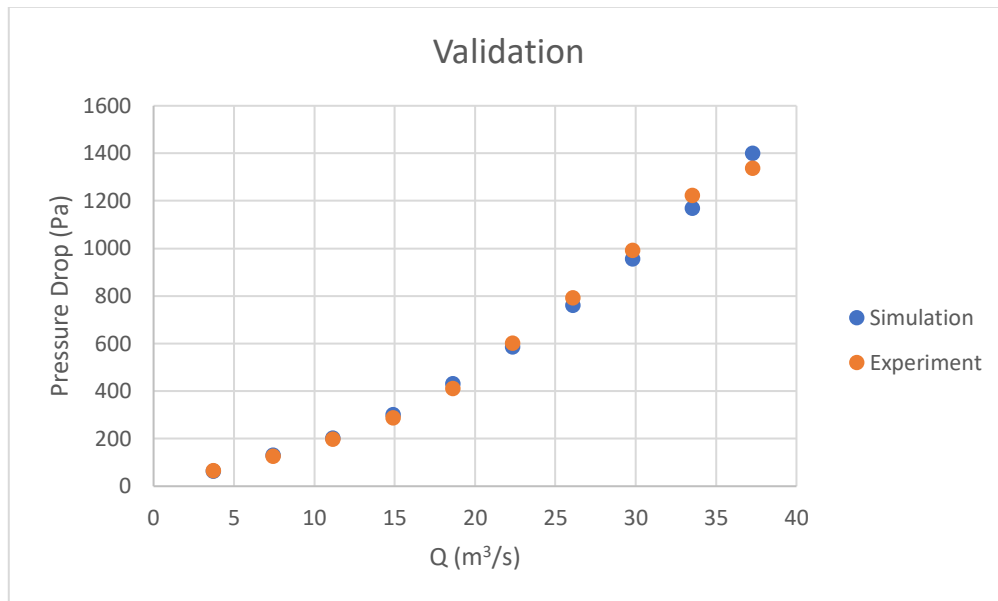


Fig. 4. Validation table

Results

Figure 5. shows that the pressure drops tend to rise as number of ratios of diameter (P/D) rising. This phenomenon could confirmed by Yanuar et al. [9] who conducted research on pitch variation of crude oil flow by experimental method.

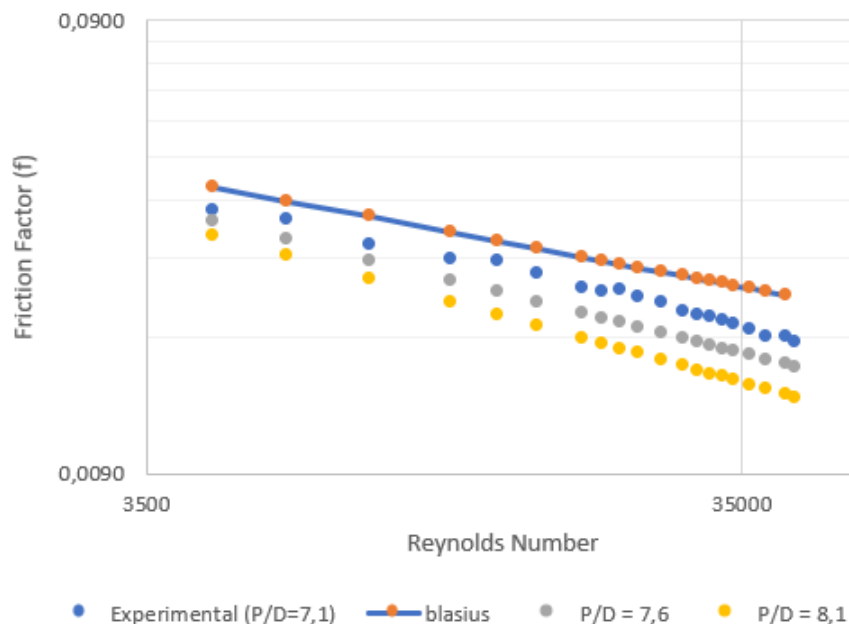


Fig. 5. Relationship between friction factor and Reynolds number

By positive correlation, it is also confirmed that as number of pitch ratio rising, DR% could tend to rise too. As the Figure 6. reveals that maximum drag reduction occurs at the number of 40% on P/D = 8.1

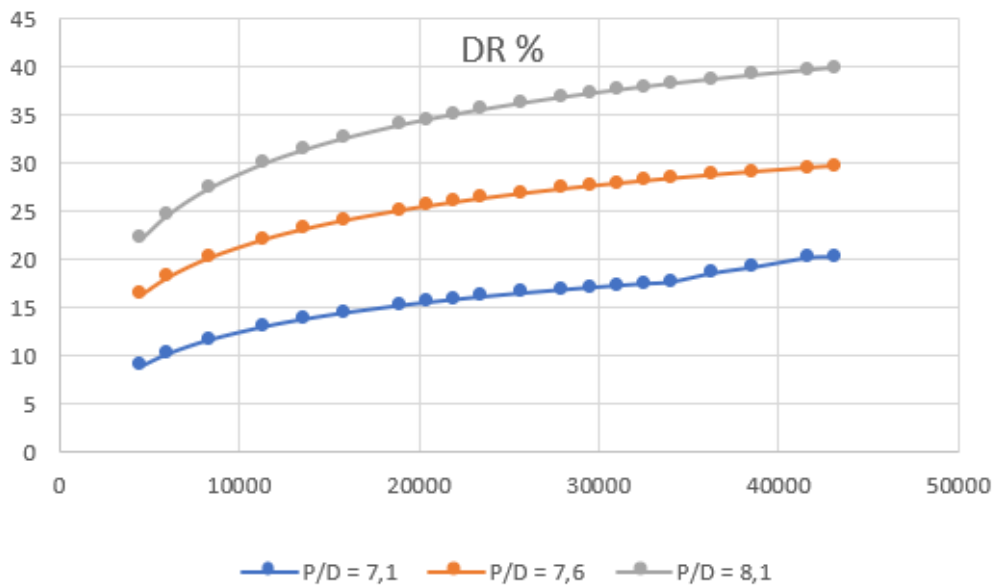


Fig. 6. Drag reduction

The highest DR% are achieved when the swirling effect of the spiral pipe are in full swing at over 40,000 in Reynolds number. Higher tangential velocity creates higher turbulence to increase the drag reduction effect [10].

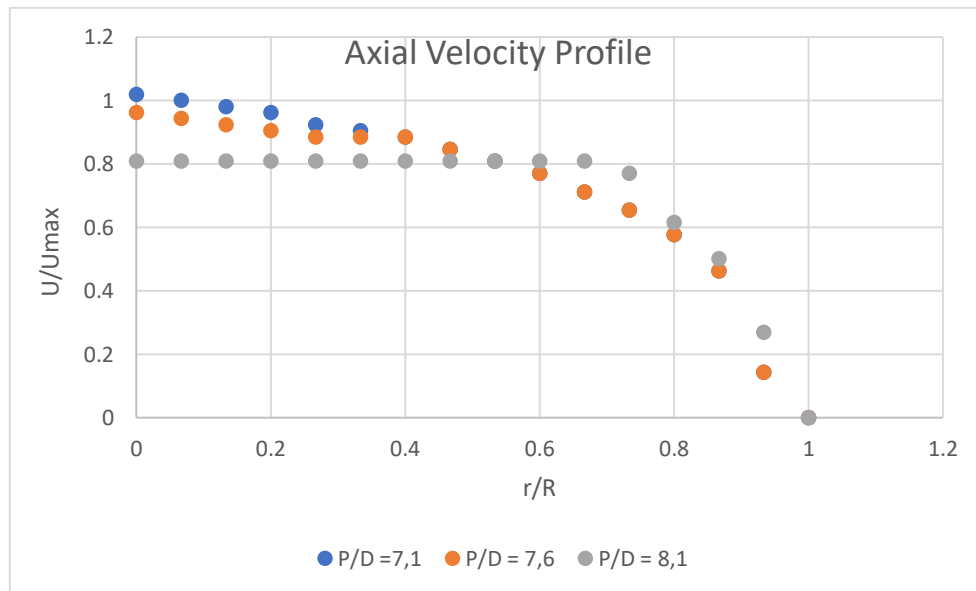


Fig. 7. Axial velocity profile

It can be seen from Figure 7. that different pitch frequency creates differences in the axial velocity profile. The best P/D variation creates a more evenly distributed local velocity magnitude. This happens because of the swirling effect. Furthermore, core of turbulence and axial velocity is highly correlated with swirl intensity [11].

Conclusions

The increase in P/D creates a better DR due to the increase of swirl number. This happens because smoother transition in swirl formation is happening in the pipe with higher P/D.

Furthermore, the major conclusion that can be pulled from this study is pitch variation has massive impact in terms of drag reduction of water flow. This result also confirmed the past study that researched the same thing by experimental method of crude oil flow [9]. The others, this study could complete the state of arts of past study and also confirm its conclusions [1].

Acknowledgement

This research was funded by a grant from Directorate of Research and Community Engagement Universitas Indonesia with Pit 9 Scheme. Contract number: NKB-0083/UN2.R3.1/HKP.05.00/2019

References

- [1] Sealtial Mau and Yanuar. "*Effect of Calcium Carbonate Solution on Drag Reduction in a Pentagon Spiral Pipe.*" Journal of Advanced Research in Fluid Mechanics and Thermal Sciences 46, no. 1 (2018): 41-48.
- [2] Subhashini Ghorai and Krishna Nigam. "*CFD modeling of flow profiles and interfacial phenomena in two-phase flow in pipes.*" Chemical Engineering and Processing: Process Intensification 45, no. 1 (2006): 55-65.
- [3] Yi Wang, Yan Wang and Zhe Cheng. "*Direct Numerical Simulation of Gas-Liquid Drag-Reducing Cavity Flow by the VOSET Method.*" Polymers 11 no. 4 (2019): 596-604.
- [4] Zi Guo Wei, Yong Li, Zhi Wu Ke and Xiang Ling Li. "*Numerical Simulation of Internal Flow Drag Reduction with Triangular Riblet Surface.*" Advanced Materials Research 779-780 (2013): 401-405.
- [5] Woohyun Kim, Munkyu Yoon, Moonyong Lee and Suwon Park. "*CFD Analysis of Cavitation in a Crude Oil Pipeline to an Oil Tanker.*" Computer Aided Chemical Engineering 31 (2012): 580-584.
- [6] Charles Gatski Speziale and Sivagnanam Thangam. "*Analysis of an RNG based turbulence model for separated flows.*" International Journal of Engineering Science 30, no. 1 (1992). 30(10): 1379-1388.
- [7] Roberta Costantini, Jean Paul Mollicone and Francesco Battista. "*Drag reduction induced by superhydrophobic surfaces in turbulent pipe flow.*" Physics of Fluids 30 no. 2 (2018):1-17.
- [8] Yanuar, Marcus Alberth Talahatu, Sealtial Mau, Kurniawan teguh Waskito and Winda Wulandari. "*The Effect of Calcium Carbonate (CaCO₃) Nanoparticles on the Flow through a Pentagon Spiral Pipe.*" International Journal of Technology 8, no. 7 (2017): 1258-1265.
- [9] Yanuar, Kurniawan Teguh Waskito, Gunawan and Budiarto. "*Drag Reduction and Velocity Profiles Distribution of Crude Oil Flow in Spiral Pipes.*" International Review of Mechanical Engineering 9 no. 1 (2015):1-10.
- [10] Smeeta Fokeer, Ian Lowndes and David Hargreaves. "*Numerical modelling of swirl flow induced by a three-lobed helical pipe.*" Chemical Engineering and Processing: Process Intensification 49 no. 5 (2010): 536-546.
- [11] F. Chang and Vijay Kumar Dhir. "*Turbulent flow field in tangentially injected swirl flows in tubes.*" International Journal of Heat and Fluid Flow 15 no. 5 (1994): 346-356.

THE EFFECTS OF MAGNETIC BLOOD FLOW IN AN INCLINED CYLINDRICAL TUBE USING CAPUTO-FABRIZIO FRACTIONAL DERIVATIVES

Dzuliana Fatin Jamil¹

Salah Uddin¹

M Ghazali Kamardan¹

Rozaini Roslan¹

¹Department of Mathematics and Statistics, Fakulti Sains Gunaan dan Teknologi, Universiti Tun Hussein Onn Malaysia, 84600 Panchor, Johor, Malaysia

Abstract: The Caputo-Fabrizio fractional derivative was used to study the effect of magnetic blood flow in an inclined cylindrical tube. The resultant blood flow was due to the oscillating pressure gradient with the existence of external magnetic field. The analytical solutions were obtained by means of the Laplace and finite Hankel transforms. The effects of the fluid parameters such as Hartmann (Ha) and Reynolds (Re) numbers to the blood and magnetic particles velocities were graphically presented with the aid of MATHCAD. The results showed that the applied magnetic field reduced the blood and magnetic particles velocities due to the Lorentz forces. Furthermore, by increasing the Reynolds number caused the velocity profiles of the blood and magnetic particle increased. The velocity of magnetic particles is always less than the velocity of blood regardless the presence of magnetic field.

Keywords: Caputo Fabrizio derivative, Blood flow, Magnetohydrodynamics, Cylindrical domain

Introduction

Biomagnetic Fluid Dynamics (BFD) revolves around the study of fluid flow under the interference of magnetic field. As reported by [1], biomagnetic fluid exists in human body and its motion can hence be influenced by the magnetic field. Many researches in BFD have been carried out since the past centuries. BFD is mainly applied in bioengineering and medical sciences such as developing magnetic tracer, targeting transport of medical drugs using magnetic particles as drug carriers, developing magnetic cell separation devices, treating cancer tumor, etc. The underlying fluid dynamics has attracted the interest of many researchers [2].

Blood is one of the biological fluids that can conduct electricity due to the strong presence of erythrocytes (red blood cells). Erythrocytes are indeed negative charge carriers that can provide magnetic effect on the vessel walls. This may change the pulsatile nature of the blood flow [3]. According to [4], Ferro-Hydrodynamics (FHD) has the same characteristics as BFD. The flow of an electrically conductive liquid is just merely affected by the fluid magnetization and there is no influence of induced current at low Reynolds number. Therefore, similar to Magneto-Hydrodynamics, fluid magnetization is considered in the mathematical model of BHD.

The study of MHD is very useful in bio-engineering especially in the study of Magnetic Resonance Imaging (MRI), heart attack and cancer treatment [5]. [6] studied the mathematical model of blood flow in tiny blood vessel subjected to magnetic field. They considered the blood to be Newtonian fluid with the suspension of erythrocytes and the cell-free layer surrounding the core. Their findings indicated that the velocity and flow rate reduced as the strength of the magnetic field increased. [7] studied the blood flow in the catheterized stenosed artery

subjected to slip velocity at the wall in the presence of a transverse magnetic field. They found that the wall shear stress increased as the strength of the transverse magnetic field increased. [8] analyzed the MHD blood flow with gold nanoparticles. The blood was threatened as third grade nanofluid in a hollow blood vessel. This problem was solved by applying the Least Square Method (LSM), the Galerkin method (GM) and the fourth order Runge-Kutta method. It was found that the velocity declined in the presence of magnetic field. A mathematical model based on fractional derivative was derived to study the effect of magnetic field on the blood flow inside the oscillatory arteries [3]. The blood flow was driven by the periodic pressure gradient and the body acceleration. The exact solution was obtained by using the Hankel and Laplace Transforms. They found that fractional derivative is valuable in controlling the blood temperature and velocity.

The use of fractional order derivative in mathematical modeling has found numerous applications such as those in physics, fluid mechanics, mathematical biology and electrochemistry [9]. Besides that, as compared to the ordinary traditional calculus the model derived from fractional calculus is more general and accurate. Considering the significance of fractional derivatives, [10] used fractional calculus to study the flow of Oldroyd-B fluid in stenosed arteries. The derived mathematical model of tapered stenosed artery in the presence of pressure gradient might help medical practitioners in treating cardiovascular diseases. [11] studied the free convection flow of an incompressible fractional second-grade fluid near the vertical plate by using Caputo and Caputo Fabrizio derivatives. Based on the comparative study, the effects of flow and fractional parameters on the temperature and velocity profiles were obtained. They found that the temperature values obtained from both fractional models decreased when Prandtl number increased.

In a general physiological system, some arteries are not perfectly horizontal or vertical. As a result, the effect of gravity should be considered in the flow calculation involving inclined arteries. [12] analyzed the unsteady non-Newtonian blood flow in an inclined catheterized and overlapping stenosed artery. The finite difference method was used in solving the stated problem. They showed that the axial velocity, flow rate and resistance impedance would be significantly affected by parameters such as wall slip and inclination angle. [13] analyzed the pulsatile flow of Herschel-Bulkley fluid through multiple inclined stenoses with periodic body acceleration. They revealed that the velocity increased as body acceleration increased. [14] investigated the influence of heat transfer on the peristaltic transport of blood through a porous medium. The blood was treated as the incompressible MHD second grade fluid asymmetric channel. They used the analytical approach such as perturbation method in solving the problem. Based on the study, the pressure gradient and the size of the trapped bolus increased as the inclination angle increased.

Motivated from the fore mentioned literatures, the blood vessel was modeled as an inclined cylindrical tube subjected to an inclined magnetic field. Blood flow occurred due to the oscillating pressure gradient in the z -direction. A fractional mathematical model of Caputo-Fabrizio was developed in cylindrical coordinate system to study the magnetic blood flow. The fractional models of blood flow and particle motion under the influence of magnetic field were solved analytically by using the joint method of Laplace and finite Hankel transforms. The analytical solutions were plotted using the mathematical software, i.e. Mathcad.

Methodology

The circular section of the artery was shown in Figure 1. In this study we considered blood flow occurred in an inclined cylindrical tube of radius R_0 under the influence of uniformly distributed magnetic particles. The magnetic blood particles that flowed in the axial direction

(i.e. z - direction) were driven by oscillating pressure gradient. Meanwhile an external magnetic field has been applied perpendicularly throughout the inclined cylindrical vessel.

In this study the magnetic Reynolds number was taken small enough as the entire blood flow stream was subjected to an external magnetic field. In other words, the induced magnetic field was assumed negligible compared to the applied magnetic field. At $t=0$, the cylindrical tube, the blood and the magnetic particles were treated as stationary.

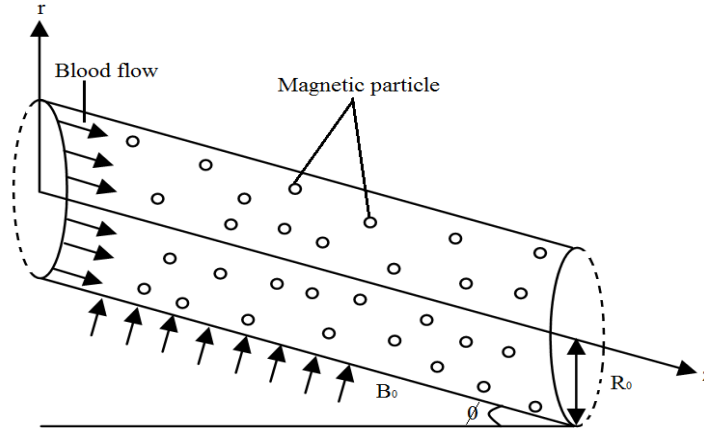


Fig. 1.: Geometry figure of the magnetic blood flow

The governing equations are the Navier-Stokes equations describing the blood flow, the Maxwell's relations describing the magnetic field and the Newton's second law describing the particle motion.

The Maxwell equations are

$$\nabla \cdot \vec{B} = 0, \nabla \times \vec{B} = \mu_0 \vec{J}, \nabla \times \vec{E} = -\frac{\partial \vec{B}}{\partial t}, \quad (1)$$

where $\vec{B}$ is the magnetic flux intensity, μ_0 is the magnetic permeability, $\vec{E}$ is the electric field intensity and $\vec{J}$ is the current density given by

$$\vec{J} = \sigma(\vec{E} + \vec{V} \times \vec{B}), \quad (2)$$

Here σ is the electrical conductivity and $\vec{V}$ is the velocity vector. The electromagnetic force $\vec{F}_{em}$ is defined as

$$\vec{F}_{em} = \vec{J} \times \vec{B} = \sigma(\vec{E} + \vec{V} \times \vec{B}) \times \vec{B} = -\sigma B_0^2 u(r, t) \vec{k}, \quad (3)$$

where $\vec{k}$ is the unit vector in the z -direction and $\vec{V} = u(r, t) \vec{k}$ is the axial velocity of the blood.

The force $\vec{F}_{em}$ will be included in the momentum equations.

The unsteady blood flow in an axisymmetric cylindrical tube of radius R_0 under the influence of uniform transverse magnetic field and pressure gradient of the form

$$-\frac{\partial p}{\partial z} = A_0 + A_1 \cos(\omega t), \quad A_0 > 0, \quad (4)$$

was considered, where the constants A_0 and A_1 are the amplitudes of the pulsatile magnetic field and pressure gradient that give rise to systolic or diastolic pressure.

The momentum equation for fluid flow in cylindrical coordinates system (r, θ, z) is

$$\frac{\partial u}{\partial t} = -\frac{1}{\rho} \frac{\partial p}{\partial z} + \nu \left(\frac{\partial^2 u}{\partial r^2} + \frac{1}{r} \frac{\partial u}{\partial r} \right) + \frac{KN}{\rho} (v - u) - \frac{\sigma B_0^2 \sin \theta}{\rho} + g \sin \phi, \quad (5)$$

where ρ is the fluid density, ν is the fluid kinematic viscosity, p is the fluid pressure, N is the number of magnetic particles per unit volume, K is the Stokes constant, u is the fluid velocity and v is the particle velocity. The term $\frac{KN}{\rho} (v - u)$ is the force due to the relative motion between fluid and magnetic particles. It was assumed that the Reynolds number of the relative velocity was small. As such, the force between the magnetic particles and the blood is proportional to the relative velocity.

The motion of magnetic particle is governed by the Newton's second law

$$m \frac{\partial v}{\partial t} = K(u - v), \quad (6)$$

where m is the average mass of the magnetic particles.

In order to consider the time-fractional model, Equations (5) and (32) can be multiplied by $\lambda = \sqrt{\frac{R_0 \rho}{A_0}}$, to yield a term with dimension of time t . The governing equations of the time-fractional model are

$$\begin{aligned} \lambda^\alpha D_t^\alpha u = & -\frac{\lambda}{\rho} (A_0 + A_1 \cos(\omega t)) + \lambda \nu \left(\frac{\partial^2 u}{\partial r^2} + \frac{1}{r} \frac{\partial u}{\partial r} \right) + \frac{KN\lambda}{\rho} (v - u) \\ & - \frac{\sigma B_0^2 \lambda}{\rho} + g \lambda \sin \phi, \end{aligned} \quad (7)$$

and

$$\lambda^\alpha D_t^\alpha v = \frac{K\lambda}{m} (u - v), \quad (8)$$

where the Caputo-Fabrizio derivative operator is

$${}^{CF} D_t^\alpha u(r, t) = \frac{1}{1-\alpha} \int_0^t \exp\left(-\frac{\alpha(t-\tau)}{1-\alpha}\right) \frac{\partial u(r, \tau)}{\partial \tau} d\tau. \quad (9)$$

The Laplace transform of the Caputo-Fabrizio time derivative can be written as

$$L\{{}^{CF} D_t^\alpha u(r, t)\} = \frac{sL\{u(r, t)\} - u(r, 0)}{(1-\alpha)s + \alpha}. \quad (10)$$

The initial boundary conditions of the fluid inside the cylindrical domain of radius R_0 are

$$\begin{aligned} u(r, 0) = 0, \quad v(r, 0) = 0, \quad r \in [0, R_0] \\ u(R_0, t) = 0, \quad v(R_0, t) = 0, \quad t > 0 \end{aligned} \quad (11)$$

For dimensionless study, the following non-dimensional parameters can be introduced

$$\begin{aligned} r^* = \frac{r}{R_0}, \quad t^* = \frac{t}{\lambda}, \quad u^* = \frac{u}{u_0}, \quad v^* = \frac{v}{u_0}, \\ A_0^* = \frac{\lambda A_0}{\rho u_0}, \quad A_1^* = \frac{\lambda A_1}{\rho u_0}, \quad \omega^* = \lambda \omega, \quad g^* = \frac{g}{u_0^2 / R_0} \end{aligned} \quad (12)$$

where u_0 is the characteristics velocity.

By introducing the above parameters and dropping the * notation the non-dimensional of Equations (7), (8), and (11) are

$$D_t^\alpha u = A_0 + A_1 \cos(\omega t) + \frac{1}{\text{Re}} \left[\frac{\partial^2 u}{\partial r^2} + \frac{1}{r^*} \frac{\partial u}{\partial r^*} \right] + R(v - u) - Ha^2 u + \frac{\sin \phi}{F}, \quad (13)$$

$$G \cdot D_t^\alpha v = u - v, \quad (14)$$

where $\text{Re} = \frac{R_0}{\lambda \nu}$ is the Reynolds number, $R = \frac{KN\lambda}{\rho}$ is the particles concentration parameter,

$Ha^2 = \frac{\sigma B_0^2 \lambda \sin \theta}{\rho}$ is the Hartmann number and $F = \frac{R_0}{\lambda u_0 g}$ is the inclination angle parameter.

The non-dimensional boundary conditions are

$$\begin{aligned} u(r, 0) = 0, \quad v(r, 0) = 0, \quad r \in [0, 1] \\ u(1, t) = 0, \quad v(1, t) = 0, \quad t > 0 \end{aligned} \quad (15)$$

Laplace transform well suited when the temporal variable t is adopted in the blood flow model (see Equations (7), (8)) and boundary condition (11). After the transformation process, we obtain

$$\begin{aligned} \frac{s\bar{u}(r, s)}{s + \alpha(1-s)} = \frac{A_0}{s} + \frac{A_1 s}{s^2 + \omega^2} + \frac{1}{\text{Re}} \left[\frac{\partial^2 \bar{u}(r, s)}{\partial r^2} + \frac{1}{r} \frac{\partial \bar{u}(r, s)}{\partial r} \right] + R\bar{v} \\ - (R + Ha^2)\bar{u}(r, s) + \frac{\sin \phi}{sF}, \end{aligned} \quad (16)$$

$$G \frac{s\bar{v}(r, s)}{s + \alpha(1-s)} = \bar{u}(r, s) - \bar{v}(r, s), \quad (17)$$

$$\bar{u}(1, s) = 0, \quad \bar{v}(1, s) = 0. \quad (18)$$

From Equation (17), the following equation can be obtained:

$$\bar{v}(r, s) = \frac{s + \alpha(1-s)}{Gs + s + \alpha(1-s)} \bar{u}(r, s). \quad (19)$$

Substituting $\bar{v}(r, s)$ from Equation (19) into Equation (16) can be obtained:

$$\begin{aligned} \left[\frac{s}{s + \alpha(1-s)} - R \left(\frac{s + \alpha(1-s)}{s + sG + \alpha(1-s)} \right) + R + Ha^2 \right] \bar{u}(r, s) = \\ \frac{A_0}{s} + \frac{A_1 s}{s^2 + \omega^2} + \frac{1}{\text{Re}} \left[\frac{\partial^2 \bar{u}}{\partial r^2} + \frac{1}{r} \frac{\partial \bar{u}}{\partial r} \right] + \frac{\sin \phi}{sF}. \end{aligned} \quad (20)$$

Applying the finite Hankel transform of order zero (i.e. applying the boundary condition (11) in Equation (20)) the following equation can be obtained:

$$\begin{aligned} \left[\frac{s}{s + \alpha(1-s)} - R \left(\frac{s + \alpha(1-s)}{s + sG + \alpha(1-s)} \right) + R + Ha^2 \right] \bar{u}_H(r_n, s) = \\ \left[\frac{A_0}{s} + \frac{A_1 s}{s^2 + \omega^2} + \frac{\sin \phi}{sF} \right] \frac{J_1(r_n)}{r_n} - \frac{1}{\text{Re}} r_n \bar{u}_H(r_n, s), \end{aligned} \quad (21)$$

where $\bar{u}_H(r_n, s) = \int_0^1 r \bar{u}(r, s) J_0(r_n r) dr$ represents the finite Hankel transform of the

velocity function $\bar{u}(r, s) = LT[u(r, t)]$ and $r_n, n = 1, 2, \dots$ are the positive roots of the equation $J_0(x) = 0$. Here J_0 is the Bessel function of order zero and it belongs to the first kind. By simplifying the coefficient of $\bar{u}_H(r_n, s)$ in Equation (21), the following equations can be formulated:

$$\bar{u}_H(r_n, s) = \frac{s^2 x_{5n} + s x_{6n} + \alpha^2}{s^2 x_{2n} + s x_{3n} + x_{4n}} \left[\frac{1}{s} \left(A_0 + \frac{\sin \phi}{F} \right) + \frac{A_1 s}{s^2 + \omega^2} \right] \frac{J_1(r_n)}{r_n}, \quad (22)$$

$$\bar{u}_H(r_n, s) = \left[\frac{x_{9n}}{s - x_{7n}} + \frac{x_{10n}}{s - x_{8n}} \right] \left[\frac{1}{s} \left(A_0 + \frac{\sin \phi}{F} \right) + \frac{A_1 s}{s^2 + \omega^2} \right] \frac{J_1(r_n)}{r_n}, \quad (23)$$

$$\begin{aligned} \bar{u}_H(r_n, s) = & \left(A_0 + \frac{\sin \phi}{F} \right) \left[\frac{s^{-1}}{s - x_{7n}} x_{9n} + \frac{s^{-1}}{s - x_{8n}} x_{10n} \right] \frac{J_1(r_n)}{r_n} \\ & + A_1 \frac{s}{s^2 + \omega^2} \left[x_{9n} \frac{1}{s - x_{7n}} + x_{10n} \frac{1}{s - x_{8n}} \right] \frac{J_1(r_n)}{r_n}, \end{aligned} \quad (24)$$

Note, the parameters in Equations (22) and (23) introduced for simplifying the coefficient of $\bar{u}_H(r_n, s)$ are:

$$\begin{aligned} x_{1n} &= Ha^2 + R + \frac{r_n}{Re}, \\ x_{2n} &= 1 + G - \alpha - R - R\alpha^2 + 2R\alpha + x_{1n} + \alpha^2 x_{1n} - 2\alpha x_{1n} + Gx_{1n} - G\alpha x_{1n}, \\ x_{3n} &= \alpha + 2R\alpha^2 - 2R\alpha - 2x_{1n}\alpha^2 + 2\alpha x_{1n} + G\alpha x_{1n}, \quad x_{4n} = \alpha^2 x_{1n} - R\alpha^2, \\ x_{5n} &= 1 + \alpha^2 - 2\alpha + G - G\alpha, \quad x_{6n} = -2\alpha^2 + 2\alpha + G\alpha, \\ x_{7n} &= \frac{-x_{3n} + \sqrt{x_{3n}^2 - 4x_{2n}x_{4n}}}{2x_{2n}}, \quad x_{8n} = \frac{-x_{3n} - \sqrt{x_{3n}^2 - 4x_{2n}x_{4n}}}{2x_{2n}}, \\ x_{9n} &= \frac{x_{7n}^2 x_{5n} + x_{7n} x_{6n} + \alpha^2}{x_{7n} - x_{8n}}, \quad x_{10n} = \frac{x_{8n}^2 x_{5n} + x_{8n} x_{6n} + \alpha^2}{x_{8n} - x_{7n}}. \end{aligned} \quad (25)$$

The Laplace transform of the image function $\bar{u}_H(r_n, s)$ discussed in Equation (24) can be obtained by using the Robotnov and Hartley's functions:

$$LT^{-1} \left[\frac{1}{s^w + y} \right] = F_w(-y, t) = \sum_{n=0}^{\infty} \frac{(-y)^n t^{(n+1)w-1}}{\Gamma((n+1)w)}, \quad w > 0, \quad (26)$$

$$LT^{-1} \left[\frac{s^z}{s^w + y} \right] = R_{w,z}(-y, t) = \sum_{n=0}^{\infty} \frac{(-y)^n t^{(n+1)w-1-z}}{\Gamma((n+1)w-z)}, \quad \text{Re}(w-z) > 0, \quad (27)$$

$$\begin{aligned} \bar{u}_H(r_n, t) = & \frac{J_1(r_n)}{r_n} \left[\left(e^{x_{7n}t} - 1 \right) \left(\frac{A_0 x_{9n}}{x_{7n}} + \frac{x_{9n} \sin \phi}{x_{7n} F} \right) + \left(e^{x_{8n}t} - 1 \right) \left(\frac{A_0 x_{10n}}{x_{8n}} + \frac{x_{10n} \sin \phi}{x_{8n} F} \right) \right. \\ & \left. + A_1 x_{9n} e^{x_{7n}t} * \cos(\omega t) + A_1 x_{10n} e^{x_{8n}t} * \cos(\omega t) \right]. \end{aligned} \quad (28)$$

By inverting the finite Hankel transforms (i.e. Equation (28)) we obtain

$$u(r, t) = 2 \sum_{n=1}^{\infty} \frac{J_0(rr_n)}{r_n J_1^2(r_n)} \times u_H(r_n, t), \quad (29)$$

$$u(r, t) = 2 \sum_{n=1}^{\infty} \frac{J_0(rr_n)}{r_n J_1^2(r_n)} \left[\left(e^{x_{7n}t} - 1 \right) \left(\frac{A_0 x_{9n}}{x_{7n}} + \frac{x_{9n} \sin \phi}{x_{7n} F} \right) + \left(e^{x_{8n}t} - 1 \right) \left(\frac{A_0 x_{10n}}{x_{8n}} + \frac{x_{10n} \sin \phi}{x_{8n} F} \right) + A_1 x_{9n} e^{x_{7n}t} * \cos(\omega t) + A_1 x_{10n} e^{x_{8n}t} * \cos(\omega t) \right]. \quad (30)$$

The magnetic particle velocity can then be obtained from Equation (17):

$$\bar{v}(r, s) = \frac{s + \alpha - \alpha s}{s + Gs + \alpha - \alpha s} \bar{u}(r, s), \quad (31)$$

$$v(r, t) = x_{12n} (1 - x_{11n}) \left[u(r, t) * e^{x_{12n}t} \right], \quad 0 < \alpha < 1. \quad (32)$$

In Equations (30) and (32), $f * g$ represents the convolution product of f and g . The parameters introduced in Equation (32) are:

$$x_{11n} = \frac{1 - \alpha}{G - \alpha + 1}, \quad x_{12n} = \frac{\alpha}{G - \alpha + 1}. \quad (33)$$

The convolution product of f and g can be calculated as

$$(f * g)(t) = \int_0^t f(\tau) g(t - \tau) d\tau. \quad (34)$$

Results and discussions

The velocities of the fluid and the magnetic particles (Equations (30) and (32)) can be solved numerically by using the positive roots of the Bessel function J_0 . In the numerical computations, a series consisting of 500 terms was generated for solving the inverse Hankel transform equation. The following parameters were adopted while performing the simulation using Mathcad $A_0 = 0.5$, $A_1 = 0.6$, $G = 0.8$, $R = 0.5$, $Re = 4$, $\omega = \pi / 4$ and $Ha = 2$ (see [15]). The velocities of the blood and the magnetic particles against the r -direction were shown in Figures (2) – (11). The influence of the fractional order α , at various Hartmann numbers Ha , Reynolds numbers Re and inclination angles ϕ is studied in order to establish better understanding of the stated problem.

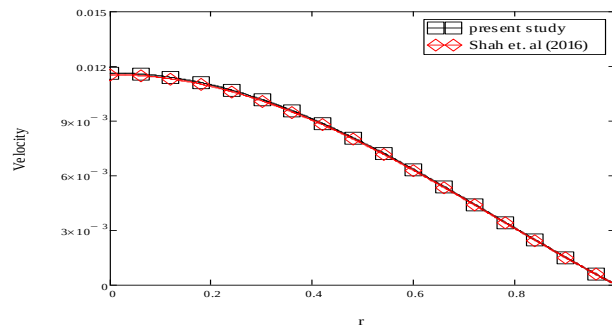
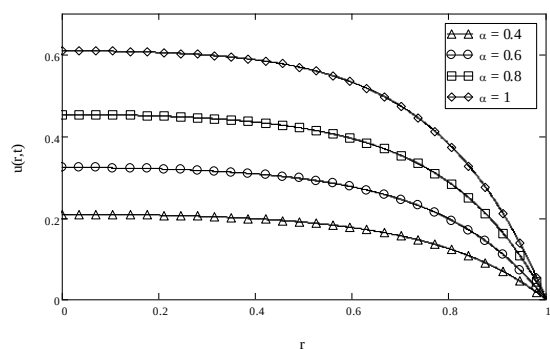
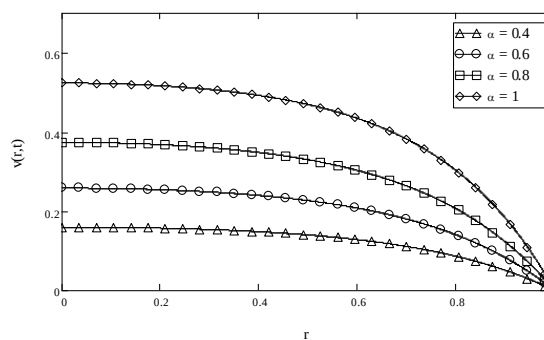


Fig. 2. Geometry figure of the magnetic blood flow

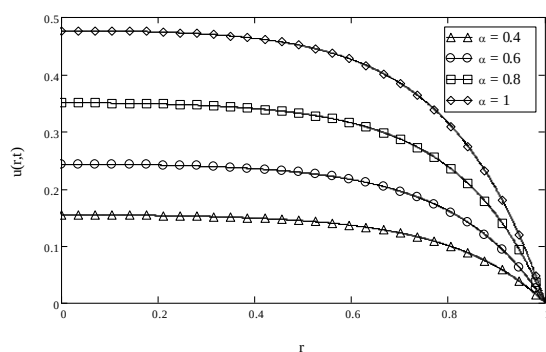


Blood flow velocity at $Ha = 2$

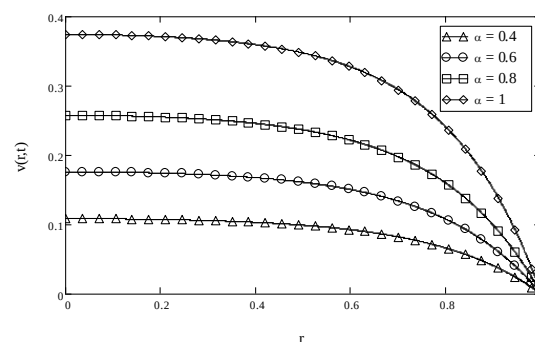


Magnetic particle velocity at $Ha = 2$

Fig. 3. Profiles of axial velocities $u(r,t)$ and $v(r,t)$ at different fractional parameters against r

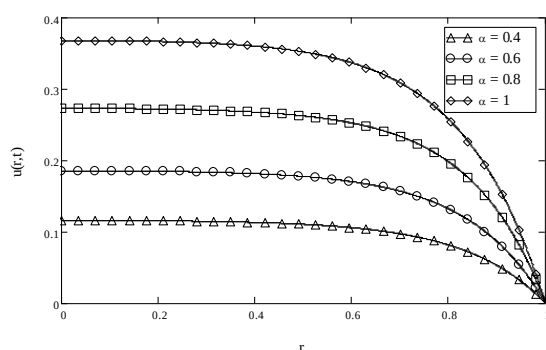


Blood flow velocity at $Ha = 2.5$

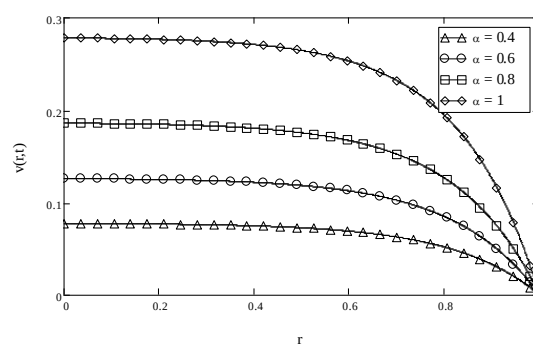


Magnetic particle velocity at $Ha = 2.5$

Fig. 4. Profiles of axial velocities $u(r,t)$ and $v(r,t)$ at different fractional parameters against r

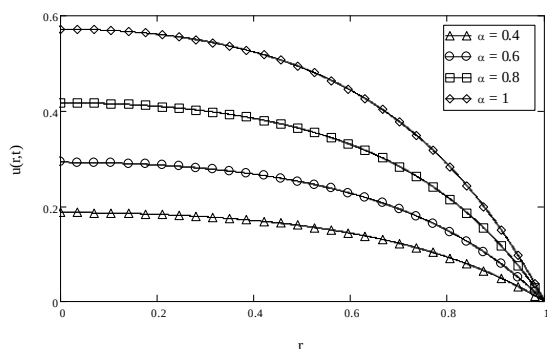


Blood flow velocity at $Ha = 3$

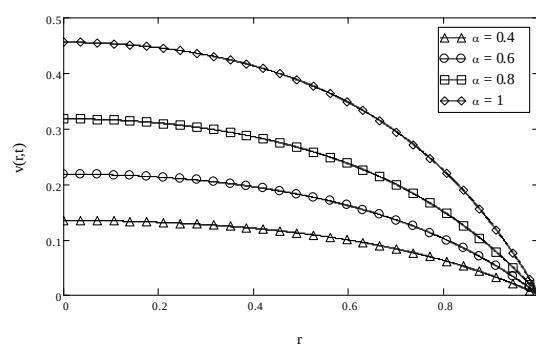


Magnetic particle velocity at $Ha = 3$

Fig. 5. Profiles of axial velocities $u(r,t)$ and $v(r,t)$ at different fractional parameters against r

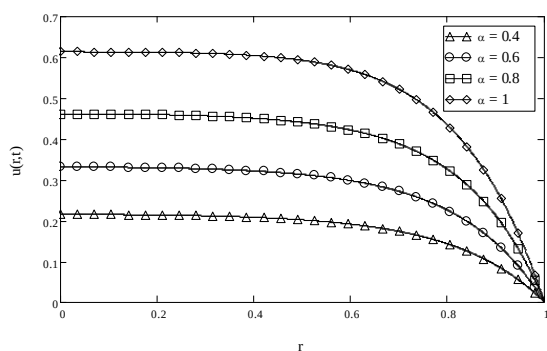


Blood flow velocity at $Re = 2$

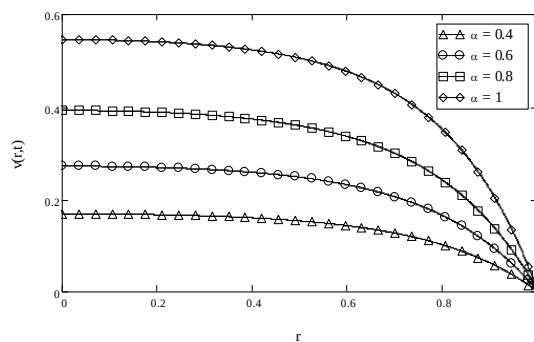


Magnetic particle velocity at $Re = 2$

Fig. 6. Profiles of axial velocities $u(r,t)$ and $v(r,t)$ at different fractional parameters against r

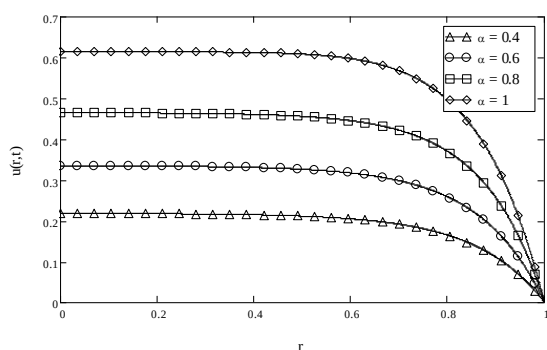


Blood flow velocity at $Re = 6$

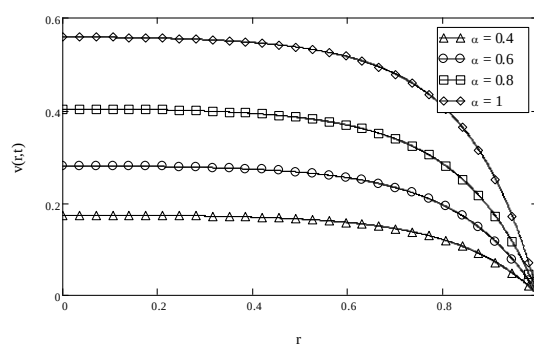


Magnetic particle velocity at $Re = 6$

Fig. 7. Profiles of axial velocities $u(r,t)$ and $v(r,t)$ at different fractional parameters against r

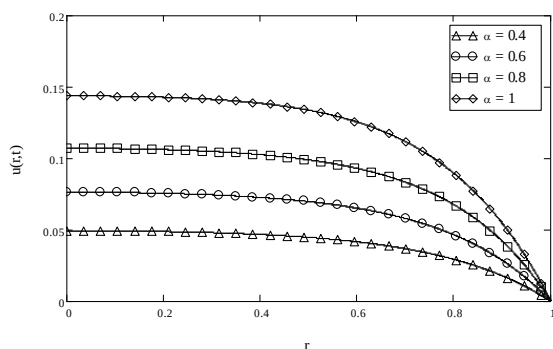


Blood flow velocity at $Re = 10$

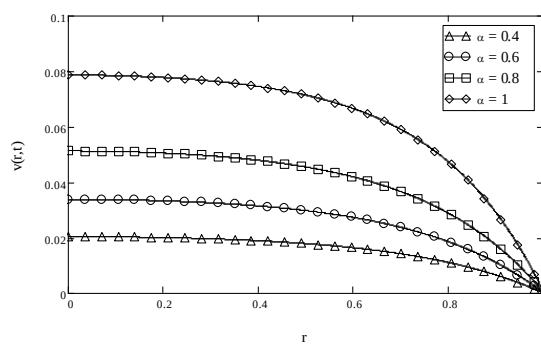


Magnetic particle velocity at $Re = 10$

Fig. 8. Profiles of axial velocities $u(r,t)$ and $v(r,t)$ at different fractional parameters against r

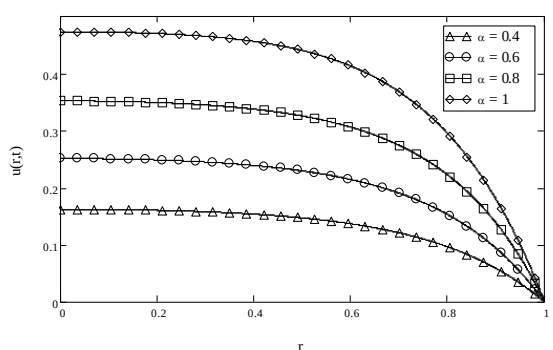


Blood flow velocity at $\phi = 0$

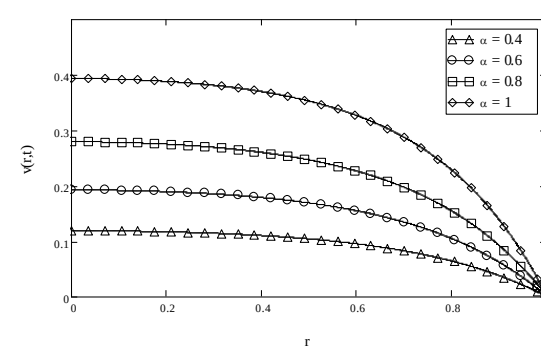


Magnetic particle velocity at $\phi = 0$

Fig. 9. Profiles of axial velocities $u(r,t)$ and $v(r,t)$ at different fractional parameters against r

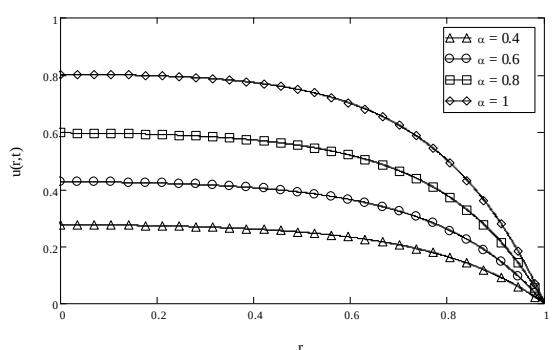


Blood flow velocity at $\phi = \frac{\pi}{6}$

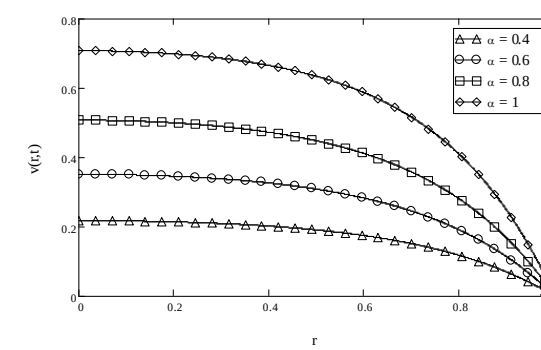


Magnetic particle velocity at $\phi = \frac{\pi}{6}$

Fig. 10. Profiles of axial velocities $u(r,t)$ and $v(r,t)$ at different fractional parameters against r



Blood flow velocity at $\phi = \frac{\pi}{2}$



Magnetic particle velocity at $\phi = \frac{\pi}{2}$

Fig. 11. Profiles of axial velocities $u(r,t)$ and $v(r,t)$ at different fractional parameters against r

To validate the current model, the results presented in Figure (2) were compared against those from [15] (by setting $\alpha = 1$ in Equation (32)). The agreement is encouraging.

Figures (3) – (5) show the effect of inclined magnetic field on the velocities of blood and magnetic particle. Seemingly, the velocities magnetic reduced appreciably as Hartmann number increases. Generally, the presence of magnetic field in the blood stream would retard the blood motion due to rise in resistive drag force. In order to identify the inclined magnetic field effect on both fluid and particle distribution, Figures (3) – (5) are depicted. It revealed that both velocities are appreciably reduced with increase in Hartmann number. Generally, the presence of magnetic field in the blood flow tends to slow down the velocity distributions due to rise in resistive force or Lorentz force. This finding agrees well with those found in [4]. This observation might be useful in some treatment processes.

Figures (3) – (8) show the effects of Reynolds number, Re on the velocities of blood and magnetic particles. In general, Re increases with respect to the velocities, implying that there is a drop in viscosity of the blood flow. This finding is in good agreement with those reported by [3]. The $u(r,t)$ and $v(r,t)$ profiles at different inclination angles ϕ were shown in Figures (9) – (11). As observed, the fluid and particle velocities increases in the inclined artery are higher than those in the non-inclined artery. It should be noted that the magnetic particle velocity is lower than the blood velocity due to the resistive force.

Conclusions

A mathematical model has been developed to analyse the blood flow in an inclined cylindrical artery subjected to the transverse magnetic field. The new definition of Caputo-Fabrizio fractional derivative was used to obtain the fluid and magnetic particle velocities. The governing non-dimensional fractional partial differential equation has been solved analytically by using both Laplace (with respect to t) and finite Hankel transforms. Based on the numerical results, the blood and magnetic particles velocities decrease when Hartmann number increases. Nevertheless, the velocities increase at increasing Reynolds number. The numerical results show that the inclination angle has a significant impact on the blood and magnetic particle velocities. This finding might be useful in the diagnosis and therapeutic treatment of some medical problems. Also the current findings could lead to the designs of better pads and machines. In order to obtain a better form of protective pads and machines.

Acknowledgement

This research was funded by a grant from Research Management Centre University Tun Hussein Onn Malaysia under Tier 1/H072 grant.

References

- [1] C. J. Pao, V. M. Haik. Y, Chen, "Bio-Magnetic Fluid Dynamics," Fluid Dyn. interfaces, pp. 439–452, 1999.
- [2] E. E. Tzirtzilakis, "A mathematical model for blood flow in magnetic field," Phys. Fluids, vol. 17, no. 7, pp. 1–15, 2005.
- [3] C. D. K. Bansi, C. B. Tabi, T. G. Motsumi, and A. Mohamadou, "Fractional blood flow in oscillatory arteries with thermal radiation and magnetic field effects," J. Magn. Magn. Mater., vol. 456, pp. 38–45, 2018.
- [4] Y. Haik, V. Pai, and C. Chen, "Development of magnetic device for cell separation," vol. 194, pp. 254–261, 1999.
- [5] C. Das, G. Wang, and F. Payne, "Some practical applications of magnetohydrodynamic pumping," Sensors Actuators, A Phys., vol. 201, pp. 43–48, 2013.
- [6] R. Bali and U. Awasthi, "Mathematical Model of Blood Flow in Small Blood Vessel in the Presence of Magnetic Field," Appl. Math., vol. 2, no. 2, pp. 264–269, 2011.

- [7] M. K. Sharma, K. Singh, and S. Bansal, "Pulsatile MHD Flow in an Inclined Catheterized Stenosed Artery with Slip on the Wall," *J. Biomed. Sci. Eng.*, vol. 07, no. 04, pp. 194–207, 2014.
- [8] M. Hatami, J. Hatami, and D. D. Ganji, "Computer simulation of MHD blood conveying gold nanoparticles as a third grade non-Newtonian nanofluid in a hollow porous vessel," *Comput. Methods Programs Biomed.*, vol. 113, no. 2, pp. 632–641, 2014.
- [9] R. L. Bagley and P. J. Torvik, "A Theoretical Basis for the Application of Fractional Calculus to Viscoelasticity," *J. Rheol. (N. Y. N. Y.)*, vol. 27, no. 3, pp. 201–210, 2002.
- [10] H. Bakhti, L. Azrar, and D. Baleanu, "Pulsatile blood flow in constricted tapered artery using a variable-order fractional Oldroyd-B model," *Therm. Sci.*, vol. 21, no. 1A, pp. 29–40, 2017.
- [11] M. I. Asjad, N. A. Shah, M. Aleem, and I. Khan, "Heat transfer analysis of fractional second-grade fluid subject to Newtonian heating with Caputo and Caputo-Fabrizio fractional derivatives: A comparison," *Eur. Phys. J. Plus*, vol. 132, no. 8, 2017.
- [12] A. Zaman, N. Ali, and M. Sajid, "Slip effects on unsteady non-Newtonian blood flow through an inclined catheterized overlapping stenotic artery," *AIP Adv.*, vol. 6, no. 1, 2016.
- [13] R. Agarwal and N. K. Varshney, "Pulsatile flow of Herschel-Bulkley fluid through an inclined multiple stenoses artery with periodic body acceleration Raja," vol. 7, no. 3, pp. 102–113, 2016.
- [14] K. Ramesh and M. Devakar, "Effect of heat transfer on the peristaltic transport of a MHD second grade fluid through a porous medium in an inclined asymmetric channel," *Chinese J. Phys.*, vol. 55, no. 3, pp. 825–844, 2017.
- [15] N. Ali Shah, D. Vieru, and C. Fetecau, "Effects of the fractional order and magnetic field on the blood flow in cylindrical domains," *J. Magn. Magn. Mater.*, vol. 409, pp. 10–19, 2016.

NATURAL CONVECTION HEAT TRANSFER IN HYBRID RECTANGULAR-TRIANGULAR ENCLOSURES WITH ISOTHERMAL VERTICAL HEATING

K. M. Fathurrahman¹,
C. S. Nor Azwadi²,
M.R. Nor Mahani¹

¹Department of Mechanical Engineering, Politeknik Sultan Abdul Halim Mu'adzam Shah, 06000, Bandar Darulaman, Jitra, Kedah, Malaysia

²Department of Mechanical Engineering, Malaysia-Japan International Institute of Technology (MJIT), Universiti Teknologi Malaysia (UTM), Jalan Sultan Yahya Petra, 54100 Kuala Lumpur

Abstract: A numerical analysis of the natural convection heat transfer inside a hybrid shape rectangular-triangular enclosure under various local heating intensity has been studied to measure the temperature distribution rate achieving target average temperature, T_{avg} of 335K within specific transient time range, τ_s from 60 to 120 seconds for specific design requirements. A vertical heater with a constant surface temperature at 370K placed on the partition between the cavities, facing the hypotenuse wall and immersed in the air with the initial ambient temperature at 300K with Prandtl number, $Pr = 0.7$. Three heater length (λ), varies at 100 mm, 200 mm and 300 mm with the Rayleigh number, $Ra = 106 \sim 107$. The rectangular enclosure with dimension of 420 mm (L) x 400 mm (H) x 400 mm (W) attached to a triangular enclosure with 13° top angles (ϕ), 400 mm (H) and 92 mm (L), which having a constant base surface temperature at 300K. The other inner sidewalls were adiabatic. Both cavities share an upper and bottom 20 mm airway to allow air circulation between two cavities regions, initiated by the air density difference from heat transfer. The governing equations solved using the Finite Volume Method. The flow and energy equations are modelled using the Realizable $k-\epsilon$ model in FLUENT CFD software. From the isotherms visualization, applying λ at 300 mm demonstrated the fastest transient time to achieve average temperature distribution at 335K within 50 seconds. The slowest time recorded when λ at 100 mm applied with transient time of 259 seconds. The analysis proceeds by observing variations in the isotherms with respect to the different Ra of the local heat source and found that when the heater length increased, heat transfer rate enhanced due to heat intensity increment. Results suggest the use of heater length of 200 mm, with heating capability to reach 335K within 90 seconds for the optimum design consideration.

Keywords: Natural convection; transient; vertical heating; hybrid enclosure; Realizable $k-\epsilon$

Introduction

Natural convection is a preferable selection mode of heat transfer in many engineering applications due to its cost effectiveness for engine room cooling, electronics cooling, building's ventilation, nuclear reactors, solar dryer, as well as thermal food container technology [1]. Natural convection heat transfer occurs when fluid motion is generated by buoyancy force due to density variations of fluids with temperature[2]. There are increasing numbers of studies on natural convection behaviors inside various geometrical shapes of enclosures. Some recent studies reported on the rectangular enclosures [3-5], triangular enclosures [6-7], cylindrical enclosures [8-9], concave enclosures [10] and other various shapes enclosures [11-12].

Natural convection in an enclosure varies based on boundary conditions, geometrical aspects and the fluid flow area as mentioned by Karatas *et al.* [13]. The heat transferred into the enclosure could be clustered as conduction, convection and radiation. Serth *et al.* [14] summarized that the fluids in the gaseous or liquid phase transfer or absorb the heat once it flows passing a surface with a temperature different value by itself. For the natural convection, fluid move by the buoyancy forces created from the density differences within the fluids. Buoyancy is due to the combined presence of a fluid density gradient and a body force that is proportional to density. In practice, the body forces is usually gravitational.

The vertical heating improve the convective heat transfer to adjacent cold wall. Kaluri *et al.* [15] concluded that the distributed heating is more energy-efficient method than heating from the bottom wall surface. The convective heat fluxes depend on the temperature differences and flow pattern of the fluid [16]. The heat flow behaviors of the natural convection of air in an enclosures investigated by Liu *et al.* [17] and Staats and Brisson [18] for design optimization.

Mariani *et al.* [19], concluded that the difference in temperature between the walls impacting the natural convection and measured by a Rayleigh number (Ra). The observation made under variations of the isotherms and streamlines at different Ra , aspect ratios and of the local heat source placement. The findings revealed that any changes in the parameter values influenced the value of Nu on both walls. Different flow patterns observed, and literally, the convection can be controlled by the local heating.

Several numerical studies focused on flow transition from the conductive state to convection due to buoyancy force. Ceng *et al.* [20] conducted a study to understand the transient behaviour of a buoyancy periodic flow in a cavity. By implementing the finite volume control method, they predicted the periodic flow and heat transfer characteristics on different cross-sectional shaped lid-driven cavities. Uniform temperature in the cavity achieved influenced by vortex formation and re-circulation cells help to enhance mixing [21]. The temperature might have an increment or reduction and wall surface heat transfer could be improved by altering the hydrodynamics blockages on boundary layers inside the flow regimes near the active walls [22].

The buoyancy driven flow field inside a cubic enclosure investigated numerically under various of Ra numbers and extensively reported by preveous researchers [23-27]. Recently, Cianfrini *et al.* [28] studied on natural convection of air inside a tilted square enclosures cavity. Two adjacent walls cooled and the other two opposite walls heated in order to investigate the impact of the cavity inclination on the convection transfers and on the nature of the heat flow. A computational model was used to calculate the mass, momentum, and energy transfer governing equations for different Ra in the range of $10^4 \leq Ra \leq 10^6$, and of the tilting angle in the angle range $0^\circ \leq \gamma \leq 360^\circ$.

A rectangular channel shaped solar collector studied by Gareh [29] to investigate the phenomenon of the laminar flow inside the enclosures by using the Finite Volume Method (FVM) to solve the mathematical formulation of the Navier-Stokes with the schematic SIMPLE programming and thermal energy equation. Tang *et al.* [30] applied a control volume-based finite difference technique in his study of combined heat and moisture transport in an enclosure with free ports. They concluded that the moisture transport and convective heat flow along with horizontal ports transport rates were greatly rely on the properties of porous media.

A comprehensive study on buoyancy driven flow in air filled tilted square cavities has been studied by Bairi [31]. Both the cold and hot walls inside the cavity temperatures maintained at T_c , and T_h respectively and the channel of the cavity were kept adiabatic. A number of analysis and simulations were carried out for various physical and thermal variations. The finite volume method implemented in solving the momentum, mass, and energy transfer governing

equations. The Rayleigh number ranged at 10^3 to 10^{10} . Nu_{avg} - Ra correlations proposed for sizing structures for different angles inclination varies from 0° to 360° .

Avedissian *et al.* [32] performed a numerical study to investigate the influence of Rayleigh number, aspect ratio, and blind geometry on free convection in a tall vertical enclosure with an internal louvered metal blind. Stavrakakis *et al.* [33] has carried out a numerical simulation of the airflow inside enclosures that contains a local heat source to investigate burning room model. The two turbulence models were selected for his study; the *standard* $k-\varepsilon$ model and the *RNG* $k-\varepsilon$ model. Calculated utilizing the CFD results, their findings reveals that both predictions are similarly performed on the bi-directional flow existing through the burning-room vent models.

Saha [34] investigated the fluid flow pattern and the heat transfer characteristics inside a triangular enclosure heated instantaneously on the inclined walls. The wall heating process was categorized as a start-up stage, a transitional stage and a steady state stage. Studies on natural convection heat transfer within triangular enclosures are quite attractive and have been attended by many researchers in different conditions. Through experimental and numerical analysis by Yesiloz and Aydin [35], a natural convection in a right-angled triangular cavity was investigated. The numerical analysis obtained by a finite volume method applied to visualized flow where the particle tracing method involved. The effects of Rayleigh number, Ra on the Nusselt number, Nu as well as temperature and velocity fields investigated with Ra ranged at 10^3 to 10^7 . Their numerical and experimental and finding results met an agreement with a correlation for Nu developed.

Saha and Gu [36] worked on an isosceles triangular enclosure with uniform base heating and stratified cooling on the inclined surfaces. Since the two surfaces were at different temperature, the flow stream being unstable. Transient flow observed in the enclosure clustered into three distinct stages, including an early stage, a transitional stage, and a steady state. They revealed that the Rayleigh number, Ra , and significantly impact the flow inside the enclosure. The governing equations applied with Boussinesq assumption.

Since this study considering the use of a vertical isothermal heating, it is crucial to understand the principle for design approach. Pantokratoras [37] conducted a study on natural convection over a flat vertical isothermal plate. For the natural convection in his case studied, they found that the velocity is zero at the heated body (no-slip boundary condition), then increase rapidly in a thin boundary layer adjacent to the body before back to zero once more when travelling far from the body surface. Meanwhile, Vogel *et al.* [38] in his study found that the heat transfer enhancement due to natural convection increases with a vertical heater geometrical setup.

Pitts and Sissom [39] stated, that any vertical surface exposed to a cooler (or warmer) surrounding fluids, then natural convection will occur. The boundary layer development adjacent to a heated vertical plane surface. From the illustration, the boundary layer is laminar at the outset; but the transition to the turbulent flows occurs at some distance from the leading edge, based on the fluid properties and thermal gradient. The transition from laminar to turbulent occurs when the Ra number equals approximately 10^9 .

The Rayleigh number gained from Grashof-Prandtl number, where $Ra = Gr_L Pr$, with Gr_L written as follows [39]:

$$Gr_L = \frac{g\beta(T_s - T_\infty)L^3}{\nu^2} \quad (2.1)$$

In (2.1) β is the coefficient of volume expansion and defined by

$$\beta = -\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T} \right)_p = \frac{1}{\rho} \frac{p}{RT^2} = 1/T_f \quad (2.2)$$

The Nusselt number, Nu , represents the ratio of convective to conductive heat transfer in a fluid over a characteristic length, L . The Nusselt number for free convection is the function of the Grashof and Prandtl numbers. An empirical correlation in engineering data for both laminar and turbulent regimes, correlate well for many geometric configurations in a single equation, written as [39]:

$$\overline{Nu} = \frac{\bar{h}}{k} L = C (Gr_L Pr)^a \quad (2.3)$$

where,
for vertical plate configuration;

constant for laminar : $C = 0.59$, $a = 1/4$

constant for turbulent : $C = 0.13$, $a = 1/3$

L : the characteristic length of vertical dimensions

All fluid properties evaluated at the mean film temperature,

$$T_f = (T_s + T_\infty) / 2.$$

In summary, in most studies, the study of natural convection heat transfer in rectangular and triangular cavities has received significant attention. Most of reviewed analysis studies focus on single shape enclosure and still lack of diversity in hybrid shape configurations to explain the thermal distribution regarding with the final retaining temperature inside specific enclosures. Alternatively, applying hybrid shaped enclosure promising better approach to counter the design challenge and having the potential to improve the temperature distribution rate using adaptive numerical methods from previous literatures. However, there is still a lack of studies reporting work on the natural convection heat transfer inside a hybrid rectangular-triangular enclosure.

It seems that the literature on temperature distribution in enclosures is surprisingly scarce in terms of numerical problems with hybrid rectangular-triangular geometrical shape with internal heat source. There is still missing information regarding with the heat flow visualization on the flat vertical heating surface when the buoyancy driven flow applied when interconnected with othe cavity. Limited data on the heat intensity and arrangement variations also made the literature void for the condition to be set up in an engineering challenge influencing a thermal distribution. Therefore, a study of the temperature distribution inside a hybrid rectangular-triangular with vertical partitioning heating with buoyancy driven flow (natural convection) carried out to fill the gap.

Methodology

The present work is a numerical analysis of the natural convection heat transfer inside a hybrid shape rectangular-triangular enclosure under the various local heating intensity. The aim was to measure the temperature distribution rate achieving target average temperature, T_{avg} of 335K within specific transient time range, τ_s from 60 to 120 seconds for specific design requirements.

Geometrical and Computational Domain

The geometrical domain of the system consisting a rectangular cavity with 420 mm (L) x 400 mm (H) x 400 mm (W), attached with a triangular enclosure sized 400 mm (H) and 92 mm (L), with top angles (ϕ), at 13. A vertical heater with a constant surface temperature at 370K placed on the partition's surface between the cavities, facing the hypotenuse wall and immersed in the air with the initial ambient temperature at 300K with Prandtl number, $Pr = 0.7$. Three heater length (λ), varies at 100 mm, 200 mm and 300 mm with the Rayleigh number, $Ra = 10^6 \sim 10^7$. Both cavities share an upper and bottom 20 mm ventilation opening and share a partitioning wall as shown in Figure 1 and Figure 2 below. The heater attached facing triangles' hypotenuse wall.

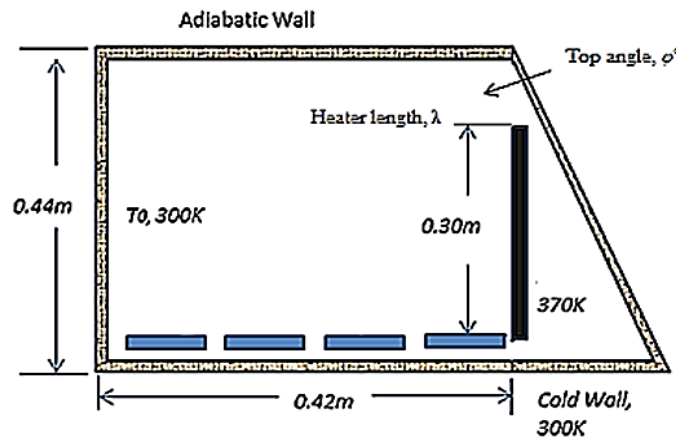


Figure 1.0 Geometrical of the hybrid rectangular-triangular enclosure

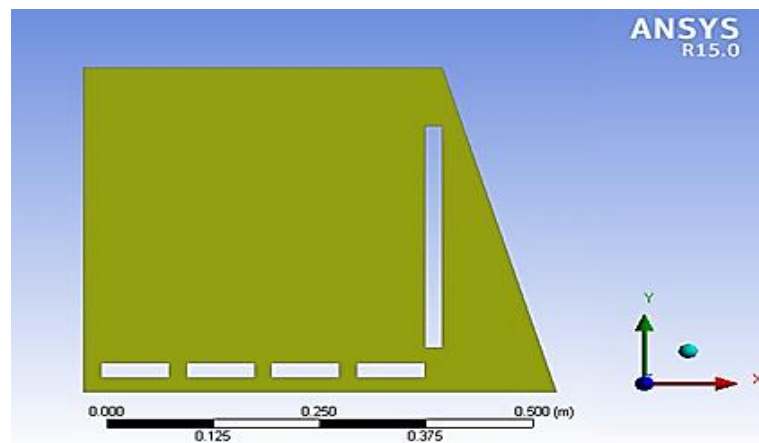


Figure 2.0 Computational domain of hybrid rectangular-triangular enclosure

Governing Equation and Boundary Condition

The working medium is considered incompressible, two-dimensional and satisfying the *Boussinesq approximation*. The buoyancy effects are confined to the momentum equation. The continuity, energy and x-momentum equations for a vertical flat plate simplified follows [39]:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (2.4)$$

$$u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \alpha \frac{\partial^2 T}{\partial y^2} \quad (2.5)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = g\beta(T - T_\infty) + \nu \frac{\partial^2 u}{\partial y^2} \quad (2.6)$$

Boundary conditions for an isothermal plate:

$$\text{at } y = 0: \quad u = 0, v = 0, T = T_s$$

$$\text{at } y = \infty: \quad u = 0, T = T_\infty, \frac{\partial u}{\partial y} = 0, \frac{\partial T}{\partial y} = 0$$

Solver: The Realizable k-ε Turbulence Model

Numerical calculation carried out using the Realizable k-ε by model proposed by Shih et al. [40] for solving the mathematical equations in (2.7) – (3.2). The realizable k-ε turbulence model applied having a new introduction of eddy-viscosity formula that involving a variable Reynolds and amodel equation representing dissipation (ε) based on the dynamic equation of the mean-square vortices fluctuation. Compared to the standard k-ε model and RNG k-ε model, the realizable k-ε correctly predicts the flow in round jets, well suited for swirling flows and the flows involving separation. The transport equations for k and ε in the *Realizable k-ε* modeled as [40]:

$$\frac{\partial}{\partial x}(\rho k) + \frac{\partial}{\partial x_j}(pku_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_m + S_k \quad (2.7)$$

$$\frac{\partial}{\partial x}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(p\varepsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} G_b + S_\varepsilon \quad (2.8)$$

Where

$$C_1 = \max \left[0.43, \frac{\eta}{\eta + 5} \right], \eta = S \frac{k}{\varepsilon}, S = \sqrt{2S_{ij}S_{ij}}$$

According to the equations, G_k is the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy, Y_m is the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate, and C_1 and C_2 are constants.

The terms of σk and $\sigma \varepsilon$ refer to the turbulent Prandtl numbers for k and ε respectively, with user-defined sources represented by as S_k and S_ε . The eddy viscosity for k-ε models is computed by combining k and ε as follow [40]:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (2.9)$$

$$C_\mu = \frac{1}{A_0 + A_S \frac{kU^*}{\varepsilon}} \quad (3.0)$$

$$U^* = \sqrt{S_{ij}S_{ij} + \tilde{\Omega}_{ij}\tilde{\Omega}_{ij}} \quad (3.1)$$

where;

$$\tilde{\Omega}_{ij} = \Omega_{ij} - 2\epsilon_{ijk}\omega_k \text{ and } \Omega_{ij} = \overline{\Omega_{ij}} - \epsilon_{ijk}\omega_k$$

Ω_{ij} is the mean rate-of-rotation tensor viewed in a moving reference frame with the angular velocity, ω_k . The model constants A_0 and A_s are given by:

$$A_0 = 4.04, \quad A_s = \sqrt{6}\cos\phi$$

$$\phi = \frac{1}{3}\cos^{-1}[\sqrt{6}W], W = \frac{S_{ij}S_{jk}S_{ki}}{\tilde{S}^3}, \tilde{S} = \sqrt{S_{ij}S_{ij}}, S_{ij} = \frac{1}{2}\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right) \quad (3.2)$$

The constant C_μ is a function of the mean strain and rotation rates represent the angular velocity of the system rotation, and the turbulence fields (k and ϵ). C_μ standard value is 0.09 for an inertial sublayer in an equilibrium boundary layer and constants are $C_1\epsilon=1.44$, $C_2=1.9$, $\sigma_k=1.0$ and $\sigma_\epsilon=1.2$.

Based on the unique cases studied (hybrid enclosures), the numerical procedure adapted from Saha and Gu [36] as a benchmarking. The governing equations of (2.4) – (2.6) listed in previous literatures were calculated conditions using the SIMPLE algorithm scheme. In order to discretize the equations, the finite volume method scheme selected. For an advection term approximation, the QUICK scheme selected. Second order accurate has been chosen for the diffusion discretization. The second order upwind for momentum and kinetics selected, while PRESTO scheme used for pressure discretization. Iterations residuals set at 10^{-6} for convergence. The air filled inside the cavity was treated as a two dimensional, with laminar and at a steady state. The air properties were assumed to be constant. An exceptional applied on the buoyancy term in the momentum equations based on Boussinesq approximation. The viscous dissipation neglected in the radiation and momentum term to reduce instability solution [41].

Meshing and Validation

The grid independence test has been conducted on the numerical scheme to evaluate the mesh quality for ensuring the results accuracy. Solutions have been obtained on three non-uniform grid patterns from coarser to finer size of 5613 elements, 8940 elements and 10826 elements. The average Nusselt number, Nu_{avg} has been measured number along the left cold wall in the rectangle cavity (for the test, the wall was set to be isotherm at 300K). The heater having a constant temperature at 370K with the length, $\lambda = 0.3$ m at $Ra = 10^7$. The value that has been predicted using the mesh selection was compared in Figure 3.0.

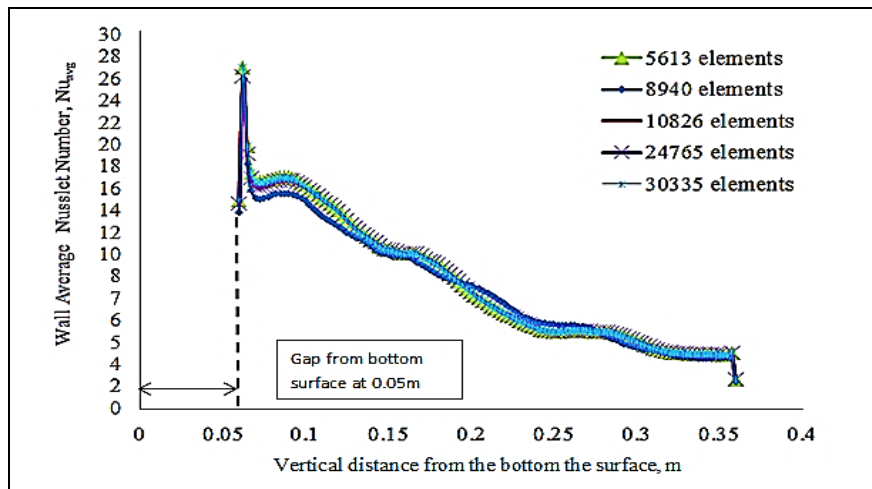


Figure 3.0 The comparison between the Nu_{avg} along the cold surface at $Ra = 10^7$

From the comparisons, the percentage of differences in the value obtained between of 5613 elements and that of 8940 elements was 1.95%. Meanwhile between the 8940 elements and 10826 elements was only 0.03 %. Using smaller size, the incremental not significantly influence the Nu_{avg} number. Hence, 8940 elements of mesh have been used in all the simulation in this study.

Table 1

Comparison of $Rk-\epsilon$ and $k-\epsilon$ turbulence model performance.

Parameters	Santosh [42]	Present $k-\epsilon$ model	Present $Rk-\epsilon$ model	Error, % $k-\epsilon$ model	Error, % $Rk-\epsilon$ model
Nusselt, Nu_{avg}	2.188	2.012	2.170	3.91	0.82
T-max, K	363	342	363	5.79	0.00
T-mid1, K	353	331	352	6.23	0.23
T-mid2, K	343	321	342	6.41	0.29
T-min, K	333	312	331	6.31	0.60

From Table 1, the predictions from the present numerical setup using $Rk-\epsilon$ turbulence model indicated smaller errors (less than 1.0%) as compared with Santosh results for Rayleigh number, $Ra = 10^4$. Meanwhile, applying standard $k-\epsilon$ model for case studied gain more than 5% of error. From the comparison of computing results, a good agreement with the benchmark data achieved when $Rk-\epsilon$ turbulence model is applied. This validation gives a good confidence in the current numerical code.

Results and Discussion

Observation on the numerical simulations of the transient flow development classified into three distinct stages; an early stage, a transitional stage and a quasi-steady state as studied by Saha and Gu [36]. Analysis done for the early simulation time, τ_s , range at $0 < \tau_s < 30$ s, the transitional stage range at $30 \leq \tau_s < 180$ s and the quasi-steady state range at $180 \leq \tau_s < 300$ s. The effect of parameters variation and discussion of the temperature distribution of the three

phases summarized for overall performance of cases studied. The initial condition at $\tau_s = 0$ assumed to be no heat transfer calculation solved.

Early Stage Transient Flow

Initially, at transient time $\tau_s = 0$ s, the air contained in the entire cavity is quiescent and does not generate heat internally. The air in the both rectangular and triangular enclosures was initially isothermal and stationary at 300K. At the start-up of the simulation, 370K isothermal conditions are presented over the flat vertical heater surface on vertical wall inside the triangular cavity. With 300K isothermal bottom triangular surface, it creates a tendency for fluid motion to find their way since greater temperature gradient influencing density differences of air.

At $0 < \tau_s < 5$ s, the evolution of heat transfer stage in all cases most dominated by conduction. For example, at $\tau_s = 2$ s, in most cases; the vertical boundary layer had started to growth. The corresponding isotherms are parallel to the hot wall, indicating that the heat energy is initially transferred from the hot wall surface only by conduction. This condition meet the agreement with works by Saha [34]. The heater on vertical wall maintains 370K introduces stronger thermal gradient between hot wall and air as time progresses.

From $5 < \tau_s < 30$ s, the heated air from the vertical heater surface moves through the boundary layer reaching to the top aperture point. The air density changed, where more dense air starts fall along the adiabatic inclined wall, creating clockwise rotating from the top down to the bottom by gravitational forces inside triangular cavity. On the other hand, the less dense hot air with higher momentum inside the triangle cavity rise and accumulated at the top portion on cavity, just before entering the rectangular cavity as visualized in Figure 4.0 under variation of the Ra .

Transitional Stage Flow

When $30 \leq \tau_s < 180$, the air motion gains momentum and convective mode of heat transfer dominates the temperature distribution. From this moment, the boundary layer grows in size since it received heat from heated wall. Further progress in time makes the clockwise rotating flow occurs in the triangular cavity. This transitional stage is characterized by the appearance of convective instabilities as a form of rising and sinking plumes.

The horizontal thermal boundary layer is formed in the triangle cavity due to heating on the side surface, which is potentially unstable since the lower portion receiving upcoming hot air from rectangular cavity.

Alongside the inclination wall, the boundary layers get distorted slightly due to growth in volume of recirculate hot air from bottom ventilation. Thus, promotes chimney effect and buoyancy forces gain momentum to distribute more air to the rectangular cavity creating air circulation between two cavities. The chimney effect influenced by the temperature gradient found in this study demonstrate a similar trend with previous works by Pinard et.al [43], Tu et.al [44] and Zhai et.al [45]. The thermal boundary layer becomes unstable as soon as the critical condition is met.

As a result, the hot air plume start to enter the rectangular enclosure and the cold air has drawn to the bottom surface. As time progress the flow between two regions becomes stronger and eventually asymmetric as a result rotating flow. It is interesting to observe that temperature distribution gain for faster towards a quasi-steady state.

Due to the occurrence of the convective instabilities the observation made until the temperature distribution inside the rectangular cavity achieved minimum 335K and simulation time recorded. The temperature distributions obtained from the numerical results shown in Figure 5.0.

Quasi-Steady State Flow

When $180 \leq \tau_s < 300$ s, it is seen in the studied regime that continuous heating within the enclosure rise up the whole portion of the cavity in time. Temperature in both cavities almost distribute evenly after equilibrium state reached. From the early stage to the transitional stage, the continuous movement of hot air to occupy the colder portion creates circulation flow between two cavities. The results obtained are shown in Figure 6.0. The interaction between hot and cold air lead to create equilibrium state with the air and almost stagnant. As a consequence, circular heat flow motion almost minimal.

During the quasi-steady state condition, it found that achieving steady state condition is quite challenging and hard to reach steady state condition since two walls with different surface temperature interact each by fluids movement. The results similar with studied by Sheu and Lin [46] in bifurcation scenario due to the buoyancy-driven flow natural convection in a cavity. In the transitional stage that the dominant plume which is formed at the edge of aperture vent affecting the temperature stratification in rectangular cavity. However, before getting steady state, air pushing downside of the cavity since there is a colder area created by a constant colder plate surface at the bottom of triangular cavity. The occurrence of the convective instability results in a number of small secondary convective cells in addition of the large dominant cell in the enclosure.

The value of final quasi-steady state temperature much affected by heat intensity and find the agreement with previous study by Zamora and Sancho [46] which remarked that the transient evolution of the flow is strongly affected by heating intensity. The list sorted by the fastest timing to achieve average temperature distribution, T_{avg} at 335K inside the rectangular cavity at various Ra . The rest of numerical results obtained shown in Table 2. The simulation time taken for all parameters combination and configurations simplified as shown in Table 3.

Table 2

Simulation time, τ_s at corresponding Ra , and Nu_{avg} on the entire heater length surface when achieving T_{avg} at 335K.

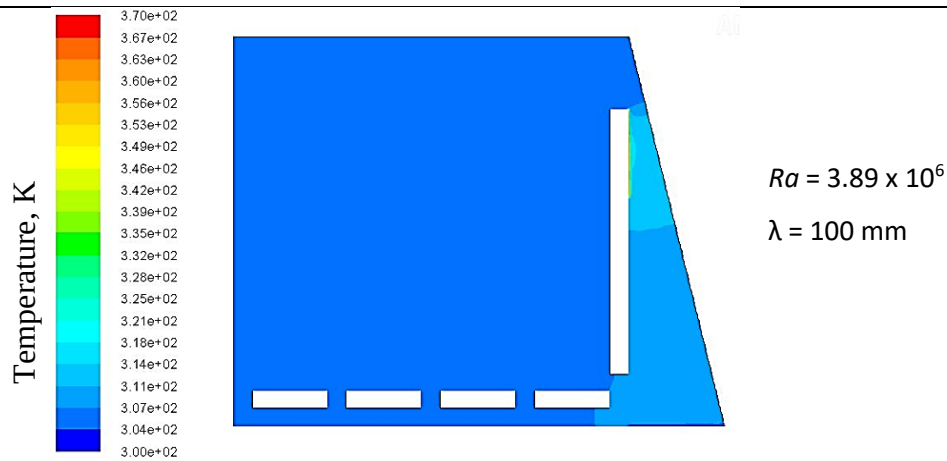
Simulation Time, τ_s s	Top angle, φ°	Heater length, λ mm	Rayleigh number, Ra	Average Nusselt number, Nu_{avg}	Convection coefficient, h_{avg} , W/m ² .K
50	13	300	1.05×10^8	59.7	59.7
90	13	200	3.10×10^7	44.0	6.3
259	13	100	3.89×10^6	26.2	7.6

Table 3

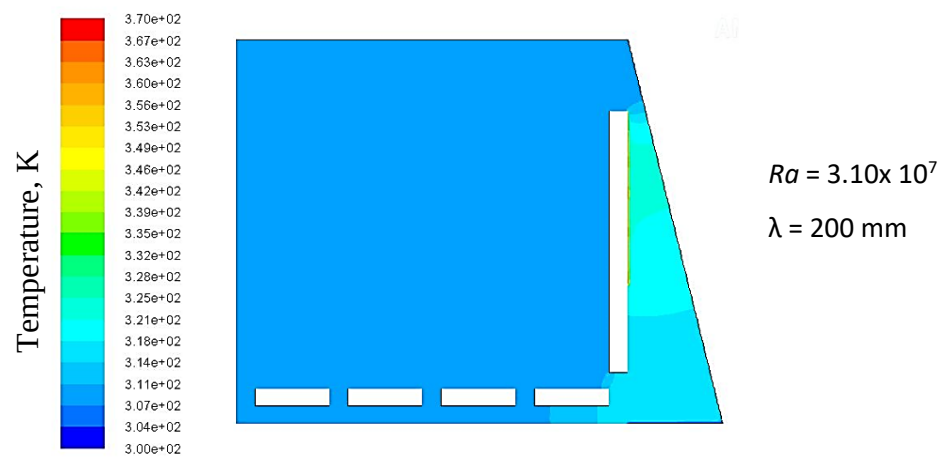
Values of averaged temperature distribution, T_{avg} inside the cavity at $\varphi = 13^\circ$.

Triangle Top angle: $\varphi=13^\circ$	
Heater length,	Transient time, τ_s sec.

λ mm	0	2	5	30	50	60	90	120	150	180	240	300
100	300	302	305	317	321	322	325	327	329	330	333	338
200	300	304	308	326	330	332	335	338	340	343	348	352
300	300	306	311	332	338	339	343	345	348	351	355	359



Contour of Temperature (K)



Contour of Temperature (K)

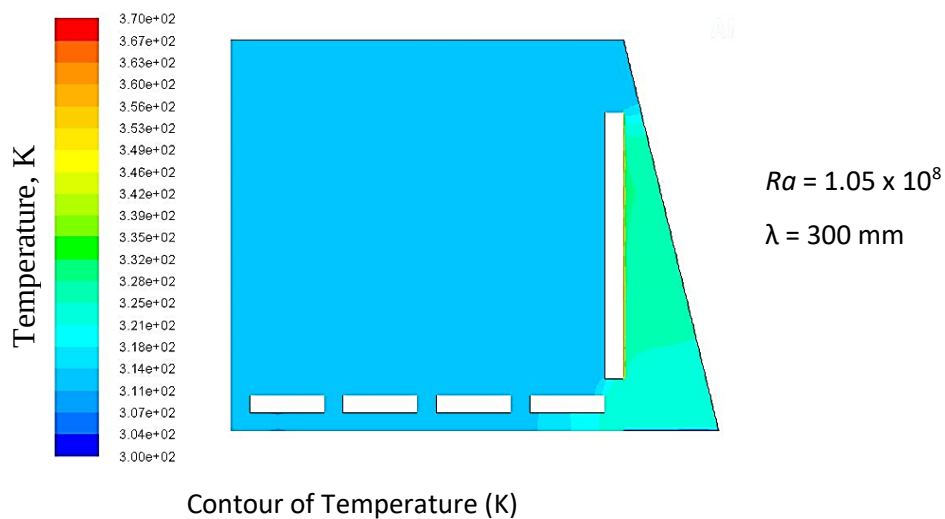
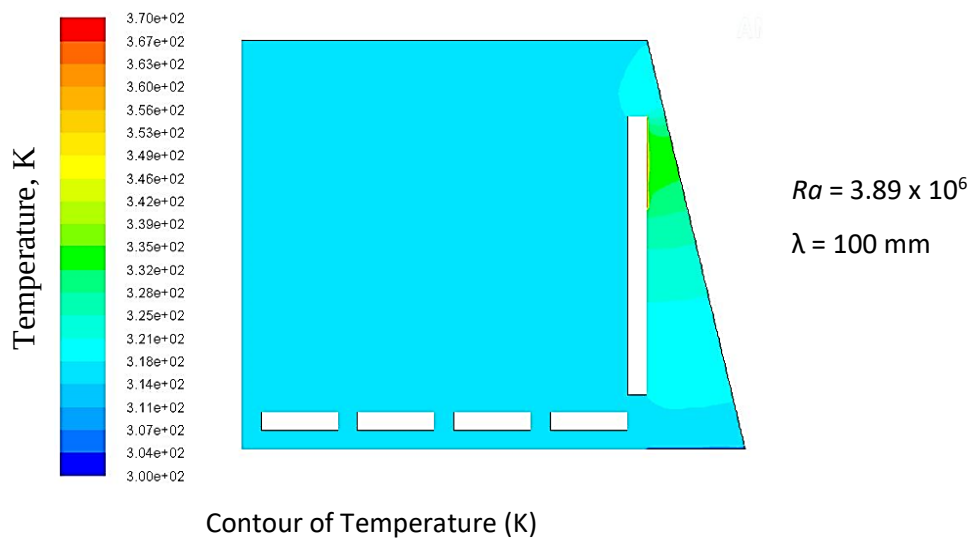


Figure 4.0 Isotherms with top angle (φ) 13° , transient time (τ_s), 5 s



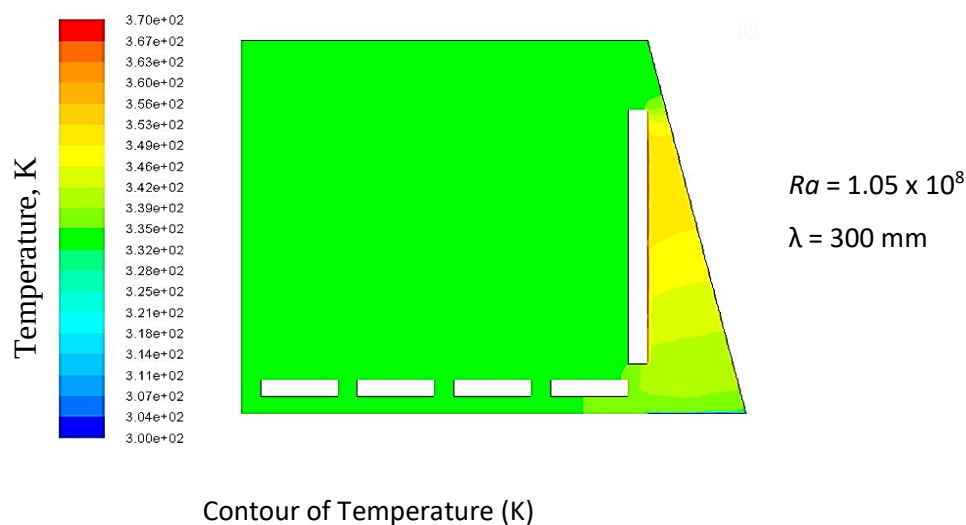
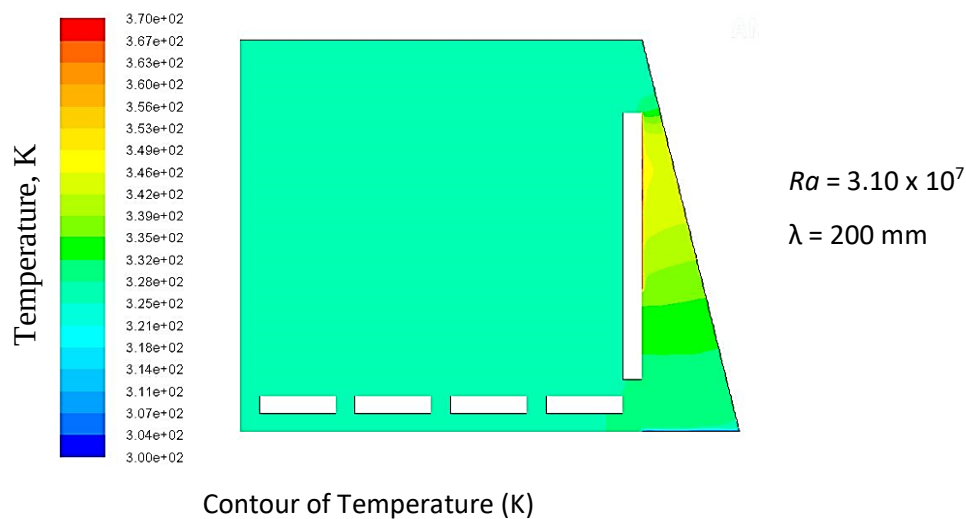
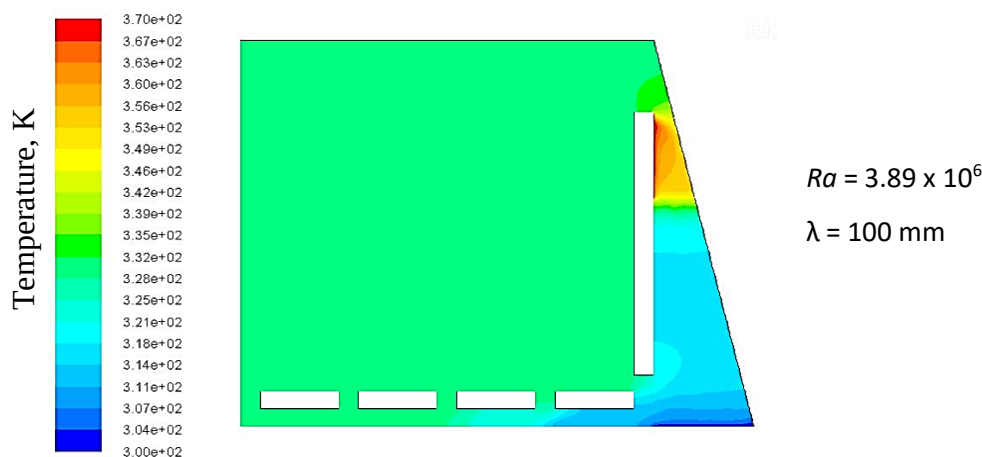
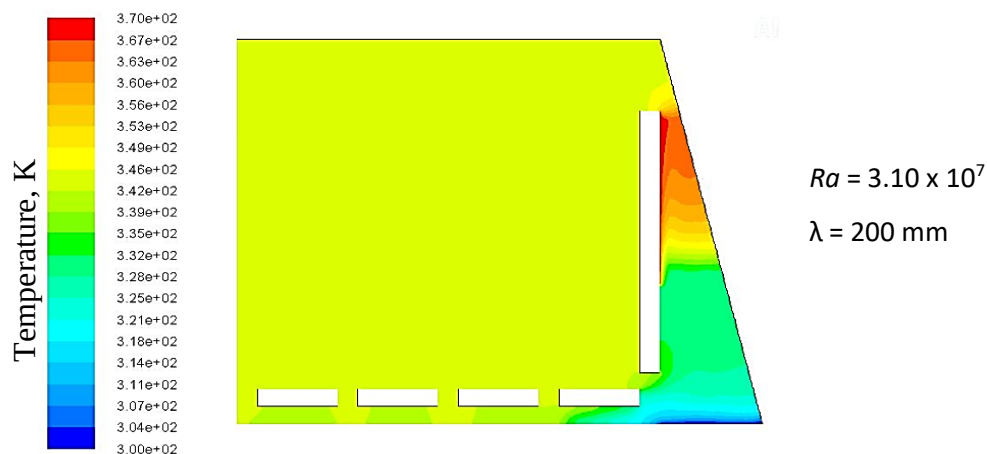


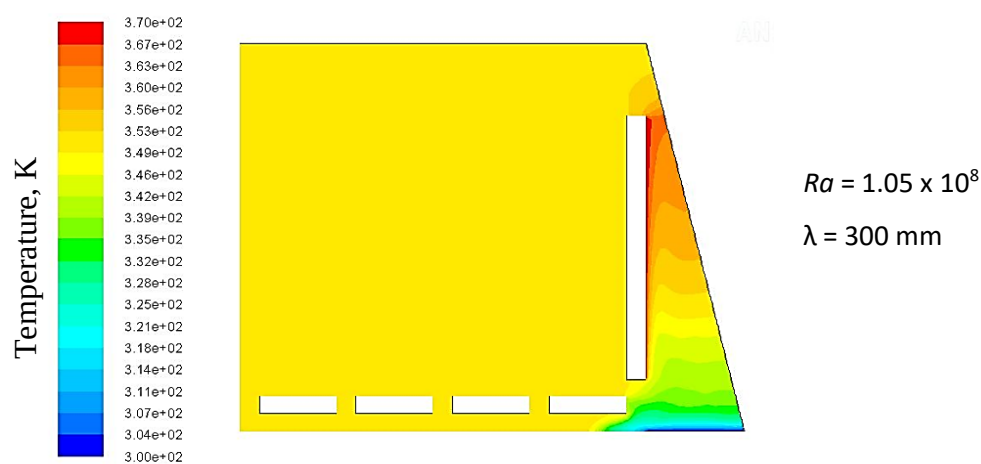
Figure 5.0 Isotherms with top angle (φ) 13°, transient time (τ_s), 30 s



Contour of Temperature (K)



Contour of Temperature (K)



Contour of Temperature (K)

Figure 6.0 Isotherms with top angle (φ) 13°, transient time (τ_s), 180

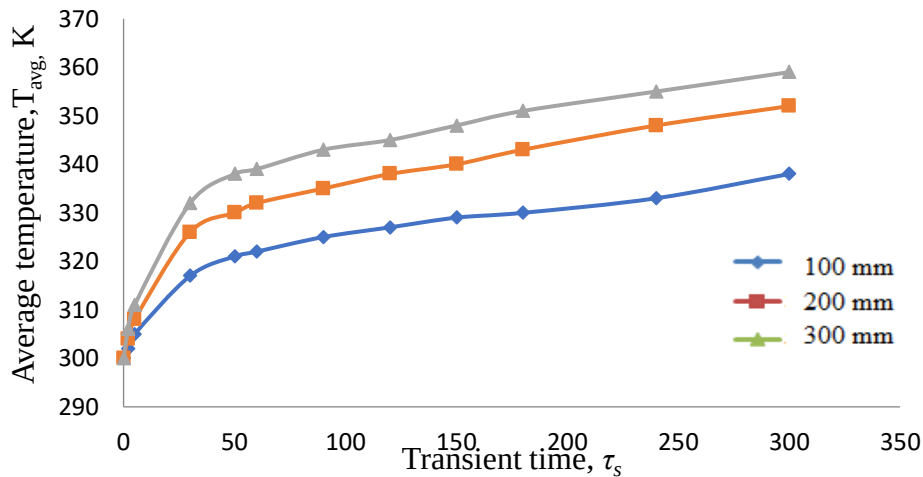


Figure 7.0 The average temperature, T_{avg} inside the rectangular cavity at $\phi = 13^\circ$

Conclusions

The present study was an attempt to numerically measure the temperature distribution inside the hybrid rectangular-triangular enclosure with faster recovery time to achieve the target temperature of 335K in order to fulfill the design requirement. The combinations of geometrical heater length, λ and top angle, ϕ was numerically investigated and visualized in isotherms. The results statistically plotted on Figure 7.0. From the isotherms visualization, applying λ at 300 mm demonstrated the fastest transient time to achieve average temperature distribution at 335K within 50 seconds. The slowest time recorded when λ at 100 mm applied with transient time of 259 seconds. The analysis proceeds by observing variations in the isotherms with respect to the different Ra of the local heat source and found that when the heater length increased, heat transfer rate enhanced due to heat intensity increment. Results suggest the use of heater length of 200 mm, with heating capability to reach 335K within 90 seconds for the optimum design consideration.

Acknowledgement

This research was funded by a grant from Ministry of Education Malaysia (T-ARGS BIL. 2/2019 Grant Document No: 91003697).

References

- [1] P. Biswal and T. Basak, "Bejan's heatlines and numerical visualization of convective heat flow in differentially heated enclosures with concave/convex side walls," *Energy*, vol. 64, pp. 69–94, Jan. 2014.
- [2] B. E. Yuce and E. Pulat, "Forced, natural and mixed convection benchmark studies for indoor thermal environments," *Int. Commun. Heat Mass Transf.*, vol. 92, no. February, pp. 1–14, 2018.
- [3] I. V. Miroshnichenko and M. A. Sheremet, "Turbulent natural convection heat transfer in rectangular enclosures using experimental and numerical approaches: A review," *Renew. Sustain. Energy Rev.*, vol. 82, no. July 2017, pp. 40–59, 2018.
- [4] F. Wu, W. Zhou, and X. Ma, "Natural convection in a porous rectangular enclosure with sinusoidal temperature distributions on both side walls using a thermal non-equilibrium model," *Int. J. Heat Mass Transf.*, vol. 85, pp. 756–771, 2015.

- [5] M. Salari, A. Rezvani, A. Mohammadtabar, and M. Mohammadtabar, "Numerical Study of Entropy Generation for Natural Convection in Rectangular Cavity with Circular Corners," *Heat Transf. Eng.*, vol. 36, no. November 2014, pp. 186–199, 2014.
- [6] H. Cui, F. Xu, and S. C. Saha, "A three-dimensional simulation of transient natural convection in a triangular cavity," *Int. J. Heat Mass Transf.*, vol. 85, pp. 1012–1022, Jun. 2015.
- [7] D. Das and T. Basak, "Role of discrete heating on the efficient thermal management within porous square and triangular enclosures via heatline approach," *Int. J. Heat Mass Transf.*, vol. 112, pp. 489–508, 2017.
- [8] S. I. Moldovan, A. M. Balasoiu, and M. Braun, "Experimental investigation of natural convection flow in a laterally heated vertical cylindrical enclosure," *Int. J. Heat Mass Transf.*, vol. 139, pp. 205–212, 2019.
- [9] M. Sheikholeslami, M. Gorji-Bandpy, I. Pop, and S. Soleimani, "Numerical study of natural convection between a circular enclosure and a sinusoidal cylinder using control volume based finite element method," *Int. J. Therm. Sci.*, vol. 72, pp. 147–158, 2013.
- [10] P. Biswal and T. Basak, "Entropy generation based approach on natural convection in enclosures with concave/convex side walls," *Int. J. Heat Mass Transf.*, vol. 82, pp. 213–235, Mar. 2015.
- [11] S. Pandey, Y. G. Park, and M. Y. Ha, "An exhaustive review of studies on natural convection in enclosures with and without internal bodies of various shapes," *Int. J. Heat Mass Transf.*, vol. 138, pp. 762–795, 2019.
- [12] T. Basak, R. Anandalakshmi, S. Roy, and I. Pop, "Role of entropy generation on thermal management due to thermal convection in porous trapezoidal enclosures with isothermal and non-isothermal heating of wall," *Int. J. Heat Mass Transf.*, vol. 67, pp. 810–828, Dec. 2013.
- [13] H. Karatas and T. Derbentli, "Natural convection and radiation in rectangular cavities with one active vertical wall," *Int. J. Therm. Sci.*, vol. 123, pp. 129–139, 2018.
- [14] R. W. Serth and T. G. Lestina, *Process Heat Transfer*. Elsevier, 2014.
- [15] R. S. Kaluri and T. Basak, "Role of distributed heating on enhancement of thermal mixing for liquid food processing with heat flow visualization method," *Innov. Food Sci. Emerg. Technol.*, vol. 18, pp. 155–168, Apr. 2013.
- [16] L. M. Raffinsoe, "Master Thesis Thermal Performance of Air Flow Windows," *Civ. Eng.*, no. July 2007, pp. 157–198, 2007.
- [17] D. Liu, F.-Y. Zhao, H.-Q. Wang, E. Rank, and G.-X. Kou, "Inverse determination of building heating profiles from the knowledge of measurements within the turbulent slot-vented enclosure," *Int. J. Heat Mass Transf.*, vol. 55, no. 17–18, pp. 4597–4612, Aug. 2012.
- [18] W. L. Staats and J. G. Brisson, "Active heat transfer enhancement in air cooled heat sinks using integrated centrifugal fans," *Int. J. Heat Mass Transf.*, vol. 82, pp. 189–205, Mar. 2015.
- [19] V. C. Mariani and L. S. Coelho, "Natural convection heat transfer in partially open enclosures containing an internal local heat source," *Brazilian J. Chem. Eng.*, vol. 24, no. 3, pp. 375–388, 2007.
- [20] C. H. Cheng and C. L. Chen, "Buoyancy-induced periodic flow and heat transfer in lid-driven cavities with different cross-sectional shapes," *Int. Commun. Heat Mass Transf.*, vol. 32, no. 3–4, pp. 483–490, 2005.
- [21] D. Cooper, T. J. Craft, K. Esteifi, and H. Iacovides, "Experimental investigation of buoyant flows in inclined differentially heated cavities," *Int. J. Heat Mass Transf.*, vol. 55, no. 23–24, pp. 6321–6339, 2012.

- [22] D. Iyi, R. Hasan, and R. Penlington, "Experimental and numerical study of Buoyancy-driven low turbulence flow in rectangular enclosure partially filled with isolated solid blockages," *Int. J. Heat Mass Transf.*, vol. 127, pp. 534–545, 2018.
- [23] T. W. H. Sheu and R. K. Lin, "Three-dimensional bifurcations in a cubic cavity due to buoyancy-driven natural convection," *Int. J. Heat Mass Transf.*, vol. 54, no. 1–3, pp. 447–467, 2011.
- [24] G. E. a Lau, G. H. a b Yeoh, V. a Timchenko, and J. A. a Reizes, "Large-eddy simulation of turbulent buoyancy-driven flow in a rectangular cavity," *International Journal of Heat and Fluid Flow*, vol. 39, pp. 28–41, 2013.
- [25] F. M. Mahfouz, "Buoyancy driven flow within an inclined elliptic enclosure," *Int. J. Therm. Sci.*, vol. 50, no. 10, pp. 1887–1899, 2011.
- [26] F. Hammami, B. Souayah, N. Ben-Cheikh, and B. Ben-Beya, "Computational analysis of fluid flow due to a two-sided lid driven cavity with a circular cylinder," *Comput. Fluids*, vol. 156, pp. 317–328, 2017.
- [27] A. Arefmanesh, M. Najafi, and S. H. Musavi, "Buoyancy-driven fluid flow and heat transfer in a square cavity with a wavy baffle - Meshless numerical analysis," *Eng. Anal. Bound. Elem.*, vol. 37, no. 2, pp. 366–382, 2013.
- [28] C. Cianfrini, M. Corcione, and P. P. Dell'Omo, "Natural convection in tilted square cavities with differentially heated opposite walls," *Int. J. Therm. Sci.*, vol. 44, no. 5, pp. 441–451, May 2005.
- [29] S. Gareh, "Numerical Study of Forced Convection in a Rectangular Channel," vol. 1, pp. 7–11, 2014.
- [30] L. Tang, D. Liu, F.-Y. Zhao, and G.-F. Tang, "Combined heat and moisture convective transport in a partial enclosure with multiple free ports," *Appl. Therm. Eng.*, vol. 30, no. 8–9, pp. 977–990, Jun. 2010.
- [31] A. Baïri, "Nusselt–Rayleigh correlations for design of industrial elements: Experimental and numerical investigation of natural convection in tilted square air filled enclosures," *Energy Convers. Manag.*, vol. 49, no. 4, pp. 771–782, Apr. 2008.
- [32] T. Avedissian and D. Naylor, "Free convective heat transfer in an enclosure with an internal louvered blind," *Int. J. Heat Mass Transf.*, vol. 51, no. 1–2, pp. 283–293, Jan. 2008.
- [33] G. M. Stavrakakis and N. C. Markatos, "Simulation of airflow in one- and two-room enclosures containing a fire source," *Int. J. Heat Mass Transf.*, vol. 52, no. 11–12, pp. 2690–2703, May 2009.
- [34] S. C. Saha, "Unsteady natural convection in a triangular enclosure under isothermal heating," *Energy Build.*, vol. 43, no. 2–3, pp. 695–703, Feb. 2011.
- [35] G. Yesiloz and O. Aydin, "Laminar natural convection in right-angled triangular enclosures heated and cooled on adjacent walls," *Int. J. Heat Mass Transf.*, vol. 60, no. 1, pp. 365–374, 2013.
- [36] S. C. Saha and Y. T. Gu, "Natural convection in a triangular enclosure heated from below and non-uniformly cooled from top," *Int. J. Heat Mass Transf.*, vol. 80, pp. 529–538, Jan. 2015.
- [37] A. Pantokratoras, "A note on natural convection along a convectively heated vertical plate," *Int. J. Therm. Sci.*, vol. 76, pp. 221–224, 2014.
- [38] V. A. F. Costa, "Natural convection in partially divided square enclosures: Effects of thermal boundary conditions and thermal conductivity of the partitions," *Int. J. Heat Mass Transf.*, vol. 55, no. 25–26, pp. 7812–7822, Dec. 2012.
- [39] D. R. Pitts and L. E. Sissom, "Vertical Flat Plate," in *Theory and Problems of Heat Transfer*, Asian Stud., McGraw-Hill Book Company, 1986, pp. 195–205.

- [40] and J. Z. T.H. Shih, W. W. Liou, A. Shabbir, Z. Yang, “A New k- ϵ Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation.,” *Comput. Fluids*, vol. 24, no. 3, pp. 227–238, 1995.
- [41] N. S. Gibanov and M. A. Sheremet, “Natural convection in a cubical cavity with different heat source configurations,” *Therm. Sci. Eng. Prog.*, vol. 7, no. March, pp. 138–145, 2018.
- [42] S. K. M.K., “CFD ANALYSIS OF NATURAL CONVECTION IN DIFFERENTIALLY HEATED ENCLOSURE Department of Mechanical Engineering National Institute of Technology Rourkela CFD ANALYSIS OF NATURAL CONVECTION IN DIFFERENTIALLY HEATED ENCLOSURE Department of Mechanical Engineering,” 2009.
- [43] S. Pinard, G. Fraisse, C. Ménézo, and V. Renzi, “Experimental study of a chimney enhanced heat emitter designed for internal renovation of buildings,” *Energy Build.*, vol. 54, pp. 169–178, Nov. 2012.
- [44] Q. Tu, S. Mao, Y. Feng, and D. Guo, “Heating control strategy in fresh air processor matched with variable refrigerant flow air conditioning system,” *Energy Convers. Manag.*, vol. 52, no. 7, pp. 2542–2554, Jul. 2011.
- [45] X. Q. Zhai, Z. P. Song, and R. Z. Wang, “A review for the applications of solar chimneys in buildings,” *Renew. Sustain. Energy Rev.*, vol. 15, no. 8, pp. 3757–3767, Oct. 2011.
- [46] B. Zamora and G. Sancho, “Effects of heating intensity on the transient natural convection flows in open cavities,” *Int. J. Heat Mass Transf.*, 2016.

3RD INTERNATIONAL SYMPOSIUM ON FLUID MECHANICS AND THERMAL SCIENCES (3RD ISFMTS 2019)

ORGANIZING COMMITTEE:

Chairman:

Assoc. Prof. Dr. Nor Azwadi Che Sidik

Treasurer:

Zafira Zainudin

Technical Committee:

Nuratikah Amid Dudin

Technical Reviewer:

Assoc. Prof. Dr. Maizirwan Mel

Dr. Mohd Zuki Salleh

Dr. Wira Jazair Yahya

Dr. Ahmad Muhsin Ithnin

Dr. Haslina Ahmad

Nor Hidayah Mat Taib

Dr. Fudhail Abdul Munir

Dr. Teoh Yew Heng

Dr. Ahmad Zahran Md Khudzari

Dr. Aklilu Baheta

Dr. How Heoy Geok

Dr. Makatar Wae-hayee

Dr. Nadiah Ahmad

Dr. Yu Kok Hwa

Wan Mohd Arif Aziz Japar

Hong Wei Xian

Tung Hao Kean

Ir. Dr. Mohd Rosdzimin bin Abdul Rahman

Dr. Mohamed Sukri Mat Ali

Dendy Adanta

Dr. Azlin Mohd Azmi

Dr. Wan Azani Wan Mustafa

Mohamad Shukri Zakaria

Dr. Mohd Sharizal Abdul Aziz

Dr. Zulkhibri Ismail

Rozanti A. Hamid

Liaison Officer:

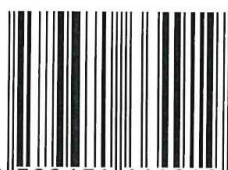
Nur Hajar Binti Mohamad Fadzil

3rd IS-FMTS



Published by:
Global Academic Excellence (M) Sdn. Bhd.
(1257579-U)
KELANTAN, MALAYSIA

eISBN 978-967-16649-1-9



9 789671 664919

3rd IS-FMTS