TRANSIENT REMOVAL OF CONTAMINANT FROM A CHANNEL WITH DIFFERENTIALLY HEATED WALL OF CAVITY

AHMAT RAJAB BIN KHAIRUL YUSRI

UNIVERSITI TEKNOLOGI MALAYSIA

TRANSIENT REMOVAL OF CONTAMINANT FROM A CHANNEL WITH DIFFERENTIALLY HEATED WALL OF CAVITY

AHMAT RAJAB BIN KHAIRUL YUSRI

A thesis submitted in fulfilment of the
requirements for the award of the degree of
Master of Philosophy

School of Mechanical Engineering
Faculty of Engineering
Universiti Teknologi Malaysia

JULY 2022

ACKNOWLEDGEMENT

In preparing this thesis, I was in contact with many people, researchers, academicians, and practitioners. They have contributed towards my understanding and thoughts. In particular, I wish to express my sincere appreciation to my main thesis supervisor, Assoc. Prof. Dr. Nor Azwadi bin Che Sidik, for encouragement, guidance, critics and friendship. Without his continued support and interest, this thesis would not have been the same as presented here.

I am also indebted to Universiti Teknologi Malaysia (UTM) for funding my Ph.D study. My fellow postgraduate student should also be recognised for their support. My sincere appreciation also extends to all my colleagues and others who have provided assistance at various occasions. Their views and tips are useful indeed. Unfortunately, it is not possible to list all of them in this limited space. I am grateful to all my family member.

ABSTRACT

Cleaning accumulated deposits inside pipe cavity are by disassembling and cleaning it part by part. Hydrodynamic cleaning of the cavity is an alternative method to clean accumulated deposits or contaminants inside the pipe cavity instead of dissembling them part by part is a tedious process or using a solvent which is not suitable in the food processing industry. This study aims to investigate the contaminants removal process from a cavity by resorting to natural flow to clean the deposits in different cavity sizes and includes different heating locations with different flow configurations. An experimental method is used to visualize the flow behaviour inside the cavity of a channel at a large aspect ratio in isothermal conditions. These results are used to validate numerical results obtained in isothermal flow conditions. For numerical study, Constrained Interpolated Profile (CIP) method is used for the advection phase of momentum and energy equation, and central difference is used to solve the non-advection phase of momentum and energy equations. The numerical studies include different aspect ratios (AR), 1 to 4, various Reynolds numbers (Re), 50 to 1000, and different locations of the heated wall inside the cavity (left wall, bottom wall, & right wall) for three different Grashof numbers (Gr), 1000, 10 000, and 100 000. The particles removal percentage at the transient and steady states are then compared and discussed. A larger aspect ratio and a more significant Reynolds number for isothermal conditions will give a higher percentage of contaminants removal except for AR = 4 and Re = 50. This particular flow shows a higher percentage of contaminant removal than AR = 4; Re = 100, 200, and 400. For mixed convection flow, one typical result can be concluded: at small Gr, the contaminant removal percentage is not changing significantly for all different heated wall positions. It is also shown that a more significant aspect ratio will produce a better contaminant removal process, and a higher Grashof number will improve the contaminant removal process. It is also found that when Gr equals 1000 and 10000, there is no significant change in the contaminant removal process and constant heat flux from the bottom wall for Gr = 100,000 gives the highest contaminant removal percentage for every aspect ratio. The highest percentage removal of contaminant is 98.94% for Gr =100 000, AR=4.

ABSTRAK

Membersihkan mendapan terkumpul di dalam rongga paip adalah dengan membuka dan membersihkannya pada setiap bahagian. Pembersihan rongga hidrodinamik adalah kaedah alternatif untuk membersihkan mendapan atau bahan cemar yang terkumpul di dalam rongga paip dan tanpa membuka setiap bahagian yang merupakan proses yang membosankan atau menggunakan pelarut yang tidak sesuai dalam industri pemprosesan makanan. Kajian ini bertujuan untuk menyiasat proses penyingkiran bahan cemar dari rongga dengan menggunakan aliran semula jadi untuk membersihkan mendapan dalam rongga yang berbeza saiz dan termasuk lokasi pemanasan yang berbeza dengan konfigurasi aliran yang berbeza. Kaedah eksperimen digunakan untuk menggambarkan kelakuan aliran di dalam rongga saluran pada nisbah aspek yang besar dalam keadaan seisoterma. Keputusan ini digunakan untuk mengesahkan keputusan berangka yang diperoleh dalam keadaan aliran isoterma. Untuk kajian berangka, kaedah *Constrained Interpolated Profile* (CIP) digunakan untuk fasa adveksi momentum dan persamaan tenaga, dan pembezaan pusat digunakan untuk menyelesaikan fasa bukan adveksi bagi persamaan momentum dan tenaga. Kajian berangka termasuk nisbah aspek (AR), 1 hingga 4, pelbagai nombor Reynolds (Re), 50 hingga 1000, dan lokasi dinding dipanaskan yang berbeza di dalam rongga (dinding kiri, dinding bawah, & dinding kanan) untuk tiga dinding yang berbeza. Nombor Grashof (Gr), 1000, 10 000, dan 100 000. Peratusan penyingkiran zarah pada keadaan sementara dan mantap kemudiannya dibandingkan dan dibincangkan. Nisbah aspek yang lebih besar dan nombor Reynolds yang lebih ketara untuk keadaan isoterma akan memberikan peratusan penyingkiran bahan cemar yang lebih tinggi kecuali AR = 4 dan Re = 50. Aliran tertentu ini menunjukkan peratusan penyingkiran bahan cemar yang lebih tinggi daripada AR = 4; Re = 100, 200, dan 400. Untuk aliran perolakan bercampur, satu keputusan tipikal boleh disimpulkan: pada Gr kecil, peratusan penyingkiran bahan cemar tidak berubah dengan ketara untuk semua kedudukan dinding dipanaskan yang berbeza. Ia juga menunjukkan bahawa nisbah aspek yang lebih ketara akan menghasilkan proses penyingkiran bahan cemar yang lebih baik, dan nombor Grashof yang lebih tinggi akan meningkatkan proses penyingkiran bahan cemar. Juga didapati bahawa apabila Gr bersamaan dengan 1000 dan 10000, tiada perubahan ketara dalam proses penyingkiran bahan cemar dan fluks haba malar dari dinding bawah untuk Gr = 100,000 memberikan peratusan penyingkiran bahan cemar yang tertinggi bagi setiap nisbah aspek. Peratusan tertinggi penyingkiran bahan cemar ialah 98.94% untuk Gr =100 000, AR=4.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

AR	-	Aspect Ratio
NSE	-	Navier – Stokes Equation

LIST OF SYMBOLS

β	-	Coefficient of thermal expansion
θ	-	Dimensionless temperature
μ	-	Dynamic viscosity of fluid
υ	-	Kinematic viscosity of fluid
Ψ	-	Dimensionless streamline
ψ	-	streamline
Ω	-	Dimensionless vorticity
ω	-	vorticity
A	-	Area cross section of pipe/duct
C_D	-	Coefficient of drag
D_h	-	Hydraulic diameter of pipe/channel
D_p	-	Dimensionless diameter of particle
F_d	-	Drag force
g	-	Gravity acceleration
Gr	-	Grashof number
L	-	Cavity's length
L_e	-	Entrance length
L_o	-	Outlet length
m_p	-	Particle mass
Pr	-	Prandtl number
Q	-	Flow rate
Re	-	Reynolds number
Re_p	-	Reynolds number associate with particle
Ri	-	Richardson number
T	-	Dimensionless time
t	-	time
X	-	Dimensionless space in horizontal direction
Y	-	Dimensionless space in vertical direction

LIST OF APPENDICES

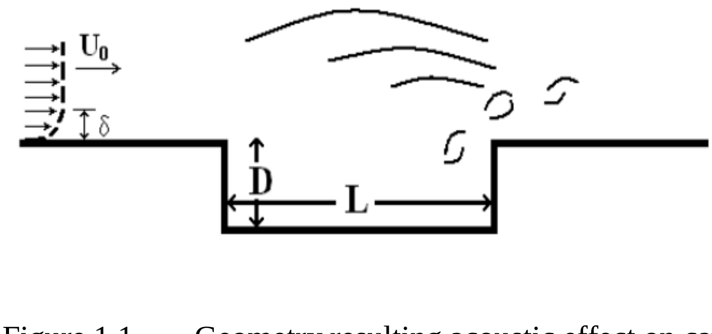
CHAPTER 1

INTRODUCTION

1.1 Overview

Cavity flow can be found in flow past a panel, flow in organ pipes, flow past a sunroof of vehicle, flow past a window, flow around a weapon bay, landing gear of an aircraft and etcetera. There is a lot more applications on cavity flow such as flow over street canyon which involving environmental study related to air pollutant control.

Details on properties and effects of cavity onto main flow have been reported by many researchers such as effect of oscillation from cavity flow onto main flow (Meganthan, 2000), flow acoustic effect resulting from circulation past cavity as shown in Figure 1.1 (Ebrahimi, 2011) and many more. These studies focusing on effect of cavity onto flow itself. Parameter use for cavity flow such as depth of cavity, D and length of cavity, L will results different flow structure inside cavity. Different inlet velocity will produce different free stream velocity, U_0 and different boundary layer thickness, δ. On the other hand, there is also research on contaminated cavity flow (Saadun et al, 2013; Jahanshaloo et al, 2014) where the contaminated cavity is studied on their particle removal process.



U_0 = free stream velocity
δ = boundary layer thickness
D = depth
L = length

Figure 1.1 Geometry resulting acoustic effect on cavity flow (Ebrahimi, 2011)

A contaminated cavity can be seen inside hydraulic components such as metal exposed to water and resulting rust particle accumulate inside cavity (Figure 1.2) (Gannon, 2018). Another example of a contaminated cavity is contaminated pipeline resulting from improper fitting of pipe joint. Therefore, cleaning process of the contaminated cavity becomes an important process to maintain hydraulic and pipeline to working properly. Furthermore, cleaning contaminated cavity can be a tedious process because need to dissemble them and clean them part by part.

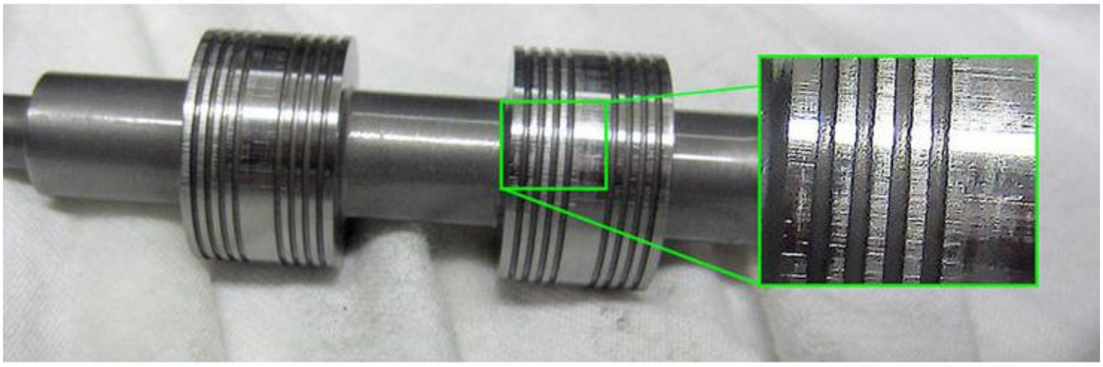
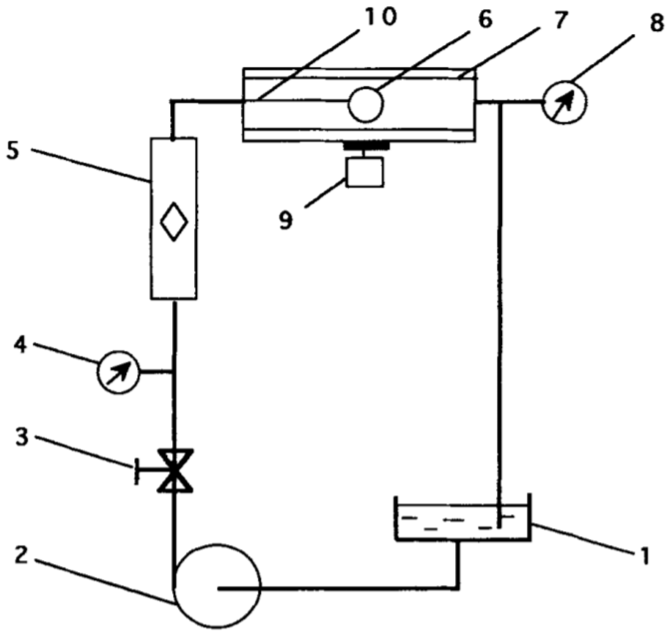


Figure 1.2 Rust particles due to water contamination in hydraulic spool. (Gannon, 2018)

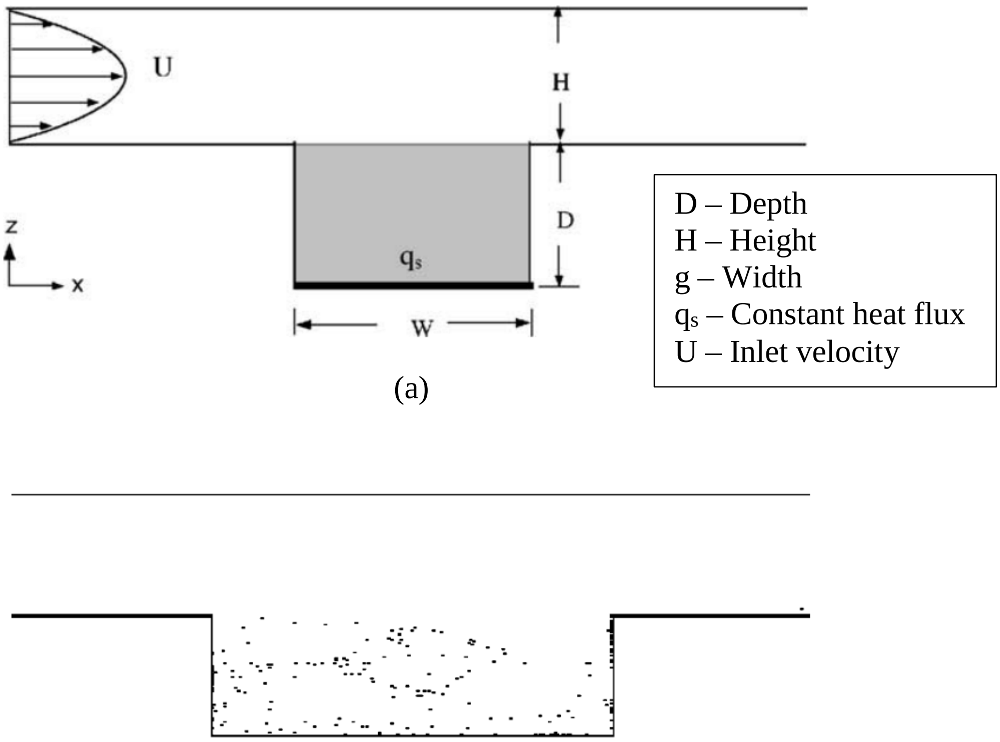
Focusing on the poor fitting of pipeline, hydrodynamic cleaning is one of simple methods to clean the contaminated cavity without dissembling it part by part. There are many studies on hydrodynamics cleaning of components, parts and pipelines that known as one research area as a method in cleaning process in pipes. One of them is by using a restrained ball and let lateral vibration of the ball clean the wall of pipe as done by Grinis and Korin (1997). They were focusing on harvesting the levitation effect of a ball inside a pipe to clean sediment and rust inside a pipeline as shown in Figure 1.3. The components of the experiment as follow: item 1 is tank, item 2 is pump, item 3 is valve to control flow rate, item 4 & 8 are manometer, item 5 is flow meter, item 6 is ball used for cleaning purpose, item 7 is pipe and item 10 is flexible wire to restrain the ball. The ball will rotate due to effect of wall of pipe and provide lateral vibration of the pipe to clean the pipe.



1 - Tank
2 - Centrifugal pump
3 - Valve
4 - Manometer
5 - Flow meter
6 - Ball
7 - Pipe
8 - Manometer
9 - Integrating vibrating meter
10 - Flexible wire

Figure 1.3 Experiment equipment sketch by Grinis and Korin (1997)

On the other hand, the effect of mixed convection flow in enhancing contaminant removal process is also one of the research areas in hydrodynamic cleaning in pipe. Zain (2012) has studied the effect of heated bottom wall to the particle removal process from cavity. Similar study as shown in Figure 1.4 reported by Fang (2003), where the bottom wall of cavity supplied with constant heat flux to enhance the removal process inside the cavity. The constant heat flux will change the flow structure inside cavity due to thermal buoyant effect to the fluid flow.



(b)

Figure 1.4 Work by Fang (2003) (a) Sketch of a contaminated cavity with constant heat flux at the bottom wall of cavity, (b) Remaining contaminant of cavity for aspect ratio 4 at steady state.

Another examples of application-related on problem arise from flow over cavity with thermal effect is cooling computer chip and computer hardware. In order computers to function properly, cooling process of its hardware is crucial as advanced computer chip provides faster processing time but also will produce heat faster. As the computer component such as processor can be costly to replace if it is burned due to overheating, cooling system for computer components is also has become an interest in the computer industry. In cooling system of computer, there are many methods such as blow cooler air to the heated component and sometimes air conditioner is used to provide faster cooling to the computer. Due to the importance of application related to flow over cavity, it is become one of interest field to further study to expanding knowledge.

Aforementioned above regarding problem arise on cavity flow, it is important to explore and study them for better understanding. In order to study on flow over the cavity, there is three methods can be used to get the results. There are 3 types of method to solve fluid dynamics as shown in Figure 1.5 which are doing analytical calculation for solution, experimental analysis and numerical method analysis. For analytical calculation can be solved by some mathematical calculation by applying correct boundary and initial condition. Results obtained were due to simplification of Navier-Stokes equation and match to the real situation but it is can only apply to very simple cases such as inviscid flow. Experimental analysis is very reliable because it is done according to the real-life situation with minimal simplification and assumptions. The major concern of conducting experiment is that the test rig can be so expensive that researcher always tries to do non-destructive test to their test rig. For numerical method, it is a cheaper method to use as it can produce significant results together with the ability to control the boundary condition and parameter of study easily. There is a lot of available numerical methods, which can be used to study flow over cavity and this will be discussed in detail in Chapter 2.

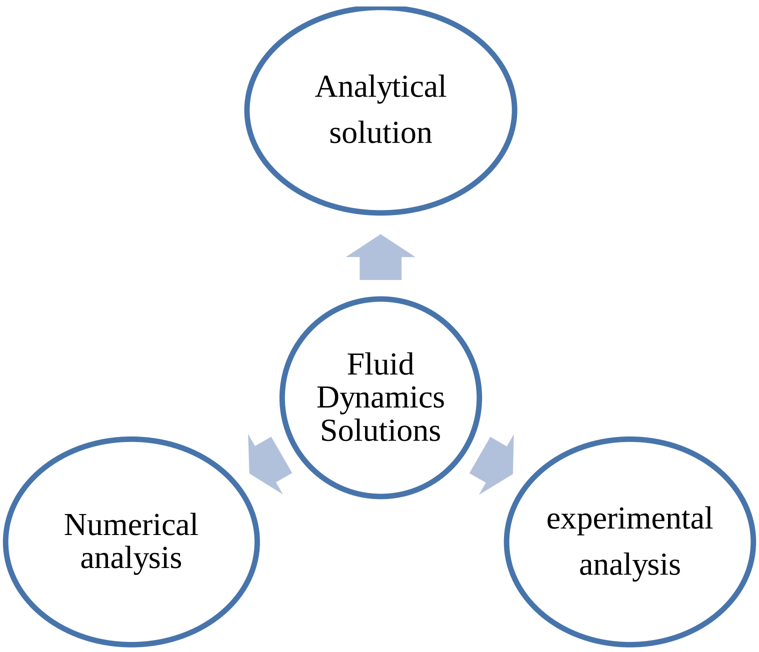


Figure 1.5 Fluid dynamics solution

Recently, Abdelmassih, Vernet and Pallares (2016) have studied the flow over cavity with heated bottom wall inside the cavity by using a numerical method which is a three-dimensional direct numerical simulation and experimentally for channel flow over cavity. They have reported that there is periodic flow at Re=100 and Ri=10 in their mixed convection flow which related to heat removal process from the bottom of the cavity. Nevertheless, they captured the fluid velocity by using Particle Image Velocimetry (PIV) where small particles are seeded in the water and motion of the particles were captured and flow velocity was obtained. The summary of the result is shown in Table 1.1 and they reported that Re>500 will experience turbulent behaviour to the flow for Richardson number, Ri=1 and Ri=10.

Table 1.1: List of studied cases of Abdelmassih et al. (2016).

Re	Ri			
	Isothermal	0.1	1	10
100	Steady state	Steady state	Steady state	Periodic flow
500	Steady state	Steady state	Turbulent	Turbulent
1000	Steady state	Steady state	Turbulent	Turbulent
1500	Steady state	Steady state	Turbulent	Turbulent

There are also such similar cases study on flow over the cavity but the cavity is contaminated and hydrodynamic flow is used for contaminant removal from the cavity such as done by Fang (2003). In this case, the heated wall inside cavity is located at the bottom wall of the cavity. His study was focused on the effect of aspect ratio of cavity and effect of Grashof number to the contaminant removal effectiveness. He found that higher Grashof number will significantly improve the contaminant removal from cavity. His study includes different aspect ratio from 0.25 to 4 and Grashof number 1 to 4000. His conclude that different flow pattern can be found by imposing heat flux from bottom of cavity. It is also shows that different aspect ratio provide different contaminant removal percentage but only limited for heated bottom wall of cavity.

In different study (Farsani, Ghasemi and Aminossadati, 2014), the removal of contaminant and flow behaviour due to magnetohydrodynamic effect was studied. They were using numerical method to study the heat transfer performance and the removal process of fluid particles. In general, stronger buoyant flow is reported to improve the removal process from the cavity at higher Grashof number and higher Reynolds number. However, their report provided very limited cases which are only 3 cases for different Reynolds number and 3 cases for different Grashof number.

As mixed convection flow is known by applying constant heat flux from the bottom wall of the cavity can change the flow structure, there are also right vertical wall and left vertical wall of the cavity that also can contribute to changing the flow structure. There is literature available such as the one done by Stiriba et al. (2010) where the right vertical wall of the cavity is remaining at constant temperature that higher than ambient temperature as shown in Figure 1.6. Another study by Aminossadati & Ghasemi (2009), the heated wall is at same location but only part of the wall is heated which reported only half of the wall is heated in the middle of the wall as shown in Figure 1.7. Their study provides data on the effect of different heated wall inside the cavity to the flow behaviour without contaminant inside the cavity.

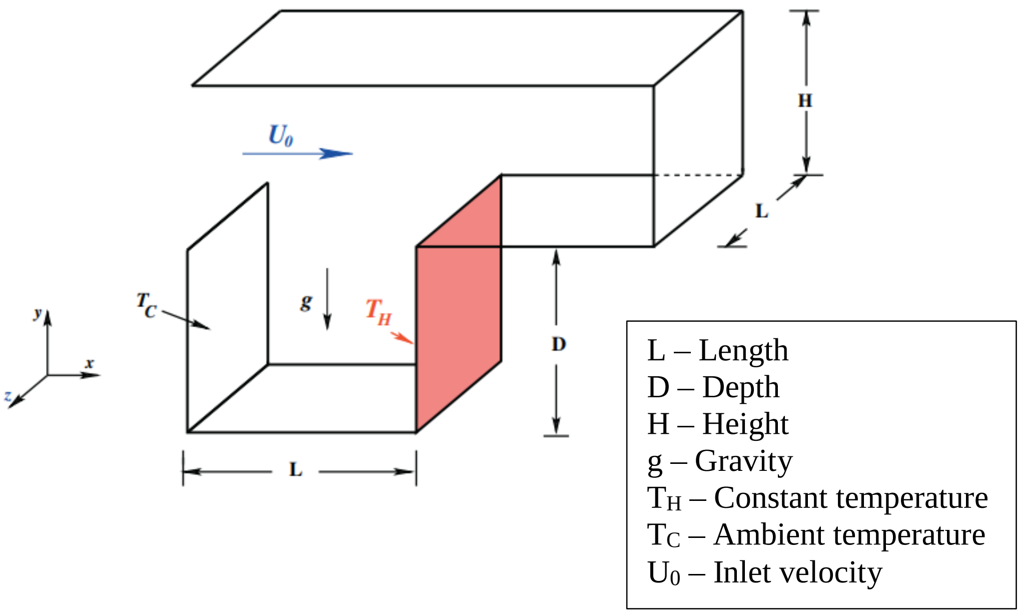


Figure 1.6 Sketch of cavity flow with heated right wall Stiriba et al. (2010)

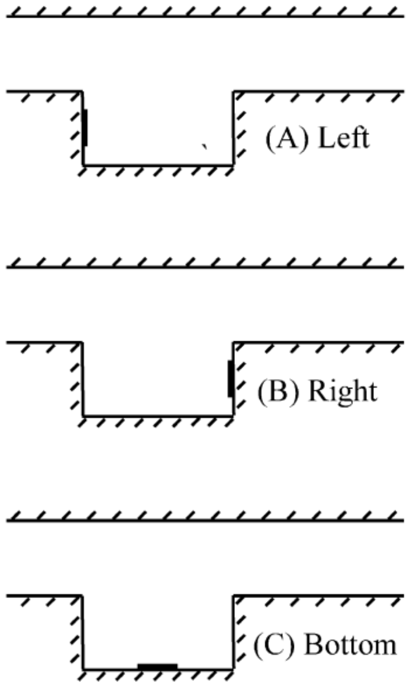


Figure 1.7 Heat source inside cavity at (a) left wall, (b) right wall, and (c) bottom wall (Aminossadati et al, 2009)

Manca,et al. (2003) has study numerically for temperature distribution and stream function using air for 3 different heated wall position for aspect ratio 2. Three different heated wall location set up as shown in Figure 1.8 where H is channel height, u_i is inlet velocity, D is cavity depth, L is cavity length and q is heat flux. Experimental study was done later by Manca, Nardini & Vafai (2006) for heated left wall of cavity. Nevertheless, they also provide experimental study for heated right wall of cavity in Manca Nardini & Vafai (2008). Eventhough their study focused on temperature distribution, their studies also shown that different heated wall will produced different vortex structure inside cavity but their studies are limited for temperature distribution and flow structure without contaminant removal process. It also didn’t includes data for heated bottom wall of cavity.

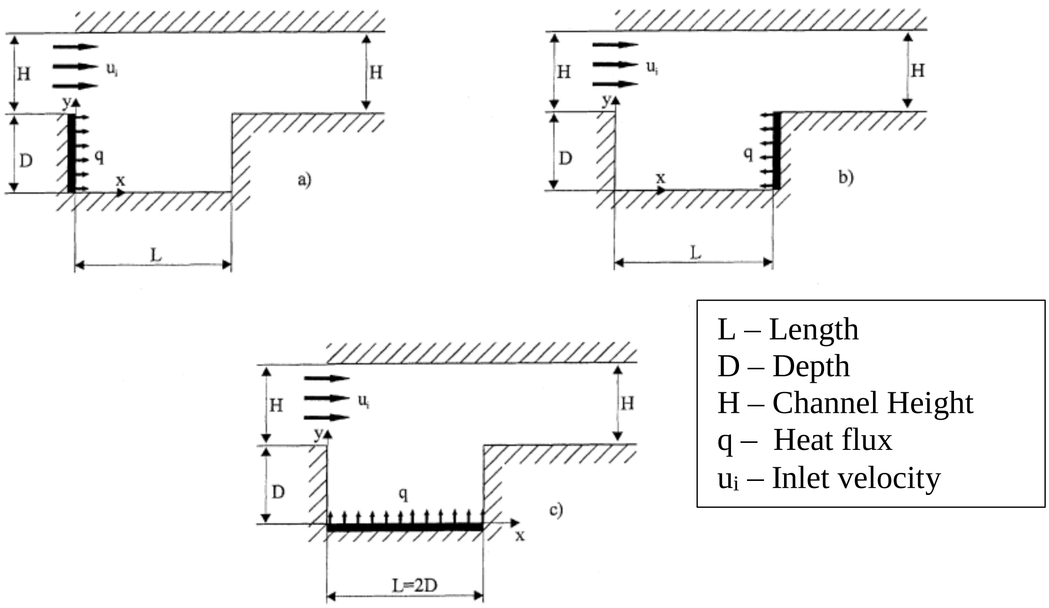


Figure 1.8 Three different heated wall position by Manca et al (2003) (a) Left wall, (b) Right wall & (c) Bottom wall.

It appears from the aforementioned study that investigation has been conducted regarding mixed convection cavity flow but some of their studies are more on heat removal process efficiency and flow structure inside cavity. It is appeared that removal of contaminant from cavity with mixed convection flow study can be broaden in term of different flow condition such as different Reynolds number and Grashoff number. It is also can be notify various study that used different aspect ratio of cavity and different heat source location can change the flow structure and flow behavior. Therefore, research on contaminant removal inside cavity by utilizing mixed convection flow sources from cavity wall is still a gap of knowledge in engineering and it is necessary to go for deep research on effect of Grashof number and location of heated wall to the effect of contaminant removal from the cavity.

1.2 Problem of statement

Conventional method to clean pipe with accumulated deposits is to dissemble it and clean it part by part. This happened where cavity flow occurred inside a pipe. There is a lot of methods to clean pipe such as hydrodynamic cleaning, vibrating of a restrain ball inside a pipe, using solvent and many more to remove the accumulated deposit and to clean the pipes. When working with food industries, the solvent is not recommended as it will affect the taste or composition of the products (Zain, 2012). Hydrodynamic cleaning can be used instead of using solvent or dissemble the pipes for cleaning. Cavity flow problem such as poor-fitting of pipe joint in two-dimensional flows in a pipe with square cavity still need to study extensively. This included various sizes of cavity flow and will be represented by various aspect ratios. Introducing constant heat flux at different cavity wall is still a question whether this can improve the removal of contaminant and accumulated deposits. Fang (2003) found that higher Grashoff number will improved the contaminant removal from cavity but this is only limited for heated at bottom wall. Study by Striba et al (2010) shown that different location of heated wall (right vertical wall) can also change the flow structure although they use air as the fluid. On the other hand, numerical study (Manca et al., 2003) and experimental studies (Manca et al., 2006 & Manca et al., 2008) on three different heated wall but limited to temperature distribution and without contaminant removal process. Therefore it is a gap in knowledge to know the best heating wall location to get better contaminant removal. This study aims to find the percentage of contaminants removal from cavity by using different heated wall properties such as different Grashoff number and different heated wall such as left vertical wall, bottom wall and right vertical wall of the cavity. In addition, flow criteria such as different Reynolds number and cavity geometry such as different cavity length to height ratio also will be included in this study to determine the effectiveness of contaminant removal process from cavity.

1.3 Objective of the study

For the hydrodynamic cleaning in pipes, several aspects of variable is included in the research. These include various aspect ratio of cavity inside pipe, various Reynolds number and location of thermal effect. To get a better understanding of the flow structure and numerical prediction, experiment set up for isothermal flow will be included in this study. To utilize cheaper research cost and flexibility, simulation and numerical study will be implemented more within this study. These ideas can be accomplished by following these objectives.

1. To experimentally investigate the flow behaviour in the cavity at a large aspect ratio by capturing the flow structure inside cavity using water and dye.
2. To numerically develop the flow structure in the cavity using CIP method by using streamlines plot.
3. To analyse the interaction of particle and flow structure in the process of particle removal in the cavity.
4. To evaluate the thermal boundary effect on the rate of particle removal from heated cavity.

1.4 Scope of the study

In order to achieve the aforementioned objectives, several assumption and limitation of study need to define so that the study can be narrowed and focussed. The scope of this study is defined as follow:

i. Two-dimensional flows will be implemented along the study so Navier-Stokes equation is written and discretize based on two-dimensional method with dimensionless parameter.
ii. Laminar flow is considered for whole study and parabolic flow (fully developed flow) will be the flow profile before the flow enters the cavity for isothermal and mixed convection cases. The flow is limited to a Reynolds number of 50 to 1000.
iii. Grashof number will be used for Mixed convection and the maximum of Gr=100,000 is simulated to show that natural convection is dominant in the flow.
iv. A number of changes in parameter during simulation are neglected as the change is insignificant such as Prantl number will remain constant at 7.0 at any temperature distribution along the simulation. This is also implemented to density of water (incompressible flow).
v. The contaminants (particles) is treated as a hard-sphere model. The size of contaminants is considered relatively small and will be treated as a point in the numerical grid.

REFERENCES

Abdelmassih, G., Vernet, A., & Pallares, J. (2016). Steady and unsteady mixed convection flow in a cubical open cavity with the bottom wall heated. *International Journal of Heat and Mass Transfe*r, 101, 682–691.

Altınsoy, N., Tuğrul, A. B., Baytaş, F., Baydoğan, N., Karatepe, N., Hacıyakupoğlu, S., & Büyük, B. (2013). Investigation of helical flow by using tracer technique. *EPJ Web of Conferences*, 50, 01001.

Aminossadati, S. M., & Ghasemi, B. (2009). A numerical study of mixed convection in a horizontal channel with a discrete heat source in an open cavity. *European Journal of Mechanics - B/Fluids*, 28(4), 590–598.

Beetstra, R., van der Hoef, M. A., & Kuipers, J. A. M. (2006). A lattice-Boltzmann simulation study of the drag coefficient of clusters of spheres. *Computers & Fluids*, 35(8–9), 966–970.

Biringen, S., & Chow, C. Y. (2011). *An Introduction to Computational Fluid Mechanics by Example*. New Jersey, USA: Wiley.

Botella, O., & Peyret, R. (1998). Benchmark spectral results on the lid-driven cavity flow. *Computers & Fluids*, 27(4), 421–433.

Brown, N. M., & Lai, F. C. (2005). Correlations for combined heat and mass transfer from an open cavity in a horizontal channel. *International Communications in Heat and Mass Transfer*, 32(8), 1000–1008.

Burgos, J., Cuesta, I., & Salueña, C. (2016). Numerical study of laminar mixed convection in a square open cavity. *International Journal of Heat and Mass Transfer*, 99, 599–612.

Chang, T. S., & Tsay, Y. L. (2001). Natural convection heat transfer in an enclosure with a heated backward step. *International Journal of Heat and Mass Transfer*, 44(20), 3963–3971.

Che Sidik, N. A., Ahmad Sofianuddin, A. S., & Ahmat Rajab, K. Y. (2014). Numerical Prediction of Contaminant Removal from Cavity in Horizontal Channel by Constrained Interpolated Profile Method. *Applied Mechanics and Materials*, 695, 384–388.

Chu, K. W., & Yu, A. B. (2008). Numerical simulation of complex particle–fluid flows. *Powder Technology*, 179(3), 104–114.

Crook, S.D., (2011), "A study of the flow within shallow, narrow cavities at low Reynolds number" Ph.D. Thesis, The University of Adelaide, Adelaide, Australia.

Deen, N. G., Annaland, M. V. S., & Kuipers, J. A. M. (2006). Detailed computational and experimental fluid dynamics of fluidized beds. *Applied Mathematical Modelling*, 30(11), 1459–1471.

Deen, N. G., Van Sint Annaland, M., Van der Hoef, M. A., & Kuipers, J. A. M. (2007). Review of discrete particle modeling of fluidized beds. *Chemical Engineering Science*, 62(1–2), 28–44.

Ebrahimi, M. (2011). Investigation of Nonlinear Acoustic Effects in Two-Dimensional Open Cavity Flow. *Applied Mechanics and Materials*, 110–116, 4578–4588.

Fang, L. C., Nicolaou, D., & Cleaver, J. W. (1999). Transient removal of a contaminated fluid from a cavity. *International Journal of Heat and Fluid Flow*, 20(6), 605–613.

Fang, L. C. (2003). Effect of mixed convection on transient hydrodynamic removal of a contaminant from a cavity. *International Journal of Heat and Mass Transfer*, 46(11), 2039–2049.

Farsani, R., Ghasemi, B., Aminossadati, S. (2014). Magnetohydrodynamic mixed convection effects on the removal process of fluid particles from an open cavity in a horizontal channel. *Journal of Heat and Mass Transfer Research*, 1(2), 67-74.

Fayyaz, H. and Shah, A. (2018). Steady viscous flow inside deep, shallow and skewed cavities by an implicit Navier-Stokes solver. *International Journal of Advanced and Applied Sciences*, 5(1), 170-176.

Fujimatsu, N., & Suzuki, K. (2010). New interpolation technique for the CIP method on curvilinear coordinates. *Journal of Computational Physics*, 229(16), 5573–5596.

Fukumitsu, K., Yabe, T., Ogata, Y., Oami, T., & Ohkubo, T. (2015). A new directional-splitting CIP interpolation with high accuracy and low memory consumption. *Journal of Computational Physics*, 286, 62–69.

García, F., Treviño, C., Lizardi, J., & Martínez-Suástegui, L. (2019). Numerical study of buoyancy and inclination effects on transient mixed convection in a channel with two facing cavities with discrete heating. *International Journal of Mechanical Sciences*, *155*, 295–314.

Grinis, L., & Korin, E. (1997). Hydrodynamic method for cleaning inner surfaces of pipes. *Chemical Engineering & Technology*, 20(4), 277–281

Gustafson, K. and Halasi, K. (1987). Cavity flow dynamics at higher Reynolds number and higher aspect ratio. *Journal of Computational Physics*, 70(2), pp.271-283.

Gustafson, K., & Halasi, K. (1986). Vortex dynamics of cavity flows. *Journal of Computational Physics*, 64(2), 279–319.

Hayden, K. S., Park, K., & Curtis, J. S. (2003). Effect of particle characteristics on particle pickup velocity. *Powder Technology*, 131(1), 7–14.

Hoomans, B. P. B., Kuipers, J. A. M., Briels, W. J., & van Swaaij, W. P. M. (1996). Discrete particle simulation of bubble and slug formation in a two-dimensional gas-fluidised bed: A hard-sphere approach. *Chemical Engineering Science*, 51(1), 99–118.

Hou, S., Zou, Q., Chen, S., Doolen, G., & Cogley, A. C. (1995). Simulation of Cavity Flow by the Lattice Boltzmann Method. *Journal of Computational Physics*, 118(2), 329–347.

Hu, C., Yang, K.-K., & Kim, Y. (2010). 3-D numerical simulations of violent sloshing by CIP-based method. *Journal of Hydrodynamics*, 22(S1), 253–258.

Ida, M., & Yabe, T. (1995). Implicit CIP (Cubic-Interpolated Propagation) method in one dimension. *Computer Physics Communications*, 92(1), 21–26.

Jahanshaloo, L., Sidik, N. A. C., Salimi, S., & Safdari, A. (2014). The Use of Thermal Lattice Boltzmann Numerical Scheme for Particle-Laden Channel Flow with a Cavity. *Numerical Heat Transfer, Part A: Applications*, 66(4), 433–448.

Jiang, N., Yao, S., Feng, L., Sun, H., & Liu, J. (2017). Experimental study on flow behavior of breathing activity produced by a thermal manikin. *Building and Environment*, 123, 200–210.

John D Anderson, Jr. (1995) 'Computational fluid dynamics : the basics with applications'. New York : McGraw-Hill

Kartusinsky, A. I., Michaelides, E. E., Hussainov, M. T., & Rudi, Y. (2009). Effects of the variation of mass loading and particle density in gas–solid particle flow in pipes. *Powder Technology*, 193(2), 176–181.

Khanafer, K., & Vafai, K. (2000). Buoyancy-driven flow and heat transfer in open-ended enclosures: elimination of the extended boundaries. International *Journal of Heat and Mass Transfer*, 43(22), 4087–4100.

Khanafer, Khalil, Vafai, K., & Lightstone, M. (2002). Mixed convection heat transfer in two-dimensional open-ended enclosures. *International Journal of Heat and Mass Transfer*, 45(26), 5171–5190.

Laín, S., & Sommerfeld, M. (2013). Characterisation of pneumatic conveying systems using the Euler/Lagrange approach. *Powder Technology*, 235, 764–782.

Lawson, S. J., & Barakos, G. N. (2011). Review of numerical simulations for high-speed, turbulent cavity flows. *Progress in Aerospace Sciences*, 47(3), 186–216.

Leong, J. C., Brown, N. M., & Lai, F. C. (2005). Mixed convection from an open cavity in a horizontal channel. *International Communications in Heat and Mass Transfer*, 32(5), 583–592.

Liu, O. Z., & Cai, J. S. (2011). Numericl Analysis Supersonic Cavity Unsteady Flow. *Applied Mechanics and Materials*, 110–116, 783–789.

Liu, Y., Dou, Z. G., & Duan, L. W. (2015). Numerical Investigation of Cavity Flow Field Characteristics in Supersonic Flow. *Applied Mechanics and Materials*, 789–790, 368–372.

Malhotra, A., & Vaidyanathan, A. (2016). Aft wall offset effects on open cavities in confined supersonic flow. *Experimental Thermal and Fluid Science*, 74, 411–428.

Manca, O., Nardini, S., Khanafer, K. and Vafai, K., 2003. Effect Of Heated Wall Position On Mixed Convection In A Channel With An Open Cavity. *Numerical Heat Transfer, Part A: Applications*, 43(3), pp.259-282.

Manca, O., Nardini, S. and Vafai, K., 2006. Experimental Investigation of Mixed Convection in a Channel With an Open Cavity. *Experimental Heat Transfer*, 19(1), pp.53-68.

Manca, O., Nardini, S. and Vafai, K., 2008. Experimental Investigation of Opposing Mixed Convection in a Channel with an open Cavity Below. *Experimental Heat Transfer*, 21(2), pp.99-114.

Mando, M., & Yin, C. (2012). Euler–Lagrange simulation of gas–solid pipe flow with smooth and rough wall boundary conditions. *Powder Technology*, *225*, 32–42.

Mary Gannon, (2018), 'Hydraulic system contamination: causes and solutions'. Retrieved from https://www.mobilehydraulictips.com/hydraulic-system-contamination-causes-and-solutions/

Matsumoto, Y., & Seki, K. (2008). Implementation of the CIP algorithm to magnetohydrodynamic simulations. *Computer Physics Communications*, 179(5), 289–296.

Maurya, P. R., Rajeev C., Vinil Kumar R.R., & Aravind Vaidyanathan, (2015). Effect of aft wall offset and ramp on pressure oscillation from confined supersonic flow over cavity. *Experimental Thermal and Fluid Science*, 68, 559–573.

Mega, F., Eduardo, & Morais, S., Vinicius, & Vieira, Edson, & Mansur, Sérgio., (2007). Experimental visualization of the flow inside open cavities. 19th International Congress of Mechanical Engineering

Meganthan, A.J. (2000), "An experimental study of low-speed open-cavity flows", Masters thesis, University of Tennessee, Tennessee.

Mehran S., (2011) Numerical investigation of blood flow in a stenosed artery using cubic interpolation profile scheme. Masters thesis, Universiti Teknologi Malaysia, Faculty of Mechanical Engineering.

Mehrez, Z., Bouterra, M., El Cafsi, A., & Belghith, A. (2013). Heat transfer and entropy generation analysis of nanofluids flow in an open cavity. *Computers & Fluids*, 88, 363–373.

Michael, V., S., (2007), "Supersonic Flow and Its Control over Highly Three-Dimensional Cavities" Master Thesis, The Florida State University

Mizoe, H., Yoon, S. Y., Josho, M., & Yabe, T. (2001). Numerical simulation on snow melting phenomena by CIP method. *Computer Physics Communications*, 135(2), 154–166.

Molki, A., Khezzar, L., & Goharzadeh, A. (2013). Measurement of fluid velocity development in laminar pipe flow using laser Doppler velocimetry. *European Journal of Physics*, 34(5), 1127–1134.

Nakamura, T., & Yabe, T. (1999). Cubic interpolated propagation scheme for solving the hyper-dimensional Vlasov—Poisson equation in phase space. *Computer Physics Communications*, 120(2–3), 122–154.

Nakamura, T., Tanaka, R., Yabe, T., & Takizawa, K. (2001). Exactly Conservative Semi-Lagrangian Scheme for Multi-dimensional Hyperbolic Equations with Directional Splitting Technique. *Journal of Computational Physics*, 174(1), 171–207.

Nor Azwadi C. S, and Rosdzimin, M. A. R (2009). Cubic-interpolated-pseudo-particle thermal BGK lattice Boltzmann numerical scheme for solving incompressible thermal fluid flow problem. *Malaysian Journal of Mathematical Sciences*. 3(2), 183-202.

Nor Azwadi C. S, A.S. Ahmad Sofianuddin, K.Y. Ahmat Rajab. (2015) Numerical Prediction of Contaminant Removal from Cavity in Horizontal Channel by Constrained Interpolated Profile Method. *Applied Mechanics and Materials,* Vol. 695, pp. 384-388, 2015

Ogata, Y., & Yabe, T. (1999). Shock capturing with improved numerical viscosity in primitive Euler representation. *Computer Physics Communications*, 119(2–3), 179–193.

Ozalp, C., Pinarbasi, A., & Sahin, B. (2010). Experimental measurement of flow past cavities of different shapes. *Experimental Thermal and Fluid Science*, 34(5), 505–515.

Pialat, X., Simonin, O., & Villedieu, P. (2007). A hybrid Eulerian–Lagrangian method to simulate the dispersed phase in turbulent gas-particle flows. *International Journal of Multiphase Flow*, 33(7), 766–788.

Pirker, S., Kahrimanovic, D., Kloss, C., Popoff, B., & Braun, M. (2010). Simulating coarse particle conveying by a set of Eulerian, Lagrangian and hybrid particle models. *Powder Technology*, 204(2–3), 203–213.

Saadun, M. N. A., Azwadi, C. S. N., Hazwani, M. N., Zakaria, M. S., Manaf, M. Z. A., & Mohd Hanafi, M. H. (2013). Numerical Analysis on the Effects of Cavity Geometry with Heat towards Contaminant Removal. *Applied Mechanics and Materials*, 393, 851–856.

Safdari, A., & Kim, K. C. (2015). Lattice Boltzmann simulation of the three-dimensional motions of particles with various density ratios in lid-driven cavity flow. *Applied Mathematics and Computation*, *265*, 826–843.

Schreiber, R., and Keller, H.B. (1983) 'Driven cavity flows by efficient numerical techniques', *Journal of Computational Physics*, 49(2), pp. 310-333.

Shen, F., Chen, C., & Liu, Z. (2014). Numerical and Experimental Study of the Flow Field Structure Evolution in the Circular Recess of Oil Cavity. *Mathematical Problems in Engineering*, 2014, 1–11.

Sommerfeld, M., & Huber, N. (1999). Experimental analysis and modelling of particle-wall collisions. *International Journal of Multiphase Flow*, 25(6–7), 1457–1489.

Sonin AA (2001), Dimensional analysis. Technical report, Massachusetts Institute of Technology

Stiriba, Y. (2008). Analysis of the flow and heat transfer characteristics for assisting incompressible laminar flow past an open cavity. *International Communications in Heat and Mass Transfer*, 35(8), 901–907.

Stiriba, Y., Ferré, J. A., & Grau, F. X. (2013). Heat transfer and fluid flow characteristics of laminar flow past an open cavity with heating from below. *International Communications in Heat and Mass Transfer*, 43, 8–15.

Stiriba, Y., Grau, F. X., Ferré, J. A., & Vernet, A. (2010). A numerical study of three-dimensional laminar mixed convection past an open cavity. *International Journal of Heat and Mass Transfer*, 53(21–22), 4797–4808.

Takewaki, H. and Yabe, T. (1987). The cubic-interpolated pseudo particle (CIP) method: application to nonlinear and multi-dimensional hyperbolic equations. *Journal of Computational Physics*, 70(2), pp.355-372.

Takewaki, H., Nishiguchi, A. and Yabe, T. (1985) Cubic interpolated pseudo-particle method (CIP) for solving hyperbolic-type equations, *Journal of Computational Physics*, 61(2), 261-268.

Takizawa, K., Tanizawa, K., Yabe, T., & Tezduyar, T. E. (2007). Ship hydrodynamics computations with the CIP method based on adaptive Soroban grids. *International Journal for Numerical Methods in Fluids*, 54(6–8), 1011–1019.

Takizawa, K., Yabe, T., Chino, M., Kawai, T., Wataji, K., Hoshino, H., & Watanabe, T. (2005). Simulation and experiment on swimming fish and skimmer by CIP method. *Computers & Structures*, 83(6–7), 397–408.

Thu, A. M., Byun, Y. H., & Lee, J.-W. (2012). Dye Visualization of the Vortical Flow Structure over a Double-Delta Wing. *Journal of Aerospace Engineering*, 25(4), 541–546.

Toda, K., Ogata, Y., & Yabe, T. (2009). Multi-dimensional conservative semi-Lagrangian method of characteristics CIP for the shallow water equations. *Journal of Computational Physics*, 228(13), 4917–4944.

Tongpun, P., Bumrungthaichaichan, E., and Wattananusorn, S., (2014). Investigation of entrance length in circular and noncircular conduits by computational fluid dynamics simulation. *Songklanakarin Journal of Science and Technology*. 36. 471-475.

Treviño, C., Lizardi, J., & Martínez-Suástegui, L. (2019). Numerical study of magnetohydrodynamic mixed convection and entropy generation of Al2O3-water nanofluid in a channel with two facing cavities with discrete heating. *International Journal of Mechanical Sciences*, *155*, 295–314.

Utsumi, T., & Koga, J. (2002). New numerical method for the solutions of the MCDF equations based on the CIP method. *Computer Physics Communications*, 148(3), 281–290.

Utsumi, T., Kunugi, T., & Aoki, T. (1997). Stability and accuracy of the Cubic Interpolated Propagation scheme. *Computer Physics Communications*, 101(1–2), 9–20.

Utsumi, T., Yabe, T., Koga, J., Aoki, T., Sekine, M., Ogata, Y., & Matsunaga, E. (2004). A note on the basis set approach in the constrained interpolation profile method. *Journal of Computational Physics*, 196(1), 1–7.

van der Hoef, M. A., van Sint Annaland, M., & Kuipers, J. A. M. (2004). Computational fluid dynamics for dense gas–solid fluidized beds: a multi-scale modeling strategy. *Chemical Engineering Science*, 59(22–23), 5157–5165.

Vegendla, S. N. P., Heynderickx, G. J., & Marin, G. B. (2011). Comparison of Eulerian–Lagrangian and Eulerian–Eulerian method for dilute gas–solid flow with side inlet. *Computers & Chemical Engineering*, 35(7), 1192–1199.

Washino, K., Tan, H. S., Salman, A. D., & Hounslow, M. J. (2011). Direct numerical simulation of solid–liquid–gas three-phase flow: Fluid–solid interaction. *Powder Technology*, 206(1–2), 161–169.

Weinan, E., and Liu, J. G. (1996) 'Vorticity boundary condition and related issues for finite difference schemes', *Journal of Computational Physics*, 124(2), pp. 368–382.

Xiao, F., & Yabe, T. (1994). A method to trace sharp interface of two fluids in calculations involving shocks. *Shock Waves*, *4*(2), 101–107.

Xiao, F., Yabe, T., & Ebisuzaki, T. (1999). An oscillation suppressing semi-Lagrangian solver for advection equation. *Computer Physics Communications*, 116(1), 121–135.

Xiao, F., Yabe, T., & Ito, T. (1996). Constructing oscillation preventing scheme for advection equation by rational function. *Computer Physics Communications*, 93(1), 1–12.

Xiao, F., Yabe, T., Nizam, G., & Ito, T. (1996). Constructing a multi-dimensional oscillation preventing scheme for the advection equation by a rational function. *Computer Physics Communications*, 94(2–3), 103–118.

Xiao, F., & Yabe, T. (2001). Completely Conservative and Oscillationless Semi-Lagrangian Schemes for Advection Transportation. *Journal of Computational Physics*, 170(2), 498–522.

Yabe, T, Ogata, Y., Takizawa, K., Kawai, T., Segawa, A., & Sakurai, K. (2002). The next generation CIP as a conservative semi-Lagrangian solver for solid, liquid and gas. *Journal of Computational and Applied Mathematics*, 149(1), 267–277.

Yabe, T., & Aoki, T. (1991). A universal solver for hyperbolic equations by cubic-polynomial interpolation I. One-dimensional solver. *Computer Physics Communications*, 66(2–3), 219–232.

Yabe, T., & Xiao, F. (1995). Description of complex and sharp interface with fixed grids in incompressible and compressible fluid. *Computers & Mathematics with Applications*, 29(1), 15–25.

Yabe, T., Ishikawa, T., Wang, P. Y., Aoki, T., Kadota, Y., & Ikeda, F. (1991). A universal solver for hyperbolic equations by cubic-polynomial interpolation II. Two- and three-dimensional solvers. *Computer Physics Communications*, 66(2–3), 233–242.

Yabe, T., Xiao, F., & Mochizuki, H. (1995). Simulation technique for dynamic evaporation processes. *Nuclear Engineering and Design*, 155(1–2), 45–53.

Yabe, T., Takizawa, K., Xiao, F., Aoki, T., Himeno, T, Takahashi, T. & Kunimatsu, A., (2004). A New Paradigm of Computer Graphics by Universal Solver for Solid, Liquid and Gas. *Jsme International Journal Series B-fluids and Thermal Engineering - JSME INT J SER B*. 47. 656-663.

Yabe, T., Aoki, T., Sakaguchi, G., Wang, P.-Y., & Ishikawa, T. (1991). The compact CIP (cubic-interpolated pseudo-particle) method as a general hyperbolic solver. *Computers & Fluids*, 19(3–4), 421–431.

Yabe, T, Chinda, K., & Hiraishi, T. (2007). Computation of surface tension and contact angle and its application to water strider. *Computers & Fluids*, 36(1), 184–190.

Yabe, T., Mizoe, H., Takizawa, K., Moriki, H., Im, H.-N., & Ogata, Y. (2004). Higher-order schemes with CIP method and adaptive Soroban grid towards mesh-free scheme. *Journal of Computational Physics*, 194(1), 57–77.

Yabe, T., Takizawa, K., Tezduyar, T. E., & Im, H.-N. (2007). Computation of fluid–solid and fluid–fluid interfaces with the CIP method based on adaptive Soroban grids—An overview. *International Journal for Numerical Methods in Fluids*, 54(6–8), 841–853.

Yabe, T., Xiao, F., & Utsumi, T. (2001). The Constrained Interpolation Profile Method for Multiphase Analysis. *Journal of Computational Physics*, 169(2), 556–593.

Ye, M., van der Hoef, M. A., & Kuipers, J. A. M. (2004). A numerical study of fluidization behavior of Geldart A particles using a discrete particle model. *Powder Technology*, 139(2), 129–139.

Yoon, S. Y., & Yabe, T. (1999). The unified simulation for incompressible and compressible flow by the predictor-corrector scheme based on the CIP method. *Computer Physics Communications*, 119(2–3), 149–158.

Zain A. A. M, (2012), "Contaminant removal from rectangular shaped pipeline joints using constrained interpolated profile method". Masters thesis, Universiti Teknologi Malaysia,

Zhu, H. P., Zhou, Z. Y., Yang, R. Y., & Yu, A. B. (2007). Discrete particle simulation of particulate systems: Theoretical developments. *Chemical Engineering Science*, 62(13), 3378–3396.

Zhu, H. P., Zhou, Z. Y., Yang, R. Y., & Yu, A. B. (2008). Discrete particle simulation of particulate systems: A review of major applications and findings. *Chemical Engineering Science*, 63(23), 5728–5770.

LIST OF PUBLICATIONS

1. Nor Azwadi Che Sidik, A.S. Ahmad Sofianuddin, K.Y. Ahmat Rajab, "Numerical Prediction of Contaminant Removal from Cavity in Horizontal Channel by Constrained Interpolated Profile Method", Applied Mechanics and Materials, Vol. 695, pp. 384-388, 2015

2. Nor Azwadi Che Sidik, K.Y. Ahmat Rajab, A.S. Ahmad Sofianuddin, "Removal of Contaminant Effectiveness in Cavity Channel Flow with Different Heated Wall Position", Applied Mechanics and Materials, Vol. 695, pp. 428-432, 2015

3. Nor Azwadi Che Sidik, A.S. Ahmad Sofianuddin, K.Y. Ahmat Rajab, "Transient Removal of Contaminants in Cavity of Mixed Convection in a Channel by Constrained Interpolated Profile Method", Applied Mechanics and Materials, Vol. 554, pp. 312-316, 2014

4. Che Sidik, Nor Azwadi & Rajab, A & Sofianuddin, A. (2014). Four-Sided Lid-Driven Cavity Flow using Time Splitting Method of Adams-Bashforth Scheme. International Journal of Automotive and Mechanical Engineering. 9. 1501-1510. 10.15282/ijame.9.2014.2.0124.