THE ENGINEER STORY

e-ISSN: 3009 - 0792 Volume 4, 202X, 8-11

# EFFECT OF COMPUTATIONAL DOMAIN SIZE FOR FLOW THROUGH PERFORATED PLATE

Chua Kai Feng *, Muhammad Noor Afiq Witri Muhammad Yazid
Faculty of Mechanical Engineering, Universiti Teknologi Malaysia,
81310 UTM Johor Bahru, Johor, Malaysia.
* Corresponding author: chuafeng@graduate.utm.my

## ABSTRACT

Flow straightener is a device capable of eliminating swirl and provide a uniform flow in the presence of high turbulence. One such device is perforated plate or perforated baffle. It is essentially a plate with many small holes on it. Simulating fluid flow across perforated plate by means of computational fluid dynamics (CFD) requires large computational efforts. The objective of this study is to investigate whether the computational domain size with the consideration of symmetry boundary conditions could have a profound effect on the flow quantities. In this study, holes are arranged in staggered manner and three different computational domain sizes are examined based on varying hole quantities: 2, 3, and 4 holes. OpenFOAM has been chosen as the CFD solver in this study, operating under the assumptions of turbulence and steady flow. The outcome of this study revealed that the differences between all computational domain sizes are negligible, exhibiting identical flow characteristics with a velocity percentage difference of less than 0.5%. It can be concluded that a computational domain containing 2 holes is sufficient to replicate the flow characteristics of a full-sized perforated plate, but with the least possible computational effort.

## KEYWORD

Computational fluid dynamics (CFD), flow straightener, perforated plate, computational domain size, pressure drop

## INTRODUCTION

Turbulent flow is caused by highly distorted flow of motion and often requires a flow straightener or flow conditioner to control the flow and make it becomes smooth [1]. Flow straightener is a device which eliminates the swirl and provided fully developed flow from upstream [2]. For a good flow conditioner, it must have the requirements of low pressure loss when passing through device, easier to install, short length of downstream from the source and compact of the combination of device and straightener [3].

The perforated plates flow straightener has numbers of the hole and it make the results of the fragmentation to a flow. As a result, it provides the homogenization of the velocity profile [4]. However, performing computational fluid dynamics (CFD) simulation of the fluid flow across perforated plate with the inclusion of tiny holes can be prohibitively expensive, as there are a lot of number of holes on it. It is a very time-consuming process for 3D CAD modelling and meshing. The large volume of domain and number of cells of meshes require larger computational effort to run simulations. The reason is that the model needs to resolve the smaller scale eddies for every holes of the plate. In this case, reducing the size of the computational domain is one of the most effective ways to reduce the CFD simulation effort as compared to using the entire plate to run the simulation. However, it remains unclear whether the reduction of the computational domain size with the incorporation of symmetry boundary conditions may have a detrimental effect on the accuracy of CFD prediction.

In this study, the effects of different computational domain size selections on fluid flow characteristics across perforated plate is examined. The method of domain selection is by considering only few holes covering partial area of the perforated plate with the assumption of symmetrical flow at the cut plane in CFD simulation.

## MATERIAL AND METHODOLOGY

The perforated plate considered in this study is taken from an actual perforated plate used at the water tunnel facility, located at Fluid Mechanics Laboratory, Faculty of Mechanical Engineering, UTM. As can be seen in Figure 1, the plates are arranged at the rectification section of the water tunnel to ensure good flow quality is achieved at the test section. The perforated dimensions are 1230mm (W) x 215mm (H) with 1.5 mm in thickness. The holes arrangement for the actual perforated plate is staggered and the medium of fluid is water. In this study, SALOME was used to model and discretize the domain whereas OpenFOAM was used to simulate the flow across the perforated plate. The flow conditions imposed in CFD are similar with the typical flow conditions of the facility. The inlet velocity was set to 0.0612 m/s corresponding to a low flow rate value (about 178 L/min) of the water tunnel.

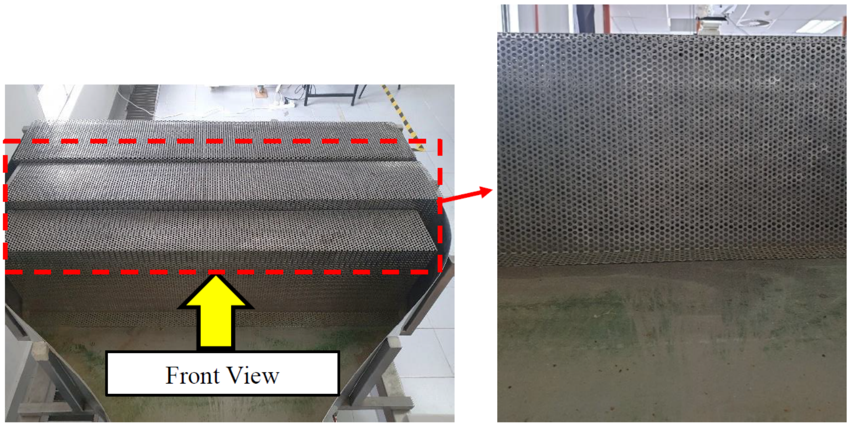


Figure 1: Perforated plate inside water tunnel.

For studying the effect of computational domain sizes, only partial area of the whole perforated plate is considered, which is based on setup found in Bayazit's study [5]. In this study, three different domain sizes was chosen, covering partial area of the whole plate with different number of holes; 2, 3 and 4 holes, as shown in Figure 2. The coverage area was chosen based on the fact that the flow can be considered symmetry at the centreline of each hole.

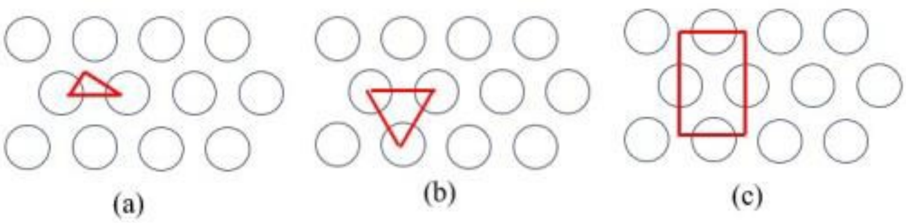


Figure 2: Schematic diagrams for the domain selection from perforated plate (a) domain selection with 2 holes (b) domain selection with 3 holes (c) domain selection with 4 holes.

## RESULT AND DISCUSSION

Figure 3 and 4 shows velocity and pressure distribution, respectively for the three computational domain sizes taken at symmetry plane. Based on the Figure 3, the velocity of fluid accelerated at the hole area for all of the 3 domains because of the obstruction of the plate wall which the cross sectional area decreases. On the others hand, the pressure of fluid reduced significantly when the flow across perforated plate for all of the 3 domains because of the head loss caused by the existing of perforated plate.

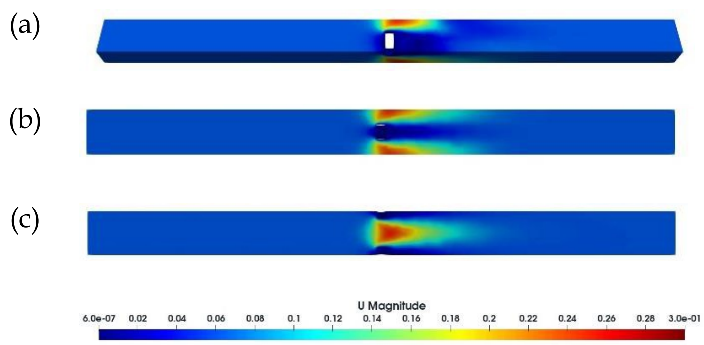


Figure 3: Contour of velocity distribution: (a) domain selection with 2 holes (b) domain selection with 3 holes (c) domain selection with 4 holes.

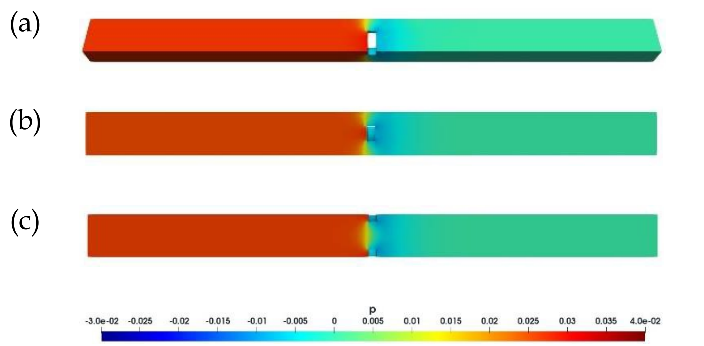


Figure 4: Contour of pressure distribution: (a) domain selection with 2 holes (b) domain selection with 3 holes (c) domain selection with 4 holes.

Figure 5 shows the pressure drop across the perforated plate where for the 2 holes domain, the pressure drop is 0.02702 Pa, the 3 holes domain is 0.02544 Pa and the 4 holes domain is 0.02609 Pa. Based on the results, the domain that had the highest pressure drop is 2 holes domain whereas the lowest pressure drop is 3 holes domain. For the percentage difference of pressure drop, the highest value is 6.21% when compared with 2 and 3 holes domains. For the comparison of 3 and 4 holes domain, the percentage difference for pressure drop is 2.56%. In addition to pressure drop, the difference among the three domain sizes is also compared in terms of percent of velocity difference between the inlet and the velocity at 0.04m from the back of perforated plate. Referring to Figure 6, for 2 holes domain, the percentage of different is the highest with a value of 1.22%. The percentage of difference for 3 holes domain is 1.12% whereas the percentage of difference for velocity is 0.79% for 4 holes domain which is the lowest. Therefore, the percentage difference between 2 holes and 4 holes domain is 0.43% and it can be summarised that differences between all computational domain sizes are negligible and provide the same results when either one of domains is chosen.

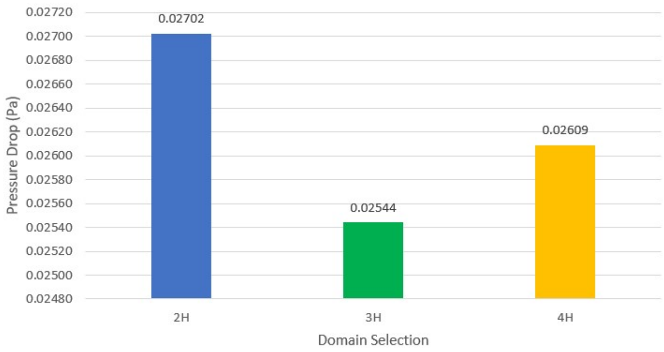


Figure 5: Pressure drop across perforated plate at inlet velocity = 0.0612m/s.

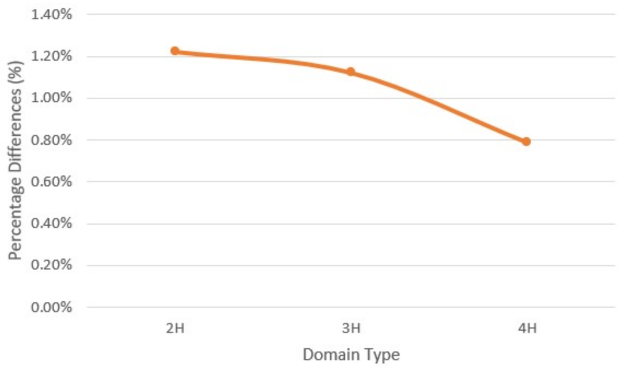


Figure 6: Percentage difference of x-velocity.

**CONCLUSION**

In conclusion, the effect of three different domain sizes on the fluid flow across perforated plate is insignificant and almost have the same characteristics. Quantitatively, the percentage different is 0.43% in terms of velocity. Therefore, two (2) holes domain size is considered to be the most suitable to simplify CFD simulation as it has the lowest volume and number of cell during meshing which can bring the contribution of reducing computational effort and processing time.

**REFERENCE**


[1] N. Ihsak, M. Elyas Muiz, A. Shaffie, M. Amirul, A. M. Azam, and M. A. Razali, "Journal of Complex Flow Experimental Study of Mixing Efficiency in Laminar Flow for Circular Pipe with Perforated Plate Mixer," 2019. [Online]. Available: www.fazpublishing.com/jcf

[2] G. Chen and G. Liu, "Performance Evaluation and Analysis of a New Flow Conditioner Based on CFD," in IOP Conference Series: Materials Science and Engineering, Institute of Physics Publishing, Aug. 2018. doi: 10.1088/1757-899X/394/3/032049.

[3] A. Ahmadi, "Experimental study of a new flow conditioner on disturbed flow in orifice plate metering," Journal of Fluids Engineering, Transactions of the ASME, vol. 131, no. 5, pp. 0511041–0511048, May 2009, doi: 10.1115/1.3114677.

[4] A. T. P. Alho, C. A. L. Conceição, S. H. Sphaier, and P. de Tarso T. Esperança, "Performance of perforated wall for the conditioning of the flow in an ocean basin," in Proceedings of the International Conference on Offshore Mechanics and Arctic Engineering - OMAE, 2008, pp. 487–494. doi: 10.1115/OMAE2008-57653.

[5] Y. Bayazit, E. M. Sparrow, and D. D. Joseph, "Perforated plates for fluid management: Plate geometry effects and flow regimes," International Journal of Thermal Sciences, vol. 85, pp. 104–111, 2014, doi: 10.1016/j.ijthermalsci.2014.06.002.