OPTIMIZATION OF OPEN CHANNEL WATER TUNNEL DESIGN

by Ismail Ismail

THE 15[TH] INTERNATONAL CONFERENCE QUALITY IN RESARCH (QIR) 2017 TEMPLATE

OPTIMIZATION OF OPEN CHANNEL WATER TUNNEL DESIGN

Ismail[a], Erlanda Augupta Pane[a*], Damora Rhakasywi, Eko Prasetyo and Pekik Bayu Asmoro[a]

[a]*Department of Mechanical Engineering, Faculty of Engineering, Universitas Pancasila Srengseng Sawah, Jagakarsa – Jakarta 12640, Indonesia*

ABSTRACT

This research was conducted to develop an open channel water tunnel and analysis of the flow distribution in the test section, where geometry of the water tunnel, discharge of water flow and turbulence intensity are being basic parameters which influence Reynolds number to reach the best condition, so that the hydrodynamic found to be optimal. The previous researches of water tunnel explained that if the Reynolds number is the same fluid, the hydrodynamic on a similar model. The methods of research used numeric analysis and Computational Fluid Dynamics (CFD) simulations. The step of water tunnel design is used to determine the Area Ratio (AR) between the contraction and the test section. Area Ratio value is 5, which take it to calculate water tunnel design. Test section geometry has length is 0.6 m, width is 0.3 m, and height is 0.3 m. Pump PMP010 type is used to pass of water with various capacities. It has capacity about 900 L/min, 1000 L/min, and 1100 L/min, respectively. The optimum result is geometry water tunnel design has length inlet plenum is 0.76 m, contraction is 1.1 m included honeycombs is 0.2 m, test section is 0.6 m and outlet module is 0.51 m, So that, the total length of water tunnel design is 2.97 m. The design of water tunnel using discharge is 900 L/min; water speed is 0.167 m/s, turbulence intensity 0.044 and Froud number 0.097.

Keywords: Froud numbers; Hydrodynamic; Reynolds number; Turbulence intensity; Water tunnel

1. INTRODUCTION

Water tunnel are used for a variety of reasons such as testing a structure under flowing fluid, which affect the forces on the submerged body (Dol 2015). The benefit of water tunnel is the Reynolds number in water flow will be about fifteen times and dynamic head (H_d) in water flow is about 800 times the corresponding than the air flow, So that the water can be a much simpler task to measurement of turbulent pressure fluctuations than in the air (Arakeri & Govinda 1988). Water tunnel has three types are High Speed Water Tunnel, Low Speed Water Tunnel, and Open Channel Flow Water Tunnel (Ahmad Zulfadhli 2008). The previous researches of water tunnel explained that if the Reynolds number is the same

* Corresponding author.
E-mail address : pane_erlanda@yahoo.com

fluid, the hydrodynamic on a similar model. The previous researches of water tunnel design can be showed in Table 1.

Table 1. The previous of design of Water Tunnel

No.	Model Type	Testing Type		Researches
		Design	Experiment	
1	IISC-HSWT water tunnel	✓		(Arakeri & Govinda 1988)
2	DSTO water tunnel		✓	(Erm 1836)
3	NTU water tunnel	✓		(Lee et al. 2005)
4	Close circuit wind/water tunnel design	✓		(Gordon & Imbabi 2016)
5	Hills Research Corporation 1520	✓		(Dol 2015)
6	J.B Herbich water tunnel	✓		(Ko 1971)
7	K15 cavitation water tunnel of SSSRI		✓	(Yao et al. 2015)
8	DTMB high speed water tunnel	✓		(Nedyalkov 2012)
9	Oscillating water tunnel (OWT)		✓	(Yuan & Madsen 2014)
10	K15 cavitation water tunnel of SSSRI		✓	(Chuan-jing & Ting 2011)

This research was conducted to develop an open channel water tunnel. Open channel water tunnel have the six main components are inlet plenum, baffle, setting chamber, inlet module, test section, and outlet module. The six main components of open channel water tunnel can be showed in Figure 1.

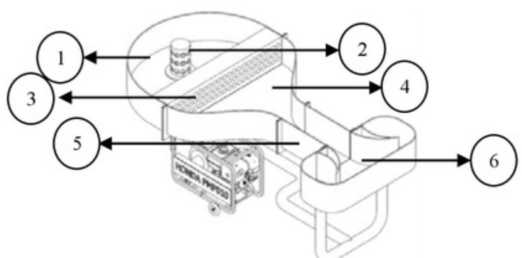


Figure 1 Open channel tunnel design. 1) Inlet Plenum; 2) Buffle Design; 3) Settling Chamber; 4) Inlet Module; 5) Test Section; 6) Outlet Module

The smooth surface of water tunnel can reduce friction of water flow and increase flow effectiveness (Dol 2015). This is being indicated by the flow speed changes with time, where the small different of flow speed along the length of the channel, it suggests the characteristic of this flow is nearly similar with the characteristic of uniform flow (Cengel & Cimbala 2006). Uniform flow has connection with Contraction Ratio (CR) shape, which it has a well design between 5 and 10; it is worthy to note that increasing the contraction ratio increases the maximum width of the contraction while a higher contraction ratio will damp flow perturbations (Kalyankar et al. 2015), can decrease of turbulence intensity (Nedyalkov 2012). The high Contraction Ratio (CR) definitely has a favourable influence on the flow quality in test section to become uniform flow (Arakeri & Govinda 1988). So that, the test section is an important part to determine of water flow characteristic through of inlet module. The settling chamber take the honeycombs model also as an important part to determine of water flow, which reduces the lateral turbulence and straighten the flow (Kalyankar et al. 2015).

The lateral turbulence and straighten the flow, could defined characteristic of water flow was the laminar flow. The characteristic of water flow can be recognized by Reynolds number (Pritchard & Leylegian 2011). The flow will change from laminar to turbulent as the value of Reynolds number increase, which the flow can be considered to be laminar below the value 3 x 10^5, and turbulent above the value 3 x 10^6 (Chuan-jing & Ting 2011). The Reynolds number can influence material drag coefficient, which the increase Reynolds number can make the drag coefficient effect decreases gradually (Yao et al. 2015). This research was conducted to develop an open channel water tunnel and analysis of the flow distribution in the test section, where geometry of the water tunnel, discharge of water flow and turbulence intensity are being basic parameters which influence Reynolds number to reach the best condition, so that the hydrodynamic found to be optimal.

2. METHOD

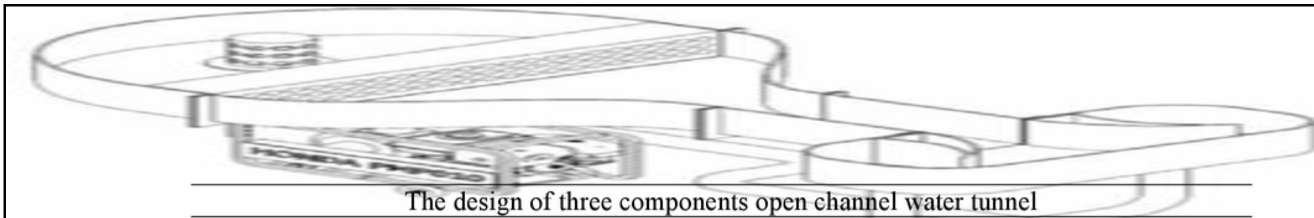
The design of open channel water tunnel used two approaches that the calculation of mathematics models and analysis simulation use Computational Fluid Dynamics (CFD). The calculation of mathematics models has designed the six components open channel water are the inlet plenum, baffle, settling chamber (honeycombs model), test section, inlet module, and outlet module. The calculation of mathematics model uses the basic parameters that discharge capacity and the dimension of test section which have determined by researcher use the Area Ratio (AR). The results of design have simulated in CFD to see the results of water speed, Reynolds number, turbulence intensity, and Froud Number.

II.1. Project and Planning Definition

The project definition of open channel flow water tunnel use the Pump PMP010 type, which it have the discharge capacity various such as 900 l/min, 1000 l/min, and 1100 l/min. Area Ratio (AR) which definition of contraction ratio in inlet module, and dimension of test section. The assumption value of Area Ratio (AR) is 5 and contraction length is 0.9 m, where the both of them can determine of discharge capacity. The test section dimension would be design by the water speed maximum and the lowest of turbulence intensity. The dimension of test section open channel flow water tunnel has length 0.6 m, width 0.3 m and height 0.3 m. The Reynolds number; turbulence intensity and Froud number of water flow must be known. The equations to calculate the three components of open channel water tunnel can be described in Table 2 below that.

Table 2 Calculate of three components open channel water tunnel and basic parameters

No	Parameters	Equations
1	Reynolds number	$R_e = (\rho \, . \, V \, . \, R_h) / \mu$
2	Turbulence Intensity	$I = 0.16 \, . \, R_e^{-1/8}$
3	Froud Number	$F_r = V / (g.Y)^{1/2}$



The design of three components open channel water tunnel		
No	Components	Equations
1	Test Section	$AR = A_2 / A_1$
2	Inlet Module	$Y_c = Y_{C1} - (Y_{C1} - Y_{C0})\ [6(X_c/L_c)^5 - 15(X_c/L_c)^4 - 10(X_c/L_c)^3]$
3	Settling Chamber	$V_{HC} = V_2 / CR$; $V_2 = (A_1 . V_1) / A_2$; $V_1 = Q/A$
		M = The width of inlet module / The number of honeycombs hole

The other section of inlet plenum, baffle, and outlet module can be explained by the assumption value, which the dimension of inlet plenum, baffle, and outlet module is the development the previous open channel water tunnel design.

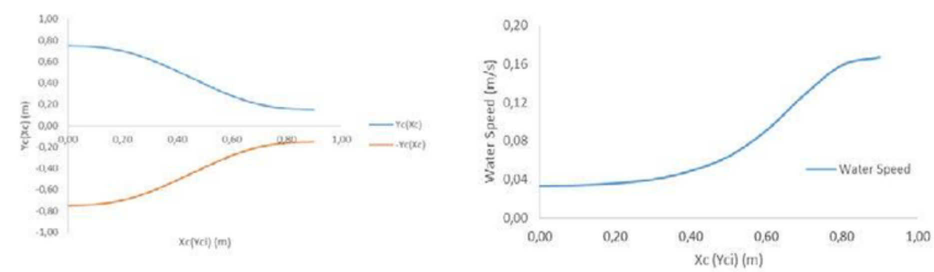
II.2. CFD Simulations

The CFD simulations can be done after calculate of open channel water tunnel design use mathematics model. The basic parameters to CFD simulation are discharge capacity and the calculate result of open channel water tunnel design. The results of CFD can show the water speed, Reynolds number, turbulence intensity, and Froud number.

3. RESULTS AND DISCUSSION

The design calculation have concept the six parts of open channel water tunnel such as inlet plenum design, baffle design, settling chamber which use the honeycombs model, test section design with Area Ratio (AR) optimum between 5 and 10, inlet module design and outlet module. Moreover, the calculate design have been determined the water speed, Reynolds number and turbulence intensity. The six parts of open channel water tunnel can be showed in Figure 1.

The graph in Figure 2 has given the information about the curvatures of geometry inlet module. The smooth curvatures can make the water tunnel distribution are quietly and uniform in Figure 2 (a). The X_c is the distance of contraction between the entrance inlet module and the middle section of inlet module (Y_c) give the effect to water speed, whereas if the length of contraction very high, can make the decrease of water speed because friction between water and inlet module surface area. The Y_c is the start of curvatures in the middle section of inlet module, which the small width of Y_c influences to increase water speed. The water speed in inlet module can be showed in Figure 2 (b).



(a) (b)

Figure 2 The curvatures of inlet module geometry a); The water speed b)

III.4. Design and Validation

The calculation of open channel water tunnel will produce the optimum design with the best result of the basic parameter. The discharge capacity is the ones of basic parameter to determine the optimum open channel water tunnel design. The dimension and parameter design can be showed that in Table 3.

The optimum of open channel water tunnel can influence of Reynolds number, which given the effect to hydrodynamic test. The best of Reynolds number determines the flow characteristic, which it can produce the laminar or turbulent flow. The improvement of kinematic viscosity gives the effect to increase water flow. Based on that condition, the hydrodynamic test on the open channel water tunnel can be optimal.

Table 3 Table of dimension and parameter design

No.	Section	Note
1	Area Ratio (R)	5
2	Discharge (Q) Water Speed (V_s)	900 l/min; 0.033 m/s (load speed)
	Discharge (Q) Water Speed (V_s)	1000 l/min; 0.037 m/s (load speed)
	Discharge (Q) Water Speed (V_s)	1100 l/min; 0.041 m/s (load speed)
3	High of water	300 mm
4	Material	Acrylic with the roughness of 3.92 nm

The CFD simulation, open channel water tunnel design have been seen the water speed, and turbulence intensity in the test section with the discharge capacity and position in the test section become basic parameter. The discharge capacity use the three variant that 900 l/min, 1000 l/min, and 1100 l/min. The simulated position in the test section use five positions that position A (0 m), B (0.15 m), C (0.30 m), D (0.45 m), and the end of test section E (0.60 m) based on the entrance of test section. The positions in the test section can be showed that in Figure 3.

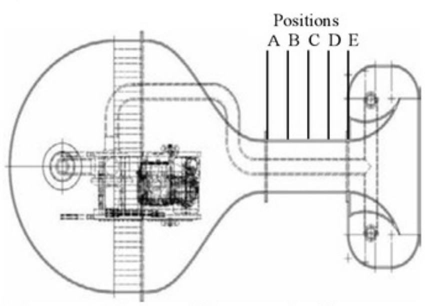
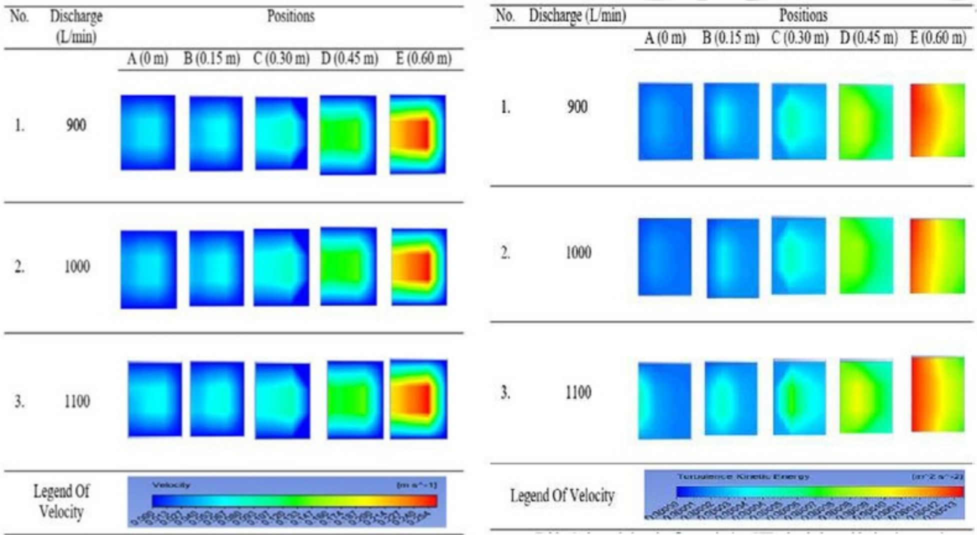


Figure 3 The position test in the test section

The result of CFD simulation with the parameter that position and discharge capacity can be determines the water speed and turbulence intensity. The result simulation can be showed that in the Figure 4.



(a) (b)

Figure 4 water speed a) and turbulence intensity b) in Test Section

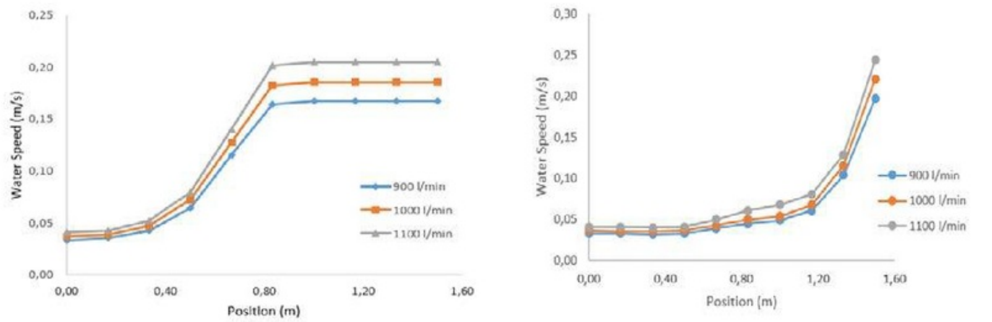
Figure 4 a) showed that the water speed CFD simulation with the three several of discharge capacity in the five positions. The trend of the water speed value between the three variant of discharge capacity have been improved. This is can be indicated by the change of colour when the calculation of water speed is being tested by CFD simulation. Improvement of water speed influenced by discharge capacity and the area of surface test section. The two factors have been relations with the water speed; this is can be showed in the discharge equations. If the discharge capacity increases, the water speed can be increased when the surface area constant. Figure 4 b) showed that the turbulence intensity CFD simulation in the five positions with the three variant of discharge capacity. The trend of turbulence intensity has been similarity with the water speed. The turbulence intensity has been increased in the three variant of discharge capacity. The turbulence intensity between 900 l/min, 100 l/min, and 1100 l/min has the different value.

III.5. The result of validate calculation

The result of validate calculation and the calculate of open channel water tunnel design simulated by Computational Fluid Dynamics (CFD) on the three various discharge capacity to determine water speed in Figure 5, Reynolds number in Figure 6 and turbulence intensity in Figure 7. This simulation have the aims to consideration result from the calculate design and the simulations, moreover this simulation can be managed of the operational cost to build the open channel water tunnel.

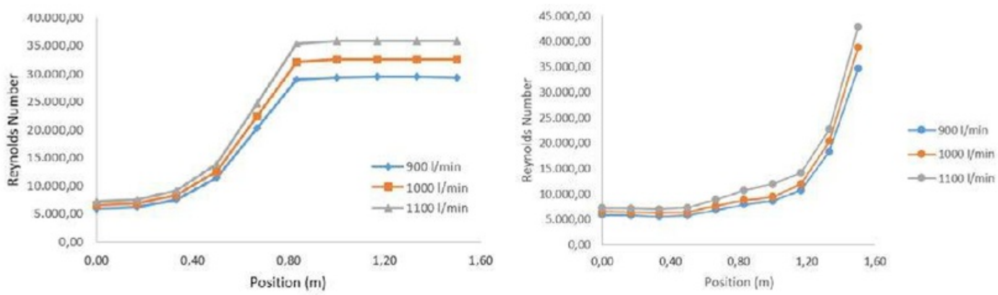
The graph of water speed showed in Figure 5(a) the different water speed between the three several of discharge capacity. The discharge capacity 900 l/min has the lowest water speed than the other. This is have the relation with the surface of area test section. The constant of surface area with the change of discharge capacity can give the effect to water speed. This condition can be concluded that the increasing discharge capacity can make the increase of water speed. The trend of water speed CFD simulation in Figure 5(b) similarity with the calculate design, but the different of graph water speed both of them on the path line. The parameter that makes the different is external parameter such as roughness material parameter, when it's become the entrance data simulations.

The water speed influence the Reynolds number in Figure 6 (a). The Reynolds number can determine the flow characteristic that laminar or turbulent flow. The graph of Reynolds number open channel water tunnel design showed that the discharge capacity 900 l/min has the lowest of Reynolds number. The trend of Reynolds number has been similarity with the water speed. Graph of Reynolds number CFD simulation in Figure 6 (b). The graph of Reynolds number CFD simulation different than the graph of Reynolds number validates calculation, because the entrance data of water speed in CFD simulations. The highest of Reynolds number makes the flow characteristic become as a turbulent flow.



(a) (b)

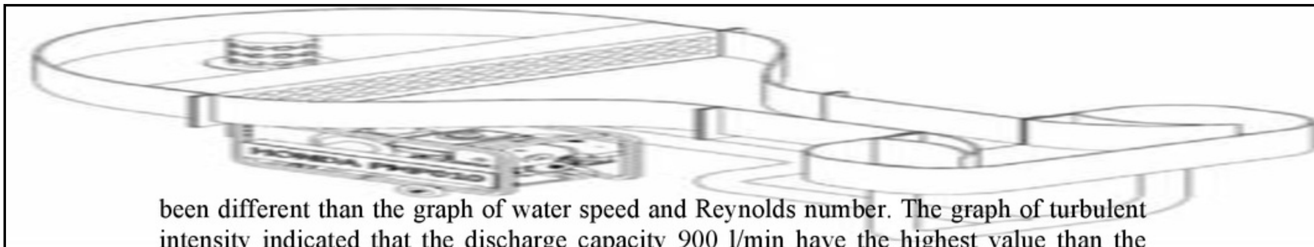
Figure 5 Water speed a); Validate Calculation b); CFD simulations



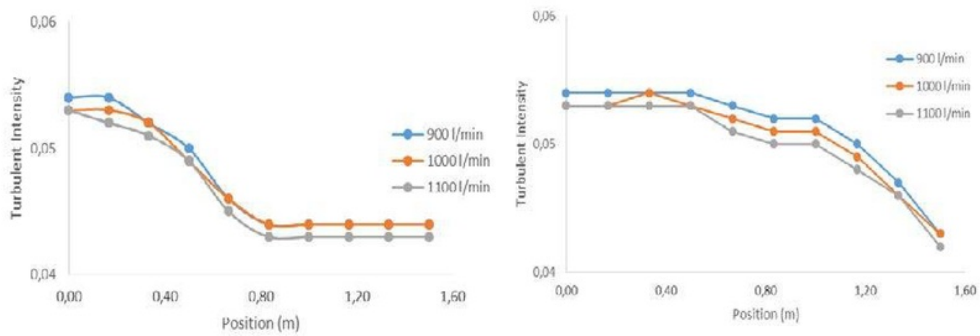
(a) (b)

Figure 6 Reynolds number a); Validate Calculation b); CFD simulation

Besides of water speed and Reynolds number, the turbulence intensity can influence the open channel water tunnel design. The graph of turbulence intensity in Figure 7 (a) has



been different than the graph of water speed and Reynolds number. The graph of turbulent intensity indicated that the discharge capacity 900 l/min have the highest value than the other discharge capacity. This can conclude that the highest of turbulent intensity needed to the lowest of Reynolds number and makes the increase of water speed. The turbulence intensity of CFD simulation in Figure 7(b) and calculate design have been different. Graph of turbulence intensity CFD simulation when the water flow through the position E will be fall down to the lowest point. The lowest point caused by Reynolds number influence the turbulence intensity parameter.



(a) (b)

Figure 7 Turbulent Intensity a); Validate Calculation b); CFD simulation

III.6. Optimal Design

The design of open channel water tunnel has been optimized by the discharge capacity 900 l/min. The characteristic of optimal design open channel water tunnel have the length of contraction in 0.9 m, and the dimension of test section with the length 0.6 m, width 0.3 m, and the height 0.3 m. The figures of water speed in test section can be coughed from 0 m/s until 0.3048 m/s. The test visualization of open channel water tunnel uses the colour spray on the surface model. The best of water speed from the test visualization is 0.09144 m/s until 0.1524 m/s. The optimum of open channel water tunnel design can be showed that in Figure 1. The optimized of open channel water tunnel with the discharge capacity 900 l/min have been selected. The parameters of open channel water tunnel design such as water speed, Reynolds number, and turbulence intensity between calculation and CFD simulation can be considerate ones to each other, and can be showed that in Figure 8.

From the graph, it seems the different result between the calculate design and CFD simulations. The graph of water speed can be showed in Figure 8(a). The graph of water speed in the calculate design have the extreme increase when the water flow through the position C and position D, and the constant water speed in position E. The different result caused by the added external parameter, such as the roughness material test section when the open channel water tunnel design simulated by CFD simulations. The graph of Reynolds number between calculates design and CFD simulations have different result in

Figure 8 (b). The different caused by the constant value of material characteristic, when material characteristic data in CFD simulations different than in calculate of design. The graph of turbulence intensity between calculate design and CFD simulations have been different in Figure 8 (c), but the value in the entrance of test section, both of them have the similarity value. The value of turbulence intensity calculate design have been extreme decreased path line than the smoothly path line of turbulence intensity CFD simulations. The graph of Froud number between calculates design and CFD simulation in Figure 8(d). Froud number of water flow in CFD simulations have the decreasing path, but in the position of 0.8 m Froud number in CFD simulations increase, the increase of Froud number because the water flow through test section, the function of test section to make the water flow quickly and smoothly.

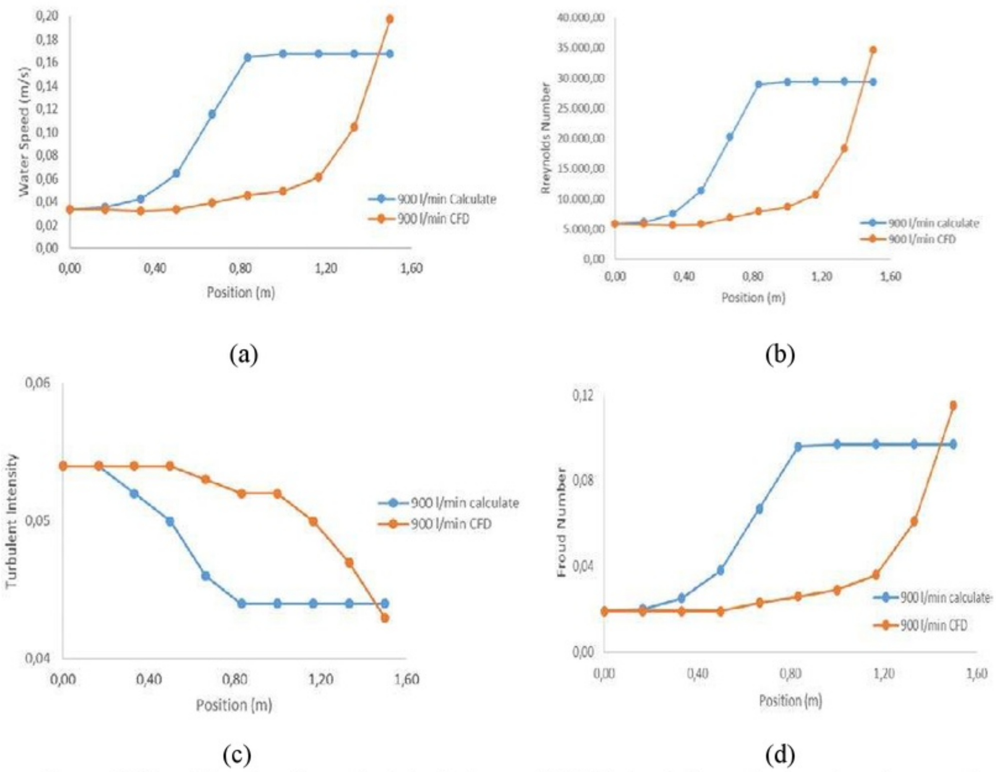
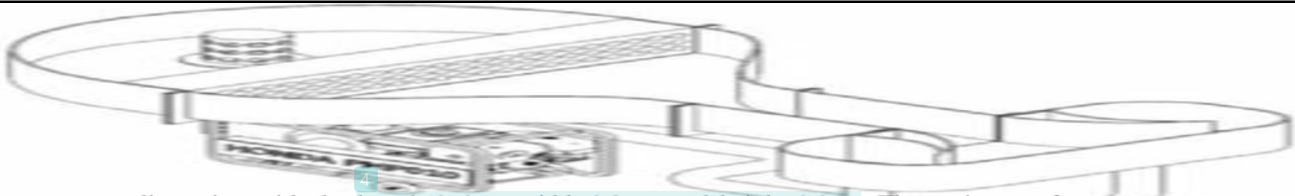


Figure 8 Consideration the calculate design and CFD simulations design a) water speed b)Reynolds number;c)Turbulent Intensity;d) Froud Number

4. CONCLUSION

The research of open channel flow water tunnel can be concluded that the optimum of open channel flow water tunnel model with the discharge capacity 900 l/min. This model has the total length 2.97 m. The length of contraction is 0.9 m included the honeycombs with the dimension of length 0. 2 m and the diameter of cell 0.05 m. The test section has the



dimension with the length 0 6 m, width 0.3 m, and height 0.3 m. The optimum of open channel water tunnel have the optimum water speed with the value is 0.167 m/s , the lowest Reynolds number 29368, and turbulence intensity is 0.044 or with the percentage is 4.4 %, which have the means is intermediate turbulence intensity and Froud number 0.097. If the water flow rate increase, its produce the Reynolds Number high and the Reynolds Number will be constant when the water flow entrance to test section, this is being caused by the constant test section geometry. If the flow discharge increases, its produce turbulence intensity low and the turbulence intensity will be constant when the water flow entrance to test section, this is being also caused by constant test section geometry.

5. ACKNOWLEDGEMENT

The authors are grateful to the support simulation team in laboratories (Mr. Kismono, and Mr. Eko Prasetyo).

6. REFERENCES

Ahmad Zulfadhli, 2008. *Design, Analysis, and Fabricate of Low Speed Water Tunnel*. Universiti Malaysia Pahang.

Arakeri, V.H. & Govinda, H.S., 1988. The High Speed Water Tunnel Facility at The Indian Institute of Science. *Sadhana*, 13(December), pp.223–235.

Cengel, Y. & Cimbala, J., 2006. *Fluid Mechanics Fundamentals and Applications*, New York: McGraw-Hill.

Chuan-jing, L.U. & Ting, S.I., 2011. Water Tunnel Experimental Investigation on The Drag Reduction Characteristics of The Traveling Wavy Wall *. *Journal of Hydrodynamics*, 23(1), pp.65–70.

Dol, M.Z. and S.S., 2015. Design and Development of Low-Cost Water Tunnel for Educational Purpose. *Material Science and Engineering*, 78, pp.1–8.

Erm, L.P., 1836. *Development and Use of a Dynamic-Testing Capability for the DSTO Water Tunnel*, Australia.

Gordon, R. & Imbabi, M.S., 2016. CFD Simulation and Experimental Validation of a New Closed Circuit Wind / Water Tunnel Design. *Fluids Engineering*, 120, pp.311–318.

Kalyankar, H. et al., 2015. Design and Analysis of Low Speed Water Tunnel for Flow Visualization of Bluff Body. In *2nd International Conference on Advances in Mechanical Engineering and Its Interdisciplinary Areas*. Kolaghat, India: Academia.edu, pp. 49–57.

Ko, S.C., 1971. *Design and Construction of a water tunnel*, Pennsylvania.

Lee, C.S. et al., 2005. Development of 3-Component Force-Moment Balance for Low Speed Water Tunnel. *Modern Physics Letters B*, 19, pp.1575–1578.

Nedyalkov, I., 2012. *Design of Contraction , Test Section , and Diffuser for a High-Speed Water Tunnel*. Chalmers University of Technology.

Pritchard, P.J. & Leylegian, J.C., 2011. *Introduction to Fluid Mechanics* Eighth Edi., Manhattan: John Wiley & Sons, INC.

Yao, Y. et al., 2015. Water tunnel experimental investigation on drag reduction of coating surface wall. *Procedia Engineering*, 126, pp.247–253.

Yuan, J. & Madsen, O.S., 2014. Experimental study of turbulent oscillatory boundary layers in an oscillating water tunnel. *Coastal Engineering*, 89, pp.63–84.

OPTIMIZATION OF OPEN CHANNEL WATER TUNNEL DESIGN

ORIGINALITY REPORT

12%	3%	9%	4%
SIMILARITY INDEX	INTERNET SOURCES	PUBLICATIONS	STUDENT PAPERS

PRIMARY SOURCES

1	Submitted to Universitas International Batam Student Paper	3%
2	M Zahari, S S Dol. "Design and Development of Low-Cost Water Tunnel for Educational Purpose", IOP Conference Series: Materials Science and Engineering, 2015 Publication	2%
3	V H Arakeri. "The high speed water tunnel facility at the Indian Institute of Science", Sadhana, 12/1988 Publication	1%
4	www.mdpi.com Internet Source	1%
5	Zahari, M, and S S Dol. "Design and Development of Low-Cost Water Tunnel for Educational Purpose", IOP Conference Series Materials Science and Engineering, 2015. Publication	1%

Yan YAO, Chuan-jing LU, Ting SI, Kun ZHU.

6	"Water tunnel experimental investigation on the drag reduction characteristics of the traveling wavy wall", Journal of Hydrodynamics, Ser. B, 2011 Publication	1%
7	Yu-chun Lin. "Pulsatile Flow Past a Cylinder: An Experimental Model of Flow in an Artificial Lung", ASAIO Journal, 11/2006 Publication	1%
8	homework.northport.k12.ny.us Internet Source	<1%
9	Agri Suwandi, Tresna P. Soemardi, Gandjar Kiswanto, Widjajalaksmi Kusumaningsih, I. Gusti Agung I. G. W.. "Development of friction and wear full-scale testing for TKR prostheses with reliable low cost apparatus", AIP Publishing, 2018 Publication	<1%
10	Submitted to Universiti Teknologi MARA Student Paper	<1%
11	Yasaman Farsiani, Brian R. Elbing. "Characterization of the Custom-Designed, High Reynolds Number Water Tunnel", Volume 2, Fora: Advances in Fluids Engineering Education; Cavitation and Multiphase Flow; Fluid Measurements and Instrumentation, 2016 Publication	<1%

12	www.carmeliet.arch.ethz.ch Internet Source	<1%
13	shareok.org Internet Source	<1%
14	espace.curtin.edu.au Internet Source	<1%
15	www.freepatentsonline.com Internet Source	<1%
16	Yan Zhang, Eldin Wee Chuan Lim, Chi-Hwa Wang. "Pneumatic Transport of Granular Materials in an Inclined Conveying Pipe: Comparison of Computational Fluid Dynamics−Discrete Element Method (CFD−DEM), Electrical Capacitance Tomography (ECT), and Particle Image Velocimetry (PIV) Results", Industrial & Engineering Chemistry Research, 2007 Publication	<1%

Exclude quotes	Off	Exclude matches	Off
Exclude bibliography	On		

OPTIMIZATION OF OPEN CHANNEL WATER TUNNEL DESIGN

GRADEMARK REPORT

FINAL GRADE

/0

GENERAL COMMENTS

Instructor

PAGE 1

PAGE 2

PAGE 3

PAGE 4

PAGE 5

PAGE 6

PAGE 7

PAGE 8

PAGE 9

PAGE 10