

MALAYSIA WATER RESEARCH JOURNAL

Vol. 8 (Bil. 1, 2018)

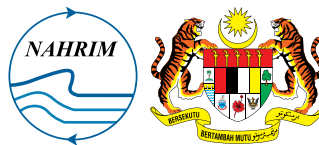


INSTITUT PENYELIDIKAN HIDRAULIK KEBANGSAAN MALAYSIA (NAHRIM)
KEMENTERIAN AIR, TANAH DAN SUMBER ASLI (KATS)



MALAYSIA WATER RESEARCH JOURNAL

Vol. 8 (Bil. 1, 2018)



Institut Penyelidikan Hidraulik Kebangsaan Malaysia (NAHRIM)

National Hydraulic Institute of Malaysia (NAHRIM)

Kementerian Air, Tanah dan Sumber Asli

Ministry of Water, Land and Natural Resources (KATS)

Lot 5377, Jalan Putra Permai, 43300 Seri Kembangan, Selangor

Chief Editor

YBhg. Datuk Ir. Dr. Azuhan b. Mohamed

Deputy Chief Editor

Ir. Hj. Mohd Fauzi b. Mohamad

Associate Editor

Ir. Dr. Safari b. Hj. Mat Desa

Ismail b. Hj. Tawnie

Abdul Taib b. Abdillah

Hj. Mohd Kamarul Huda b. Samion

Muhammad Rizal b. Razali

Secretariat

Al Imran b. Abdul Manan

Nur Amira bt. Zahari

©Malaysia Water Resources Journal is copyright of National Hydraulic Research Institute of Malaysia (NAHRIM). Any reprinting, rewrite, reuse or modifications must be subject to approval.

Print by

Ezrich Group Sdn Bhd

Printed year

November 2018

ISSN No

2232-0016



Table of Contents

	PHYSICAL HYDRODYNAMIC MODELLING STUDY FOR WAVE INTERACTION ONTO
6	SUBMERGED BREAKWATER Safari Mat Desa, Othman Karim & Kamarul Huda Samion
10	PUMP SUMP MODEL TESTING FOR SUNGAI BELIBIS PUMP STATION Mohd Kamarul Huda Samion, Mohd Fauzi Mohamad & Khairul Azuan Adam
18	EXPERIMENTAL STUDY ON PROPELLER PERFORMANCE OF A VESSEL NEAR BANK AND IN SHALLOW WATER Mehdi Nakisa, Fatemeh Behrouzi, Adi Maimun & Mohamed Yasser Ahmed
28	COST-EFFECTIVE, LAB-SCALE AUTOMATED SPATIAL AND TEMPORAL MEASUREMENT SYSTEM FOR LOCAL SCOUR AROUND BRIDGE PIERS Mojtaba Porhemmat & Wan Hanna Melini Wan Mohtar
35	STUDY OF FLOW CHARACTERISTICS IN MENGKUBAU DAM SPILLWAY Che Mohamad Amirus Shafiq Che Ismail, Mohd Remy Rozainy Mohd Arif Zainol, Ismail Abustan, Mohd Zulkifli Abdullah & Nuridah Sabtu
41	OPTIMUM POSITIONING OF BUFFLE BLOCKS IN STILLING BASIN FOR MENGKUBAU DAM Ismail Abustan, Nuridah Sabtu, Mohd Remy Rozainy Mohd Arif Zainol, Mohammad Khairy A. Wahab & Mohd Azmi Ismail
52	AN EXPERIMENTAL ASSESSMENT OF BOAT-GENERATED WAVE BY LANGKAWI GEOFOREST KARST TOURIST BOATS Omar Yaakob, Faizul Amri Adnan, Mohd Arif Ismail, Mohamad Hidayat Jamal, Ahmad Hadi Mohamad Rashidi, Wan Ahmad Hafiz Wan Mohd Azhary, Mohd Kamarul Huda Samion, Andiknurdiyana Bachok, Badruzzaman Ahmad & Nur Shafira Ahmad
67	CALIBRATION OF A DRAG TILT CURRENT VELOCIMETER FOR AN OPEN CHANNEL APPLICATION Ibnu Akil Maitar Muharram, Zool Hilmi Ismail & Kamarul Bahrin Mohamad



PHYSICAL HYDRODYNAMIC MODELING STUDY FOR WAVE INTERACTION ONTO SUBMERGED BREAKWATER

Safari Mat Desa^{12a}, Othman A. Karim¹, Mohd Kamarul Huda Samion²

ABSTRACT

Pre-fabricated narrow crest submerged breakwater provide option for modeler in hydrodynamic wave modeling. Wave transmission is one of the components resulted from interaction between incident wave and coastal structure. Hydrodynamic factors influenced wave transmission were wave period, wave height and water depth whilst structural factors were structural height, bottom width as well as crest width. A systematic test program was undertaken in a monochromatic unidirectional wave condition. This narrow crest submerged breakwater also acts as artificial reef which protect and converse marine biology as well as enhance the marine ecological environment. This 2-D modeling study presents transmission coefficient as the reference index of breakwater's wave attenuation in the effect of submergence depth and wave height.

Keywords: wave attenuate, pre-fabricated structure, transmission coefficient, monochromatic regular wave.

¹ Faculty of Engineering and Built Environment, National University of Malaysia.

² Senior Research Officer, National Hydraulic Research Institute of Malaysia (NAHRIM). (Corresponding Author: safari@nahrim.gov.my)

Introduction

In the past two decades submerged breakwaters have been increasingly used in coastal management for protecting beaches from erosion without spoiling landscape. This is increasingly important in recreational and residential coastal developments. In addition, submerged breakwaters absorb some of incoming wave energy thus diminishing the transmitted wave energy. The effects of breakwater submergence, wave period, crest width of breakwater, different depth of submergence on wave hydrodynamics were analysed depending on site condition, different parameter of wave propagation and structural type (Ahmad Mustaffa et al. 2014). Owing to the outlined engineering importance, researchers and engineers have tended to focus on the interactions between breaking solitary wave (Mead S.T et al., 2002), non-breaking regular wave (Losada et al., 2005), directional random wave (Klopman et al. 2008), multi-directional random wave (Hur D.S et al., 2003) and submerged breakwaters. Studies of hydrodynamic effects of submerged breakwaters on the sloping bed can be fully applied to the essential characteristics in practice. This paper present pre-fabricated narrow crested structure, Wave Breaker and Coral Restorer (WABCORE), a simple trapezoidal design of narrow crest subject to hydrodynamic character of each and different places developed by NAHRIM.

Experimental Set-Up.

A 2-D hydrodynamic physical model were carried out in National Hydraulic Research Institute Of Malaysia (NAHRIM). Generally, the largest possible scale is selected subjected to the test facility capability. The smaller the scale, might affect repeatability by which the small scale wave can be reproduced in the laboratory.

Thus, the flume capability is vitally important parameter before any scale effect were chosen. For this particular experiment, Froude Law with scale effect of 1:4 has been determined. Waves forces were calibrated in the flume before the structure is built to ensure the incident wave measurements are not corrupted by reflected waves. The first, second and third wave probe were respectively installed at 17, 21 and 23 meter down the flume whilst WP4 and WP5 were set at 2 meter and 5 meter after the centre line of structure.

WABCORE Construction.

The pre-fabricated of WABCORE were tested in arrangement of bottom up, 4-3-2-1 trapezoidal shape in which comply to geometric similarity while wave propagation compliance to kinematic and dynamic similarity.

Test Parameter.

Actual primer wave data were collected earlier in Pulau Tinggi, Johor by using of Acoustic Doppler Current Profiler (ADCP) as shown in Table 1.

Table 1: Test Parameter.

Wave Type	Monochromatic
Wave Period, T_p (s)	2.50, 2.0, 1.75, 1.50
Wave Height, H_i (m)	0.10, 0.15, 0.20, 0.25
Water Depth (m)	0.50, 0.75, 1.0, 1.25
Frequency (hz)	0.40, 0.50, 0.57, 0.66

Result and Discussions

Transmission coefficient (C_t) of the wave over the submerged breakwater was quantified by the transmitted wave height divided by the incident wave height, $C_t = H_t/H_i$. The greater the value of C_t , the worse effect of wave attenuate to the leeside of the breakwater and vice versa.

Effect of Submergence Depth for Transmission Coefficient.

Crest level height plays a very significant role in wave transmission. High crest level gives low transmission as compared to low crest level. Incident wave energy dissipated when relative submergence depth, d_s/H_i more than 0. Transmission coefficient increase when submergence depth increase due to the decreasing in effective structure height as shown in Figure 1.

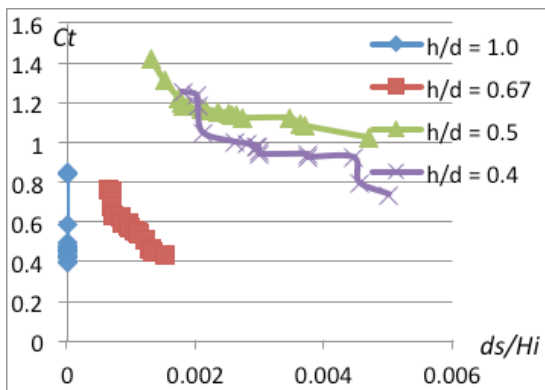


Figure 1: Effect of Submergence Depth

Effect of Incident Wave Height for Transmission Coefficient.

Effect Of Incident Wave Height For Transmission Coefficient. Different incident wave heights were induced by changing the wave generator setting parameter. Result shows at the minimum water depth and with small incident wave height, C_t value is extremely high which indicates low impact of wave transformation. Nevertheless at the minimum water depth and bigger incident wave height, C_t value becoming lesser that proved wave height is an important factor for wave attenuate. At the relative water depth, $h/d=1.0$ and incident wave height generated at 0.25m, value of C_t is the smallest at 0.429

shows wave dissipated tremendously as shown in Figure 2.

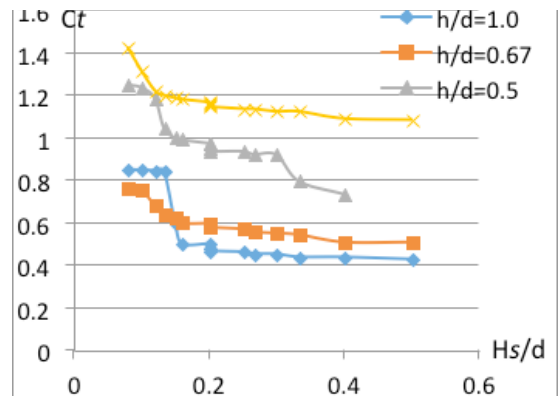


Figure 2: Effect of Incident Wave Height

Conclusion

Plotted C_t for WABCORE shows an amazing performance for wave transmission. Subject to different sea state condition such as water depths (m), wave periods (s), incident wave height (m) and wave length (m), indicated high wave suppression and capability of WABCORE that promotes wave breaking and structural friction to wave motion. Physical sea state condition such as bigger incident wave height, effective submergence depth, crest width span and wave period shows correlation to the performance of WABCORE.

The dissipation of wave nearly 60% of incident wave height in the test limit of relative water depth, $0.4 \leq h/d \leq 1.0$; submergence depth, $0.0 \leq d_s/H_i \leq 0.00049$ and incident wave height, $0.08 \leq H_s/d \leq 0.5$ in regular monochromatic wave.

References

- Ahmad Mustafa Hashim et al. Wave Attenuation of Interlocking Concrete Unit - V (ICU-V). Applied Mechanics and Materials, Vol. 567, 313–318, 2014.
- D.S.Hur, N. Kawashima, K. Iwata: Similarity solution of oblique impact of wedge shaped water column on wedged coastal structures. Journal of Coastal Engineering, 56(4), 400–407, 2003.
- I.J. Losada,. R. Silva, M.A. Imada: Modeling of velocity and turbulence fields around and within low crested rubble mound breakwaters. Coastal Engineering Journal, 52, 887–913, 2005.
- Klopman, G. & Van der Meer, J. W. 1998. Random wave measurements in front of reflective structures. Journal of Waterway, Port, Coastal and Ocean Engineering. Vol 125. pp. 39-45.
- J. Chen, C.B. Jiang, Y.G. Cao et al.: Science paper Online.7, p.516, 2008.
- Mead, S. T. & Black, K. . Multi-Purpose Reefs Provide Multiple Benefits – Amalgamating Coastal Protection, High-Quality Surfing Breaks and Ecological issue November (2002). pp. 1-17, 2002.

PUMP SUMP MODEL TESTING FOR SUNGAI BELIBIS PUMP STATION

Mohd Kamarul Huda Samion, Mohd Fauzi Mohamad & Khairol Azuan Adam

ABSTRACT

A new Pekan, Pahang flood mitigation project was implemented by Malaysian Government due to frequent flooding event causing various losses of property and lives. A pump station was establishing just downstream of the Belibis river for diverting the discharge to reduce the impact of the flooding. Hydraulic model studies are necessary for the pump sumps intake due to the operational requirements of the pumps in a limited space environment which can leads to hydraulic problems. The purposes of this hydraulic model study is to identify undesirable flow conditions in the pump sump model such as vortices, pre-rotation of flow and uneven distribution of flow and propose improvements to the pump sump prototype based on model testing results. Based on the optimum model discharge, an undistorted scale model of 1:10 was adopted. The model features four pumps (7.91 l/s for pump 1 and 2, and 4.74 l/s for pump 3 and 4) with different values of water depth in total of 4 cases of study. Pump flow rate measurements were obtained by ultrasonic flow meter and swirl angle in the suction intakes were measured by a vortimeter. No vortices were occurred near the suction intake for all cases but an uneven flow through the suction intake were detected in some cases. A minor modification is required by installation of buffer block directly under the pump intake column. During detail testing and analysis, the installation of the buffer shown a better flow distribution on all cases.

Keywords: Hydraulic, Physical Model, Pre-rotation flow, Pump Sump, Vortices.

¹ National Hydraulic Research Institute of Malaysia (NAHRIM). (Corresponding Author: kamarul@nahrim.gov.my)

Introduction

Pekan city in state of Pahang, Malaysia and its adjacent areas have been experiencing frequent flooding causing hardship to the local population and significant damages to the economy. To reduce the impact, a new Pekan Flood Mitigation Project was implemented by Malaysian Government whereby pump station was established just downstream of the Belibis river for diverting the excess flows. Hydraulic model studies are necessary for the pump sumps intake due to the operational requirements of the pumps in a limited space environment which can lead to hydraulic problems such as attached surface vortices; submerged vortices; air entrainment from inflow; swirl and undulating flow; and dead flow regions.

Physical modelling is commonly used during design stages to optimize a structure and to ensure a safe operation of the structure (Chanson, 1999). When used in tandem with numerical modelling, this approach leads to great success for fluid-structure interaction studies and discharge capacity evaluation (Archembau et al., 2001; Piroton et al., 2001; Piroton et al., 2003). Physical model testing is recommended for pumping stations in which the geometry differs from recommended standards, particularly if previous experience with them is not available. Model testing can also be employed to seek solutions to problems in existing installations. If the source of a problem is unknown, it can be less expensive to determine the cause and find remedies by model studies rather than by trial and error at full scale. In this pump sump model test, the pumping station will house 2 nos. of submersible pumps (main pump) and 2 nos. of submersible pumps (jokey pump).

The objectives for this hydraulic studies of pump sump model are to identify undesirable flow conditions in the pump sump model such as vortices and pre-rotation flow and also to propose improvements to the pump sump prototype based on model testing results.

The scope of work for the studies are design and construct the physical model of the pump sump; Carry out hydraulic studies on the pump sump model to confirm the suitability of the designed pump sump; Recommendation on modification of pump sump design based on the outcomes of the initial hydraulic studies on the pump sump model; And also carry out subsequent testing on modified pump sump model to confirm the performance of the modified pump sump design.

Hydraulic Model Simulation

Froude number modelling is typically used when friction losses are small and the flow is highly turbulent (Chanson, 1999). As the flows studied were mainly controlled by gravity and as the friction losses could be negligible, studies were adopted with the same ratio between inertia and gravity forces as on the prototype. This similarity results in the conservation between model and prototype of the non-dimensional number of Froude (Ercicum et al., 2008). After thorough considerations for the most optimum configuration, an undistorted model scale of 1:10 ratio was decided. Thus, in compliance with the Froude Law the corresponding model and prototype relationships was summarised in Table 1.

Table 1. The relationship between prototype and model values.

Property	Scale Relationship	Model Value
Geometric	$S = L_m/L_p$	1/10
Velocity	$V_m = V_p S^{0.5}$	1/3.16
Flow	$Q_m = Q_p S^{2.5}$	1/316
Time	$T_m = T_p S^{0.5}$	1/3.16

A geometrically pump sump model with a scale ratio of 1:10 similar to the prototype had been constructed for this study. Figure 1 and 2 below shows the layout plan and side view of the study area.

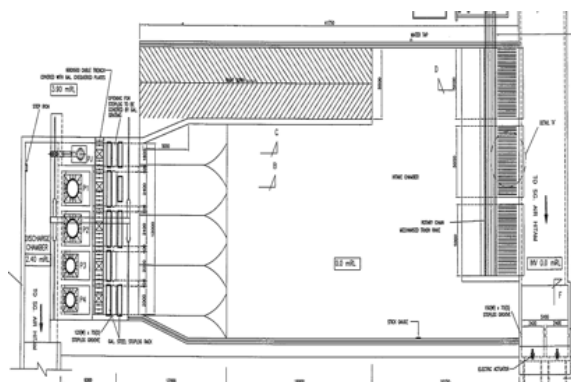


Figure 1. Detail plan view of the pump sump.

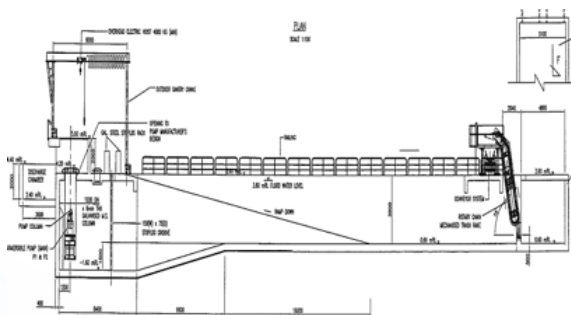


Figure 2. Overall side view of the pump sump.

In order for the pump sump model to maintain the water level at a constant value during observation, a closed loop system using a circulating pump had been built (please refer to Figure 3). This system also enables the regulation of water level in the pump sump model by changing the

volume of water contained. In the closed loop of flow system, the flow will be drawn through the pump sump model to the suction side of the external circulating pumps and then into the intake structure of the pump sump model. All the flow for the model testing is measured using the ultrasonic flow meter. In order to control the regulation of flow and the delivery head, a butterfly valve is incorporated in each of the external circulating pumps as a control valves.



Figure 3. Water circulation system and scale model of the pump sump.

The assessment of the pump sump model test involved observing the approach flow pattern towards each operating pump intake with the aid of a blue dye tracer. This gave a good insight as to how the pumps would respond since any flow that departs significantly from the one of the steady flow is undesirable. This uneven distribution of flow approaching the pumps can cause swirl and vortex formation at the pump intake.

Test scenarios

Tests were undertaken under steady conditions with each duty pump operating at the specified design flow rate and water levels provided by client. Using Froude Number similarity, the model flow capacity for each major pump (P1

& P2) is 7.9 l/s and the capacity for minor pump (P3 & P4) is 4.74 l/s. The test scenarios (please referred Table 2) in this study are as follows:

- A single pump, the P4 (minor pump) in operation at MSL +0.40 which is equivalent to 260 mm from the bottom of the sump (minor pump),
- Two pumps, P4 (minor pump) and P1 (major pump) in operation at MSL +0.90 which is 310 mm from the bottom of the sump (minor and major pumps),
- Three pumps, P4 (minor pump), P1 (major pump) and P2 (major pump) in operation at MSL +1.50 which is 370 mm from the bottom of the sump (minor and major pumps), and
- All pumps in operation at MSL +1.80 which is equivalent to 400 mm from the bottom of the sump.

Table 2. The relationship between prototype and model values.

Test	Water Level MSL	Major Pump 1	Major Pump 2	Major Pump 3	Major Pump 4
Run 1	+1.80	✓	✓	✓	✓
Run 2	+1.50	✓	✓	-	✓
Run 3	+0.90	✓	-	-	✓
Run 4	+0.40 (Min level)	-	-	-	✓

Result and Discussion

Vortices

Ideally, the flow of water into a pump should be uniform, steady, without swirl and without air, either entrained from a free surface or released from local low pressure regions. Lack of uniformity can lead to reduction of efficiency.

Unsteady flow will result in fluctuating loading of the propeller, leading to noise and vibration. Swirl in the intake can cause a change in flow, reduction in the pump efficiency and power. It may result in vortices leading from the free surface or from a bounding solid surface into the pump. These vortices can become strong enough for the cores to be air filled or cavitation. Vortices from the water surface can draw air continuously into the pump; solid surface vortices, often called submerged vortices, provide discontinuities in the flow around the propeller blades.

The free vortices are generated by rotation and separation combined with the effects of accelerated flow at the pump intake. Coherent subsurface swirl is often apparent, sometimes penetrating into the intake but again, since only gravitational and inertial forces are involved Froude Number similarity accurately predicts the magnitude of such effects. In the extreme, air core vortices will be generated which may have serious consequences. Such vortices, with air core extending near to a model intake, will be subjected to scale effects. In a full-size installation, due to the absence of scale effects, the transition from a surface dimple and coherent subsurface swirl to an air core will occur more readily than in the model.

In order to achieve the objectives, it is necessary to investigate the performance of the pump sump as designed and to make modifications to overcome any encountered problems. The assessment of the pump sump involved observing the approach flow pattern towards each operating pump intake with the aid of dye tracer. The tests were undertaken under steady conditions with each duty pump operating at the specified design flow rate and water levels. When a single pump in operation it was conducted at

lower water level of 260 mm (from the bottom sump) and at 400 mm water level when all major and minor pumps are in operation. Observations were made from the dosing of the blue dye at the entrance and immediately downstream of the slope intake showed that the dye had a tendency to flow towards the centre of the suction column for all test scenarios as in Figures 4. Thus, this indicates that the proposed pump sump design work well during all pumps operation cases.

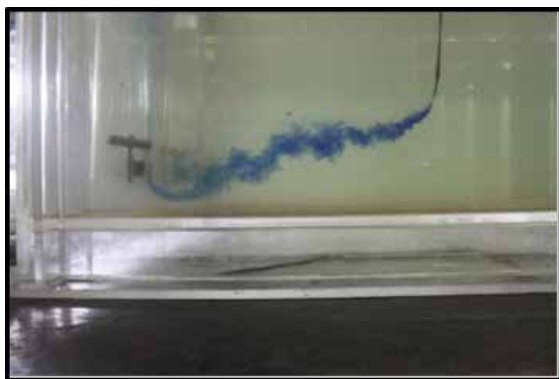


Figure 4. The approach flow pattern towards pump intake with the aid of dye tracer.

Pre-rotation and swirl at pump intake

Swirl is a general term for any flow condition where there is a tangential velocity component in addition to a usually predominating axial flow component. Pre-rotation is a specific term to denote a cross sectional average swirl in suction line of a pump or, in case of a vertical wet pit pump, upstream of the impeller. Pre-rotation will influence the pump performance since the flow approaching the impeller already has a rotational flow field which may oppose or add to the impeller rotation, depending on direction. The design of the pump impeller vanes i.e. shape and angle assumes no pre-rotation and the existence of pre-rotation implies flow separation along one side of the impeller vanes. The degree

that should be of concern is dependent on the type of pump and may not always be known. Pre-rotation should be quantified in a model by an average cross sectional swirl angle, determined by detailed velocity measurement or by reading on a swirl meter. Since swirl decays along a pipe as a result of wall friction, internal fluid shear and turbulence, the swirl meter in the model suction pipe should be located near the impeller. The method to determine the degree of vortices is using Vortimeter (please referred Figure 5) developed by Aden Research Laboratory. The Vortimeter was constructed and mounted in each of pump column intake suction for swirl angle measurement. The swirl angle was determined using the following relationship:

$$\tan \theta = \left(\frac{V_R}{V_A} \right) \quad [1]$$

Where:

V_R is the rotational velocity of the swirl meter vanes, and

V_A is the axial velocity in the pump intake.



Figure 5. Vortimeter used to detect swirl or rotational flow at the pump intake.

The counting swirl angles are indicated that the results are within the acceptable limit with the installation of the proposed anti-swirl device on

the bottom of intake column. As recommended by Ansar (1997), if the swirl angles are less than 5° , the model indicated that the incoming swirl is very minimum and insignificant. In sum, these results could be considered within the acceptable limit.

From Table 6 through 9, it was found 3 cases do not meet the requirement of the vortimeter angle. Pump 1 and pump 2 for the Case 1 (all pumps in operation), pump 2 for the Case 2 (3 pumps in operation) and finally for pump 4 in the Case 4. These mean that distributions velocity entered the inlet column with relative larger angle (more than 5°) that the limit suggested by Ansar (1997). There is possibility for the inflow angle leads to unbalanced pump impellers. Thus, it is recommended to install a triangle buffer block directly under the intake column.

Table 6. Pump configuration and swirl angle degree for case 1.

Test 1	PI	P2	P3	P4	Remark
swirl angle ($^\circ$)	4.8	0.0	0.9	0.4	cw
swirl angle ($^\circ$)	0	15.2	2.8	2.1	acw

Table 7. Pump configuration and swirl angle degree for case 2.

Test 1	PI	P2	P3	P4	Remark
swirl angle ($^\circ$)	4.0	5.3	-	3.0	cw
swirl angle ($^\circ$)	0	1.2	-	1.6	acw

Table 8. Pump configuration and swirl angle degree for case 3.

Test 1	PI	P2	P3	P4	Remark
swirl angle ($^\circ$)	0.6	-	-	0.1	cw
swirl angle ($^\circ$)	0.2	-	-	0.6	acw

Table 9. Pump configuration and swirl angle degree for case 4.

Test 1	PI	P2	P3	P4	Remark
swirl angle ($^\circ$)	-	-	-	9.1	cw
swirl angle ($^\circ$)	-	-	-	0.0	acw

Modification and improvement at pump intake

Since the initial proposed design indicated the existence of some degree of uneven flow distribution in the intake column sections, modification tests are required. The retests were conducted with a triangle buffer blocks directly under all of the pump intake column.

The detail modifications of a triangle buffer block directly under the pump intake column are shown Figure 6. The buffers divide the under current flow on the pump sump and direct them straight to the intake column. These buffers prevent from the occurrences of cross current directly under the intake column. Therefore, these could lead to a better flow distribution.



Figure 6. Modification of a triangle buffer block directly under intake column at all pumps.

The reading and the calculated angle values of vortimeter with a triangle buffer block are shown in Table 10 through 13. The modification minimizes the angle of flow and provide a better flow distribution straight to the pump intake column. These buffers also prevent under current to cross each other. Further, the 45° face of the buffer provide a smooth transition of incoming front flows.

Table 10. Swirl angle degree with a triangle buffer block for case 1.

Test 1	PI	P2	P3	P4	Remark
swirl angle (°)	0.9	0.4	0.3	0.0	cw
swirl angle (°)	0.0	0.6	3.3	1.9	acw

Table 11. Swirl angle degree with a triangle buffer block for case 2.

Test 1	PI	P2	P3	P4	Remark
swirl angle (°)	0.9	1.4	-	0.7	cw
swirl angle (°)	0	0.8	-	1.0	acw

Table 12. Swirl angle degree with a triangle buffer block for case 3.

Test 1	PI	P2	P3	P4	Remark
swirl angle (°)	0.6	-	-	0.3	cw
swirl angle (°)	0.0	-	-	0.2	acw

Table 13. . Swirl angle degree with a triangle buffer block for case 4.

Test 1	PI	P2	P3	P4	Remark
swirl angle (°)	-	-	-	3.0	cw
swirl angle (°)	-	-	-	0.0	acw

Conclusion

The results from the initial testing conclude that the proposed design for the Belibis River Pump Sump are not acceptable at three pump protocols. First when all 4 pumps in operations (Case 1 - 2 major and 2 minor). The occurrences of uneven flow distribution into the pump intake columns that indicated by the entrance degree of vortimeter greater than 5° . Second when 3 pumps in operations (Case 2 - 2 major and 1 minor). The occurrences of uneven flow distribution into the pump intake columns that indicated by the entrance degree of vortimeter greater than 5° . And the last one when only 1 pump in operation (Case 4 - 1 minor). The occurrence of uneven flow distribution into the pump intake column that indicated by the entrance degree of vortimeter greater than 5° . Thus, a minor modification is required by installation a triangle buffer block directly under all of the pump intake column as shown in Figure 6. During detail testing and analysis, the installation of the buffer shown a better flow distribution on all cases.

References

- Ansar, M. (1997). Experimental and Theoretical Studies of Pump Approach Flow Distributions at Water Intakes. PhD Thesis, University of Iowa, U.S.A.
- Archambeau, P., Epicum, S., Mouzelard, T. and Piroton, M. (2001). Experimental - numerical model interaction: example of a large dam project in Laos. Water Resources Management 2001, WIT Press, Southampton, Boston, 365-374 pp.
- Chanson, H. (1999). The Hydraulics of Open Channel Flow, Arnold, London, U.K.
- Epicum, S., Dewals, B., Archambeau, P., Detrembleur, S., Fraikin, C. and Piroton, M. (2008). Scale modelling and similarity laws for the study of an under pressure settling structure. 9th International Symposium on River Sedimentation, Tsinghua University Press, China.
- Piroton, M., Archambeau, P., Epicum, S. and Mouzelard, T. (2001). Water management of large dams. International Symposium on Environmental Hydraulics, Arizona State University, U.S.A.
- Piroton, M., Lejeune, A., Archambeau, P., Epicum, S. and Dewals, B. (2003). Numerical experimental interaction in hydrodynamics: an integrated approach for the optimal management of hydraulic structures and hydrographic basins. 9th International Conference on Enhancement and Promotion of Computational Methods in Engineering and Science, University of Macau, Hong Kong.

EXPERIMENTAL STUDY ON PROPELLER PERFORMANCE OF A VESSEL NEAR BANK AND IN SHALLOW WATER

Mehdi Nakisa¹, Fatemeh Behrouzi¹, Adi Maimun², * Mohamed Yasser Ahmed¹

ABSTRACT

Ship-bank hydrodynamic interaction effect in shallow water on ship manoeuvrability, steering performance and navigation in restricted waters has been a main challenge to safe navigation. The objectives of this research are firstly to investigate the effect of non-symmetric sloped bank and shallow water on hydrodynamic characteristics of propeller for a LNG ship, secondly to determine the flow wake pattern at propeller plane in different water depth and ship-bank distance. The numerical and experimental research methodologies were applied to achieve the objectives. The data contributes to the surface-ship resistance and propulsion model-scale database for computational fluid dynamics validation, as part of an international collaborative project between Malaysia International Shipping Corporation Berhad (MISC) and Hydrodynamic Research Group of MTC in UTM Institute (HRG-MTC-UTM). The tests were conducted in Marine technology Centre (MTC) – Universiti Teknologi Malaysia (UTM) to validate the computational fluid dynamic simulations using ANSYS-CFX 14. The resistance increased around 40 percent and thrust deduction factor decreased 25 percent due to separation and suction effect in stern region of ship above Froude number 0.08. Also, wake fraction increased 62 percent, the propeller thrust and torque increased 22 percent and 24 percent respectively. These results can be applied for as a valuable guideline for evaluating the manoeuvring and navigation of the vessel in restricted waters to consider the effect of restricted waters on propulsion performance and ship manoeuvring in channels.

Keywords: Towing Tank Tests, Propeller Performance, Numerical Modelling, Deep and Shallow Water conditions, Bank Effect.

¹ Member of Hydrodynamic Research Group, Marine Technology Centre (MTC), Universiti Teknologi Malaysia (UTM), Malaysia

²,* Head of Hydrodynamic Research Group, Faculty of Mechanical Engineering, Marine Technology Centre (MTC), Universiti Teknologi Malaysia (UTM), Malaysia
(Corresponding Author: adi@mail.fkm.utm.my)

Introduction

In restricted waterways, ship operation is affected by boundaries of the channels, ports and harbours such as the bottom, horizontal restrictions due to banks and quay walls. Lataire and Vantorre (2008) examined the different bank geometries to investigate the ship-bank interaction. They discovered that the ship manoeuvring behaviour is considerably affected by waterways restrictions, both in vertical and lateral dimensions, due to close proximity of the ship to the bottom and of the side walls, respectively.

Ship-bank interaction effect varies depending on bank shape, water depth, ship speed and ship-bank distance. In Lataire and Vantorre (2008), seven different bank geometries are installed in towing tank to simulate manoeuvres in low water depths in Flanders Hydraulics Research-Ghent University in a towing tank of 88m length x 7m breadth x 0.5m depth. Their fully automated towing tank can execute captive tests up to 35 tests per day.

Experimental Survey

Lataire and Vantorre (2008) showed that the relationship between ship's centre and half draft of bank ($TY/2 - Y_{ship}$) and the lateral hydrodynamic force which are shown in Figures 1.1 and 2.2 respectively.

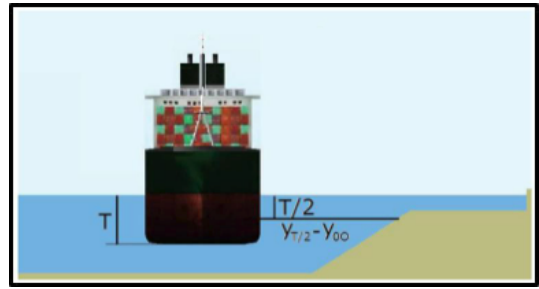


Figure 1.1: Ship bank distance in half draft (Lataire and Vantorre, 2008).

Lataire et al., 2007 determined the ship-bank interaction using surface piercing and with submerged horizontal platform to find the effect of bank distance on ship behaviour in case of sway force and yaw moment (Figure 1.3). In Norrbin (1974) and Dand (1981), flooded banks and surface piercing banks were studied. It was discovered that sway force and yaw moment were affected significantly in the case of a bank submerged under a low depth of water. Indeed, one of the advantages of a submerged bank is the test accuracy because the waves due to the bank geometry are damped and the reflexed waves take longer time to come in contact with the ship hull and propulsion system as compared to surface piercing.

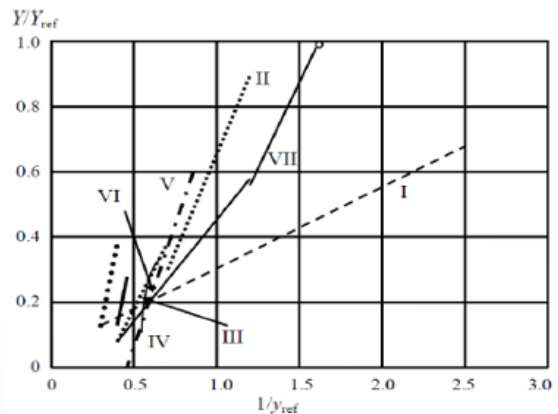


Figure 1.2: Relation between ship-bank distance and lateral force (Lataire and Vantorre, 2008).

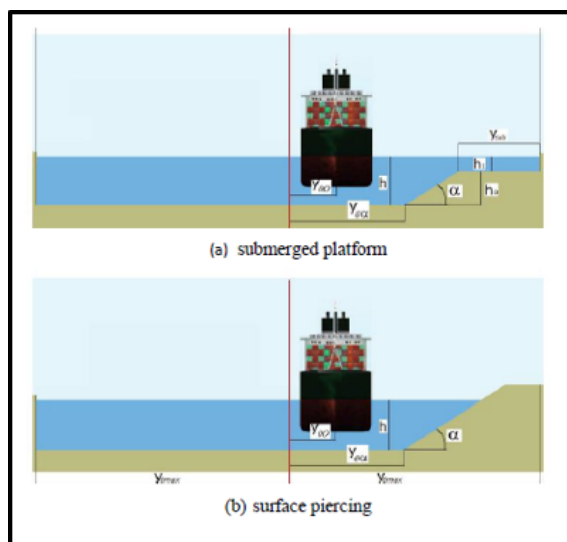


Figure 1.3: Characteristics of (a) submerged platform and (b) surface piercing bank (Lataire et al., 2007).

Zhou et al. (2013) investigated hydrodynamic interactions between ship and bank when ship is sailing along a bank in restricted waters. They applied computational fluid dynamics (CFD) to numerically simulate the flow around the ship appended with a rudder whilst navigating parallel to a bank.

Model scale experiments have been conducted by Duffy (2002) to study ship bank interaction in ship navigation. Parameters such as speed of ship, vessel draft, bank height, slope of bank, water depth and ship-bank distance were investigated. Empirical equation was developed from experimental data to estimate sway force and yaw moment produced in the case of surface piercing banks and submerged banks. The tests were performed at Australian Maritime College (AMC) towing tank facility.

On the other hand, Duffy (2002) could develop a parameter that superposition yaw moment and sway force produced by port and starboard

banks linearly. To produce empirical equations which can help to estimate the sway force and yaw moment, a number of regression analyses were carried out on the experimental results. Research of Duffy (2002) was extended by Maimun et al. (2009) to study the manoeuvring of Liquefied Natural Gas (LNG) ship in shallow water with ship-bank interaction effects.

Norrbinn (1971, 1974 and 1985) carried out extensive model tests and simulation on manoeuvring in general and effects of bank in particular (Figure 1.4). The model tests with a propelled tanker model ($L = 5.024$ m, $B = 0.852$ m, $T = 0.339$ m, $CB = 0.821$) were conducted to deduce lateral forces and yawing moments acting on a ship due to interaction with a vertical bank based on geometrical ship characteristics, water depth, ship-bank distance and propeller loading. The results showed that the propeller effect initially is amplified if the forward speed increased, up to a certain critical value. The application point of the additional force due to propeller action appeared to move forward with decreasing under keel clearance and increasing speed, and move aft with decreasing bank clearance.

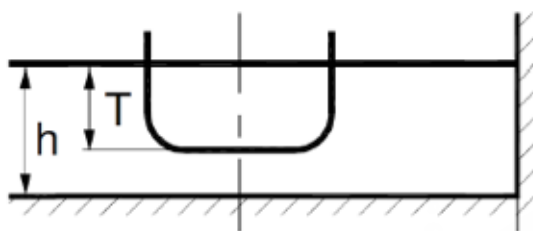


Figure 1.4: Vertical bank configuration (Norrbinn, 1971, 1974 and 1985).

Kobylnski, (2014) studied on ships travelling in inland waterways with restricted width

and depth and with different geometry of the channel's cross section (Figure 1.5). There are several hydrodynamic effects which exist in inland waterways. He also mentioned that the resistance of vessel, propeller's hydrodynamics parameters and as a result, propulsion system performance is changed due to the narrow fairways in which the vessel is navigated.

Cross sections of sloped bank with large and small under keel clearance are shown in Figures 1.5-A and 1.5-B respectively. Figure 1.5-C shows the canal with a step on the bank surface. The sloped channel with submerged horizontal platform is shown in Figure 1.5-D. There are many banks and waterway configurations with different slope angles with which the hydrodynamic interactions can be investigated (Kobylnski, 2014).

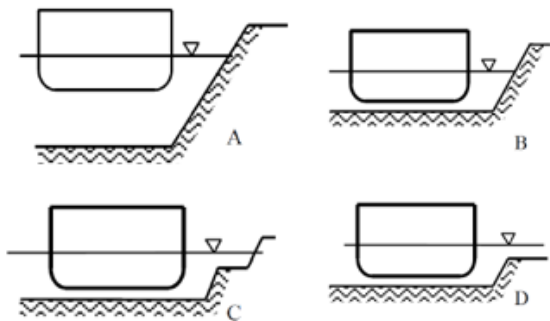


Figure 1.5: Different geometries of bank's cross section (Kobylnski, 2014).

Computational Fluid Dynamic Simulation Survey

CFD has indeed made significant progress in replacing expensive model tests and this is proven by the extensive use of viscous flow and potential flow CFD codes in design. Model tests are now only limited to the much later stages

in the project. In marine applications, CFD has also been used to simulate the flow around marine bodies such as hulls, propellers and ship appendages. In following parts, literature review is focused on evaluation of propeller working in open and confined water conditions. Figure 2.1 and Table 2.1 show the suitable computational domain and dimensions for simulations that they used. Hexa-structured meshes and hybrid-unstructured meshes were compared in the study and the turbulence model used in the commercial CFD solver was Shear Stress Transport (SST).

Morgut and Nobile (2012) found that for the propeller in open water propulsion characteristics, the hexa-structured and hybrid-unstructured meshes can guarantee similar levels of accuracy. Nevertheless, hybrid-unstructured meshes seemed to exhibit a more diffusive character than hexa-structured meshes, and also they were suited for detailed investigations of the flow field (Morgut and Nobile, 2012; Zhang, 2010; Zadavec et al., 2007 and Li, 2006).

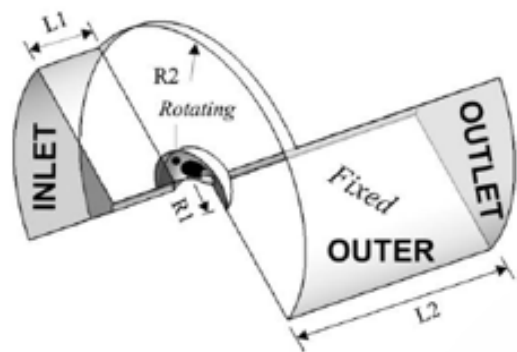


Figure 2.1: Computational domain of propeller model (Morgut and Nobile, 2012).

Table 2.1: Dimension of computational domain (Morgut and Nobile, 2012).

Values	P5168		E779A	
	Rotating	Fixed	Rotating	Fixed
R1	0.72D		0.70D	
Lmid	0.75D		0.76D	
R2		5D		5D
L1		3D		3D
L2		5D		5D

For the case of the flow around a marine propeller and prediction of the hydrodynamic characteristics, the numerical predictions using solvers are based on the Lifting-Surface-Theory that have been first carried out by Kerwin and Lee, (1978) and Streckwall, H. (1986). The viscous RANS (Reynolds Averaged Navier Stokes) approach, on the other hand, was applied later by Kim and Stern, (1990).

Seo et al, (2010) applied no-slip condition on the hull surface and free slip condition on top and bottom boundaries. Total pressure including hydrostatic one was applied on the upstream and side boundaries. Through the downstream boundary, the static pressure was extrapolated, as for resistance test. The sliding mesh technique was adopted around the propeller to apply non-conformal interface between the propeller and hull region as shown in Figure 2.2.

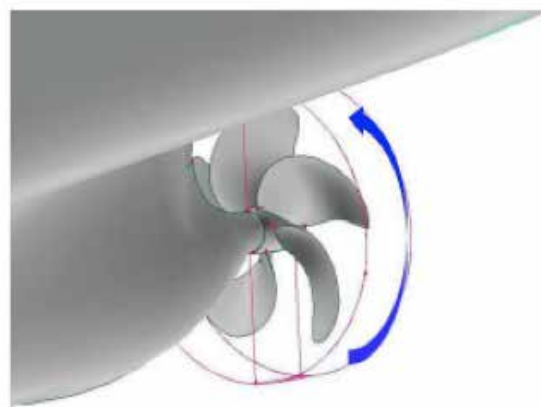


Figure 2.2: Configuration of sliding mesh (Seo et al. 2010).

Ship Near Bank and in Shallow Water

Figure 3.1 shows the scheme of ship position in shallow water and near bank. The conditions of ship in shallow water and near the bank are listed in Table 3.1 to show the strategy of research methodology which is conducted the resistance and propulsion tests numerically and experimentally in different ship speeds and propeller revolutions.

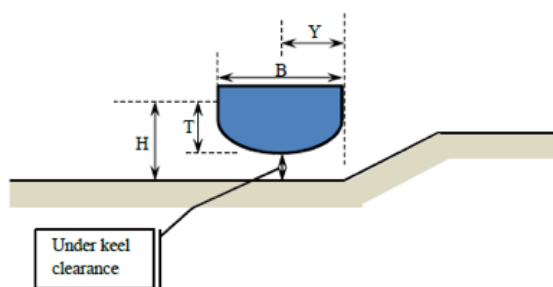


Figure 2.1: Scheme of ship position near bank and in shallow water.

Table 3.1: Ship conditions near bank and in shallow water.

H/T Y/B	1.2	1.4	1.6	Deep water
0.5	✓	✓	✓	---
0.8	✓	✓	✓	---
Without bank	✓	✓	✓	✓

Computational Fluid Dynamic simulation

Nowadays, CFD methods have been developed in completed units in one set of code. Ansys 14.0 has been utilised to simulate the ship bank interaction effect on propeller performance and flow around the ship's hull in shallow water. There are three main steps involved for CFD simulations which are Pre-processing, Solver and Post-processing. The shear stress transport (SST) turbulence model had been used in this study, because it gave the best results in comparison with other turbulence models. The equations are shown as follows (Eq. 1 and 2):

Equation of κ :

$$\frac{\partial}{\partial t}(\rho\kappa) + \frac{\partial}{\partial x_i}(\rho\kappa u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_\kappa \frac{\partial \kappa}{\partial x_j}\right) + G_\kappa - Y_\kappa + S_\kappa \quad (1)$$

Equation of ω :

$$\frac{\partial}{\partial t}(\rho\omega) + \frac{\partial}{\partial x_i}(\rho\omega u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right) + G_\omega - Y_\omega + D_\omega + S_\omega \quad (2)$$

Where G_κ and G_ω express the generation of turbulence kinetic energy due to mean velocity gradients and ω . Γ_κ and Γ_ω express the active diffusivity of κ and ω . Y_κ and Y_ω represent the dissipation of κ and ω due to turbulence. D_ω expresses the cross-diffusion term, S_κ

and S_ω are user-defined source terms (Sticker, et al. 1994 and Choi, et al. 2010).

Figures 3.2 and 3.3 show the mesh elements of two computational domains. The main domain includes flexible free surface (between air and water) with ship model and propeller domain respectively:

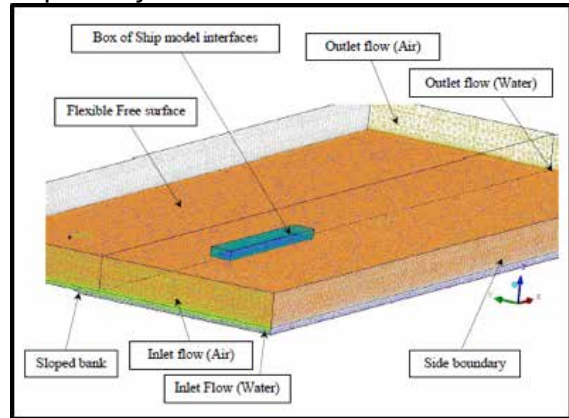


Figure 3.1: Mesh elements of Stationary computational domain

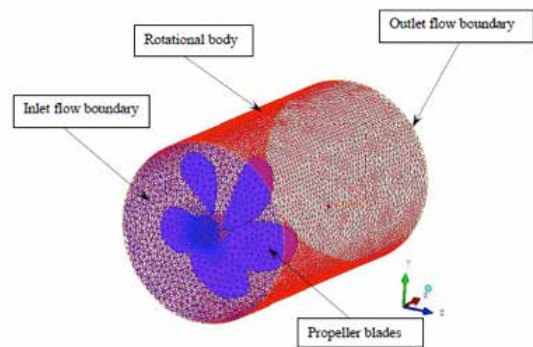


Figure 3.2: Mesh elements of the rotational domain

Model Test and Validation

Hydrodynamic propeller performance and wake pattern at propeller plane of the LNG model were experimentally investigated in open water, near bank and in shallow water conditions to validate

the numerical simulation. The experimental works were performed in Marine Technology Centre (MTC) - Universiti Teknologi Malaysia (UTM) which is shown in Figures 4.1 and 4.2.



Figure 4.1: The LNG model attached to the carriage in towing tank of UTM-MTC.



Figure 4.2: LNG carrier model test in near bank and shallow water conditions.

Results and Discussion

The propeller open water test was conducted in towing tank of Universiti Teknologi Malaysia (UTM) - Marine Technology Centre (MTC). In according to the numerical and experimental results it can be seen that the results of both methods have similar trend (Figure 5.1). The numerical results were verified by the experimental results from open water test in Universiti Teknologi Malaysia

(UTM) - Marine Technology Centre (MTC).

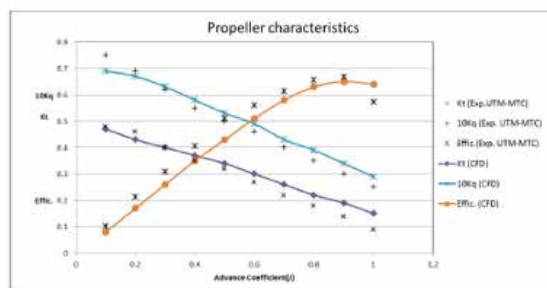


Figure 5.1. Propeller performance curves of open water test.

The self-propulsion test has been simulated to determine the changing of the propeller thrust (T) due to the propeller load variation in open, shallow and limited water conditions numerically. The thrust of the propeller behind the LNG ship model in open, shallow is shown in Figures 5.1 and 5.2 respectively.

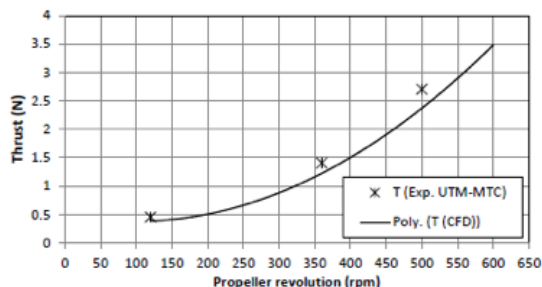


Figure 5.2: Self-propulsion test results to validate the CFD calculation results of hydrodynamic thrust (T) for the propeller behind the LNG ship model in open water condition with constant ship speed ($V_m = 0.35\text{ m/s}$, $V_s = 6\text{ knots}$) versus variable propeller revolution speed.

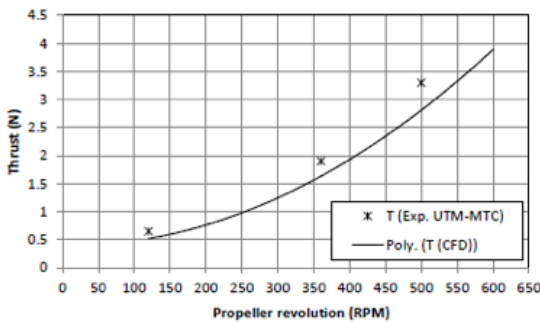


Figure 5.3: CFD calculation of hydrodynamic thrust (T) of the propeller behind the LNG ship model in shallow water condition ($H/T=1.2$) with constant ship speed ($V_m=0.35$ m/s, $V_s=6$ knots) versus propeller revolution speed.

Nominal wake pattern at propeller plane of the LNG ship model is investigated numerically in open water and restricted water conditions. Figures 5.1 show the half body velocity contours of axial velocity at the propeller plane in open water condition. The nominal wake pattern in deep and unrestricted water is symmetric due to the symmetrical incoming flow to the propeller disc.

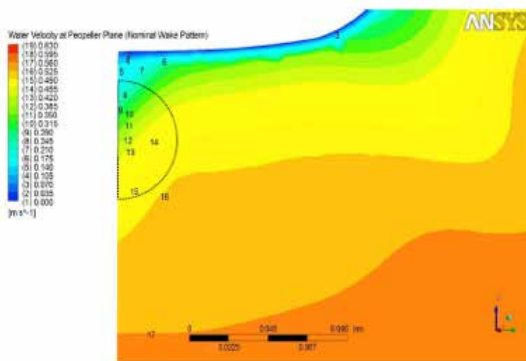


Figure 5.4: CFD calculation of nominal wake pattern in open water condition for the LNG ship model at $V_s=10$ knots.

Conclusion

This research has investigated the hydrodynamic propeller performance of LNG vessel near bank and in shallow water. Investigation conducted using a numerical simulation method. The conclusions are listed as follows:

- i. The water depth to draft ratio (H/T) and ship-bank distance (Y/B) effected significantly on propeller performance. For a certain propeller revolution (RPM) and ship speed, in the lower water depth and bank distance, the thrust and torque increased due to sea bead and wake fraction effects.
- ii. Ship resistance increased in lower water depth and ship-bank distance, because flow velocity increased under the ship bottom. This increment of resistance occurred due to turbulence flow under the ship bottom in shallow water depth and also the resistance increased dramatically near the bank.

Acknowledgment

This research was supported by Universiti Teknologi Malaysia (UTM) – Marine Technology Centre (MTC).

References

- Choi, J. E., Min, K. S., Kim, J. H., Lee, S. B., and Seo, H. W. (2010). Resistance and propulsion characteristics of various commercial ships based on CFD results. *Ocean engineering*. 37(7), 549-566.
- Dand, I. (1981). On ship-bank interaction. *Royal Institution of Naval Architects*. 25- 40.

- Duffy, J. T. (2002). Prediction of Bank Induced Sway Force and Yaw Moment for Ship Handling Simulator. Ship Hydrodynamics Centre, Australian Maritime College (AMC).
- Kerwin, J. E. and Lee, C. S. (1978). Prediction steady and unsteady marine propeller performance by numerical lifting surface theory. transaction of the society of naval architect and marine engineers.86.
- Kim, H. T. and stern, F. (1990). Viscous flow around a marine propeller-shaft configuration with infinite pitch rectangular blades. Journal of propulsion and power. 6(4), 434-444.
- Kobyliński, L. (2014). Bank effect and operation of inland waterways vessels. Scientific Journals Maritime University of Szczecin, Zeszyty Naukowe Akademia Morska w Szczecinie. 37(109), 50–55.
- Lataire, E., and Vantorre, M. (2008). Ship-bank interaction induced by irregular bank geometries. Symposium on Naval Hydrodynamics, 27th, Proceedings. 2, 511–524.
- Lataire, E., Vantorre, M., Laforce, E., Elout, K., and Delefortrie, G. Navigation in Confined Waters: Influence of Bank Characteristics On Ship-Bank Interaction. 2nd international conference on marine research and transportation. 28-30 June 2007, Ischia Naples Italy.2007.
- Li, D. Q., Leer-Andersen, M., Ottosson, P., and Tragardh, P. Experimental investigation of bank effects under extreme conditions. In Practical Design of Ships and Other Floating Structures. Proceedings of the Eighth International Symposium on Practical Design of Ships and Other Floating Structures. 1.2001.
- Maimun, A., Priyanto, A., Rahimuddin, S., Baidowi, A., and Nurcholis. Manoeuvring of a Vessel in Shallow Water with Ship-Bank Interaction Effect, Proceedings of International Conference in Ocean Engineering, ICOE 2009 IIT Madras, Chennai. India. 1-5 Feb.2009.
- Morgut, M., and Nobile, E. (2012). Influence of grid type and turbulence model on the numerical prediction of the flow around marine propellers working in uniform inflow. Ocean Engineering. 42, 26-34.
- Norrbin, N.H. Bank effects on a ship moving through a short dredged channel. Proceedings 10th Symposium on Naval Hydrodynamics, Cambridge, MA, USA. 24-28 Jun .1974.71-88.
- Norrbin, N.H. Bank effects on a ship moving through a short dredged channel. Proceedings 10th Symposium on Naval Hydrodynamics, Cambridge, MA, USA. 24-28 Jun .1974.71-88.
- Norrbin, N. H. Bank clearance and optimal section shape for ship canals, In: 26th PIANC international navigation congress, Brussels, Belgium. Section 1, Subject 1.1985. 167–178.
- Seo, J. H., Seol, D. M., Lee, J. H., and Rhee, S. H. (2010). Flexible CFD meshing strategy for prediction of ship resistance and propulsion performance. International Journal of Naval Architecture and Ocean Engineering. 2(3), 139-145.
- Stanier, M. The application of 'RANS' code to investigate propeller scale effects. In: 22nd Symposium on Naval Hydrodynamics. Washington, DC, USA.1999.
- Streckwall, H. (1986). A method to predict the extent of cavitation on marine propellers

by lifting-surface-theory. International Symposium on Cavitation. Sendai, Japan.

Zadravec, M., Basic, S., and Hribersek, M. (2007). The influence of rotating domain size in a rotating frame of reference approach for simulation of rotating impeller in a mixing vessel. *Journal of Engineering Science and Technology*. 2(2), 126-138.

Zhang, Z. R. (2010). Verification and validation for RANS simulation of KCS container ship without/with propeller. *Journal of Hydrodynamics, Ser. B*. 22(5), 932-939.

Zou, L., and Larsson, L. (2013). Computational fluid dynamics (CFD) prediction of bank effects including verification and validation. *Journal of Marine and Science Technology*. 18,310–323.

Zou, L., and Larsson, L. (2013). Computational fluid dynamics (CFD) prediction of bank effects including verification and validation. *Journal of Marine and Science Technology*. 18,310–323.



COST-EFFECTIVE, LAB-SCALE AUTOMATED SPATIAL AND TEMPORAL MEASUREMENT SYSTEM FOR LOCAL SCOUR AROUND BRIDGE PIERS

Mojtaba Porhemmat & Wan Hanna Melini Wan Mohtar

ABSTRACT

This article describes the process of developing a fully-automated, laboratory scale spatial and cost-effective for spatial and temporal measurement of local scour around bridge piers. Most of the studies in laboratory scale faced difficulties in obtaining a holistic local spatial variability around the bridge pier, particularly for continuous small time interval. Long experimental period, which can be up to few days, does not permit a consistent time interval spatial scour measurement, particularly due to physical constraints in the lab. Furthermore, readily available equipment or instrument is either expensive or not suitable for application in the laboratory. This study proposed a self-developed, locally sourced transverse-camera system capable to obtain the changes in both spatial and temporal of local scour. The system allows measurement using web camera at an adjustable distance (± 0.1 mm) and angle ($\pm 0.1^\circ$) from the origin position, which is programmed and controlled using Arduino. The application of Arduino allows a fully automated instruction to the transverse and camera for image capturing in four sequential steps. The coded programme controls the camera position and subsequently ordered the camera to capture the images at specified vertical and radial positions. The images were sequentially saved associated to the spatial interval, allowing a complete radial scour profile. The system was tested to monitor and measure the temporal and spatial local scour for long hours of experiments. The results showed that the location of the maximum scour depth can vary in different interval time from downstream to the upstream. The system was able to provide a holistic view of both spatial and temporal variability in the development of local scour at laboratory scale.

Keywords: spatial and temporal local scour, cost-effective, Arduino

¹ PhD Student, Department of Civil & Structural Engineering, Faculty of Engineering & Built Environment, Universiti Kebangsaan Malaysia.

² Senior Lecturer, Department of Civil & Structural Engineering, Faculty of Engineering & Built Environment, Universiti Kebangsaan Malaysia

Introduction

The studies on investigating the profile of local scour around bridge piers have been extensively conducted since 1950s (Laursen & Toch, 1956). The scouring process, if exceeds the permissible maximum scour depth may resulted in catastrophic bridge failure events which lead to major financial and infrastructural losses (Diab, 2011). As such, researchers examined the evolution of local scour profile for various shapes and sizes of piers, flow profile and sediment characteristics. The assessment of scour pattern gives an overview of suitable countermeasures and scour protection for specific type of pier and its location.

The temporal progression in scour depth around the piers have been extensively investigated (Diab, 2011). Long experiments (which may up to a week) were conducted to study the maximum scour depth obtained for specific type of pier, flow and sediment attributes. Majority of the studies obtained the maximum scour depth based on one-point measurement, mostly obtained in a water-free environment (that is the water was drained prior the determination of scour dimension). This procedure is rather conventional as the scour depth obtained only represents the profile after the determined equilibrium time. Furthermore, detailed temporal measurement requires laborious experimental work (using the conventional method) and constant human interaction throughout the experimental period.

Measuring the scour profile both in spatial and temporal without disruption is the way forward to fully understand the behaviour of scour. As such, several methods have been developed to obtain such accurate measurement using

camera, acoustic approach and laser (Debnath & Chaudhuri, 2010b; Diab, 2011). These methods have been proved to give favourable outcome and we strived to have similar measurement type for our work. Therefore, we explored possibilities from these three methods as our measurement technique for laboratory scale experimental work.

Measuring the scour profile using acoustic technique is commonly used at the offshore oil rig piers as monitoring procedure for its local scour. Acoustic measurement permits a plane profile which the bed morphology can be directly obtained and analysed. This method was an obvious first option due to its practicality and accuracy, however, searching for lab-scale readily available acoustic instrument (for this purposes) was time consuming and the most suitable apparatus found was beyond our existing financial budget.

Next, we tried our luck with the laser method. The technique of laser is attractive as it is non-intrusive and fast. Most of the laser equipment (for scour measurement) were developed to be used at field with large vertical measurement (up to 30 m) and is not suitable for lab-scale work. More importantly, the minimum distance from the transducer to the measurement area is 1 m, which is not feasible in a flume experimental setup. Although laser is a preferred choice, the dual medium of air-water proved to be quite challenging as the wavenumber of laser must be able to measure accurately in dual medium. A preliminary test was conducted to see the feasibility of one laser short distance instrument. The instrument was able to determine the distance accurately in air (as it was supposed to), but due to rather big fluctuation measurement

obtained in dual medium made us abandoned this option altogether.

After two attempts, we ventured out to measuring scour depth using image (captured by camera). This method is also non-intrusive albeit more efforts are required to pre-process the images and analysis. A rather convincing preliminary assessment for both spatial and temporal measurement using this technique managed to persuade us to go further. A detailed description on how the scour measurement was conducted will be further discussed in the next section.

Methodology

Developing a fully automated transversal system To obtain a spatial measurement, the camera is needed to capture images in the 360 angle of the pier. In this study, a low-cost, wide angled web camera with dimension of 0.06 m was employed. Note that the image capturing was feasible as the camera was inserted in a transparent Perspex made pier. The camera is mounted on a self-developed transverse system, controlled by Arduino with accuracy of 1 mm. Arduino is an open source micro-controller which can be purchased at reasonably low price whilst the programming software is freely available.

This user-friendly device (i.e. Arduino MEGA), shown here in Figure 1, was used in this study to control the transverse system, which was designed to move in vertical and rotational movement. The developed code gives instruction to the system to place the camera in the exact determined position (± 1 cm) and angle ($\pm 1^\circ$) and permit the image capturing in a complete plane dimension.

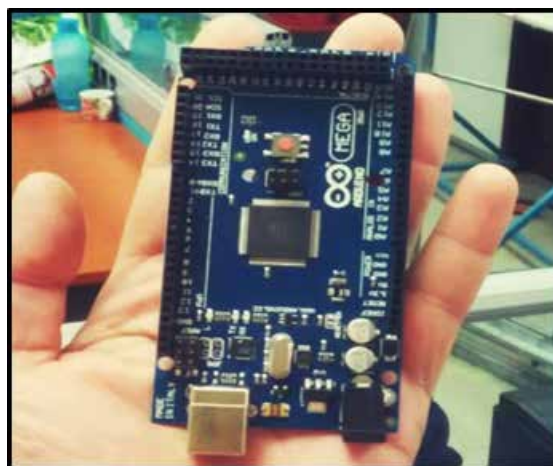


Figure 1: Arduino MEGA micro controller

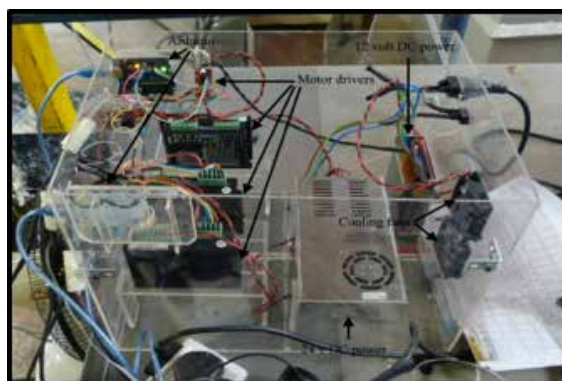


Figure 2: Central Control Unit.

The vertical and radial movement was able with stepper motor for each. The motors are connected to the stepper motor drivers, which were housed in the Central Control Unit as shown in Figure 2. Besides the motor drivers, this unit comprises of cooling fan, Arduino microcontroller and processor and DC power.

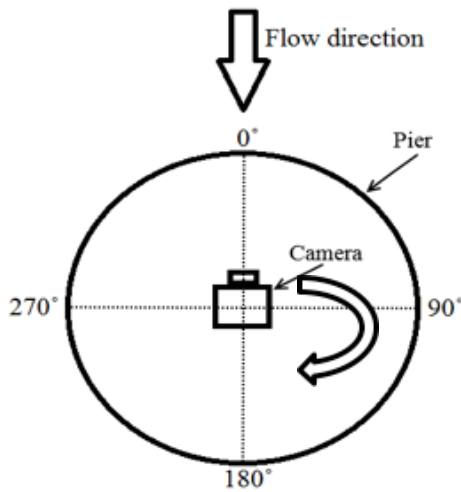


Figure 3: Schematic diagram of the position of the camera inside the pier. The arrow shown indicate the clock-wise direction of camera during image capturing.

To illustrate the radially camera movement, Figure 3 shows the schematic position of the camera inside the pier. The upstream nose of the pier (in the same direction as the flow) is always set at $\alpha = 0^\circ$, where α is the radial position around the pier.

The transverse-camera system is operated based on four-step sequences, here shown as flow diagram in Figure 4. The first step requires the coded programme to be uploaded into a computer. Then, it gives instruction to the Arduino which subsequently orders the motor drivers to go at the specific locations, bringing along the camera. After the camera has been set at the specified position, the Arduino sends affirmative feedback to the computer through the serial port, denoted here as Step 2. Next in Step 3, the camera is instructed to capture image at that position. Following Step 3, the camera sends the feedback of image captured, which is automatically saved and recorded in the accordingly specified folder. This process is

organised, saves time in pre-processing (as the files were specifically named according to its position) and assists in further analysis.

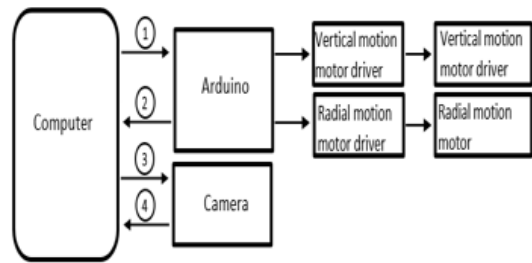


Figure 4: Flow diagram of the system. The arrow pointing to the right ($\rightarrow$) denotes instruction while the arrow pointing to the left ($\leftarrow$) denotes feedback. The number inside the circles indicate the steps in the system.

Experimental setup

The efficacy of the developed transverse and camera system was tested in a tilt-able flume with dimension of 1 x 1 x 8 m³ at the Coastal and Water Resources Laboratory in Universiti Kebangsaan Malaysia. The system was installed at the working area, located about 5 m from the inlet of flume. Figure 5 displays the pictorial placement of the system in the flume, including the position of camera in the pier, spot light to enhance the lighting (for better image) and where the vertical transverse system was fixed at.



Figure 5: The placement of the transverse-camera system in a laboratory flume.

Preliminary experiments were conducted in a 0.35 m deep and 2 m long sediment recess of artificial fine sand with $d_{50}=152.7 \mu\text{m}$, $\sigma_g = 1.73$ (i.e. the standard deviation calculated as d_{84}/d_{16}). Note that d_{84} and d_{16} denote the percentage of sediment at 84% and 16%, respectively.

The camera was placed in a 0.1 m diameter of Perspex made cylindrical pier and was ensured a free rotational movement without interaction to the pier. The flow was controlled by a submersible pump with capacity up to 0.14 m³s⁻¹. In these experiments, the velocity and the water depth was fixed at 21 cm/s and 30 cm, respectively giving the flow velocity is about 5% under the calculated critical velocity for the sediment used. The velocity was consistently measured to ensure its constancy using an ADV velocimeter, which was installed on a (different) traverse, placed at the upstream of the pier.

The experiments were left to run continuously for about 80 hours and was terminated after no scour was observed for at least five consecutive hours, following the suggestion given by (Debnath & Chaudhuri, 2010a). Images for each

specified vertical location and radial position were stored for further analysis.

Results and Discussion

Although panoramic photos were available for the 80 hours long of experiment, here only shown the data for the first five hours. Figure 6 shows the panoramic photo made from the photographs captured at varying time intervals, namely $t = 1, 2, 3, 4$ and 5 hours, and at varying angles of between $\alpha = 0^\circ$ to $\alpha = 360^\circ$. Data shows that the scour profile is symmetrical along its mid-plane, which is in-line with the other findings.

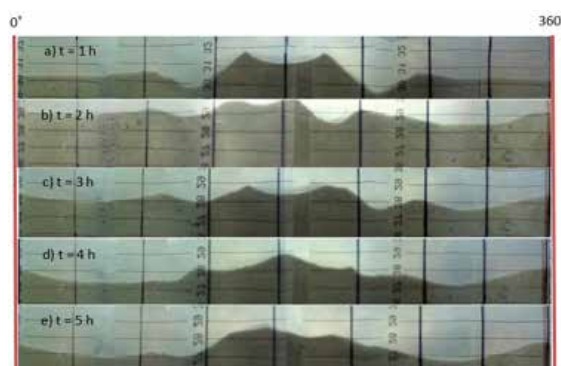


Figure 6: Panoramic photos of scour at different angles between 0° to 360° . a) after 1 hour, b) after 2 hours, c) after 3 hours, d) after 4 hours, e) after 5 hours.

The captured panoramic photos permit a detailed analysis on both spatial and temporal local scour evolution. The scour depth was obtained using image processing for finer scale and allowed more comprehensive understanding on how the scour was developed.

The maximum and minimum scours and the specific location can be inferred from this figure. The maximum scour is almost 5.4 cm and is located at the angle of $\alpha = 235^\circ$, whereas the minimum scour is almost 3.2 and is found at the angle of $\alpha = 152^\circ$. An analysis of the scaled

photographs shown in Figure 6a to 6e using the image processing technique produced a radial profile of the scouring at different intervals. The calculated depth from Figure 6 is transferred into a plot for better representation of the scour profile. Figure 7 presents the data derived from panoramic photos in Figure 6 using the Image software for a step of 15° angle and a time interval of 1 to 5 hours. Here $\alpha/\alpha_{\max} = 0$ to 0.25 and $\alpha/\alpha_{\max} = 0.75$ to 1 indicate the locations at the upstream of the pier while the $\alpha/\alpha_{\max} = 0.25$ to 0.75 denotes the downstream of the pier.

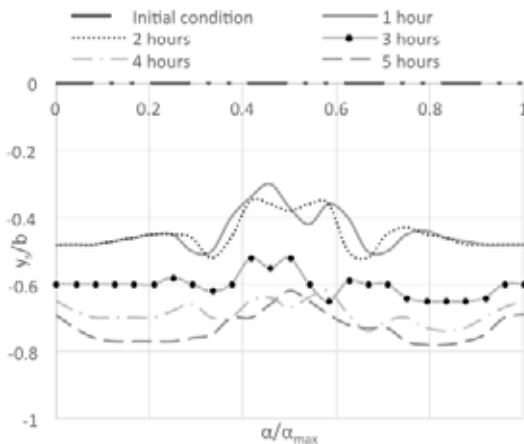


Figure 7: Local scour radial profile at different angles for 1-5-hour time interval, where y_s is the local scour depth, b is bridge width, α is the angle and $\alpha_{\max}$ is the maximum angle, namely 360° .

The curves, shown for different time (where the different symbols for specific time were illustrated) depicted the dynamic changes of the position of maximum scour depth. Initially, the maximum scour depth was observed close to the upstream at $\alpha/\alpha_{\max} = 0.2$ (at $t = 1$ hour) before progressively moved towards the downstream $\alpha/\alpha_{\max} = 0.375$ just an hour later and was remained constant until at $t = 5$ hr.

Conclusion and Recommendations

The self-developed transverse-camera system for laboratory scale local scour measurement was successfully and cost-effectively made. The system allows a full profile measurement for both spatial and temporal of local scour around bridge piers. Although at the moment, it only manages to capture the profile next to the pier, the evolution of maximum scour depth at finer time scale was able to be determined nonetheless. Dynamic changes of the position of maximum scour depth was observed.

References

- Debnath, K., & Chaudhuri, S. (2010a). Bridge Pier Scour in Clay-Sand Mixed Sediments at Near-Threshold Velocity for Sand. *Journal of Hydraulic Engineering*, 136(9), 597. doi:10.1061/(ASCE)HY.1943-7900.0000221
- Debnath, K., & Chaudhuri, S. (2010b). Laboratory experiments on local scour around cylinder for clay and clay-sand mixed beds. *Engineering Geology*, 111(1-4), 51–61. doi:10.1016/j.enggeo.2009.12.003
- Debnath, Koustuv, and Susanta Chaudhuri. 2010. "Bridge Pier Scour in Clay-Sand Mixed Sediments at Near-Threshold Velocity for Sand." (September): 597–609.
- Diab, Reda Mahmoud Abd Elaal Ali. 2011. "Experimental Investigation on Scouring around Piers of Different Shape and Alignment in Gravel." PhD thesis.
- Diab, R. (2011). Experimental Investigation on Scouring around Piers of different Shape and Alignment in Gravel. Retrieved from <http://tuprints.ulb.tu-darmstadt.de/id/eprint/2561>

Laursen, E. M., & Toch, A. (1956). Scour Around Bridge And Abutments, (4), 64. Retrieved from http://www.iowadot.gov/operationsresearch/reports/reports_pdf/hr_and_tr/reports/hr30.pdf

Laursen, Emmett M., and Toch, Arthur. 1956. Scour Around Bridge Piers and Abutments. Bulletin No. 4, Iowa Highway Research Board, Iowa Institute of Hydraulic Research.



STUDY OF FLOW CHARACTERISTICS IN MENGKUBAU DAM SPILLWAY

Che Mohamad Amirus Shafiq Che Ismail, Mohd Remy Rozainy Mohd Arif Zainol,
Ismail Abustan, Mohd Zulkifli Abdullah & Nuridah Sabtu

ABSTRACT

This paper presents a study of flow over a unique spillway model in Mengkubau Dam, Brunei. The model was subject to hydraulic test at Hydraulic lab of Universiti Sains Malaysia with 1 to 20 spillway scale model was used. A unique curve after labyrinth on the entrance to the chute needs a deeper understanding to the behaviour of the flow such as water depth and instantaneous velocity. Several critical locations have been selected after a test for the study. The study of the model is based on several design period discharge which are for 100, 1000, 10000 years, and Probable Maximum Flood (PMF). The main focuses of the study area are in relation between the water discharges, water profile and velocity along the model.

Keywords: physical model; water profile; velocity

¹ Research Assistant, School of Civil Engineering, Universiti Sains Malaysia (USM)

^{2*} Associated Researcher, CEGEOTECH, Centre of Excellent Geopolimer and Green Technology, Universiti Malaysia Perlis (UNIMAP) *E-mail: ceremony@usm.my

³ Dr. Eng., Professor, School of Civil Engineering, Universiti Sains Malaysia

⁴ Dr. Eng., School of Mineral and Resources Engineering, Universiti Sains Malaysia (USM)

⁵ Dr. Eng., School of Civil Engineering, Universiti Sains Malaysia (USM)

Introduction

The existing Mengkubau Dam reservoir with the storage capacity of the FSL of 28.7 m seems could not meet for the upcoming demand. To upgrade the dam's reservoir capacity, the modeller (ANGKASA Sdn. Bhd.) designed the dam crest to be raised from 31.00m BSD to 34.7m BSD. This enables the FLS to be increased to 32.9m BSD from the existing FSL of 28.7 m BSD for the Brunei Muara area community. To limit the flood rising level, a special Labyrinth type spillway will be constructed and it will efficiently convey the excess flood water to the downstream through the existing chute (Kamanbedast and Gholizadeh, 2012).

Hydraulic modelling is creating a scaled down model of structures where all possible factors and cost are being considered. The model is then used by the researcher to study the natural behaviour of the water towards hydraulics structures as a basis to predict the possible occurrences (Ismail et al, 2015), (Salleh et al, 2011), (Ho et al, 2006) and (Eric et al, 2014). The researcher is able in a very short time to test and record various scenarios without risking the life or equipment. The data collected from these tests can be used to further determine the feasibility of a site and to ensure that the proper methods are designed for the project with extensive experimental works previously (USBR, 2006).

Details of the model

An undistorted model scale of 1:20 (Bhajantri et al, 2006) was constructed at the Hydraulic Laboratory of Civil Engineering School, USM. The model will include an inlet portion of the reservoir, the spillway approach channel, the Labyrinth weir, the chute, the stilling basin and

a portion of apron which is the downstream channel (McPherson et al, 2014) and (Amador et al, 2007). The scope of the study included the evaluating the water profile and velocity along the model. The model contains unique shape which is a curve slope and wide in length for the chute. As to control the flow after the curve slope, a small drain was designed to control and release surplus water along the left side of the model. An approximately wide rectangular stilling basin for energy dissipation and a curve with drain in the middle of the apron was designed for the model to withstand the flow in the downstream area.

For water pump, the model was supply with significant amounts which in total of 21 external pumps, Pentex CST300/3, were used in this simulation study in which each pump has a capacity of 1000 L/min (16.67 L/s) and head of 7 meters for maximum discharge.



Figure 1: Mengkubau Spillway Model

The length of the upstream approach channel and the downstream channel will be able to reproduce the flow conditions before and after passing through the spillway channel. The physical model is 11.594m long, 2.900m wide and 1.719m height. All parts of the model spillway where the water is flowing will be made of approved material complying with the roughness and turbulent flow scaling

requirements. The inlet corner of the water will be properly designed so that uniform flows without surface waves are achieved at the approach channel. The toe of the stilling basin will be constructed of material which will provide a conservative estimate of downstream scouring. The model is considered appropriate and this results in the following model: prototype relationship:

Length	=	1:20
Velocity	=	1: 4.47
Flow	=	1:3125

Since the physical hydraulic spillway model is modelled on the basis of Froude Number similarities and of 1/20th full size scale (S), the calculation of flow capacity of the requested hydraulic spillway model to actual designed flow can be derived. Thus, using Froude Number similarity, the maximum model flow capacity for each pump is 201 L/s.

Selected Case Studies

The studies are based on the discharge of the design period. The discharge is 46 L/s, 63 L/s, 82 L/s and 203 L/s which the design period 100,1000,10000 years and the Probability Maximum Flood (PMF) respectively as displayed on Table 1.

Table 1: Detail discharges requirement for hydraulic spillway model and prototype

Return Period	Q _{prototype} (m ³ /s)	Q _{model} (L/s)	Froude Number
PMF	360	201	0.18
10000	146	82	0.04
1000	133	63	0.09
100	82	46	0.06

Objectives

The main objective of the study is;

- to construct a physical model of the Mengkubau Dam spillway,
- to determine the water profile of the spillway under a range of design discharges, and
- to determine the velocity profile of the flow based input discharge to the model

Scope of Study

The following scope of are required in order

- Locate and determined the critical location along the model for the study.
- Measure the water depth along the model and develop the water profile for all cases.
- Measure the velocity along the model for all cases.

Measurement Points of Surface Water Depth and Velocity

The measurement points have been pre-determined during the initial test in which the critical surface waters could be noticed. The detailed locations of measurement points are shown in Figure 2 as marked based on lines (L) and points along the selected lines (Ismail et al, 2015). There are 26 measurement locations starting at the upstream of the Labyrinth weir until after the discharge channel. On each line, there are several measuring points, P1 to P10 at all the critical cross-sections or lines. For example, there are 10 measurement points on L1 and each point is represented by P1 to P10 starting from left to right. However, the number of measurement points was determined by the available space at the specific location. For example, L23 to L26 only have 5 measurement points each.

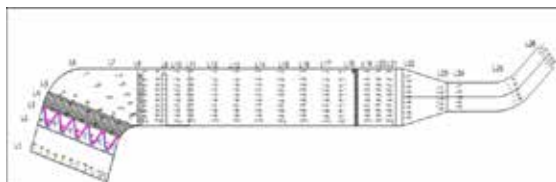


Figure 2: Selected Points for study

Observation Results

Throughout the hydraulic spillway model testing, measurements of water depth and velocities were recorded at pre-selected measurement points as discussed previous section.

General description of flow behaviour over the steps

Good design of a hydraulic structure can be displayed by its functionality after tested in the lab by several low and high discharges. In the Figure 3, the model is perfectly fine when subjected to a lower discharge of 46 L/s but seem barely to cater the fluctuated flow for the PMF discharge which is at 203 L/s.



Figure 3.1: Low discharge (46 L/s) flow at the downstream area



Figure 3.1: High discharge (203L/s) flow at the downstream area

Water Profile

Water level or water depth was measured using a digital point gauge for all 26 lines along the spillway; 11 measurement points at L1 to L3 representing upstream, on and downstream of the Labyrinth weir. In the slope area section, there were two lines (L4 and L5) and 5 points each to measure. For the flat area (before the chute), L6 has 6 measurement points while L7 and L8 have 5 measurement points each. In addition, L9, L10 and L11 consisted of 10 measurement points each. In the chute section, 8 measurement lines that were selected consist of 10 each measurement points. The study also designed detailed measurement points for stilling basin in measuring water depth and velocity. In this section, it is crucial to determine the water level and velocity difference for different cases tested. This is the reason for why only 5 lines; L22 to L26 in the apron and channel sections were selected for measurement as these sections are important for tail water observation.

The graphical water profiles for all discharge along the labyrinth spillway until the apron are shown in Figures 4. In the transition slope between the weir and the flat area (L4 to L6), the flows were very turbulent averaging between

105 to 151mm in height. In this section, accurate measurement is difficult to achieve since the water jumped from the weir. However, the water level remained relatively calm in the flat area with the average depth of 105mm to 135mm (L6 – L11). In the chute section (L12 to L19), the average depth is observed decreasing from 71mm to 2.5mm. This indicates that the water level changes with the distance of the chute. This is probably due to the changing of flow velocity from the upstream of the chute to the chute bottom.

For all model discharge, water flow depths were found catered in the sidewall of all model's section. However, in the case of 201 L/s, the water depth was found to surpass the model's sidewall at L24 (Chainage 272.337m). The stage-discharge for 201L/s results clearly show that the water levels in the stilling basin (L20 to L22) to be of similar maximum height with the most downstream part of the chute (L19).

The testing of the physical model shows that for the spillway discharge of 201 L/s, the flow water depth can be catered in all sections (labyrinth weir, slope area, flat area, chute and stilling basin) except for the downstream of apron channel (L24 and L25) as shown in in Figure 3. A rapid flow jet was observed when the water flow entered the apron section. The observed water depth in the apron reached the range average of 390mm to 540mm at L24. The average water depth upstream of the labyrinth weir was 160mm to 292mm (L1) and the average water depth on the top of the weir was only 39mm to 130mm (L3). On the flat area, the average water depths ranged from 178mm (L6 and 11) to 256mm (L7). The average water levels on the chute were decreasing from the upstream

of 156mm (L12) to 51mm (L19). Further, in the stilling basin, on average, the water level in the same region was between 49mm to 53mm. The water levels started to increase drastically like a water jet that reached 540mm and overtopped apron section when the water passed the settling basin. Only partial water flowed into the downstream of the apron, as the rest overtopped the model's sidewall.

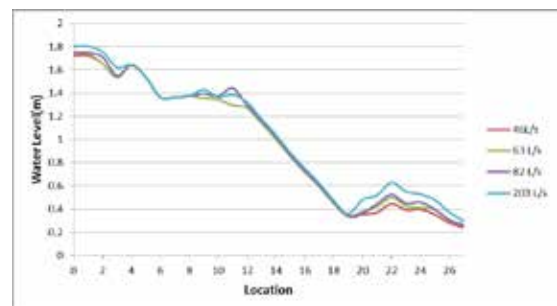


Figure 4: Water Profile for all Design Period for Mengkubau Spillway

Velocity

The detailed velocity profiles in each measurement points for model discharges of 46lps, 63L/s, 82 L/s and 203 L/s are displayed in Figure 5. The flow velocities remained calm in the upstream of labyrinth weir (L1) and started to move with velocity of 0.09 to 0.33 m/s at L2 and L3 respectively. In the slope section, which start from location 12 to 19 where the maximum is, the turbulent velocity first observed is due to rapid change of the level; the reduced flow velocity that follows in the region is due to flat areas (L7 to L11). The flow velocity was observed to speed up again when the water flowed in the chute with velocity starting at 1.12 to 1.82 m/s (L12) and then increased to 3.36 to 4.48 m/s at the bottom of the chute (L18). The flow velocity reduces when it entered stilling basin (L19); probably due to momentum mass of the water

flux leaving the chute. The water flow continued on with velocity 0.17 to and 0.73m/s at L23 and increases to the end of the model.

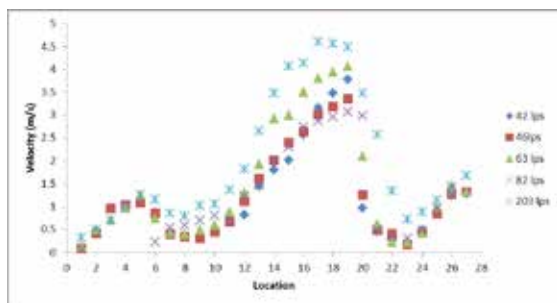


Figure 5: Velocity Profile

Conclusion

The results in the apron and natural channel (up to CH 326.837) indicate that in all cases, the flow depth is higher than both structures. For PMF, the most critical case is due to having highest velocity and flow depth as the discharge will cause erosion at the downstream area. Therefore, for PMF, more protection on the apron section and natural channel is required in order to prevent and reduce scouring effect due to relatively high velocity (1.97 – 7.51m/s for the prototype). The Mengkubau Dam Spillway model covered several aspects of the hydraulics behaviour that needed to be assessed as part of the design process.

References

- Amador, A., Sanchez-Juny J. and Dolz, 2007. Velocity and pressure field in skimming flow in stepped spillway, Hydraulic, Maritime and Environmental Eng. Dep., UPC, Barcelona, Spain.
- Bhajantri, M.R., Eldho, T.I. and Deolalikar P.B., 2006. Hydrodynamic modelling of flow over a spillway using a two-dimensional finite volume-based numerical model, *Sādhanā* Vol. 31, Part 6, pp. 743–754.
- C. M. A. S. Che Ismail, Mohd Remy Rozainy M.A.Z., I. Abustan, A.M. Mustafa Al Bakri, 2015. The Study of Mengkuang Dam's Chute Spillway by Physical Modelling.
- Eric, L., Blake, T. and Darryl, A. 2014. Stepped Spillway Model Pressures Characteristics, Susu Dam Malaysia.
- Ho, D.K.H., Cooper, B., and Donohoo, K.R., and S., 2006. 1. Application of Numerical Modeling to Spillways in Australia.
- Kamanbedast A., Gholizadeh, B., 2012 The Study of siphon spillway hydraulic by Modeling (Physical and Software), *Journal of Applied Sciences Research*, 8(5): 2548-2553
- McPherson, B.L., Lesleighter, E.J., Scriven D.C. and Bollaert, E.F.R. 2014. Physical and Computational Modelling System Analysis-Case Study for Paradise Dam, Queensland.
- Salleh, H. M., Khor, C. H., and Lai, K. F., 2011. Preparation of Interaction Plan for Expansion of Mengkuang Dam, Pulau Pinang
- USBR, Bureau of Reclamation, Design of Small Dams 2006. A Water Resources Technical Publication.

OPTIMUM POSITIONING OF BUFFLE BLOCKS IN STILLING BASIN FOR MENGKUBAU DAM

Ismail Abustan, Nuridah Sabtu, Mohd Remy Rozainy Mohd Arif Zainol,
Mohammad Khairy A. Wahab & Mohd Azmi Ismail

ABSTRACT

This paper attempts to study the effect of baffle block in terms of energy dissipation and hydraulic jump in stilling basin in a physical model of Mengkubau Dam spillway. Due to high velocity flow in the upstream may cause scouring, damaging and erosion on downstream area. The experimental procedures include the data collection using a velocity meter and flow meter with three flow discharges (63L/s for initial design, 82L/s for 10000 years return period and 201L/s for PMF or most critical flow discharge). The results from the initial design testing conclude that the proposed design for the Mengkubau Dam spillway are not acceptable when in operation without any modification in stilling basin. After the proposed modification in the stilling basin, the test results had shown a better improvement in term of energy dissipation and hydraulic jump. With these modification of rows baffle installed in the stilling basin, the high velocity for 201L/s case from upstream were reduced from 21.95m/s to 15.47m/s for 1st modification and reduced the depth of water in the stilling basin. Based on testing and analysis, the 1st modification is recommended to be implemented in the final design.

Keywords: stilling basin, baffle block, physical model

¹Dr. Eng., Professor, School of Civil Engineering, Universiti Sains Malaysia (Penang, 14300, Malaysia)

²Dr. Eng., School of Civil Engineering, Universiti Sains Malaysia (Penang, 14300, Malaysia)

³Associate Researcher, CEGeoGTech, University Malaysia Perlis (Perlis, 01000, Malaysia)

⁴Student, School of Civil Engineering, Universiti Sains Malaysia (Penang, 14300, Malaysia)

⁵Dr. Eng., School of Mechanical Engineering, Universiti Sains Malaysia (Penang, 14300, Malaysia)

Introduction

Energy dissipation and hydraulic jump are important in workability for stilling basin in order to prevent the scouring at downstream area and damaging the downstream structures (Alikhani et al., 2010) (Farhoudi et al., 2010) (Tiwari et al., 2015) (Farhoudi, 2009). Many researchers investigate the basin workability in order to develop new efficient in energy dissipation and also economical model for pipe outlet in comparison to USBR VI stilling basin model by inserting the plain splitter block along with sloping end sill and impact wall (Tiwari et al., 2011) (Tiwari et al., 2015).

The baffle block was introduced by Harleman (1955), and studied the role of baffle blocks in functioning of stilling basins and their effects on flow characteristics. Farhoudi and Narayanan (1991) studied experimentally the drag forces induced by hydraulic jump on baffle blocks in a stilling basin downstream of sluice gate. Firoto and Rinaldo (1992b), studied the features of hydraulic jump downstream of sluice gate, where the Froude number ranged between 5 and 9.5 (Farhoudi, 2010).

The shape, height, width and spacing between of the baffle block are important role in increases with the Froude number (Peterka, 1978). The baffle shows effectiveness to stabilize the jump formation, dissipate energy by impact action, and increase turbulence to assist in energy dissipation (FEMA, 2010). Position of baffle blocks would be best if as close as possible to the entrance of basin. However, the design engineer should be careful to put the baffle blocks far enough from the chute blocks to prevent the

interference of chute blocks and baffle blocks performances (Farhoudi, 2009).

Model Description

The model was constructed geometrically with scale ratio of 1:20 for this study, as shown in Figure 1 and the physical model is 11.594m long, 2.900m wide and 1.7.19m height.



Figure 1: The physical model of spillway model

Measurements and Instrumentation

The following measurements were required for the model:

Flow rates

The total model flow rate was measured using ThermoPolysonics Flow Meter, made in West Germany (DFD Flow Meter Hydra SX30) (Serial No. 72/3/34811C) to obtain the desired flow discharge in the physical hydraulic model (Polysonics, 2000). Figure 2 shows the Thermo Polysonics Flow Meter that will be used in this study and reading be carried out before, during and after running the physical model experiment for each study case. Twenty numbers of external pumps, Pentex CST300/3, were used in this simulation study in which each pump has a

capacity of 1000 L/min (16.67 L/s) and head of 7 meters.

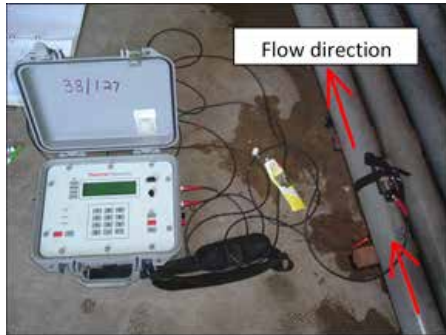


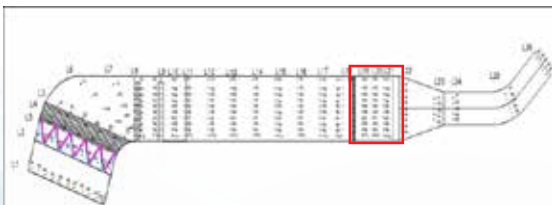
Figure 2: Thermo Polysonics Flow Meter

The model scale of 1/20 full size is considered appropriate. This will result in the following model/prototype relationships:

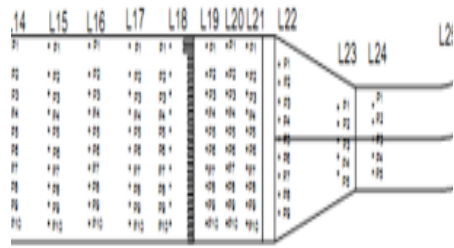
Length	= 1:20
Velocity	= 1:4.47
Flow	= 1:3125

Velocity Distribution

Velocities were measured with a miniature Nixon Streamflo Velocity Meter model 403 Standard low speed velocity probe (Serial No. 2319) for the range 5.0 to 150 cm/sec. (Streamflo, 2003). The velocity values were measured at seven different flow rate. The location of the measurement points in stilling basin are showed in Figure 3.



(a)



(b)

Figure 3: (a) Overall measurement points at spillway (b) measurement points at stilling basin

Calculation of Required Model Discharge

Since the pumping station sump structure is modelled on the basis of Froude Number similarity and of 1/20th full size scale (S), the calculation of flow capacity of the model in requested to actual designed flow can be derived as:

$$Q_m = \frac{Q_p}{S^{2.5}}$$

$$Q_m = \frac{360,000}{10^{2.5}}$$

$$Q_m = 201 \text{ L/s}$$

Thus, using Froude Number similarity, the model flow capacity for each pump is 201 L/s. Table 1 shows detail flow requirements provided by the Angkasa Consulting Services Sdn. Bhd.

Table 1: Detail discharges requirement for hydraulic spillway model and prototype

Return Period	$Q_{\text{prototype}}$ (m ³ /s)	Q_{model} (L/s)
PMF	360	201
10000	146	82
1000	133	63

The Test

The initial test was to determine the workability of the physical model and to discuss any modification(s) that is/are needed. There were 8 testing cases that had been simulated as shown in Table 2 and only tested on three discharge of 63L/s, 82L/s and 201L/s. The main different of the cases are the arrangement of buffer configurations as per suggestion by Angkasa Consultant Services Sdn. Bhd. and as per reference to the Small Dam Design by US Bureau Reclamation.

Table 2: Testing cases of the hydraulic spillway model

Test no.	Model Flow Rate (L/s)	Prototype Discharge (m ³ /s)	Buffer Configuration
1	63	133	initial design
2	201	360	initial design
3	63	133	single row buffers
4	82	146	single row buffers
5	201	360	single row buffers
6	63	133	double row buffers
7	82	146	double row buffers
8	201	360	double row buffers

Note:

Actual size and configuration of tender drawing

Actual size and configuration of tender drawing with single row of buffers

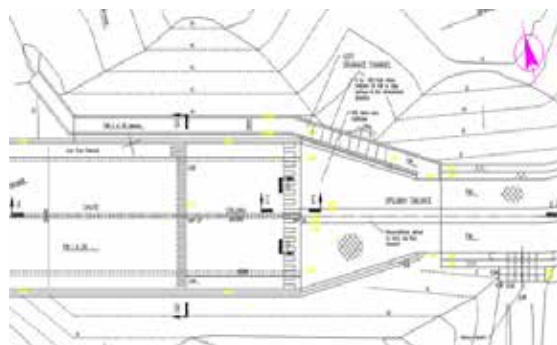
Actual size and configuration of tender drawing with double row of buffers

Test Program

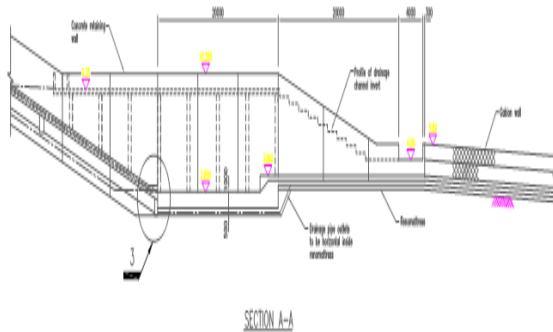
Test conducted to evaluate the performance and determine the most severe condition(s), it is necessary to investigate the performance of the stilling basin as designed and to make modifications to overcome any encountered problems.

Initial Design Testing

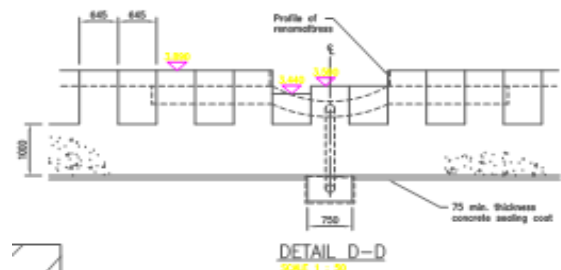
Water level or water depth was observed and measured using a digital point gauge at L6 (before the chute) has 6 measurement points while L7 and L8 have 5 measurement points each. In addition, L9, L10 and L11 consisted of 10 measurement points each. In the chute section, 8 measurement lines that were selected consist of 10 each measurement points. Only two discharge (63L/s and 201L/s) were tested per recommendation by Angkasa Consulting Services Sdn. Bhd. Figure 4 shows the initial design of stilling basin and dissipating hydraulic jump buffer.



(a)

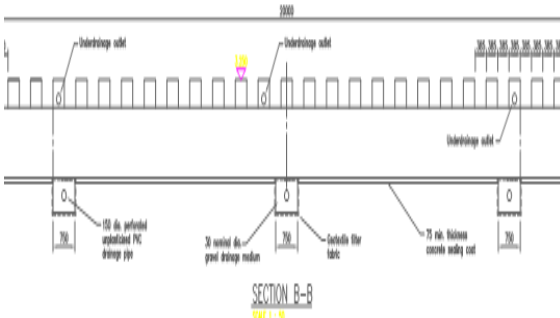


(b)

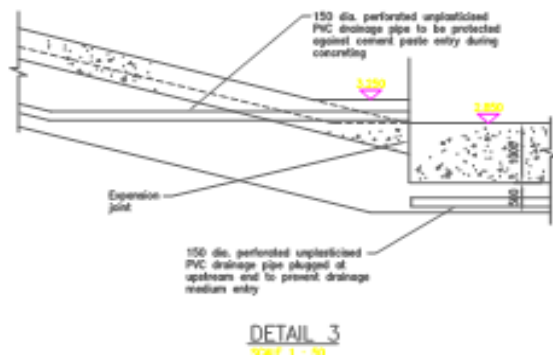


(e)

Figure 4: (a) plan view (b) side view (c), (d), (e) details of stilling basin (Source: Angka Consultant Services Sdn. Bhd.)



(c)



(d)

From the results on Table 3, shown that the highest water depth occurred in stilling basin at L22 P10 for the model discharge of 63L/s. In spite of that, the stilling basin wall height (485mm) could be catered this maximum water. The surface water levels were high in the upstream of labyrinth spillway with the average depth of 240mm from the bottom of the weir. The average measured overflow on the labyrinth weir is 43mm (L3). In the transition slope between the weir and the flat area (L4 and L5), the flows were very turbulent averaging between 105 to 151mm in height. In this section, accurate measurement is difficult to achieve since the water jumped from the weir. However, the water level remained relatively calm in the flat area with the average depth of 105mm to 135mm (L6 – L11). In the chute section (L12 to L19), the average depth is observed decreasing from 71mm to 2.5mm. This indicates that the water level changes with the distance of the chute. This is probably due to the changing of flow velocity from the upstream of the chute to the chute bottom

Table 3: Model water levels for 63.0 L/s (initial design)

Initial		Water Depth (cm)										average	
		1	2	3	4	5	6	7	8	9	10	11	
Labyrinth	1	25.0	25.1	25.2	25.2	25.1	24.6	24.5	24.4	24.2	24.2	24.2	24.7
	2	25.1	5.4	24.4	5.5	24.4	4.7	24.3	7.2	24.2	7.7	24.4	16.1
	3	5.2	4.2	4.2	4.5	4.1	4.5	4.4	4.2	4.2	4.0	3.9	4.3
sloped area	4	15.2	14.3	15.1	15.0	15.2	15.4	15.5					15.1
	5	10.4	10.3	11.0	10.6	10.2							10.5
flat area	6	15.0	14.1	14.8	12.5	12.1	12.2						13.5
	7	8.2	15.6	12.8	12.7	12.4							12.3
	8	8.2	13.9	14.2	12.7	12.6							12.3
	9	8.2	11.5	11.6	11.9	12.0	10.4	9.8	9.1	10.0	10.7		10.5
	10	12.9	11.2	11.3	11.6	11.7	10.4	9.8	9.1	10.0	10.7		10.9
	11	13.5	12.3	12.7	12.8	13.4	12.2	12.0	12.0	11.5	11.5		12.4
chute	12	8.0	6.6	6.3	6.6	6.5	6.8	8.7	7.8	6.9	6.9		7.1
	13	4.5	3.5	2.5	3.6	4.1	4.9	4.7	4.7	4.9	4.7		4.2
	14	4.1	3.4	3.1	3.3	4.1	3.6	3.6	4.3	4.7	4.8		3.9
	15	4.1	2.3	2.2	3.0	2.6	3.0	3.5	3.8	3.9	4.3		3.3
	16	2.5	2.2	2.3	2.5	2.6	2.6	3.0	3.8	3.8	4.1		2.9
	17	3.3	2.3	2.2	2.3	2.3	2.6	2.8	3.4	3.6	3.7		2.9
	18	3.5	1.6	1.6	1.6	1.6	2.6	2.9	3.2	3.4	3.6		2.6
	19	3.8	1.5	1.5	1.8	2.1	2.5	2.8	3.0	3.3	2.9		2.5
stilling basin	20	14.6	12.6	13.1	13.2	14.4	19.0	18.0	18.0	17.0	16.0		15.6
	21	18.9	19.1	18.8	21.1	21.2	20.0	22.0	23.0	22.0	21.0		20.7
	22	21.3	20.4	21.8	21.6	21.8	24.0	24.0	23.0	24.0	25.0		22.7
Apron	23	19.4	19.6	21.2	17.9	21.7	25.0	23.0	22.0	22.0	22.0		21.4
	24	18.5	18.6	21.6	20.0	19.5							19.6
	25	12.9	13.6	18.6	12.5	14.0							14.3
	26	8.1	9.2	11.8	12.0	13.0							10.8
	27	8.9	7.5	11.2	9.0	9.2							9.2

The result of the physical model in Table 4 for the spillway discharge of 201 L/s shows that, the flow water depth can be catered in all sections (labyrinth weir, slope area, flat area, chute and stilling basin) except for the downstream of apron channel (L24 and L25). A rapid flow jet was observed when the water flow entered the apron section. The observed water depth in the apron reached the average of 540mm at L24. The average water depth upstream of the labyrinth weir was 292mm (L1) and the average water depth on the top of the weir was only 130mm (L3). On the flat area, the average water depths ranged from 178mm (L6 and 11) to 256mm (L7). The average water levels on the chute were decreasing from the upstream of 156mm (L12) to 51mm (L19). Further, in the stilling basin, on average, the water level in the same region was between 49mm to 53mm. The water levels

started to increase drastically like a water jet that reached 540mm and overtopped apron section when the water passed the settling basin. Only partial water flowed into the downstream of the apron, as the rest overtopped the model's sidewall.

The graphical stage-discharge for 66L/s and 201L/s along the labyrinth spillway until the apron are shown in Figures 5 and 6. In the 66L/s model discharge, all water flow depths were found catered in the sidewall of all model's section. However, in the case of 201 L/s, the water depth was found to surpass the model's sidewall at L24 (Chainage 272.337m). The stage-discharge for 201L/s results clearly show that the water levels in the stilling basin (L20 to L22) to be of similar maximum height with the most downstream part of the chute (L19).

Table 3: Prototype water levels for 201 L/s (initial)

Initial		Water Depth (cm)										average	
		1	2	3	4	5	6	7	8	9	10		11
Labyrinth	1	29.2	29.2	29.2	29.2	29.2	29.2	29.2	29.2	29.2	29.2	29.2	29.2
	2	28.5	23.5	28.5	28.5	23.5	28.5	23.5	28.5	23.5	28.5	23.5	26.2
	3	8.0	16.0	7.5	18.0	7.5	18.0	7.5	18.0	7.5	18.0	17.5	13.0
Sloped area	4	19.2	19.4	19.8	19.5	18.9							19.4
	5	17.2	16.7	17.4	17.2	16.0							16.9
Flat area	6	31.0	29.0	27.0	21.0	11.0	10.0	6.0	5.5	20.0			17.8
	7	31.0	29.0	27.0	21.0	20.0							25.6
	8	29.0	25.0	23.0	22.0	20.0							23.8
	9	29.0	25.0	23.0	22.0	22.0	19.0	18.5	19.5	19.0	18.0		21.5
	10	23.0	19.0	20.0	21.0	20.0	20.0	19.0	19.5	19.0	19.0		20.0
	11	19.0	17.0	18.0	18.5	19.0	17.5	17.5	17.5	17.0	17.0		17.8
Chute	12	18.0	17.0	18.0	19.0	19.5	12.5	13.0	12.5	13.0	13.0		15.6
	13	13.0	12.0	12.0	12.0	12.0	9.0	9.0	9.5	9.5	10.0		10.8
	14	10.0	8.0	8.0	8.5	8.5	8.0	8.0	8.0	8.5	9.0		8.5
	15	7.1	6.2	6.4	6.4	6.8	6.5	7.0	7.0	7.5	7.5		6.8
	16	6.6	5.3	5.3	5.6	5.6	6.5	7.0	7.0	7.5	7.5		6.4
	17	6.3	4.7	5.2	5.7	5.7	6.0	7.0	6.5	7.0	6.5		6.1
	18	5.7	4.3	5.1	5.3	5.1	5.5	5.5	5.5	6.0	6.0		5.4
	19	5.6	4.6	4.7	4.5	4.9	5.5	5.0	5.5	5.5	5.5		5.1
Stilling basin	20	5.9	4.2	4.4	4.4	4.7	5.0	5.0	5.0	5.0	5.0		4.9
	21	6.2	4.8	4.8	4.8	4.7	5.0	5.0	5.0	5.0	5.0		5.0
	22	6.3	5.1	4.9	4.7	4.7	5.5	5.5	5.5	5.5	5.5		5.3
Apron	23	20.1	20.4	20.2	20.6	22.0	20.1	20.4	19.5	18.5			20.2
	24	53.5	54.2	54.8	54.2	53.4							54.0
	25	41.6	41.2	41.3	42.2	41.6							41.6
	26	24.0	20.0	16.0	16.4	18.0							18.9
	27	24.5	24.0	20.0	13.0	16.0							19.5

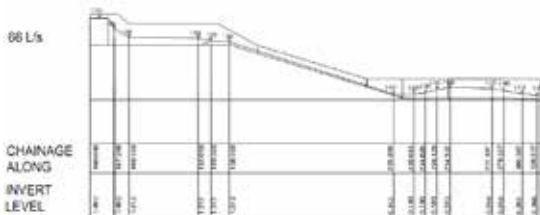


Figure 5: Stage - Discharge for 66 L/s (initial)



Figure 6: Stage - Discharge for 201 L/s (initial)

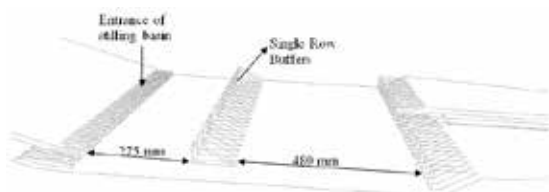
Modification Testing

The main objective of modification is to increase the energy dissipation by reducing the height of hydraulics jumps and minimizing the flow velocity in the settling basin. The installation of buffer blocks will do the necessary actions on this matter. Thus, the buffers provide a smooth transaction of flow from the high velocity at the end of spillway chute and to a manageable velocity and height of hydraulics jumps in order to sustain downstream vicinity from massive erosion and scouring when the labyrinth spillway weir is overtopped. The installation of buffer blocks will do the necessary actions on these matters in Figure 8 and 9. The row/s of buffers provide a smooth transaction of flow from the high velocity at the end of spillway chute and to a manageable velocity and height of hydraulics

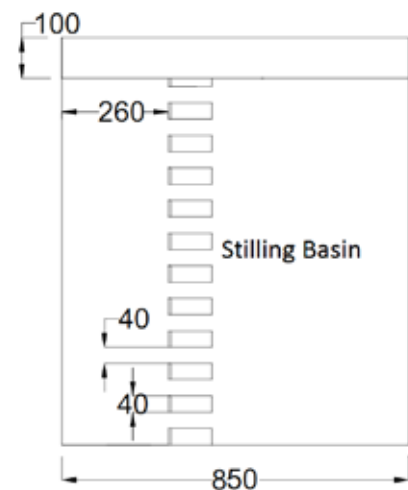
jumps in order to sustain downstream vicinity from massive erosion and scouring when the labyrinth spillway weir is overtopped.



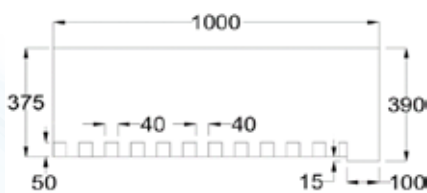
(a)



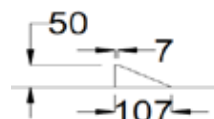
(b)



(c)

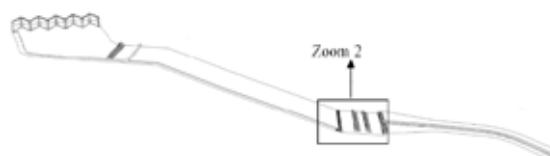


(d)

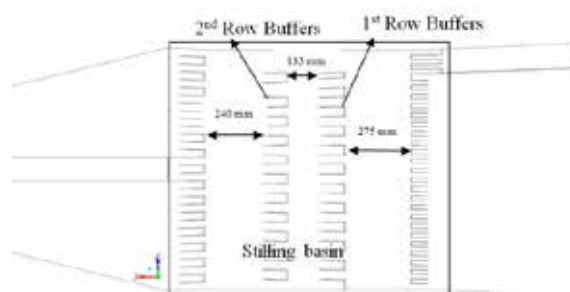


(e)

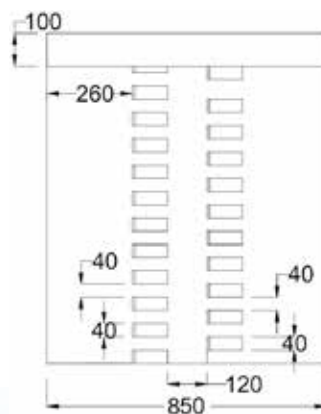
Figure 7: (a) overview of spillway (b) close-up of stilling basin (c) plan view details of stilling basin 1st modification (d) front view details of stilling basin 1st modification (e) details of baffles 1st modification



(a)



(b)



(c)

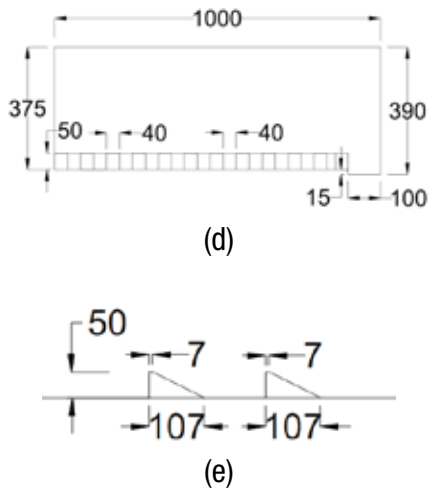


Figure 8: (a) overview of spillway (b) close-up of stilling basin (c) plan view details of stilling basin 2nd modification (d) front view details of stilling basin 2nd modification (e) details of baffles 2nd modification

The 1st and 2nd Modifications

Observations and measurements were made on the height and length of hydraulics jumps and the velocity at the end of stilling basin. During modification activities, it can be observed that there are some positive changes in terms of height and velocity in the stilling basin. Detailed results of reduction of hydraulics jumps height are displayed in Table 4. These results indicate that the proposed minor 1st modification has some degree in dissipating the hydraulics jump energy and reducing basin velocities as to 2nd modification. The maximum height of the settling basin wall is 380mm and the total length of the settling basin is 850mm. In Table 4, the height of hydraulic jumps in the 2nd modification (double rows of buffers) exceeded 51mm above the basin wall height (431mm) when the model discharge ran at 201L/s. The comparison of hydraulic jump height between 1st and 2nd modifications shows that the average heights of 1st modification are lower than 2nd modification

in all cases. In the down-stream of the stilling basin, the imaginary walls were placed at the height of 300mm in order to cater the flowing water from overtopped the existing channel (L24 to L26) of 100mm height.

From the results in Table 4, in term of velocity in the stilling basin, the initial design shows the highest magnitude since there are no buffers to block flow from the chute. For example, the average maximum velocity for the PMF reached 4.91m/s for the model. In other words, the prototype velocity is 21.95m/s (4.91×4.47).

Table 4: The range of average height and velocity of hydraulics jump in the settling basin for the Initial Design, 1st Modification and 2nd Modification

Discharge (L/s)	Buffer Configuration	Height (mm)	Velocity (m/h)
63	Initial Design	156 – 227	2.58 – 4.39
	1st Modification	165 – 203	0.36 – 2.52
	2nd Modification	205 – 260	0.34 – 2.97
82	Initial Design	-	-
	1st Modification	183 – 275	0.93 – 4.67
	2nd Modification	223 – 317	1.60 – 4.67
201	Initial Design	49 – 53	4.56 – 4.91
	1st Modification	230 – 370	1.16 – 3.46
	2nd Modification	227 – 431	0.95 – 3.74

Conclusion

The results from the initial testing conclude that the proposed design for the Mengkubau dam spillway are not acceptable when operation without any modification. After the proposed modification, the test results shown a better improvement in term of flow velocity and height of the hydraulics jumps in the stilling basin.

- a. The velocity and pressure distribution showed that the most critical case was at the PMF or maximum discharge of 201L/s or equivalent to 360m³/s in the prototype. From observation, the maximum designed discharge capacity of 360m³/s caused the highest velocity of 21.95m/s (4.91m/s x4.47) in the prototype for the initial designed. However, after installation of buffers, the velocity had been reduced to 15.47m/s and 16.72m/s for 1st and 2nd modifications respectively.
- b. The 1st modification with a row of buffers shows that it could cater the height of the hydraulics jumps in the stilling basin. The 1st modification reduces depth of water in the stilling basin and reduces overall velocity as well. In sum, the 1st modification works better than 2nd modification.
- c. The results in the apron and natural channel (up to CH 326.837) indicate that in all cases, the flow depth is higher than both structures. For PMF, the most critical case is due to having highest velocity and flow depth as the discharge will cause erosion at the downstream area. Therefore, for PMF, more protection on the apron section

and natural channel is required in order to prevent and reduce scouring effect due to relatively high velocity (1.97 – 7.51m/s for the prototype).

During detail testing and analysis, the modeler recommends the 1st modification to be implemented in the final design.

References

- Alikhani, A., Behrozi-Rad, R., & Fathi-Moghadam, M. (2010). Hydraulic jump in stilling basin with vertical end sill. *International Journal of Physical Sciences*, 5(1), 25-29.
- Farhoudi, J. (2009). Total Pressure around Chute Blocks of SAF Stilling Basins. *International Journal of Civil Engineering*, 7(4), 271-279.
- Farhoudi, J., Sadat-Helbar, S. M., & Aziz, N. I. (2010). Pressure fluctuation around chute blocks of SAF stilling basins.
- Federal Emergency Management Agency (2010). *Technical Manual*.
- Nixon Flowmeters Ltd. (2003). *Streamflo Velocity Meter*. Glos, UK.
- Outlet Works Energy Dissipators. (FEMA P-679). Retrieved from http://www.damsafety.org/media/Documents/DownloadableDocuments/ResourcesByTopic/P679_EnergyDissipators2010.pdf
- Peterka, A. J. (1978). *Hydraulic design of stilling basins and energy dissipators*. Washington: United States Department of the Interior, Bureau of Reclamation.
- Polysonics, Inc. (2000). *Instruction Manual: HYDRA SX30, Portable Dual Frequency Doppler Flowmeter*. Texas, USA.

Tiwari, H. L., Gahlot, V. K., Goel, A., & Tiwari, S. Y. S. (2015). Improvement of USBR VI Stilling Basin Models for Pipe Outlet.

Tiwari, H. L., Goel, A., & Gahlot, V. K. (2011). Experimental Study of effect of end sill on stilling basin performance. International Journal of Engg. Sci. and Technology, 3(4), 3134-3140.



AN EXPERIMENTAL ASSESSMENT OF BOAT-GENERATED WAVE BY LANGKAWI GEOFOREST KARST TOURIST BOATS

Omar Yaakob, Faizul Amri Adnan, Mohd Arif Ismail, Mohamad Hidayat Jamal, Ahmad Hadi Mohamad Rashidi, Wan Ahmad Hafiz Wan Mohd Azhary, Mohd Kamarul Huda Samion, Andiknurdiyana Bachok, Badruzzaman Ahmad & Nur Shafira Ahmad

ABSTRACT

Sungai Kilim Langkawi, an attractive tourist spot in Malaysia, is now facing an environmental issue of eroded mangrove riverbank. It was open to tourist in 2007 once the place received their accreditation from UNESCO. The erosion problem lies on the numerous number of tourist boat used without control on the speed limit and their production of boat-generated wave (wake wash) height, which is assumed to directly affect the riverbank. To date, no other research work has been done regarding this specific area, but a lot of researches and studies have been carried out across the globe related to erosion caused by boat wake. In this study, measurement on site had been made to analyze the wake produced by the tourist boat. The most frequent boat used (27 feet long type) had been modelled and tested in the NAHRIM facility. Under controlled environment, the wake produced by the model boat had been analysed and compared with the real data. By using a wave decay analysis approach, limited data can be obtained. From the produced wake height, energy per wave height has been estimated. It is found that, 20 Joules/m energy is created in sub critical condition, while more than 20 Joules/m energy is created in super critical condition. Based on this foundation, a new criterion is suggested to be implied at the research interest sites with the additional finding on the factor related to mangrove riverbank erosion at Sungai Kilim, Langkawi.

Keywords: ship wake, ship generated wave (SGW) height, boat hydrostatics and modelling, wave energy

-
1. Marine Technology Centre, Universiti Teknologi Malaysia
 2. Faculty of Mechanical Engineering, Universiti Teknologi Malaysia
 3. Faculty of Civil Engineering, Universiti Teknologi Malaysia
 4. Centre for Coastal and Oceanography Studies, National Hydraulic Research Institute of Malaysia
 5. Hydraulic & Instrumentation Laboratory, National Hydraulic Research Institute of Malaysia

Introduction

Kilim Mangrove Park in Langkawi secured UNESCO Geopark listing status in 2007 and gazetted as “Protected Forest” under the Forestry Act 1984. The listing has promoted the area to become a tourist destination. Visitors to Kilim Karst Geoforest alone rose from 78,145 in 2006 to 307,889 in 2014.

The increasing number of tourists has, however, exerted an excessive demand on the environmental ecosystem of the Kilim Geological Park and threatened the sustainability of this tourism product. The impact of boat-generated waves and sea level rise has negatively affected the ecosystem, particularly the mangrove forests.

Among the main contributors to the problems are high waves caused by speeding tourist boats, boat overcapacity, poorly trained tourist guides and conflicting and overlapping of jurisdictions, as well as lack of scope and responsibilities of the stakeholders to monitor and manage the tourism activities.

This study highlights the effect of waves generated by the tourist boats on river bank erosion. Ships and boats generate waves called as ship generated waves (SGW), wake wash or ship wake. The characteristics of these waves alter significantly with changes in speed, depth of water and hull form (Katsuhido and Kazuki, 1984). According to Abdul Ghani (2008), there are two sets of waves, which are divergent waves that move out at an angle from the centreline of travel, and transverse waves that move out from the stern perpendicular to the centreline of travel. Figure 1 illustrates the wave's pattern at depth water. SGW effects of the waterway on the

boat are categorized in terms of the water depth (h), waterway breadth, ship speed (V) and wave speed (c). Usually, the speed ranges of a ship in a restricted waterway are defined in term of depth by Froude number (Fr_h), which is defined as:

$$Fr_h = \frac{V}{\sqrt{gh}} \quad (1)$$

at critical speed, $Fr_h = 1.0$. Speeds $< Fr_h = 1.0$ are known as subcritical speeds while speeds $> Fr_h = 1.0$ are known as supercritical speeds.

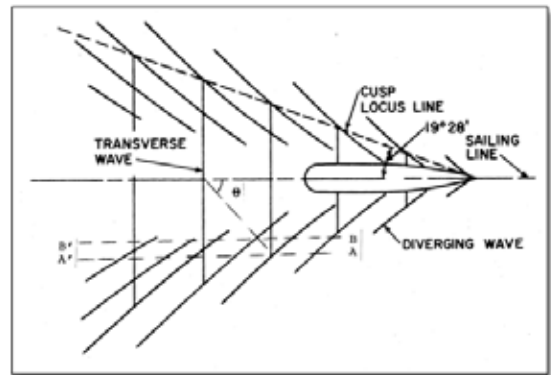


Figure 1.1: Deep water wave crest pattern by moving vessel (Katsuhido and Kazuki, 1984)

Studying the effect of SGW in rivers is prominent because some rivers including Sungai Kilim, Langkawi have shallow water and narrow channels at certain location. Stumbo (2000) found that the energy of wake wash in shallow water is higher than that of deep water, exposing it to more wave energy. Traffic congestion, hull form design, and depth of river water are some other complex factors that contribute to the critical condition because of wave-generated energy created by boat operators navigating their vessels at high speed. Similarly, Velegrakis et al. (2007) found that SGW also affect beaches as additional energy to win-generated waves.

In recent years, several studies focused on the sedimentation and riverbank erosion (Erwan et al., 2008; Sheng Jeng et al., 2014). Since ecotourism industry in Malaysia is increasingly becoming the main source of income for people living in waterway areas, the preservation of mangrove ecology and most importantly the river environment has to be taken seriously.

In short, researches on the environmental impact of generated waves by tourist boat are important because of their damaging effects in the long run. As the tourism industry in Malaysia contributes to the nation's economy and a part of the population is depending on tourism for their livelihood, a careful study should be done especially on boats speed and its wave energy produced, to ensure minimal negative impact of waves generated by boat on riverbank erosion. This paper presents an investigation on SGW by carrying out tests on a model of Sungai Kilim tourist boat. This preliminary assessment on tourist boat wake is basically to compare the laboratory scale test results with those of a full scale measurement of generated wave heights. From this study, relationship between wave heights with its expected energy produced is quantified. The research was started with boat measurement; the selected boat was most commonly used boat in Sungai Kilim. A scale model was constructed using fiberglass and tested in National Hydraulic Research Institute of Malaysia (NAHRIM) facility.

Boat Model Construction

Field measurement: Boat particulars

The particulars, geometry and shape of typical tourist boat (27 feet -V planing hull) had been measured and fed into the MaxSurf ship modelling software.

Model Boat Construction

A 1:7 scale model of the 27 feet boat was constructed using fibre glass. The boat particulars for the full and scale model and their lines plan are shown in Table 2.1 and Figure 2.1 respectively. The loading and speed were set similar with appropriate ratio to the full scale testing. The scaled down speed and weight was calculated by using Froude Number and law of similarity.

Table 2.1: Boat Particular

Particulars	Full Scale	Model Scale
LOA (m)	7.85	1.12
Breadth (m)	2.12	0.30
Height (m)	1.12	0.16
Draft (m)	0.28	0.04
Displacement (kg) (with 7 persons)	950	2.7
Speed (m/s)	11.11	4.2
Speed (knot)	21.6	8.2

Froude Number (Fn) is defined by:

$$Fn = \frac{V}{\sqrt{gL}} \quad (2)$$

where:

V = boat speed (m/s)

L = boat length (m)

g = gravitational acceleration (9.81 m/s²)

As stated by the law of similarities, the Froude Number for real condition boat and scaled down boat must be equal, calculated by using Equation (3):

$$\left[\frac{V}{\sqrt{g \times L}} \right]_p = \left[\frac{V}{\sqrt{g \times L}} \right]_m \quad (3)$$

The final boat speed equation was defined by using Equation (4):

$$V_m = \frac{V_p}{\sqrt{\lambda}} \quad (4)$$

where:

V_p = real condition boat speed (m/s)

V_m = scaled down boat speed (m/s)

λ = scale factor = $\frac{L_p}{L_m}$

For scaling of weight, Equation (5) was used:

$$W_m = \frac{W_p}{\lambda^3} \quad (5)$$

where:

W_p = real condition boat weight (kg)

W_m = scaled down boat weight (kg)

which provided the minimum voltage required for the model boat to move at desired speed during the model tests.

Table 2.2 shows the weight of the components in boat model. It was found that despite all efforts to reduce the weight, the total weight still exceeded the target by 0.8kg, which is equivalent to the full scale boat carrying 11 persons, instead of 7 persons.

Table 2.2: Weight of boat model components

Component	Weight (Kg)
Motor, 660kV	0.5
Battery, 14.8V	0.5
Propeller unit	0.13
Rudder unit	0.24
Shaft	0.15
Servo system controller	0.17
Compartment mounting	0.3
Cooling system	0.01
Constructed fiber glass boat	1.5
TOTAL	3.5

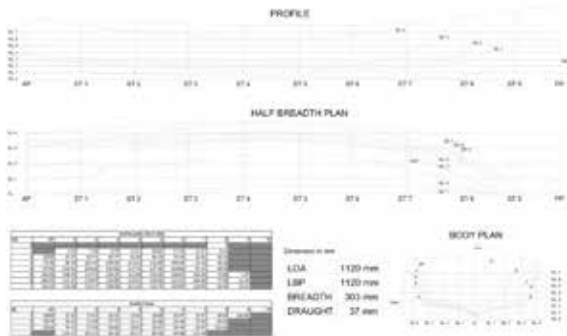


Figure 2.1: Lines plan for boat model

One of the challenges in the construction was the need to make the model as light as possible, while maintaining strength and structural integrity. The boat model was inclusive of battery, control and the propulsion system should weigh 2.7 kg, equivalent to the full scale boat with 7 persons onboard. To reduce the overall weight, electric propulsion system was used, instead of gasoline engine. The propulsion system of the boat was powered by a small 14.8V battery,

Figure 2.2 and Figure 2.3 show the model in the making and the arrangement for propulsion system of the boat model, respectively. Several pre-testing runs were made at Marine Technology Centre (MTC), UTM and UTM Recreational Lake to check its stability and alignment before running actual test at NAHRIM. Pre-testing of the boat is shown in Figure 2.4.



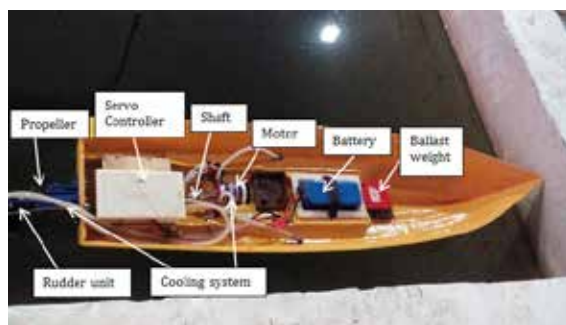
(a)



(b)



(c)

Figure 2.2: Boat moulding and on water stability check**Figure 2.3:** Finished Model complete with control and propulsion system

(a)



(b)

Figure 2.4: Model pre-testing runs at MTC, UTM and UTM Recreational Lake

The tests were carried out in the coastal engineering basin at NAHRIM. A scaled slope with a riverbank/beach was provided, and wave probes were located at strategic locations near the slope.

Preparation of Test Location

As the boat model testing is a reflection of the real situation as experienced by actual boat in Sungai Kilim, Langkawi, a slope was provided along the side of the basin near the boat path. This slope resembled the slope of the riverbank and bed at Sungai Kilim, referred from NAHRIM bathymetry data at selected location. Figure 3.1 shows the scaled down particulars of the slope. The slope was created to have 15m length and was simplified to a straight line. Figure 3.2 shows the slope preparation made before conducting the test.

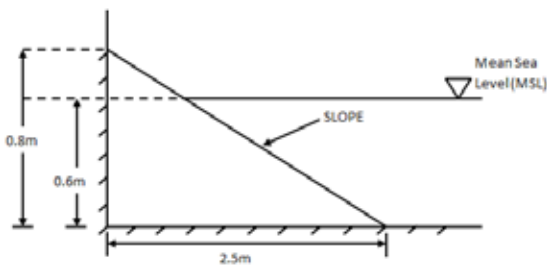


Figure 3.1: Slope setup for model testing

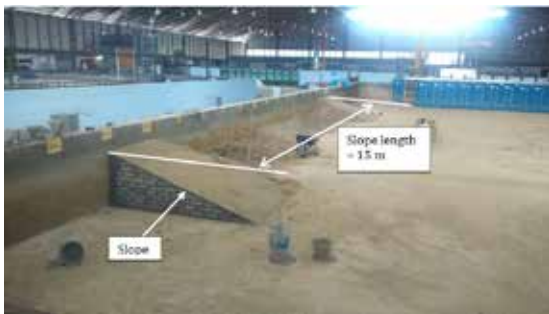


Figure 3.2: Slope setup at NAHRIM facility for model testing

Wave data was assessed using resistance wave probes. The wave probes were situated similar to the position of the tide gauge during the field measurement process, for comparison process later. Figures 3.3 and 3.4 show the location of tide gauge during the field test and its schematic

diagram, respectively.



Figure 3.3: Location of tide gauge during field test

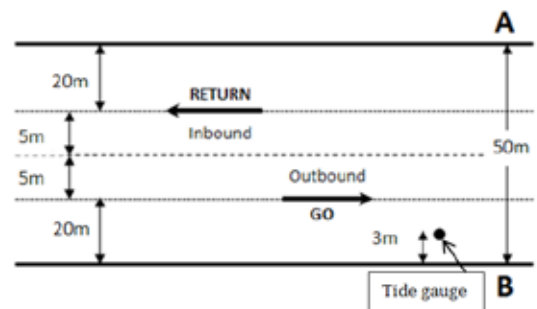


Figure 3.4: Schematic diagram from field test

By using the same scale factor for boat model construction (1/7th), equivalent dimensions were obtained for model testing. For model tests at NAHRIM, more than one wave probes were used for further analysis and comparison process, for better understanding of the produced SGW. However, they could not be located exactly at the same corresponding location because the relatively large size of probes would touch the slope if installed close to the 'shore'. The locations of the wave probes are shown in Figure 3.5.

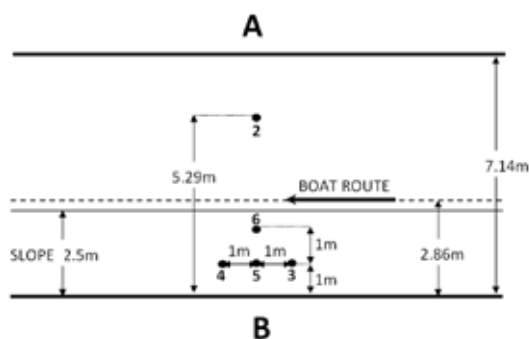


Figure 3.5: Schematic diagram for test procedure (numbers indicate wave probe numbers)

Wave probes 3, 4 and 5 were arranged in a straight line. These three wave probes were expected to have similar wave heights, with appropriate time delays. Wave probe 2 was arranged as a mirror to wave probe 5. Before starting any test, the wave probes needed to be calibrated first. 20 seconds were allocated for measurement for each run using data acquisition rate of 50 data per second. Figure 3.6 shows the wave probes locations and the starting and finishing lines which were separated by a distance of 13 meters.



Figure 3.6: Wave probe locations and model testing distance line at NAHRIM facility

Speed measurement

The boat did not have a speed logger; therefore, speed was manually measured. The speeds of the boats were calculated from the duration for the boat to traverse the distance from the starting to the finishing lines. It was difficult to accurately control the speed using the remote controller. The boat model was run at various speeds as close as possible to the desired speed of 4.2 m/s, equivalent to 21.6 knots by the full scale boat. The boat was also tested at low speeds (1.3-2.3 m/s equivalent to 2.5-4.5 knots full scale) to see the effect of low speeds on the reduction of SGW. Table 3.1 shows the speed variation during the test. Figure 3.7 shows a typical test run.

Table 3.1: Speed variation for model testing

THROTTLE	FROM	TO
Full	4.3 m/s	5.0 m/s
Half	3.5 m/s	4.2 m/s
Low	1.3 m/s	2.3 m/s



Figure 3.7: Typical test run

Data Analysis

Results: Data and discussion

SGW data measured by the wave probes were fed directly into the data acquisition system in NAHRIM facility, by which each probe being

connected to the data acquisition system via a channel, which was numbered according to the probe number, i.e. probe 2 being connected to Channel 2 etc. This section presents the results of the analysis of the data, in three sub-sections:

- Raw wave data results
- Wave data similarities
- Wave decay envelope analysis

Raw Wave Data Results

Table 4.1 shows the time taken by the boat model to finish the run distance of 13 meter at various speeds, based on records from four time keepers (t1 to t4). Depth Froude Number was calculated using Equation (1):

$$Fn_h = \frac{V}{\sqrt{gh}} \quad (5)$$

where:

V = ship speed (m/s)

g = gravitational acceleration (9.81 m/s²)

h = water depth (m)

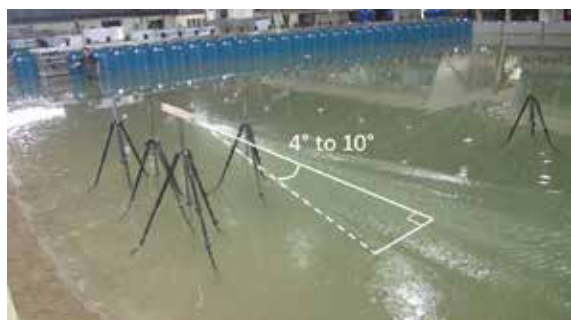
Using the depth Froude number criteria could determine whether the boat was in a supercritical or subcritical condition.

Table 4.1: Time taken for various speed condition and Depth Froude Number

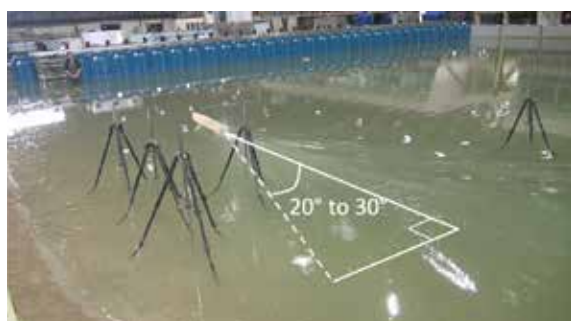
No.	t1	t2	t3	t4	average time(s)	dis. (m)	MODEL	PROTOTYPE			Depth Froude No.	
							Speed (m/s)	m/s	knots	km/h		
run 1	3.09	3.13	3.1	0	3.11	13	4.18	11.07	21.52	39.86	1.645	Super-critical (Fn _h >1)
run 2	3.1	2.88	3.2	0	3.06	13	4.25	11.24	21.85	40.46	1.670	
run 3	2.4	2.6	2.7	0	2.57	13	5.06	13.40	26.05	48.24	1.991	
run 4	3.9	3.5	3.7	3.7	3.70	13	3.52	9.30	18.08	33.48	1.382	
run 5	2.6	2.5	2.6	2.6	2.58	13	5.05	13.36	25.96	48.09	1.984	
run 6	3	3.2	2.97	0	3.06	13	4.25	11.25	21.87	40.51	1.671	
run 7	9.77	9.8	10	9.5	9.77	13	1.33	3.52	6.84	12.68	0.523	Sub-critical (Fn _h <1)
run 8	6.2	5.76	6	6.4	6.09	13	2.13	5.65	10.98	20.33	0.839	
run 9	7.1	6.76	6.9	7.1	5.57	13	2.33	6.17	12.00	22.22	0.917	

Figure 4.1 shows the crest angles for Run 6 (4.25 m/s, supercritical) and Run 8 (2.13 m/s, subcritical) SGW produced by the boat. The angles of 4° to 10° and 20° to 30° were for supercritical and subcritical conditions,

respectively. The test results showed that Run 1 to Run 6, which had speeds between 3 m/s to 5 m/s experienced supercritical condition while at speeds lower than 2.5 m/s, the boat experienced subcritical condition.



(a) Run 6 (4.25 m/s) for supercritical condition



(b) Run 8 (2.13 m/s) for subcritical condition

Figure 4.2: SGW crests for super and subcritical condition

An example data from the data acquisition system for each run and channel was then tabulated, as shown in Table 4.2. The data used for the analysis were actually prototype data, as given in column 5 and 6.

Table 4.2: Wave data tabulation in Excel from 0s to 0.12s

Scale	1	Units	mm		
Model Time (s)	Raw Data	Processed Data	Calibrated Data	Pro-to-type Time (s)	Pro-to-type Data
0	0.023462	0.023462	0.880996	0	0.881
0.02	0.025599	0.025599	0.961212	0.02	0.961
0.04	0.02865	0.02865	1.075804	0.04	1.076
0.06	0.030787	0.030787	1.156021	0.06	1.156
0.08	0.033533	0.033533	1.259155	0.08	1.259

0.1	0.03628	0.03628	1.362288	0.1	1.362
0.12	0.037806	0.037806	1.419586	0.12	1.420

The screen capture of the raw data graph was plotted referring to the data acquisition system, as shown in Figure 4.3.

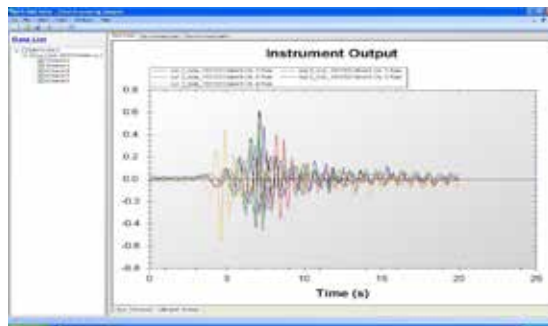


Figure 4.3: All channel wave data provision for one run.

Wave Data Similarities

As mentioned previously, it was expected that the wave probes which lay in a line parallel to the direction of the boat would experience the same wave pattern. This can be seen as in Figure 4.4 that shows wave heights from Channels 3, 4 and 5 for boat speed of 4.25m/s (Run 2). The wave patterns for these 3 channels were very similar, albeit with time shift. Further analysis should thus use only data from one of the channels, in which Channel 5 was selected.

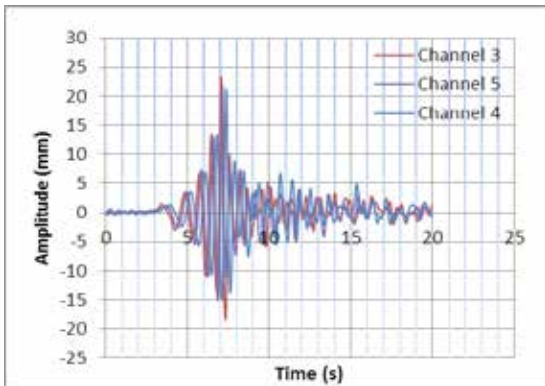


Figure 4.4: Channel 3, 4 and 5 wave height and wave pattern.

Wave Decay Envelope Analysis

In general, wave envelope is a smooth curve outlining its extremes (peaks) in the function of an oscillating signal, which in this case study, is SGW by the boat. The purpose of using this wave envelope was to predict the wave heights at other locations where wave probes were not present particularly at the same location of the tide gauge during the field scale measurement. During the field work in Sungai Kilim, the tide gauge was placed 3 meters from the river bank, equivalent to 0.43m at 1:7 scale. Due to the nature of the wave probes and the presence of the slope in the model experiment, the wave probes could not be located at equivalent distance. Instead, they were located at a distance of 1m. Thus, to obtain the wave heights produced by the boat model at 0.43m, wave decay envelope analysis had been carried out.

Supercritical and subcritical (calculated using Depth Froude number) conditions are associated with different wave crest angles. In order to extrapolate the results to the actual tide gauge, estimation was made according to the wave crest angles produced by the boat model. Figure 4.5 shows the method to determine the exact

distance and the time taken for the waves to reach the scaled down tide gauge distance from shore, which was 0.43m. The analysis of wave decay envelope was simplified by using a linear decay line. Only the maximum possible angle was taken into account because of the very small difference in the output-calculated time, between each condition.

In Figure 4.5, the supercritical condition is illustrated as blue line while subcritical condition is shown as red line. The routes or paths of the boat in both conditions were the same, but shown slightly shifted in this figure to differentiate them.

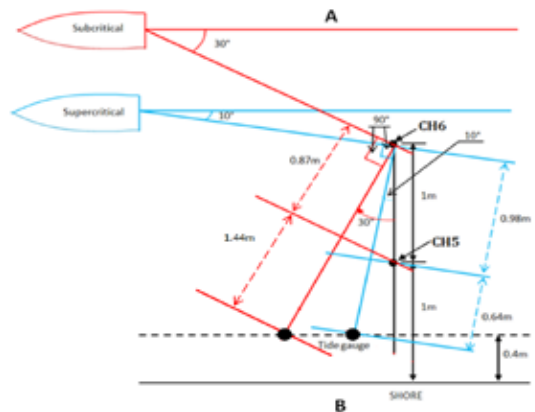


Figure 4.5: Tide gauge location and wave angle at two conditions

By using Channel 5 and Channel 6 wave data, the wave height at the exact location of the tide gauge during field-testing could be obtained through interpolation. As the waves travelled at their supercritical or subcritical condition and were perpendicular to the wave probe, the distance and wave travel time could be calculated.

Figure 4.6 shows the wave decay envelope sample at speed of 5.06 m/s. By using the time

taken from the interpolation step referring to Figure 4.5, the wave heights could be measured, whose results are shown in Table 4.3. The rest of the wave decay envelope patterns had been analysed and discussed as well.

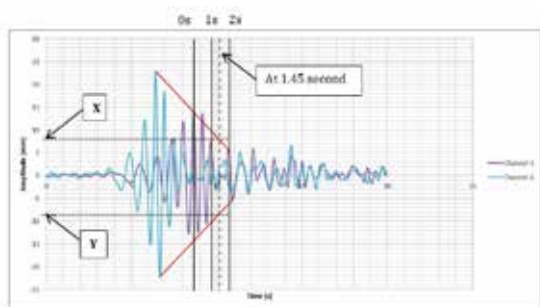


Figure 4.6: Wave height and wave decay envelope for speed of 5.06 m/s

Table 4.3: Wave height results from test (wave decay envelope method)

	Speed (m/s)	Peak to peak of 0.98m time (s)	Tide gauge distance (0.64m) time (s)	X	Y	Model scale wave height (mm)	Full scale wave height (m)
	5.06	2.22	1.45	8.0	8.5	16.5	0.116
	5.05	2.28	1.50	9.0	8.0	17.0	0.119
	4.25	2.26	1.50	6.5	8.5	15.0	0.105
	4.25	2.76	1.80	7.5	7.5	15.0	0.105
	4.18	2.22	1.45	9.5	7.0	16.5	0.116
	3.52	2.22	1.45	7.5	8.0	15.5	0.109

	Speed (m/s)	Peak to peak of 0.87m time (s)	Tide gauge distance (0.98m) time (s)	X	Y	Model scale wave height (mm)	Full scale wave height (m)
	2.33	2.11	2.40	6.0	3.5	9.5	0.067
	2.13	2.24	2.50	6.0	2.5	8.5	0.060
	1.33	2.56	2.90	3.0	2.5	5.5	0.039

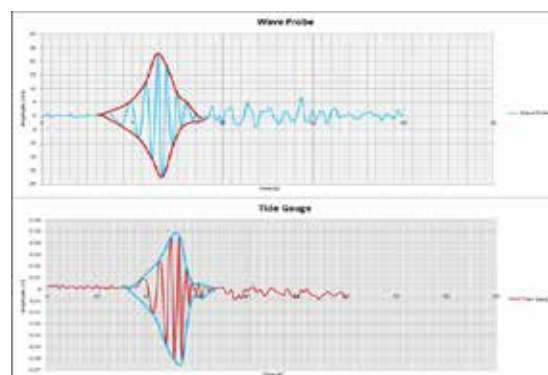


Figure 4.7: Wave envelope for Model Testing (above) and Field measurement (below)

Comparison of SGW envelope pattern showed good similarity. Figure 4.7 shows full wave envelopes from model testing (top) and field measurement (bottom), which indicate that the boat model produced similar SGW pattern to that of actual boat in Sungai Kilim, Langkawi. This suggests that the model results reflected the actual full scale results.

The data for field test was measured during the field measurement using the exact number of person boarding, which is four to seven persons. Wave height data was measured using tide gauge attached to a pole, as shown in Figure 3.3. Thus, the data from field could be tabulated

and compared with the data from model testing (up scaled wave height), as shown in Figure 4.8.

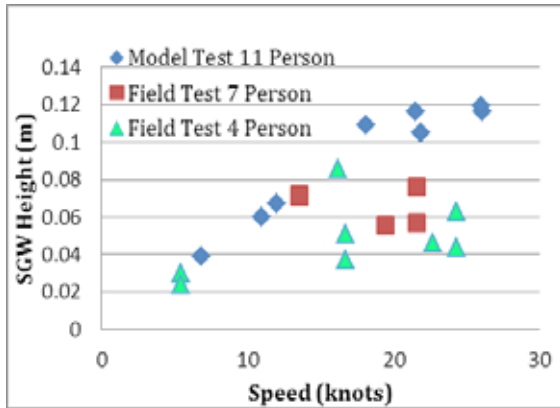


Figure 4.8: SGW height of 27 feet boat versus speed

The points show that when the speed of the boat was increased, the wave height increased perpendicularly. This was true for load of 11 people on board during the model testing. Other than that, for 7 and 4 people on board from field measurement data, the value was randomly distributed, due to bathymetry effect, where the sailing line of the boat might have sailed beneath a non-constant water depth. The overall pattern however, when reaching a speed of 12 knots, the wave height still tended to increase, but with a small difference for all three conditions as a response to the speed condition of the boat. This might have happened due to the planing mode the boat experienced at certain speeds; in this case, 12 knots and above. During displacement and transition condition (acceleration) the v planing hull experienced increment in resistance, as a response directly to the SGW height produced. For fewer number of passenger (low draft), the wave height produced by the boat was smaller compared to more number of passenger on board (high draft). Apart from that, the position of passenger on

board also plays an important role in producing wake. These positions will determine whether the boat is in trim or no trim condition, which will later affect the magnitude of wave height produce. Proper position of the passengers on board helps to increase boat stability as good ballast. This is a preliminary assessment on the effect of number of passenger to the SGW height produced by boat.

Wave Energy per Maximum Wave Height Analysis

In relation to erosion experienced by the riverbank, wave energy needed to be estimated as it might be the main contributor instead. To assess the energy produced by the boat model, energy formulation as suggested by Gourlay (2011) had been used. In deep water, d is larger than $\lambda/2$, where d is water depth and λ is the wavelength. Since the boat model was assumed to travel beneath a depth water condition, the transmitted wave energy was estimated by using Equation (6) as follows:

$$E = \frac{\rho g^2 H^2 T^2}{64\pi} \quad (6)$$

where:

- E = Energy per meter of crest length (J/m)
- ρ = ship speed (m/s)
- g = gravitational acceleration (9.81 m/s²)
- H = wave height (m)
- T = wave period (s)

However, only energy per maximum wave height was studied in this research. The amount of energy per maximum wave height is tabulated in Table 4.4. For estimating the maximum allowable energy per maximum wave height, a permissible 75mm wave height as used at River Gordon, Australia was referred. Another

criterion used was by implying the maximum wave energy criteria by MacFarlane and Cox 2004; in which the maximum allowable energy is less than 60 Joules/m as used at River Noosa, Australia. The environment or vegetation for the criterion chose may be different with current research condition, but with some similarities such as compact shorelines type and often sheer soil banks, characteristics of these rivers can be used as benchmark on the allowable maximum produced SGW height. Thus, an extensive study for the criteria applicable at the interest research site of Sungai Kilim should further be carried out so it could have its own and unique criterion.

Table 4.4: Wave energy per maximum wave height for 11 people on board condition gained from model test data

Velocity Model (m/s)	Velocity Full Scale (knots)	F _{hh}	H _{max} Full Scale (m)	T _{peak} Full Scale (s)	Energy H _{max} (Joules/m)
5.06	26.023	2.086	0.116	1.32	22.017
5.05	25.972	2.082	0.119	1.48	29.318
4.25	21.857	1.752	0.105	1.48	22.825
4.25	21.857	1.752	0.105	1.43	21.224
4.18	21.497	1.723	0.116	1.32	22.017
3.52	18.103	1.451	0.109	1.48	24.372
2.33	11.983	0.960	0.067	1.88	14.736
2.13	10.954	0.878	0.060	1.85	11.464
1.33	6.840	0.548	0.039	1.85	4.800

From the model testing carried out in NAHRIM facility, all produced maximum wave heights, resulting in a small amount of energy (less than 60 Joules/m). During this model testing, it was easy to recognize the subcritical mode experienced by the boat, as it had energy per maximum wave height not exceeding 20

Joules/m at speed range of 6 to 12 knots. This made the speed range from 13 to 26 knots to create supercritical condition, producing more than 20 Joules/m energy. From the table, it can be concluded that in most cases, with higher SGW height, higher wave energy would be produced. However, the energy produced did not relate directly to the boat speed but only to the wave height and wave period produced. For instance, when the boat model was run under 5.06 m/s, the SGW height and produced energy were lower than those of 3.52 m/s speed. It was also found that the speed of 5.06 m/s and 4.18 m/s produced the exact amount of energy at maximum wave height. The difference and similarities might be because the sailing line in which the model boat travelled was not kept in constant straight line even though a careful control was implemented. The model boat was quite difficult to be controlled at faster speed. This resulted in different depth during sailing, and the boat might have surged over the setting slope. Since there was change in water depth, it directly affected the wave height and its period. In the model testing, it was still possible to control the condition. However, the wave height and period during field measurement resulted in random values since the bathymetry and topology beneath were different and it was quite impossible to make a constant sailing line. Nevertheless, a clear and robust relationship could be identified between the boat speeds, hull form and boat draft according to the people on board to the production of boat wake; made possible by using physical model testing and computational approach.

Conclusion and Recommendations

Preliminary assessment has been carried out accordingly using the facility in NAHRIM to determine the wave height and wave period produced under certain speed and draft. These are the main parameter related to the energy production of the boat. Data have been calculated and analysed based on the wave decay analysis. A comparison made between testing data and experimental work concludes that the wave undergoes the same enveloping process. Thus, a decay analysis is true for extrapolating the wave data. With a limitation of being unable to reduce the weight of the model, the final model resembles the draft of 11 people on board. It is, nevertheless, comparable to data boarded by 4 to 7 people, as shown in Figure 4.8.

The energy has been quantified using the energy per maximum wave height approach. 75 mm and more wake wave heights are found to produce more energy compared to the less height. The energy however, is slightly lower than the allowable energy of 60 Joules/m per maximum wave height produced in one full wave development. In conclusion, the allowable energy should be less than 60 Joules/m produced by the maximum wave height at the research interest cites, as well as the maximum wave height since by these criteria, Sungai Kilim riverbank erosion would still happen. It is also recommended that the total energy of wave train and the frequency number of boat at some period of time should be taken into account to relate the wake produced with erosion rate in current research. In other words, the total energy produced in a cumulative wave train at one SGW development and other parameter such as sediment transportation may still have effects on the rate of erosion and be

closely related to the condition of wake, which however, were not analysed in this study.

References

- Abdul Ghani.M.P., Rahim. M.N.A. (2008). The Prediction of Wake Wash in the Towing Tank. *Jurnal Mekanikal - Universiti Teknologi Malaysia*, 26, 129-140.
- Erwan Garel, Laura Lopez Fernandez, Mike Collins. (2008). Sediment resuspension events induced by the wake wash of deep-draft vessels, *Geo-Marine Letters* 28(4): 205-211 (2008)
- Gourlay, T. (2011). Notes on shoreline erosion due to boat wakes and wind waves (Rep.). Retrieved March 26, 2016, from http://cmst.curtin.edu.au/wp-content/uploads/sites/4/2016/05/Gourlay_2011_Notes_on_shoreline_erosion.pdf
- Katsuhiko Kurata and Kazuki Oda (1984). *Ship Waves in Shallow Water and Their Effects on Moored Small Vessel*, pp. 3258-3273.
- Macfarlane, G. J., & Cox, G. (2004). The development of wave wake criteria for the Noosa and Brisbane Rivers in South East Queensland. In *Fifth International Conference on Coastal Environment 2004* (Vol. 10, pp. 55-72).
- Sheng Cheng, J.I, Abdellatif, O., Hassan, S. Philippe, S., (2014). 3D Modelling of sediment movement by ships-generated wakes in confined shipping channel. *International Journal of Sediment Research*, 29, pp 49-58.

Stumbo, S., Fox, K. Elliott, L. (2000) An Assessment of Wake Wash Reduction of Fast Ferries at Supercritical Froude Numbers and at Optimized Trim. Proceeding of the International Conference on Hydrodynamic of High Speed Craft, November 7-8, London: RINA.

Velegrakis, A. F., M. I. Vousdoukas, Vagenas, A.M., Karambas, Th., Dinou, K., Zarkadas, Th. (2007). Field observations of waves generated by passing ships: A note. Coastal Engineering. 54(4), 369-375

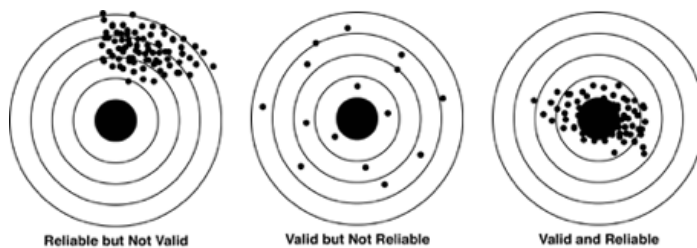
CALIBRATION OF A DRAG TILT CURRENT VELOCIMETER FOR AN OPEN CHANNEL APPLICATION

Ibnu Akil Maitar Muharram, Zool Hilmi Ismail & Kamarul Bahrin Mohamad

ABSTRACT

Water is one of the prime elements responsible for life on earth with two thirds of the earth's surface covered by it. Managing water resources effectively can promote conservation and make the best use of our limited water resources. Having knowledge about flow information on the river networks can be very beneficial for applications such as hydropower, transportation, irrigation, flood mitigation, water treatment, industrial and domestic needs. The development of simple and inexpensive instrument (<RM500) is to help researchers to produce and measure flows on spatial river at a very low cost. The analytic approximation was derived for giving relationship of the instrument's tilting behavior on water velocity. Calibration process was held and the data are used to investigate the relationship between the measurement collected from the experiment and the known values. Analyzing the experiment results, it was concluded that the instrument tilting in response to the water velocities. Random and systematic errors arise during calibration processes which could contribute to the high uncertainties of the instrument. The documented experiments, procedure and facility used for the calibration are given in this paper. It is shown that the accuracy relationship of this instrument against the analytic approximation identified is 66.45%.

Keywords: Low cost; flow meter; calibration; reconfiguration system



¹ Senior Electrical Engineer, Pasukan Projek Penyelidikan SHRE, Caw. Kejuruteraan Elektrik, JKR Malaysia, Kuala Lumpur, Malaysia

² Centre of Artificial Intelligence and Robotic, Universiti Teknologi Malaysia, Jalan Sultan Yahya Petra, Kuala Lumpur, Malaysia

³ Senior Principle Assistant Director (Electrical), Pasukan Projek Penyelidikan SHRE, Caw. Kejuruteraan Elektrik, JKR Malaysia, Kuala Lumpur, Malaysia

Introduction

The development of Drag Tilt Velocimeter (DTV) has brought in the simplest technology and hydrological concepts into measuring the velocity of water at the river and streams. This innovative approach in developing an electronic flow meter is based on the drag force that a body will experience when immersed in a fluid stream and has proven excellently in measuring flow in the ocean for aquatic study (R.Marchant, T.Stevens, S.Choukroun, G.Coombes, M.Santarossa, J.Whinney and P.Ridd, 2014). This DTV is meant to be deployed in the river as it is required for measuring water velocity in field for the mini hydro project. It will serve the purpose of judging the river conditions to locate river flows which will suit the turbine placement in order to assure a highly profit and low risk project development as well as boosting the investor's confidence (JKR Malaysia and UNIMAS Sarawak, 2010).

There are many flow measurement techniques as well as different types of flow meter to measure the river flow but these all are too costly and require extra effort to monitor and additional safety measures to figure in as well as the maintenance itself. Various types of flow meters are available in the market and Table 1 below lists out the different types of flow meter based on various principles (Ria Sood, 2013).

Table 1: Flow meter category based on various principles

Principle	Types
Differential Pressure	Orifice Plate type of meter, Rota Meter, Flow Nozzle, Pitot type Tube, Elbow Tap, Venturi Tube
Positive Displacement	Oval Gear type, Rotating Disc type, Rotary Vane type, Reciprocating Piston

Velocity	Turbine type, Vortex Shedding Electro-magnetic, Ultrasonic Doppler type, Ultrasonic Transit Time type.
Mass	Coriolis, Thermal
Open Channel	Weir, Flume

The sensor used to measure velocity can be either contactable type with water or non-contactable. Despite the ability to give highly accurate and precise water velocity, the technology of ultrasonic and acoustic Doppler comes with a price where their deployments at site are limited. The deployment of low cost, robust, user friendly and accurate instrument can be beneficial for the spatial dense of river study. The accuracy of the measurement is an essential step for the qualitative and economic points of view. This paper will observe the behavior of the drag-tilt velocimeter during calibration process and analyze the error made by it.

Method of Approach

The Fundamentals of Drag Tilt Velocimeter

Our design of drag tilt velocimeter uses a negatively buoyant object which is mounted under the pontoon or any rigid structure. This technique is not new, but our approach of measuring the tilt angle using a cheap and low power sensing devices is much more accurate than any of the previous systems which manually estimated the angle by visual, which will cause uncontrolled random error and low range of measurement (B.Kjerve, 1982). The measurement ranges of DTV which relies on tilt angle also increases. There are possibilities for communication through wireless sensor network (Mohd Fauzi Othman, 2012) or cabling direct to the console. The assembly of fittings is also different from others where we introduced a

simple and commercially available part made by galvanized iron tube as the drag tilt object where an accelerometer is attached at the bottom of the tube.

Figure 1 below shows the schematic of the Shre Drag Tilt Current Velocimeter (DTCV). The self-weight of the tube should be more than the buoyancy force to act as a restoring force when there is no current and the tilt angle is zero. A downward 'zero' position kept by the negative buoyancy of the tube in a situation of no current.

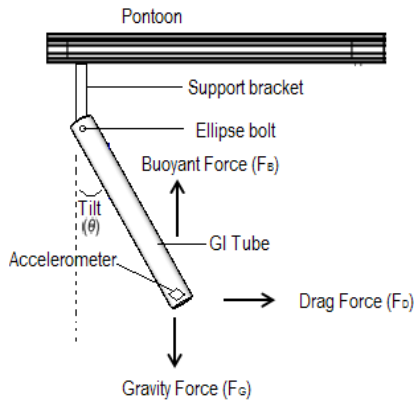


Figure 1: Shre DTCV Concept

The Analytic Approximation

The technique's principle is that; a drag force will make the submerged tube deviate from the vertical line. An analytic approximation of the system of forces was performed to relate tilt, current velocity, tube size and buoyancy. We consider a simplified static solution with the assumption of horizontal flow than a dynamic solution which is more complicated and nonlinear with flow from various angles.

From energy conservation, the drag force, F_D equals to:

$$F_D = \frac{1}{2} v^2 \rho C_D A \quad (1)$$

where,

F_D is drag force, (N)

v is current velocity, (ms⁻¹)

ρ is water density, (kgm⁻³)

C_D is the drag coefficient, (Unitless) and

A is the cross section area of the tube, (m²).

The drag coefficient, C_D varies with the various body size of tube (Streeter, 1961), (Monalisa Malik, 2014).

By equating the drag and restoring forces (B.Kjerve, 1982), it follows that:

$$v = k (\sqrt{\tan \theta}) \quad (2)$$

where,

k is sensitivity factor, (Unitless)

θ is the tilt angle, (radian)

while,

$$k = \sqrt{\frac{2 m_{sub} g}{\rho C_D A}} \quad (3)$$

where,

v is current velocity, (ms⁻¹)

m_{sub} is the submerge tube mass, (kg)

g is the gravity coefficient, (9.81 ms⁻²)

ρ is water density, (kgm⁻³)

C_D is the drag coefficient, (Unitless) and

A is the cross sectional area of the tube, (m²)

m_{sub} , the submerged mass of the tube assembly is calculated by

$$m_{sub} = m_{tube} - \rho V \quad (4)$$

where,

$m_{\text{(tube)}}$ is the mass of the tube, (kg)

ρ is water density, (kgm^{-3})

V is the volume of the tube, (m^3)

The Instrument Setup

The Shre DTCV was manufactured using commercially available materials. We used a bracket as the upper body and ellipse bolt as the swivelled connector for the lower body. The bolt will also prevent the tube from rotating about its longitudinal axis. The dimension of lower body made by galvanized iron is 25mm diameter and 400mm length. The 25mm diameter of pipe allows the insertion of the accelerometer sensor to the bottom of it for data logging. The tube allows the measurement of tilt direction.

Automated Data Collection

The ability to collect useful measurement information on water flow is dependent on the timing and frequency of data collection over time. A very quick response on water flow measurement is a benefit of automated and real time data collection. The installation of the drag-tilt velocimeter under the pontoon based will support the quick and real time collection of the data at any weather condition as the controller and cabling systems of the sensors can be done on the surface of the pontoon. The tilt was measured with an ADXL335 triple axis accelerometer by Analog Devices and supported by Arduino microcontroller.

Selection of Sensor

The main electronic device that detects and measures water velocity through drag and tilt pipe is an accelerometer. It acts as a sensory organ for the brain in the controller, giving its information

to make calculation. As the water flows through the tube, it tilts and the accelerometer inside the tube will also be displaced according to the tilt movement. The ADXL335 is a triple axis accelerometer with extremely low noise and low power consumptions shown at Figure 2 below.



Figure 2: Accelerometer

Even though this sensor is called accelerometer, we use it as a tilt sensor. The reason why we just do not simply get the acceleration output is because we want the reading in velocity but it is hard to convert acceleration to velocity. That is why we use the formula to get the velocity output by using the formula given by Eq. (2).

The reading will be much more accurate based on the object mass and volume, cross-sectional area, water density, coefficient of drag and gravity. We will use the output angle from the ADXL335 into the formula to calculate the velocity of the water in ms^{-1} .

The operating voltage range is only 1.6V - 3.6V. Oversupply of the voltage will spoil the sensor. The ADXL335 will connect to the 3.3V on the Arduino UNO board. The most important thing is, when we attach ADXL335 to something, when it is at rest or static, the velocity or angle reading must be zero. Otherwise, the rest of the reading will not be accurate. Table 2 below lists the absolute maximum ratings of the accelerometer sensor.

Table 2: Absolute Maximum Rating of Accelerometer

Parameter	Rating
Acceleration (Any Axis, Unpowered)	10,000g
Acceleration (Any Axis, Powered)	10,000g
Vs	-0.3 V to +3.6 V
All Other Pins	(COM -0.3V) to (Vs + 0.3V)
Temperature Range (Powered)	-55°C to +125°C
Temperature Range (Storage)	-65°C to +150°C

Associated Errors

An error in the scientific context is a point to the uncertainty that affects all the measurement. Errors are inevitable. The best one can do is ensuring the errors are as small as reasonably possible and how large they can go could be estimated (J.Taylor, 1997). For every new instrument, it would be recommended to be calibrated to make sure the instrument accuracy indicated achieved. Occurrences of error in instrument may cause by many factors such as drift, environmental, electrical supply, addition component to the output loop, type of material used, mechanical wear and tear, process changes, etc. (Johnstone, 2008).

Experimental error can be classified into two categories; namely random and systematic errors. Random Error results from unknown or unpredictable variations during experiment such as temperature or in-line voltage fluctuations. These accidental errors can be reduced by repeating the measurement, taking average value or improving the experimental technique (Strupstad, 2009).

Systematic Errors are associated with particular instruments or techniques. Reducing the systematic errors always depends on the skill of the experimenter to detect, prevent and correct them since improperly calibrating the instruments are the sources of this error (Strupstad, 2009). Measurements with relatively small determinate error are of high accuracy.

Fitting experimental data using the least square method are mostly chosen. The n experimental data points (x_i, y_i) , where $i=1, 2, \dots, n$, can be modeled by the function $f(x_i, p_m)$ with m adjustable coefficients. The “best” of least square method is defined when the sum of squared residual, $\sum SS_{res}$ is minimal. Given by (J.Taylor, 1997), the $\sum SS_{res}$ is computed as:

$$SS_{res} = \sum_{i=1}^n (y_i - f(x_i, p_m))^2 \quad (5)$$

It is defined as the difference values of the dependent variables y_i and the predicted values from the estimated model $f(x_i, p_m)$.

The goodness of the fit statistical model is typically reported as R-square either it is a line or a curve. An R^2 closer to 1 indicates a better fit where the regression line perfectly fits the data. If $\bar{y}$ is the mean of observed data, given by (J.Taylor, 1997),

$$\bar{y} = \frac{\sum_{i=1}^n y_i}{n} \quad (6)$$

The variability of the data set can be measured using this sum of squares formula;

$$SS_{tot} = \sum_{i=1}^n (y_i - \bar{y})^2 \quad (7)$$

and R2 is computed by;

$$R^2 = 1 - \frac{SS_{res}}{SS_{tot}} \quad (8)$$

Experimental Setup

Measurement Facility

Most of the time, current meter calibration is carried out by towing instrument meters along a tank at a series of predetermined speed. At National Hydraulic Research Institute Malaysia (NAHRIM), Seri Kembangan, Selangor there are only 60m long flume which can operate reliably at lower speed over a velocity range of 0-0.7ms⁻¹. The flume section is 60 meters long, 2 meters deep and 1.2 meters wide. It has a combination of concrete walls and glass side wall. Figure 3 below shows the plan view diagram of the experimental set up for this calibration process.

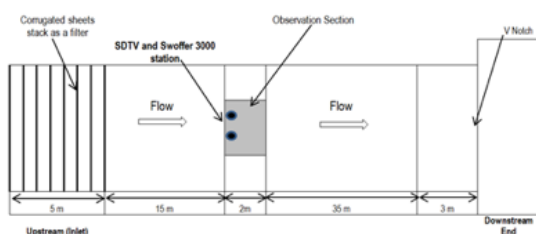


Figure 3: Plan view of the experimental set up

The width result is dimensionally stable and repeatable. Figure 4 below shows the general view of the flume.



Figure 4: Flume Facilities at NAHRIM

To perform the experiment, the flume was filled with water up to 1.2 meters deep. The flow was generated by circulating the flow from the holding pond at the downstream end of the flume back to the upstream end of the flume. To accomplish the process, 2 units of variable heavy duty electrical powered 4hp pump was installed to force the water up with the pump inlet connected to the downstream holding pond then the pump outlet was connected to an inlet of the upstream flume structure which goes through a series of filters. The entrance to the flume was designed to give disturbance free uniform velocities by series of filters so fairly uniform laminar flows go through the flume.

Calibration Procedure

At the early stage, the water left stagnant and stable without any flow to locate a downward 'zero' position kept by negative buoyance of the drag tilt tube. In this no current situation, a tilt reading was recorded digitally by accelerometer to the computer and the real flow reading recorded visually by Swoffer 3000, a propeller types current meter.

To measure the response to differing flow rate, water velocity was varied from 0 ms⁻¹ to 0.7 ms⁻¹. It was done by running the water through the flume by controlling the water pump power

switch to get variations of speed. Until the flow becomes perfectly stable at the location identified, the flow readings were recorded. The drag tilt velocimeter was steadily installed about 15 meters from the upstream end, while the Swoffer 3000 was attached adjacent to it as per Figure 5 below.

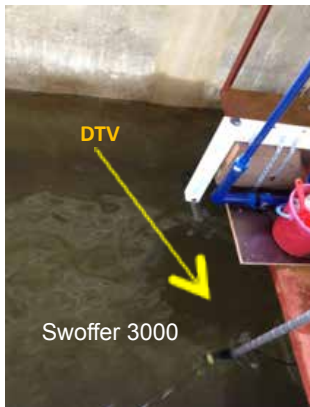


Figure 5: Arrangement of DTV and Swoffer 3000 current meter at Flume

Three series of flow run were conducted at different constant velocities. The data of calibration curve of three series of run was analyzed on the basis of analytic approximation curve as described in Section 2.2 above. The three run of calibration data was combined to check the overall performance of the Shre DTCV.

Results and Discussions

The behavior and performance of the Shre DTCV are interpreted in the following graphs.

Analysis of calibration curves:

From the experiment, the variations in the value of velocities were observed. The graph was drawn for the variations in the value of velocities,

v against the tube tilt inside the water, rad. The graphs of runs no. 1-3 (Figures 6, 7, 8) show that the tube continuously tilts from 0-0.7 radian as velocities increase

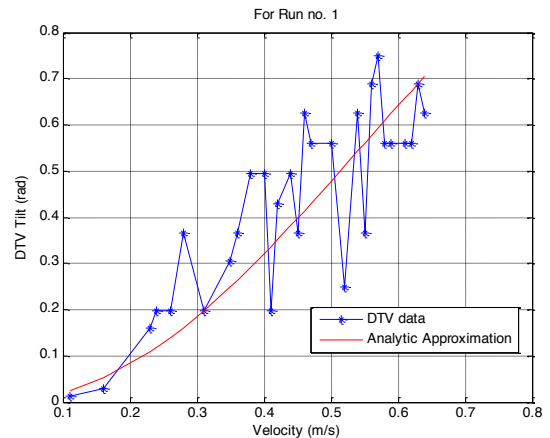


Figure 6: Graph for Run no. 1

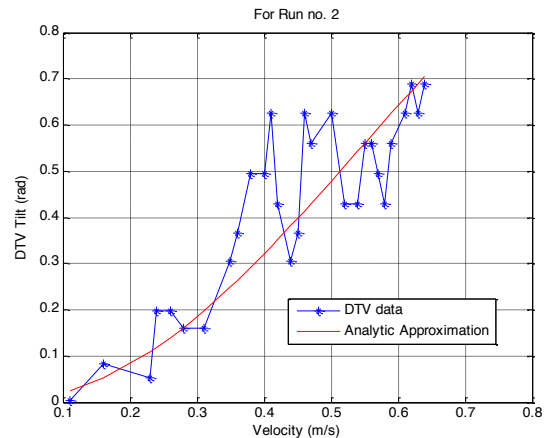


Figure 7: Graph for Run no. 2

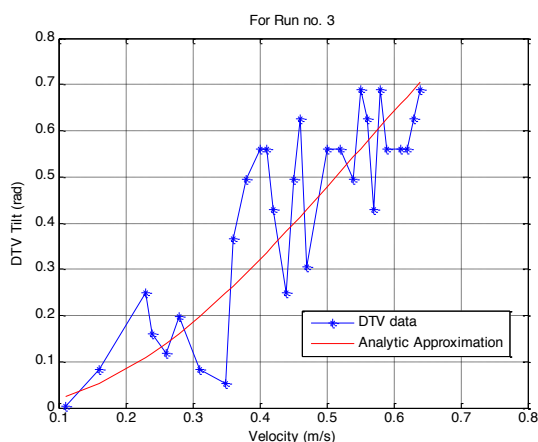


Figure 8: Graph for Run no. 3

As per Table 3 below, the fluctuations of radian tilt show that the tube tilting is not 100% following the preset value of analytic approximation curve as there are random and systematic type of errors during the calibration process. Only 62.36% of calibration data fitted the analytic curve for test run 1. While for test run 2 it is 68.99% and test run 3 is 67.80%. Standard deviation for run 1 and 2 is similar which is 0.041, while run 3 is 0.047.

Table 3: Details of Calibration Results

Test Run	Standard deviation	R-Squared
No. 1	0.041	62.36%
No. 2	0.041	68.99%
No. 3	0.047	67.80%

Analysis of combining all calibration curves:

All the three sets of the calibration data were combined to discover more information when the data was fitted into the analytic curve. The graph is as shown at Figure 9 and the details calibration results as per Table 4 below. It

was found that the R-squared and standard deviations were not much change compared to the individual run test with 66.45% and 0.042 values each. Points that fall off the trend lines decreased the R-squared value.

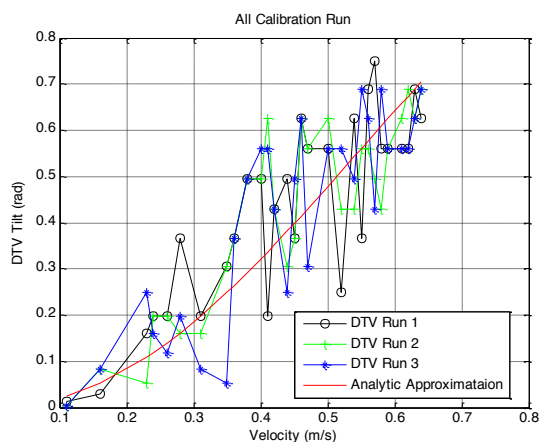


Figure 9: Graph for Combination Run

Table 3: Details of Calibration Results

Test Run	Standard deviation	R-Squared
Combine Test Run	0.042	66.45%

The low accuracy points mean that the calibration process and procedure is less accurate or not properly done. It happened consistently at all three sets of calibration data and exists either at low flow or high flow region. The increase of error may be associated with the procedure or the instrument itself.

Conclusions

The root cause of wide spread variations on the data calibration values can be the instrument configuration systems and calibration process. Thus, a detail configuration of system needs to be executed on the Shre DTCV in order to increase

the accuracy of the instrument. Furthermore, a proper calibration technique and procedures can be conducted at an accredited lab and performed by skilled control system technician where their equipment, standard and procedure are traceable to a national or international standard. Calibration or zeroing a flow meter instrument in flow meter measurement is critical and should be done properly to avoid a systematic offset introduced into the system. The technique of using the drag-tilt principle in measuring water velocity is still reliable. Their low cost, rugged, attractive and user friendly system require much more effort on research in order to increase the accuracy of the instrument. Analytic approximation can be made as a guideline in designing the instrument. At the same time, introducing an artificial intelligence technique or modeling can be beneficial to increase the instrument's accuracy. As there is no instrument which has 100% accuracy, as yet the drag-tilt instrument can help researchers to get spatial dense of river study at a very low cost.

Acknowledgement

The authors would like to thank KETTHA and JKR Malaysia for supporting under grant number R.J130000.7309.4B161. NAHRIM, on the facilities of calibration process and Mohamad Syakir for helping with manufacturing of the instrument.

Reference

- B.Kjerve. (1982, December). Calibration of Estuarine Current Crosses. *Estuarine Coastal and Shelf Science*, 15, 553-559.
- JKR Malaysia and UNIMAS Sarawak. (2010). *JARIMAS Turbine Project, Energy Mapping Study at Schools along Sungai Balleh, Kapit, Sarawak*.
- Johnstone, D. (2008). *Current Meter Calibration Equations: A review of the adequacy of limited-range equations for New Zealand requirements*. Christchurch, New Zealand: National Institute of Water & Atmospheric Research Ltd.
- J.Taylor. (1997). *Error Analysis (2nd Edition)*. University Science Book.
- Mohd Fauzi Othman, a. K. (2012). *Wireless Sensor Network Applications: A Study in Environment Monitoring System*. *Procedia Engineering*, 41, 1204-1210.
- Monalisa Malik, A. (2014). *Study on Drag Coefficient for the Flow Past a Cylinder*. *International Journal of Civil Engineering Research*, 5(4), 301-306.
- Ria Sood, M. K. (2013). *Design and Development of Automatic Water Flow Meter*. *International Journal of Computer Science, Engineering And Applications (IJCSEA)*, Vol.3, No.3.
- R.Marchant, T.Stevens, S.Choukroun, G.Coombes, M.Santarossa, J.Whinney and P.Ridd. (2014, January). *Peer-Reviewed. Technical Communication - A Buoyant Tethered Sphere for Marine Current Estimation*. *IEEE Journal of Oceanic Engineering*, 39(No. 1).
- Streeter, V. (1961). *Handbook of Fluid Dynamics (1961 ed.)*. New York: McGraw-Hill.
- Strupstad, A. (2009). *Error Analysis of Flow Experiments*. Trondheim: Norwegian University of Science and Technology, Department of Petroleum Engineering and Applied Geophysics.

Contents

PHYSICAL HYDRODYNAMIC MODELLING STUDY FOR WAVE INTERACTION ONTO
SUBMERGED BREAKWATER

Safari Mat Desa, Othman Karim & Kamarul Huda Samion

PUMP SUMP MODEL TESTING FOR SUNGAI BELIBIS PUMP STATION

Mohd Kamarul Huda Samion, Mohd Fauzi Mohamad & Khairul Azuan Adam

EXPERIMENTAL STUDY ON PROPELLER PERFORMANCE OF A VESSEL NEAR BANK AND IN
SHALLOW WATER

Mehdi Nakisa, Fatemeh Behrouzi, Adi Maimun & Mohamed Yasser Ahmed

COST-EFFECTIVE, LAB-SCALE AUTOMATED SPATIAL AND TEMPORAL MEASUREMENT
SYSTEM FOR LOCAL SCOUR AROUND BRIDGE PIERS

Mojtaba Porhemmat & Wan Hanna Melini Wan Mohtar

STUDY OF FLOW CHARACTERISTICS IN MENGKUBAU DAM SPILLWAY

Che Mohamad Amirus Shafiq Che Ismail, Mohd Remy Rozainy Mohd Arif Zainol,
Ismail Abustan, Mohd Zulkifli Abdullah & Nuridah Sabtu

OPTIMUM POSITIONING OF BUFFLE BLOCKS IN STILLING BASIN FOR MENGKUBAU DAM

Ismail Abustan, Nuridah Sabtu, Mohd Remy Rozainy Mohd Arif Zainol,
Mohammad Khairy A. Wahab & Mohd Azmi Ismail

AN EXPERIMENTAL ASSESSMENT OF BOAT-GENERATED WAVE BY LANGKAWI GEOFOREST
KARST TOURIST BOATS

Omar Yaakob, Faizul Amri Adnan, Mohd Arif Ismail, Mohamad Hidayat Jamal, Ahmad Hadi
Mohamad Rashidi, Wan Ahmad Hafiz Wan Mohd Azhary, Mohd Kamarul Huda Samion,
Andiknurdiana Bachok, Badruzzaman Ahmad & Nur Shafira Ahmad

CALIBRATION OF A DRAG TILT CURRENT VELOCIMETER FOR AN OPEN CHANNEL
APPLICATION

Ibnu Akil Maitar Muharram, Zool Hilmi Ismail & Kamarul Bahrin Mohamad
