



Study of Flow in the Test Section of a Circulating Water Channel by Varying the Honey Comb Cross-section

Kiran George Varghese, Nair Vinay Gopi, Avinash Godey, and Prof. (Dr.) Sunil Kumar PG

Abstract

The Circulation Water Channel (CWC) is a research facility available at Indian Maritime University, Visakhapatnam Campus. The investigation of a flow, in different configurations of the CWC set up, for obtaining a required flow velocity in the test section, is complex. To study the flow, a physical model of the CWC, with different configurations, should be made, which in overall is a complicated and time-consuming exercise. But this difficulty can be overcome through using Computational Fluid Dynamics (CFD) analysis, as in this study, where a CFD analysis is done using 'STAR-CCM+' software.

A CFD model of the existing CWC [corresponding to the 1:4 scale setup at IMUV], is first made, and its validity is checked, by comparing the results of the CFD analysis, against those results obtained from the experimental analysis.

The test section is the area in the CWC, or its CFD model, where the experimental activities are carried out. It is important to have a smooth streamlined flow in the test section, and to ensure this, a honeycomb structure is placed such that the flow enters the test section through the structure.

The optimum honeycomb structure, which produces a smooth flow in the test section of a CWC, is found out, by conducting analyses for different shapes - i.e. for shapes ranging from rectangular to hexagonal and also for different inlet velocities

Keywords —Circulating Water Channel (CWC), Computational Fluid Dynamics (CFD), Vanes, Honeycomb Section, Test Section.

INTRODUCTION

The CWC can be used for various hydrodynamic research activities such as flow around ship, Resistance Experiment, Wake Analysis, Trim measurements etc., by generating a controlled flow environment. [10]. The study was based on the 1:4 scale Circulation Water Channel (CWC) situated Indian Maritime University Visakhapatnam Campus (IMUV).

THE IMUV CIRCULATING WATER CHANNEL

The Test section is of 2 x 0.5 x 0.5 m in dimension, Velocity of water flowing through the test section is varied by varying the impeller rpm, to provide uniform flow to the test section. A Honeycomb structure, Guide vanes, Contraction section, Transition sections, Bend sections, Diffuser are present. The function of the honeycomb, which consists of a set of closely spaced horizontal and vertical plates, is to straighten the flow and remove any cross flow and rotational velocity components [3]. The function of the guide vanes is to turn the flow through 90 degrees with minimum variation in velocity across the width of the flow and with minimum energy loss. [3]. An optimally designed CWC will have a contraction section at the inlet to the test section [2]. However, the IMUV CWC there is an accessories section between the test section and the contraction section which consist of the trough with increased breadth and depth to accommodate the accessories [3], which must be flush with the test section by suitable plates, the plates which are flushed are not properly fixed resulting in reduced beam at the accessories section and a sudden increase in beam to the test section.[1].

Table 0-1 Dimensions Of CWC

Item	Dimension
Test Section Length	2.25m
Test Section Width	0.5m

Test Section Depth	0.5m
Water Speed	0-1.5m/s
Impeller rpm	0-2500rpm
Impeller Diameter	240mm
Ship Model Length	250mm
Ship Model Beam	25mm
Ship Model Draught	25mm

Parts

The various parts of a circulating water channel are:

1. Test Section
2. Sand trap for sedimentation studies
3. First bend
4. Transition section (square to circular)
5. Second bend
6. Impeller section
7. Adjustment piece
8. Transition section (circular to square)
9. Diffuser (expansion section)
10. Third bend
11. Transverse (or vertical) section
12. Fourth bend
13. Honeycomb
14. Contraction section
15. Accessories section (wave maker, flow regulator, current generator etc.). [3]

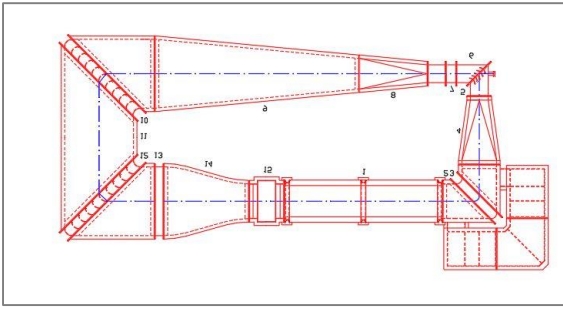


Figure 0-1 Circulating Water Channel Plan view

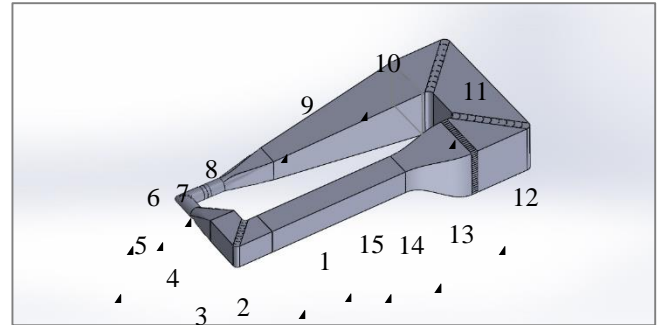


Figure 0-1 CWC Model

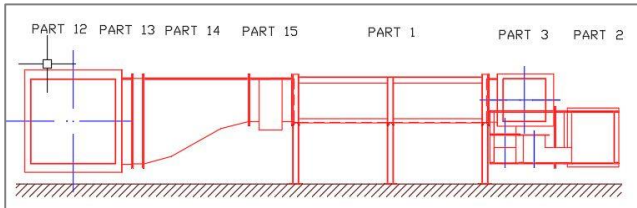


Figure 0-2 Circulating Water Channel Elevation View

PROBLEM FORMULATION

This numerical simulation work was carried out in STAR CCM. The basic steps are:

- Problem specification and geometry preparation.
 - It involves the specification of the problem, flow conditions which are requirements for the simulation.
 - In this study a part of CWC was modelled and with different honeycomb cryosections varying from rectangular, hexagonal and circular, also considering the influence of vanes.

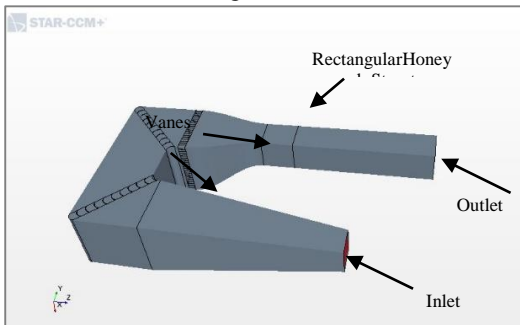


Figure 0-1 CWC Part (Open Loop)

- Selection of governing equations and boundary conditions
 - Specified velocity on inlet
 - A set of inlet velocities ranging from 0.5m/s, 1m/s, 1.5m/s, 2m/s was provided at velocity inlet.

MODELING AND MESHING

Modelling of CWC was done in Star CCM+, for creating the fluid domain, to create the fluid domain, the hollow CWC is made solid and the solid components such as vanes and honey comb were made hollow. [1]

- Selection of gridding strategy and numerical method
 - The grid size was selected from the previous work. [1]

CFD software using STARCCM+ has been used to generate the grid for the mode, so that the flow domain was able to be discretized. Trimmed mesh with a base size of 0.1 meter, with six number of prism layer to the mesh model.

Table 0-1 Mesh Values

GROUP		MESHER	
Surface Meshers		Surface Remesher	
Core Volume Meshers		Trimmed	
NODE	PROPERTY	SETTINGS	
Base Size	Value	0.1 metre	
Target surface	Relative to base	75%	
Minimum surface size	Relative to base	10%	

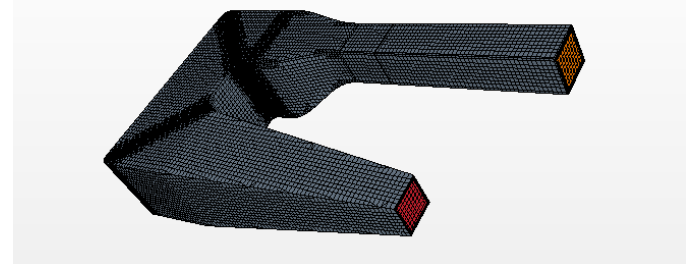


Figure 0-2 CWC Meshed

Table 0-2 Physics Values

Space	Three Dimensional	
Time	Steady	
Material	Liquid	
Flow	Segregated Flow	
Equation Of state	Constant Density	
Viscous Regime	Turbulent	
Reynolds-Averaged	K-Epsilon	
Turbulence	Turbulence	
Optional Models	Gravity	
Node	Property	Setting
Stopping Criteria		



International Symposium on Marine Design and Construction 2019 (SMDC 2019)



Maximum Iterations	Inner	Value	1000
--------------------	-------	-------	------

Material	Equation of state
Liquid	Constant Density

The direction and magnitude of the gravity vector are set using the **Reference Values** node. In this case, a gravity force is applied in the negative y-direction.

VALIDATION

The validation was done by comparing the flow pattern of the CFD model with the real-life scenario. For the purpose of validation our previous work was referred [1].



Figure 0-1 Disturbance in Actual Real Life CWC

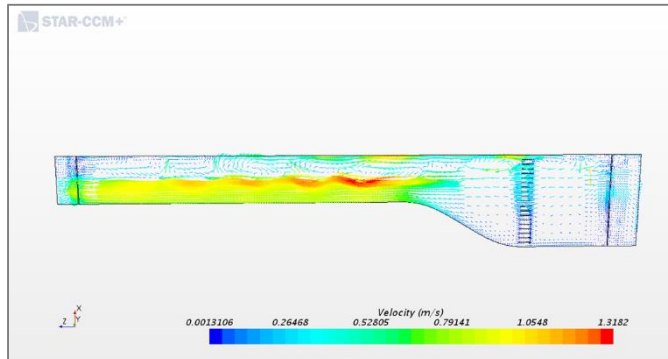


Figure 0-2(Disturbance Pattern from CFD Result (Star CCM+))

The disturbance created in the test section of the real life CWC was captured in the CFD model.

For a better and credible validation, velocity was measured at a particular region in the test section of the real life CWC, for an impeller rpm of 700. The velocity at the same location was measured in the CFD model, for an input rpm of 700. These two velocities were compared.

$$\text{Velocity VCFD} = 1.19 \text{ m/s.}$$

The average velocity measured in test section of the actual CWC at IMUV, using a pitot tube is:

$$\text{Velocity VEXP} = 1.01449 \text{ m/s.}$$

The difference in velocity was comparable and accounted to a velocity difference of 0.17 m/s.

VI. RESULTS

The flows were compared by varying the honey comb cross

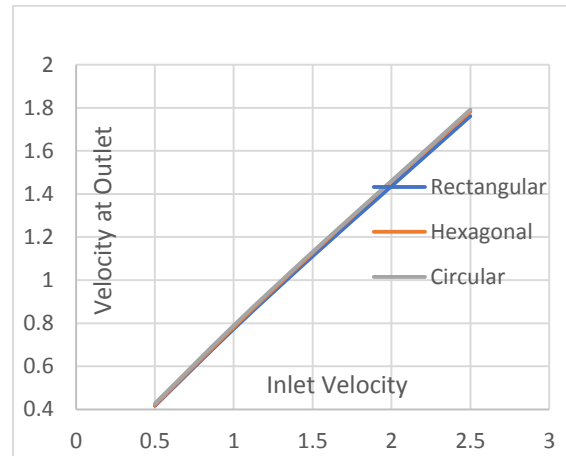
sectional shape, from rectangular to hexagonal, and finally to circular. The velocity in the test section was used as one of the parameters, to compare the shapes.

The velocity of the circular shape was found to be the highest for all inlet velocities varying from 0.5 to 2.5 m/s, at an interval of 0.5 m/s.

Table 0-1 Velocity Comparison

Input Velocity (m/s)	Output Velocity(m/s)		
	Rectangular	Circular	Hexagonal
0.5	0.416	0.427	0.419
1	0.773	0.789	0.781
1.5	1.110	1.131	1.126
2	1.436	1.460	1.459
2.5	1.761	1.791	1.781

Error! Not a valid link.



GRAPH 0I-1 Comparison of Velocities

The comparison was also done for the flow pattern attained in the test section.

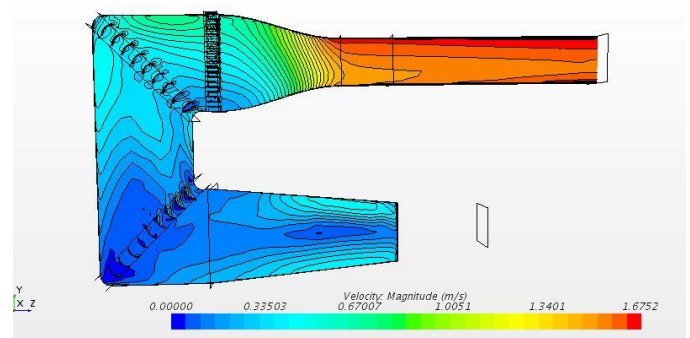


Figure 0I-0-3: Rectangular Honeycomb

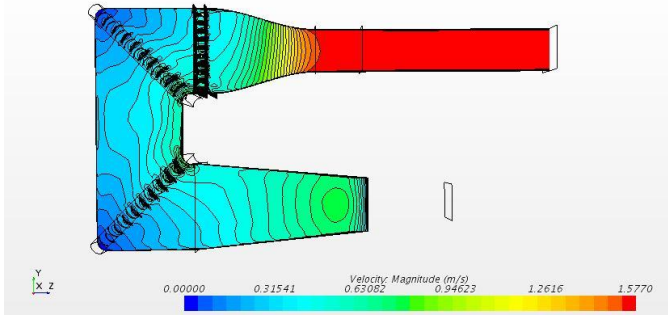


Figure 0I-0-4: Hexagonal Honeycomb

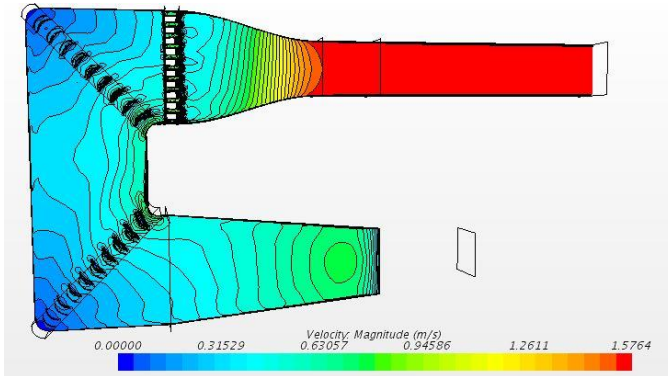


Figure 0I-0-5: Circular Honeycomb

The comparison was done using stream line flow pattern.

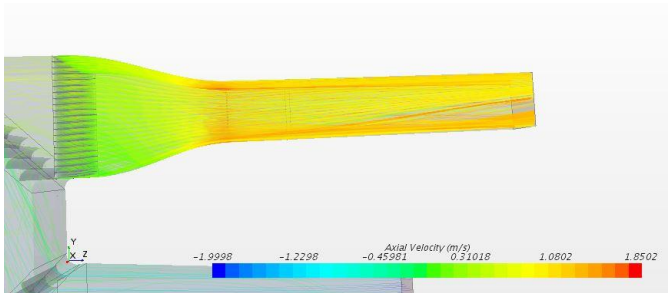


Figure 0I-0-6 Rectangular Honeycomb Stream Line Pattern

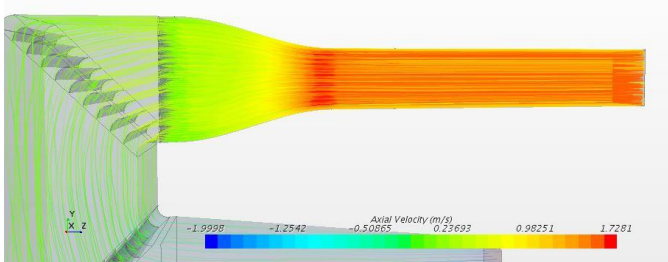


Figure 0I-0-7 Hexagonal Honeycomb Stream Line Pattern

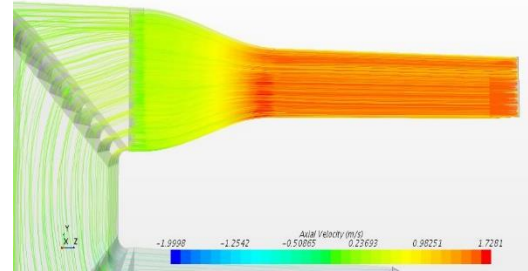


Figure 0I-0-8 Circular Honeycomb Stream Line Pattern

CONCLUSION

The flow pattern and the stream line flow, obtained for the circular and hexagonal sections, are better and smooth, compared to the rectangular shape.

For a given input velocity, the maximum output velocity can be achieved for a circular shape, honeycomb cross section. That is, for a low input velocity, a higher output velocity can be achieved in the test section, by using a circular shape. Thus, we can infer that, in a closed loop system, a higher velocity can be achieved in the test section, for a given input impeller rpm, which would help us improve the workability of the CWC.

Though the hexagonal and circular cross sections, both deliver nearly the same results, we may say that the circular section holds an upper footing.

From a construction point of view, a circular honeycomb shape can be achieved by the stacking of pipes.

SCOPE FOR FUTURE WORK

1. In the present study, analysis was done by changing the honey comb shape, keeping the area of fluid domain constant. The area was selected from the real life CWC. Future study ought also include the optimization of area for different shapes.
2. Future analysis ought include the partially filled condition too.
3. In future studies, the closed loop analysis for the circular CWC ought be carried out against different impeller input rpms.
4. Closed loop study ought to be carried out, by varying the type of impeller, and its influence in the flow in the test section.

ACKNOWLEDGMENT

"Teach me, my God and King, / In all things Thee to see, /
And what I do in anything / To do it as for Thee."

-The Elixir, George Herbert



International Symposium on Marine Design and Construction 2019 (SMDC 2019)



REFERENCES

- [1] Kiran George Varghese, Nair Vinay Gopi and Avinash Godey *CFD as a tool to validate and modify the flow in the test section of a Circulating Water Channel*
- [2] M.G. Pullinger and J.E. Sargison; *Using CFD to improve the design of a circulating water channel.*
- [3] School of Maritime Design and Research Indian Maritime University Visakhapatnam Campus, Study Project on *Flow Around Ships in a Hydrodynamic Test Facility*
- [4] Dr. R. K. Bansal; *Fluid Mechanics and Hydraulic Machines*
- [5] J.P. Ghose & R.P. Gokarn; '*Basic ship propulsion*'.
- [6] Edward.V.Lewis; *Principles of Naval Architecture Vol II.*
- [7] Star CCM+ User Guide.
- [8] H.K.Versteeg&W.Malalasekera; *An Introduction to Computational Fluid Dynamics The Finite Volume*
- [9] Pradeep JS Bhavaraju; *Holistic Approach to PROJECT CWC with ALTAIR.*
- [10] Pradeep JS Bhavaraju; *Qualitative and Quantitative Analysis in Comparing Prospective CWCs.*
- [11] Gustavo Roque da Silva Ássi, Julio Romano Meneghini, José Augusto Penteado Aranha, Walter George P. Coleto; *Design, Assembling and Verification of a Circulating Water Channel Facility for Fluid Dynamics Experiments.*
- [12] Captain Harold E.Saunders , Clyde W. Hubbard ;*The Circulating Water Channel of the David W. Taylor Model Basin.*

1 AUTHORS BIOGRAPHY

Avinash Godey is with the Indian Maritime University Visakhapatnam Campus, Visakhapatnam, Andhra Pradesh-530011, India. He is an Assistant professor in the Department of Naval Architecture and Ocean Engineering. He can be contacted at avinashgodey@gmail.com, or at his phone number at **+91 9490489010**.

Prof. (Dr.) Sunil Kumar PG is the Dean (Research) at Sree Narayana Gurukulam College of Engineering, Kolenchery, Kerala- 682311, India. An alumnus of the Cochin University of Science and Technology, IIT Delhi and University College London, he can be contacted at snlkmrpg@gmail.com, or at his phone number at **+91 8281815134**.

Nair Vinay Gopi was with the Indian Maritime University Visakhapatnam Campus. He too is an alumnus of the 2016-'18 M. Tech Naval Architecture and Ocean Engineering Batch. He is now a Junior Naval Architect at XShip Design and Analytics Pvt Ltd, South Kalamassery, Ernakulam, Kerala- 682022, India. He can be contacted at vinay.nair17@gmail.com, or at his phone number at **+91 955877085**.

Kiran George Varghese was with the Indian Maritime University Visakhapatnam Campus. He is an alumnus of the 2016-18 M. Tech Naval Architecture and Ocean Engineering Batch. He is now an Assistant Professor at with the Department of Naval Architecture and Ship Building at Sree Narayana Gurukulam College of Engineering, Kolenchery, Kerala- 682311, India. He is he corresponding author of this paper and can contacted either at his mail address at kirangeorgevarghese@gmail.com, or at his phone number at **+91 6282592822**.