Journal of Naval Architecture and Marine Engineering
December, 2021
http://dx.doi.org/10.3329/jname.v18i2.45982 http://www.banglajol.info

A STUDY FOR VALIDATING, RECTIFYING AND OPTIMIZING THE FLOW IN THE TEST SECTION OF A CIRCULATING WATER CHANNEL

Kiran George Varghese[1], Vinay Gopi Nair[2], Avinash Godey[3] and P. G. Sunil Kumar[4]
[1]Assistant Professor, Department of Naval Architecture and Ship Building, SNG College of Engineering, Kolenchery, Affiliated to APJ Abdul Kalam Technological University, Kerala, India. Email: kirangeorge93@gmail.com
[2]Junior Naval Architect, XShip Design and Analytics Pvt. Ltd., Kerala, India. Email: vinay.nair17@gmail.com
[3]Assistant Professor, Department of Naval Architecture and Ocean Engineering, Indian Maritime University, Visakhapatnam, Andhra Pradesh, India. Email: avinashgodey@gmail.com
[4] Professor and Dean (Research), Department of Naval Architecture and Ship Building, SNG College of Engineering, Kolenchery, Affiliated to APJ Abdul Kalam Technological University, Kerala, India. Email: snlkmrpg@gmail.com

Abstract:

The Circulation Water Channel (CWC) is an experimental facility available at Indian Maritime University, Visakhapatnam Campus. A study for comparing the flow pattern and velocity in the test section, for different configurations of the CWC, 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.

On successfully validating the results, modifications are suggested for rectifying the disturbance which is present in the test section. The test section is the area in the CWC where experimental activities are carried out. In order to carry out the experiments with a certain degree of accuracy, it is important to have a smooth streamlined flow in the test section. To ensure this, a honeycomb structure is positioned such that the flow enters the test section through the honeycomb, which streamlines the flow.

On successfully rectifying the disturbance, studies are carried out to improve the streamlined flow in the test section, for which, different configurations of honeycomb structure are studied. 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 circular, against different inlet velocities.

The present paper sums up the findings of our earlier work which were presented in conferences at IIT Madras and IMU Visakhapatnam respectively.

Keywords: Circulating water channel (cwc), computational fluid dynamics (CFD), vanes, honeycomb section, test section

1. Introduction

Experimental facilities such towing tanks and water channels are used in research laboratories, to study the various flow behavior of floating and submerged bodies. These experiential facilities are equipped with various instrumentations for flow visualizations and measurements. The Circulation Water Channel (CWC) can be used for various hydrodynamic research activities such as studying the flow pattern around ships, conducting resistance experiments, wake analyses, trim measurements etc., by generating a controlled flow environment (Bhavaraju, 2015). The above experiments can also be performed in a towing tank, but the CWC has an advantage that the flow conditions in the test section can be maintained for a very long period of time. Also, the flow visualization can be performed better on a CWC, compared to a towing tank. (Ássi, 2005; Chen, 2006; Santos, 2015).

1813-8535 (Print), 2070-8998 (Online) Received on: March 2020

The present study is based on the 1:4 scale CWC, situated at Indian Maritime University, Visakhapatnam Campus (IMUV).

A CFD analysis is performed as part of the study, in order to improve the flow in the test section of the CWC at IMUV. A CFD model of the 1:4 scale CWC is first made and validation is carried out by comparing the experimental and CFD results. After validation, corrections and modifications in design that would help improve the flow are suggested, based on the results of the analysis.

2. The IMUV Circulating Water Channel

The test section is of 2 x 0.5 x 0.5 m in dimension. It is made of glass for the purpose of flow visualization [Alho, 2010]. The 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, contraction section, diffuser, guide vanes, and transition and bend sections are present. The function of the honeycomb, which consists of a set of closely spaced horizontal and vertical plates, is to remove any cross-flow and rotational velocity components. 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, with minimal energy loss. An optimally designed CWC will have a contraction section at the inlet to the test section (Pullinger, 2007). However, in the IMUV CWC, there is an accessories section between the test section and the contraction section (Ramesh, 2011). The accessory section is provided with an increased beam to accommodate any instruments for flow measurement, but it has not been flushed properly with a glass plate, as it ought to have been when no instruments are present (Ramesh, 2011; Godey, 2018).

Table 2.1: Dimensions of CWC (Ramesh, 2011)

Item	Dimension
Test Section Length	2.25 m
Test Section Width	0.5 m
Test Section Depth	0.5 m
Water Speed	0 - 1.5 m/s
Impeller rpm	0 - 2500 rpm
Impeller Diameter	240 mm
Ship Model Length	250 mm
Ship Model Beam	25 mm
Ship Model Draught	25 mm

2.1 Parts

The various parts of the CWC 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.)

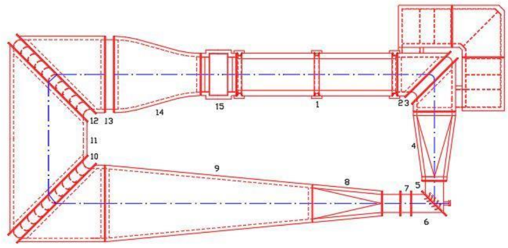

Figure 2.1.1: CWC (Plan View)

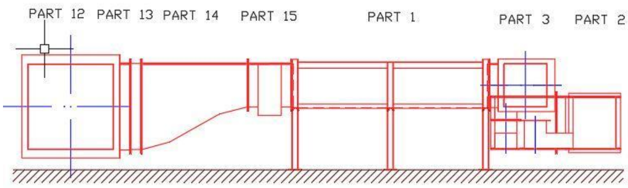

Figure 2.1.2: CWC (Elevation View)

3. Problem Formulation

This numerical simulation work is carried out in STAR-CCM+. The basic steps are:

i. Problem specification and geometry preparation.
   - It involves the specification of the problem and flow conditions, required for simulation.
   - A closed-loop CWC, similar to real life, is modeled and this is used in the study for validation and rectification.

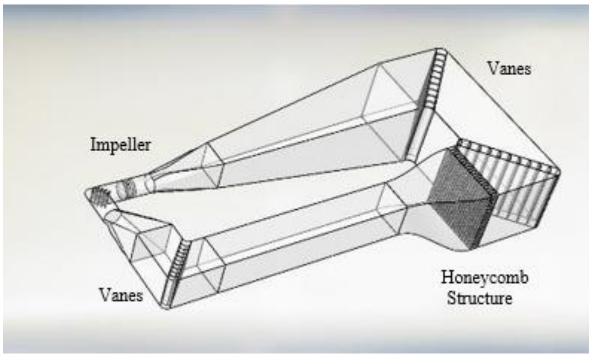

Figure 3.1: Closed Loop CWC with Vanes, Honeycomb and Impeller

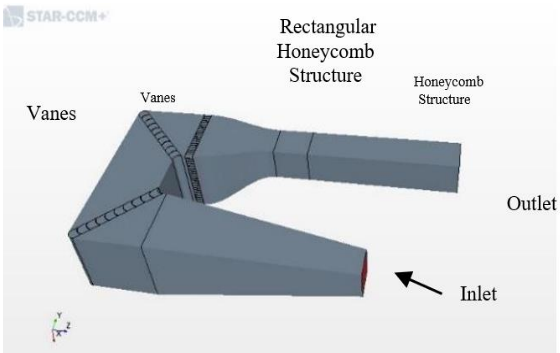

Figure 3.2: CWC Part (Open Loop)

   - For the study on optimization of CWC, a part of CWC is studied (for ease of computation). The study is focused on optimizing the honeycomb shape, for which different honeycomb cross-sections, varying from rectangular to hexagonal and circular, are considered.

ii. Selection of boundary conditions.
   - In case of a closed-loop CWC with impeller, an rpm of 700 is given as the input.
     - The level of water in the test section is initially set at 25 cm.
     - The actual real-life CWC is provided with the exact same conditions.
   - In the case of a CWC part, the velocity at the inlet is specified as follows:
     - A set of inlet velocities having values 0.5 m/s, 1 m/s, 1.5 m/s, and 2 m/s is provided at the inlet.

4. Modeling and Meshing

The modeling in CFD analysis is to be carried out in the fluid domain. The solid obstacles in the path of the flow such as the impeller, the honeycomb structure, and the vanes need to be eliminated before analysis is carried out for the water that flows through the CWC.

The actual geometry of the honeycomb structure is as in the left side of Figure 4.1 below (It is but for visualization purposes only). The right side represents the fluid domain used in the analysis. The fluid domain behaves as in the real-life scenario during the analysis.

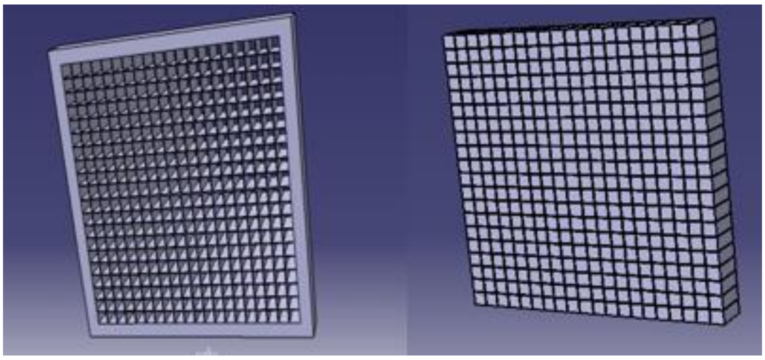

Figure 4.1: Honeycomb structure

The modelling of the CWC is done in CATIA and SOLIDWORKS. To create the fluid domain, the hollow CWC is made solid and the solid components such as honeycomb, impeller and vanes are made hollow (Godey, 2018).

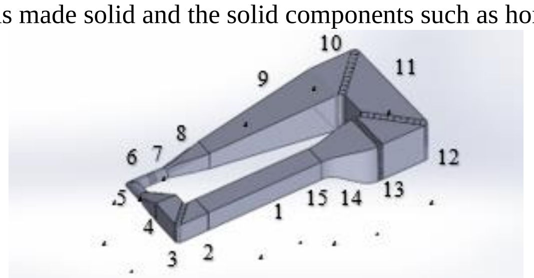

Figure 4.2: CWC Model

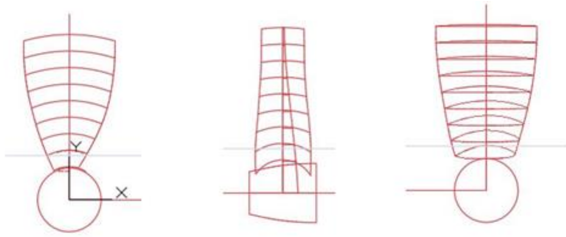

Figure 4.3: Impeller Drawings

The modelling of the Ka-Series impeller (as in the real life CWC at IMUV) is done in SOLIDWORKS and AutoCAD 2018. To model the blades, expanded blade sections are obtained from Ka Series Table (Carlton, 2007). A two-dimensional drawing is made in AutoCAD. The developed blade sections are obtained by rotating through their respective pitch angles, and the rotated sections are projected onto right circular cylinders of respective radii (Godey, 2018).

Selection of gridding strategy and numerical method:

- The grid size is selected from the previous work *CFD as a Tool to Validate and Modify the Flow in the Test Section of a Circulating Water Channel*.

STAR-CCM+ has been used to generate the grid for the model.

4.1. Mesh Setting for Closed Loop CWC

Table 4.1.1: Mesh Values (Godey, 2018)

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 %
Volumetric Control	Parts	Free surface, parts near to impeller
Controls > Surface Trimmer	Custom isotropic size	Enabled
Values > Custom size	Size type	Relative to base
	Percentage of base	10

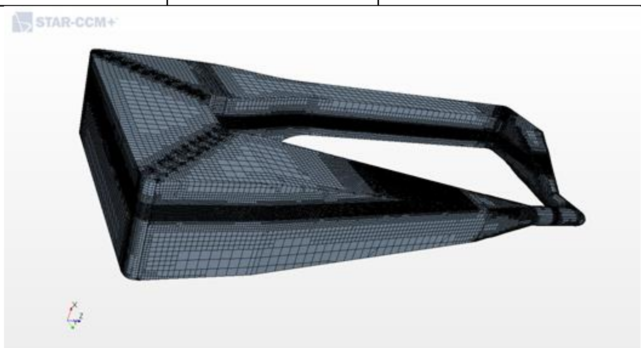

Figure 4.1.1: CWC Meshed with Dense Mesh at Air Water Interface Level

Table 4.1.2: Physics Values (Godey, 2018)

Space	Three Dimensional	
Time	Implicit Unsteady	
Material	Eulerian Multiphase	
Eulerian Multiphase Model	Volume of Fluid	
Flow	Segregated Flow	
Equation Of state	Constant Density	
Viscous Regime	Turbulent	
Reynolds-Averaged Turbulence	K-Epsilon Turbulence	
Optional Models	Gravity	
NODE	**PROPERTY**	**SETTING**
Implicit Unsteady	Time-step	0.01 s
Segregated flow > Velocity	Under Relaxation Factor	0.5
Segregated flow > Pressure	Under Relaxation Factor	0.5
Stopping Criteria		
Maximum Physical Time	Value	10 s
Maximum Inner Iterations	Value	5
Property	Setting	
Axis Direction	[0, 0, 1]	
Rotation Rate	700 rpm	

As there are two fluids, the materials corresponding to each, against 'Eulerian phases' nodes, must be defined (Siemens Digital Industries Software, 2017).

To define the phases, and set the material properties, two phases are generated by:

- Selecting physics continuum – “Models > Eulerian Multiphase > Eulerian Phases” node

Two phases are thus created and are named as water and air respectively.

The following models are used to define the two fluids:

Table 4.1.3: Phase Definition (Godey, 2018)

Material	Equation of state
Liquid	Constant Density
Gas	Constant Density

Two field functions are defined, in order to define the level of water and air. This is done by creating two scalar field functions.

- **“**Tools > Field Functions” node and two field functions (“New > Scalar**”)** are created from the pop-up menu and named water level and air level respectively.

Select the field node (water level and air level) and set the following properties:

Table 4.1.4: Definition of Water Level (Godey, 2018)

Property	Settings
Function Name	Water Level
Definition	($$Position <=0.25)? 1:0
Function Name	Air Level
Definition	($$Position >0.25)? 1:0

Node	Property	Settings
Volume Fraction	Method	Composite
Composite > Water	Method	Field Function
	Scalar Function	Water Level
Composite > Air	Method	Field Function
	Scalar Function	Air Level

4.2. Mesh Setting for Open-loop CWC

Table 4.2.1: Mesh Values (Varghese, 2019)

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 %

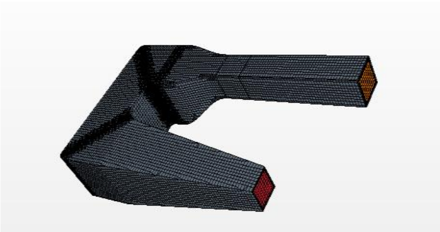

Figure 4.2.1: CWC Part Meshed

Table 4.2.2: Physics Values (Varghese, 2019)

Space	Three Dimensional	
Time	Steady	
Material	Liquid	
Flow	Segregated Flow	
Equation Of state	Constant Density	
Viscous Regime	Turbulent	
Reynolds-Averaged Turbulence	K-Epsilon Turbulence	
Optional Models	Gravity	
NODE	**PROPERTY**	**SETTING**
Stopping Criteria		
Maximum Inner Iterations	Value	1000

Material	Equation of state
Liquid	Constant Density

The direction and magnitude of the gravity vector are set using the 'Reference Values' node. For both cases, a gravity force is applied in the negative y-direction (gravity acting downwards, as in real life scenarios).

5. Validation

The validation is done by comparing the flow pattern of the CFD model with the real-life scenario. For the purpose of validation, our previous work, *CFD as a Tool to Validate and Modify the Flow in the Test Section of a Circulating Water Channel*, is referred.

Figure 5.1: Disturbance in Actual Real Life CWC

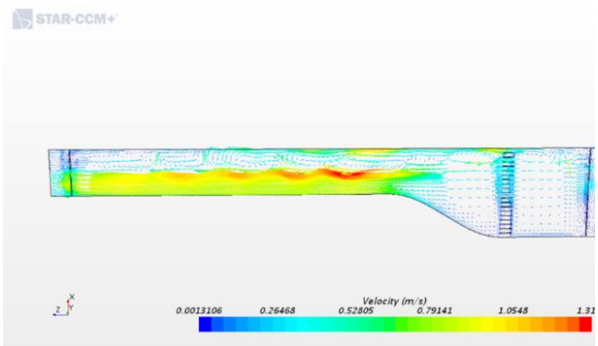

Figure 5.2: Disturbance Pattern from CFD Result (STAR-CCM+)

The disturbance created in the test section of the real life CWC is captured in the CFD model.
For a better and credible validation, velocity is measured at a particular region, in the test section of the real life CWC, against an impeller rpm of 700. The velocity at the same locations are measured in the CFD model, against an input rpm of 700, using velocity probes.

Pitot tubes are used to measure the velocity of flow in the test section of the CWC, corresponding to particular values of impeller rpm. It is based on the principle that if velocity of flow at a point becomes zero, the pressure there is increased due to the conversion of kinetic energy into pressure energy (Bansal, 2010).

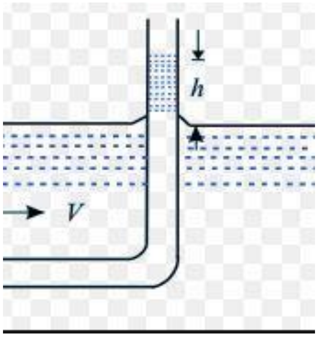

Figure 5.3: Diagrammatic Representation of Pitot Tube

Velocity of flow at any point V = $Cv\sqrt{2gh}$

Where, Cv: Coefficient of discharge, g: Acceleration due to gravity, h: Rise of water level

In its simplest form, a pitot tube consists of a glass tube, bend at right angles. A simple pitot tube is made by bending a simple straight glass tube into an 'L' shaped tube, with the help of flames from a burner. The 'L' shaped tube is then attached to a wooden piece, by making a groove in it and the tube is fixed using glue. The value of Cv is taken as unity.

Table 5.1: Velocity measured at Different Locations in the Test Section of the CWC, Using a Pitot Tube

The different pitot tube locations for velocity measurement			Rise of water level	Velocity measured
x	y	z	h	V_{EXP}
cm	cm	cm	m	m/s
85	10	20	0.047	0.960
85	15	20	0.049	0.980
85	20	20	0.049	0.980
85	25	20	0.049	0.980
85	10	15	0.052	1.010
85	15	15	0.054	1.029
85	20	15	0.054	1.029
85	25	15	0.05	0.990
85	10	10	0.055	1.039
85	15	10	0.055	1.039
85	20	10	0.057	1.058
85	25	10	0.048	0.970
85	10	5	0.054	1.029
85	15	5	0.055	1.039
85	20	5	0.056	1.048
85	25	5	0.056	1.048
Average Velocity (V_{EXP})				**1.014**

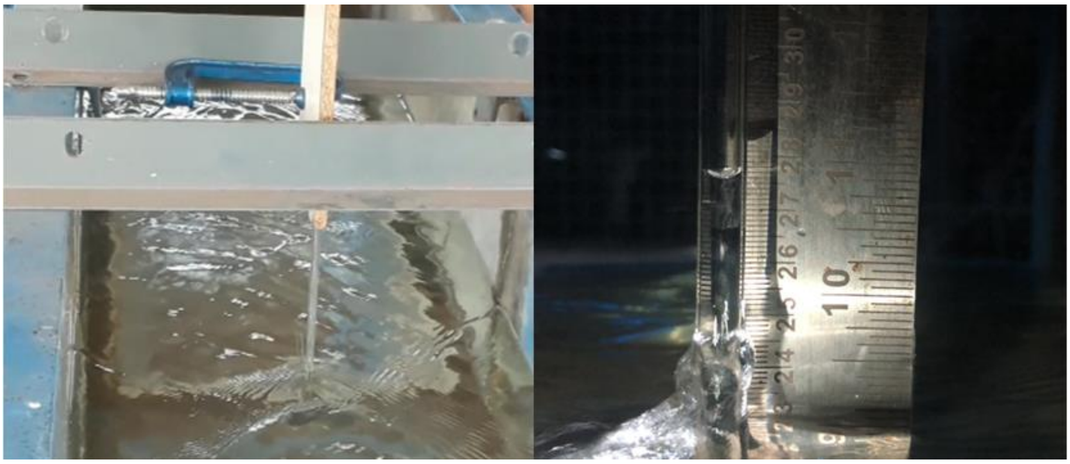

Figure 5.4: Velocity Measurement in the Pitot Tube

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

Velocity V_{EXP} = 1.014 m/s

The velocity obtained at the exact same location, from CFD analysis is:

Velocity V_{CFD} = 1.19 m/s

The difference in velocity is comparable and is accounted to a velocity difference:

$V_{DIFFERENCE}$ = 0.17 m/s

6. Rectification

A disturbance is caused due to improper flushing or positioning of the glass plate in the accessory and test sections. It is discovered that an anomaly that occurred during the fabrication of the CWC setup has led to this. The dimensions of the accessory and test sections of the CWC veer slightly from those recommended in the original part drawings. The accessory section is provided with an increased beam to accommodate any instruments for flow measurement, but it has not been flushed properly with a glass plate (Fig. 6-2), as it ought to have been (Fig. 6-1), when no instruments are present (Ramesh, 2011). The present beam at the test section is 480 mm. It ought to have been 500 mm (Fig. 6-4), as per the part drawings and it is this that leads to the disturbance in the accessory and test sections. This can but be rectified through the proper flushing, or positioning, of the glass plate.

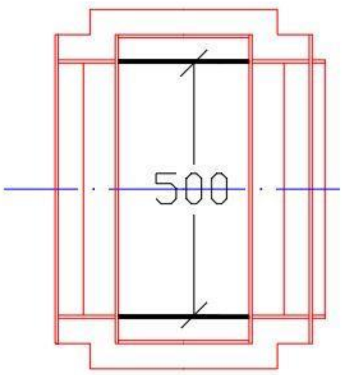

Figure 6.1: Beam, Required at the Accessory Section of the CWC, at IMUV (All dimensions are in mm)

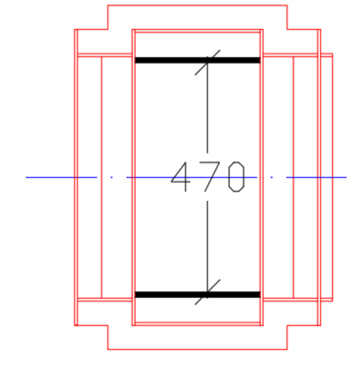

Figure 6.2: Beam, Present at the Accessory Section of the CWC, at IMUV (All dimensions are in mm)

Figure 6.3: Improperly Flushed Glass Plate

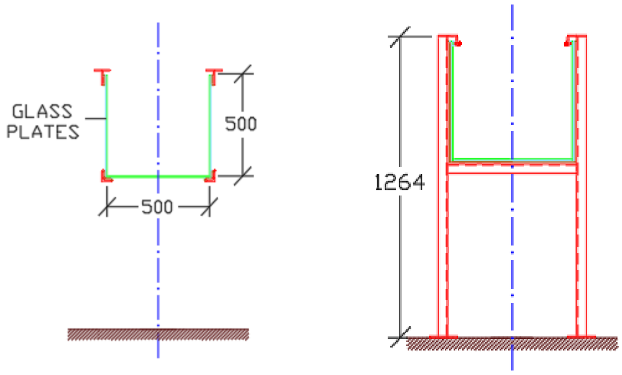

Figure 6.4: Beam, Required at the Test Section in CWC, at IMUV (All dimensions are in mm)

The accessory and test sections are properly modeled in STAR-CCM+ and analyses are carried out. We can see that the disturbances that had been present earlier have been eliminated now, as the sections have been properly flushed (Godey, 2018).

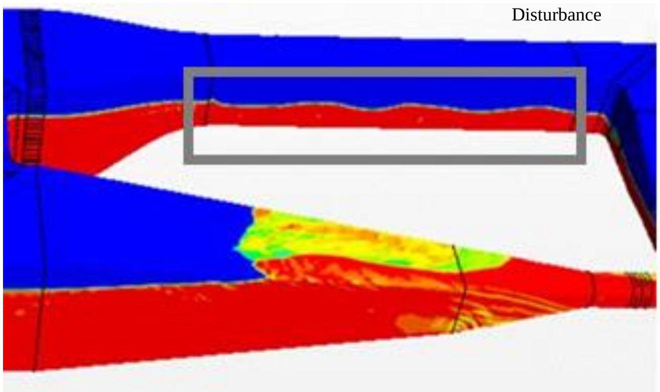

Figure 6.5: Disturbance Pattern [in the Test Section] Due to Improper Flushing of Glass

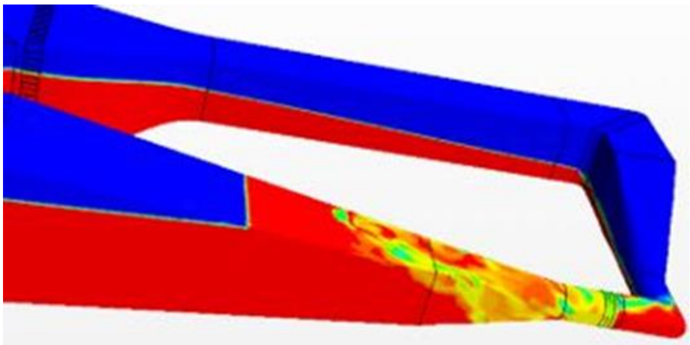

Figure 6.6: Smooth Flow in Test Section Due to Proper Flushing of Glass

7. Optimization

The flows are compared by varying the honeycomb cross-sectional shape, from rectangular to hexagonal, and finally to circular. The dimensions in each case are varied in such a way that the area through which the flow occurs remains constant - that is, the area of flow in the honeycomb section remains constant. In case of circular cross-section, the area of flow of the noncircular region is also taken into account in the study.

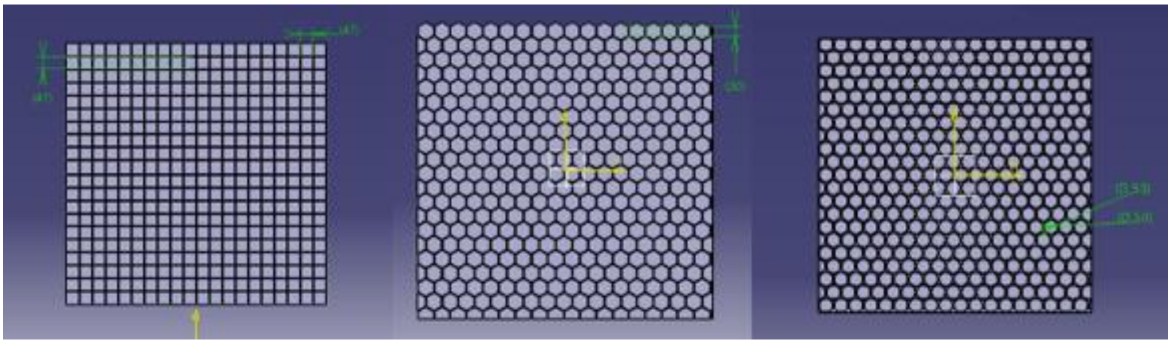

Figure 7.1: Honeycomb structure domain shapes - rectangular, hexagonal and circular (All dimensions are in mm)

The velocity in the test section is used as one of the parameters, to compare the shapes (Varghese, 2019). The velocity is considered as a parameter because, while studying the effect of the flow pattern for different honey comb arrangements, it was observed that the flow was more or less identical in certain cases, as can be seen in figures 7-2 to 7-7. If more velocity is obtained at the test section, it can be inferred that the CWC configuration is capable of working at a higher velocity range.

The velocity obtained at the test section, in case of the circular shape is 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. The inlet velocity is provided by the user at the inlet region (represented in Fig 3.1). The output velocity is measured using velocity probes at the test section

Table 7.1: Velocity Comparison

Input velocity	Output velocity (m/s)		
(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

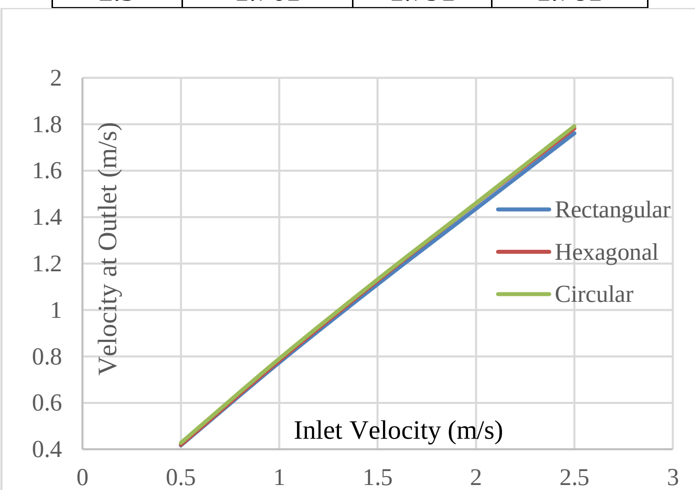

Figure 7.1: Comparison of Velocities

A comparison is carried out, of the flow patterns of various honey comb structures.

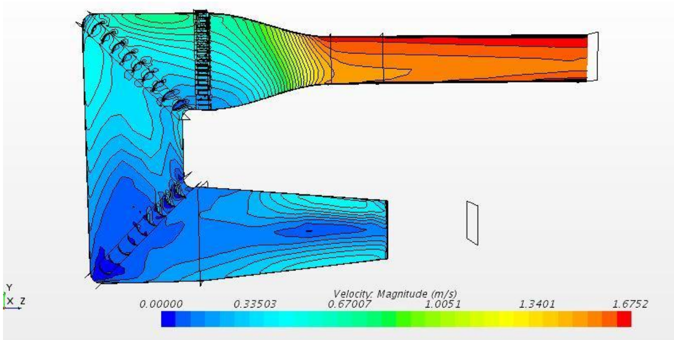

Figure 7.2: Rectangular Honeycomb

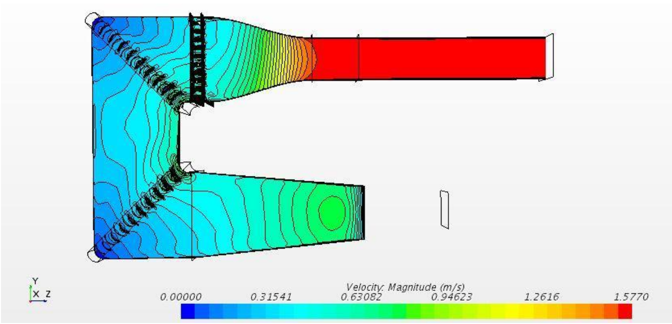

Figure 7.3: Hexagonal Honeycomb

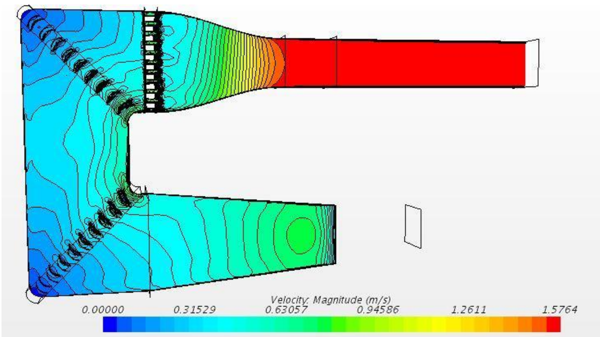

Figure 7.4: Circular Honeycomb

The comparisons are carried out for obtaining a better stream lined flow in test section, for a given honeycomb structure.

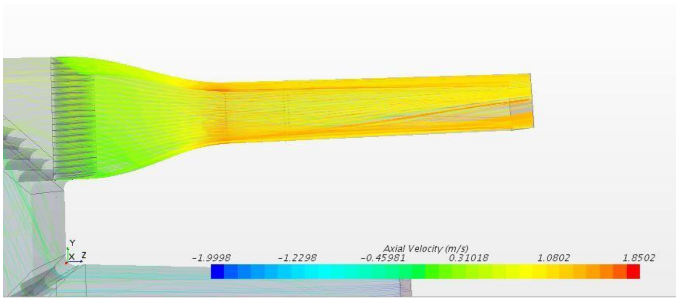

Figure 7.5: Rectangular Honeycomb Stream Line Pattern

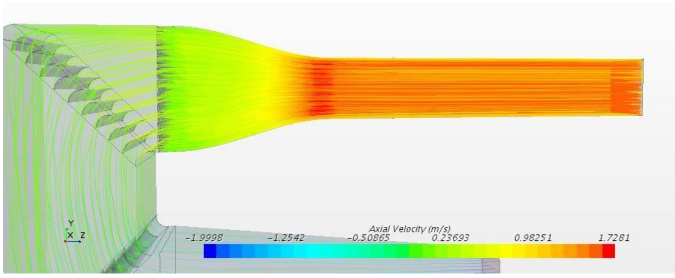

Figure 7.6: Hexagonal Honeycomb Stream Line Pattern

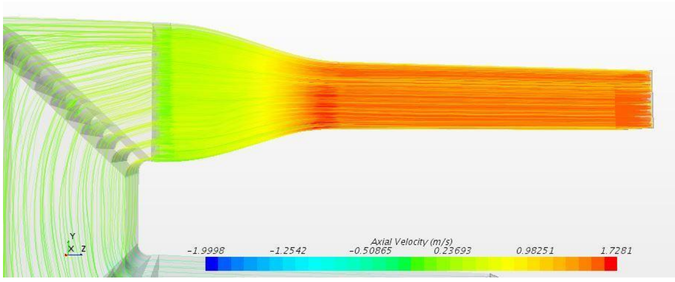

Figure 7.7: Circular Honeycomb Stream Line Pattern

8. Conclusion

1. The disturbance created in the test section, due to improper flushing of glass plates (in the accessory region and test section), can be eliminated by properly flushing, or positioning, the plates.
2. The actual impeller is operated at 700 rpm. When the rpm is increased to 800, air is getting entrapped (at the circular to square transition section; Part no. 8, in Fig. 4-1). A proper study using CFD should be carried out to rectify this particular anomaly and thereby enable the impeller to operate at a higher rpm range.
3. The velocity is measured using a simple pitot tube, which might give rise to errors due to the inaccuracy prevailing in the particular setup. Velocity measurements should rather be done using better flow measuring devices, such as velocity meters or flow meters (which is but costly) (Godey, 2018).
4. The flow pattern and the stream line flow, obtained for the circular and hexagonal sections, are better and smooth, compared to the rectangular shape (Varghese, 2019).
5. For a given input velocity, the maximum output velocity can be achieved for a circular shaped honeycomb structure. 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.
6. Though the hexagonal and circular cross sections both deliver nearly the same results, we may say that the circular section holds an upper footing (Varghese, 2019).
7. From a construction point of view, a circular honeycomb shape can be achieved by the stacking of pipes, which is comparatively simple and cost effective.

9. Scope for Future Work

1. In the present study, analysis was done by varying the honeycomb 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. In future studies, the closed loop analysis for the circular CWC ought be carried out against different impeller input rpms.

3. Closed loop study ought to be carried out, by varying the type of impeller, and its influence in the flow

in the test section.

4. Analysis for a vertical type CWC could be carried out to study the comparison between a vertical type and a horizontal type CWC.

5. The current analysis was done for a 1:4 scale model. The analysis for 1:1 model ought to be done.

Acknowledgements

"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

References

Alho, A. T. P., Maria, H. F. and José, L. S. N. (2010): On the Design of a Circulating Water Channel for the Brazilian National Institute of Metrology – Inmetro, 15th Flow Measurement Conference (FLOMEKO), October 13-15, 2010, Taipei, Taiwan.

Ássi, G. R.S., Julio, R. M., José, A. P. A. and Walter, G. P. C. (2005): Design, assembling and verification of a circulating water channel facility for fluid dynamics experiments, Proceedings of COBEM 2005. 18th International Congress of Mechanical Engineering.

Bansal, R. K. (2011): Fluid Mechanics and Hydraulic Machines, Laxmi Publications (P) Ltd, Bangalore, pp. 287-85.

Bhavaraju, P. J. (2015). Holistic approach to project CWC with Altair, Altair Technology Conference.

Carlton, J. S. (2007). Marine Propellers and Propulsion (2nd ed.). Butterworth-Heinemann. pp. 118-17.

https://doi.org/10.1016/B978-0-7506-8150-6.X5000-1

Chen, Z., Yumi, K. and Hitoshi, N. (2006): CFD Application on the development of circulating water channel, The Sixteenth International Offshore and Polar Engineering Conference, May 2006, San Francisco, California, USA.

Godey, A., Kiran, G. V. and Nair, V. G. (2018): CFD as a tool to validate and modify the flow in the test section of a circulating water channel, Proceedings of the International Conference on Computational and Experimental Marine Hydrodynamics, The Royal Institution of Naval Architects and IIT Madras.

Pullinger, M. G. and Sargison, J. E. (2007): Using CFD to improve the design of a circulating water channel, Paper Presented at the 16th Australasian Fluid Mechanics Conference, Gold Coast, Queensland.

Ramesh, U. S., Simha, B. P. J., Lakshmy, O., and Srinivasa, R. B. (2011). Study project on flow around ships in a hydrodynamic test facility, School of Maritime Design and Research, Indian Maritime University.

Santos, A. M., Alho, A. T. P. Garcia, D. A., Farias, M. H., Massarim, P. L. and Silva, V. V. S. (2015): Experimental studies toward the characterization of Inmetro's circulating water channel, 8th Brazilian Congress on Metrology (Metrologia 2015).

https://doi:10.1088/1742-6596/733/1/012002

Siemens Digital Industries Software (2017): Simcenter STAR-CCM+ User Guide, ver. 12.02. In Modeling Multiphase Flow, p. 3425. Siemens.

Varghese, K. G., Nair, V. G., Avinash, G., and Sunil K. P. G. (2019): Study of flow in the test section of a circulating water channel by varying the honeycomb cross-section, Proceedings of the International Symposium on Marine Design and Construction 2019 (SMDC 2019), Indian Maritime University, Visakhapatnam.