PRELIMINARY ANALYSIS FOR A CIRCULATING WATER CHANNEL USING CFD

B Pradeep Jaya Simha[1], Prof. (Dr.) S C Misra[2], Prof. (Dr.) R P Gokarn[3]

[1]Scientist B, Indian Maritime University, Visakhapatnam Campus, Visakhapatnam, India
[2]Director, Indian Maritime University, Visakhapatnam Campus, Visakhapatnam, India
[3]Ex-Professor of Naval Architecture and Ex-Dean of post graduates studies IIT Kharagpur, India

ABSTRACT

Sustainability in shipping includes improved designs that reduce power consumption. This requires advanced experimental techniques for hull form and propulsor development. A Circulating Water Channel (CWC) is used to generate a controlled flow environment and is used for various hydrodynamic research activities including flow around ships besides predicting its maneuvering behavior, studies for fishing nets, etc. While the experimental facilities such as towing tank facility, sea keeping and maneuvering basin are expensive, a relative low cost CWC, recognized by the ITTC community, is designed to facilitate academic and research activities and better flow studies.

A preliminary Computational Fluid Dynamic analysis is carried out to minimize the variation of flow velocity through the guide vanes and across the width and study means of improving the flow uniformity in the test section of the CWC. A full scale CWC is modeled in commercially available software Altair Hyperworks 11.0 and a mesh is generated for the same. A CFD Analysis is carried out for 85°, 90°and 95° vane angles for an inlet velocity of 1.5 m/s. The results obtained are used to analyze and improve the flow uniformity in the test section of the CWC.

1. INTRODUCTION

The Indian Maritime University came into being through an Act of Parliament (Act 22) on 14th November 2008 as a Central University and is poised to play a key role in the development of trained human resource for the maritime sector. The IMU Visakhapatnam Campus, erstwhile National Ship Design and Research Center (NSDRC), emerges from the confluence of the centrally established Indian Maritime University at Chennai, and India’s premier ship design and maritime research institution, NSDRC.

The CWC is one of the research facilities that is being established currently in the campus and is used for various hydrodynamic research activities including flow around ships besides predicting its maneuvering behavior, studies for fishing nets, shallow water analysis, flow visualization, velocity and pressure distributions around models of ships, rudders, fins, submerged bodies, offshore structures, wake measurements, propulsions tests, windloads on super structures and offshore structures, etc.

The Circulating Water Channel consists of three bends at the corners to divert flow into the channel. The bend consists of three parts bolted together, namely an entrance, guide vane section, and an exit. The function of the guide vanes is to turn the flow through 90° with minimum variation in velocity across the width of the flow and with minimum energy loss. Hence, to check for minimum energy losses and improving flow uniformity in the working section, a preliminary CFD analysis is done in this paper.

2. EXISTING CWCs

Data for 26 existing CWCs categorized into Country and Year established, orientation, working section, maximum speed and motor power is obtained from the “Catalogue of Facilities” published by International Towing Tank Conference, ITTC as shown in Table 1.

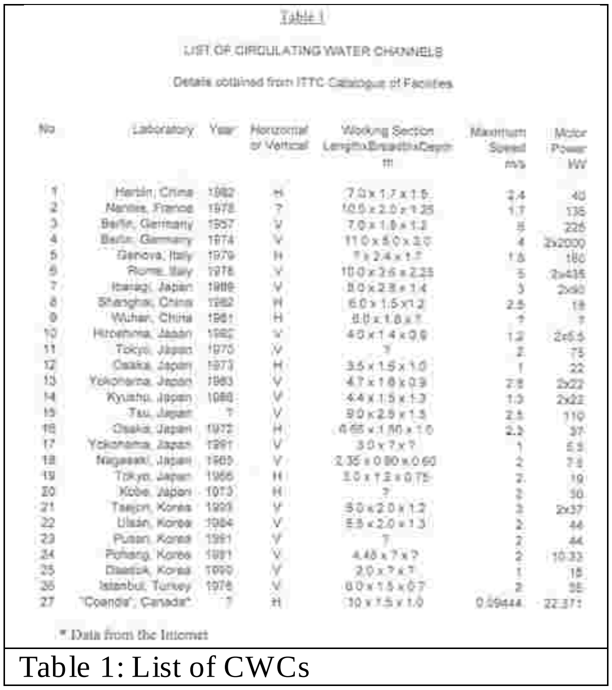

Table 1

LIST OF CIRCULATING WATER CHANNELS

Details obtained from ITTC Catalogue of Facilities

No	Laboratory	Year	Horizontal or Vertical	Working Section Length x Breadth x Depth m	Maximum Speed m/s	Motor Power kW
1	Harbin, China	1982	H	7.0 x 1.7 x 1.5	2.4	40
2	Nantes, France	1978	?	10.5 x 2.0 x 1.35	1.7	135
3	Berlin, Germany	1957	V	7.0 x 1.8 x 1.2	8	225
4	Berlin, Germany	1974	V	11.0 x 5.0 x 3.0	4	2x2000
5	Genova, Italy	1979	H	7 x 2.4 x 1.7	1.5	180
6	Rome, Italy	1978	V	10.0 x 3.6 x 2.25	5	2x435
7	Ibaragi, Japan	1989	V	8.0 x 2.3 x 1.4	3	2x90
8	Shanghai, China	1982	H	6.0 x 1.5 x1.2	2.5	18
9	Wuhan, China	1981	H	8.0 x 1.8 x ?	?	?
10	Hiroshima, Japan	1982	V	4.0 x 1.4 x 0.9	1.2	2x5.5
11	Tokyo, Japan	1975	V	?	2	75
12	Osaka, Japan	1973	H	3.5 x 1.5 x 1.0	1	22
13	Yokohama, Japan	1983	V	4.7 x 1.8 x 0.9	2.8	2x22
14	Kyushu, Japan	1988	V	4.4 x 1.5 x 1.3	1.3	2x22
15	Tsu, Japan	?	V	9.0 x 2.5 x 1.5	2.5	110
16	Osaka, Japan	1972	H	4.65 x 1.80 x 1.0	2.2	37
17	Yokohama, Japan	1991	V	3.0 x ? x ?	1	5.5
18	Nagasaki, Japan	1965	V	2.35 x 0.80 x 0.60	2	7.5
19	Tokyo, Japan	1966	H	3.0 x 1.2 x 0.75	2	19
20	Kobe, Japan	1973	H	?	2	30
21	Taejon, Korea	1993	V	9.0 x 2.0 x 1.2	3	2x37
22	Ulsan, Korea	1984	V	5.5 x 2.0 x 1.3	2	44
23	Pusan, Korea	1981	V	?	2	44
24	Pohang, Korea	1991	V	4.48 x ? x ?	2	10.33
25	Daeduk, Korea	1990	V	2.0 x ? x ?	1	15
26	Istanbul, Turkey	1976	V	8.0 x 1.5 x 0.7	2	55
27	"Coanda", Canada*	?	H	10 x 7.5 x 1.0	0.09444	22.371

* Data from the Internet

Table 1: List of CWCs

A study of the existing channels shows that CWCs with vertical circuits are more popular than those with the horizontal circuits. CWCs with vertical channels are evidently preferred because of the quality of flow in the working section (the part of the CWC in which the experimental is carried out) is better than in the channels with horizontal circuits.

However, a study of the layout drawings of these channels shows that to obtain this better flow quality, it is necessary to incorporate additional features such as surface flow accelerators and vacuum systems.

Since there are nine internationally accredited laboratories with horizontal CWCs, it is considered that an acceptable quality of flow can be obtained even with a horizontal circuit avoiding complications of surface flow accelerators and vacuum systems.

3. CWC AT IMU VISAKHAPATNAM CAMPUS

The dimension of the CWC was finalized from the existing channels within the constraints of space and costs. A 9m x 2m x 2m working section was considered for the 11m x 24.35m overall plan area on par with the existing CWCs as shown in Figure 1 and Figure 2.

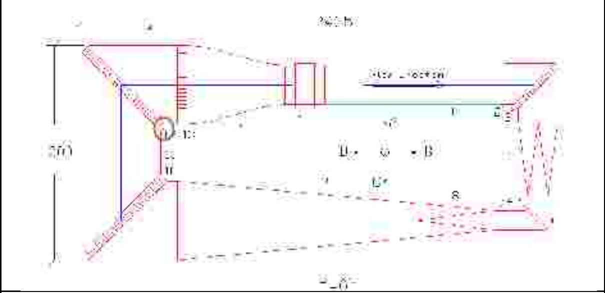

Figure 1: Plan view of the CWC

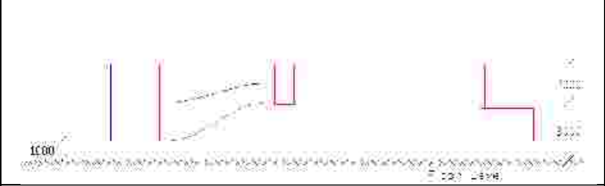

Figure 2: Front view of the CWC

4. CFD ANALYSIS USING ACUSOLVE

4.1 GEOMETRY MODELING

A CFD analysis for 85°, 90°and 95° vane angles is considered here. A 10mm-thick 90-degree subtending angle vane is considered at the turning vane section of the CWC as show in Figure 3. A 95° vane angle inclines towards the working section of the CWC whereas 85° indicates away from it.

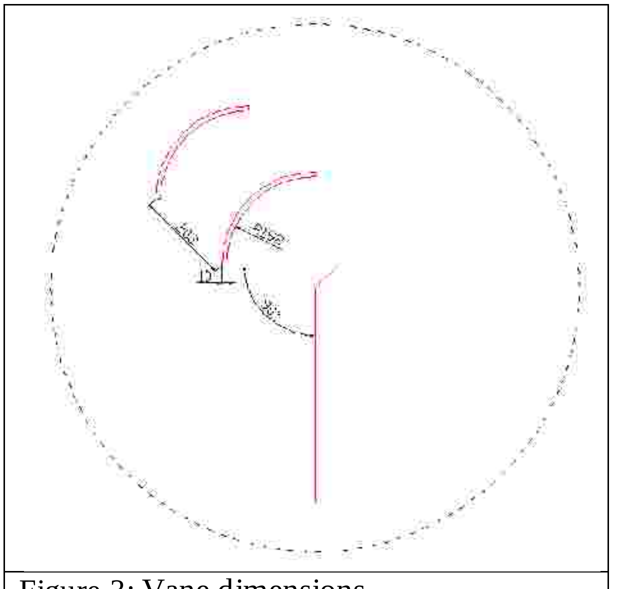

Figure 3: Vane dimensions

The measurements were taken from the Preliminary Concept Design and the geometry is modeled for the upstream of the working section as shown in Figure 4.

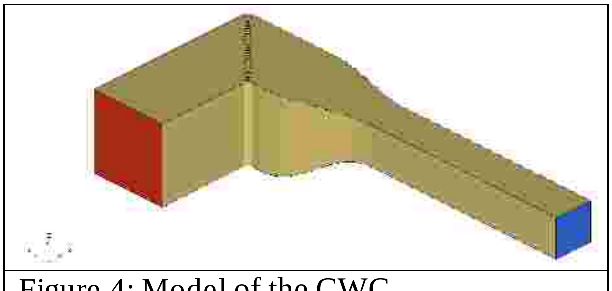

Figure 4: Model of the CWC

4.2 MESH SPACING

An unstructured tetrahedral mesh is generated across all the surfaces and a boundary layer is generated at the vane section and walls with a suitable y+ value in accordance with the Spalart Allamaras turbulence model.

A coarse mesh is considered for the analysis due to the limited size of computation power and speed.

4.3 FLOW SOLUTION

AcuSolve is a general purpose incompressible flow solver. Its technology is based on the Galerkin/Least-Squares (GLS) finite element method. It uses unstructured meshes of low order tetrahedron, pyramid, wedge, and hexahedron elements, with nodal-based field variables.

The GLS formulation provides second order accuracy for spatial discretisation of all variables. In addition to satisfying conservation laws globally, the formulation ensures local conservation for individual elements. Equal-order nodal interpolation is used for all working variables, including pressure and turbulence equations. The semi-discrete generalized-alpha method is used to integrate the equations in time for transient simulations. The resultant system of equations is solved as a fully coupled pressure/velocity matrix system using a preconditioned iterative linear solver.

Once the mesh is checked for element quality, it is exported into AcuSolve. The boundary conditions are specified conveniently and sufficient under-relaxation factors are employed. Spalart Turbulence model with standard wall function is used to model turbulence. 100 iterations are made to get converged solution.

4.4. VANE ANGLE ALTERATION

The effect of three vane angles of 85°, 90° and 95° is considered. The analysis is repeated for the same parameters and the results are plotted.

5. RESULTS AND CONCLUSIONS

A CFD analysis was performed for three vane angles of 85°, 90° and 95° using commercially available Altair Hyperworks and AcuSolve softwares.

The pressure contour plot shown in Figures 5(a), 5(b), 5(c), 6(a), 6(b), 6(c) clearly indicate that a uniform pressure distribution is achieved faster towards the working section of the CWC for 85° than for the other two angles.

The velocity profile shown in Figure 7(a), 7(b), 7(c) indicate that a uniform velocity is achieved for all the three cases at working section. However, the plot for the 95° indicate a faster attainment of convergence of uniform velocity close to the vanes as indicated in Figure 8(a), 8(b), 8(c).

Though the study indicates a preliminary analysis, further studies for different parameters needs to be completed for characterizing their effect to produce uniform flow.

7. FIGURES

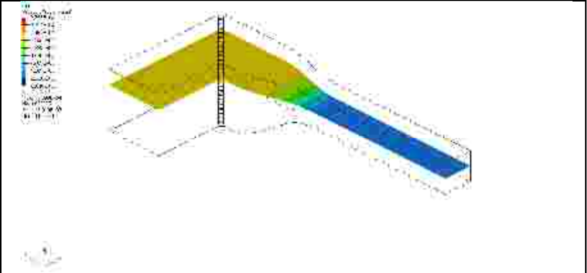

Figure 5(a): Pressure contour plot-85°

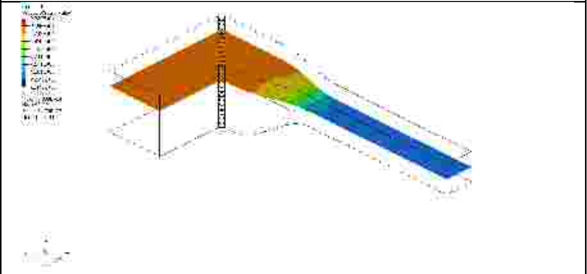

Figure 5(b): Pressure contour plot-90°

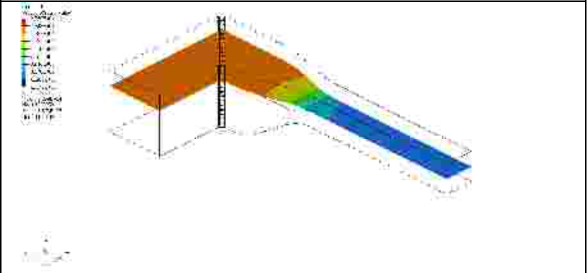

Figure 5(c): Pressure contour plot-95°

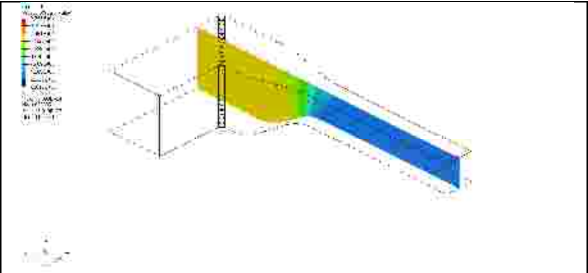

Figure 6(a): Pressure contour plot-85°

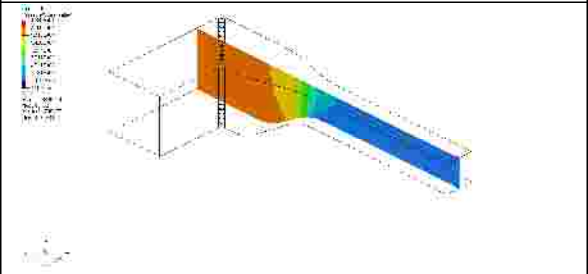

Figure 6(b): Pressure contour plot-90°

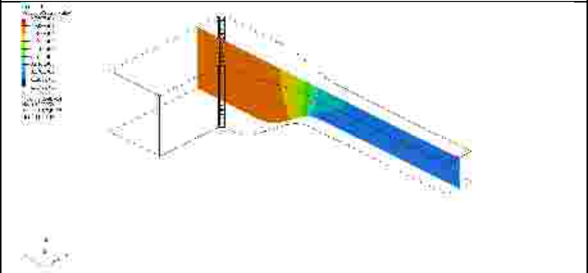

Figure 6(c): Pressure contour plot-95°

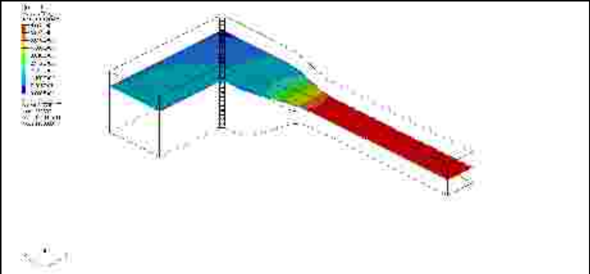

Figure 7(a): Velocity mag. contour plot-85°

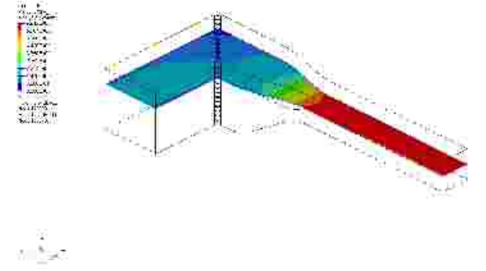

Figure 7(b): Velocity mag. contour plot-90°

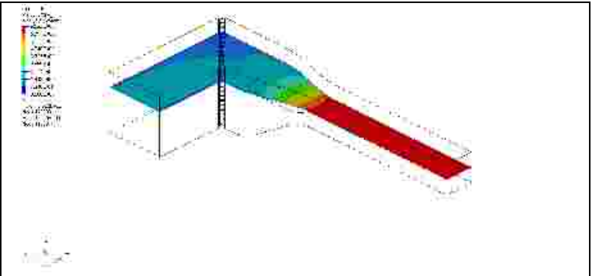

Figure 7(c): Velocity mag. contour plot-95°

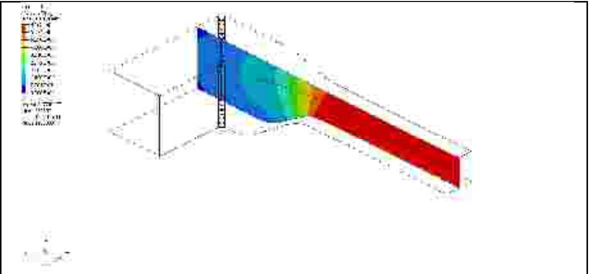

Figure8(a): Velocity mag. contour plot-85°

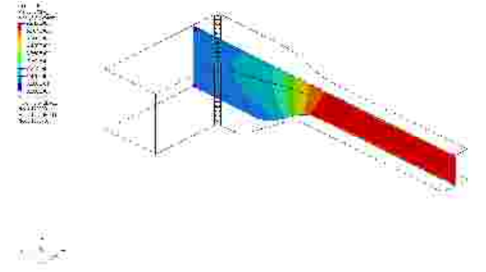

Figure 8(b): Velocity mag.contour plot-90°

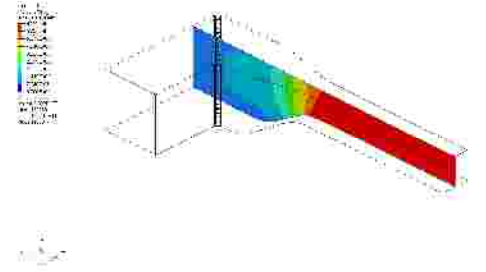

Figure 8(c): Velocity mag. contour plot-95°

8. REFERENCES

[1] Concept Design Document, Aarnav Shipping Technological Research Sanstha.

[2] M.G. Pullinger and J.E. Sargison, 2007, Using CFD to improve the design of a circulating water channel, School of Engineering, University of Tasmania, Hobart, TAS, 7001 AUSTRALIA

[3] AMHRC. 2006. Circulating Water Channel (Flume Tank). Australian Maritime Hydrodynamics Research Centre. http://www.amhrc.edu.au/facilities/flumetank-info.html

[4] Altair Hyperworks 11.0 Documantation.

[5] AcuSolve Documentation.

[6] ANSYS Fluent Documentation.

9. AUTHORS BIOGRAPHY

Pradeep Jaya Simha Bhavaraju currently works at Indian Maritime University Visakhapatnam Campus as Scientist B. He completed his M.E (Marine Engineering) from Andhra University, Visakhapatnam in 2011 and obtained his B.Tech (Mechanical Engineering) from GVP College of Engineering Visakhapatnam, in June, 2008.

S.C.Misra obtained a B.Tech (Hons) degree in Naval Architecture from the Indian Institute of Technology, Kharagpur in 1970 and Ph. D from University of New Castle upon Tyne, UK in Applied Science. He has been a Professor of Naval architecture at IIT Kharagpur and currently holds the position of Director, IMU Visakhapatnam campus.

R.P. Gokarn obtained a B.Tech (Hons) degree in Naval Architecture from the Indian Institute of Technology, Kharagpur in 1962 and the Ph.D in 1971. He currently heads the faculty at Aarnav Shipping Technological Research Sanstha- Mumbai and has over 40 years of teaching experience at IIT Kharagpur. He is also a visiting professor at the Indian Maritime University, Visakhapatnam Campus.