

Computational Flow Analysis of Hull Shape of the Aranmula Snake Boat

Dr.Yogesh M¹, Anand S², Akash Ramesh³, and Munna Krishna R⁴

¹*Professor Department of Mechanical Engineering, St. Thomas College of Engineering & Technology, Chengannur*

^{2,3,4}*Students of Mechanical Engineering, St. Thomas College of Engineering & Technology, Chengannur*

Abstract— This study examines the hydrodynamic performance of the Aranmula Snake Boat, a traditional watercraft from Kerala, India, using Computational Fluid Dynamics (CFD). The primary aim is to optimize the boat's hull shape to enhance speed by reducing drag forces. Initial measurements of the boat's U-shaped hull were taken and recreated in AutoCAD and with the CATIA V5 software created the CAD model for CFD analysis in ANSYS Fluent. A comparative analysis was performed between the original U-shaped hull and a newly designed V-shaped hull of the Aranmula Snake boat which is inspired by the Chundan Vallam. Results indicate a reduction in drag force from 1866.57N for the U-shaped hull to 1835.80N for the V-shaped hull, suggesting potential speed improvements. The study highlights the significance of preserving traditional designs while optimizing performance for modern applications.

Index Terms— Aranmula Snake Boat, Computational Fluid Dynamics (CFD), Hull Optimization, Drag Reduction, Speed Enhancement, Traditional Design Optimization, Chundan Vallam

I. INTRODUCTION

A. RESEARCH BACKGROUND

Snake boat racing is a traditional water sport deeply rooted in the culture of Kerala, India.

The Kerala culture is used in Vallamkali (boat race). Similar types of boat races are conducted in some other parts of the world including the Ottawa Dragon Boat Race Festival, Canada, Sydney Dragon Blades, Australia, Dragon Boat Race Festival, Hong Kong and Dragon Boat Race, Germany. The snake boats of Kerala are constructed according to specifications in the Sthapathya Veda, an ancient treatise on the building of wooden boats and are a marvelous example of ancient India's prowess in Naval Architecture. The Aranmula Snake Boat, also known as Palliyodam, holds a special place in this rich tradition. These long and slender boats, often exceeding 100 feet in length, are manned by a large

crew of rowers who paddle to the rhythm of traditional boat songs. The Aranmula Snake Boat Race, held annually on the Pampa River in the village of Aranmula, attracts thousands of spectators and is a significant cultural event. In recent times, computational fluid dynamics (CFD) has emerged as a powerful tool for analyzing the hydrodynamics of various watercraft, including traditional boats. This study focuses on the computational flow analysis of the Aranmula Snake Boat, aiming to gain insights into its hydrodynamic performance and to increase the speed of the Snake Boat. The major objective of this work is to collect data of a typical snake boat to enable its characterization and thereafter to use this data to perform CFD analysis of its hull form to understand its resistance characteristics. It is extremely important that data on these traditional boats are documented as otherwise it is very likely that this indigenous knowledge base may be lost forever. Also, it is hoped that this study may yield some exciting results which could be useful in the development of new hull forms of vessels which are highly efficient. This is of considerable practical significance when viewed in the background of the ongoing efforts to revive the inland navigation system in Kerala.



Figure 1: Aranmula Snake Boat

B. MOTIVATION

The Chundan Vallam used for the Nehru Trophy is renowned for its speed, attributed to its longer length,

narrow bottom (hull shape), and the amount of area immersed in water. In our study, we focus on how the hull shape contributes to the Chundan Vallam's speed increase. For this purpose, we draw inspiration from the Chundan Vallam's hull shape and aim to optimize the hull shape of the Palliyodam (Aranmula snake boat) accordingly.



Figure 2: Chundan Vallam

C. HULL OPTIMIZATION

Hull optimization of a Snake boat refers to the process of improving the design of the hull, which is the main body of the boat that floats on the water. The goal of hull optimization is usually to enhance the boat's performance in terms of speed, stability, and other factors. Here we are more focusing on enhancing the speed by reducing the drag force.

This process involves using computer-aided design (CAD) software and computational fluid dynamics (CFD) analysis to study and modify the shape of the hull.

Firstly, we created a U-shaped CAD model of the hull of the Aranmula snake boat. To achieve the V-shape, we referenced measurements from the Chundan Vallam. Subsequently, we scaled and optimized the designs to achieve the V-shape for the Palliyodam. Finally, we created a CAD model in CATIA V5 based on the obtained shape and used this model for CFD analysis in ANSYS.



Figure 3: U shaped hull of Snake boat



Figure 4: V shaped hull of Chundan Vallam

II. METHODOLOGY

The computational flow analysis of a snake boat, such as the Aranmula snake boat, involves using computational fluid dynamics (CFD) techniques to study the fluid flow around the boat. This analysis helps in understanding the hydrodynamic performance of the boat, optimizing its design for better efficiency, and improving its overall performance. Here's a general methodology for conducting computational flow analysis

A. Objectives and Design Methodologies

The objectives of the computational flow analysis. This could include understanding the drag forces, optimizing the hull shape, studying the flow patterns around oars, or evaluating the boat's performance under different conditions. In this study we mainly focused on the drag reduction of the snake boat by optimizing the cross sections where it immersed in the water when fully loaded. To achieve this, we have visited the site for data collection and then redraw the sections in AutoCAD software in a trial-and-error method.

B. DATA COLLECTION



Figure 5: Data Collection

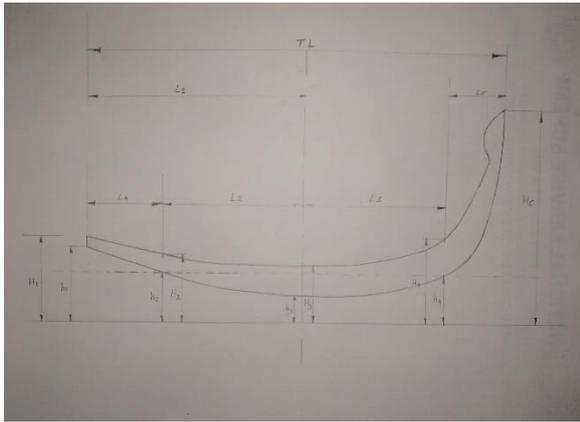


Figure 6: Hand-sketched drawing of the Aranmula snake boat.

TL = Total Length

L1 = Length from the front end to middle

L2 = Length from the front immersed portion hull to middle

L3 = Length from the middle to back immersed portion of hull

L4 = Frond Immersed portion to front end

L5 = Back Immersed portion to back end

H1 = Base line to Frond top end

h1 = Base line to Frond bottom end

H2 = Base line to Top frond portion

h2 = Base line to Bottom immersed frond portion of boat

H3 = Base line to Top middle portion

h3 = Base line to Bottom middle portion

H4 = Base line to Top Back height

h4 = Base line to Bottom immersed back position of boat

H5 = Base line to Back end height

C. CROSS - SECTION GENERATED IN AUTOCAD

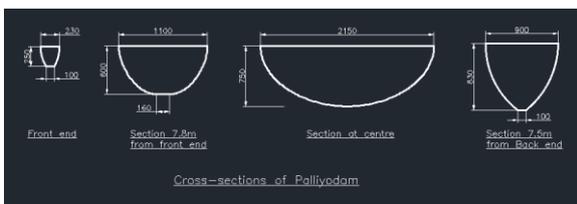


Figure 7: Cross-sections of Palliyodam

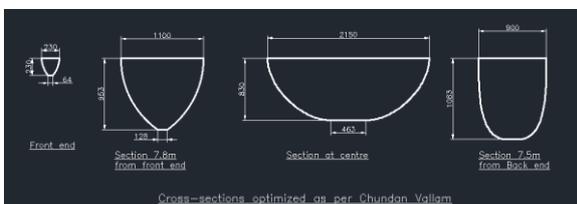


Figure 8: Cross-sections of Optimized shape

Our team visited the Aranmula snake boat (Palliyodam) in Aranmula. We took measurements of the boat using tape, paying extra care to measure the portion immersed in water. The measurements collected from the site were used to create cross-sections in AutoCAD software, helping us to recreate the U-shaped structure accurately. Similarly, for creating the V-shape, we used references from measuring the Chundan Vallam, especially the portion immersed in water. The measurements collected were used to create cross-sections in AutoCAD software. We then scaled and optimized the designs to achieve the V-shape for the Palliyodam.

D. GEOMETRY AND MODEL FOR FLUID DOMAIN

Create a detailed 3D model of the Aranmula Snake Boat, including the design section. The model should accurately represent the boat's shape and features. Simplify the geometry as needed for computational efficiency. In this project we have created the main model and fluid domain in CATIA V5 software, then the geometry is exported as STEP file. To simplify the computation time, we have only considered the water immersed portion of boat for the analysis.

Created the cross sections in AutoCAD software from the data collected at the Site. In CATIA we created the guide curve as per the measurement taken from the site. Then positioned the cross-sections at the exact locations in the guide curve. Generated the boat surface shape using the wireframe and surface design workbench in CATIA. From the generated surface we and modeled the immersed portion of the boat as solid model. Then created the C-shaped fluid domain by subtracting the solid boat portion from the C-shaped fluid domain.

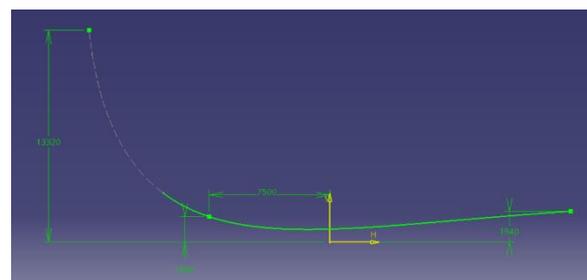


Figure 9: Guide Curve



Figure 10 : Cross-sections with guide curve for preparing the portion of boat immersed in water

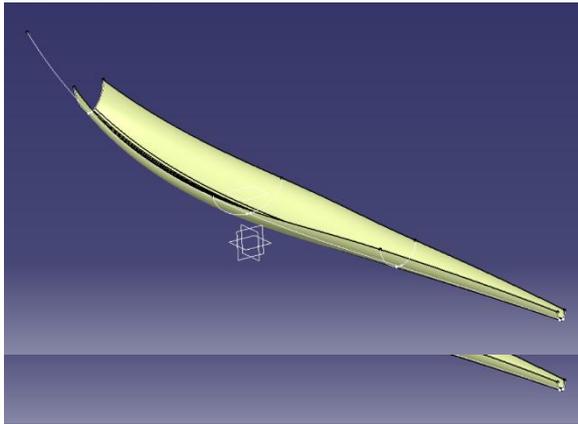


Figure 11: Surface Generated using the Cross-sections and Guide Curve

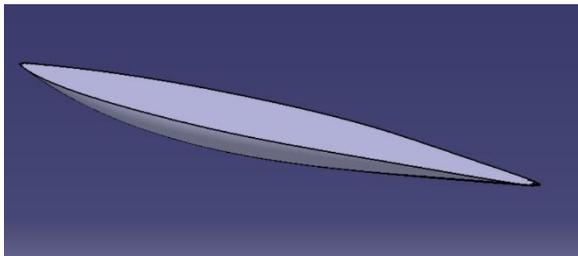


Figure 12: Solid Geometry Extracted from the surface

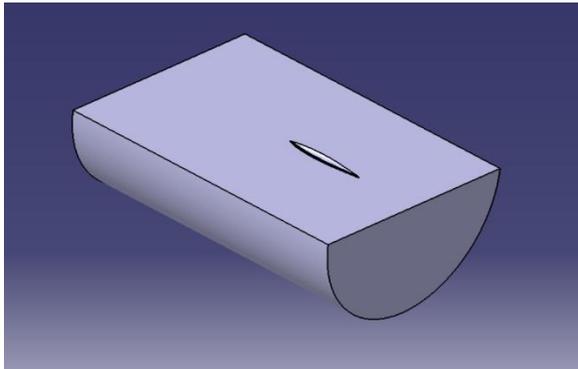


Figure 13: Developed fluid domain for CFD simulation

E. CFD ANALYSIS

Ansys can be used for computational fluid dynamics (CFD) simulations. Here's a general guide on how we approach this task. Creating a detailed computational flow analysis of the Aranmula Snake Boat design section in ANSYS Fluent requires a step-by-step process

- Geometry Preparation
- Mesh Generation
- Setup Boundary Conditions
- Select Physics Models

- Solver Settings
- Run the Simulation
- Post-Processing
- Interpret Results
- Optimization
- Documentation

1. CFD Simulation using ANSYS Fluent.

In this project we performed a comparative study of two cross-sectional shapes (U and V in the Palliyodam to check whether the Cross-sectional change reduces the drag force or not.

In the first phase study we analyzed the actual Palliyodam model to get the drag force value. Later in the second phase we replaced the cross-sections with the optimized cross-sections those were inspired from the Chundan vallam.

Steps involved in the simulation

- Imported the fluid domain prepared in the CATIA. Then Rotated the model in ANSYS Design Modeler to match with the Global Axis to manage the flow direction of Fluid.
- We meshed the model using tetrahedral elements in ANSYS Mechanical Module.
- Assigned Boundary name for the faces like Inlet, Outlet and Boat Surface, then transferred the file to ANSYS Fluent.
- In the fluent software we converted the tetrahedral mesh to polyhedral mesh to get more quality elements.
- Then set up the model for analysis.
- Then run the analysis to solve the case.
- After the completion of solution phase collected the data for verification.

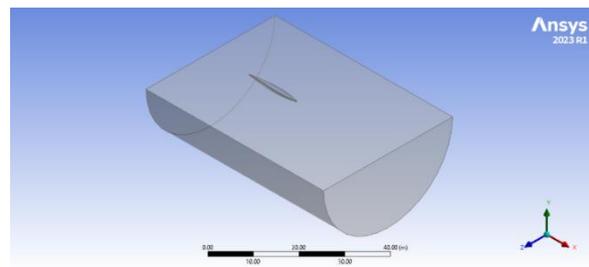


Figure 14: Geometry Imported into ANSYS Fluent

2. Mesh Generation

Generate a high-quality mesh for your geometry. The mesh should be fine enough to capture the important flow features around the boat. Pay attention to the

near-surface regions and areas of flow separation. The mesh divides the geometry into small elements for computational analysis. Here we converted the tetrahedral mesh to polyhedral mesh to get more quality elements.

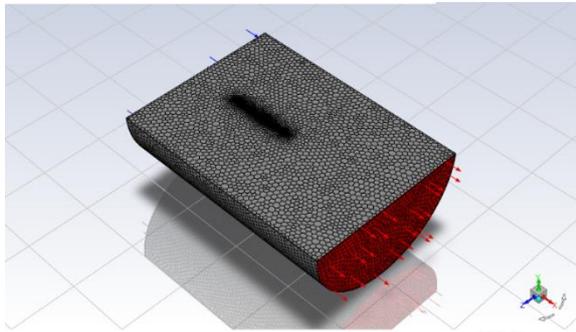


Figure 15: Tetrahedral Mesh generated

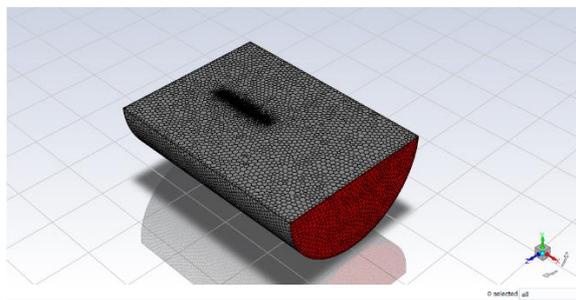


Figure 16: Remeshed with polyhedra elements

3. Boundary Condition

Define the boundary conditions for the simulation. Specify the Turbulent Model, Inlet velocity, and outlet condition.

CFD Solver: - Viscous model, K-Epsilon 2 equation model

Boundary Conditions: -

Inlet Velocity: 30km/hr

Outlet Gauge Pressure: 0 Pa

4. Fluid Properties

Define the properties of the fluid, such as density and viscosity. Here we are considering steady flow condition.

5. Numerical Solver

Choose an appropriate numerical solver for your simulation. Common CFD solvers is Finite Element Method (FEM). Selected the solver that is well-suited for simulating the fluid-structure interaction in which the boat's motion is involved.

6. Simulation Setup

Set up the simulation parameters, such as time step, convergence criteria, and turbulence models if

applicable. Specify the solver settings and ensured that they are appropriate for the type of simulation conducting

7. Running the Simulation

Run the simulation, and monitor its progress. It may take several iterations to achieve convergence. Check for total pressure at the boat surface.

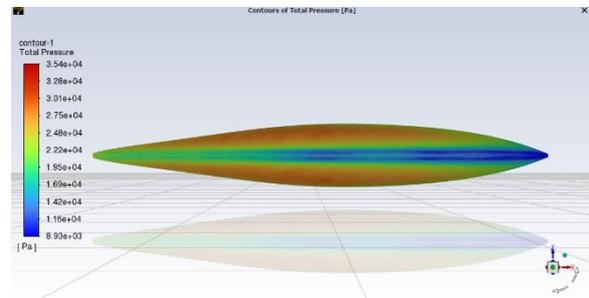


Figure 17: Total Pressure Plot of Actual model at boat surface

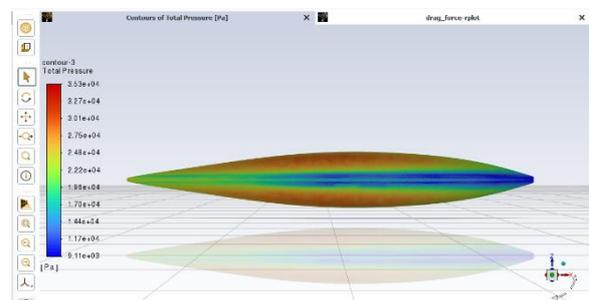


Figure 18: Total Pressure Plot of optimized model at boat surface

F. Result

Now we completed the analysis of two cross-sectional shapes (U and V) in the Palliyodam to check whether the Cross-sectional change reduces the drag force or not.

In the first phase study we analyzed the actual Palliyodam model and obtained the drag force value. Later in the second phase we replaced the cross-sections with the optimized cross-sections those were inspired from the Chundan vallam to obtained the drag force value corresponding to this shape.

III. CONCLUSIONS

The Chundan Vallam used for the Nehru Trophy is renowned for its speed, attributed to its longer length, narrow bottom (hull shape), and the amount of area immersed in water. In our study, we focus on how the hull shape contributes to the Chundan Vallam's speed

increase. For this purpose, we draw inspiration from the Chundan Vallam's hull shape and aim to optimize the hull shape of the Palliyodam (Aranmula snake boat) accordingly.

Our team visited the Aranmula snake boat (Palliyodam) in Aranmula to collect measurements. We carefully measured the boat using tape, with particular attention to the portion immersed in water. These measurements were used to create accurate cross-sections in AutoCAD software, allowing us to recreate the U-shaped structure faithfully. To create the V-shape, we referenced measurements from the Chundan Vallam, focusing on the portion immersed in water, and used these measurements to create cross-sections in AutoCAD software. We then scaled and optimized the designs to achieve the V-shape for the Palliyodam. Subsequently, we created a CAD model in CATIA V5 based on the obtained shape and used this model for CFD analysis in ANSYS.

Based on the simulation results, the redesign of the Aranmula snake boat from a U to a V shape resulted in a noticeable change in the drag force experienced by the boat. The U shape exhibited a higher drag force of 1866.57N, while the V shape showed a lower drag force of 1835.80N. Although the difference in drag force between the two shapes is relatively small, it indicates that the V shape has the potential to reduce drag and improve the boat's speed.

This finding is significant, suggesting that small changes in the snake boat's design can have a measurable impact on its performance. By reducing drag, the V shape design has the potential to increase the boat's speed.

However, it is essential to consider practical constraints, such as the space needed for passengers to be seated comfortably and safely. The Aranmula snake boat is not just a racing vessel but also a cultural symbol with a long tradition. Therefore, our optimized V-shaped hull of the palliyodam takes care and focuses on balancing improved performance with respect for the boat's heritage and functionality.



Figure 19: Drag force of actual model at boat surface

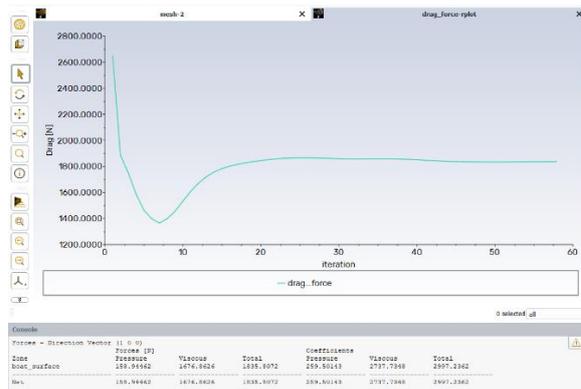


Figure 20: Drag force of optimized model at boat surface

REFERENCES

- [1] Subbaiah, B.V., Thampi, Santosh.G. and Mustafa, V. (2015) 'Modelling and CFD analysis of traditional snake boats of Kerala', Aquatic Procedia, 4, pp. 481–491. doi:10.1016/j.aqpro.2015.02.063.
- [2] Miyata, H., Akimoto, H. and Hiroshima, F. (1997) 'CFD performance prediction simulation for hull-form design of sailing boats', Journal of Marine Science and Technology, 2(4), pp. 257–267. doi:10.1007/bf02491532.
- [3] A., D. et al. (2010) 'Modelling hydrodynamic drag in swimming using computational fluid dynamics', Computational Fluid Dynamics [Preprint]. doi:10.5772/7112.