iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 9 **Issue:** III **Month of publication:** March 2021

DOI: https://doi.org/10.22214/ijraset.2021.33525

Conceptual Design and Fabrication of Water Tunnel

Dhruv Panchal[1], Dhaval Mahyavanshi[2], Bhavik Nanera[3], Anand Jadvani[4], Swati Chauhan[5]
[1, 2, 3, 4]B.Tech Students, [5]Assistant Professor, Department of Aeronautical Engineering, Parul University, Vadodara, Gujarat, India

Abstract: We designed, fabricated and installed a low-speed water tunnel to conduct fluid-structure interaction (FSI) experiments. The tunnel is custom-designed to deliver uniform flow up to 3 m/s through a square test section. This work presents an overview of water tunnels, their primary components, and their functions. It details the design and fabrication of key components of the water tunnel and provides justification for design decisions. It discusses instruments for measuring pressure and for flow visualization. The qualification procedure and results are also described
Keywords: Water Tunnel (WT) Low-speed, Fluid-structure interaction, qualification, Design, CFD Analysis, & Fabrication.

I. INTRODUCTION

Water tunnels are used for a variety of reasons such as testing a structure under flowing fluid, which affect the forces on the submerged body. Water tunnels are also used for preliminary measurements of prototypes (e.g. vehicles, buildings, towers and bridges) early design such as to conduct flow visualization around models. Water tunnels are generally preferred over wind tunnels for measurement like Particle image velocimetry (PIV). This is because flow visualization in wind tunnels is harder to control and it dissipates quickly. This paper describes a low speed subsonic closed recirculating type water tunnel facility that has been designed.

II. METHODOLOGY

The modelling of the structure is done by using engineering software. After that we are doing CFD analysis of solid model to validate the design of water tunnel after completion successful CFD analysis of solid model we are fabricate the water tunnel as per the solid model design.

1) *Phase 1:* Design calculation.
2) *Phase 2:* Solid Model Design.
3) *Phase 3:* CFD Analysis.
4) *Phase 4:* Fabrication.
5) *Phase 5:* Experimental analysis.
6) *Phase 6:* Result

A. Component of WT

1) *Main Component of WT.*
   a) Test Section
   b) Diffuser
   c) Propeller Part
   d) Contraction Cone

2) *Helping/Other Component.*
   a) Turning sections.1
   b) Transitions 1
   c) Transitions 2
   d) Settling Chamber
   e) Turning Section

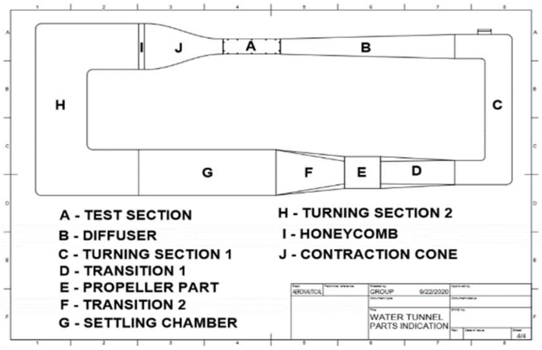


Fig: - 1 Component of Water Tunne

III. CONCEPTUAL DESIGN

Basically, conceptual design is the very first stage of the product design process, where drawings and other illustrations or models are used. In which the broad outlines of function and form of something are articulated. It includes the design of interactions, experiences, processes, and strategies.

A. Test Section

The test section is where models are tested and data are collected. Test section designs a significant consideration in the design of a water tunnel. The tunnel's intended uses affect decisions regarding the test section's cross section and size, which ultimately affect the cross section and size requirements for the entire tunnel. After determining the desired cross section and sizing requirements, numerous factors can be assessed, including flow speed, cavitation ranges, boundary layer growth, center line velocity, and blockage ratios. Other constraints, such as the simplicity of installation and cost, must also be considered relative to the importance of each factor.

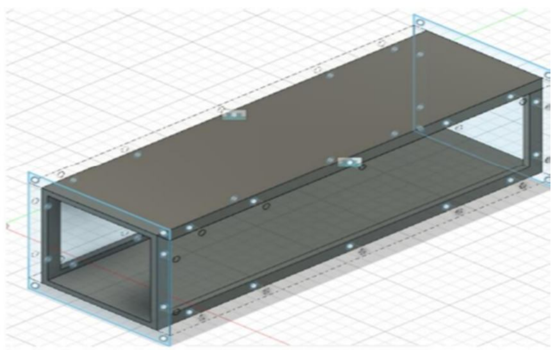


Fig :- 2.1 Test Section Design

B. Dimensions Of Test Section

Sr No	Parameter	Dimension
1	Length (lts)	30cm
2	Height (hts)	9cm
3	Width (Wts)	9cm
4	Area Cross-section (Ats)	81 cm^2
5	Water temperature (T)	20^0C = 293 K
6	Water density (ρw)	1000 kg/m^3
7	Minimum contraction ratio (rc)	9
8	Maximum flow rate ($Umax$)	3m/s

1) *Calculation:* Now we have to calculating mass flow rate in the test section because it is very important parameter for selecting motor and also important for study about flow in test section.

a) Using Mass flow rate Equation,

Mass flow rate: - $\dot{m} = U_{max} * \rho_w * A_{ts}$

$\dot{m} = 3*1000*81*10^{-4}$

$\dot{m} = 24.3$ kg/s

C. *Diffuser*

The diffuser decelerates high velocity flow exiting the test section while avoiding excessive energy loss or decreasing flow quality throughout the rest of the tunnel. A gradual increase in the cross-sectional area aids in minimizing pressure losses in the system, but if the expansion half-angle is too large, flow separation may occur, resulting in high energy loss and flow instabilities that would ultimately create disturbances in the test section. If the half-angle is too small, it results in a diffuser too long to meet size constraints.

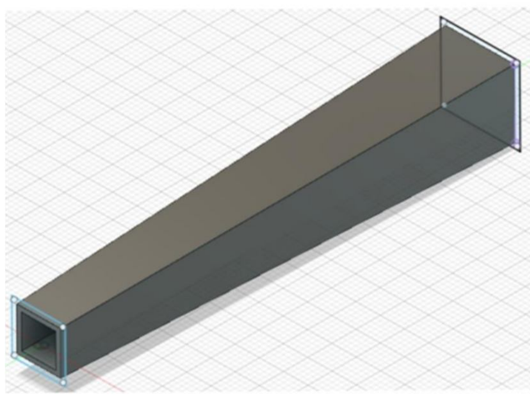


Fig 2.2:- Diffuser Design

1) *Dimensions Of Diffuser*

Sr. No	Parameter	Dimension
1	Inlet cross section	81cm^2
2	Outlet cross section	203.06cm^2
3	Diffuser half angle	6^0
4	Length	91.5cm

The length of the diffuser was determined using the effective diffuser half angle.

$$\emptyset_{half} = tan^{-1}\left(\frac{\sqrt{\frac{4a_{out}^2}{\pi}} - \sqrt{\frac{4a_{in}^2}{\pi}}}{2L_D}\right)$$

(From this equation diffuser half angle we got 6 *o* and length of the diffuser (*LD*) we got 91.53 cm.)

As per the calculations we found out dimensions of turning section 1 which is shown below.

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue III Mar 2021- Available at www.ijraset.com

D. Turning Section 1

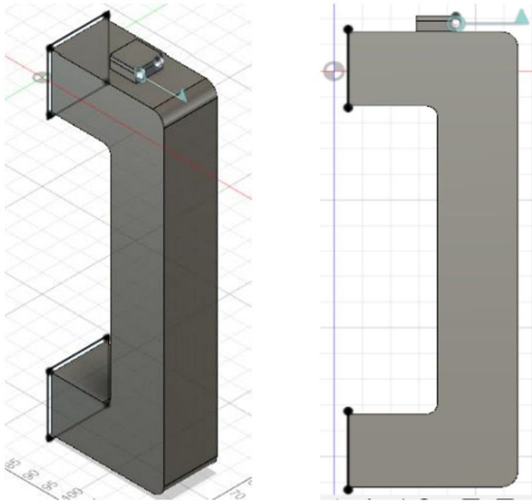


Fig 2.3:- Turning Section 1 Design

As per the calculations we found out dimensions of turning section 1 which is shown below

1) Dimensions Of Turning Section 1

Sr.No	Parameter	Dimension
1	Inlet area (cross section)	203.06 cm^2
2	Outlet area (cross section)	203.06 cm^2
3	Length	30 cm
4	Width	14.25 cm
5	Height	88.5 cm
6	Outside edge radius	3 cm
7	Inside edge radius	2 cm

2) Turning Section 2

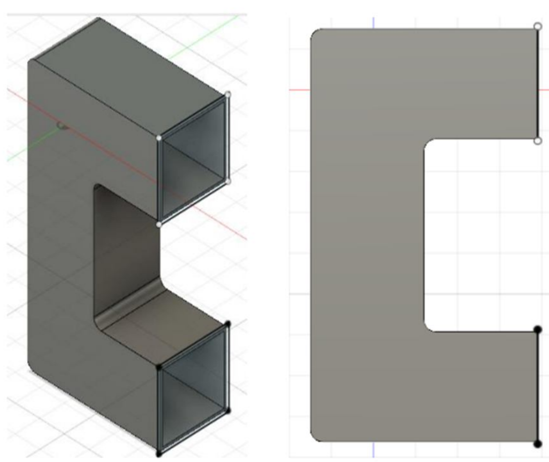


Fig :- 2.4 Turning Section Design

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue III Mar 2021- Available at www.ijraset.com

3) Dimensions Of Turning Section 2

Sr.No	Parameter	Dimension
1	Inlet area (cross section)	729cm^2
2	Outlet area (cross section)	729cm^2
3	Length	54 cm
4	Width	21 cm
5	Height	101.25 cm
6	Outside edge radius	3 cm
7	Inside edge radius	3 cm

E. Transition 1

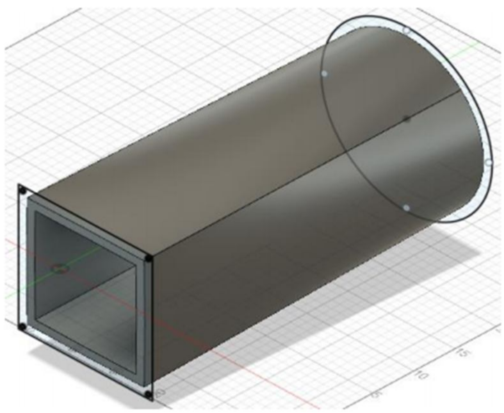


Fig 2.5 Transition 1 Design

1) Dimensions Of Transition 1

Sr.No	Parameter	Dimension
1	Inlet area (cross section)	203.06 cm^2 (square)
2	Outlet area (cross section)	276.12 cm (circle)
3	Length	39 cm

2) Transition 2

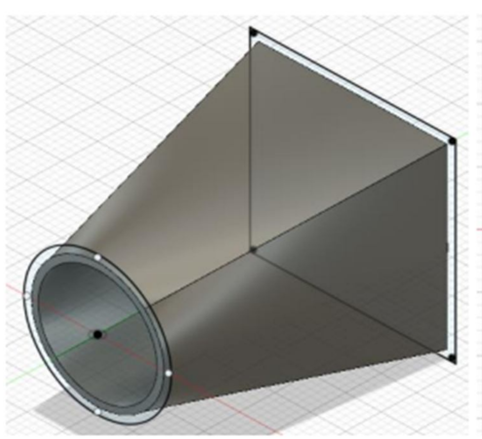


Fig 2.5 :- Transition 2 Design

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue III Mar 2021- Available at www.ijraset.com

3) Dimensions Of Transition 2

Sr.No	Parameter	Dimension
1	Inlet area (cross section)	203.06 cm^2 (square)
2	Outlet area (cross section)	276.12 cm^2(circle)
3	Length	39 cm

F. Circular Part

Propeller is fitting in propeller circular part which is placed between two transitions. Its inlet is connecting with outlet of transition 1 and outlet is connecting with inlet of transition 2.

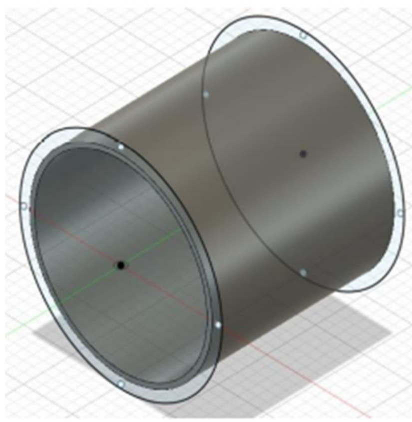


Fig 2.7 Circular Part Design

1) Dimensions Of Propeller Circular Part

Sr.No	Parameter	Dimension
1	Inlet area (cross section)	276.12 cm^2(circle)
2	Outlet area (cross section)	276.12 cm^2 (circle)
3	Length	18.75 cm

G. Settling Chamber

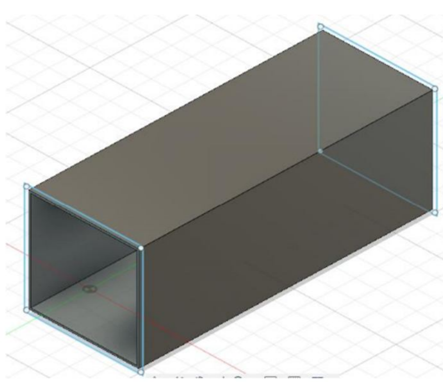


Fig :- 2.8 Settling Chamber Design

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue III Mar 2021- Available at www.ijraset.com

1) Dimensions Of Settling Chamber

Sr.No	Parameter	Dimension
1	Inlet area (cross section)	729 cm^2
2	Outlet area (cross section)	729 cm^2
3	Length	72 cm

H. Contraction Cone

The primary purpose of the contraction cone is to minimize turbulence while increasing flow speed. The inlet to outlet ratio, also known as the contraction ratio, greatly affects the cone's effectiveness in minimizing the increase in turbulence. Greater contraction ratios lead to lower turbulence and therefore a more uniform velocity profile within the test section.

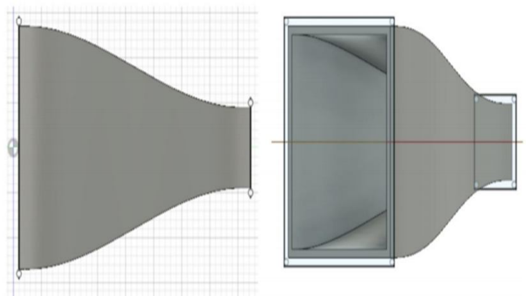


Fig:-2.9 Contraction Cone Design

1) Dimensions Of Settling Chamber

Sr.No	Parameter	Dimension
1	Inlet area (cross section)	729 cm^2
2	Outlet area (cross section)	81 cm^2
3	Length	41.25 cm

I. Propeller

The purpose of the pump is to generate a pressure differential large enough to overcome the head losses in the system and drive the flow loop. The velocity and pressure ranges play a large role in determining the appropriate motor/pumping system. The maximum power of the motor sets the upper limit for tunnel speed.

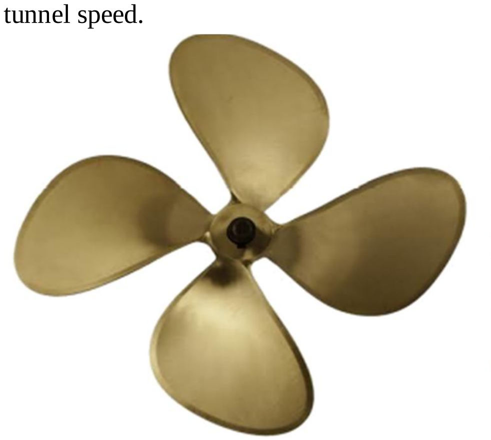


Fig 2.10:- Propeller Design

1) Calculation

The power required without accounting for fluid losses

$Pnl = Umax * \rho w * At$

Pnl = 3*1000*531441*10^{-7}

Pnl = 159.4323 W

The power required accounting for fluid losses

$$Pfl = Pnl\left(1 + \frac{1}{re}\right)$$

$$Pfl = 159.4323\left(1 + \frac{1}{5}\right)$$

$$Pfl = 191.31876 \text{ W}$$

Calculations shown in the above Section were used to determine the power requirements for the motor and mass flow rate for the propeller. Calculations show that a 191.31876 W motor is the smallest that can successfully fulfill these requirements but a 200 or 225 W motor was selecting for convenience.

IV. CFD ANALYSIS OF CAD MODEL

CFD is carried out to a huge range of studies and engineering troubles in lots of fields of look at and industries, which includes aerodynamics and aerospace evaluation, climate simulation, natural science and environmental engineering, industrial machine design and analysis, organic engineering, fluid flows and heat transfer, and engine and combustion analysis.

Computational fluid dynamics (cfd) is a branch of fluid mechanics that uses numerical evaluation and records structures to analysis and clear up troubles that involve fluid flows. Computers are used to perform the calculations required to simulate the unfastened-movement glide of the fluid, and the interaction of the fluid (beverages and gases) with surfaces described by using boundary situations. With high-speed supercomputers, higher answers may be accomplished, and are often required to resolve the biggest and most complicated issues. Ongoing studies yields software that improves the accuracy and speed of complex simulation scenarios together with transonic or turbulent flows. Preliminary validation of such software program is generally performed the use of experimental apparatus which includes wind tunnels. Further, formerly achieved analytical or empirical analysis of a specific hassle may be used for evaluation. A very last validation is frequently finished the use of full-scale testing, consisting of flight checks.

After making the solid model of water tunnel we are going to doing the CFD (computational fluid dynamics) analysis of solid model in ANSYS R17.1 and we found out the results are shown below.

A. Geometry Of Water Tunnel

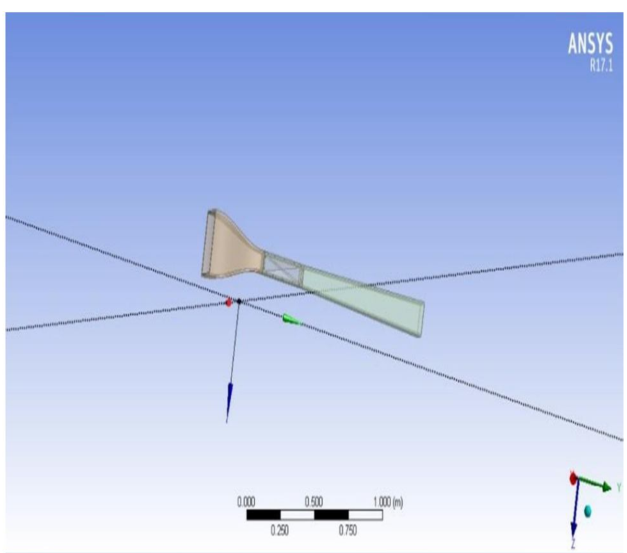


Fig 3.1 Graphics OF WT

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue III Mar 2021- Available at www.ijraset.com

B. Meshing Of Water Tunnel

A mesh divides a geometry into many elements. These are used by the CFD solver to construct control volumes. Terminology: ... The shapes of control volumes depend on the capabilities of the solver. Structured-grid codes use quadrilaterals in 2D and hexahedrons in 3D flows.

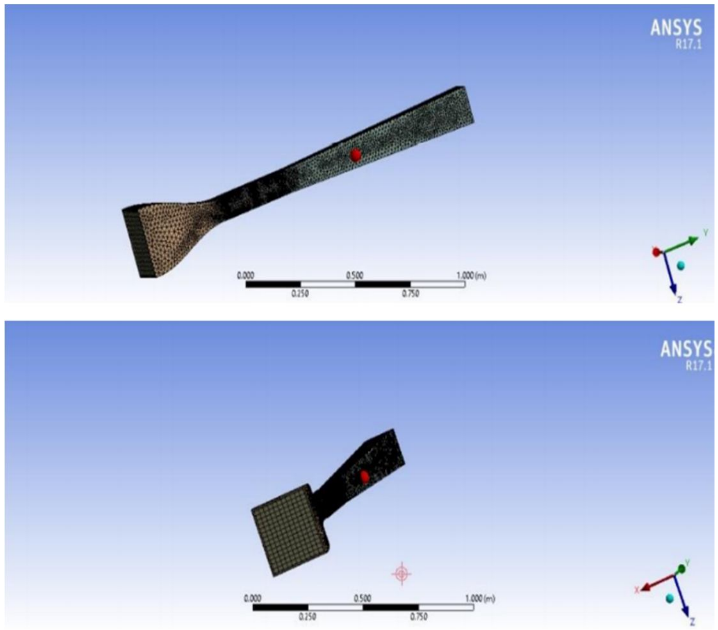


Fig 3.2 :- Meshing of WT

C. Boundary Condition

1) For boundary conditions first we set mass flow rate equation
2) For material we defined solid as a “CRC iron Sheet and fluid as water.
3) For inlet we define the inlet velocity as 3m/s.
4) Density of Water is 1000 kg/m
5) Initial Water Temperature is 20^0C = 293 K

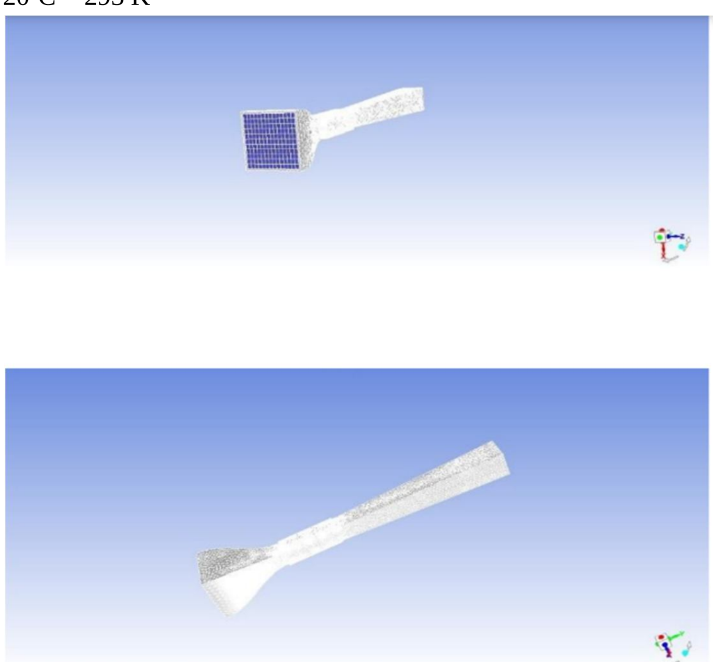


Fig 3.3:- Boundary Condition Apply

V. RESULT OF CFD ANALYSIS

A. Pressure Distribution In Water Tunnel

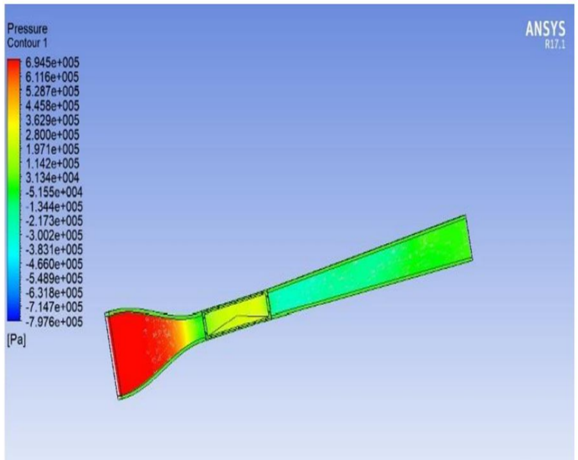


Fig 4.1 :- Pressure Distribution in WT

B. Velocity Distribution In Water Tunnel

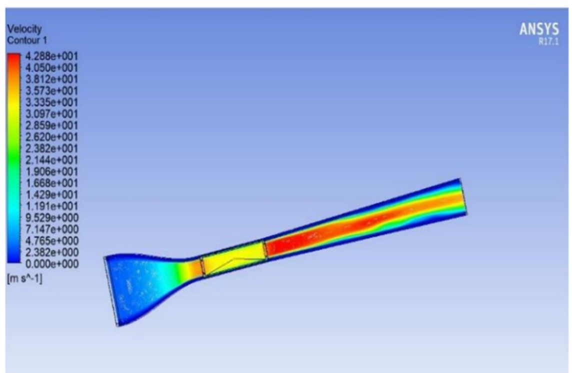


Fig 4.2:- Velocity Distribution in WT

C. Velocity Streamline Distribution In Water Tunnel

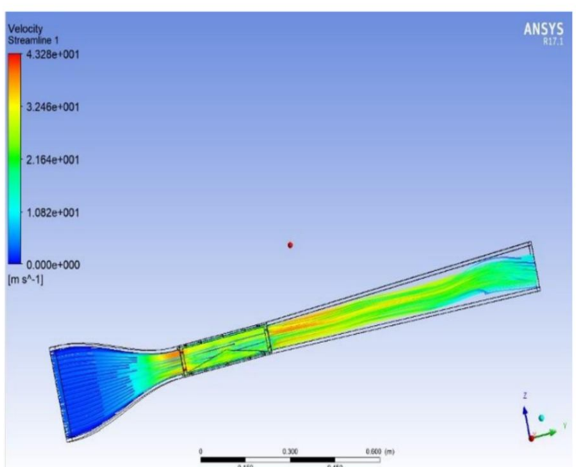


Fig 4.3:- Velocity Streamline of WT

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue III Mar 2021- Available at www.ijraset.com

D. Pressure Distribution In Test Section

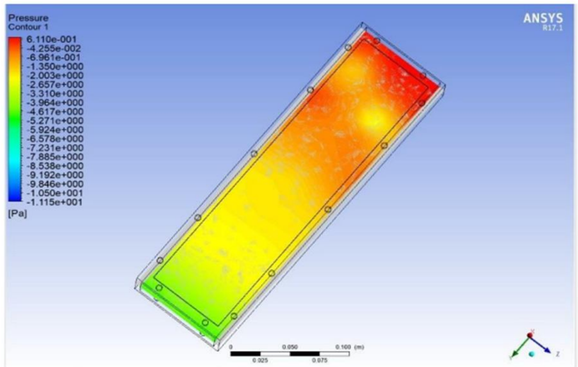


Fig 4.4:- Pressure Distribution in Test Section

VI. FABRICATION OF WATER TUNNEL

After the validation we started the fabrication of water tunnel. We used CRC (cold rolled steel sheet) of 20 gauge to made different parts of water tunnel. Cold rolled sheet offer a variety of outstanding properties, including easy formability and a smooth, clean surface.

A. Properties Of CRC (Cold Rolled Steel

1) It has good durability.
2) This material is heat resistance.
3) It has good welding performance.
4) This sheet is coming with the silver smooth surface.

B. Benefits Of CRC (Cold Rolled Steel)

1) Cost of this material sheet is comparatively low to the other metal sheets.
2) It has long life as compare to other metals.
3) Very important this material is corrosion resistance which is our prime requirement.
4) Inspection of this material is easy which is directly affected in future maintenance work due to easy inspection maintenance cost is also low as compared other material

C. Procedures Of Fabrication

1) Marking
2) Cutting
3) Bending
4) Layer Forming
5) Welding (Arc Welding)
6) Assembling

VII. INSTRUMENT USED DURING FABRICATION.

A. Machines Used In Fabrication

1) Manual Sheet Cutting Machine.
2) Manual Sheet Bending Machine
3) Sheet Rolling Machine
4) Grinder Machine
5) Welding Machine

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue III Mar 2021- Available at www.ijraset.com

B. Tools Used In Fabrication

1) Scale
2) Right Angle
3) Hammer
4) Screw Driver

VIII. FABRICATION PART OF WATER TUNNEL

A. Turning Section

Fig 6.1 Turning Section 1& 2

B. Diffuser

Fig 6.2:- Diffuser

C. Circuler

Fig 6.3 :- Circular

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue III Mar 2021- Available at www.ijraset.com

D. Setlling Chamber

Fig 6.4 Settling Chamber

E. Transition 1&2

Fig 6.5 Transition 1 & 2

IX.DEAERATION PROCESS

A. Overfill the tunnel slightly to balance water loss as the tunnel is desecrated.
B. Start the motor running at low speeds.
C. Let the motor run for about five minutes to allow the system to stabilize.
D. Ramp the motor up to a high speed.
E. While the motor is running at high speeds, slightly open the top vent. This will allow air bubbles to escape, but also carry some water out with it.
F. Allow step 5 to continue for 15-20 minutes.
G. Ramp the motor down to medium speed.
H. Repeat step five. Allow the tunnel to vent at lower speeds for another 10 minutes.
I. Ensure the water level in the tunnel is still full, but do not allow another air pocket to fill the tunnel. This can be done by slightly opening the vent and checking whether water flows out of the tunnel or air is sucked in.
J. Close the vent, and operate the tunnel as intended.

X. CONCLUSION

The tunnel, currently in operation, delivers uniform flow up to 3 m/s and is being used to conduct fluid-structure interaction experiments. The work describes the major components of the water tunnel and their functions. It also discusses the qualification procedure and results.

A. Used for analyzing the different types of airfoil and hydrofoil.
B. Water tunnel are sometimes used in place of wind tunnel to perform measurements because techniques like particle image velocimetry (PIV) are easier to implement in water.
C. Ship model basin, used in naval architecture to study the behavior of sea vessels.
D. Wave tank, a laboratory setup for observing the behavior of surface wave.
E. Ripple tank, a shallow glass tank of water used in schools and colleges to demonstrate the basic properties of waves.

XI. FUTURE SCOPE

The use of water tunnels and scaled models to help solve the aerodynamics problems of aircraft and the hydrodynamic nature of naval ships and submarine as they are developed is a well-established method in the aerospace and naval sector. In the early days of aviation, water tunnels were used to see if and how new aircraft designs would fly and hydrodynamic nature of airfoil and hydrofoil. Nowadays, with the tube and wing design of aircraft dominant in the market, they are more likely to be used for verification and certification.

XII. ACNKNOWLEDGEMENT

We have taken efforts in this project. However, it would not have been possible without the kind support and help of faculties. We would like to extend our sincere thanks to all of them with immense pleasure we express our deep and sincere gratitude, regards and thanks to our project guide Prof. SWATI CHAUHAN for her excellent guidance, invaluable suggestions and continuous encouragement at all the stages of our project work. As a guide she has a great influence on us, both as a person and as a professional.

REFRENCES

[1] Seyyed Mostafa Seyyedi, Rouzbeh Shafaghat, “Design Algorithm of a Free Surface Water Tunnel to Test the SurfacePiercing Propellers (SPP); Case Study Water Tunnel of Babol Noshirvani University of Technology”, INTERNATIONAL JOURNAL OF MARITIME TECHNOLOGY Vol.6/ Summer 2016.

[2] Sanket V. Kalgutkar, Shubham K. Khobragade, Kishor U. Masurkar, Sayalee Sawant, Vivek Tople and Shruti Wasnik, “Design, Fabrication and Analysis of Low Speed Water Tunnel”, Journal of Material Science and Mechanical Engineering (JMSME) Volume 3, Issue 3; April-June, 2016.

[3] Stephen C. Ko, “design and construction of a water tunnel” , Fritz Engineering Laboratory Department of Civil Engineering Lehigh University Bethlehem, Pennsylvania January 1971.

[4] Ivaylo nedyalkov, “Design of Contraction, Test Section, and Diffuser for a High-Speed Water Tunnel” , Department of Applied Mechanics Division of Fluid Mechanics chalmers university of technology Gothenburg, Sweden 2012 Master’s thesis.

[5] Lincoln P. Erm and Michael V. OL, “An Assessment of the Usefulness of Water Tunnels for Aerodynamic Investigations” , Air Vehicles Division DSTO Defence Science and Technology Organisation, 506 Lorimer, St Fishermans Bend, Victoria 3207, Australia, December 2012.

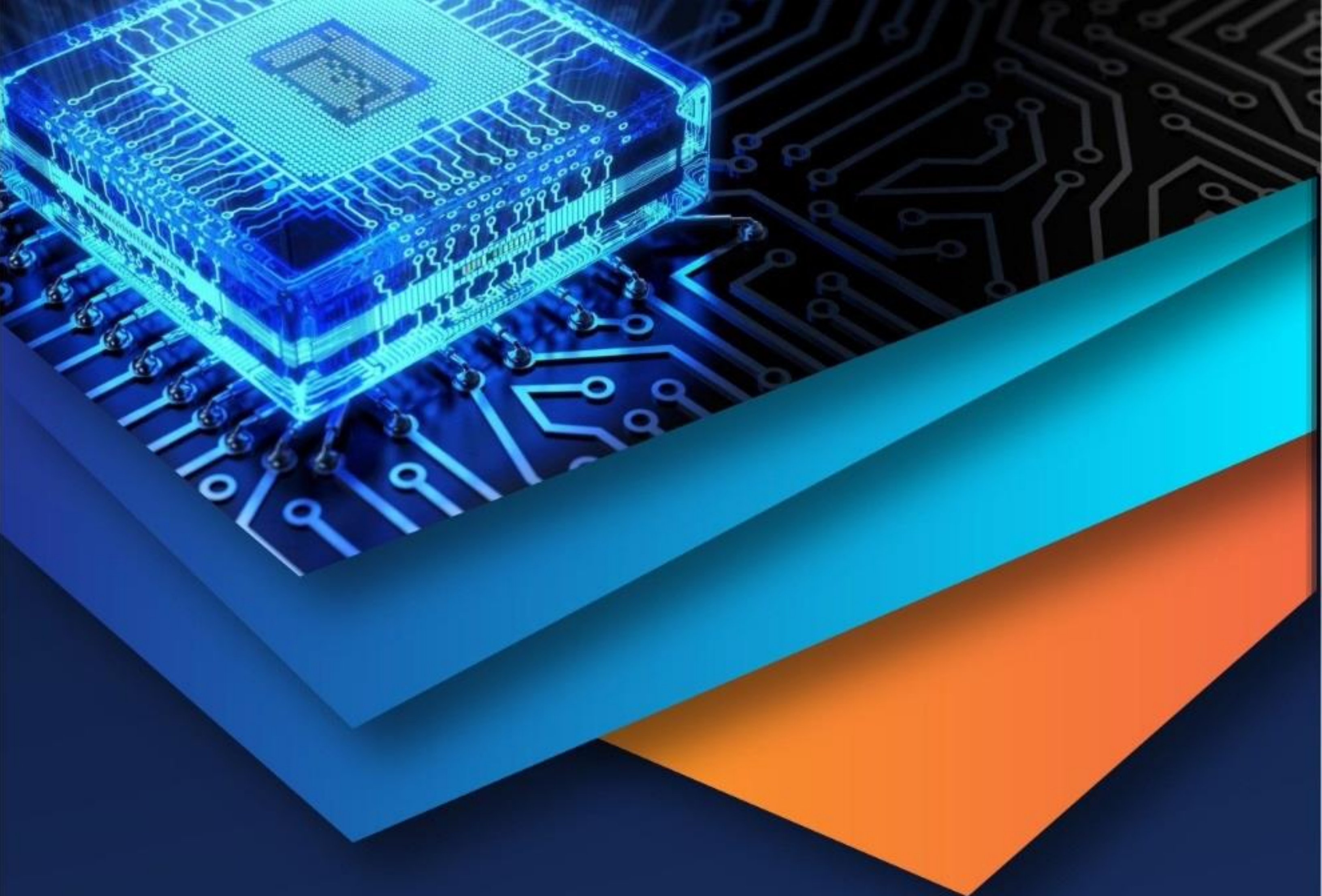
[6] Frits H. Post, Theo van Walsum, “FLUID FLOW VISUALIZATION” , Conference Paper · January 1991.

[7] JAMES W. DAILY, “The Water Tunnel as a Tool in Hydraulic Research”, PROCEEDINGS OF THE THIRD HYDRAULICS CONFERENCE Bulletin 31, University of I owa Studies in Engineering 1947

[8] J. L. Harp, V. 0. Hoehne, J. M. Allen, and G. R. Whitacre, “WATER TUNNEL DESIGN AND RELATED INSTRUMENTATION CONSIDERATIONS AND A SELECTED BIBLIOGRAPHY ON INCOMPRESSIBLE INTERNAL FLUID FLOW”, Research Branch Redstone Scientific Information Center, Directorate of Research and Development, U. S. Army Missile Command, Redstone Arsenal, Alabama.

[9] JE Sargison, AF Barton, GJ Walker, PA Brandner, “Design and calibration of a water tunnel for skin friction research”, Article in Australian Journal of Mechanical Engineering · January 2009.

[10] M Zahari and S S Dol, “Design and Development of Low-Cost Water Tunnel for Educational Purpose”, IOP Conference Series: Materials Science and Engineering.



10.22214/IJRASET

INDEX COPERNICUS 45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY