Journal of Material Science and Mechanical Engineering (JMSME)
p-ISSN: 2393-9095; e-ISSN: 2393-9109; Volume 3, Issue 3; April-June, 2016 pp. 224-230

http://www.krishisanskriti.org/Publication.html

Design, Fabrication and Analysis of Low Speed Water Tunnel

Sanket V. Kalgutkar[1], Shubham K. Khobragade[2], Kishor U. Masurkar[3], Sayalee Sawant[4], Vivek Tople[5] and Shruti Wasnik[6]

[1,2,3,4,5,6]*Priyadarshini College of Engg. Nagpur, Maharashtra, India*
E-mail: [1]sanket8791994@rediffmail.com, [2]shubhamkhobragade07@yahoo.in, [3]kishormasurkar786@gmail.com, [4]sawant.sayalee@gmail.com, [5]vivek.s.tople@gmail.com, [6]shrutiwasnik28@gmail.com

Abstract—*The low speed recirculating type water tunnel was successfully designed and fabricated. The designing was done by taking into consideration of Reynolds number, flow velocity, boundary layer formation, cavitation number and Froude's number. Theoretical and computational studies were made to decide the shape and the size of the various component of the low speed water tunnel. The setup consists of an inlet plenum with baffle and honeycomb structure to reduce turbulence or splashing, converging section, test section, wide angle diffuser and outlet module. The water was circulated using this motor and appropriate plumbing system. All the components were designed, analyzed, constructed and installed successfully. General purpose polymer was used as a ground material for the fabrication purpose.*

1. INTRODUCTION

A water tunnel is an experimental setup for studying the hydrodynamic behavior of the submerged bodies in the flowing water. It is analogous to that of the wind tunnel except that the working fluid is water. Water tunnel are used in place of the wind tunnels to calculate different forces such a lift and drag on the different test bodies. But water tunnel are widely used for visualization over the submerged bodies. It is also experimentally used for measurements of particle image velocimetry (PIV) as it is easy and sophisticated to implement it in water rather than in other working fluid. As long as the Reynolds number is controlled and well within the limits, the results are valid to calculate the details same as that in air for most of the cases. For low Reynolds number flows, tunnel can be made to run oil instead of water. The advantage is that the increased kinematic viscosity will allow the flow to be at faster speed for a lower Reynolds number.

Water tunnels belong to the category of experimental aerodynamics, as almost universally they are of scale and scheme adequate with what one finds in university laboratories. Water-tunnel model can also be built cheaper than wind-tunnel models.

2. LITERATURE REVIEW

- A test section of 10 x 15 x 70 cm was used in this water tunnel. A flow speed of 0 to 21 cm/s can be achieved with this setup by using a pump of flow rate 19.08m^3/hr. At last a desired velocity of 8 cm/s was achieved. The setup was recirculating and was used external dye injection system. [1]
- An iterative design for three dimensional contractions installed on small, low speed wind tunnel. The contraction section was developed with this method considering the laminar boundary layer condition which was justified later. [2] The contraction section was designed with the help of 5th order polynomial equation which is mention below :

$$Y_c = Y_{c1} - (Y_{c1} - Y_{c0})\left[6\left(\frac{X_c}{L_c}\right)^5 - 15\left(\frac{X_c}{L_c}\right)^4 + 10\left(\frac{X_c}{L_c}\right)^3\right]$$

- The design used an open circuit blower tunnel without exit diffuser was used. The success blower tunnel design was mainly based on data collected from exiting tunnel data.[3] The following wide angle diffuser equation is used:

A = (1.14*K+1)

3. DESIGN AND FABRICATION

3.1 Inlet Plenum

Inlet plenum acts as reservoir tank, main function of the inlet plenum is to store the sufficient amount of water. When water pumps out from inlet through pipe then water is collected in inlet plenum because the area of inlet plenum increases drastically, velocity of flow reduces and the rate of reduction is higher with respect to the diversion section. It is shown is Fig. 3.1.

3.1.1 Baffle

"Baffle" is the concept which is used to cover inlet delivery point of the water tunnel. It is useful to overcome the problems like overflow of the water. It reduces the turbulence energy of the incoming water and it gives laminar and smooth flow which is required during analysis in water tunnel.

3.1.2 Honeycomb Structure

It's purpose is to convert the turbulence flow to laminar flow and reduce the transverse component of velocity fluctuation. It is necessary to obtain the laminar flow while analyzing the models. Honeycomb must be place before converging section to obtain the laminar flow in the test section.

3.2 Convergent Section

To accelerate the velocity from low value employed for economy in recirculation to high value required in the test section and at the same time provide unifying counteractive influence on the velocity profile. The convergent section wall shape was made by using 5^{th} degree polynomial equation. The working and manufacturing dimensions and various other parameters are shown in table 3.1 and Fig. 3.2

Table 3.1: Working and Manufacturing dimensions and various other parameters

Parameters	Operational	Manufacturing
Contraction Ratio	6	
Length (L_c)	38 Cm	
Length to Height (L/H)	1.06	
Convergent Inlet Area	726 Cm^2	924 Cm^2
Convergent Outlet Area	121 Cm^2	154 Cm^2
Length to Inlet Dia. (L/D)	1.85	1.515
Height	11 Cm	14 Cm

3.3 Test Section

Following parameters were considered while designing the test section:

- Square test section was design to get equal pressure intensity as well as equal velocity distribution.
- The velocity can be regulated between the range of 2 to 8 cm/s.
- Reynolds and Froundes Number at water level of 11 cm and flow velocity of 2cm/s are 2747 and 0.0192 respectively.

3.4 Wide Angle Diffuser

A wide angle diffuser is used to reduce the length of a diffuser of given area ratio. While designing wide angle diffuser it is assume that the total pressure drop coefficient K is 2 and diffuser angle 2θ is 60°. The various design parameters of the diffuser are shown in table 3.2.

Table 3.2: Parameters of Wide Angle Diffuser

Area ratio	3.28
Diffuser angle	30°
Length of diffuser	21.733 cm
Inlet area	154 cm^2
Outlet Area	505.12 cm^2

3.5 Outlet Module

The geometry of the outlet module was design properly. If any irregularities occur in the model it leads to the reverse flow or back pressure in the system which will disturb the results in the test section. Outlets with two opening are designed so that the water flow is distributed equally. The curved surface after the diverging section guides the flow towards the holes thus making a smooth exit of the flow and prevents any back pressure in the system.

3.6 Polymer Sheet Making

- The general purpose BQTN polymer was mixed with hardener and accelerator in the proportion of 40:3:1.
- A net of fine grid with the dimension of mould was placed inside the polymer to provide it extra support and strength.
- When the solution was in a state of semi-solid, it was taken out of the mould and fixated on the PVC wall of the water tunnel.
- The acrylic glass were then attached to the test section and are allowed to dry for at least 24 hours before any further was carried out.
- Fig. 3.3 and Fig. 3.4 show Polymer sheet and actual fabricated model respectively.

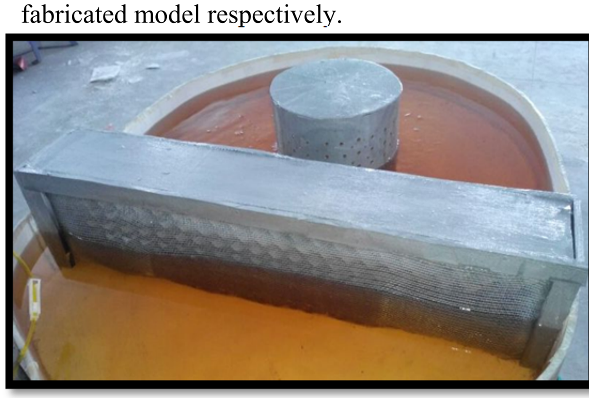


Fig. 3.1: Shows Baffle and Honeycomb Structure

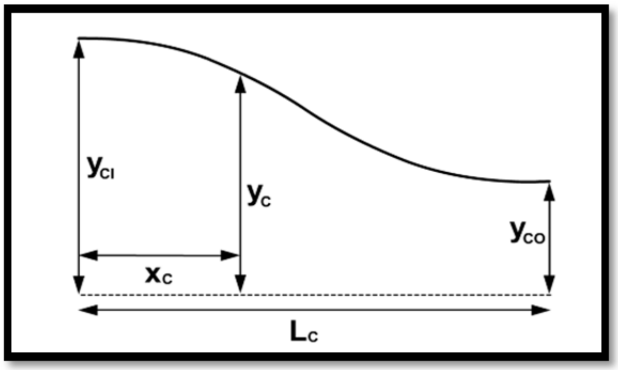


Fig. 3.2 Schematics of Contraction Shape

Fig. 3.3 Shows Polymer Sheet

Fig. 3.4 Actual Fabricated Model

4. EXPERIMENTAL RESULTS

Flow visualization over different test bodies such as Cylinder, NACA 2412, Delta, and Nozzle section was performed using dye injection system. Food liquid dye was used to obtain the flow pattern of the flowing fluid. Dye was injected through a syringe or a needle which was aligned to the local flow velocity. To obtain the streamline flow, flow velocity of the dye must match with the flow velocity of the water. **Fig. 4.1** Shows Laminar Flow in Test Section

4.1 Cylinder

In Fig. 4.2 fluid is passed over the cylinder. Initially flow is laminar, but due to abrupt resistance provided by model the flow gets turbulence and swirls are formed behind the cylinder. The coefficient of drag at 2 cm/s was 0.0433.

4.2 NACA 2412

In Fig. 4.3 laminar flow on upper surface of airfoil is observed up to maximum camber. Flow separation takes place near the trailing edge due to loss of kinetic energy of fluid on a body. Flow separation takes place near the trailing edge due to loss of kinetic energy of a fluid on the body. It is observed that flow separates near the leading edge as we increase as angle of attack. The coefficient of drag at 2 cm/s was 0.025 at zero angle of attack.

4.3 Delta

In Fig. 4.4 flow gets separate at its leading edge as flow passes over model, were velocity of flow get reduced. When flow leaves the trailing edge, vortices are formed which increases dead space behind the model. The coefficient of drag at 2 cm/s was 0.0360.

4.4 Nozzle

In Fig. 4.5 the variation of flow over different cross section in nozzle is observed and fluid follows converging path and the fluid gets distorted at minimum area which results in increase in velocity. As a result it led to increase in Reynolds number and turbulence.

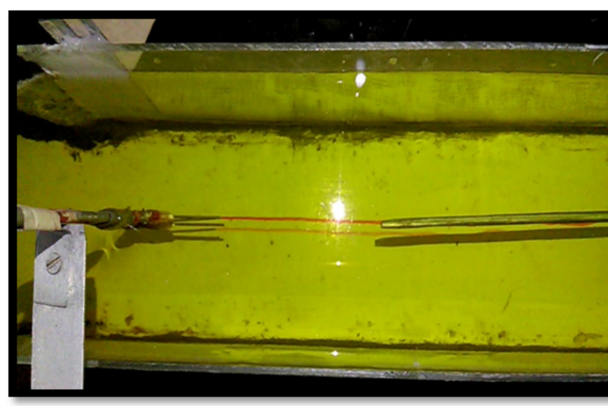


Fig. 4.2: Shows Laminar Flow in Test Section

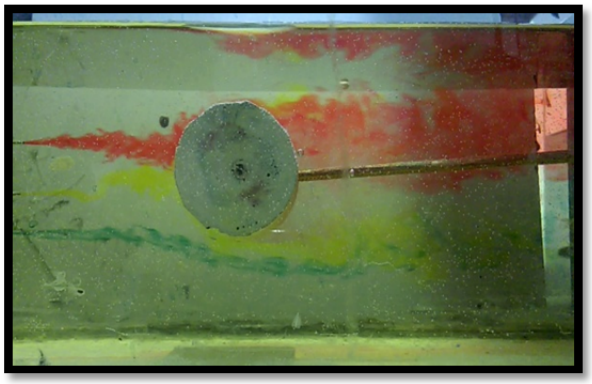


Fig. 4.2 Flow over Cylinder

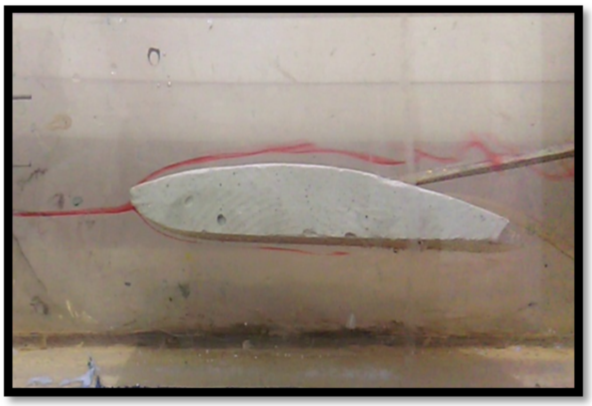


Fig. 4.3 Flow over NACA 2412

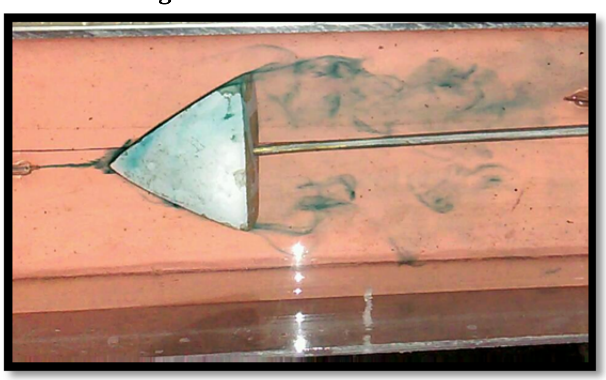


Fig. 4.4 Flow over Delta

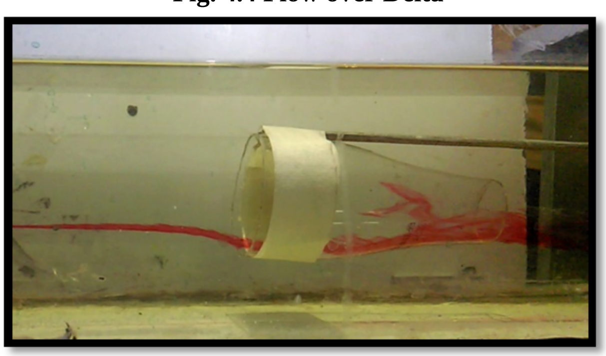


Fig. 4.5 Flow inside Nozzle Section

5. COMPUTATIONAL ANALYSIS

CFD analysis of full scale 3-D model of water tunnel and different test model was performed. All the results were compared with the experimental results.

5.1 Three Dimensional Computational Analysis

Numerical Analysis of low speed water tunnel was performed in commercial software's to observe the change in parameters in computational and experimental conditions. Further the changes in pressure and flow patterns were observed starting from inlet plenum to outlet module. Contours were display to show the results of different parameters and graphs were plotted for the same. Computational analysis was performed to finalize the design of low speed water tunnel.

5.1.1 Grid Generation

The basic step in computational analysis of 3-D model is to create geometry in commercial CAD software and save it in .IGS file format. Required modifications were made in the geometry. Spilt the geometry into map able volumes to generate quality mesh in commercial pre-processing software. Unstructured meshing was performed of Tet/Hybrid scheme and their type is of TGrid type. First length of 10^{-3} was defined at inlet and outlet to generate fine meshing and to obtain the required flow pattern. Worst Equiangle skew found is 0.907494 and aspect ratio of 6.7753 both of which lie within the safe limits for the total elements of 2079924. The boundary conditions for the mesh model were defined as follows:

Faces	Boundary Type
Inlet	Mass flow Inlet
Outlet	Outflow
Base Wall	Wall
Side Wall	Wall
Top Wall	Wall

After defining the boundary conditions export the file to the commercial processing software and save the file with extension as. msh.

5.1.2 Results

In post processing the computational results are verified with experimental results. Plot the results with respect to position along x – axis against velocity magnitude (m/s) from which velocity at different locations can be known and can be verified experimentally. Velocity streamline pattern can also be observed in the contours for proper flow pattern of water in the model.

5.2 Two Dimensional Analysis

Test over different models such as cylinder, NACA 2412, Delta and nozzle are carried out in commercial software to analyse flow pattern over different shapes and compare them with the actual test carried out in water tunnel.

5.2.1 Grid Generation

- **Cylinder:** The aspect ratio of 5.8898 and equiangle skew of 0.493626 and total number of elements were 47300.
- **NACA 2412:** The aspect ratio of 100.84 and equiangle skew of 0.76337. The total number of elements was 130050.
- **Delta:** The aspect ratio of 377 and equiangle skew of 0.34 and total number of elements were 190810.
- **Nozzle:** The aspect ratio of 2.15764 and equiangle of 0.1525 and total number of elements were 12000.

5.2.2 Results

The following figures show the computational results of the water tunnel setup, cylinder, NACA 2412, Delta and nozzle respectively.

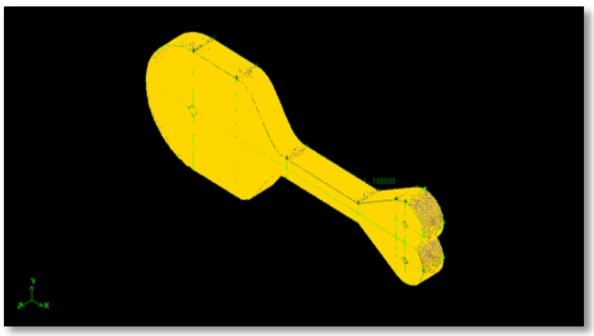


Fig. 5.1 Unstructured Mesh

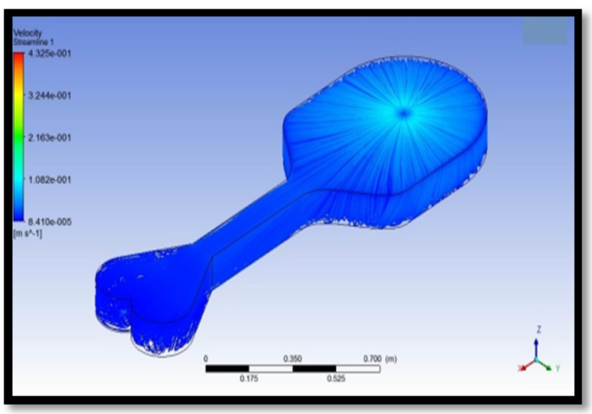


Fig. 5.2Shows Velocity Streamline Pattern

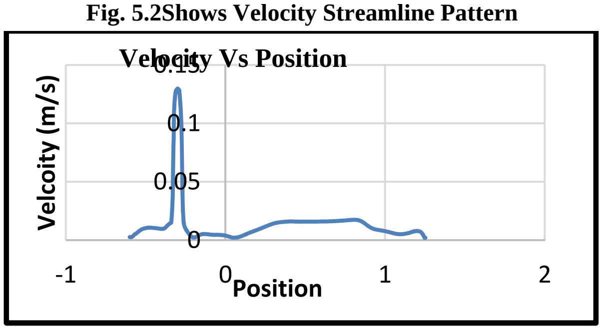


Fig. 5.3 Shows Velocity Magnitude vs. Position Graph

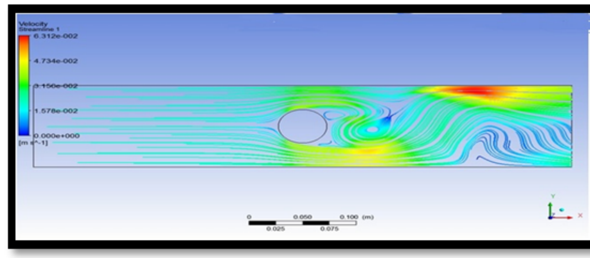


Fig. 5.4 Velocity Streamlines over Cylinder

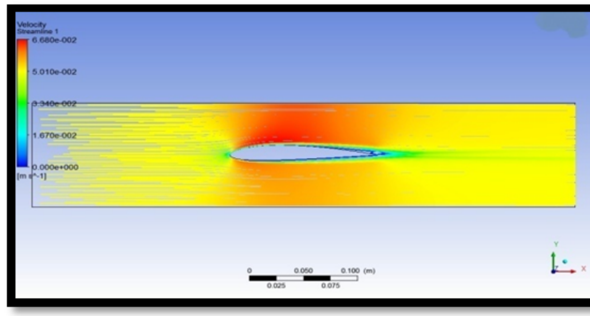


Fig. 5.5 Velocity Streamlines over NACA 2412

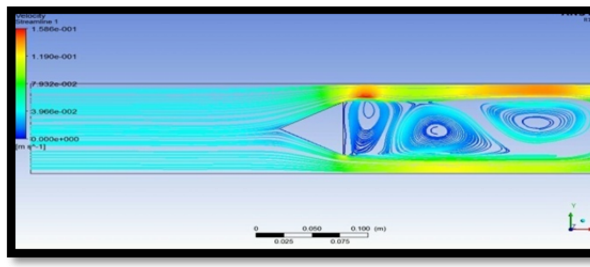


Fig. 5.6 Velocity Streamlines over Delta

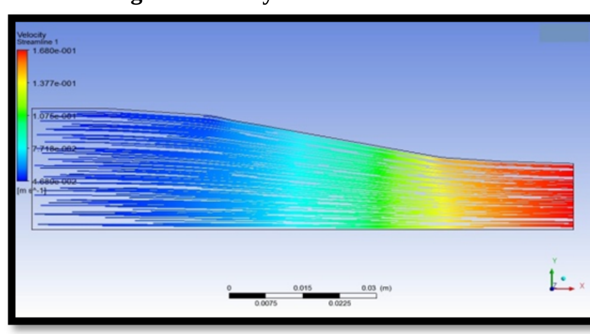
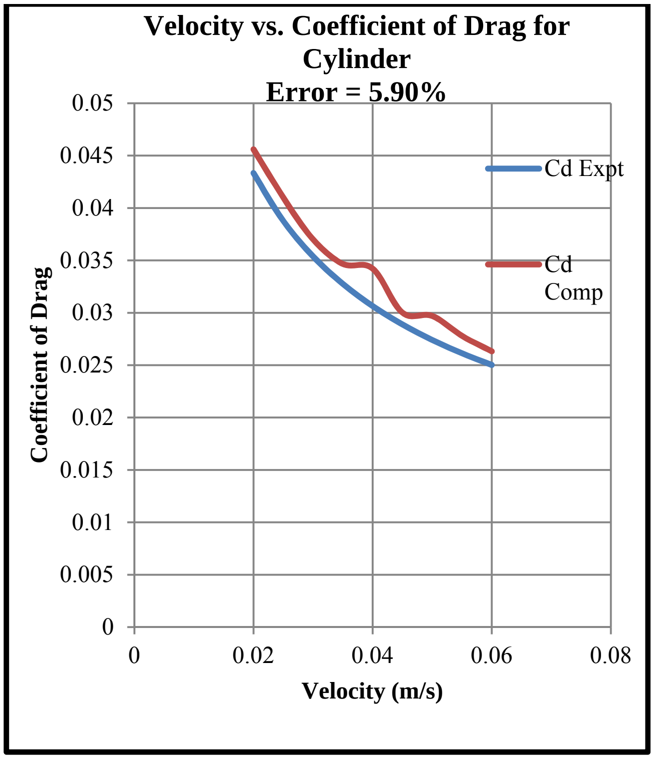
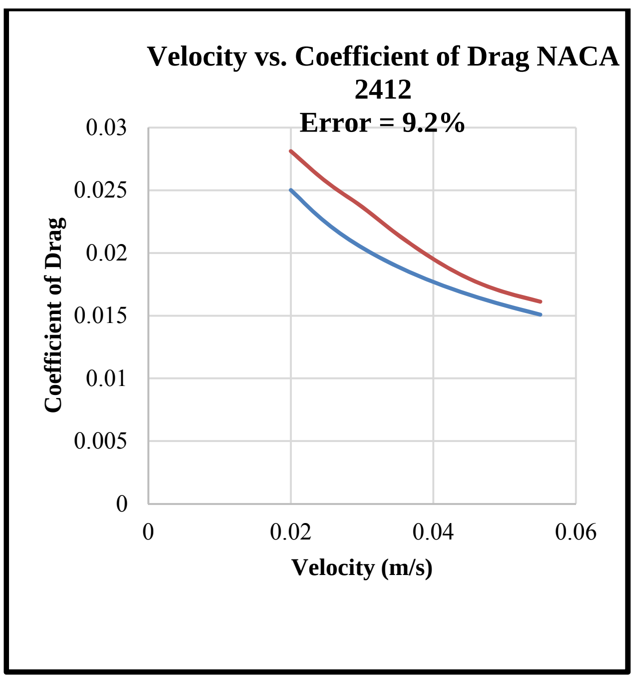
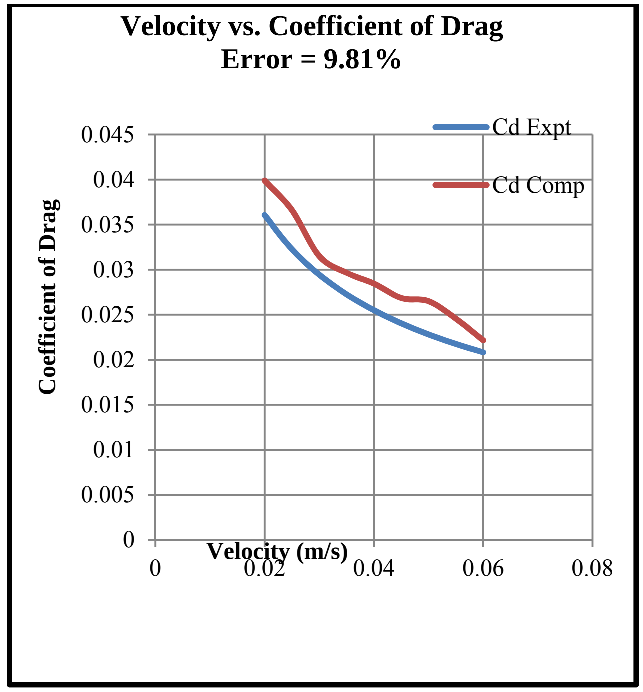


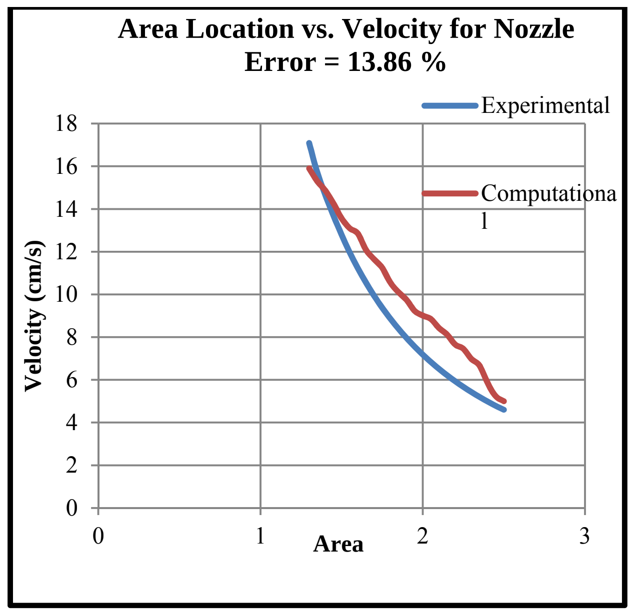
Fig. 5.7 Velocity Streamlines inside Nozzle Section

6. COMPARISON OF COMPUTATIONAL AND EXPERIMENTAL RESULTS









7. ACKNOWLEDGEMENT

This work was supported by Department of Aeronautical Engineering, Priyadarshini College Of Engineering, Nagpur under the guidance of Asstt. Prof. Pawan Kumar Karn and Asstt.Prof. Manoj Kharbade.

REFERENCES

1. Harshad Kalyankar, Design and Analysis of Low Speed Water Tunnel for Flow Visualization of Bluff Body-2015.

2. J.H.BELL and R.D. Mehta, contraction design for small low-speed wind tunnels. NASASTI/Recon Technical Report N, 89: 13753, August 1988.

3. Ranjan basak & Deboj yoti Mitra, Wide Angle diffuser Design using Two Cubic Profiles for open circuit wind Tunnel.

4. John F Ripken "Design studies for a closed- jet water tunnel" Technical paper no.9 series B, University of Minnesota, August-1951.

5. Stephen C. Ko "Design and construction of water tunnel" Lehigh, University Bethlenem, Pennsylvania, January 1971.

6. University Desktop Water Tunnel Model - 0710 by Rolling Hills Research Ltd.

7. Water Tunnel Turbulence Measurements behind a Honeycomb" by Basil E. Robbins.

8. Fluent Inc, GAMBIT 2.4 modeling Guide, 2007.