Study of different contraction design of wind tunnel for better performance by using CFD

Mr. Shrimali Darshan K.[1], Mr. Hardik B.Patel [2]

[1]Student, Dept. of Mechanical Engineering, Gokul Global university, Gujarat, India
[2]Assistant Professor, Dept. of Mechanical Engineering, Gokul Global university, Gujarat, India

---***---

Abstract - *The wind tunnel is one of the most common experimental testing facilities for the testing of fluid flow. The main aim of present work is to study effect of different configuration of wind tunnel on flow uniformity, flow separation and pressure gradient. This project aims to propose the different configuration for wind tunnel design using CFD tool and investigated experimentally. The contraction, test section and diffuser section were studied and literature is done to propose modified design numerically using CFD tool and validating it experimentally..*

Keywords:- **Wind tunnel,Diffuser, Test section, Contanction ratio, CFD**

1. INTRODUCTION

Even with today's computers, a wind tunnel is still an essential engineering tool for model tests, basic experimental research and computer code validation. Since the 193Os, when the strong effect of free-stream turbulence on shear layer behaviour became apparent, emphasis has been laid on wind tunnels with good flow uniformity and low levels of turbulence and unsteadiness. In the past, it has been difficult to devisefirm rules for wind tunnel design mainly due to the lack of understanding of flow through the various tunnel components. The first attempt at providing some guidelinesfor the complete design of low-speed wind tunnels was that due to Bradshaw and Pankhurst (1964). However, recent experimental studies of flow through individual components of a wind tunnel (Mehta, 1977, 1978 and Mehta and Bradshaw, 1979) haveled to increased understanding and design philosophy for most of the components withthe notable exception of contractions.

The first flow experiments arose around 1700 using small fans with a test object in frontof it (Pope and Harper, 1966). The fans gradually expanded to wind tunnels by adding more parts, such as a closed test sections and flow straighteners, to the design. In the 20th century, they got into the shapes as they are known nowadays. Alongside wind tunnels, computers started to gain popularity in the 1970's and 80's. It was expected that computer simulations would soon replace the wind tunnel experiments (Barlow et al., 1999, Moonen et al., 2006a). However, up to this date the physics of turbulent flows is not yet fully understood. Therefore computational data are simplifications of the reality and wind tunnel studies are needed to validate models.

2. OBJECTIVE

1 Study of different contraction design of wind tunnel for better performance by using CFD

To design wind tunnel with goals illustrated as below

A) flow uniformity in test section.
B) absence of flow separation (controlling Pressure Gradient in the Contraction).

2 Re-Designing contraction shape using numerical method and finding optimum shape using CFD simulation.

3 Investigating effect of placement of trip wire screen on the pressure distribution in contraction.

3. PROPOSED METHODOLOGY

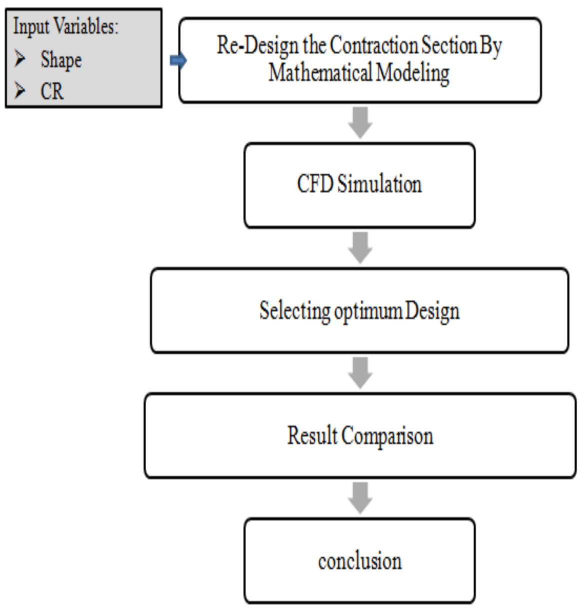


Fig-1: Proposed Methodology for present work

4. EXISTING WIND TUNNEL DESCRIPTION

The existing wind tunnel facility presented in this work is an open circuit type driven by variable speed controlled axial fan placed at the end of diffuser section with following configuration

Table-1: Wind tunnel setup configuration.

Bell-mouth Inlet cross section	90 cm X 90 cm
Contraction Inlet cross section	70 cm X 70 cm
test section inlet /Contraction outlet	30 cm X 30 cm
CR(Contraction Ratio)	9
Length of contraction	130 cm
Test section length	100 cm
Diffuser length	200 cm
Diffuser outlet diameter	54 cm
Air flow speed range	0-25 m/s
Suction motor	2.30kw(3 Hp)

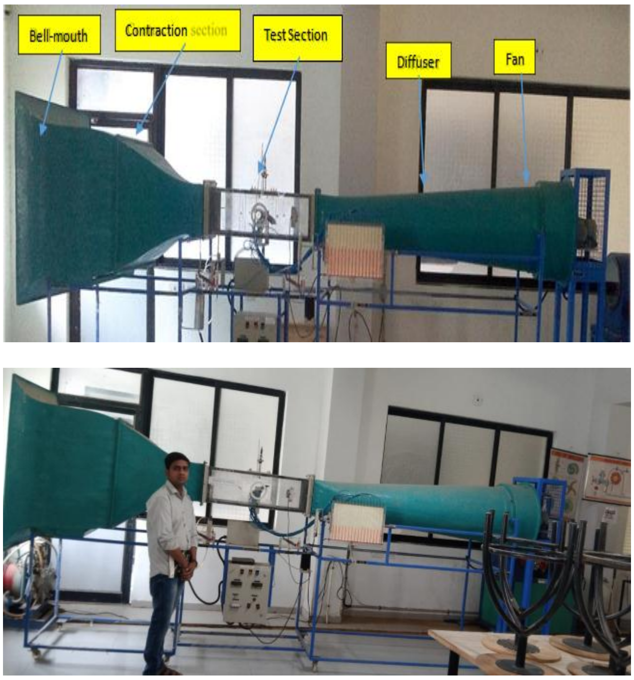


Fig-2: Open circuit Wind Tunnel Setup.

5. PROBLEM DEFINATION

From Bernoulli's Equation

$P1 + ½\, \mathcal{P}\, V_1^2 = P2 + ½\, \mathcal{P} V_2^2$

But, V_2= Stagnation point velocity = Zero

So, P_2-P_1 = $½\, \mathcal{P}_{air}\, V_1^2$

$V_1 = \sqrt{2\Delta P/\mathcal{J}}$

$\Delta P = P_2 - P_1 = \mathcal{P}_{water} * g * \Delta h$

$V_1 = \sqrt{2(\mathcal{P}water * g * \Delta h/\mathcal{P}air}$

Sample Calculation: Velocity:- 20 m/s

$V_1 = \sqrt{2(\mathcal{P}water * g * \Delta h/\mathcal{J}}$

$V_1 = \sqrt{2(1000 * 9.81 * 25.50/(1.23 * 10}$

V_1 = 20.08 m/s

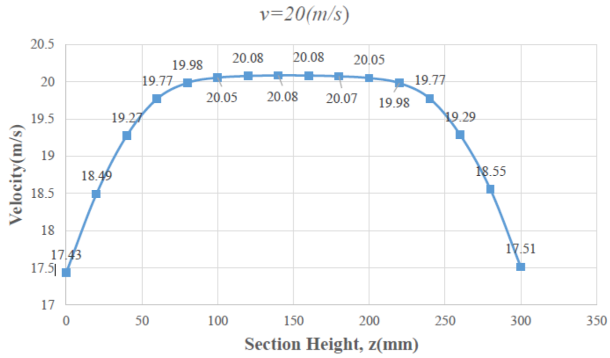


Fig-3: Velocities at different test section height for v=20m/s.

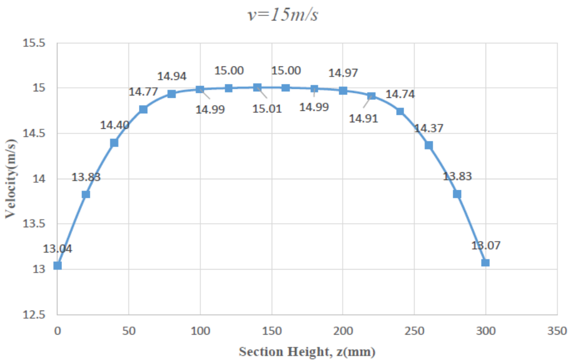


Fig-4: Sectional Velocity distribution for v=15m/s.

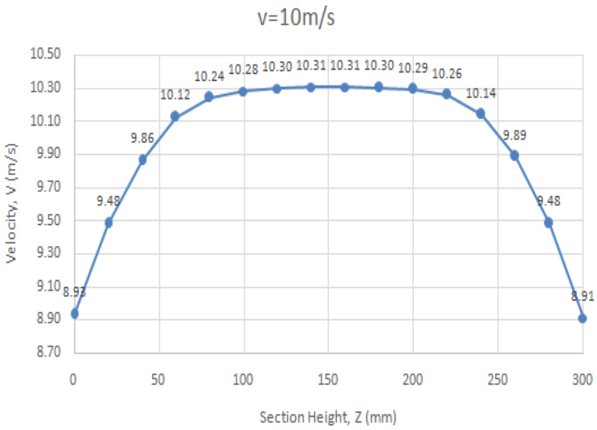


Fig-5: Sectional Velocity distribution for v=10m/s.

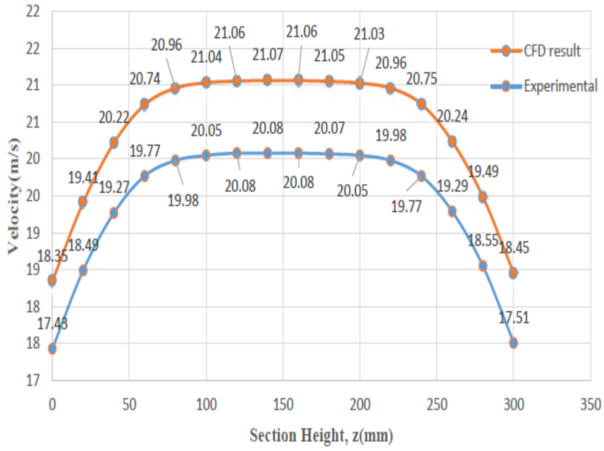


Fig-6: Experimental and CFD Velocity result comparison.

5.1 UNCERTAINITY IN VELOCITY MEASUREMENT

Table-2: Uncertainity of pitot tube measurement.

Anemometer velocities(m/s)	Pitot-tube velocity (m/s)	% error
5	2.186	-3.72
10	10.3306	-3.306
12.9	13.0198	-0.9286
14.2	14.0630	+0.97717
17.2	16.8085	+2.2762
19	18.5128	+2.5642
20	19.4031	+2.9845

5.2 CFD MODELLING OF EXISTING SETUP AND VALIDATION.

CFD model is generated by applying boundaries conditions such that at mid- section of test section we get atmospheric pressure and desired velocity assuming temperature and density of air remains unchanged. And result is compared withexperimental one as shown in figure 1.9 and found that CFD predicts somewhat high value of velocity throughout the section and it is so because in model we have not considered a trip-strip placed in bell-mouth of setup which offers some pressure drop.

6. COMPUTATIONAL MODELS

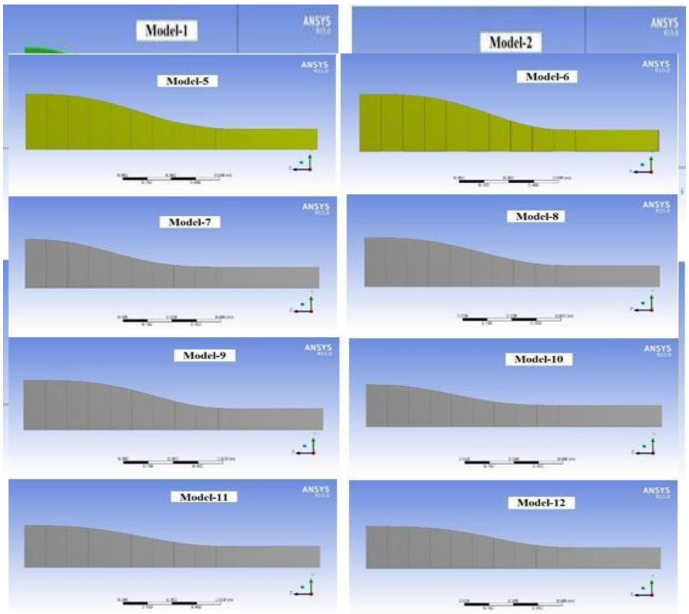


Fig-7: 3D Models

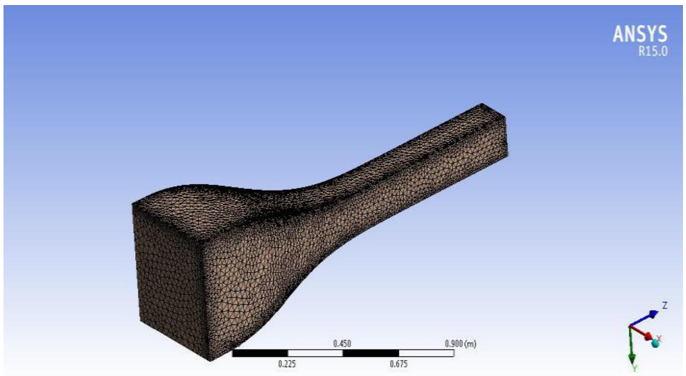


Fig-8: Meshed model

Table-3: No. of nodes and elements of meshed model

Mesh Method	Nodes	Elements
Tetrahedron	18054	87067

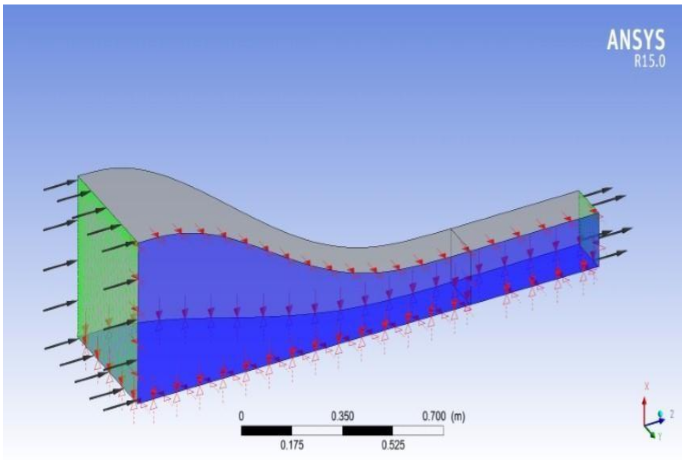


Fig-9: Boundary conditions applied to model in ANSYS CFX.

Figure 5 shows inlet, outlet boundary conditions by arrows, and wall boundary by grey and symmetry boundary by blue colour. And the conditions applied to every boundaries is summarised in above report exported by ANSYS. The Reynolds Shear Stress Transport (SST) model of turbulence was used with a specified turbulence level of 1%.

7. COMPUTATIONAL RESULTS

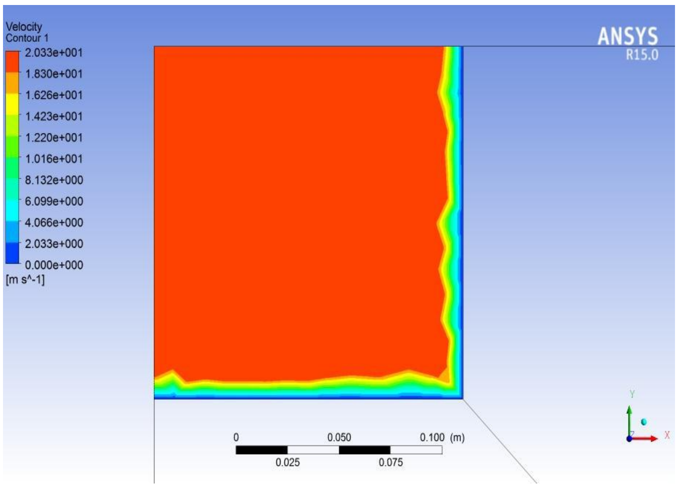


Fig-10: Velocity distribution at mid-test section

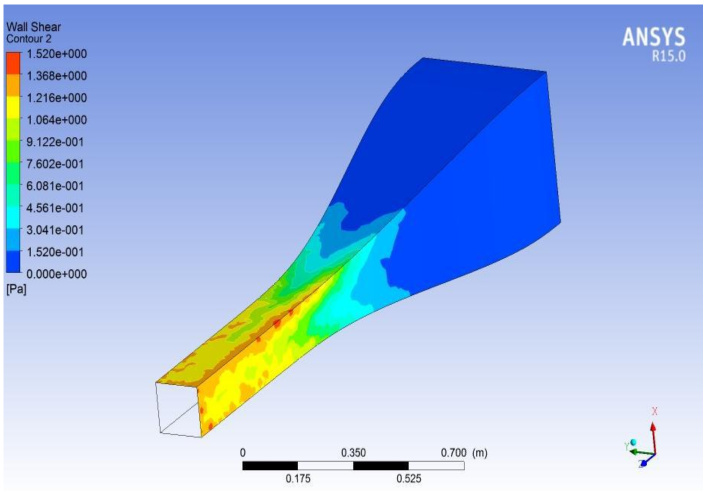


Fig-11: Wall shear (Pa) plot

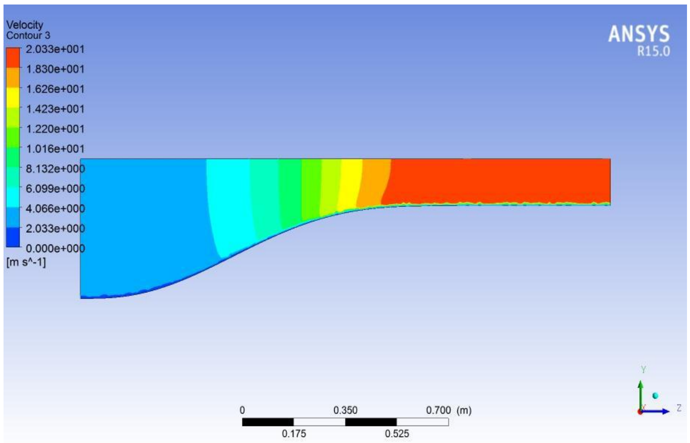
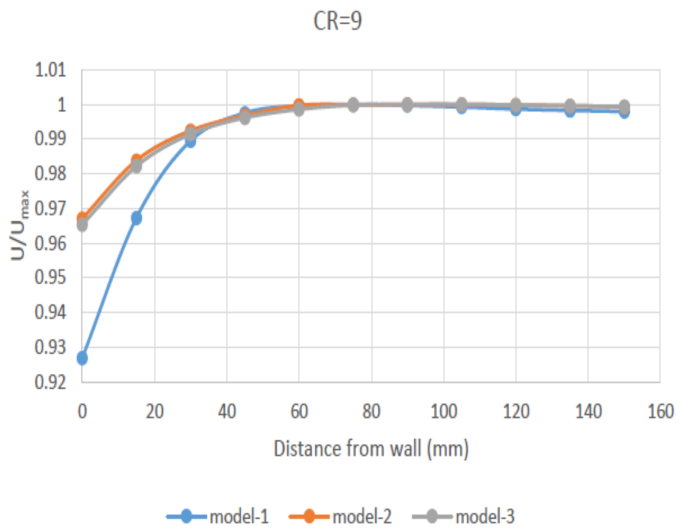


Fig-12: Velocity contour at mid-plane of contraction



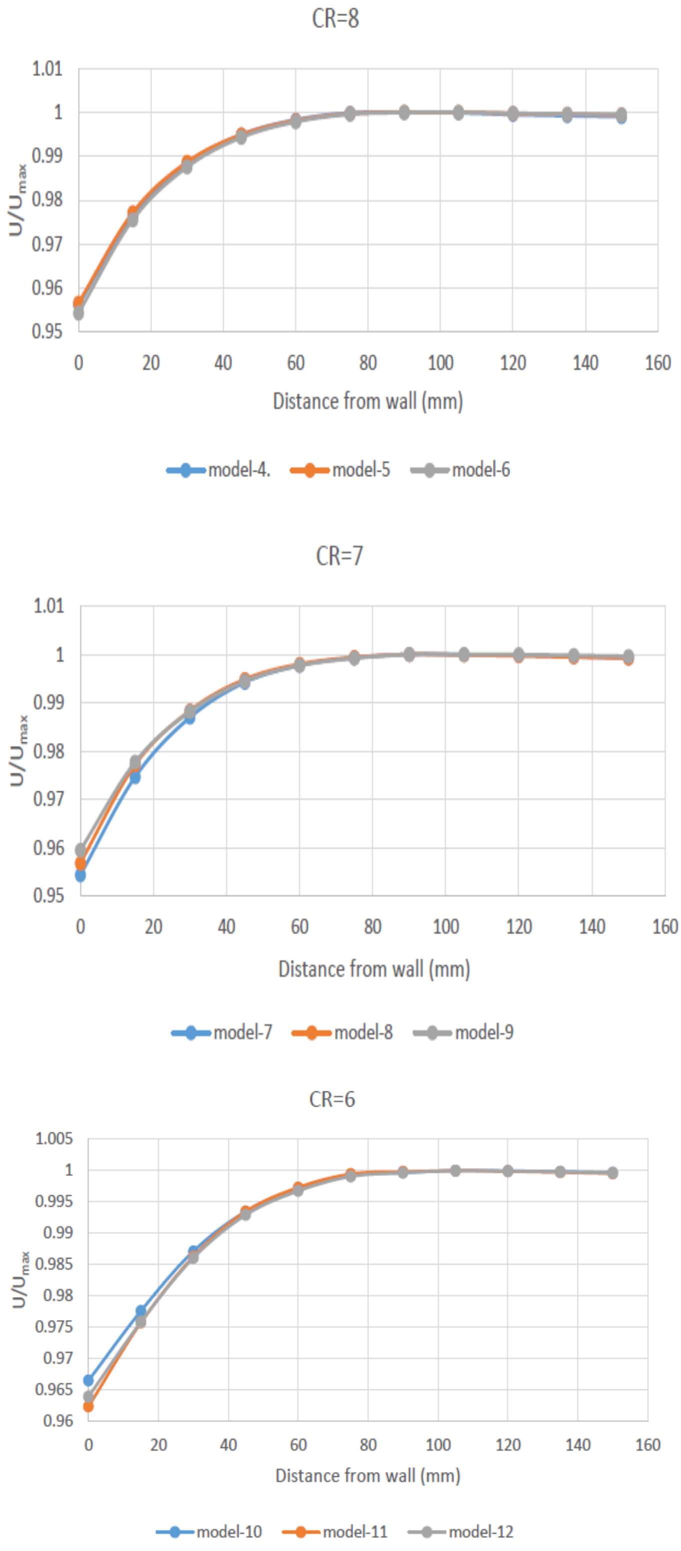


Fig-13: Velocity profile at mid working section on horizontal plane

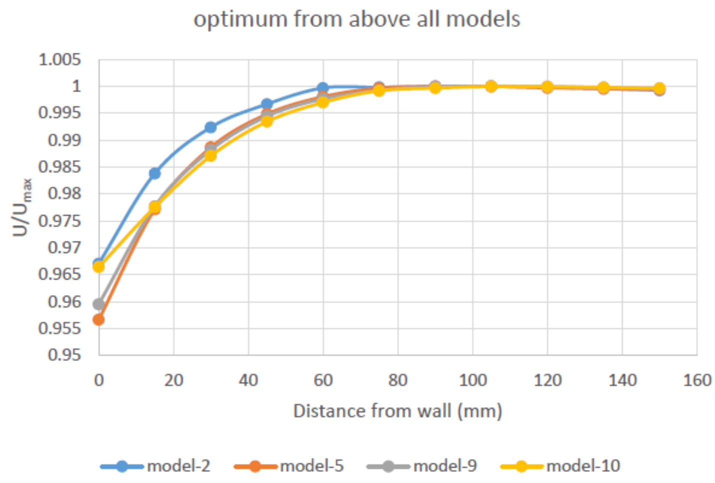


Fig-14: Optimum models from all twelve models

8. CONCLUSION

CFD has been used to optimize the design of a wind tunnel contraction. The use of CFD has increased the flexibility of shapes considered, and allowed the use of a sixth order polynomial to define the profile. The parameters of the profile that were varied were the location of the point of inflection and the Contraction Ratio. The CR 6 is used and contraction fabricated is of rectangular-to-square type now. Results show that the general behavior of the flow in the regions away from the wall is in reasonable agreement with the predicted behavior. Velocity uniformity get disturbed at corners. And at walls it deviates from CFD results this so because we have considered smooth wall which is impractical. Physical calibration of the facility has validated the CFD methods used and demonstrated that the technique can be used for future wind tunnel designs.

Further work can be done to know effect of placement of mesh-screen like honeycombat inlet of contraction mouth on velocity uniformity. Even one can work by placing guiding channel in inlet contraction to make flow parallel in test section mentioned in literature.

REFERANCES

[1] Ismail, Johanis John , Erlanda A. Pane , Budhi M. Suyitno , Gama H.N.N. Rahayu , Damora Rhakasywi ,Agri Suwandi "Computational fluid dynamics simulation of the turbulence models in the tested section on wind tunnel"(2020)

[2] Jiao Lei, Pengcheng Huang , Linhe Zhang , Yukui Yuan , Wenyang Deng , Shaohua Mao, Jun Zhang "Experimental study on flow characteristics of a large-scale open jet wind tunnel for outdoor pool fire research" (2021)

[3] Hao Su , Haoran Meng , Timing Qu , Liping Lei " Wind tunnel experiment on the influence of array configuration on the power performance of vertical axis wind turbines"(2021)

[4] Leifur Leifsson, Slawomir Koziel "Simulation-driven design of low-speed wind tunnel contraction" (2015)

[5] Keum-Yong Park, Yeol-Hun Sung, Jae-Hung Han, "Development of a cable suspension and balance system and its novel calibration methods for effective wind tunnel tests", (2020)

[6] Guiquan Fu, Xianying Xu , Xiaona Qiu , Gaoxing Xu , Wen Shang, Xuemei Yang, Peng Zhao, Chengwu Chai, Xiaoke Hu, Yunian Zhang, Qiangqiang Wang, Chuanyan Zhao, "Wind tunnel study of the effect of planting Haloxylon ammodendron on aeolian sediment transport", (2021)

[7] WeiYi, PengZhou, YiFang, JingwenGuo, SiyangZhong, XinZhang, XunHuang, GuochengZhou, BaoChen, " Design and characterization of a multifunctional low-speed anechoic wind tunnel at HKUST" , (2021)

[8] A.S. Abdelhamed, Y.El-S. Yassen , M.M. ElSakka, " Design optimization of three dimensional geometry of wind tunnel contraction, (2014)

[9] María Rodríguez Lastra , Jesús Manuel Fernández Oro,Mónica GaldoVega, Eduardo Blanco Marigorta, Carlos SantolariaMorros , "Novel design and experimental validation of a contraction nozzle for aerodynamic measurements in a subsonic wind tunnel", (2013)

[10] Ling Jin , Yunsong Gu , Xiao Bing Deng , Haisheng Sun , Tingrui Yue , Junlong Zhang, "Standing wave and its impact on the low-frequency pressure fluctuation in an open jet wind tunnel,(2020)

[11] Roberto Merino-Martínez , Alejandro Rubio Carpio , Lourenço Tércio Lima Pereira , Steve van Herk , Francesco Avallone , Daniele Ragni , Marios Kotsonis, "Aeroacoustic design and characterization of the 3D-printed, open-jet, anechoic wind tunnel of Delft University of Technology", (2020)

[12] Ismail , Johanis John , Erlanda A. Pane , Budhi M. Suyitno , Gama H.N.N. Rahayu, Damora Rhakasywi , Agri Suwandi, "Computational fluid dynamics simulation of the turbulence models in the tested section on wind tunnel", (2020)

[13] Hrvoje Kozmar , Boris Laschka, "Wind-tunnel modeling of wind loads on structures using truncated vortex generators", (2019)

[14] J. McCarthy , T. Teske , S. Lam , M. Jones, "Preliminary assessment of surface pressure measurements on a metallic, additive manufactured wind tunnel model", (2020)

[15] W.C. Niu , Y.L. Ju, " System design and experimental verification of an internal insulation panel system for large-scale cryogenic wind tunnel", (2021)