Ilkogretim Online - Elementary Education Online, 2021; Vol 20 (Issue 3): pp. 3351-3360
http://ilkogretim-online.org
doi: 10.17051/ilkonline.2021.03.348

Design And Analysis Of Tabletop Wind Tunnel

Shashank Bijalwan [1], Ritvick Pundir[1], Armaan Mansoor[1], Akash Sawant[1], Rajesh P Verma[1,*], Ritvik Dobriyal[1], Dr Sisir Kumar Jeena[2]

[1]Department of Mechanical Engineering, Graphic Era Deemed to be University, Dehradun, 248002, Uttarakhand, India.

[2] Assistant Professor, Department of Computer Science and Engineering, Graphic Era Hill University, Dehradun.

[*] rajesh_diva1@yahoo.in

Abstract. The purpose of this study is to explain the comprehensive process of designing and analysing a low-speed open-circuit table-top wind tunnel. All of the preceding research work's principles were followed in the creation of this wind tunnel. The wind tunnel is constituted of a contraction chamber, a test section, and a diffuser. This design contains a square test section with a side length of 340mm and an on-design velocity of 10m/s inside the test section. The overall pressure loss inside the table-top wind tunnel is computed, as is the minimum power sufficient to run the system. The 2D axis-symmetric modelling is done in "CATIA V5" design software, while the numerical flow simulation is done in "ANSYS Fluent".

INTRODUCTION

A wind tunnel is a tool for analysing the flow pattern formed by the object's body inside the test section. It is also used to monitor the pressure, temperature, and flow velocity surrounding the object. The table-top wind tunnel is an open-circuit wind tunnel, which means that the room's ambient air enters from the Contraction chamber and leaves from the Diffuser after passing through the tunnel. These wind tunnels have their practical applications such as anemometer calibration and analysing of the thermal characteristics of heat-producing items such as circuit boards and electronic components. They can also be used for aerodynamic studies on a smaller scale than their larger counterparts.

Bell et al. [1] tried to create an optimum contraction chamber for a small-scale low-speed wind tunnel. The VSAERO computer software is used to calculate the wall pressure distribution and the wall velocity whereas the PDMINT boundary code is utilised to calculate the generation of separation bubbles in the contraction chamber. In his investigation, he discovered that contraction ratios between 6 and 10 are best suited for small slow-speed wind tunnels. The conclusion was reached that the contraction's length to inlet height ratio is almost equal to one.

Maurya et al.[2] examined the design and fabrication of the low-speed wind tunnel, concluding the significance of the shape, size, and material chosen for flow analysis, i.e. calculating lift and drag on the object. The study also concludes that to improve the performance of the airfoil structure inside the test section, we must maximize the lift force while minimizing the drag force. Siddique et al. [3] redesigned a low-speed open type wind tunnel which was used for experimental inspection and demonstration of a compressed flow phenomenon. The design of the wind tunnel was designed that can operate in a test section of 84mph in (200 mm X 200 mm). The model of the wind tunnel in CFD was created and analysed in Fluent13 using RNG K-epsilon turbulence model with the scalable wall function.

Hussain et al. [4] done calculations for the design, construction, and testing of a low-speed wind tunnel with measuring and inspection equipment. The planned wind tunnel features a test section with a cross-section area of 0.7 m X 0.7 m and a length of 1.5 m. The greatest speed for an empty test segment was around 70 m/s, with a contraction ratio of 8.16. Mauro et al. [5] attempted to design, construct, and calibrate a small-scale open circuit wind tunnel. They used a 5 cm X 5 cm test segment with a wind velocity of 6 m/s to detect turbulence, which was roughly 0.4 percent. The experiments were carried out using the NACA 0012 aerofoil to check the wind tunnel's compatibility with the aerodynamic design and scientific literature.

Shahrukh et al. [6] has also done outstanding work in the design, fabrication, and testing of a small-scale low Reynolds number table-top wind tunnel. They built a table-top wind tunnel with a maximum flow velocity of 7m/s and a Reynolds number of 71000. They created the model in CATIA V5, which has a honeycomb structure and a sequence of displays, and then simulated it in ANSYS fluent. The entire table-top wind tunnel was made of mild steel for cost effectiveness and ease of use, and the test section was made of transparent acrylic glass for flow visibility. The pitot-static probe was used to calibrate the test portion.

Magryta et al. [7] conducted a study in an open circuit wind tunnel to investigate the lift force generated by an air foil with a velocity range of up to 28 m/s. According to the findings, active airflow has little effect on lift and drag force. The first step of his research includes testing without active air flow to validate the simulation computation, and the second phase of research assumes wind tunnel testing to fully utilise bench testing of active airflow at various overpressure and vacuum values.

We concentrated on determining the total pressure loss within the table-top wind tunnel and the least power required to drive the system using the previously applied method in this study. Furthermore, the paper is divided into sections, with section 2 containing the methodology that discusses the wind tunnel design. Section 3 has the calculations, followed by the results and conclusions, and the last section contains the references.

METHODOLOGY

Test Section

The most significant portion of the wind tunnel is the test section (fig.1 (a)), which is where all of the experiments take place and where the object is positioned. It is the initial system component to be designed. The test section design includes the geometry of the test chamber, its basic dimensions, and the intended wind velocity (table 1). The size of the test section is determined by the object that will be placed inside it. The length of the test section must be 0.5 to 3 times the hydraulic diameter of the section for uniform intake airflow [8]**.** If the length of the test section is less than 0.5 times the hydraulic diameter, boundary layer separation will occur at the exit of the test section, while lengths greater than 3 times the hydraulic diameter will result in an increase in boundary layer thickness inside the test section.

The hydraulic diameter (D_h) is calculated by the formula –

$$D_h = 2\sqrt{\frac{\Omega_{T_S}}{\pi}} \tag{1}$$

where, Ω_{T_s} is the cross-section area of test section.

Table 1. Specification of test section

Inlet cross section area	150mm X 150mm
Outlet cross section area	150mm X 150mm
Length	340mm
Hydraulic diameter	169mm
Velocity	10m/s

Contraction Chamber

This component's purpose is to accelerate the flow of wind into the test section chamber and produce a uniform flow velocity profile. It is the most difficult aspect of the system. The contraction chamber should be designed (fig. 1 (b)) to meet all of the relevant standards while being as short as possible (table 2). The ratio of the cross-section area of the contraction chamber's intake and exit should be between 6 and 10. If the ratio is less than 6, the contraction chamber has substantial pressure loss, whereas a ratio larger than 10 indicates that the entrance cross section dimension is excessive [1]. The ratio of the length to the side-length of the contraction chamber should be between 0.667 and 1.79. If the ratio is less than 0.667, an air flow detachment will occur just before the contraction outlet cross-section, whereas a ratio greater than 1.79 will result in a rise in boundary layer thickness [1].

Table 2: Specifications of contraction chamber

Inlet cross section area	400mm X 400mm
Outlet cross section area	150mm X 150mm
Length (L_0)	400mm
Area ratio	7.1
Length and side-length ratio	1

Diffuser

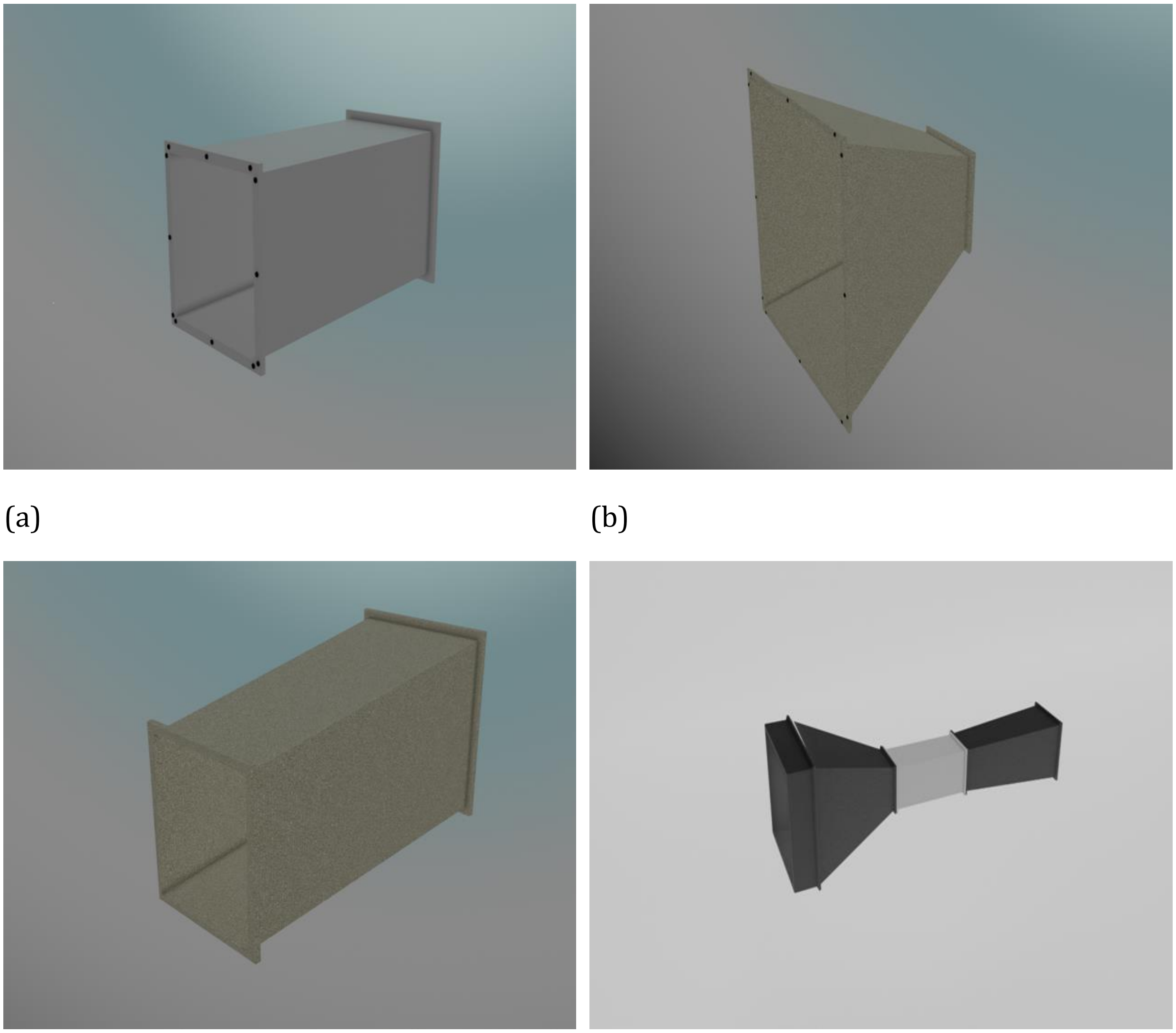
The diffuser's purpose is to produce a vacuum inside the test section, which aids in increasing or reducing the speed of the wind within the test section. The diffuser also reduces the stress on the driving system by lowering the velocity of the wind across the shortest feasible distance. The geometry of the diffuser chamber is determined by the area ratio and the diffuser angle (table 3). The area ratio should be less than 2.5, and the diffuser inclination should be between 5° and 7° [8].The diffuser chamber length (L) is calculated with the help of diffuser angle formula-

$$\theta_e = \tan^{-1}\left(\frac{1}{2}\,\frac{\sqrt{AR}-1}{\frac{L}{D_{h1}}}\right) \tag{2}$$

where, AR is the area ratio of the diffuser, D_{h1} depicts inlet section hydraulic diameter of the diffuser and θ_e is the half of the included angle of the diffuser cone. Figure 1(c) shows the design of the diffuser.

Table 2: Specifications of contraction chamber

Inlet cross section	150mm X 150mm
Outlet cross section	200mm X 200mm
Length	490mm
θ_e	6°
Inlet cross section	150mm X 150mm



(a) (b)

(c) (d)

Figure 1. (a) Test section, (b) **C**ontraction cone, (c) Diffuser and (d) Tabletop wind tunnel designed in Catia v5

Driver System

The wind flow inside the test section is maintained by the driver system, which compensates for pressure loss in the wind tunnel circuit. The driver system consists of an axial fan with a high efficiency and low turbulence air flow and a motor. An axial fan has a diameter of 180 mm and a wind speed of 8.84 m/sec at the fan section. The speed of wind at the fan section is calculated by the formula-

$$\text{Fan section speed} = \frac{\mathrm{V\ X\ A}}{0.25\ \mathrm{X}\ \pi\ \mathrm{X\ D}^2} \tag{3}$$

where, V is the velocity of wind in test section, A is the cross-section area of test section and D is the diameter of the fan.

CALCULATIONS

Reynolds numbers are commonly utilized for scaling fluid dynamics problems and can be used to distinguish between two different scenarios of fluid flow.

$$\mathrm{Re}=\frac{\rho \,\mathrm{X}\, \mathrm{V} \,\mathrm{X}\, \mathrm{D_h}}{\mu} \tag{4}$$

where, ρ is the density of air while μ represent the dynamic viscosity of wind. After the calculation the Reynolds number comes out to be 110472.

Pressure Loss in Wind Tunnel Circuit

For the calculation of pressure loss inside the wind tunnel is done by the formula: -

$$\mathrm{k_i} = {}^{\Delta \mathrm{P_i}}/_{\mathrm{q_0}} \tag{5}$$

where, q_o is dynamic pressure inside the test section and given by $q_o = \frac{1}{2}\rho_o V_0^2$.

Pressure Loss in Test Section

The pressure loss coefficient inside the test section is calculated by the formula: -

$$\mathrm{K_t} = \mathrm{f} \times {}^{\mathrm{L}}/_{\mathrm{D_h}} \tag{6}$$

where, f is friction loss coefficient.

Pressure Loss In Contraction Section

The pressure loss coefficient inside the contraction section is given by-

$$\mathrm{K_c}= \frac{\mathrm{f}}{4} \,\mathrm{X}\, \frac{\mathrm{L_0}}{\mathrm{D_i}-\mathrm{D_0}} \,\mathrm{X}\, \left(1 - \frac{\mathrm{D_0^4}}{\mathrm{D_i^4}} \right) \tag{7}$$

Pressure Loss In Diffuser Section

The pressure loss inside the diffuser section is caused by two main factors: the first is skin friction loss, and the second is expansion loss. The total pressure loss inside the diffuser section is the sum of skin friction loss and expansion loss, as calculated by the formula-

$$\mathrm{K_d}=\mathrm{K_f}+\mathrm{K_{exp}} \tag{8}$$

where, K_f depicts the pressure loss due to skin friction while $K_{exp}$is the pressure loss due to expansion loss. The pressure loss coefficient due to skin friction is calculated by the formula: -

$$\mathrm{K_f}=\left(1 - \frac{1}{\mathrm{Ar}^2}\right) \mathrm{X}\, \frac{\mathrm{f}}{8\sin\theta_e} \tag{9}$$

where, Ar is the ratio of the inlet and outlet cross section are of the diffuser section and f is the friction coefficient.

The pressure loss coefficient due to expansion is calculated by the formula: -

$$K_{exp}=K_e \text{ X } \theta_e \text{ X } \left(\frac{Ar-1}{Ar}\right)^2 \tag{10}$$

where, θ_e= Expansion angle. The value of K_e is calculated by formula, K_e=A3+B3θ_e, where the value of A3= (-0.0132) and B3= (0.05866).

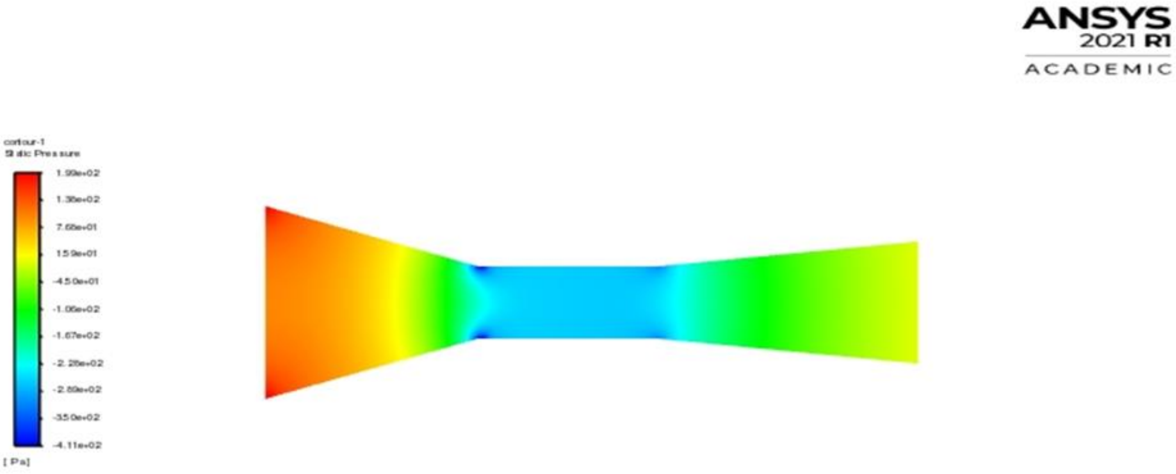
RESULTS AND DISCUSSION

Numerical Analysis

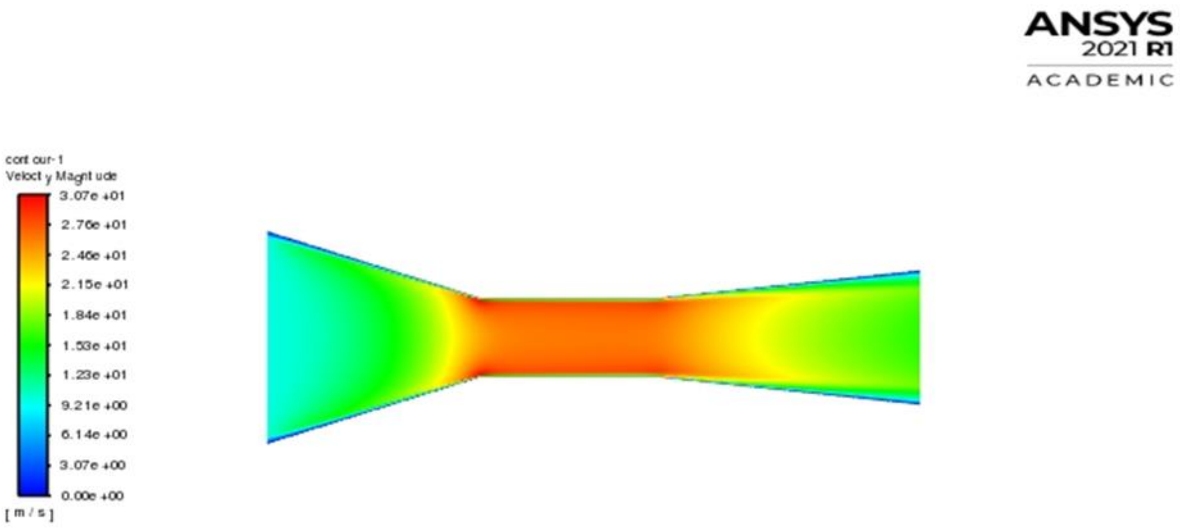
"CATIA V5" designed software is used for modeling the 2D axis symmetric model of the tabletop wind tunnel and the mesh modeling is performed in "ANSYS FLUENT" for the numerical flow analysis of the model as shown in figure 2. For saving time and easing the work for analysis of the entire model 2D axis symmetry model has been used. the parameter taken for the analysis has been given in table 3.

Table 3: Ansys parameters and models used for turbulence modeling

Parameter	Condition
Types of symmetry	Axis symmetry
Viscous model	K-epsilon (2eqn)
Pressure-Velocity coupling	SIMPLE



(a) Pressure contour



(b) Velocity contour

Figure 2. (a)Pressure and (b) velocity contours of the Wind Tunnel on post-processing done in Ansys.

Total Pressure Loss

Total pressure loss = K_i X q_0 + Pressure loss at fan section,
Where K_i= 0.3989, q_o= 61.25 and pressure loss at fan section = 47.86 Pa.
The total pressure loss inside the wind tunnel comes out to be = 72.297 Pa

Table 4: Tunnel pressure loss coefficient at test section velocity 10m/s

Tabletop wind tunnel section	Pressure loss coefficient
Contraction chamber	0.00602
Test section	0.03486
Diffuser	0.358
Total	0.3989

Power Required To Operate The Wind Tunnel

The power required to operate the wind tunnel is calculated by the formula: -

$$P=Q \text{ X } \Delta Pa = (A \text{ X } V) \text{ X } \Delta Pa \tag{11}$$

The power required after the calculation is 16.26 Watt.

CONCLUSION

The characteristics and theories relevant to the design and construction of a tabletop wind tunnel have been reviewed in this study. In this paper, we also examine the different types of wind tunnels, how Reynolds numbers are utilized in wind tunnel design calculations, and the necessity of dimensional analysis in wind tunnel design calculations.

We also talked about how to design and build a contraction chamber, test section, and diffuser. The paper also discusses the measurement of hydraulic diameter and test section length for effective test section design so that it has practically uniform intake air flow and can avoid boundary layer separation at the test section exit.

REFERENCES

1. Bell J. H., Metha R. D. Contraction Design for Small Low-Speed Wind Tunnels. NASA-CR-182747, 1988.
2. Nagendra Kumar Maurya, Manish Maurya, Avdhesh Tyagi and Shashi Prakash Dwivedi "Design & Fabrication of Low-Speed Wind Tunnel and Flow Analysis" (2018) International Journal of Engineering & Technology.
3. Saiful.A. Siddique "The design, development and CFD verification of a subsonic wind tunnel".
4. Ihsan.Y. Hussain, Makah. Majeed, Anmar Hamid Ali, Wali Sami Sarsam work "The design, construction and testing of wind tunnel with low speed with its measurement and inspection devices" (2011), Mech.Engg.Dept., College of Engineering, University of Bagdad.
5. Mauro S, Brusca S.b, Lanzafame R.a, Famoso F.a, Galvagno A.b and Messina M.a "Small-Scale Open-Circuit Wind Tunnel: Design Criteria, Construction and Calibration" International Journal of Applied Engineering Research ISSN 0973-4562 Volume 12, Number 23 (2017) pp. 13649-13662Research India Publications.
6. Syed Shahrukh, Nithin Raj N, Jayaprakash BP, Premkumar Naik R, Dr. G Balakalyani , S.Srinivasan "Design, fabrication & testing of small-scale low Reynolds number tabletop wind tunnel" Volume: 07 ,Issue: 01 ,Jan 2020, e-ISSN: 2395-0056, p-ISSN: 2395-0072.
7. Paweł Magryta "Wind tunnel research on the influence of active airflow on the lift force generated by the airfoil", Department of Thermodynamics, Fluid Mechanics and Aviation Propulsion Systems, Faculty of Mechanical Engineering, Lublin University of Technology, 36 Nadbystrzycka Str., 20-618 Lublin, Poland, e-mail: p.magryta@pollub.pl, Volume 7, No. 19, Sept. 2013, pp. 60–65.
8. Barlow J. B., Rae W. H. Jr.- Pope A. Low Speed Wind Tunnel Testing. 3rd Ed. Wiley-Interscience: New York, 1999. ISBN: 978-0-471-55774-6.

9. Bapu Joglekar and Rana Manoj Mourya “Design,Construction and Testing Open Circuit Low Speed Wind Tunnel” Vol. 2, Issue 4, pp: (271-285), Month: October - December 2014 ISSN 2348-6988 (online) Publisher: International Journal of Electrical and Electronics Research.
10. Wind tunnel research on the influence of active airflow on the lift force generated by the airfoil. Advances in Science and Technology Research Journal Volume 7, No. 19, Sept. 2013, pp. 60–65.
11. Azzawi, I. D. J., Mao, X., & Jaworski, A. J. (2016) ”Design, fabrication and characterization of low speed open-jet wind tunnel” In S. I. Ao, L. Gelman, D. W. L. Hukins, A. Hunter, & A. M. Korsunsky (Eds.), Proceedings of the World Congress on Engineering (Vol. II, pp. 883-888). (Lecture Notes in Engineering and Computer Science; Vol. 2224).
12. Panda MK and Samanta AK (2016), Design of Low Cost Open Circuit Wind Tunnel. Indian journal of science and technology, Vol 9(30), DOI: 10.17485/ijst/2016/v9i30/99195