Low Speed Wind Tunnel Design and Optimization Using Computational Techniques and Experimental Validation

Ahmed ABOELEZZ*,[a]

*Corresponding author
Institute of Aviation Engineering and Technology, Egypt,
Ahmed.helmy.3zz@gmail.com

DOI: 10.13111/2066-8201.2019.11.2.1

Received: 25 April 2019/ Accepted: 22 May 2019/ Published: June 2019

Abstract: *Generally, the experimental aerodynamics is related to wind tunnel experiments. The wind tunnel design topic is very old but the development in computational fluid dynamics led to improvement in the wind tunnel design. This paper describes the design and optimization of low speed wind tunnel using CFD techniques. The new optimum wind tunnel will replace the old one featuring poor air quality and small area with lower wind speed at the test section. A computational domain was generated and adopted using ANSYS mesh generator and the solution domain was analysed by simulation technique using FLUENT CFD code in ANSYS Workbench package. The pressure drop calculations comparison between analytical, computational and experimental is included for different sections in the wind tunnel. The contraction cone was optimized using the response surface technique. The results identified that the pressure drop and turbulence level are modified as compared to the old wind tunnel.*

Key Words: *Wind Tunnel Design, Low Speed Wind Tunnel, Wind Tunnel Optimization, CFD of Low Speed Wind Tunnel*

1. INTRODUCTION

The solution to any aerodynamics problem is conducted by three techniques, namely analytical, computational and experimental. The experimental aerodynamics in general is related to wind tunnel experiments. The wind tunnel design topic is very old but the development in computational fluid dynamics led to improvement in the wind tunnel design which previously was based on analytical and experimental data. [1] made the contraction cone design optimization using surrogate model, the turbulence was modeled by Menter's Shear Stress Transport (SST) k–ω model. He also tested his design in order to measure the turbulence level but he didn't measure the pressure drop. The problem of using the surrogate model and the 3D model in optimization is the need for a huge computational time. [2] used CFD to investigate the flow parameters in closed wind tunnel. He made a full scale CFD model of the entire wind tunnel. The authors compared various CFD turbulent models; the predicted velocity and turbulence values were evaluated against analytically and experimentally solutions. The analysis showed that the k-epsilon and k-omega standard models closely predicted the test section flow speed (3% and 4% error) and turbulence

[a] Research and Teaching Assistant

intensity (7% and 10% error). Also, they concluded that k-epsilon and k-omega were the most suitable turbulence models for this study. [3] made computational and experimental investigation focusing on turbulence level and flow quality. No pressure drop investigation was made. [4] used analytical calculation to calculate the pressure drop in the wind tunnel sections; he didn't made comparison with CFD or experiments. The design, fabrication, and preliminary characterization of an anechoic wind tunnel facility at the University of Florida were presented in this paper [5]. Though, there is no mention to the pressure drop calculation, the paper focusing on turbulence and disturbance of noise. [6] indicated a very important problem in determining the total wind tunnel pressure loss; he explained and illustrated the difference between the analytical and computational. The wind tunnel design process would start with the general wind tunnel geometry based on the requirements. Some guidelines for the initial design would be used from [7]. The low speed wind tunnel should have 50*50 cm cross section area test section, velocity 30m/s and a turbulence level 5 %. This wind tunnel would be used for educational purpose and would replace an old wind tunnel with smaller test section and lower wind speed in the Institute laboratory. The most important objective of this research is the optimization of low speed wind tunnel using computational techniques (CFD). The main flow parameters of the wind tunnel should be investigated. The computational investigation focuses on turbulence level, pressure drop and flow quality. Finally, the design optimization of the wind tunnel will improve the wind tunnel design and reduce its power loss. The important equations representing this problem are introduced in the paper. For CFD a computational domain will be generated and adopted using ANSYS mesh generator and the solution domain will be analysed by simulation technique using FLUENT CFD code in ANSYS Workbench package.

2. WIND TUNNEL DESIGN

Test section. The first step in the design of the wind tunnel is to determine the size of the test section. The cross section area of the tunnel test section will basically determines the overall size of the wind tunnel and will determine the required power [7]. The start point will be the wind tunnel test section design parameters which are dimensions, shape and desired air velocity. In our case, a square testing chamber with a 0.5 m side was used with an air velocity of 30 m/s. The test chamber length has to be in the range of 0.5 - 3 times its hydraulic diameter [7]. This led to choose the length of the test section to be 1 meter.

Contraction cone. The contraction or "nozzle" accelerates the flow from the settling chamber to the test section, further decreasing any variations in velocity. In a wind tunnel, the nozzle is the most difficult component to design [8]. Flow velocity and its uniformity within the test chamber cross-section depend on the nozzle's design. The nozzle exit cross-section dimensions and shape are matching to the test section as they are joined together. The nozzle inlet area ratio should be 'as large as possible' to reduce the total-pressure loss through the screens mounted between the settling chamber and the nozzle. Normally, the nozzle inlet/outlet cross-section area ratio should be in the range 6 – 8 [8]. Area ratios greater than 10 lead to excessive inlet dimensions while area ratios less than 6 lead to high pressure loss through the screens. In this work area ratio 7 was chosen as an average value, which led to inlet section with $1.3\times1.3\ m^2$ dimensions.

Diffuser. The diffuser outlet cross-section area is governed by the fan dimensions. It is known that the ratio between the fan cross-section area A_f and the test chamber cross-section area A_{ts} has to be in the range 2 – 3 [7]. To use an (A_f/A_{ts}) ratio value greater than 3 is not recommended because irregular flow velocities at the fan entrance may be generated [8]. To use a (A_f/A_{ts}) ratio value less than 2 is also not recommended because it may increase the overall wind tunnel dimensions (higher wind tunnel construction costs) [8]. The diffuser angle length (L) and area ratio (AR) should not exceed a maximum value 6 [8] and 7 [7].

Honeycomb. A rectangular honeycomb will be used in the wind tunnel, the honeycomb will has cell length (L_h), cell hydraulic diameter (Dh), and the porosity (βh). The porosity is the ratio of actual flow cross-section area over the total inlet area A_{total} [8]. These parameters are the mean parameters in honeycomb design. L_h/D_h Should be in the range 6-8 and $\beta h \geq 0.8$ [8]. The following equations will be used for honeycomb dimensions calculations [8]:

$$\beta_h = \frac{A_{flow}}{A_{total}} \tag{1}$$

$$\beta_h = 1 - \frac{n_d}{L} \tag{2}$$

$$\text{Area ratio } \alpha = \frac{A_{sheet}}{A_{total}} \tag{3}$$

where n_d is the number of cells, L is the honeycomb face length and A_{sheet} is the actual material area in the honeycomb.

For the new wind tunnel we need 100 cells per length [8]. There would be 101 cell wall with 0.05 cm width for the rectangular honeycomb, also $\beta_h + \alpha = 1$.
Then β_h =0.90, D_h=1.07, L_h=7.5cm.

Screens. The screens reduce velocity fluctuation in the test section. The screens to be effective in reducing turbulence it must have a porosity in the range β = 0.58 – 0.8 [8].

$$\beta = 1 - \frac{d_w}{w_w} \tag{4}$$

where d_w is the wall thickness and w_w is the cell width. It has been found that a screen combination with a spacing equivalent to about 0.2 settling chamber diameters performs successfully [8]. For our case we will use 2 screens with 0.6 porosity.

3. PRESSURE DROP CALCULATION

The pressure drop calculation will help in choosing the suitable fan-motor for the wind tunnel and will help in design optimization. That made the determination of the pressure drop through the wind tunnel very important. This section approaches the description for analytical, computational and experimental techniques to calculate and measure the pressure drops in the wind tunnel. The old wind tunnel in our faculty, shown in Fig. 1 would be used to compare the three solutions.

Fig. 1 - Old wind tunnel

Analytical methods. Analytical methods are the oldest and easiest method to determine the pressure drop through the wind tunnel, but unfortunately they are not accurate (not for all sections in the wind tunnel).

The following table summarized losses determination for the different wind tunnel sections from [8] and [6] based on section loss parameter, where *K*: Section loss parameter, *f*: Friction factor, *L*: Length of the section and *D*: Hydraulic diameter of the section.

Table 1. Wind tunnel sections loss parameters

Test section	$k_t = f\frac{L}{D_h}$	(5)
Nozzle	$k_n = .32 * f_{av}\frac{L}{D_{ts}}$	(6)
Diffuser	$k_d = k_f + k_e$	(7)
Screens	$k_s = k_{mesh}k_{Re}\sigma + \frac{\sigma^2}{\beta^2}$	(8)
Honeycomb	$k_h = \lambda\left(\frac{L_h}{D_h} + 3\right)\left(\frac{1}{\beta}\right)^2 + \left(\frac{1}{\beta} - 1\right)^2$	(9)

Computational fluid dynamics Simulation using ANSYS Fluent. First of all the geometry of the old tunnel was drawn in Ansys design modeler as shown in Fig. 2.
The CAD drawing is used to generate the required mesh as shown in Fig. 3. The mesh independed study in Fig. 4 is used to determine the best mesh element number. Increasing the mesh element number too much will increase the computational time.

The mesh is uniform and contains about 200k element with surface inflation near walls to consider the boundary condition effect in calculations.

ANSYS Fluent 17.1 was used with this 2D mesh and K-omega turbulence model to solve the fluid governing equations.

The 2D Flow analysis would be used in this design for two reasons, the optimum design in 2D will also be the optimum in the 3D analysis (without consider the flow in the corners). The different would be only in the actual pressure drop calculations, but making design optimization in the 3D would be complicated and the results would complicate the wind tunnel geometry which it may be impossible to manufacture.

As will be indicated later in the pressure drop results, the pressure drop in 2D CFD was close to the experimental result which is 3D case.

The boundary conductions used in this study were the inlet pressure, outlet pressure, the tunnel body as a stationary walls and symmetrical line. All boundary conditions are shown in Fig. 3.

As mentioned in the literature review it is possible to use K-ε or K-omega turbulences models, K-omega turbulence model would be used in this research as pressure inlet- pressure outlet boundary conditions were used.

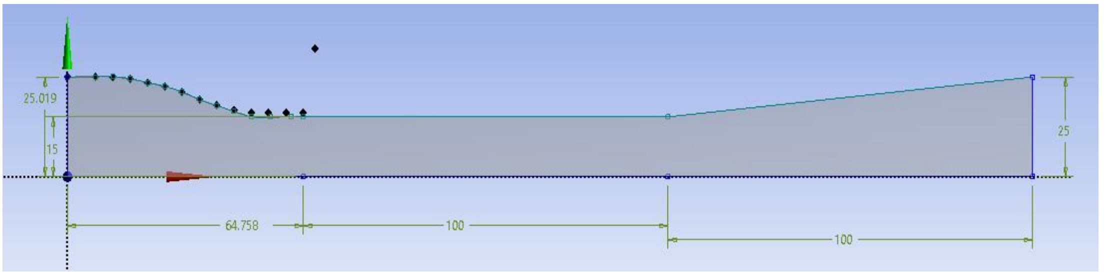

Fig. 2 - Old wind tunnel dimensions in cm

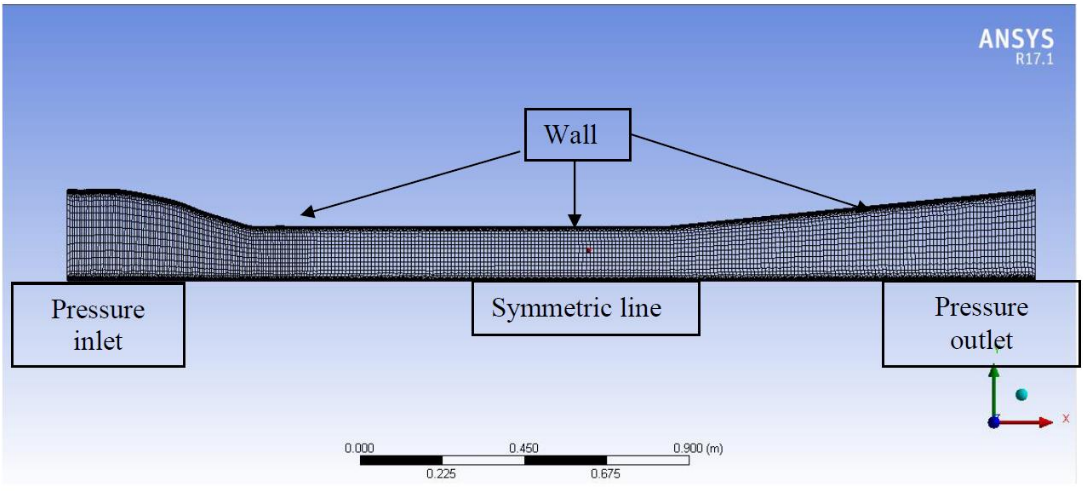

Fig. 3 - Overall meshing grid

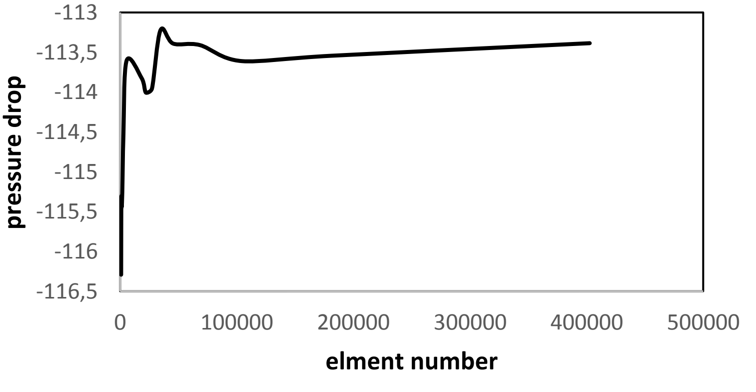

Fig. 4 - Mesh independed study

Experimental method. The most accurate method to determine pressure drop is the experimental method by measure the pressure drop on the wind tunnel. The problem with the experiment is that it should be conducted in a wind tunnel. In our case the pressure drop will be measured in the old wind tunnel to validate the CFD code.

Old wind tunnel pressure drop results. As a validation to CFD we conduct the experiment in the old wind tunnel with test section area 0.3×0.3m and speed 13.5m/s. This wind tunnel was built from wood in 2006 as a graduation project. We should mention here that this wind tunnel were designed to achieve 30m/s wind speed in the test section, but due to using the

analytical equations during design the test section velocity was 13.5m/s. Using a digital manometer enable measuring the pressure in different location along the wind tunnel.
The comparison between CFD, Experiment and analytical pressure drop calculation is summarized in Table 2. From the results it is very clear that most errors in calculation are in the nozzle part because the analytical solution assumes perfect nozzle with friction loss only [1]. It was impossible to simulate the flow in honeycomb and screens so the analytical results were used in this part. In general, the CFD gave very accurate results compared to experiment results.

Table 2. Old wind tunnel Pressure drop in Pascal

section	Analytical	CFD	experimental
honeycomb	4.675	4.675(analytical solution)	4.675(analytical solution)
Screens (3 screens)	43.578	43.578(analytical solution)	44
nozzle	5.1	75	72
Test section	12.381	3	7
diffuser	22.189	63	61
total	87.27	189.253	188.675

4. WIND TUNNEL DESIGN OPTIMIZATION

Based on the design section of the wind tunnel in this paper the initial geometry of the wind tunnel was drawn in ANSYS design modeler as shown in Fig. 5; the design optimization could be started from this initial geometry. The optimization focused on the contraction cone as it strongly affects the flow quality in the test section and the overall performance of the tunnel. A response surface technique was used with a control point in the contraction cone as the input variable shown in Fig. 5. The control point will move in the horizontal and vertical direction (this movement will change the overall shape of the contraction cone) and the optimum point would give highest flow uniformity with minimum pressure losses. The design of experiment is illustrated in Fig. 6Fig. ; in this figure it appears the variation of the design objective function with the control point movement. The variation of uniformity of the flow in the test section with the variation of control point location is shown in Fig. 7.

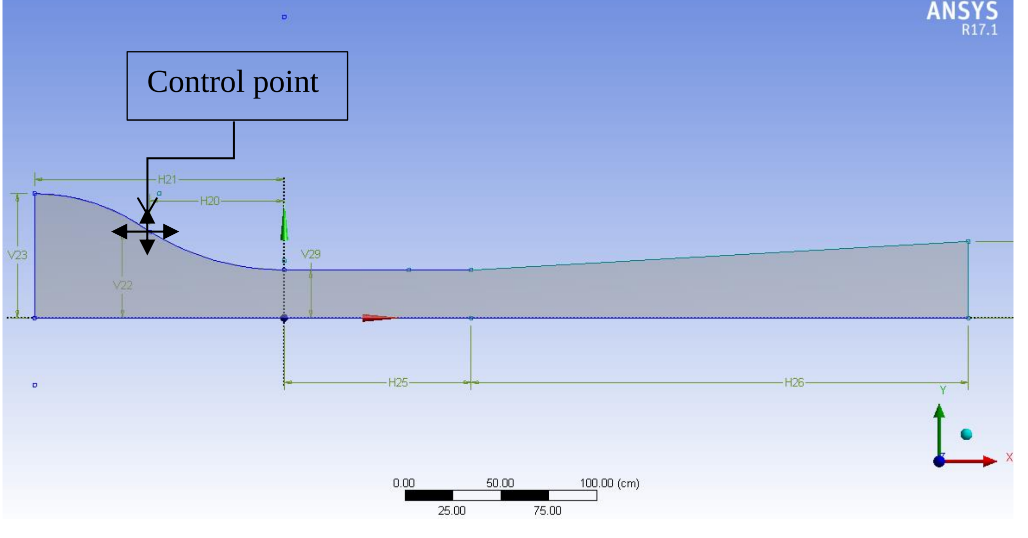

Fig. 5 - Control point location for optimization

	A	B	C	D	E
1	Name	P6 - controlpointh (cm)	P7 - controlpointv (cm)	P9 - speed (m s^-1)	P10 - uniformity
2	1	50	40	33.277	0.99594
3	2	30	40	33.096	0.99164
4	3	70	40	33.333	0.99777
5	4	50	30	33.051	0.99855
6	5	50	50	33.207	0.99435
7	6	30	30	33.234	0.99585
8	7	70	30	31.973	0.99956
9	8	30	50	33.017	0.99008
10	9	70	50	33.302	0.9966

Fig. 6 - Design of experiment

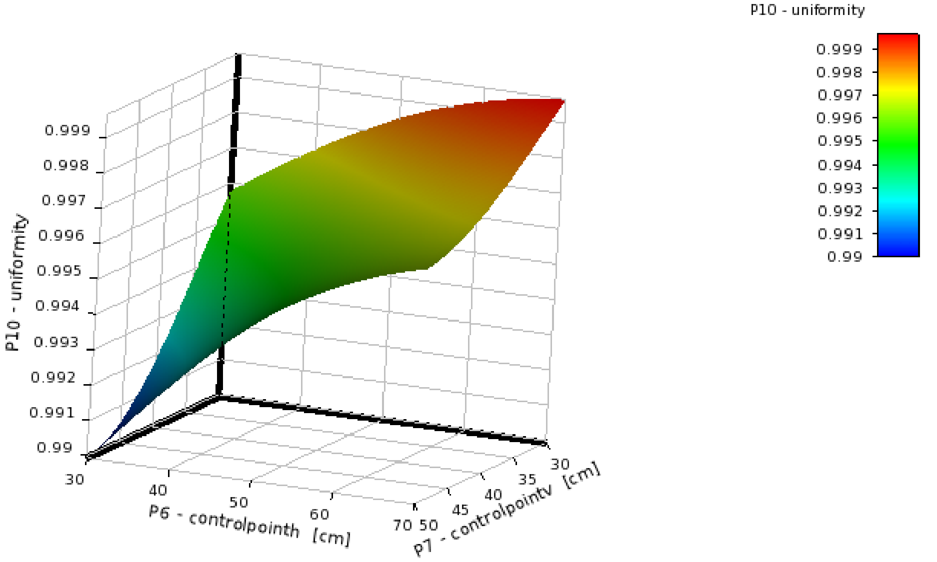

Fig. 7 - Velocity uniformity changing with control point movement

5. RESULTS AND DISCUSSIONS

ANSYS fluent with the 2D model could numerically calculate the pressure drop and flow quality in the wind tunnel test section. A new wind tunnel design was presented in the paper to replace our faculty old wind tunnel.

A response surface is constructed from a number of set of contraction cone shapes optimization process.

The optimum contraction cone gives the highest flow uniformity with minimum pressure loss; uniformity and pressure loss were calculated with ANSYS FLUENT.

The optimum contraction cone is plotted in Fig.8. The equation for this cone could be written as:

$$\frac{y}{y_{max}} = 9.8825\left(\frac{x}{x_{max}}\right)^6 - 34.196\left(\frac{x}{x_{max}}\right)^5 + 43.62\left(\frac{x}{x_{max}}\right)^4 - 23.296\left(\frac{x}{x_{max}}\right)^3 + 3.7448\left(\frac{x}{x_{max}}\right)^2 - 0.3719\left(\frac{x}{x_{max}}\right)^1 + 1.0012$$

where y_{max} is maximum height of the cone from the centerline and x_{max} is the maximum length of the contraction cone.

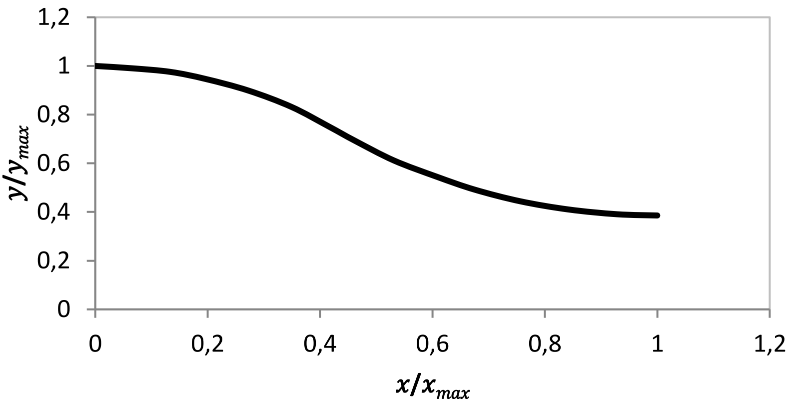

Fig. 8 - Contraction cone geometry

The velocity variation in the test section inlet is illustrated in Fig. 9 for both old and optimum wind tunnel, the velocity is plotted from the wind tunnel centerline and the test section roof.

As the two tunnels are operated with different speeds and both have different height, the figure is plotted as $\frac{y}{y_{max}}$ versus $\frac{v}{v_{max}}$.

The significant improvement in the velocity variation is very clear in the figure, even the new tunnel has higher wind speed and larger dimensions but it has better flow quality (smaller velocity variation).

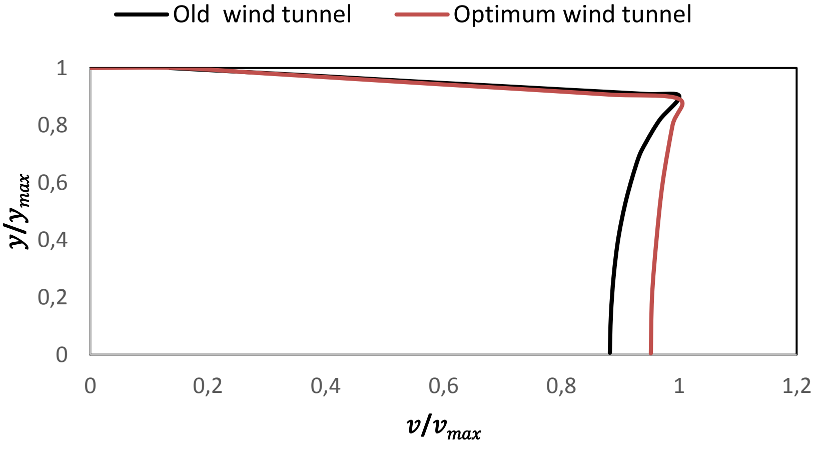

Fig. 9 - Velocity variation at the test section inlet

The velocity and pressure contours results for the old and new optimum tunnel are illustrated in Fig. 10, Fig. 11, Fig. 12 and Fig. 13, respectively.

The pressure and velocity contour illustrate smoother transition from a section to anther when comparing the results between the old and new wind tunnel. Also the new diffuser has better performance with lower separation.

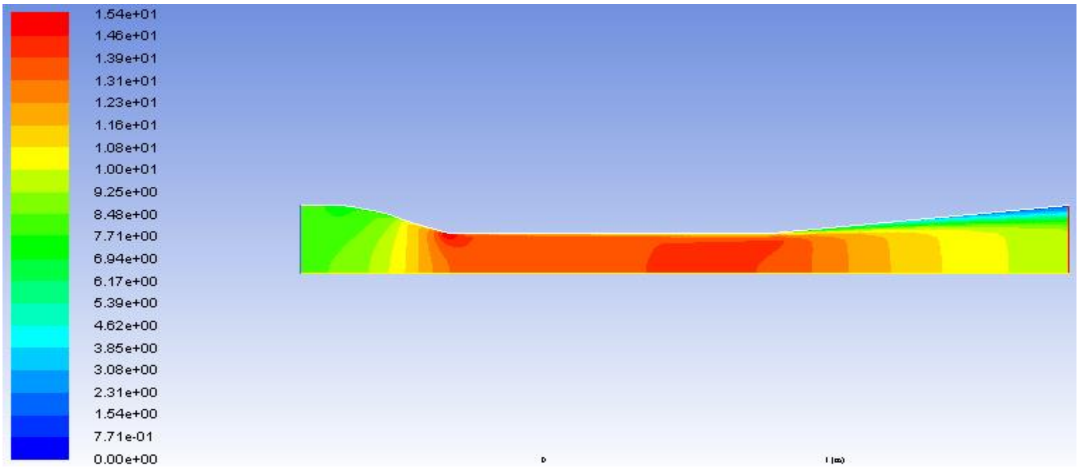

Fig. 10 - Old wind tunnel velocity contours.

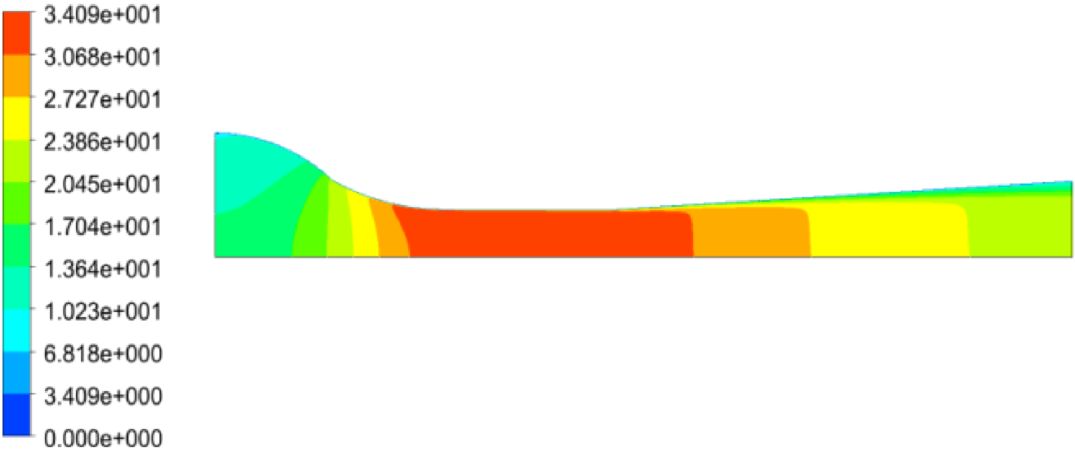

Fig. 11 - Optimum wind tunnel velocity contours.

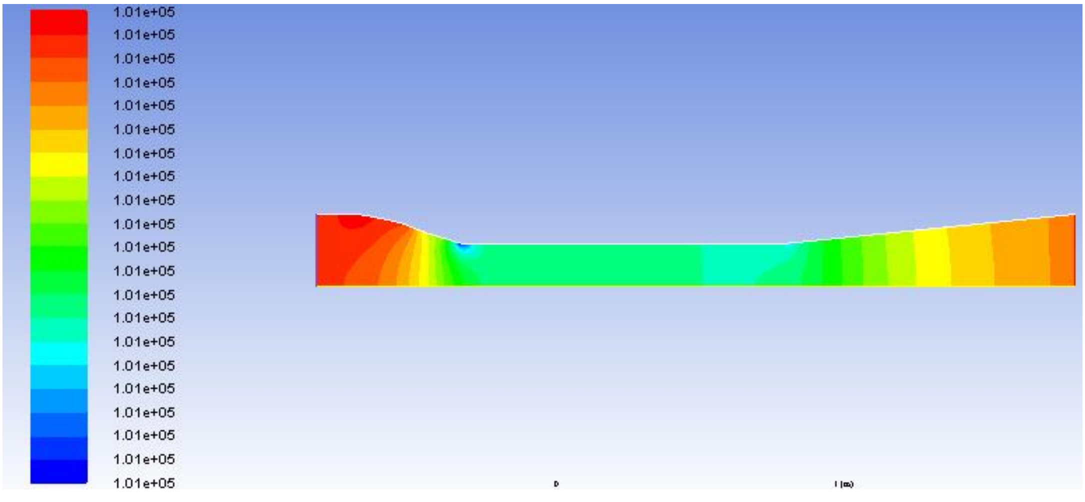

Fig. 12 - Old wind tunnel pressure contours.

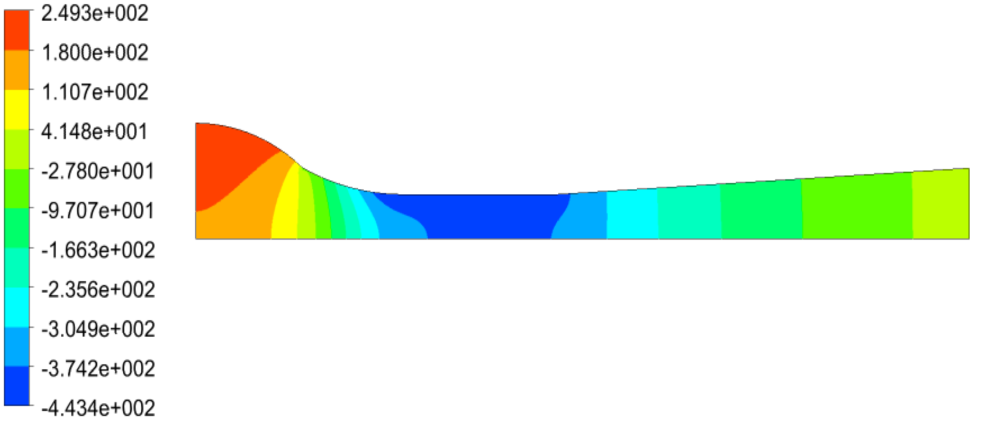

Fig. 13 - Optimum wind tunnel pressure contours.

The CFD analysis could predict the pressure drop in the wind tunnels and this will help us to determine the required fan and motor systems. The pressure drop for the old tunnel was 189 Pascal and for the new tunnel 1100 Pascal. With the analysis in this paper we could design a new wind tunnel with higher test section area, higher wind speed and the most important parameter a better flow quality in the test section.

6. CONCLUSIONS

The computational analysis could be used to calculate the pressure drop for all parts of the wind tunnel. The analysis shows that the k-ε and k-ω standard models are the most suitable turbulence models for this study.

Using 2D CFD will reduce computational time. 200k element in the mesh is very suitable number for 2D wind tunnel analysis.

The control point and response surface optimization in FLUENT could be successfully used to optimize the wind tunnel design.

This analysis will help to design a low speed wind tunnel for minimum pressure drop which finally leads to increase the tunnel efficiency.

The analytical pressure calculation is good only for honeycomb and screens case, so for any new wind tunnel design the analytical solutions will be used in these parts and the computational for the rest of the wind tunnel parts. Turbulence intensity experiment should be conducted in the future work.

NOMENCLATURE

A_f	Fan section area
A_{ts}	Test section area
D_h	Cell hydraulic diameter
d_w	The wall thickness
L_h	Honeycomb cell length
n_d	The number of cells
w_w	The cell width
β_h	Porosity
α	Area ratio

ACKNOWLEDGEMENT

The author would like to thank David Eskander, teaching assistant at the institute of aviation engineering and technology for his help in Mesh independed study.

REFERENCES

[1] L. Leifsson, S. Koziel, Simulation-driven design of low-speed wind tunnel contraction, *Journal of Computational Science*, Volume **7**, Pages 1-12, ISSN 1877-7503, 2015.

[2] J. K. Calautit, H. N. Chaudhry, B. R. Hughes, L. F. Sim, A validated design methodology for a closed-loop subsonic wind tunnel, *Journal of Wind Engineering and Industrial Aerodynamics*, Volume **125**, Pages 180-194, ISSN 0167-6105, 2014.

[3] Itimad D. J. Azzawi, X. Mao, and A. J. Jaworski, *Design, Fabrication and Characterization of Low Speed Open-jet Wind Tunnel*, Proceedings of World Congress on Engineering 2016, Vol. **2**, Newswood Limited, 2016.

[4] Mahesh K. Panda and Amiya K. Samanta. Design of Low Cost Open Circuit Wind Tunnel-A Case Study, *Indian Journal of Science and Technology*, **9**.30, 2016.

[5] J. Mathew, C. Bahr, B. Carroll, M. Sheplak and L. Cattafesta, Design, Fabrication, and Characterization of an Anechoic Wind Tunnel Facility, *11th AIAA/CEAS Aeroacoustics Conference, Aeroacoustics Conferences*, May 2005.

[6] Torrano, M. Martinez-Agirre and M. Tutar, Numerical characterization of pressure drop through a low speed wind tunnel: Some design aspects, 2013 International Renewable and Sustainable Energy Conference (IRSEC), Ouarzazate, pp. 240-245, 2013.

[7] W. H. Rae and A. Pope, *Low-speed wind tunnel testing*, John Wiley, 1984.

[8] J. D. Pereira, *Wind Tunnels: Aerodynamics, Models, and Experiments*, Nova Science Publisher's, 2011.

[9] J. Mathew, et al., Design, fabrication, and characterization of an anechoic wind tunnel facility, 11th AIAA/CEAS Aeroacoustics Conference, 2005.

[10] B. Celis, H. H. Ubbens, Design and Construction of an Open-circuit Wind Tunnel with Specific Measurement Equipment for Cycling, *Procedia Engineering*, Volume **147**, Pages 98-103, ISSN 1877-7058, 2016.