DESIGN, CONSTRUCTION AND TESTING OF LOW SPEED WIND TUNNEL WITH ITS MEASUREMENT AND INSPECTION DEVICES

Prof. Dr. Ihsan Y. Hussain* **Asst. Lect. Maki H. Majeed***

Lect. Anmar H. Ali* **Lect. Wail S. Sarsam***

* Mech. Engr. Dept., College of Engineering, University of Baghdad

ABSTRACT

A low speed open circuit wind tunnel has been designed, manufactured and constructed at the Mechanical Engineering Department at Baghdad University - College of Engineering. The work is one of the pioneer projects adapted by the R & D Office at the Iraqi MOHESR. The present paper describes the first part of the work; that is the design calculations, simulation and construction. It will be followed by a second part that describes testing and calibration of the tunnel. The proposed wind tunnel has a test section with cross sectional area of (0.7 x 0.7 m^2) and length of (1.5 m). The maximum speed is about (70 m/s) with empty test section. The contraction ratio is (8.16). Three screens are used to minimize flow disturbances in the test section. The design philosophy is discussed and methods for wind tunnel calculation are outlined. Simulation of wind tunnel using ANSYS shows no separation of flow along wind tunnel. Construction steps are also included in present work.

الخلاصة:

تم تصميم و تصنيع و بناء نفق هوائي مفتوح واطئ السرعة في قسم الهندسة الميكانيكية في كلية الهندسة – جامعة بغـداد. يمثل العمل احد المشاريع الريادية التي تبنّتها دائرة البحث و التطوير في وزارة التعليم العالي و البحث العلمي العراقية. يتطرق البحث الحالي الى الجزء الاول من العمل و هو الحسابات التصميمية ، التمثيل على الحاسبة والتصنيع، و سوف يتبعه جزء ثاني يتطرق الـى اختبار و معايرة النفق. للنفق الهوائي المقترح مقطع اختبار مساحته (٠,٧ x ٠,٧ م^{٢}) و طول (١,٥ م). سرعة الهواء القصوى فـي مقطع الاختبار و هو فارغ تبلغ (٧٠ م / ثا). نسبة التخفيض في المساحة هي (8,16). تم استخدام ثلاثة مشبكات لتقليل الاضطراب في مقطع الاختبار. سوف تتم مناقشة فلسفة التصميم و طرق الحسابات المتبعة في البحث. تم تمثيل النفق علـى الحاسـبة باسـتخدام برنامج (الانسيز) وقد بينت النتائج عدم حصول انفصال للجريان على طول النفق الهوائي. تتضمن البحث الحالي ايـضا خطـوات البناء للنفق الهوائي.

Key Words: Wind Tunnel, Low Speed, Design, Simulation and Construction

INTRODUCTION

The development of fluid mechanics involves observation and study of the physical phenomena which form the basis of the theory. Experimental aerodynamics serves to check the existing theory and also its extension. On the other hand, theoretical developments strongly influence experimental ones.

The wind tunnel is an important available device for experimental work in aerodynamics and it can be considered the main tool for aerodynamics design of aircrafts, rockets, turbo-machines…etc; its main function is to provide a uniform and controllable air flow through the test section.

At low speed, dynamic similarity requires the equivalence of Reynolds number, but since the model is visually a scaled down body, this equivalence is difficult to achieve unless a pressurized tunnels are used. However if the Reynolds number exceeds 10^5, it is assumed that the exact equivalence is not essential **(Raikan ٢٠٠١)**. The basic tunnel type is an open circuit tunnel with suction or blowing as shown in **Fig.1**. The air flows in straight path from the entrance through a settling chamber with contraction to the test section, followed by a diffuser, a fan section and an exhaust of the air. In suction wind tunnel the closed test section pressure is less than that in the surrounding medium, this make it more difficult to carry out test, and introduces inaccuracies into the determination of the forces acting on the model, since atmospheric air enters through the walls of the test section.

The design of the open circuit wind tunnel selected in the present work will be done by analyzing the flow through each component such as the contraction section, test section, and diffuser section. The available literatures that deal with the design of wind tunnel or those components are illustrated firstly. **(Pope and Harper 1966)** pointed out the most important recommendations and presented guide lines for the design of low speed wind tunnel, also by using empirical correlations they presented a design procedure for the fan required for the wind tunnel.**(Milan Valajinac 1970)** summarized the design, construction and calibration of a subsonic wind tunnel. The design philosophy is discussed and methods for wind tunnel calculations are outlined. Comparisons of measured wind tunnel parameters are shown to be in excellent agreement with design calculations. **(Bradshaw and Mehta 1979)** presented design guide lines for the main components of a wind tunnel, the fan, wide angle diffuser, settling chamber, contraction, screens, honey combs and exit diffuser for small low speed wind tunnels, also in similar way as in **(Pope and Harper 1966)**. In **(Raikan 2001),** a subsonic wind tunnel design of closed circuit type for a range of Mach No. in the test section up to 0.7 has been carried out and a mathematical models describe the flow in each component of the wind tunnel are made. The comparisons between the results obtained from their design and the literature were done and they showed that a good agreement exist with these results. In the present work the dynamic pressure in the test section was originally chosen to be more than 3 kpa as a basis for a power requirement and performance calculations, so that the particular fan and power plant selection must match the calculated wind tunnel requirements.

WIND TUNNEL DESIGN CONSIDERATIONS

Mathematical models describing the flow analysis through all components of low-speed wind tunnel are made in this article. The flow field is assumed incompressible throughout the tunnel, and the model for each part is then linked to predict power requirements. The chosen tunnel and its components are also investigated. The wind tunnel shown in **Fig.1** is to be designed and analyzed for its dimensions. It has a square cross section with side equal to (0.7 m) and length is taken to be (1.5 m), since it may be sufficient for testing large models. The total divergence angle (2α) of (1.28°) for one pair of walls is chosen as will be shown later during analysis to maintain approximately constant average Mach number along the test section.

POWER REQUIREMENTS

The power required to maintain steady flow through the wind tunnel is equal to the total losses occurring in the flow through the tunnel. These losses are due to kinetic energy being dissipated by vortices and turbulence. The loss in kinetic energy, which appears as a decrease in total pressure must be compensated by pressure rise usually provided by a fan. The power input to the fan is the motor shaft output, and the fan has efficiency (η). The equation balancing the energy input to the stream to the energy losses in the tunnel is:

$$\eta\,\square = \sum \text{circuit losses} \qquad (1)$$

As pointed-out previously the tunnel can be divided into sections with energy loss of each section written as a drop in pressure Δp or a pressure drop in coefficient $K_i = \Delta p_i / q_o$, Where q_o is the test section dynamic pressure ($1/2\rho_0 v_0^2$). The flow energy through the test section is:

$$E_0 = 1/2\rho_0 v_0^3 A_0 \qquad (2)$$

For subsonic flow with M<0.4, $\rho_0/\rho i=1$ (within 1% error) and **eq. (1)** becomes **(Milan Valajinac 1970)**:

$$\eta = 1/2\rho_0 v_0^3 A_0 \sum_i K_i \qquad (3)$$

The required power for a given test section size and flow conditions depend on the sum of the pressure drop coefficients (K_i) in various tunnel sections and a reduction in their coefficients improves the tunnel efficiency. The energy ratio defined as:

$$E.R = \frac{1/2\rho_0 v_0^3 A_0}{\eta} = \frac{1}{\sum_i K_i} \qquad (4)$$

And is a measure of the tunnel efficiency.

1. The Contraction Section

It has a square cross section of (2x2) m^2 at inlet with an area ratio of (8.16: 1). The contraction wall shape profile, see **Fig.2**, consists of two elliptic arcs matching at a point; the position of maximum slope. The dimensions have been scaled so that the width at the entrance is unity. The arcs are then uniquely specified by the length L. the equations for the arcs in Y and Z direction are given by;

$$y_w = \left(a_w - b_w\sqrt{1 - \frac{x^2}{c_w^{\,2}}}\right) \qquad 0 \le x \le \overline{p_w}$$

$$y_w = \left(d_w - e_w\sqrt{1 - \frac{(x^2 - L)}{f_w^{\,2}}}\right) \qquad \overline{p_w} \le x \le L$$

$$z_h = \left(a_h - b_h\sqrt{1 - \frac{x^2}{c_h^{\,2}}}\right) \qquad 0 \le x \le \overline{p_h}$$

$$z_h = \left(d_h - e_h\sqrt{1 - \frac{(x^2 - L)}{f_h^{\,2}}}\right) \qquad \overline{p_h} \le x \le L$$

$$a = \left(\frac{\overline{H^2} - \bar{p}\bar{T}\bar{H} - \frac{1}{(4ra)^2}}{2\bar{H} - \bar{p}\bar{T} - \frac{1}{ra}}\right)$$

$$b = \left(a - \left(\frac{1}{2ra}\right)\right)$$

$$c^2 = \left(\frac{\bar{p}b^2}{\bar{T}(a - \bar{H})}\right)$$

$$d = \left(\frac{\overline{H^2} + \bar{T}\bar{H}(L - \bar{p}) - \frac{1}{4}}{\bar{T}(L - \bar{p}) + 2\bar{H} - 1}\right)$$

$$e = \frac{1}{2} - c$$

$$f^2 = e^2(L - \bar{p})/\bar{T}(\bar{H} - d)$$

$$ra_w = \left(\frac{w1}{w2}\right)$$

$$ra_h = h_1/h_2$$

$$\bar{p} = (L/(1 + ra))$$

$$\bar{H} = \left(\frac{1}{ra + 1}\right)$$

$$\bar{T} = \left(\frac{2(ra - 1)}{L\,ra}\right)$$

(5)

Once separation is assured not to exist, simple procedure to predict performance is the regular one-dimensional flow method. The final differential equation which describes this flow affected by area change and friction is given by **(Shapiro 1953)** as follows;

$$\frac{dM^2}{M^2} = \left[\frac{1 + \frac{(\gamma - 1)M^2}{2}}{1 - M^2}\right]\left\{-2\frac{dA}{A} + \gamma M^2 \frac{4\,fdx}{DH}\right\}$$

(6)

The friction factor in this equation will be calculated from the following relation mounted by **(Shapiro 1953)**, for the subsonic flow:

$$f = \left[1 - \frac{1.12}{\log_{10}(R_{e})}\right] \quad (7)$$

Where:

$$C_D = \frac{0.42}{\log_{10}(R_e)\left[1 - \frac{\gamma - 1}{2}M^2\right]}$$

and $Re = \frac{\rho v_{av} D_H}{\mu}$ (8)

$$D_H = \frac{4 y_w z_h}{(y_w + z_h)}$$

$A = 4Y_w Z_h$

Also, the Sutherland law of viscosity **(Schlichting 1979)**, is used to calculate viscosity:

$$\frac{\mu}{\mu_0} = \left[\frac{T}{T_0}\right]^{3/2}\left[\frac{T_0 + S_1}{T + S_1}\right] \quad (9)$$

Where T_0 is reference temperature taken to be equal to (273.6 K) and μ_0 denotes the viscosity at reference temperature which is (1.708×10^{-5} Pa.s), and S_1 is constant whose value for air is (110 K).

2. Test Section

The main purpose in the design of the test section is to maintain approximately constant static pressure with slightly small change in the average Mach number throughout the test section. Therefore; the cross sectional area should gradually increases in the flow direction to compensate for the thickening of the boundary layer which cause the reduction in the static pressure along the test section, so divergence angle for one pair of the walls will be made, and this may be calculated by assuming the test section as fully turbulent and the boundary layer begins at entrance. For turbulent layer, the displacement thickness could be calculated as **(Schlichting 1979)**;

$$\delta^* = 0.479 * x/(R_x)^{1/5} \quad (10)$$

By considering the test section with constant height, the upper and lower walls are deflected to prevent boundary layer growth. This design provides good view to users. To find losses inside wind tunnel test section, the only losses comes from friction losses so that the losses are:

$$\frac{\Delta P}{1/2 \rho_0 v_0^2} = K_{ts} = f \frac{L}{D_0} \quad (11)$$

where Do represents hydraulic diameter of test section and f is a friction factor, and it is assumed constant along test section and approximately equals to (f = 0.01).

3. Diffuser

The diffuser is used to reduce the power losses due to high flow velocity. Generally; the velocity must decrease with distance without any separation of boundary layer at walls. One of the side effects of separation is the vibration of the fan which then causes change in velocity at test section. The divergence angle is:

$$\theta = \tan^{-1}\left(\frac{R_2 - R_1}{L}\right) = \tan^{-1}\left(1/2 \frac{\sqrt{AR} - 1}{L/D_1}\right) \quad (12)$$

Where $A_R = A_2 / A_1$

One of the most important aspects in diffusers design is that the divergence angle must be small to ensure no separation of boundary layer at wall. But this mean long diffuser for high aspect ratio and may be costly inefficient. **Fig.3** shows the relation between 2θ and aspect ratio for two type of diffusers 2D and axisymmetric diffuser, From **(Wallis 1983).** The pressure losses in diffuser can be calculated with the same equations used in contraction section and same procedure or by using equation listed in **(Pope and Harper 1986):**

$$\left.\begin{aligned} k_f &= \left(1+\frac{1}{AR^2}\right)\frac{f}{8\ \sin\theta} \\ k_{ex} &= k_e(\theta)*\left(\frac{AR-1}{AR}\right)^2 \end{aligned}\right\} \quad (13)$$

$$\left.\begin{aligned} &K_e=0.1709-0.117\theta+0.0326\theta^2+0.001078\theta^3-0.0009076\theta^4-0.00001331\theta^5+0.00001345\theta^6 \\ &1.5^\circ \le \theta \le 5^\circ \end{aligned}\right\} \quad (14)$$

$$K_d = k_f + k_{ex}$$

4. Settling Chamber and Straighteners

The two sections are used to decrease the turbulence in flow and make it straight by using honeycombs and screens. The velocity at these sections must be low to decrease the pressure losses in wind tunnel. Screens reduce the axial turbulence more than the lateral turbulence; they have a relatively large pressure drop in the flow direction. Honeycombs have small pressure drop and this have less effect on the axial velocities but they reduce lateral turbulence. **(Pope and Harper 1986)** presents a method for calculating these components depending on semi-empirical equations which used in the coming sections to calculate the pressure drop on the settling section.

5. Entrance

The entrance may be divided into two components. The first called (bell mouth) and the other is a converged duct as show in **Fig.1**. Bell month is an important part in wind tunnel because it improves air flow entrance which decreases the energy losses and increase the amount of air enters the wind tunnel. The design equation of the entrance given by **(Bleier 1997)** is (r = 0.14 D). Same equations and procedure used in contraction section and diffuser are used for the entrance to find energy losses.

NUMERICAL SOLUTION

The numerical solution procedure to solve the governing differential equations and auxiliary relations which are derived in mathematical model for wind tunnel component is described here.

The contraction section consists of a square cross section and with two elliptical cross-sectional areas which represent the wall shape profiles, also two parallel ducts are connected upstream of the contraction toward the last screen and downward to the test section. The flow through the contraction is considered. One dimensional compressible flow affected by area change and friction is used. The final equation which describes the one dimensional flow affected by area change and friction is given by **eq. (6)**. The numerical solution of this equation is done by dividing the contraction section into a number of equal small elements each of length Δx, and the conditions at the inlet are known, the calculation proceeds step by step using a forward finite difference scheme. So **eq. (6)** is written in forward finite difference form. The problem here is that the conditions at the inlet are unknown, but they are known at the exit, and hence the forward finite difference scheme cannot be used, so **eq. (6)** will be written in a backward finite difference form. Solution of this equation step by step gives the inlet conditions. Then the forward finite difference scheme is used with these inlet conditions to obtain the new exit conditions. The friction factor is calculated from **eq. (7)**. Three of wind tunnel parts are calculated by this procedure (Entrance, contraction section and diffuser). In the diffuser the inlet conditions are known, so that, backward difference is not needed.

A computer program was developed to perform the numerical solution formulated and described above. The computer program was written in FORTRAN language and designed to give the distribution of the flow quantities along the wind tunnel. The flow chart of the program is shown in **Fig.8.**

DESIGN CALCULATIONS

The numerical results obtained from the developed computer program in the present work which describe the flow analysis through the all wind tunnel components mentioned previously are presented here.

Test Section

The conditions at the test section are;

- Square inlet area (700*700 mm^2)
- Test section length (1500 mm)
- The dynamic pressure (3000 Pa) and velocity (70 m/s)

From **eq.(10),** the displacement thickness based on Reynolds number of ($3.3*10^6$) is calculated to be equal to (δ^*=3.55 mm). So, the longitudinal direction with width constant along the test section becomes (7.01 mm), it can be approximated to (10 mm) so that the test section dimensions become:

- test section entrance→ (700*700 mm^2)
- test section exit→ (700*710 mm^2)
- test section length→ (1500 mm)

The pressure losses occur at test section could be calculated by assuming constant cross section and the friction factor also constant. So that, from **eq. (11)** (kts = 0.021) for friction factor (f = 0.01).

Contraction Section

For the contraction ratio (8.16: 1) where inlet area is (2000x2000 mm^2) and exit area equal to (700x700 mm^2), the numerical solution was made with the following parameters see **Fig.2**:

- Number of divisions equal to 1000 small elements.
- Convergence limit is 1.0E-5

```
TUNNEL NOZZLE DIMENSIONs:-
======================
L =        1.5000      m

W1=        2.0000      m
H1=        2.0000      m

W2=        0.7000      m
H2=        0.7000      m

LW  =      1.7030      m
LZ  =      1.7030      m
CR  =      8.1633

OUTPUT PROPERTIES:-
==================
M_OUT = 0.2022
V_OUT = 70.0000            m/s
P_OUT = 98510.0000         Pa
T_OUT = 298.4000           K
RO_OUT = 1.1503            Kg/m3
PO_OUT = 101357.0715       Pa
TO_OUT = 300.8390          K

INPUT PROPERTIES:-
==================
M_IN = 0.0241
V_IN = 8.3979 m/s
P_IN = 101307.2628         Pa
T_IN = 300.8039            K
RO_IN = 1.1735             Kg/m3
PO_IN = 101340.1849        Pa
TO_OUT = 300.8390          K

CONTRACTION LOSSES K:-
=====================
```

K_n = 0.0060 this represent nozzle pressure losses.

Diffuser

The energy losses at any point in the wind tunnel depend on the cubic velocity at that point. So that, the diffuser works to decrease the velocity with minimum losses and higher back pressure. Generally it must decrease the velocity without boundary layer separation at the wall.
The relation between divergence angle 2θ and ratio of inlet diameter to length of the diffuser for two type of diffuser (2D and cone diffusers).The following are the diffuser dimensions:

- Inlet diffuser diameter D_1=700mm
- Outlet diffuser diameter D_2=1500mm = fan diameter
- Diffuser length =6000mm
- The divergence angle is = $\theta = 7.6^\circ$
- Aspect ratio of diffuser AR = 4.527

When comparing the diffuser dimension with that in **Fig.** (3) for cone diffuser, the ratio will be (N/R1) =17 this mean the divergence angle will be equal to 7.25°. So it represents good approximation for dimensions. The losses at diffuser could be calculated numerically as mentioned previously and the results are:

```
TUNNEL DIFFUSER DIMENSION:-
======================
L =        6.0000      m

W1=        0.7000      m
H1=        0.7100      m

D2=        1.5000      m
AR  =      4.5270

INPUT PROPERTIES:-
==================
M_IN = 0.2022
V_IN = 70.0000             m/s
P_IN = 98510.0000          Pa
T_IN = 298.4000            K
RO_IN = 1.1503             Kg/m3
PO_IN = 101357.0715        Pa
TO_IN = 300.8390           K

OUTPUT PROPERTIES:-
==================
M_OUT = 0.0437
V_OUT = 15.1755            m/s
P_OUT = 101133.5630        Pa
T_OUT = 300.7244           K
RO_OUT = 1.1718            Kg/m3
PO_OUT = 101534.3830       Pa
TO_OUT = 300.8390          K
```

CONTRACTION LOSSES K:-
=====================
K_d= 0.0629 This represents diffuser pressure losses.

It must be noted that the losses in contraction equal to 0.006 and in diffuser equal to 0.0629, in **(Pope and Harper 1966)** there is an experimental relation to find the losses in this section. In diffuser there are two types of losses, Friction (K_f) and expansion (K_{ex}) losses. The friction factor and density assumed constant along this section. The equations as in **(Pope and Harper 1966)** are,

$$K_f = \left(1+\frac{1}{A_R^2}\right)\frac{f}{8\times\sin\theta}$$

Friction factor could be calculated from moody diagram which its value at Re = $33*10^6$ is f = 0.01; K_f = 0.019. The expansion losses could be calculated from equation,

$$K_{ex}= K_e(\theta)\left(\frac{A_R-1}{A_R}\right)^2$$

K_e=0.1709-0.117θ - 0.03260θ^2 – 0.00178θ^3 – 0.0009076θ^4- 0.000013331θ^5 + 0.00001340θ^6

For θ = 3.8° and AR = 4.527 the expansion losses is K_{ex}= 0.0580 and total diffuser losses is,

K_d= K_e + K_{ex} = 0.019+0.0580 = 0.077

As show in calculations the two solutions are approximately same. The increase in losses is due to boundary layer, which is not considered in the previous calculations.

Settling Chamber and Straightener:-

The dimensions of settling chamber are,

- length of section = 1500 mm
- Entrance area (2000x2000) mm^2
- Three stage of screen

For strengthener the dimensions are,

- length = 450 mm
- Entrance is (700x700) mm^2
- One stage of screen

Screens are used to decrease the turbulence in the flow due to the fluctuating velocities in three components. So that the characteristics of screens are

- (8*8) 64 mesh pre cm^2
- The wire diameter of 0.39 mm

The losses in settling chamber may be calculated by taking the entrance velocity which is 8.4 m/s as calculated in contraction. The Reynolds number at this section is Re = 225.455 and from **(Pope and Harper 1986)** the losses calculation for Settling chamber and straitghtener are K = 2.2 with respect to local velocity of Settling chamber. For three stage the total losses in Settling chamber and straightened are K = 0.095 with respect to the dynamic pressure at the test section.

Entrance

The dimensions of entrance are:

- Inlet hydraulic diameter equal to 2000 mm (2000x2000) mm.
- The inlet bell mouth radius equal to 280 mm.
- Length of section 1200 mm.
- to enhance the flow enter to this section and decrease the flow velocity the dimension of entrance area increased 10 mm so that the dimension becomes 2100 mm height and 2100 width

Losses in this section becomes as,

TUNNEL ENTRANCE DIMENSION:-
======================
L = 1.2000 m

W1= 2.1000 m
H1= 2.1000 m

W2= 2.0000 m
H2= 2.0000 m
AR = 0.9070

INPUT PROPERTIES:-
==================
M_IN = 0.0220
V_IN = 7.6190 m/s
P_IN = 101300.0000 Pa
T_IN = 298.4000 K
RO_IN = 1.1828 Kg/m3
PO_IN = 101334.3358 Pa
TO_IN = 298.4289 K

OUTPUT PROPERTIES:-
==================
M_OUT = 0.0243
V_OUT = 8.4004 m/s
P_OUT = 101292.1583 Pa
T_OUT = 298.3938 K
RO_OUT = 1.1828 Kg/m3
PO_OUT = 101335.2135 Pa
TO_OUT = 298.4289 K

CONTRACTION LOSSES K:-
=====================
K_E = 0.0003 This represents the entrance pressure losses.

Fan

A major work in designing fan is that it must provide a required velocity at test section (70 m/s) and to resist the decrease in pressure along the wind tunnel. The total pressure loss can be calculated as:

$TP = K_E + K_{SC} + K_N + K_{TS} + K_D = 0.0003+0.095+0.006+0.21+0.077 = 0.1993 \approx 0.2$

Usually a safety factor may be taken as 25% thus;

$K_T = 0.25$

$$\Rightarrow \Delta P = K_T \times 0.5 * \rho_0 \upsilon_0^2 \Rightarrow \Delta P = 750.3125 \quad Pa$$

So that, the total pressure loss at fan could be found by:

$A_{fan} = 1.767\ m^2$; $V_{fan} = A_{bs} V_{ts} / A_{fan} = 1904.1$ m/s

$P_f = 0.5\ \rho_0\ V^2_{fan} = 0.5 * 1.225 * 19.41^2 = 230.75$ Pa

$\Delta P_T = \Delta P + P_f = 750.3125 + 230.75 = 981.0025$ Pa

The power required is :

Power = $A_F * V_F * \Delta P_T = 1.767 * 19.41 * 981.0025 = 33647.96$ W

Power (hp) = 33647/ 750 = 44.86 $\approx$ 45 hp

For BHP = 60% approximately then

EFF = power (hp)/ BHP $\Rightarrow$ BHP = 74.7667 $\approx$ 75 (hp)

This power is more than that needed, but for future work when velocity increases to 100m/s at test section it may give the required power.

The following are dimensions and characteristics of fan;

- Axial simple fan
- Hub diameter 500 mm
- Blade diameter 1500 mm
- Blade length is 450 mm
- Number of blades are 12
- Tilt angle of blade is 30°
- Length of convex shape of blade is 300 mm
- Outer diameter of fan is 1510 mm
- Number of revolution per minute is 1500 rpm.
- Steel sheets with thickness equal to 6mm are used in manufacturing the blades.

Wind Tunnel Simulation:

In recent years, **ANSYS** represents a good tool to find a mechanical and flow properties for complex configuration. Therefore it was used to simulate the flow inside wind tunnel of the present work.

The first step in simulation is to model the wind tunnel and to create its parts to apply the boundary conditions. These boundary conditions must be applied accurately to ensure good results. **Figs. (4, 5, and 6)** show **ANSYS** steps for the present wind tunnel (Modeling ,boundary condition, meshing and solving using **FLOTRAN** three dimensional solutions).

ANSYS solution shows that test section is approximately have a constant velocity near (70 m/s) along test section as shown in **Fig. (6),** which represents most important part of the wind tunnel. This indicate that our design is fair enough since there is no flow separation in velocity at test section or at least no thickening boundary layer at this region which may cause an error in measurement.

Wind Tunnel Construction

An open circuit wind tunnel was designed and constructed incorporating features discussed in previous articles. The following group figures show the construction of parts of the wind tunnel and its final shape.

Test Section

A test section was designed to give a velocity of 70m/s and size equal to (700x700) mm^2 inlet area and 1500mm long section. Transparent Plexiglas was used entirely in its construction to provide viewing of the model. A square cross section with upper and lower surface diverges along the section to balance the boundary layer growth. This may provide a constant pressure along test section and prevent error in measurements. A steel form was constructed and used to hold the Plexiglas's and prevent them to make convex shape due to low pressure at test section (less than atmospheric pressure).

Contraction and Settling Sections

The contraction shape and settling section were made of steel galvanized plate to prevent corrosion and it's strong enough to give safety work with velocity 70m/s. Both have square cross section. The contraction ratio is (8.16:1). The wall shape was designed using the previous equations. The settling section contains [(8x8)64 – mesh per cm^2 and diameter 0.39mm]. Both sections are mounted on stands held on the ground by bolts.

The contraction section is joined to test section by a mating flange which compresses an O-ring rubber there by sealing joints.

Inlet Section

Inlet section and bell mouth are made of galvanized steel. The bell mouth increases the efficiency of air inlet and prevents what is called vena contracta. All wall plate of inlet section made pyramid shape with inlet and outlet area. Flanges are constructed at ends of parts with O-ring sealing and lifted from ground by stand structure holder on the ground.

Diffuser

The diffuser was made of galvanized steel plate with circular cross section so that, there must be converting sections which convert the square section at test section to the circular cross section of the diffuser. So it was decided to make the diffuser and converter section in same part. This gives a gradually grading of transformation. The diffuser then has a square cross section at inlet and circular cross section at exit. A constant divergence angle of (7.6°) between upper and lower surfaces was made. The diffuser exit matches the fan inlet with (1510mm) diameter.

Four crosse shape plates placed inside diffuser before fan section and holder on the diffuser wall. These plats prevent the swirling action of the flow due to fan rotation and decrease losses in fan work

Power Plant:-

An axial fan driven by a (75hp) (1500rpm) amplitude lament motor provides the power to the tunnel. The fan is mounted at the end of tunnel and swallows the flow from tunnel inside and discharged it to outside. The power plant section mounted on the ground by a structure and connected with diffuser with a double sealing rubber to prevent vibrational motion during its operation. The fan manufactured by steel of 6mm thickness and 12 vanes with probability to change or maintenance.

Power Supply and Motor Speed Control:-

In order to have motor speed control from zero to 1500rpm a power supply and motor speed control were provided to the wind tunnel power plant.

CONCLUSIONS

An open circuit wind tunnel has been designed, simulated and constructed to obtain as speed of 70 m/s at test section of (0.7x0.7) m^2. The design procedure shows the wind tunnel losses are approximately equal to 0.25 with respect to the dynamic pressure at test section. A 75 HP AC motor with regulator are used to control its velocity from 0-70 m/s. The wind tunnel was simulated and solved using ANSYS commercial program. The results show that the test section has a constant velocity along it. The construction is then being done using the dimensions of the wind tunnel. The test work and calibration will be done in the next part of the paper.

ACKNOWLEDGEMENT

We would like here to acknowledge the (Research and Development (R & D) Office at the Iraqi Ministry of Higher Education and Scientific Research MOHESR), which adapted the present project as one of the pioneer projects, and supported it financially.

REFERENCES

Bradshaw P. and Mehta R. D., “Design Rules for Small Wind Tunnels”, Aeronautical Journal, November 1979, p.p. 443-449.

Frank, P. B., “Handbook Selection, Application and Design”, McGraw-Hill, 1997.

Milan Vilaginac, “Design, Construction and Evaluation of a Subsonic Wind Tunnel”, M.Sc. thesis, MIT, June 1970.

Pop a. and Harper j. j., “Low Speed Wind Tunnel Testing”, McGraw-Hill, 1986.

R. Allan Wallis, ”Axial Flow Fan and Ducts”, John Wiley and sons, 1983.

Raikan S. Dawas, “Proposal for Subsonic Wind Tunnel Suitable in Iraq”, M.Sc.Thesis, University of Baghdad, September, 2001

Schlichting, H., "Boundary Layer Theory", McGraw-Hill, 6th Edition, 1968.

Shapiro, A. H., "The Dynamic and Thermodynamic of Compressible Fluid Flow", Vol I, Ronald Press, New York, 1953.

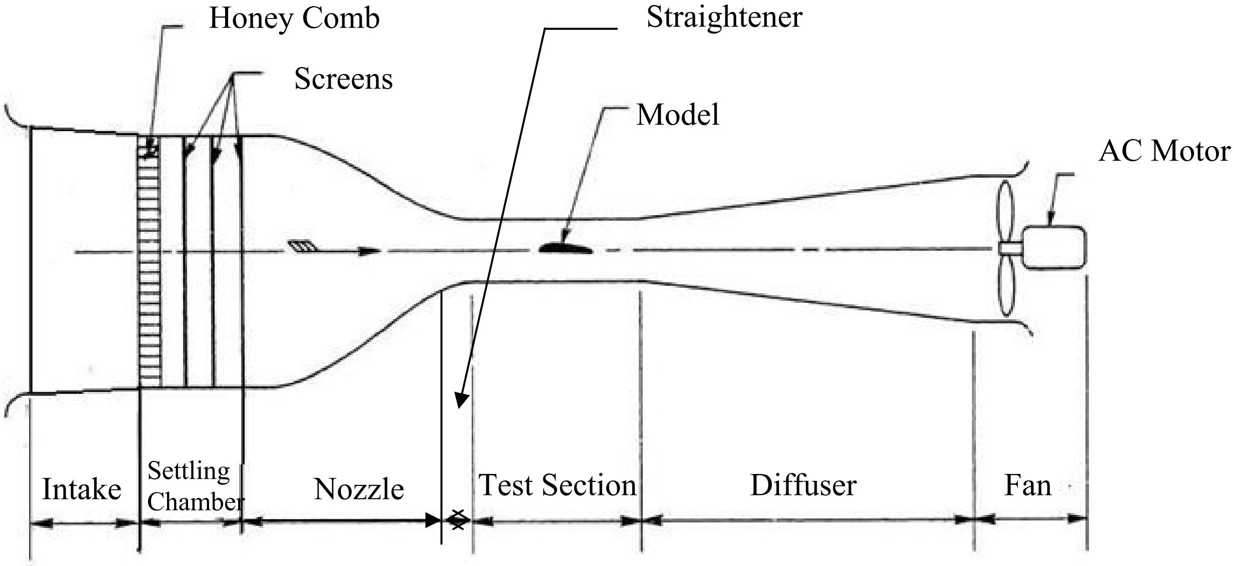

Fig.1: Wind Tunnel Parts.

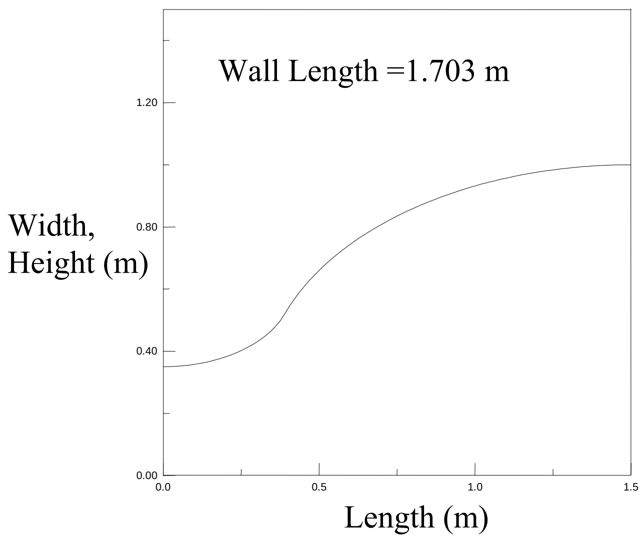

Fig.1: Contraction wall shape.

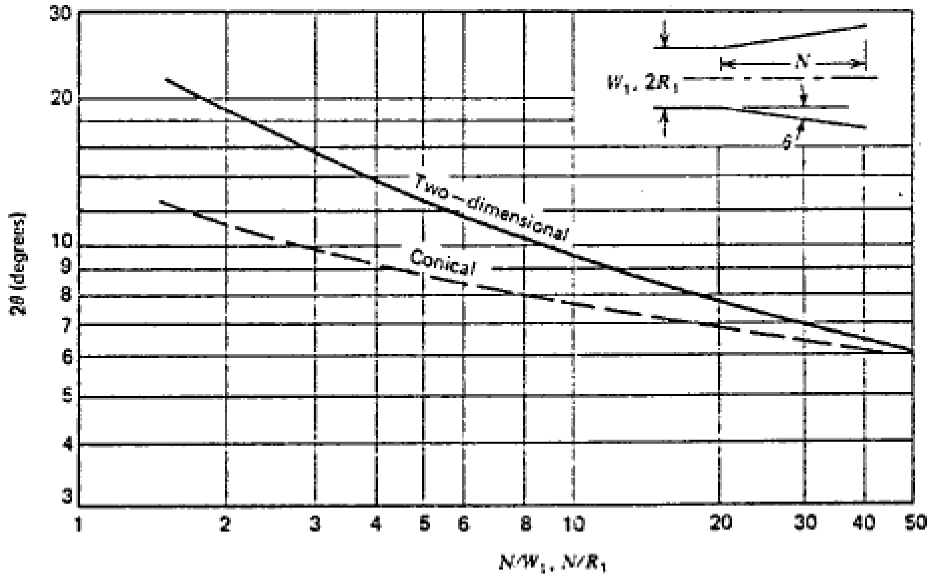

Fig.3: Diffuser Divergence Angle. (Wallis 1983)

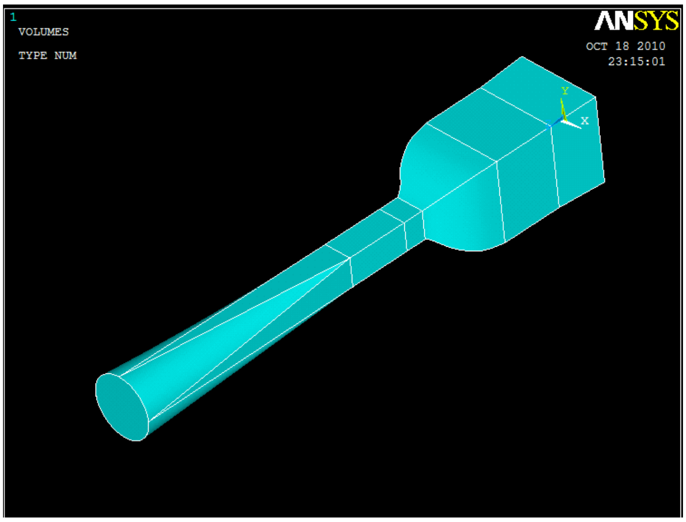

Fig.4: ANSYS Modelling of Wind Tunnel

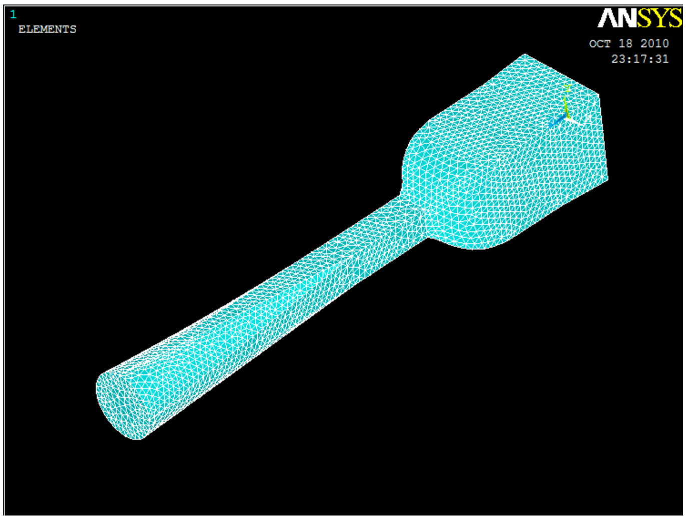

Fig.2: ANSYS Meshing of Wind Tunnel

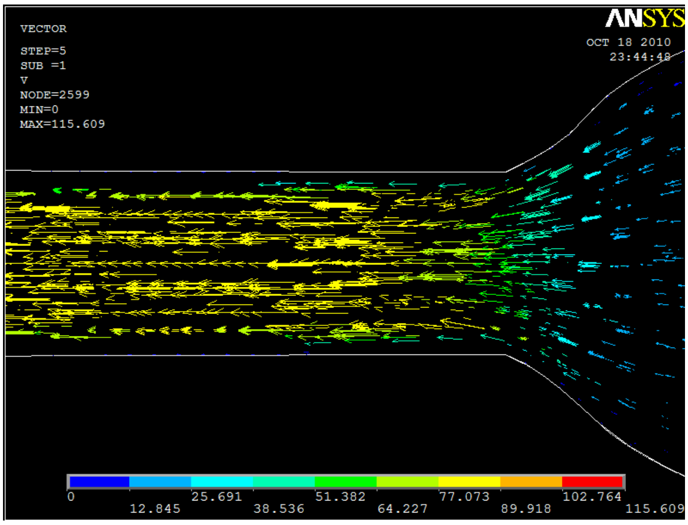

Fig.3: ANSYS Test Section Solution.

Fig.4: Wind Tunnel Construction.

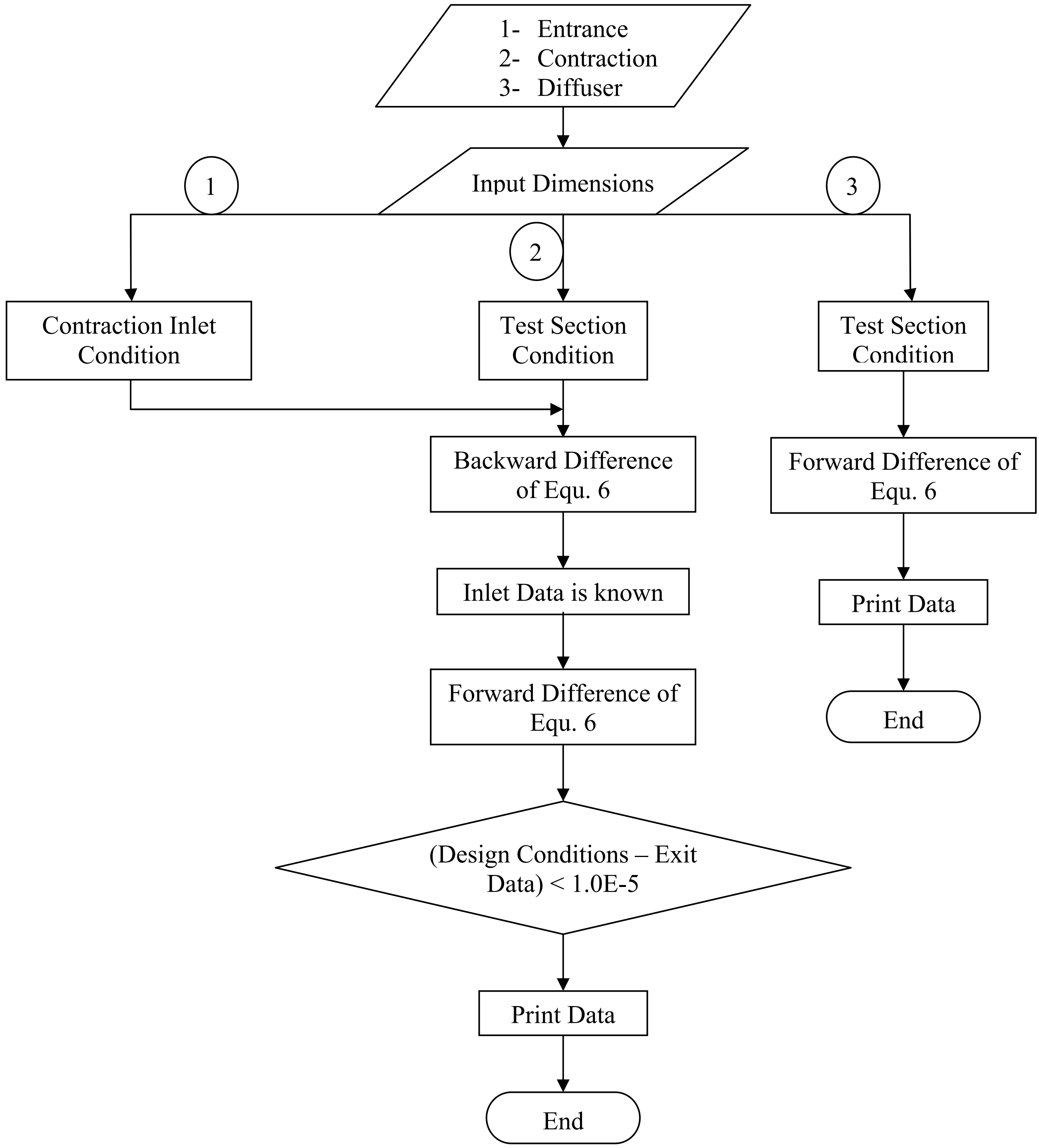

Fig.8: Flowchart of Computer Program.

Nomenclature:

Latin Symbols

A_0	Cross-section area test section	m^2
AR	Aspect ratio	
CD	Drag coefficient	
D_H	Hydraulic diameter	M
E.R	Energy ratio	
E_0	Energy at test section	W
f	Friction factor	
K_i	Pressure drop coefficient	
L	Length	M
M	Mach No.	
Re	Reynold No.	
v_0	Velocity at test section	m/s
ΔP	Pressure difference	Pa
ρ_0	Density at test section	Kg/m^3

Greek Symbols:

□	Power	W
δ^*	Displacement thickness	M
η	Efficiency	
θ	Divergence angle	Deg.
μ	Viscosity	Pa.s