UNIVERSITI PUTRA MALAYSIA

TURBULENT FLOW IN AN ACTIVE WIND-DRIVEN VENTILATION DEVICE

ABOLFAZL SOUSANABADI FARAHANI

FK 2009 88

TURBULENT FLOW IN AN ACTIVE WIND-DRIVEN VENTILATION DEVICE

By

ABOLFAZL SOUSANABADI FARAHANI

Thesis Submitted to the School of Graduate Studies, Universiti Putra Malaysia, in Fulfilment of the Requirement for the Degree of Master of Science

October 2009

Abstract of thesis presented to the Senate of Universiti Putra Malaysia in Fulfillment of the requirement for the degree of Master of Science

TURBULENT FLOW IN AN ACTIVE WIND-DRIVEN VENTILATION DEVICE

By

ABOLFAZL SOUSANABADI FARAHANI

October 2009

Chairman: NOR MARIAH BT ADAM, PhD

Faculty: Engineering

Growing concern on environmental issues has prompted house owners and industries to consider use of roof top ventilators, as a passive form of quality air circulation and comfort using only wind energy. However, many of these ventilators have evolved through trial and error and the flow physics associated with these ventilators is barely understood.

This study presents prediction of airflow using Computational Fluid Dynamics (CFD) technique code, FLUENT, so as to visualize the flow behavior around and within turbine ventilator in addition to determining the aerodynamic forces acting on a turbine ventilator during operation and comparing the simulated results to available

experimental data. The prototype used for this investigation is a wind driven ventilator from Edmonds Company with a rotor diameter of 330 mm and base diameter of 150 mm. The free stream velocities in visualization of flow are set to be 7 and 20 $m.s^{-1}$ when, for determining the aerodynamic forces are considered to be 7, 10, 14, 20 and 25 $m.s^{-1}$ corresponding to experiment. The simulated prototype is placed in a control volume with the same dimensions as open circuit wind tunnel used in experimental investigation. Also the operating pressure and fluid properties are set to be the same as experiment. Standard k-ε, Realizable k-ε, SST k-ω and RSM turbulence models are used by taking advantage of moving mesh method to simulate the rotation of turbine ventilator and the consequent results are obtained through the sequential process which ensures accuracy of the computations.

The results demonstrated that, the RSM turbulence model shows the best performance on flow visualization and predicting the aerodynamic forces acting on a turbine ventilator. Results from this study, besides ensuring the reliability of utilizing the CFD method in design process of future turbine ventilators, would lead us to a conspicuous progress on increasing the efficiency at reduced cost of wind driven ventilators and similar devices.

Abstrak tesis yang dikemukakan kepada senat Universiti Putra Malaysia sebagai memenuhi kepeluan untuk sarjana

ALIRAN BERGELORA DI DALAM PERANTI PENGUDARAAN ARUS TIUPAN ANGIN AKTIF

Oleh

ABOLFAZL SOUSANABADI FARAHANI

Oktober 2009

Pengerusi: NOR MARIAH BT ADAM, PhD

Fakulti: Kejuruteraan

Kesedaran manusia tentang isu alam sekitar pemilik rumah dan industri telah mendorong mengguna turbin pengalihudaraan bumbung, sebagai mekanisme pasif yang hanya memerlukan tenaga angin bagi menjamin mutu pengalihudaraan dan keselesaan. Namun demikian, kebanyakan pengudaraan ini berevolusi secara cuba-cuba (trial and error) dan kajian terhadapnya kurang difahami.

Kajian ini mengetengahkan simulasi pergerakan udara menggunakan kaedah dinamik bendalir berkomputer (CFD) bagi memperlihatkan mekanisme aliran di sekeliling dan di dalam turbin pengudaraan semasa operasi dan membandingkan hasil simulasi dengan data kajian sedia ada. Prototaip yang digunakan di dalam kajian ini ialah dengan diameter rotor berukuran 330 mm dan diameter tapak berukuran 150 mm

daripada Syarikat Edmonds. Aliran bebas digambarkan dalam halaju yang ditetapkan pada 7 dan 20 $m.s^{-1}$ apabila, untuk menentukan daya aerodinamiknya dipertimbangkan pada 7, 10,14, 20 dan 20 $m.s^{-1}$ merujuk kepada eksperimen. Simulasi prototaip ini diletakkan dalam volum kawalan yang mempunyai dimensi yang sama dengan terowong arus tiupan terbuka yang digunakan di dalam kajian ujian. Sifat-sifat tekanan dan bendalir kendalian juga ditetapkan sama seperti eksperimen. Model gelora standard k-ε, k-ε, k-ω SST dan RSM digunakan dengan mengambil kira kaedah jaringan bergerak untuk mensimulasikan putaran turbin pengudaraan dan hasilnya diperoleh melalui proses berurutan bagi memastikan ketepatan pengiraan.

Keputusan menunjukkan kaedah model gelora RSM memberikan prestasi terbaik dari segi menggambarkan aliran dan meramal daya aerodinamik yang bertindak terhadap turbin pengudaraan. Hasil daripada kajian ini, selain daripada memastikan kebolehpercayaan penggunaan kaedah CFD dalam reka bentuk turbin pengudaraan masa depan, juga membawa kepada peningkatan tahap kecekapan pada kos yang lebih rendah untuk pengudaraan arus tiupan angin dan peranti yang serupa.

ACKNOWLEDGEMENTS

I would like to express my appreciation for those who gave me an opportunity to complete this thesis. I am deeply indebted to my supervisor Associate. Prof. Dr. Nor Mariah bt Adam for her close guidance, inspiration and invaluable help throughout the study. Also, I would like to thank Dr. Mohd Khairol Anuar Bin Mohd Ariffin and Dr. Megat Mohamad Hamdan B Megat Ahmad as the co-supervisors for their kindness, information and suggestions during the project research.

I express my deepest gratitude to my lovely family and my adored Rey, for their continuous encouragement, understanding and support.

TABLE OF CONTENTS

LIST OF TABLES

Table **Page**

LIST OF FIGURES

LIST OF ABBREVIATIONS

CAD	Computer Aided Design
CAE	Computer Aided Engineering
CFD	Computational Fluid Dynamics
CPU	Central Processor Unit
DNS	Direct Numerical Solution
FSM	Fractional Step Model
LVT	Long Volume Turbines
NITA	Non-Iterative Time Advancement scheme
PISO	Pressure-Implicit with Splitting of Operators
RSM	Reynolds Stress turbulence Model
SST	Shear Stress Transport model
TSR	Tip Speed Ratio

NOMENCLATURE

k	Kinetic Energy
ε	Turbulent dissipation rate
Re	Reynolds number
ω	Specific dissipation rate
ρ	Density ($kg.m^{-3}$)
y^+	Dimensionless wall distance
y	Distance from the wall (m)
μ	Fluid dynamic viscosity ($kg.m^{-1}s^{-1}$)
u_τ	Friction velocity (ms^{-1})
C_d	Drag coefficient
C_l	Lift coefficient
P_s	Surface pressure (pa)
A_x, A_y, A_z	Projected surface areas (m^2)
U_∞	Free stream velocity (ms^{-1})
P_∞	Free stream static pressure (pa)
P	Local static pressure (pa)
λ	Tip Speed Ratio
U_T	Tangential velocity ($rad.ms^{-1}$)
I	Turbulence intensity
$\acute{u}$	Root-mean-square of the velocity fluctuations (ms^{-1})
u_{avg}	Mean flow velocity (ms^{-1})

ℓ	Turbulence length scale
Ω	Angular velocity ($rad.s^{-1}$)
v_f	Tunnel wind speed (ms^{-1})
θ	Degree in radian
$\emptyset$	Variable
α	Under relaxation factor
$\dot{m}$	Mass flow rate ($kg.s^{-1}$)

CHAPTER 1

INTRODUCTION

1.1 Overview

The diminution of greenhouse gases is required for the developed countries. However, it can be predicted that developing countries including Malaysia will be demanded to adhere to the Kyoto Protocol in the near future. Therefore, it is important to consider the energy saving means in developing countries in the course of its economic growth. The emissions produced from the use of air-conditioners in residential areas could be effectively performed through energy saving efforts by maximizing the use of passive cooling techniques or by means of natural ventilation. Consequently, the importance of natural ventilation has been increasingly reassessed partly due to the recent needs of energy saving.

1.2 Passive cooling techniques for hot-humid climates

The term "passive cooling" is referred to as a building design method that not only avoids outdoor heat, but also transfers indoor heat to natural heat sinks. Complete reviews in passive cooling study can be found in Cook (1989) and Abram (1986), where passive cooling techniques are categorized as follow:

1. Heat avoidance
2. Radiative cooling
3. Evaporative cooling

4. Earth coupling
5. Ventilation

1.2.1 Heat avoidance

Heat avoidance technique consists of the use of shading devices, suitable building orientation and the use of local vegetation as a simple means of reducing heat gain, (Balaras, 1996).

1.2.2 Radiative cooling

Radiative cooling method, as explained in Cook (1989), is the process whereby heat is absorbed by buildings in the daytime, and then radiated later to the cooler, night sky as infrared radiation. This technique works best in arid climates where diurnal temperature swings are significant. For hot-humid regions, high humidity and cloud cover usually slows the rate of night time radiative heat transfer, thus trapping heat inside the buildings that would have otherwise radiated to the night sky.

1.2.3 Evaporative cooling

Evaporative cooling is another technique that is currently used in passively cooled buildings in hot-arid regions. Unfortunately for hot-humid regions, high humidity prevents evaporative cooling from being effective. As described by Cook (1989), "Evaporative cooling works when the sensible heat in an air stream is exchanged for the latent heat of water droplets or wetted surfaces." However, in hot-humid climates like Malaysia, cooling with outdoor air without first removing moisture (such as with a desiccant cooler) causes the indoor air to be too humid or even to condense on surfaces, and thus causes mold and mildew to form.

1.2.4 Earth coupling

Regarding earth coupling techniques, Cook (1989) has summarized that in earth-coupled buildings, the interior space is thermally coupled to the subsoil by conduction-convection through the building slab. This requires that the ground temperature be within the comfort zone (i.e., 20 – 26/°C) so that the ground can act as a heat sink. This technique is useful in temperate climates where the average ground temperature is within the comfort zone.

1.2.5 Ventilation

In ventilation, as mentioned in Abram (1986), a cooling effect occurs by means of convection by using surrounding air as a heat sink. A lack of ventilation can cause too much humidity, condensation, overheating and creation of odours, smokes and pollutants. In commercial and industrial buildings ventilation is a part of HVAC (heating, ventilation and air-conditioning) systems which are very energy intensive; usually including of large fans, air-conditioning and heating components. In domestic buildings the most important ventilation technique is renewable in the form of air infiltration and natural ventilation through windows and openings.

1.3 Natural form of ventilation

Natural ventilation uses the natural forces of wind pressure and stack effects to redirect the movement of air through dwellings. Wind incident on a building facade produces a positive pressure on the windward side and a relative negative pressure on behind. This pressure difference beside the pressure differences inside the building will cause airflow to move. Stack effects are due to the temperature differences between the inside and outside of buildings. As long as the inside building temperature is more than the outside, warm indoor air will rise and exit then being replaced by cooler, denser air from a lower height. The stack effect is foremost during periods of low wind speed and reduces in summer periods when temperature differences are negligible. Natural ventilation is now one of the main methods in the energy efficient design of buildings.

Various wind driven ventilation techniques are used in energy efficient building design, (Khan, 2008), and they are classified as:

1. passive wind driven ventilation
2. directed passive wind driven ventilation
3. active wind driven ventilation

1.3.1 Passive wind driven ventilation

Devices and methods in this category are passive in nature and mainly using wind-induced effects as drive forces for providing ventilation. Some examples of these devices and methods are; window openings, atria and courtyards, wing walls, chimney cowls, wind towers, wind catchers, wind floor and air inlets, (Khan, 2008).