Evaluate the Airflow Propagation on the Buildings Using Computational Fluid Dynamics (CFD)

Ehab Hanafi Mahmoud*, Mohamed Ali Barakat, Hany El-Hady Ibrahim, Eman Ali Ibrahim

Abstract— Studying airflow around and through the buildings regards today from the most complicated studies, which there is no simple equations can predict airflow pattern and its behavior among buildings layout. Airflow affected by many factors like building shapes, heights, interior distances between buildings, wind direction, urban setting position and so on. The investigation of the airflow impact on the buildings provides specific measurements of the airflow properties without compromising appearance and utilities of the buildings and the architectural implications of such geometric.

This paper presents the finding from a program of research, which explores a big opportunities offered by an environmental studies approach. Moreover, explores guidance criteria for evaluating airflow impact on building using computational approach of infinite element package known as Computational Fluid Dynamics (CFD) simulation package.

Index Terms— Airflow, CFD, Environmental studies, Architecture, Wind impact, Wind tunnel, Protection.

---

1 Introduction

It is necessary to examine the characteristics of wind around buildings, building sites and streets both for urban planning and for each building's architecture design. It has recognized that the wind environment around building sites is one of the various environmental characteristics belonging to a void. In other words, the overall wind environment in an urban block thought to be a comprehensive assessment of airflow in each void.

It is great challenge of the engineerings and architects to reduce the wind impact on the tall buildings.Especially the design of group of tall buildings is the most important consideration to take care of the housing problem of the huge population. As the building becomes tall it is necessary yo take into consideration the affect of wind its design.
The subjects of wind load on buildings and structures are not new one. Inthe 17th century, Galileo andNewton has considered the effect of wind impact on buildings, but during that period. The effect of windimpact on buildings and structures has been considered for design purposes since late in the 19th century;but starting from that time up to about 1950, the studies in this field have not been considered seriously. [1]

The study of wind effect was first limited to loading and structures only, possibly because of its most effects are seen in their collapses. Researcers started the study of less dramatic, but equally important environmental aspects of flow of wind around buildings. These included the effects on pedestrains, weathering, rain penetration, ventilation, heat loss, wind noise and air pollution etc. [1]

Buildings and their components are to be designed to withstand the code specified wind impact. Calculating wind impact is important in the design of wind force resisting system, including structural members, components, and cladding against shear, sliding, overturning and uplift actions.

- *Ass. Prof. Dr. Ehab Hanafi Mahmoud*
- *is Head of the Architecture & Civil Branch, Military Technical College, Cairo, Egypt.*
  Cell: +2 0100 613 8182 *E-mail: ehabhmd@yahoo.co.uk*

Wind tunnel

A wind tunnel is a research tool developed to assist with studying the effects of airflow moving over or around solid objects. Air is blown or sucked through a duct equipped with a viewing glass port and instrumentation where models or geometrical shapes where mounted for study. Various techniques are then used to study the actual airflow around the geometry and compare it with theoretical results.[2]

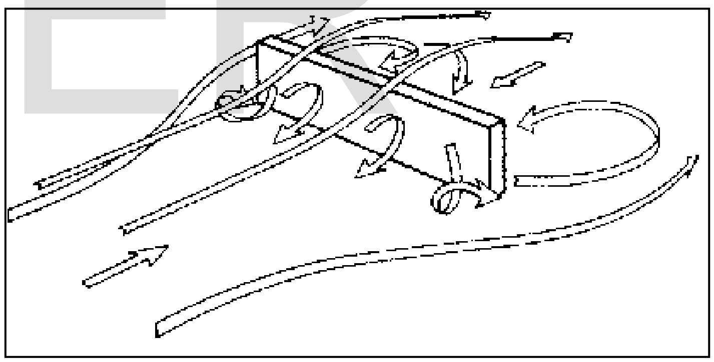


Fig.1 Wind near and around building as it expected.

Wind tunnel modeling using scale models has been the primary tool used to address these impacts for some time. Though proven to be accurate and reliable, this approach is not always feasible. In addition, since measurements can only be taken at discrete points in the model, the more accuracy needed, the more expensive the modeling becomes.

Such studies have typically been performed using wind tunnels and scale models or qualitative techniques based on past experience. In these wind tunnel studies, scale models are exposed to winds at different angles and wind speeds at critical areas are recorded using hot-wire anemometers. [2]

ISSN 2229-5518

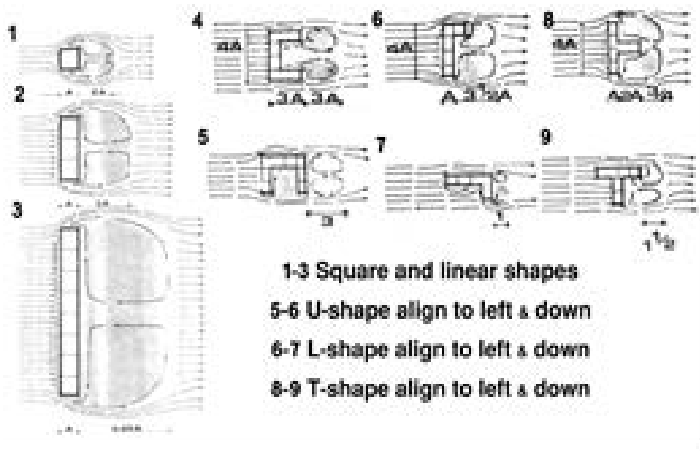


Fig. 2 a previous works for scaled models investigated by wind tunnel experiments [3]

Figure 1 presents the classification for some of buildings types which have been proven to be accurate and representative of the actual environment if the modeling technique is done correctly. However, wind tunnel modeling cannot return a full 3-dimensional view of the wind vectors and other flow features in the model.

The more accuracy that is needed, the more sensors are needed. More sensors mean added expense and more corruption of the flow itself from interaction of the sensors with the flow field. [3]The advancements in computing power today have allowed for an alternative computational approach of wind tunnel modeling known as Computational Fluid Dynamics (CFD).

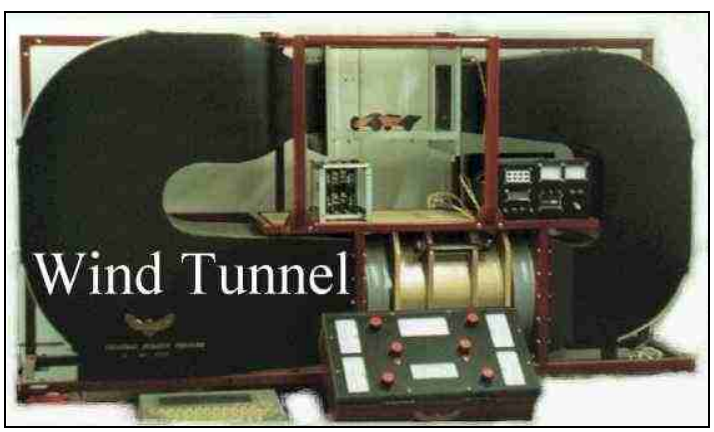


Fig.3 A Wind Tunnel used to study airflow.

2 Methodology of using Computational Fluid Dynamic (CFD)

In recent years, computational fluid dynamics (CFD), has shown promise to become a reliable alternative to wind tunnel modeling. CFD is essentially a computerized wind tunnel that solves the equations of motion to give a steady-state solution of the wind field in a model. Though the steady-state solution is directly not that useful in wind evaluation, estimates of gust speed can be calculated using the calculated values of Turbulent kinetic Energy (TKE) at locations in the model.

Though not as well established as wind tunnel modeling, CFD offers a viable alternative that has advantages. CFD allows the investigator to analyze the full domain of modeling and present results in easy-to-understand graphics rather than several dozen discrete points, as with physical wind tunnel modeling. CFD modeling has been used by Environ metrics to assess comfort levels with respect to wind climate. [4]

ANSYS Computational Fluid Dynamics (CFD) simulation software allows predicting, with confidence, the impact of fluid flows on the buildings — throughout design and manufacturing as well as during end use. The software's unparalleled fluid flow analysis capabilities can be used to design and optimize new equipment and to troubleshoot already existing installations. ANSYS fluid dynamics solutions give valuable insight into building's performance.

ANSYS fluid dynamics solutions are fully integrated into the ANSYS Workbench platform. This environment delivers high productivity and easy-to-use workflows. Workbench integrates all your fluid workflow needs (pre-processing, simulation, and post-processing) as well as multiphysics functionality (fluid–structure interaction, electromagnetic/fluid coupling). Furthermore, the ANSYS Workbench environment allows for automated and easy-to-set up optimization or design exploration studies [1]

Understanding the effect of urban setting height and layout on the surrounding building fluid is paramount in the production of wind impact on building. The purpose of this study is to create a basis to understand Computational Fluid Dynamics (CFD) programs. A simple rectangular two-dimensional structure was chosen. The layout used was that of a large domain with an inflow, outflow, upper wall, and lower wall with building. [5-6]

3 Simulation set- up

The building is incorporated into the lower wall if we illustrate it as an elevation of two-dimensional structures or incorporated into the middle of computational domain if we illustrate it as a plan of two-dimensional structures. Both Gambit and Fluent programs [6], interprets the building's properties as if it were a wall. The geometric representation is exported as a mesh and saved. Once in Fluent, the file is read, proper-

ISSN 2229-5518

ties and materials are defined, and boundary conditions are set. Solution parameters are chosen and the solution is iterated. The solution is then displayed.

In Gambit, the structure was built. In building, the domain vertices were created, marking the endpoints of each line. To facilitate the meshing extra vertices were placed along the inflow, outflow and upper wall in conjunction with the building height and width. [7]

The node distribution and mesh edges were set with gradient spacing and interval count for the opposite walls which were sized the frequently. For specific model display, attributes the boundary types were set as follows:

a) Inflow was set as a mass flow inlet.
b) Top and bottom walls were set as walls.
c) Outflow was set as out flow.

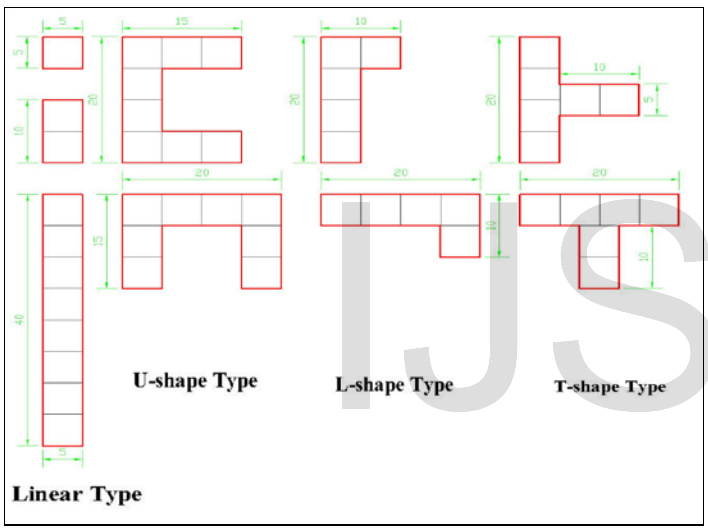


Fig. 4 a simple Tow-Dimensional Structures

This file was saved and exported as a mesh file to be read in Fluent program. Fluent 2D was selected as the CFD solver in which the mesh file was read. The model was defined as Spalart-Allmaras with one Equation along the upper and lower walls. The material selected was air. The boundary conditions were such that the pressure was at 1 atmosphere normal pressure with inlet velocity of 1 m/s. The direction of flow was normal to the inlet boundary. The solution was initialized and the iteration was selected to be 1000. [7]

4 Results

4.1 Contours of Total Pressure:

In this display, the pressure is increasing in the middle inflow region front of building geometry. This higher pressure continues both building sides. The lower pressure is found immediately behind the building. The upper and lower region of the computational domain appears to be unaffected by the presence of the building.

4.2 Contours of Velocity Magnitude

The velocity magnitude is unaffected close to the inlet side. As the solution approaches, the building there is an increase in pressure between the building and the upper wall. The velocity magnitude is greatly reduced near and past the building, creating an arch. A larger increase in velocity is found above this arch.

5 Analyses

Constructive information can be extracted. Importantly, it can be determined if and where on a structure the flow is sufficient enough for a particular wind turbine. With the development of a domain containing a structurally more complex layout, the effects of that layout on the wind flow can be examined. Applying two dimensional testing on a three dimensional problem has it limitations, however the information can be a useful tool in the selection of models that will be tested with a more expensive method, such as wind tunnel tests.

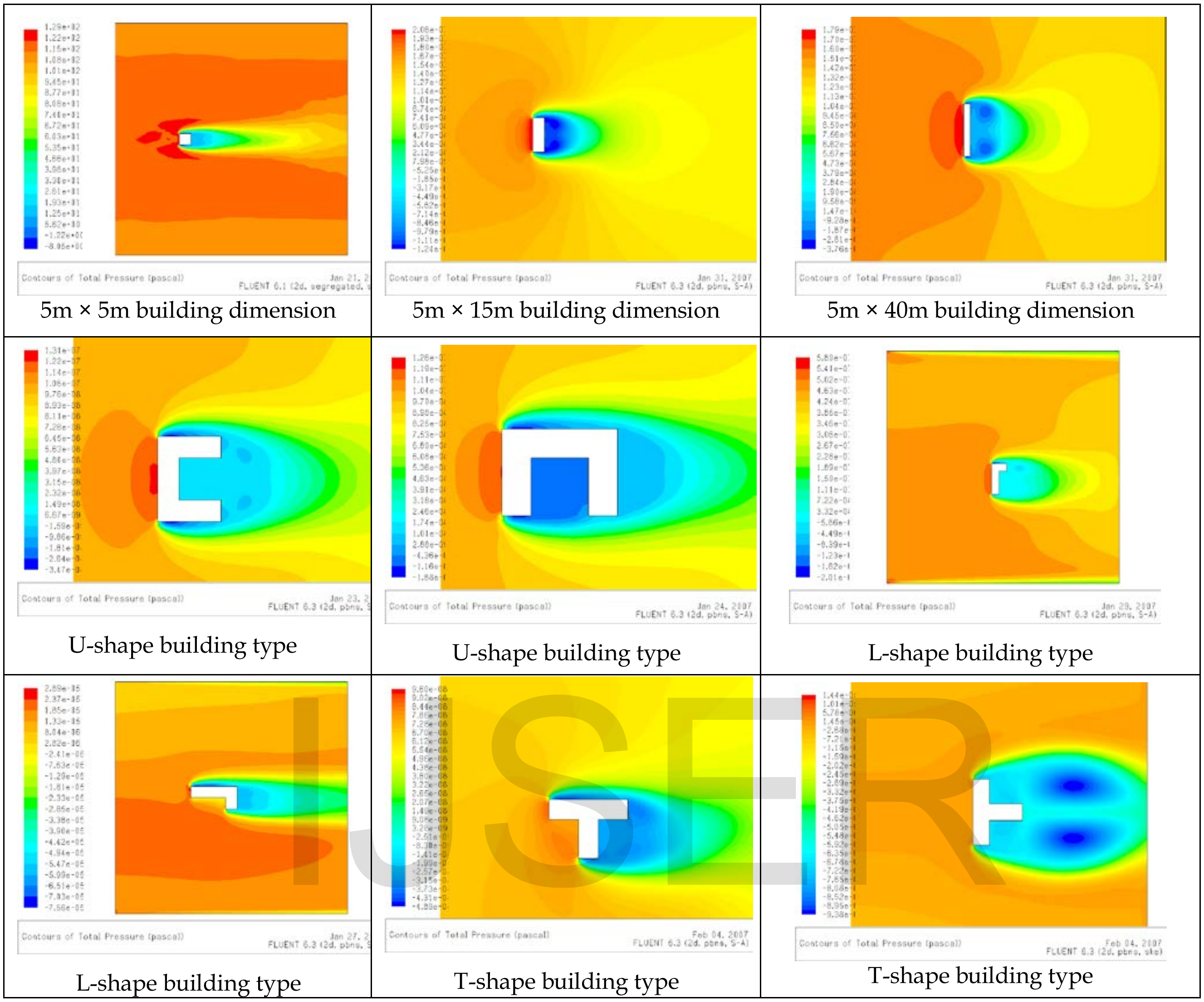
5.1 Contours of Pressure

Figure 5 presents typical simulations of pressure Contours for each model. It was found that the higher pressure on the inflow side occurs due to the presence of the building. This regional effect continues over the structure similar to the line of separation. This higher pressure influences the region after the structure. Past this structure, on the outflow area, there is a pressure drop. The drop occurs as a result of the fluid flow being blocked by the structure.

5.2 Velocity Magnitude.

Figure 6 presents typical simulations of Velocity Magnitude Contours for each model the inflow side remains unaffected until the flow approaches the structure. There is a region of velocity increase located between the upper wall and the top of the building. A reduction of space (the narrowing of the passage) creates an increase in velocity. Past the building is a stream of lower velocity. In this line of separation there, is an imbalance setting the higher velocity region in motion.

ISSN 2229-5518

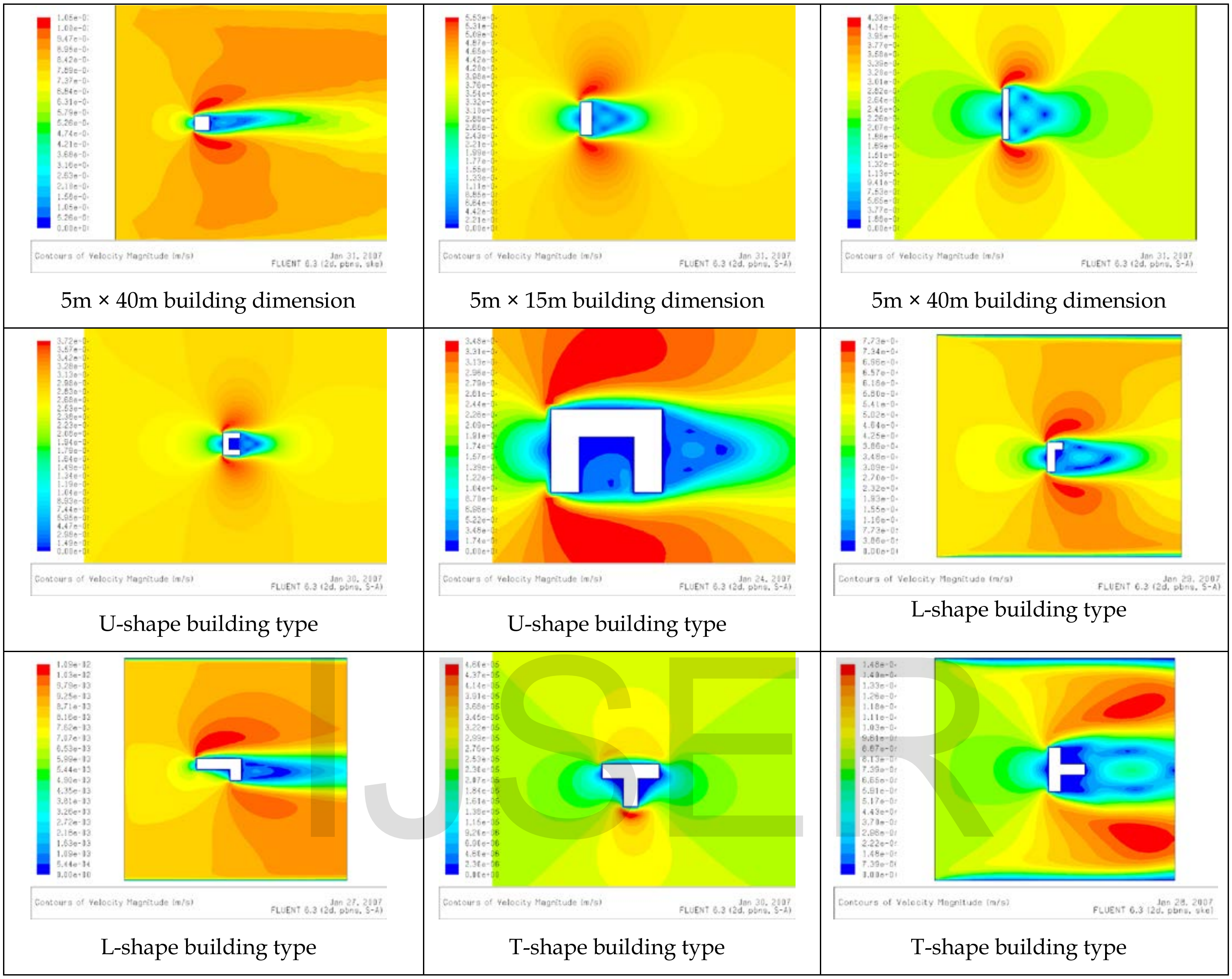


5m × 5m building dimension | 5m × 15m building dimension | 5m × 40m building dimension

U-shape building type | U-shape building type | L-shape building type

L-shape building type | T-shape building type | T-shape building type

Fig. 5 Typical simulations of pressure Contours for each model



5m × 40m building dimension | 5m × 15m building dimension | 5m × 40m building dimension

U-shape building type | U-shape building type | L-shape building type

L-shape building type | T-shape building type | T-shape building type

Fig. 6 Typical simulations of Velocity Magnitude Contours for each model

Here are the seven steps involved this methodology in conducting a CFD-based study of winds in an urban landscape:

1. A meteorological analysis of the wind climate in the selected area is conducted.
2. A mathematical model of the site is built for computerized CFD modeling.
3. A proper method of simulating the atmosphere in the model is determined.
4. Modeling is performed at different wind speeds and directions, all determined from the climatology study of the area.
5. Wind velocities and levels of turbulence are measured in the model under the different scenarios.
6. Graphics and results are displayed showing the level of impact in different areas.
7. Mitigation efforts, if needed, are developed and tested in the model.

ISSN 2229-5518

8. Risk management decisions are made to find the most economical solution that limits the risk of negative impacts.

6 Conclusion

1. In terms of performance prediction potential, there is no single best method. Each method has its own advantages and disadvantages. This declares that is the infinite elements simulations to use depend on the type of analysis that is needed at a particular stage in the design process.

2. A simulation quality assurance procedure is indispensable. Apart from the essential need for domain knowledge, parts of such procedure might be semi-automated.

3. Computational Fluid Dynamics (CFD) is a prediction tool that requires a detailed geometric model of the investigated air space and potentially it is suitable to predict dispersion phenomena where local geometry is significant forthe airflow.

4. Although much progress has been there remain many problematic issues in building airflow simulation. This is a very promising direction for future work.

References

[1]Md. Jomir Hossain, Md. Quamrul Islam and M. Ali "An Experimental Investigation of Wind Effects and Dynamic Behavior of an Octagonal Cylinder" International Journal of Material and Mechanical Engineering, (IJMME) Volumn 2 Issue 3, August 2013

[2] H. El-Hady Ibrahim, Ehab. H. Mahmoud, "Design and Planning Critiria to Evaluate Military Communities", research, MTC, Cairo, Egypt, 2009.

[3] A. El-kader & Nasmat, Dar Al-araby, "In planning & designing residuals zones, entrance & application", P&D, Cairo, Second edition, 2010

[4] J.D. Mc Alpine, "Criteria for Determining the Impact of Wind Climatology on Pedestrian Comfort in an Urban Setting " , Envirometrics, Inc. Seattle, WA held at the Nashville Convention Center, May 22-28, 2004.

[5] A. Jones, M. Abdullah "Computational Fluid Dynamics for Wind Energy Production in an Urban Setting Experiences for Undergraduates", Research; Japan Advanced Technology Florida A&M University-Florida State University College of Engineering, Civil Engineering, August 19, 2003.

[6] J.L.M. Hensen "Integrated buildings and airflow simulation: an over view ", The 9th Conference on Computing in Civil and Building Engineering, Taipei, Taiwan Center for Building & Systems, Technische Univ. Eindhoven, Netherlands, April 3-5, 2002

[7] B. Bjerg, S. Morsing, K. Svidt, G. Zhang, "Three-dimensional airflow in a livestock test room with two-dimensional boundary conditions.", Journal of Agricultural Engineering Research, 74, 267-274, 2010

Ass. Prof. Dr. Ehab Hanafi Mahmoud
PH.D., M.Phil, M.Sc., B.Sc.
- *Head of the Architecture & Civil Branch, MTC, Cairo, Egypt*
- *RMCS, Cranfield University, England, UK*
- *International Advisor CI-Premier, Singapoure*
- *Member, Research Town Plnning Institute, RTPI, England, UK*
- *Cosultant for Industrial and Residential Buildings*

Ass. Prof. Dr. Mohamed Ali Barakat
PH.D., M.Phil, M.Sc., B.Sc.
- *Military Technical College, Cairo, Egypt*
-
- *RMCS, Cranfield University, England, UK* *Cosultant for Industrial and Residential Buildings*

Hany El-Hady Ibrahim,
- *Ass. Lecture in Architecture Department, currently pursuing P.HD. degree program Military Technical College, Cairo, Egypt*

Eman Ali Ibrahim
- *currently pursuing M.Sc. degree program in Architecture Department, Port Said University, Port Said, Egypt.*