Journal of Multidisciplinary Engineering Science and Technology (JMEST)
ISSN: 2458-9403
Vol. 5 Issue 10, October - 2018

Numerical Analysis Of The Effect Of Wind Flow On Natural Ventilation For Native Housing Of Sistan Province Of Iran

Saide Doodi[1]
Department of mechanical engineering
Iran University of science and technology
Tehran, Iran
Saide.doodi70@gmail.com

Dr S.M.Hosseinalipour[2]
Department of mechanical engineering
Iran University of science and technology
Tehran, Iran
Alipour@iust.ac.ir

***Abstract*— Wind energy has become one of the most renewable energies resources in nowadays modern world . in addition to harnessing it's energy and converting it to electricity by employing wind turbines , wind power is also used for many rural natural ventilation specially in hot and hot-humid climate regions. With a total area of 181,000 km^2 and it's well known "120-days wind " with an average of 8 m/s ,the South-East region of IRAN has been an ideal location to utility wind power for local housing ventilation purpose from many years ago . computational fluid dynamic is a powerful tool for numerical simulation of flow patterns due to wind passing over and through rural buildings in order to evaluate the amount of natural ventilation can be achieved for various architectures. In this article the result of a comprehensive the numerical simulation are presented for evaluating the natural ventilation can be achieved due to local wind for the most common module of SISTAN's ancient housings. The validation of numerical result has been carried out using on site measurement which shows good agreement.**

Keywords—component; natural ventilation; wind engineering; computational fluid dynamic;

I. INTRODUCTION

Natural ventilation strategies based on wind has been used in design of buildings using opening on their both windward and leeward exposures since many years ago in different parts of IRAN. the pressure distribution across the building structure to wind blowing ,provide a natural ventilation inside the building .(fig1).

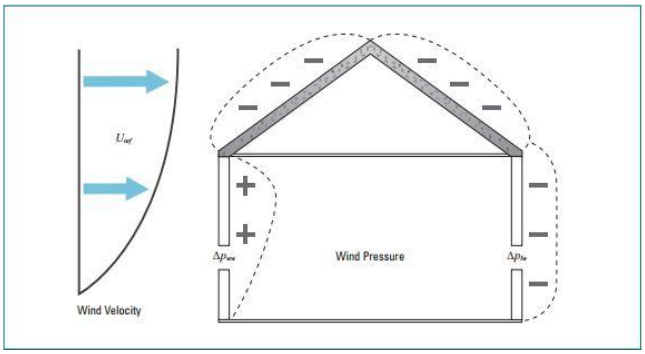


FIGURE I Pressure differential due to the wind [1]

The three main factors affecting the surface pressure are as the followings:

1- Wind speed relative to building
2- Wind direction
3- Building external shape [1]

The SISTAN province is located the south east of Iran .It covers an area of approximately 181,000 square kilometers (11% country area). Approximately half of 2.5 million populations live in rural with rural lifestyle [2].SISTAN's hot and arid climate is characterized by the strong wind ("120-day winds'') in summer which has been used since hundreds of years ago with aim of special architecture for utilization of natural ventilation in residential [3]. Due to severe climate change in this region during the last decades a vast immigration is accruing from SISTAN to other parts of country .

One key parameters which can alleviate this situation is to redesign the architecture of rural housing in order to achieve better natural ventilation in new condition Investigation and analysis on natural ventilation in enclosed spaces can be performed by several methods: full-scale measurements [4] reduced-scale measurements [5,6] , analytical methods[7] and Computational Fluid Dynamics (CFD) [8,9,10]. The Analytical approach seems to be inappropriate in complex geometries [11]. The fast growth in computational power during the last two decades CFD commercial software have been used in many investigations regarding indoor natural ventilation problems.[12]the following paragraph presents a brief review of some these attempts.

Miguel Mora-Pérez et al used the computational techniques for analysis the wind-building interactions for different building configuration regarding the natural ventilation [13]. Durrani et.al carried out numerical simulations for evaluating the performance of LES and Unsteady Reynolds-Averaged Navier-Stokes (URANS) methods for modeling multiple steady states in natural ventilation [14]. Gilani et.al performed a sensitivity analysis of stratified indoor environment in displacement ventilation [15].

Driss et.al carried out a combined experimental and numerical study on wind-induced natural ventilation in isolated building with patio [16]. Pulat and Ersan investigated the effects of inlet turbulence parameters on the air flow patterns inside a ventilated room [17]. Tominaga and Blocken carried out a comprehensive wind Tunnel experiments regarding the cross-ventilation flow of a generic single-zone building in order to complete a validation data base for CFD simulation [18].

This paper presents the results of a numerical analysis for simulation of wind flow around and inside a native housing of SISTAN province . in order to investigate the air flow pattern and natural ventilation for different configuration due to application of some opening in the base geometry such as : SOORAK ,KOLAK and DARICHE MOHABAK.

II. EXPERIMENTAL DETAIL

Climate has always been a key parameter in designing rural building in SISTAN province of Iran . the many year experiences of native architectures have provided enough knowledge for using different opening and configurations in benefit the local winds for natural order ventilations in their housing**.** The basic configuration and its different opening are introduced in the following paragraphs and figure 2.

KOLAK: it is a quarter-circle sector located at the roof for directing the air towards the interior space of the room.

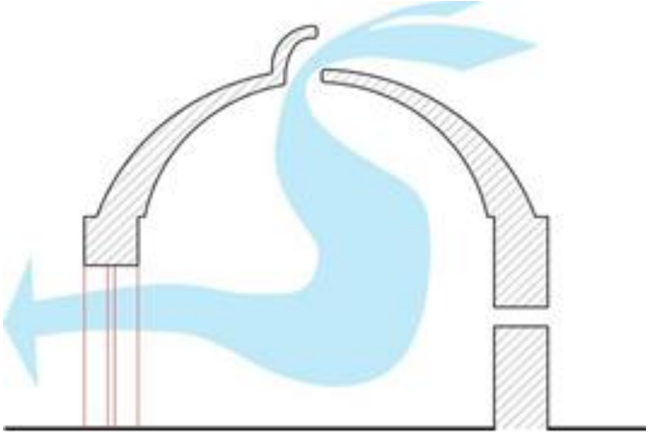


FIGURE II The cross-section of native housing

SOORAK: it is a rectangular opening with a cross section of 20*15 cm which is located on the positive pressure side wall 2 meters above the ground level.

DARICHE MOSHABAK: four squared opening with a cross section of 15*15 cm located on the positive pressure side wall 50cm above the ground level.

FIGURE III native house

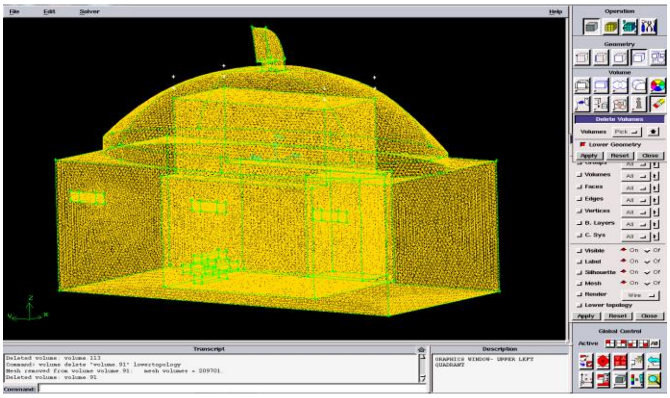
Table 1 summarizes the different investigated configurations which have been studied in this research.

TABLE 1 different form of case study

plans tools	Plan1	Plan2	Plan3	Plan4	Plan5
KOLAK	✓	✓	✓	✓	✗
SOORAK	✗	✓	✗	✗	✗
DARICHE MOSHABAK	✓	✓	✗	✓	✓
WINDOWS	✗	✗	✗	✓	✓

III. COMPUTATIONAL DOMAIN AND BOUNDARY CONDITION

Unstructured and structured-map meshes were used for external and internal spaces of the building respectively. The computational domain and meshes is presented in figures 4 and 5 .



```
Deleted volume. volume 113
Command: volume delete "volume 91" lowertopology
Mesh removed from volume volume 91:   mesh volumes = 209701
Deleted volume. volume 91
```

```
GRAPHICS WINDOW- UPPER LEFT
QUADRANT
```

FIGURE IV geometry

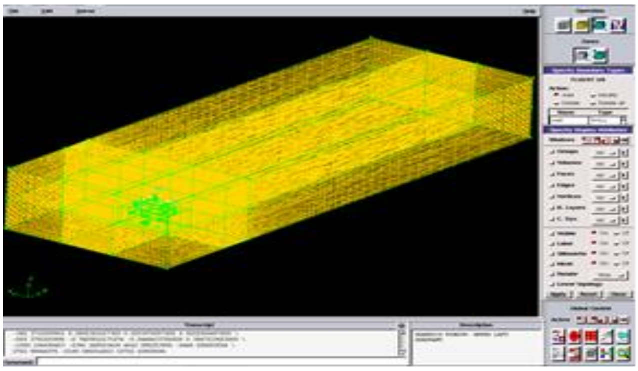


FIGURE V computational domain

The following boundary conditions were used in the simulations:

- inlet velocity at inlet boundary
- pressure outlet at the outlet boundary
- zero gradient at top and side boundaries
- no-slip at solid walls.

simulation: the commercial Fluent 16 CFD package was used in this study. the simulation were carried out for steady state situations . the well- known RNG k-ε turbulence model was activated for all case studies. The readers are addressed to the Fluent user guide for detailed information regarding the Reynold average and RNG k-ε equations**.**

IV. RESULT

5 different plans which presented in table 1 simulated and numerically analyzed . In pressure contour concluded from plan number 1, positive and negative pressure areas respectively at the *Wind ward and leeward to the wind presented at figure No 6***.**

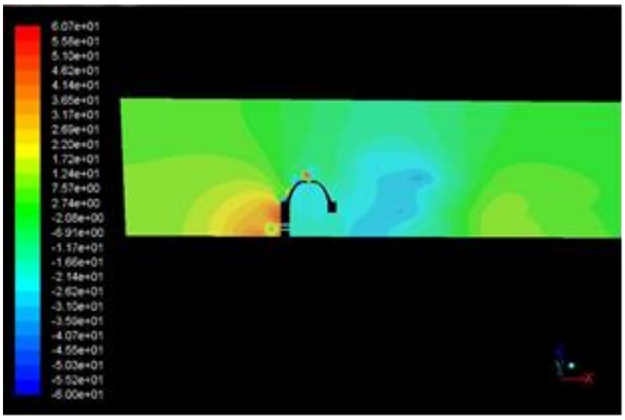


FIGURE VI pressure contour for plan1

Also, in plan number 1, velocity starts from 8 m/s at the inlet and by going through the wind inlet gates, and entering to the building inside space decreases up to 5 to 6 m/s. Maximum velocity create after wind hitting the roof of building which is approximately is 12 m/s (figure7).

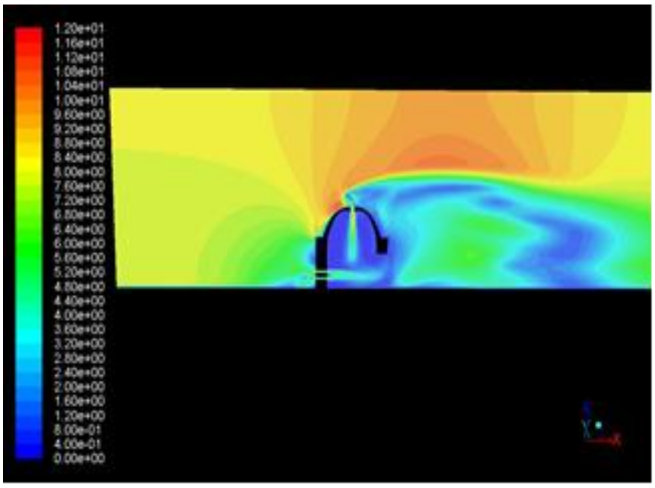


FIGURE VII velocity contour for plan 1

In plan number 2 the wind velocity at the inlet gate is 8m/s and by going to the building inside space it decrease and arrive to 4 to 5 m/s. By more spreading wind inside the building, velocity distribution rate will continue and in the other areas of roof and room will be equal to 1 to 2 m/s (figure 8) .

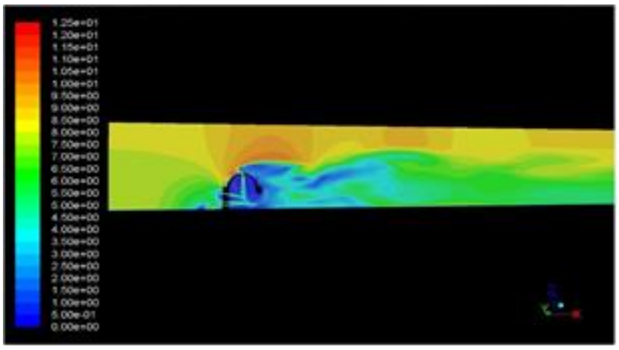


FIGURE VIII velocity contour for plan 2

Figure number 9 shows velocity distribution from top and north direction and presents created vortices around the building and after hitting wind to the building.

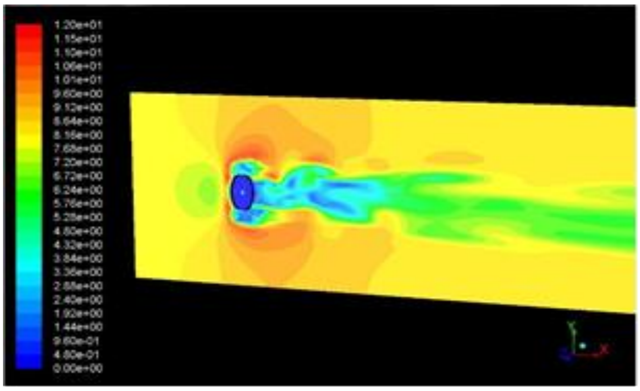


FIGURE IX velocity contour from top direction

At the contours of plan number 3 (figure10) which only includes KOLAK, it can be seen by deleting wind inlet gates on the walls, air recirculation at the inside space of building faced with problem and in a large part of interior space velocity is zero which Indicates lack of air changes in many sectors, and also presents reduced efficiency of natural ventilation.

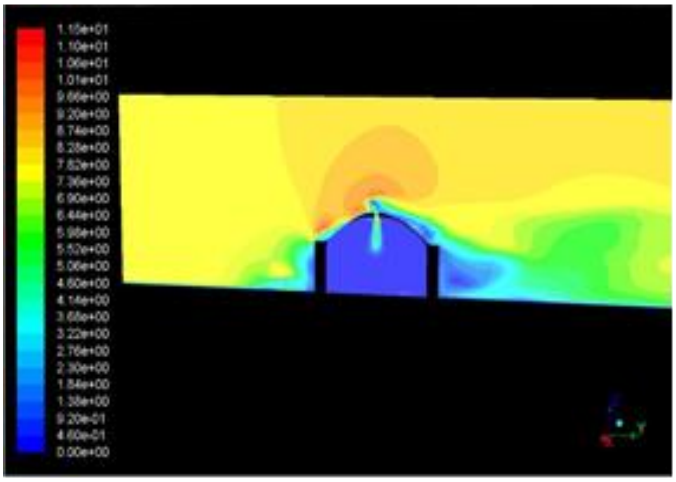


FIGURE X velocity contour for plan3

plan number 4 which additionally a window place on the leeward side of wind, the velocity contour will be shown as figure11.

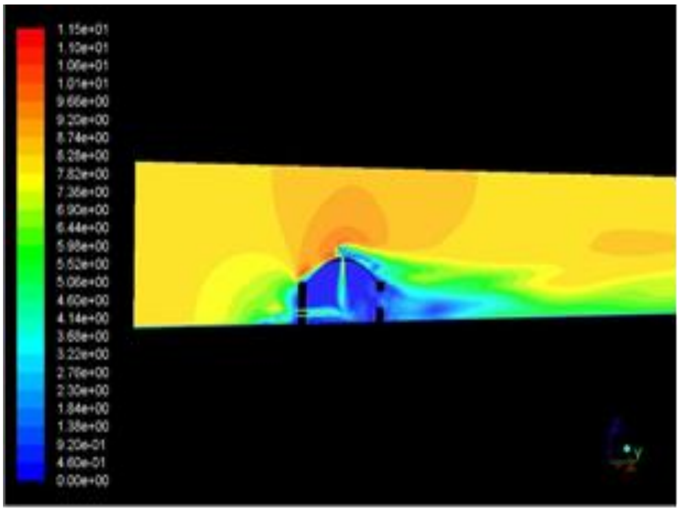


FIGURE XI velocity contour for paln 4

Analyzing results of last plan which has no air inlet on the building , shows that air ventilation done only by the holes on the wall. Wind velocity in many parts of dome inside is equal to zero which is an imperfect measure of ventilation (figure 12).

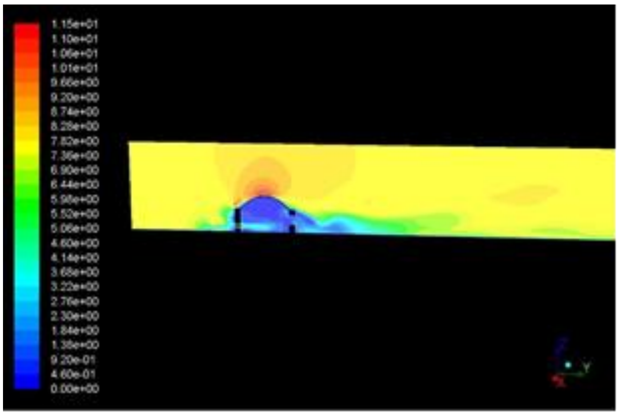


FIGURE XII velocity contour of plan5

In all presented contours, vortices behind building can cause accumulation of sands behind of windows and walls of building. The validation of numerical result has been carried out using on site measurement which shows good agreement in figure (13,14).

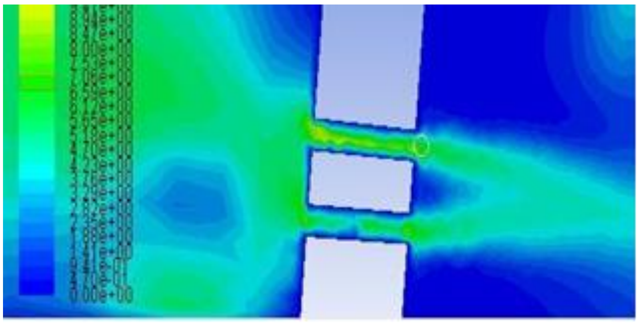


FIGURE XIII velocity distribution

FIGURE XIV experimental varfication

CONCLUSION

1- OPTIMIZATION AND UTILIZATION OF ANCIENT CONFIGURATION OF BUILDING IN UNDER INVESTIGATION REGION ,CAN BE A GOOD TEMPLATE FOR AIR CONDITION AND USING WIND AS RENEWABLE ENERGY SOURCE.

2- DUE TO ARID CLIMATE OF THE REGION AND BY USING EVAPORATING COOLING AND INCREASING RELATIVE HUMIDITY AT INLET GATES OF AIR TO THE BUILDING, APPROPRIATE AIR TEMPERATURE CAN BE REACHABLE.

3- FOR AVOIDING VORTICES AND ACCUMULATION OF SAND BEHIND OF BUILDING IN RESULT OF THEM, REDESIGNING THE BUILDING SHAPE AND REPLACING SHARP EDGES OF BUILDING WITH SOFT FORMS, CAN CHANGE IN WIND FLOW REGIME WILL BE USEFUL FOR SOLVING THE PROBLEM.

4- PRESENTED RESULTS OF NUMERICAL ANALYSIS SHOWS GOOD AGREEMENT WITH SITE MEASUREMENT

REFERENCES

[1] Engineering Guide-Natural Ventilation k-4M.
[2] M.Sharifikia, Environmental challenges and drought hazard assessment of Hamoun Desert Lake in Sistan region, Iran, based on the time series of satellite imagery Springer Science+Business Media B.V. (2012)
[3] Mohammad ali sargazi , sustainable development and its concepts in rural residential spaces of SISTAN,IRAN,(2014)
[4] M.N.A. Saïd, R.A. MacDonald, G.C. Durrant, Measurement of thermal stratification in large single-cell buildings, Energy Build. 24 (1996)
[5] P. Cooper, P.F. Linden, Natural ventilation of an enclosure containing two buoyancy sources, J. Fluid Mech. 311 (1996) .
[6] R. Tovar, P.F. Linden, L.P. Thomas, Hybrid ventilation in two interconnected rooms with a buoyancy source, Sol. Energy 81 (2007)
[7] P.V. Nielsen, A. Restivo, J.H. Whitelaw, Buoyancy-affected flows in ventilated rooms, Numer. Heat. Transf. Part A 2 (1979).
[8] A. Stamou, I. Katsiris, Verification of a CFD model for indoor airflow and heattransfer, Build. Environ. 41 (2006).
[9] H. Montazeri, B. Blocken, W.D. Janssen, T. van Hooff, CFD evaluation of new second-skin facade concept for wind comfort on building balconies: Case study for the Park Tower in Antwerp(2013).
[10] R. Ramponi, B. Blocken, CFD simulation of cross-ventilation for a generic isolated building: impact of computational parameters, Build. Environ. 53 (2012).
[11] Q. Chen, Ventilation performance prediction for buildings: a method overview and recent applications, Build. Environ. 44 (2009).
[12] P.V. Nielsen, Computational fluid dynamics and room air movement, Indoor Air 14 (Suppl. 7) (2004) .
[13] Miguel Mora-Pérez, Ignacio Guillén-Guillamón, P. Amparo López-Jiménez , Computational analysis of wind interactions for comparing different buildings sites in terms of natural ventilation (2015).
[14] Faisal Durrani , Malcolm J. Cook , James J. McGuirk , Evaluation of LES and RANS CFD modelling of multiple steady states in natural ventilation (2015).
[15] Sara Gilani , Hamid Montazeri , Bert Blocken ,CFD simulation of stratified indoor environment in displacement ventilation: Validation and sensitivity analysis (2016).
[16] Slah driss,Zied Driss,Imen Kallel kammoun , numerical simulation and wind tunnel experiments on wind –induced natural ventilation in isolated building with patio (2015).
[17] Erhan Pulat , Hifzi Arda Ersan , numerical simulation of turbulent airflow in a ventilated room:inlet turbulence parameters and solution multiplicity (2015).
[18] Yoshihide Tominaga, Bert Blocken , wind tunnel experiments on cross-ventilation flow of a generic building with contaminant dispersion in unsheltered conditions (2015).