International Journal of Scientific Engineering and Technology (ISSN : 2277-1581)
Volume No.4 Issue No.3, pp : 126-130 01 March. 2015

Performance Evaluation of Natural Ventilation Devices to Improve Thermal Comfort of a Computer Lab of University Building Using CFD as a Tool

Dipesh Kumar, Abhishek Jatav
Mechanical Department, Suresh Gyan Vihar University, Jaipur

Abstract: Air circulation study is most important part for any building; so that engineers can optimize thermal comfort for buildings. A computer lab located in university building of Jaipur city is used in this study. Main aim of this study is to optimize computer lab for two types of conditions, first is only internal fans are in operational condition and second condition is effect of room window opening and closing. Thermal comfort parameters are used for analysis. Model equations are also generated in this steady using advanced D.O.E. (design of experiment) technique.
Computational fluid dynamic (CFD) studies were carried out by using the ANSYS FLUENT 14.5 to find the effects of internal ventilation, friction loss and thermal performance factor characteristics in a computer lab at some assumptions. Simulation was performed with inlet vent and outlet vent based natural ventilation conditions with radiation modeling conditions applicable for computer lab. All experiments are designed using taguchi methods and orthogonal array methods.

Keywords: CFD, Dimensionless numbers, Turbulence models, fan boundary conditions, radiation modeling

INTRODUCTION

Hajdukiewicz et. Al. [01] presented that solar radiation are important for outdoors conditions of any building. In their study it was concluded that vertical component of inlet velocity highly influenced the indoor air temperature. So it can be said that natural ventilation is very helpful for the healthy life of occupants live in buildings.

Pascal Stabat et al [02]studied that single-sided ventilation is less efficient than cross-ventilation, it is more common practice since it fits in easily with most office building layouts. The driving forces of single-sided ventilation are due to pressure differences across the opening, resulting from wind and stack effects. Main parameters focused in this study is 1.formation of recirculation zone take place when the wind enter leeward side of building

2leeward opening at enter and outing of air, turbulence increases behind the building.

3. Help of CFD, the correlation are developed. It was concluded that when wind speed increases;-1.a positive one obtained by mixing of layer. 2.negatve effect are caused due to the reducing of ventilation rate at leeward opening.

Anastasia D. Stavridou and Panagiotis E. Prinos [03] natural ventilation promotes not only protection and restoration of indoor air, but also sustainability and energy saving. This research reveals a fascinating branch of fluid mechanics, dealing with two basic forms of ventilation: mixing and displacement ventilation. Main parameters focused in this study are 1. Natural ventilation bye buoyancy assisted through wind is fined with computational and laboratory simulation. 2. Using ethanol water is option then using salt water 3. Ethanol solution help in find difference density of air at inlet and outlet. Main outcome of this research is 1.position of outer opening plays a crucial role in experiment due to its empting of space. 2cross ventilation is found with the help of buoyancy effect.

Mathieu Barbason and Sigrid Reiter [04] try to develop a coupling between BES approach and CFD simulation. Main parameters focused in this study are insulation and lack of son protecting has widely made this use of phenomena an in the northern Europe.

1 space and time coupling is used

2. Qualitatively and quantitatively both used coupled for better result in numerical state

CFD MODELING

Systematic diagram of the CFD domain setup is shown in the fig. 1. Various fan boundary condition can use in study problem. In figure 1 only one fan boundary condition is used.

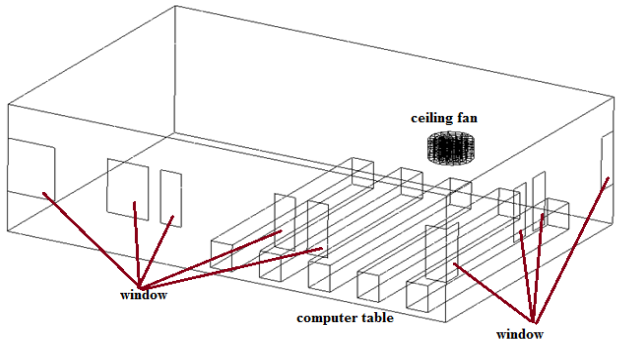

Fig. 1 Systematic diagram of Simple computer lab

3.1.1 PROBLEM STATEMENT

In this study a numerical analysis will be conducted to study heat transfer analysis of computer lab having natural ventilation conditions like only fan circulation, desert fan circulation and radiant cooling circulation. For final selection of the best design various cases are studied and selection was purely dependent on thermal comfort and temperature difference obtained. A parametric study was also done to find effect when change in flow parameters is done on system. All experiments are completed using DOE methodology named taguchi methods.

3.1.2 Methodology

In CFD simulation study, first of all geometry of the computer lab was created using ANSYS design modeler (a preprocessor software of FLUENT). After geometry formation next step is to discretize the geometrical domain, which was also completed in ANSYS ICEM CFD. Next step in ANSYS WB is to declare

continuum type and boundary types for the surfaces generated for computer lab. Last step is to generate a mesh file which is used by FLUENT14.5 (Solver Tool).

After importing mesh file in FLUENT, dimensional units scaling (if required) for CFD domain are changed. In FLUENT desired turbulence models was selected on the basis of literature review. After selection of required turbulence model boundary conditions are specified and give initial values. Fluent has capability to store values from any physical parameters for any specified point in the domain. Four points were created to store the values of physical parameters such as temperature, velocity and turbulent intensity.

FLUENT is now ready to simulate flow problem with some residual criteria. Simulation was completed for steady mode. Post processing and parametric study was completed in ANSYS WB module.

3.2 DESCRIPTION OF CFD DOMAIN

In present study CFD simulation is carried out for three conditions of natural ventilation design of computer lab as shown in figure 2. Various simulations are run to verify the best design for natural ventilation of computer lab. In present study computer lab construction is assumed to be made by building material with insulation at proper places of room body. Thickness of glass windows are assumed 5 mm. Thickness of building wall was assumed 9 inch excluding heater wall at exterior surface of system.

Present study is focused on CFD simulation results analysis and valves are assumed to be used when it goes on production stage.

Design 1 Single fan (on) having various window open conditions, all experiments are shown in next Section DOE and research methodology. (see figure 2 (a))

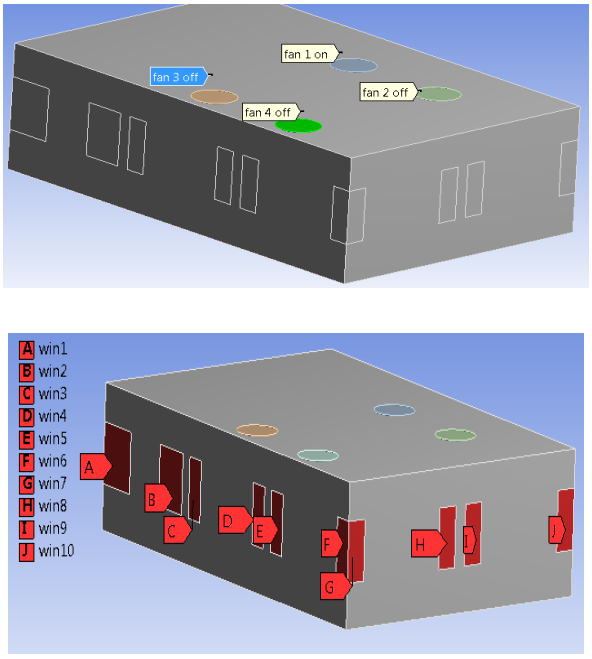

Fig. 2 (a) design 1

Design 2 two fans (on) having various window open conditions, all experiments are shown in next Section DOE and research methodology. (See figure 2 (b))

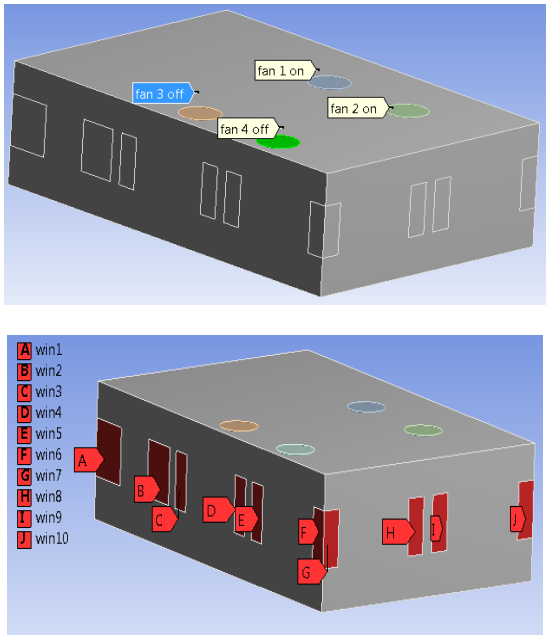

Fig. 2 (b) design 2

Design 3 three fans (on) having various window open conditions, all experiments are shown in next Section DOE and research methodology. (See figure 2 (c))

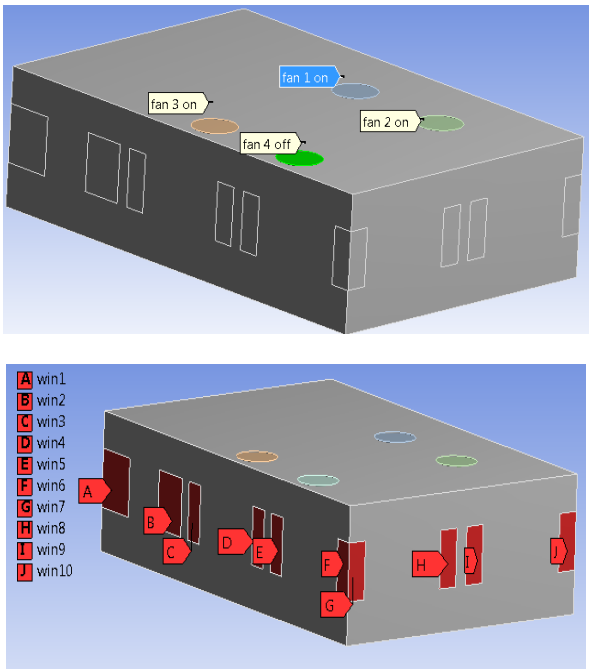

Fig. 2 (c) design 3

Design 4 four fans (on) having various window open conditions, all experiments are shown in next section DOE and research methodology. (See figure 2 (d))

International Journal of Scientific Engineering and Technology (ISSN : 2277-1581)
Volume No.4 Issue No.3, pp : 126-130 01 March. 2015

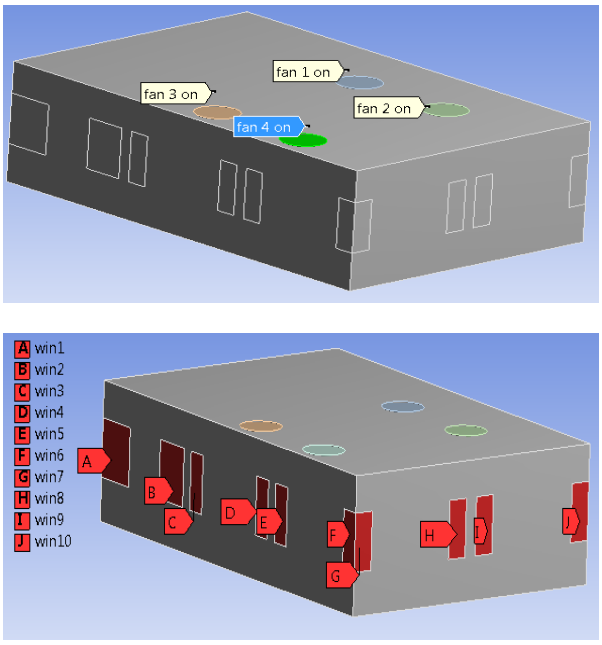

Fig. 2 (d) design 4

DOE AND RESEARCH METHODOLOGY

In present study one computer lab is used for numerical simulation having fan circulation based natural ventilation analysis, DOE was applied to both combinations to find optimum solution for thermal comfort analysis. DOE is very vast subject and it was very difficult to choose proper technique for experiments design. From literature review it was found that DOE based on Taguchi methods was used by various researchers and that's why this technique was also used in this study and was discussed in detail in next section.

FACTORS AND LEVELS

Design of DOE table was only possible by selection of proper factors and their levels. In this study four factors were selected with three levels for each case and were shown in table 4.1 to table 1.

Table 1 Summary table of Factors and Levels for case_1 (four fan conditions)

Levels	press_Jump (Pascal)	Win_in	Fan Number
1	1.5	2	2
2	2.0	3	3
3	2.5	4	4

Orthogonal Array L9 is made for CFD simulation and presented in table 2.

Table 2 L9 orthogonal Array for Case_1

Sr. No.	press_Jump	Win_in	Fan Number
1	1.5	2	2
2	1.5	3	3
3	1.5	4	4
4	2	2	3
5	2	3	4
6	2	4	2
7	2.5	2	4
8	2.5	3	2
9	2.5	4	3

Result and Discussion

Fan circulation in computer lab is numerically solved for improvement of natural ventilation which is shown in figure 1. ANSYS FLUENT 14.5 CFD software package is used for simulation purpose. All experiments were designed according to DOE technique (Taguchi orthogonal array table), which are presented in table 1 and table 2. Main outcomes focused in this study are following:

ANOVA Analysis

Signal to noise ratios analysis

Model equations generation

In this study response is air temperature at 3 feet height of domain developed during CFD simulation is selected, and all results according to L9 array experiments is presented in table 3.

Table 3 L9 orthogonal Array for Case_1

Sr. No.	press_Jump (Pascal)	Win_in	Fan Number	Air Velocity
1	1.5	2	2	0.334
2	1.5	3	3	0.425
3	1.5	4	4	0.543
4	2	2	3	0.431
5	2	3	4	0.601
6	2	4	2	0.379
7	2.5	2	4	0.642
8	2.5	3	2	0.416
9	2.5	4	3	0.534

Signal to Noise Ratio

Signal to noise ratio is simple technique to predict the effect of changing of factors according to their levels to find effect on thermal efficiency. In this study "larger is better" option is adopted as quality indicator for S/N ratio and means ratio. The response tables for S/N ratio and mean are presented in table 4 and table 5.

Table 4 response table for signal to noise ratio

Response Table for Signal to Noise Ratios
Larger is better

Level	Win Open	Loss CoE	Fan
1	-7.420	-6.895	-8.523
2	-6.720	-6.491	-6.731
3	-5.639	-6.393	-4.525
Delta	1.782	0.501	3.998
Rank	2	3	1

Table 5 response table for mean ratio

Response Table for Means

Level	Win Open	Loss CoE	Fan
1	0.4340	0.4690	0.3763
2	0.4703	0.4807	0.4633
3	0.5307	0.4853	0.5953
Delta	0.0967	0.0163	0.2190
Rank	2	3	1

International Journal of Scientific Engineering and Technology (ISSN : 2277-1581)
Volume No.4 Issue No.3, pp : 126-130 01 March. 2015

Both tables show factors importance ranking and it is clear that air velocity is most important factor, which can increase velocity magnitude at outer boundary of window.

Best and worst cases from experiment factors and their levels are also presented in this study and are calculated from figure 3 and figure 4.

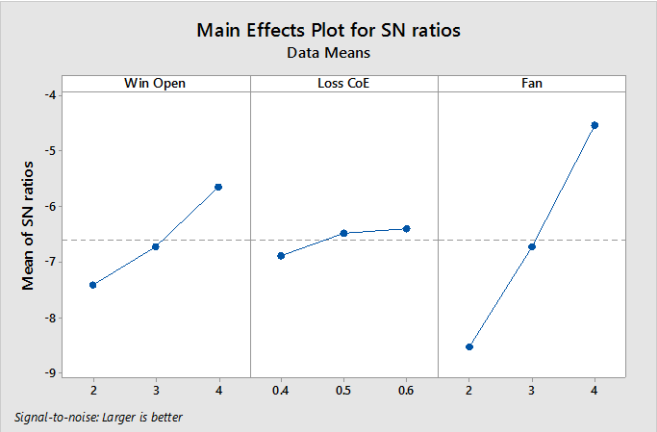

Fig. 3 Data means for larger is better for S/N ratios

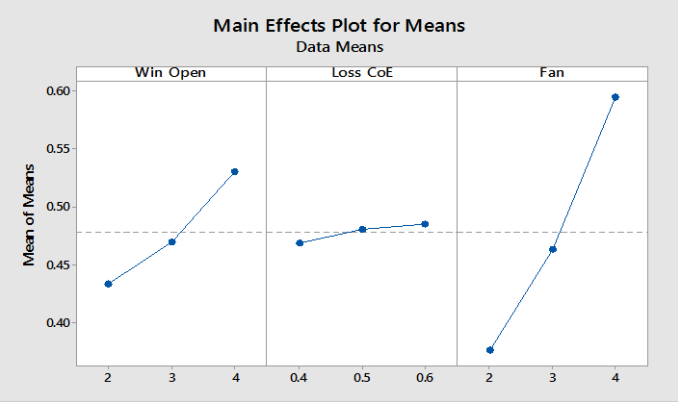

Fig. 4 Data means for mean ratios

Best case: A3 B2 C3

Worse Case: A1 B1 C1

ANOVA Analysis

The analysis of variance is calculated for this study and results are shown in table 6 respectively. In ANOVA analysis F-Test is conduct to compare a model variance with a residual variance. F value was calculated from a model mean square divided by residual mean square value. If f value was approaching to one means both variances were same, according F value highest was best to find critical input parameter.

Table 6 Analysis of Variance for velocity at 3 feet height plane

Analysis of Variance

Source	DF	Adj SS	Adj MS	F-Value	P-Value
Regression	3	0.086358	0.028786	59.34	0.000
Win Open	1	0.014017	0.014017	28.89	0.003
Loss CoE	1	0.000400	0.000400	0.82	0.405
Fan	1	0.071941	0.071941	148.29	0.000
Error	5	0.002426	0.000485		
Total	8	0.088784			

Table 6 list out one important result that F value for regression models are very high, than one and P value is very less (approx 0.0000) suggested that all cases are significant.

From literature review various researchers found that if p value was very small (less than 0.05) then the terms in the regression model have a significant effect to the responses.

Model equations for temperature are presented in table 7 ANOVA analysis with model equations.

Table 8 Model summary for ANOVA analysis

Model Summary

S	R-sq	R-sq(adj)	R-sq(pred)
0.0220257	97.27%	95.63%	91.12%

Coefficients

Term	Coef	SE Coef	T-Value	P-Value	VIF
Constant	-0.0360	0.0594	-0.61	0.571	
Win Open	0.04833	0.00899	5.38	0.003	1.00
Loss CoE	0.0817	0.0899	0.91	0.405	1.00
Fan	0.10950	0.00899	12.18	0.000	1.00

Model Equation

Air temp = -0.0360 + 0.04833 Win Open + 0.0817 Loss CoE + 0.10950 Fan

The adequacy of regression models shall be inspected to confirm that the all models have extracted all relevant information from all simulated cases. If regression equations results were adequate than the distribution of residuals should be normal distribution. For normality test, the Hypotheses are listed below

A. Null Hypothesis: the residual data should follow normal distribution

B. Alternative Hypothesis: the residual data does not follow a normal distribution Normal probability figures for all responses were shown in figure 5.

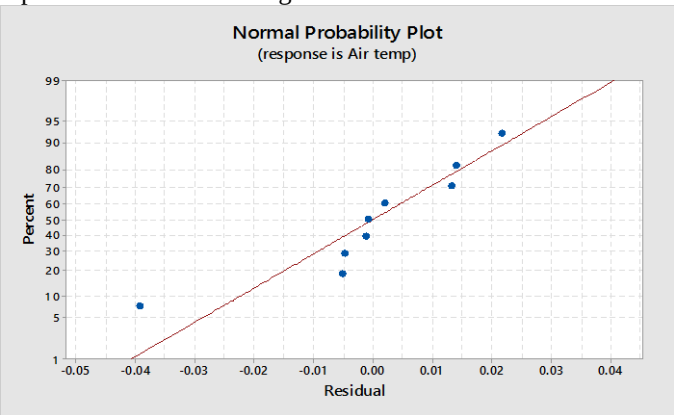

Fig. 5 Normal probability for velocity at 3 feet plane from floor

CONCLUSION

The aim of this study is to try to balance among response results and CFD simulation results for Natural ventilation in computer lab. This study utilizes L9 orthogonal array for CFD based data analysis. In this study Analysis of variance (ANOVA), and linear regression analysis is main key techniques to show response and factor relations strongly with each other. Main results are summarized as follows:

1. Best parameter combination for CFD results are following respectively

   Case_1 best Set (S/N Ratio): **A3 B2 C3**
   Case_1 best Set (Mean ratio): **A3 B3 C3**

Table 9 Summary of best cases for S/N ratio and Mean

Design	press_Jump (Pascal)	Win_in	Fan Number
S/N ratio	2.5	3	4
mean	2.5	4	4

2. Model equations for stress response was predict accurately with Minitab software and show 95% good prediction for responses and can be used by any manufacturer.

REFERENCES

i. *Hajdukiewicz Magdalena, Geron Marco and Keane Marcus M. Calibrated CFD simulation to evaluate thermal comfort in a highly-glazed naturally ventilated room [Journal] // Building and Environment. - 2013. - Vol. 70. - pp. 73-89.*

ii. Caciolo Marcello [et al.] Development of a new correlation for single-sided natural ventilation adapted to leeward conditions [Journal] // Energy and Buildings. - 2013. - Vol. 60. - pp. 372-382.

iii. Stavridou Anastasia D. and Prinos Panagiotis E. Natural ventilation of buildings due to buoyancy assisted by wind Investigating cross ventilation with computational and laboratory simulation [Journal] // Building and Environment. - 2013. - Vol. 66. - pp. 104-119.

iv. Barbason Mathieu and Reiter Sigrid Coupling building energy simulation and computational fluid dynamics: Application to a two-storey house in a temperate climate [Journal] // Building and Environment. - 2014. - Vol. 75. - pp. 30-39.