ISSN: 0976-3104
SUPPLEMENTARY ISSUE
Sorour et al.

ARTICLE OPEN ACCESS

TEMPERATURE DISTRIBUTION IN AN INDUSTRIAL HALL

Mahsa Golshan Sorour*, Saeid Niazi, Abdulhamid Ansari Nasab

Department of Mechanics Engineering, Bandar Abbas Branch, Hormozgan University, IRAN

ABSTRACT

Industrial hall in a large thermal power plant is of great importance for its operational efficacy, inspection, and maintenance. The temperature in the operational or production room is generally very high due to the large amount of steam in a thermal plant. Therefore, people working in those conditions face several difficulties. An efficient ventilation system to control ambient temperature as well as fresh air is therefore essential in such environment. Here, we distributed air flow in the room in order to improve temperature conditions. We calculated the temperature distribution of the turbine hall using fluid dynamics and heat transfer calculations were performed using Fluent software. The modeling was performed at room temperature in real dimensions. Because of symmetrical and identical conditions, a fourth hall dimensions (32 x 42 x 49 mm) is considered and k- ε turbulence model and thermal diffusion model at 34° C ambient air temperature was analyzed. Inspection of temperature distribution shows CFD methods is a good tool to analyze the situation and different scenarios for improving the situation. By examining the various models we also suggested a model to improve the situation.

Received on: 10th-March-2016
Revised on: 29th -May-2016
Accepted on: 02nd – June-2016
Published on: 5th–June-2016

KEY WORDS

Temperature distribution, FLUENT, CFD.

***Corresponding author: Email:** mahsa_golshan_s@yahoo.com

INTRODUCTION

Heat stress can lead to a series of problems such as blistering of the skin, dizziness, and anesthesia among others. Early symptoms of heat stress such are generally fatigue and drowsiness. The lack of stability and balance in decision-making can lead to serious accidents and without immediate treatment, the symptoms such as convulsions and anesthesia can expand critical condition. In Iran, the issue is severe as reported in recent studies [1-2]. Ventilation and thermal comfort providine favorable conditions to enterprises to maintain the rights of workers, reduce unintentional errors, increase focus, increase working hours, and avoid errors and high cost that cones from environmental hazards.

The aim of this study was to investigate the temperature in the salon industry using numerical modeling and software with ANSYS Fluent and Gambit [1]. Navier-Stokes equations and the energy equation in the hall, within a networked environment, and indoor temperature field around heating tools are determined. We considered horses as a heat source, and natural convection air was investigated Applying Matthews and Arndt's method (2003) to examine the ventilation in a stable horse [2].

We evaluated different models also. Natural ventilation model by Dutt et al in 2003 [3] on wind tunnel and Horak's model 2010 [4] on distribution of air flow and the temperature in buildings using computational fluid dynamics were considered. We used computational fluid analysis to evaluate the physical configuration of the intake and exhaust vents and temperature and flow patterns inside a hall.

MATERIALS AND METHODS

In order to investigate the process of heat transfer in the hall we used computational fluid dynamics. We solve the conservation equations using ANSYS FLUENT-15 modules. For Grid, we used Gambit Version 2.4.6 .

THE IIOAB JOURNAL
ISSN: 0976-3104

We used Governing equations of conservation of mass, momentum, and energy. Usually turbulent air flow created by the temperature gradient is in the hall. Therefore, from the Governing equations it can be averaged. In this approach, the entire range of computing and computational cells conservation equations of mass, momentum and energy are solved. We also included the Reynolds stresses are modeled in our study. Next we averaged the various methods to determine the final air flow and heat transfer models (k-ε and k-ω) that are having superior air flow and heat transfer with sufficient accuracy.

RESULTS AND DISCUSSION

Models

A comparison between different software and models, a simple mode was introduced in the hall geometry and numerical solution for constant temperature mode. The comparison between the results of numerical and measurement is performed to solve the model of turbulence and the heat flux output is obtained from various German levels. Comparison of results are shown in **Table-1**.

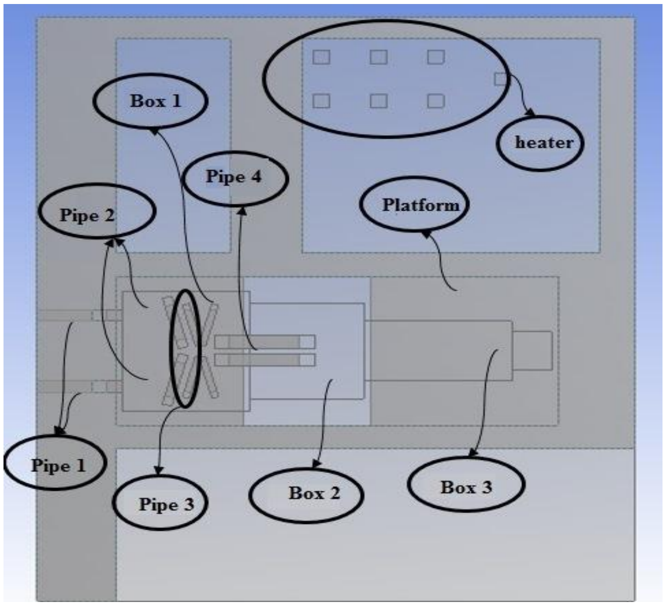


Fig: 1. View from the top model Gambit

Table: 1. Constant flux boundary condition element in the turbine hall

Difference (%)	Heat flux (w / m 2), Numerical heat flux	Heat flux (w / m 2), Numerical heat flux	Surface (m2)	Temperature (ºC)	Element
9	188	205	520	62	Heather
4	701	730	70	80	Pipe 1
6	371	395	55	68	Pipe 2
2	497	511	54	75	Pipe 3
8	355	385	102	70	Pipe 4
3	401	415	131	70	Box 1
4	137	143	140	55	Box 2
3	22	27	120	47	Box 3

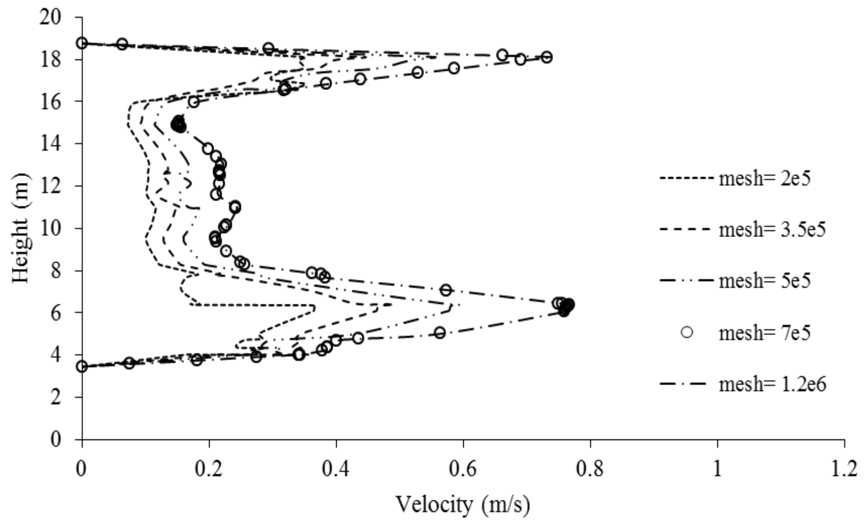
As seen in the table above, The differences between the results of manual calculations and analytical solution is 9% therefore, ither method is acceptable.

Sensitivity

To evaluate the sensitivity of grid size, we used k-ε turbulence model. In order to solve a number of different mesh was done. Numerical Solution of five modes: 200000, 350000, 500000, 700000 and 1200000 mesh was conducted and the results are compared. Boundary conditions and geometry of the solution was the same as the previous that was solved using a constant temperature levels. Hall geometry software and networked was drawn by Gambit and

ISSN: 0976-3104

the mesh of the type of organization was chosen. After defining the boundary conditions, the program initializes and implemented The convergence condition every time the program based on the criteria determined .The remaining error in the momentum balance and other quantities was less than Az4-10 and energy balance was less than 6-10. The results of Fluent suggests that even after 10,000 repetitions, the energy balance remains about 4-10 and are not changed. In a resolution CFD, when convergence is achieved, the remaining amount of velocity or temperature was less than the standard value and therefore, a convergence has been achieved. The results of the sensitivity of the network is shown in **Figure-2**. It is found that, when the mesh number is increased to more than 700,000 there is a significant change in the speed graph. It can be used to check the number of mesh. The temperature distribution in proposed model is given in **Figure-3**.



Fig; 2. Sensitivity analysis

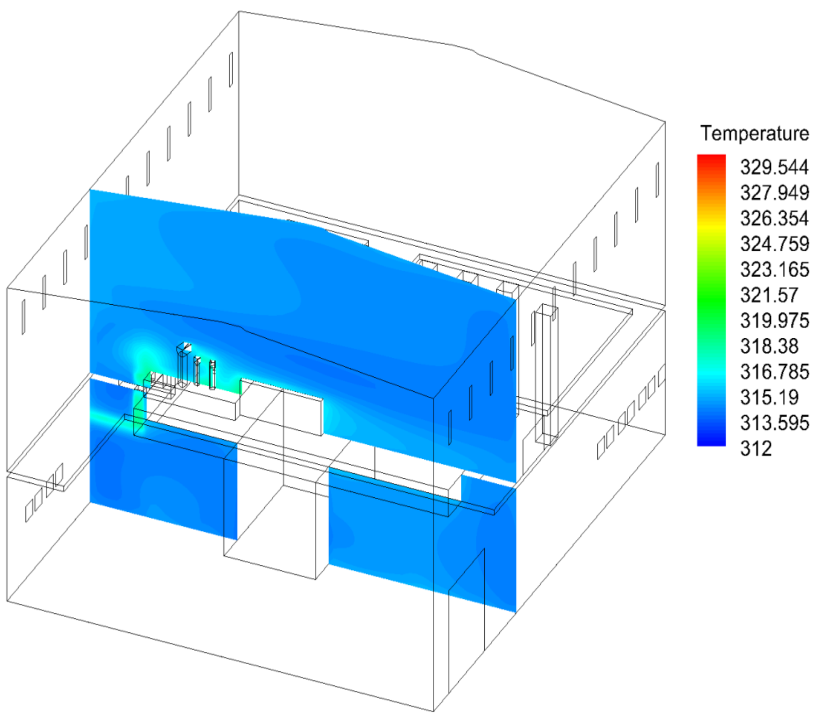


Fig:3. Temperature distribution, the proposed mode

THE IIOAB JOURNAL
ISSN: 0976-3104

CONCLUSION

Here we carried out a simplified numerical simulation of the hall. At temperature of 34°C we found the removal of heat from the outside air temperature. k-ε turbulence model found best in our stydy.

ACKNOWLEDGEMENT

We thank for Islamic Azad University (Sanandaj branch) for support this thesis.

CONFLICT OF INTEREST

There is no any form of conflict of interested.

FINANCIAL DISCLOSURE

No financial support was received for this work.

REFERENCES

[1] Mathews EH, D. Arndt. [2005] Validation of models to predict the thermal and ventilation performance of horse stables. Building and environment 38(2): 237-246.

[2] Dutt AJ, RJ.de Dear, Parthiphan Krishnan. [1992] Full scale and model investigation of natural ventilation and thermal comfort in a building. Journal of Wind Engineering and Industrial Aerodynamics 44(1): 2599-2609.

[3] Hurak Bradley. [2011] Computational Fluid Dynamics Analysis of Air Flow and Temperature Distribution in Buildings. http://hdl.handle.net/1811/48777

[4] Hoof P.[2012] Application of computational fluid dynamics in building design: aspects and trends.Indoor and Built Environment, 15(4): 305-313

[5] Wang Yi, et al. [2014] Influence of convection and radiation on the thermal environment in an industrial building with buoyancy-driven natural ventilation." Energy and Buildings,75: 394-401.