Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 146 (2016) 632 – 639

Procedia Engineering

www.elsevier.com/locate/procedia

8th International Cold Climate HVAC 2015 Conference, CCHVAC 2015

The indoor thermal environment simulation and testing validation of a power plant turbine room in extreme cold area

Guofu DU[a],Meiying Lei[b],Xi Chen[a],Congxin Huang[c],Yanjie Lv[a],Hui Kang[b],Ruixue Zhang[a],Baosheng Zhang[d] *

[a]*China Academy of Building Research, Beijing, 100013，China*
[b]*China Power Engineering Consulting Group Co., Ltd.,Beijing,100120 China*
[c]*Northwest Electric Power Design Institute of China Power Engineering Consulting Group,Xian, 710075，China*
[d]*North Electric Power Design Institute of China Power Engineering Consulting Group,Beijing,,100011，China*

Abstract

This paper conducts an analysis study on indoor thermal environment of a steam turbine room in power plant by CFD. Refer to a typical steam turbine room in an actual thermal power plant which has been conducted field test, the typical numerical simulation model is built including a reasonable indoor heat conditions, structural parameters and envelope architectural opening, flow boundary conditions. Indoor air temperature distribution and air velocity distribution of steam turbine room is obtained. Comparing the simulation results with the corresponding field measurement data on typical location show that two sets of results are very close. So accuracy and applicability of CFD simulations is proved. It is also proved that complete method for CFD simulations of the paper is appropriate for interior thermal environment study of typical steam turbine room and thus laid the foundation for the further studies of a large number of universal cases.

Peer-review under responsibility of the organizing committee of CCHVAC 2015

Keywords: power plant; steam turbine room; extreme cold region; temperature distribution; cold air infiltration

0. INTRODUCTION

Steam turbine room is one of the major buildings of thermal power plant and is usually considered as tall huge thermal building which owns extremely large interior space. Its indoor thermal comfort requirement is relatively less restrict comparing with conventional civil building. However, the pros and cons of the thermal environment have great significance on safe operation of electrical equipment.

* Corresponding author. Tel.: +0-086-13811151134.
E-mail address: 13811151134@163.com

Peer-review under responsibility of the organizing committee of CCHVAC 2015
doi:10.1016/j.proeng.2016.06.418

By testing the indoor thermal environment of typical power station turbine room, the main characteristics of turbine room thermal environment can be concluded as follows (Chen Y, 2014): (1) The tremendous amount of heat that gained from equipment causes a hot air rising and thus the large vertical temperature gradient which leads to the uneven layer distribution. Usually the temperature of ground floor is low in winter and may even add to freeze risk. Meanwhile the temperature of high floor is high and may reach above 30℃; (2) The thermal requirements are significantly different between workplace and non-workplace; (3) Comparing with the large amount of equipment heat, the temperature difference caused by solar radiation can be neglected; (4) The horizontal temperature distribution is not even. While the temperature near wall side is much lower than around the equipment in the central room.

Because of the complexity of turbine room thermal environment, long experimental period and the high cost, early analysis on indoor thermal environment mainly focus on ventilation patterns via calculating analysis (Bao, 2005; Huang, 2004; Li G, 1996; Wang X, 2000). Recently, optimization analysis using CFD are also adopted in the thermal environment analysis (Feng Z, 2010; Liu Y, 2009; Wang J, 2005; Yang Q, 2011; Zhang Q,2006). Although particularity of thermal environment has been pointed out and some measures to improve the main building thermal environment have been proposed, limitations still exist: (1) the study on temperature stratification is still qualitative and lack of quantitative data support; besides, there is no study about the difference of thermal environment between different floor which is important; (2) As the main research approach, the accuracy of CFD simulation model directly affects the reliability of the calculating results. The models that proposed in the literature lack verification and most of the models are referring to ordinary civil research method; (3)In terms of research content, above studies only focus on specific power plant and thus lack of universality. This paper establish an indoor thermal analysis model by successfully verifying simulation data and field measured data for typical turbine room in extreme cold regions . This can be a foundation for .studies of a large number of universal cases.

1. THE TURBINE ROOM SIMULATION MODEL

1.1. Simulation software

This paper chooses STREAM as the CFD simulation tool. STREAM is structural-grid-based heat transfer analysis software. Ever since released in 1984, STREAM has become a functional, fast calculating, and simply operating software. Now STREAM has been widely used in various industries which play a key role in various products developing process. STREAM has many advantages and is suitable for this project simulation and the main characters of STREAM can be concluded as: The model file can be established by importing from other software, thus significantly reducing the modeling time; STREAM own component model library and can help improve the design efficiency; STREAM can simulate various phenomena, such as diffusion, multi-phase flow, chemical reaction, particle tracing, and solidification, and so on; STREAM have the functions of multi-Block and automatic meshing; orthogonal matrix solver help improve the calculating speed and computational stability; excellent visual capabilities.

1.2. The physical model

The establishment of the CFD physical model is based on the typical steam power plant building which has been conducted field test research team. The details of test process and results of turbine room can be found in Ref (Chen Y, 2014). According to the actual construction scale, the opening mode of doors and windows, and the ventilation openings on the floor settings, the basic physical model is set up, as shown in Figure 1-Figure 5. The setting of interior heating sources and pipes are simplified.

1.3. Boundary conditions setting

The purpose of this study is to validate the CFD model results by field testing results, so boundary conditions settings comply with field measurement in maximum extend.

1.3.1 Outdoor temperature

The outdoor temperature in the simulation is an average temperature during the whole testing procedure, exact values can be found in Ref (Chen Y, 2014).

1.3.2 Flow boundary condition settings for openings

Since there are many doors and windows in a turbine room, infiltration in winter are the main parameters that affect the indoor thermal environments. The flow boundary conditions of all opening are set as constant flow boundary conditions and the data are acquired from Ref (Chen Y, 2014). During the winter test, the openings are all closed to prevent equipment frozen that caused by infiltration and the outdoor are blocked using hang-up curtain. The inlet louvers and sash windows are also closed to prevent air intake. But with strong indoor thermal pressure, infiltration still existed, and thus affected indoor thermal environment. According to the testing data, the flow rate of the infiltration is shown in Table 1.

1.3.3 Interior heating source

The intensity and the distribution of the interior heat source can significantly affect the indoor thermal environment. Ref (Chen Y, 2014) calculates the intensity of the interior heat sour according to the testing data with the heat balance method and the results will be used in this paper. Zone scheme method is used to allocate the total heat source to different zones according to distribution of equipment and pipes. Firstly, heat source is set up individually for the three stories of turbine room. Secondly, as the certain amount of heaters and motors are placed between BC axis and the heat intensity shaft and horizontal intensity distribution between BC axis and AB axis is significant different, the intensity of the two places are set separately. For turbine room, the heat source of each part can be allocated according to zone area. The heat source of each part is shown in the figure 6.

1.3.4 Heat transfer coefficients of building envelope

Envelope heat transfer boundary conditions are set directly by constant heat transfer coefficient. The heat transfer coefficient of the profiled steel sheet which is the main building envelope are from the thermal laboratory testing. Other parts of the heat transfer coefficient refer to the civil building thermal specifications. The specific heat transfer coefficient are shown in Table 2

1.3.5 Other

In addition to the settings that mentioned above, there are also some other settings used in the simulation.

1) The porous medium model boundary condition

Ventilation louvers are installed at the openings that located in the bottom layer and intermediate layer of the turbine room. Grilles are installed at ventilation openings between different floors. Air flows through above openings may accompany pressure loss. Therefore, these openings should be considered as porous medium. Based on the relationship between flow coefficient and resistance loss, and the resistance loss coefficient are set to reflect the different characteristics of these parts.

2) Virtual heat source and pressure loss model

As described above, it is hard to model all the equipment and pipes in details and corresponding heat can't be shown in every specific equipment, or pipe. All the heat source is arranged in a fixed virtual area, with a uniform heat intensity. Meanwhile, taking the flow resistance caused by piping into account; the virtual flow resistance coefficient is set in a virtual heat source. Based on the above two points, the numerical model is closer to the actual situation of the internal thermal environment of power plants building.

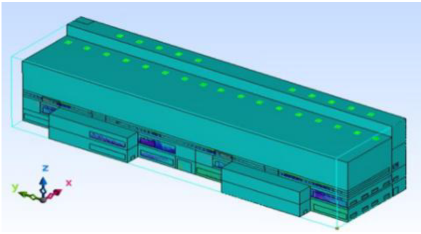
Fig. 1. The outline of the turbine room

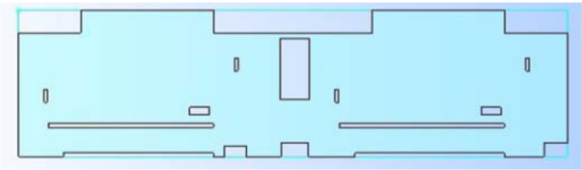
Fig. 2. Ventilation holes setting at the height of 6.85m

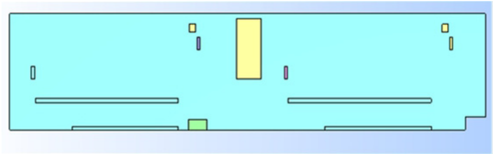
Fig. 3. Ventilation holes setting at the height of 13.70m

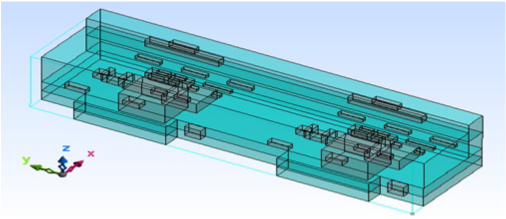

Fig. 4. The equipment settings inside the turbine room

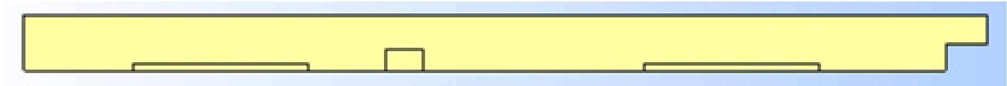
Fig. 5 Ventilation holes setting at the height of 26m

Table 1 The infiltration flow rate

	Location	Infiltration flow rate(m^3/h)
13.7m floor	North	140556
	East	36311
	West	18508
6.85m floor	West	18000
	East	3402
	North	61793
0.00m floor	North gate	30000
	West door	15000
	East door	12000
	North side door	3000
	North	32529
	East	19217
	West	24274
26.00m floor	South	3046

Table 2 The specific heat transfer coefficient

Location	Structure	Heat transfer coefficient	Notation
Window	5+9A+5 Plastic hollow window	2.8	empirical value
Concrete wall	10 ceramic + 15 cement mortar +370 non-clay porous brick +15 cement mortar	1.2	theoretical value
Profiled sheet	100mm glass wool insulating profile sheet	0.62	testing value
Roof	20 cement mortar + waterproof +20 1:3 cement mortar + 80XPS board + 30LC lightweight aggregate concrete	0.31	theoretical value

2. THE COMPARATIVE ANALYSIS OF TURBINE ROOM THERMAL ENVIRONMENT IN WINTER BETWEEN TEST AND CFD RESULTS

2.1. CFD simulation results in winter

After 98 iterations, the numerical simulation convergence is achieved. Fig. 7 -Fig.8 show the cross section temperature distribution and air velocity distribution of the steam turbine room. It can be seen from Figure 7 and Figure 8 that the temperature and velocity distribution can reflect the indoor thermal environment of the turbine room. It can be seen that in the area near the roof, high-temperature region is observed. In the positions that is close to ground outer wall, because of the infiltration, the temperature is low. The numerical simulation results effectively show the thermal stratification.

Figure 9 ~ 11 shows the temperature distribution of occupied zone (1.5 ~ 2m) of different floor in turbine room. From Figure 9 ~ 11 we can know that, the numerical simulation results effectively show uneven distribution at the same level in steam engine room and basically reflect the characteristics of thermal environment of the steam turbine room in winter.

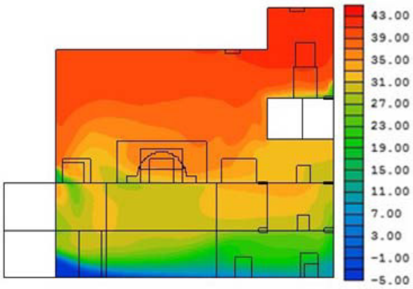

Fig. 7. Temperature distribution in the chambers(Y=83.7) of steam engine room

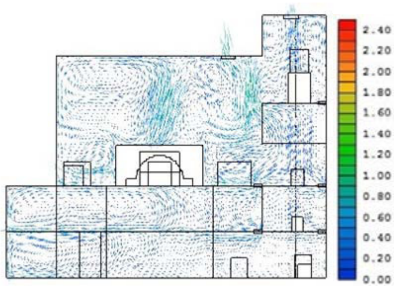

Fig. 8. Velocity distribution of the cross -section (Y=112.2) in steam engine room

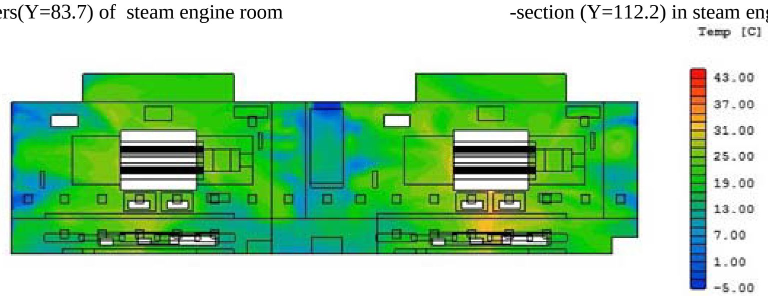

Fig. 9. The temperature distribution of the operation floor of the turbine room (Z=1.8m)

Fig. 10. The temperature distribution of the operation floor of the turbine room (Z=8.5m)

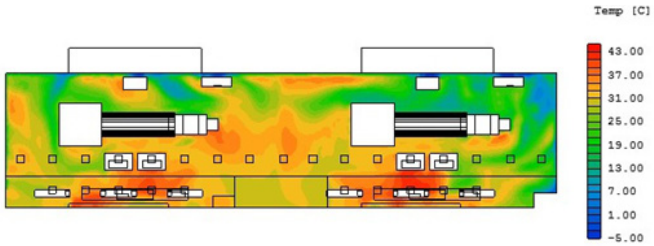

Fig. 11. The temperature distribution of the operation floor of the turbine room (Z=15.5m)

2.2. Comparison of CFD results and field test results in winter

Table 3 and table 4 shows comparison between the CFD simulations results with field test result in Ref (Chen Y, 2014) at typical position of steam turbine room.

Table 3.The comparison between the simulation result and the field test data at different level

Height of the level	Measured value	Simulation value	Error	Data source
27.5m	38.9	39.2	0.77%	Suspension self-counter
24.5m	37.9	38.6	1.85%	Suspension self-counter
21.5m	37.3	37.7	1.07%	Suspension self-counter
18.5m	36.5	35.1	3.84%	Suspension self-counter
15.5m	31.2	32.1	2.88%	Suspension self-counter and Fixed point measurement
14.5m	30	30.1	0.33%	Suspension self-counter
8.5m	32	31.2	2.50%	Fixed point measurement
1.8m	20.6	19.7	4.37%	Fixed point measurement

Table 4.The comparison between simulation result and the field test data at fixed position

Term	Position	Measured value	Simulation value	Error
Velocity	6.85m level grill(Axis B)	0.30	0.27	10%
	6.85m level grill (Axis C)	0.45	0.36	20%
	13.70mlevel grill (Axis B)	0.43	0.37	14%
	13.70mlevel grill(Axis C)	0.66	0.54	18%
Temperature	6.85m level grill(Axis B)	33.0	32.5	2%
	6.85m level grill(Axis C)	33.1	32.7	1%
	13.70m level grill(Axis B)	37.9	37.6	1%
	13.70m level grill(Axis C)	38.2	38.6	1%

It can be seen from Table 3 and Table 4:

1) The simulation results capture the interior temperature stratification phenomena and the relative error between testing data and simulation data is basically below 5%.

2) Comparison of air velocity data at specific location shown in table 6, the relative error is commonly below 10% except three relative errors is among 10%~20%. The cause of this error may be the small geometry, the simplification of the model, the relatively low value of flow velocity which is difficult to test accurately at multi-point .The integrity and the trend of temperature distribution in the whole room were mainly concerned, so the relative big error in ventilation rate can be accepted.

Combining with Table 3 and Table 4, the CFD simulation model under winter condition can well depict the interior thermal environment with a relative low error and thus can be proved effective and reliable.

4. SUMMARY AND OUTLOOK

In order to establish the CFD simulation method for the indoor thermal environment of the steam turbine room of the power plant in the extreme cold area, the comparison between the CFD simulation results and the field test data is carried out. It can be seen that the numerical simulation model is basically consistent with field test data. The main conclusions are as follows:

1) The CFD simulation method for the thermal environment of the steam engine room in the plant power is achieved. From the comparison results, the accuracy and practicability of the CFD model are verified. The CFD model can basically reflect the actual situation of the power plant.

2) This paper validated the boundary conditions including the heat source amount, the distribution of heat source, the indoor ventilation openings and the setting of the porous media. The simplified method of inside conditions and the boundary conditions are reasonable and reliable, which can be used widely in further research.

3) The results of this research can be applied to the detailed and comprehensive study on the ventilation problems of main building in thermal power plant. It can be applied to the analysis and calculation of main building heating energy consumption of thermal power plant. It is helpful to achieve building energy efficiency.

In this paper, a quantitative study of the thermal environment of the main building is conducted, including the quantitative study of CFD simulation and field measurement. Comparison result between the two methods verify the feasibility, accuracy and applicability of the CFD method. This paper has particularly studied the steam turbine room. The complete set of research methods can be used to the research of similar steam turbine room in different plant power. Parts of the results can be applied to the research on the energy consumption of the heating, ventilation and air conditioning system.

References

[1] Bao M. Heating effect improvement by roof ventilators in a large workshop. Journal Heating Ventilating and Air conditioning, 2005(11): 51-55.

[2] Chen Y, Lei M, Chen X, et, al. Analysis on the Test of Indoor Thermal Environment in Main Power House of a Thermal Power Plant in Severe Cold Zone. Building Science, 2014, 30(6):19-26.

[3] Feng Z. Ventilation Optimization in Nuclear Power Plant Turbine Room. 2010, Donghua University, Shanghai, China, 100 pages.

[4] Huang J. Analysis of ventilation design of main factory building of 5th and 6th extension project in Xuanwei power plant. *Yunnan Electric Power*, 2004(2): 42-44.

[5] Li G. Ventilation in thermal power plant in severe cold region. Building Energy and Environment, 1996(4):9-15.

[6] Li S, Kang H,Sun X. HVAC design manual for thermal power plant and substations. In:*China Electric Power Press*, 2011, Beijing, China.

[7] Liu Y. Energy Conservation Simulation Research on Ventilation and Heating Design in Steamer Room. 2009, Huazhong University of Science and Technology, Anhui, China, 62 pages.

[8] Wang J. Study on artificial environment simulation and ventilation adjustment programs for large steam turbine room. 2005, Zhejiang University, Zhejiang, China,70 pages.

[9] Wang X, Liu Y. Discussion on ventilation plans for thermal power plant in cold region. SCI/TECH Information Development & Economy, 2000, 10(6): 54-55.
[10] Yang Q. The Model Test and Ventilation Optimization in Nuclear Power Plant Turbine Room. 2011, Donghua University, Shanghai, China, 127 pages.
[11] Zhang Q. CFD Simulation and Energy Saving Strategy of ventilation and heating in power plant turbine room. 2006, Zhejiang University, Zhejiang, China,79 pages.