A REVIEW PAPER ON CFD ANALYSIS TO PREVENT OVERSIZING OF DUCTS

N.ASHOK KUMAR, B.PHANINDRA KUMAR, MOHAMMED ARBAAZ, P.ARAVIND,
MECHANICAL Department, GURUNANAK COLLEGE

ABSTRACT

This paper focus on study of design modifications to a ductusing computational fluid dynamics(CFD). In the power plant the duct is the major part of the system. The duct system that is used to release the fuel gas from the boiler outlet to the economizer input port. The present review is intended to encompass the fluid flow analysis and heat transfer related work analysis and research. Curved ducts with various shapes and sizes play a very important role in many applications like heat transfer, refrigeration and air conditioning etc. Flowrate and heat transfer in curved duct is different from straight ducts because of secondary flow. Flow of fluids like air, water or nanofluid are investigated with variation of aspect ratio, dimension of duct, Reynolds number etc and obtained result in form of average nussult numbers, friction factor etc.

KEYWORDS; CFD (Computational Fluid Dynamics), Duct, oversizing,prevention.

INTRODUCTION

The duct is an essential part of the air conditioner that carries the air flow rate. The main purpose of ducting system is to transfer the fuel gas from the boiler to the other systems like economizer and air preheater. By using the duct we can reduce the pressure drop and equal flow of the gas to the other systems. The duct can be made by using the Galvanized steel,aluminum, and Polyurethane and phenol insulation panels, fiberglass duct board. In this project we will come to know that the stability of a duct and pressure drops and how they determine the available static pressure, which then leads to the total effective length of our duct system. Related to this project we will come to know that various pressures acting on the duct as well as on the of the inner walls of the duct.

LITERATURE REVIEW

1.**MADHULIKA SINGH, SHAH ALAM**said that "CFD Approach to Design and Optimization of Air, FlueGas Ducting System" In this paper mainly focused on ducting system is used to transfer of air or gas from one place to anotherplace and chamfering the required section to have the uniform distribution of the fuel gas through the duct. And alsopre circulate the gas before entered into the duct. The main objective of this paper is by using the CFD analysis to provide the improved flow condition in air and fuel gas in the duct and also to reduce the pressure loss and lowturbulence.

2.A.ARAVINDKUMAR, works on "Analyzing the gas flow in CFD for various ducts". In this paper focused on to form a vibration less duct and good flow distribution the number of plates provided in the inlet of the ducting system. The number of plates has increased in each case for reduce the vibration and also provide the even distribution of flow. The mainview of this paper is analyzing the new economizer in CFD analysis And Finally the existing design is compared with the new modified design.

3.SREEKANTH S, Dr. BENNY PAUL,on his paper named "CFD analysis of waste heat boiler". In this paper

focused on to guide the flow properly through the RA duct with guide plates to eliminate the vortex generation and circulation induced which was the primary cause for vibrations developed in the duct assembly. The vibration developed reduced Turbulent flow of the gas. So the suitable solution was found that the guide plates are introduced to reduce the vibration. And also reduce the cost of the process.

4.J.J. Bezuidenhout, Y. Yang and J.J. Eksteen has investigated waste heat boiler in which the waste-heat boiler is used within the supplied flash smelting process as the main dust and energy recovery unit. Computationalfluid dynamics (CFD) is done within a study to model the flow analysis and heat transfer distribution throughout theducts. This study focus on geometric modifications and calculations to the boiler, which includes elevation for theceiling, placement of flow- obstructing baffles and also to the radiation plates parallel within the flow system. The geometricmodifications said the desired effect for increasing the volumetric utilization and further enhancing heat transferbetween the boiler surface and also the gas stream and dust segregation.

5.Gonzalo Sánchez et. Al(2019) Design parameters of HVAC installations in high-performance hospital operating theatres were evaluated according to UNE 100713, ASHRAE Standard 170, and pre-standard EN 16244-2. All of them establish a range of valuefor thermodynamic conditions. It was said that ASHRAE Standard is the most tolerant in values proposed for room overpressure. Pre-standard maintains minimum values proposed by UNE standard and does not define a maximum, so in both, the value of this parameter is to designer criteria. ASHRAE Standard recommends is smaller number of filtering stages and less efficiency. The pre-standard adds an additional levels of pre-filtering over UNE standards.

6.M N Rahman Y et. al(2018) The CFD simulation result for Minor Operation Theatre demonstrated differences in real measurement about 19% for air flow velocity and for the 0.04% temperature. The errors that occurred for the uncontrolled mesh density and walls of Minor Operations Theatres in actual Minor Operations Theatre which hard to be quantified for CFD analysis. Based on the observation, better location of AC unit must be proposed for better distribution of air flow in Minor Operation Theatre and it reveals that ANSYS Fluent can be utilized for air simulation in Minor Operation Theatre.

7.Essam E. Khalil et. al(2017)The air is not just a medium but it can be regarded as a guard in the critical health applications. The proper direction for the airflow increase the possibilities for successful pollutant scavenging from healthcare application. The numerical tools, used here, was found to be so effectiveness to predict the airflow pattern in the healthcare facilitieat reasonable cost and acceptable accuracy and Good architectural design allows the HVAC system designers to properly locate the supply outlets and also the extraction ports in optimum locations.

8.Rachael Hopwood Jarvis et. al(2015) It is desirablefor only young air surrounding the patient to reduce risk of contamination and also the continual flow of the clean air into the room from above designed to the ensurepatient is continuously washed in clean air. The simulation indicates the effectiveof this air-wash, indicates the typical flow analysis patterns and average-age-of-air at various locations in room, and the transient simulations also indicates variation seen in these patterns and distributions due to unsteadiness in the flow analysis.

9.Yunyu Chen et. al(2018) In addition to ensure that the indoor temperature and humidity and differential pressure are within the standard range, the clean air conditioner in the clean surgery department also needs to ensure that the indoor air concentration of bacteria and dust particles are within the standard range in order to reduce the risk of wound infection in patients and ensure the health of health care workers. Therefore, the operating effect of clean air conditioning is extremely important.

10.MuhammadIdrusAlhamid et. al(2014)The existing condition showed to have ACH value at 8 ACH. In order to comply with air dischargerates standard of 20 ACH, the freshly air discharges required by the operating room was 1,525 ft3/ min (2,585 m3/ h) . The air flow in the operating room of the existing condition was not very good for sweeping the particles towards RAG because turbulence flow brought back the particles to the operating table. In the redesign condition, the airflow turbulence was minimized by lowering speed and increasing number of RAGs in the room.

11.S L Sinha et. al(2014)Horizontal airflow pattern can provide an important alternative to operating room ventilation airflow design. When the air supply and exhaust facilities are installed on the same lateral wall to generate a horizontal unidirectional flow ventilation in the operating room, the medical lamp and thermal plume induced by temperature difference between people and environment have no obvious influence on the airflow pattern of the occupant area, and the airflow pattern is easy to keep unidirectional. This system performs well in avoiding the influence for the surgeons to the patient. Particles released by surgeons can be wiped off effectiveness, without leading to the increase the particle concentration surroundingthe patient. The risk of the post-operative infection can be decreased significantly. The relative position of source and layout of the operating room highly influences the particle concentration surrounding around the patient. Low particle concentration surrounding the patient, especially around the wound area can be maintained by prescribing the direction of the patient correctly, according to the character of the surgical. The main principle to set the layout is to make sure that the main source of particle is in a downstream location of the wound area.

12.Muneera Abs Farjet.al(2014) They were also ineffective in reducing the OT temperature, the presence of recirculation zones was most prominent and this might trap the contamination inside the OT.The effect of adding two inlets at the bottom showed enhanced flow inside OT. The temperature in the zone of the interest was also reduced significantly. The major advantage of all the three-horizontal inlet setup is that the airflow is not blocked by the primary and secondary light sources. Vertical Inlet Generally vertical inlet cases outperform the horizontal inlet cases, both in terms of velocity and temperature distribution inside the OT. The room was effectively cooled in both the cases, the only drawback was the presence of recirculation zone room. Also Slight increase in temperature in the central region was noticed. it effectively maintain the room temperature, reducing the recirculation zones and good velocity distribution throughout the room. This case will be used for further analysis, to effectively track the contamination removal inside OT.

13.Carla Baloccoet.al(2014)Numerical simulations of airflow, thermal fields and contaminant concentration distributions were carried out for a real OT under different ventilation schemes for supplying and recovering indoor air. Our investigation provides understanding for which ventilation scheme can guaranteebest compromise between IAQ levels and also comfort requirements under real use conditions of the OT. Results confirms the strong effects of a correct ventilation system designs and location of the air supply diffuser on compliance of microclimatic conditions with suggested standard limits, thermal comfort andIAQ levels guarantee and on satisfactory contaminant removal results, with noticeable low contamination levels at wound site.

14.Clive B. Begg et. al(2012) From the foregoing discussion, it is clear that ventilation systems for general wards and patient rooms are specified using criteria that differ little from those used for nonclinical spaces. The guidelines in both the United Kingdom and the United States avoid any discussion of the risks posed by airborne microorganisms, but focus on providing a comfortable environment. This is understandable, given that patient comfort is of great importance and that the clinical risk

posed by many airborne pathogens is unclear. Noneother, there is growing evidence on the aerial dispersion of some nosocomial pathogens is widespread environmental contamination that may lead promoting CFD.

15.Manish Shankar et. al(2016) The design and development of air conditioning equipment needs data and results of the research on the thermal comfort and performance parameters of the system to be designed. These parameters are evaluated from the numerical and experimental studies. Thermal conditions and air distributions have close relation with each other. The literaturereviewshows that air distribution is gaining importance in development of better air conditioning effect and also thermal comfort. Apart from the experimental methods, numerical methods are being used by the researchers for being the effective and low cost technology analysis and simulation. The role of air motion and temperature as the main parameters of the air conditioning needs to be optimized for the physical conditions of the conditioned space and the air distribution method.

16.CĂTĂLIN GEORGE POPOVICI et. al(2015)The HVAC system of the real building was studied by using a 2D model, with specific software ANSYS-Fluent. The HVAC system is subjected to an analysis of its functionality, by evaluating the supplied parameters when operating at maximum load. After an overall outlook of the results, it can be stated with certainty that the recently implemented HVAC system reaches its task and provides adequate comfort conditions inside the spectacle hall during both seasons. It must be noticed that in winter the heat transfer balance is sometimes reversed, determining the necessity to realize the cooling of the audience hall, while the other rooms need to be heated.

17.M N Shami R et. al 2018The CFD simulation result for Minor Operation Theatre demonstrated differences in experimental result are about 3.06% for RH. The errors that occurred because of the uncontrolled mesh density and wall of Minor Operation Theatres in actual Minor Operation Theatre which hard to quantified for CFD simulation. Based on the study observationand it reveals that ANSYS Fluent can be utilized for RH simulation in Minor Operation Theatres.

CONCLUSION

By studying all those papers we conclude that Computational fluid dynamics analysis provides the power to quantify flow improvement and to quickly predict the behavior of several designs. The CFD analysis has offered a comprehensive range of output including velocity distribution, pressure profile and turbulence levels.

REFERENCES

[1]MADHULIKA SINGH, SHAH ALAM on “CFD Approach to Design and Optimization of Air, Flue Gas Ducting System”Research Technology Management, (Sept- Oct), 25-32

[2]A.ARAVIND KUMAR, on “Analyzing the gas flow in CFD for various ducts”UNE 100713ASHRAE Standard 170, and pre-standard EN 16244, -2.

[3].SREEKANTH S, Dr. BENNY PAUL**,** “Guidelines for the Fluid Dynamics Design of power plant Ducts,” Electric Power Research Institute California, 1988.

[4].Bezuidenhout, J.J. Yang, Y. and Eksteen , J.J. “CFD Analysis of a waste-heat boiler associated with flash smelting of base metal sulphides”.

[5] Gonzalo Sánchez et. Al(2019) Design parameters of HVAC installations in high-performance hospital operating theatres were evaluated according to UNE 100713ASHRAE Standard 170, and pre-standard EN 16244, -2.

[6] Menke, Michael M. (1994), Improving CFD Decisions and Execution, Research Technology Management, (Sept-Oct), 25-32.

[7]. S. Jayanti “Techniques for flow passage optimization in air and flue gas ducting in boilers”, BHEL

Nationalconference on CFD Application ,Hyderabad, 17-18 Nov,2006.

[8] Bezuidenhout, J.J. Yang, Y. and Eksteen , J.J. “CFD Analysis of a waste-heat boiler associated with flashsmelting of base metal sulphides”.

[9] Hajee Mohammed, M. and Wilson,V.H. “Analysis of Boiler Super Heater Tubes from High Flue GasTemperature”

[10] Blake Robert Eisner “CFD simulation with fine and a recenty developed turbulence model, the scale adaptivesolution (SAS)model.

[11] CEE, 2002. High Efficiency Commercial Air Conditioning Efficiency

[12] Specifications, Boston, MA. Consortium for Energy Efficiency.

[13] Davis, R., P.Francisco,M. Kennedy, D. Baylon, B.Manclark. 2002. Enhanced Operations & Maintenance Procedures for Small Packaged Roof top HVAC Systems: Protocol Development, Field Review, and Measure Assessment. Submitted to Eugene Water and Electric Board. Seattle, WA .Ecotope ,Inc.

[14] Eley, 2002. Measure Analysis and Life-Cycle Cost. 2005California Building Energy Efficiency Standards, Part 1.Preparedfor the California Energy Commission. San Francisco, CA. Eley Associates, Inc. P400-02-011

[15] 1Energy Design Resources,1998a.IntegratedEnergy Design. DesignSource,www.energydesignresources.com.

[16] Energy Design Resources,1998b.BuildingCommissioning. Design brief prepared by E Source and Architectural Energy Corporationwww.energydesignresources.com.EnergyDesign Resources, 1999.

[17] Energy Design Resources,1998c.Skylighting Design Guidelines. Design guide by Heschong -Mahone Group,www.energydesignresources.com

[18] Energy Design Resources,2001.Economizers. Design brief prepared by

[19] Financial Times Energy, www.energydesignresources.com.

[20] Energy Design Resources,2002.Design for Your Climate. Design brief prepared by Architectural Energy Corporation, available atwww.energydesignresources.com

[21] Energy Design Resources,2003.Integrated Design for Small Commercial HVAC. Design brief prepared by Architectural EnergyCorporation, availablatwww.energydesignresources.com

[22] EWEB, 2003. EWEB “Premium” Outside Air Economizer Requirements, Eugene, OR.Eugene Water and Electric Board.

[23] Faramarzi, R. B. Coburn, R. Sarhadian, 2002.Performance Evaluation of Typical Five Ton Roof top Air Conditioning Units under High Ambient Temperatures,Irwindale, CA. Southern California Edison Company, Refrigeration and Thermal Test Center.

[24] Felts, D. 1998. Pacific Gas and Electric Company Roof Top Unit Performance Analysis Tool Program—Final Report. San Francisco, CA. Pacific Gas andElectric Company.

[25] Henderson, H., Y. Huang, D. Parker, 1999. Residential Equipment Part Load Curves for Use in DOE-2,Berkeley, CA. Lawrence Berkeley National Laboratory,LBNL-42175