ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

CFD ANALYSIS OF GAS COOLER FOR ASSORTED DESIGN PARAMETERS

B Nageswara Rao[*] & K Vijaya Kumar Reddy
[*]Head of Mechanical Department, Govt. Polytechnic, Gannavaram-AP, India
Professor, Department of Mechanical Engineering, JNTU, Hyderabad-500085, India

ABSTRACT

Spray cooling is the most economical and simplest method of exhaust hot gas cooling. Water spray coolers are extensively used in all industries where hot exhaust cooling is mandatory. The spray coolers also play a crucial role in space research and defense research organizations during ground simulation testing to cool the exhaust gas. In spray cooler chamber, the hot gas is cooled by injecting the high pressure water as fine droplets through the orifices circumferentially arranged in different injection planes. The performance of the spray cooler is a function of different variables like mass flow rate of water, number of orifices, orifice diameter, droplet size, length of the spray cooler etc. A spray cooler is to be designed in such a way that it should deliver best performance at lowest cost. In this research paper, a detailed CFD analysis has been carried out on the spray cooler and optimized the design parameters.

KEYWORDS: spray cooler, CFD, Exhaust gas, Design parameters

INTRODUCTION

At any incineration process like waste incineration, steel or cement kiln incineration the hot exhaust gas needs to be cleaned before escapes into our atmosphere. The hot gas has to be cooled before it passes through the filtering equipment; otherwise, the material of the filter cannot with stand for the hot gas temperature. Similarly, in high altitude simulation test facilities, the hot exhaust gas released from the test object has to be delivered to the atmosphere through diffuser –ejector system. In this case also, to protect the hardware of ejector system, the exhaust gas has to be cooled in the spray cooler. The amount of the water that needs to be added in order to cool a certain amount of hot exhaust gas, could be simply derived in balancing the enthalpy. The spray liquid droplets use the sensible heat of the exhaust gas to warm up until evaporation. This lowers the gas temperature as the liquid absorbs the heat energy during the phase change. The vapor then continues to absorb energy as it warms to the surrounding temperature. The droplet size plays a major role to make sure that all the liquid evaporates before it hits the wall surface. The main trade off is the energy cost of creating the smallest possible droplets. Another complicated and challenging task is, complete evaporation of the water droplets inside the given length of the spray cooler. Therefore, in the present research paper, a numerical investigation has been conducted to design, analysis and optimize the parameters of the spray cooler to cool the hot exhaust gas.

CONFIGURATION OF THE SPRAY COOLER

The hot exhaust gas enters the spray cooler on left side. Number of nozzles fixed circumferentially in different planes to the spray cooler. The pressurized water is supplied to the nozzle from external source. Water injected at high pressure is atomized into a fine spray and as the droplets of the spray travel along the length of the cooler. Simultaneous heat and mass transfer takes place between the water and hot gas. The design data given in the table No.: 1 is used for the analysis.

Table No.1: Specifications of the spray cooler

Spray cooler diameter	3.0m
No of injectors	500, 1000, 2000
Exhaust gas temperature	3000K
Spray cooler length	3m, 6m, 12m, 18m
Water mass flow rate	101.322 kg
Injection planes	At 0.1m,0.4m
Orifice diameter	1mm, 2mm, 3mm

NUMERICAL MODELING AND SIMULATION

An axi-symmetric 3D model of the spray cooler is considered for the numerical investigations. Simulations are carried out with commercially available ANSYS-CFX software. The input parameters considered for the analysis are shown in the table No.1Computational simulations are carried out for different spray cooler lengths with different injector diameter and the injectors are arranged in different injection planes. Particles are uniformly injected with 100micron diameter and mass flow rate is divided equally among all the orifices. The enhanced TAB (ETAB) model utilizes the droplet deformation dynamics used in the standard TAB model with a new strategy for the description of the droplet breakup process. In this model, the rate of product drop creation is assumed proportional to the number of the product droplets, with the proportionality constant depends on the breakup regime. Transient simulations are carried out, choosing time step slightly higher than particle time breakup and total time for simulation more than particle travelling time required for leaving the domain from exit. Constant property model is assumed for the analysis.

RESULTS AND DISCUSSIONS

Computational fluid dynamics analysis is carried out for a wide range of design parameters. The parameters such as mass flow rate, number of orifices and the orifice diameter would automatically predict the injection pressure required for creating the water spray. Initially in case (1), the analysis is carried out with fixed orifice diameter of 3mm, mass flow rate of 101.322 kg at staggered injection planes at 0.1m and 0.4m. The spray cooler length is varied from 3m to 18m in four steps (3m, 6m, 12m, & 18m). Secondary droplet breakup model ETAB is adopted for communication with initial droplet diameter of 100micron. The temperature contours on central vertical plane are represented in the fig (1). The hot gas parameters at the exit of the cooler are shown in Table No. (2).

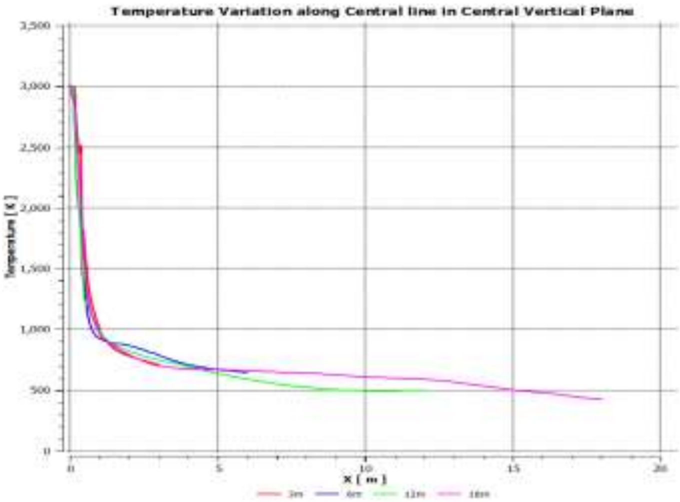


Fig.1 : Temperature variation along the central vertical plane

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

Table No.2: Hot gas properties at the spray cooler exit

spray cooler length in m	Area averaged temperature in K	% of un evaporated water
3	836.612	34.59
6	744.435	25.31
12	660.485	12.71
18	560.751	6.76

After that, in case (2), analysis has been carried out for the effect of the second injection plane position. The first injection plane is positioned at 0.1m from the entrance of the spray cooler. The position of the second injection plane is varied from 0.4m to 3.0m. The particles are uniformly injected from orifice and mass flow rate equally divided among the orifices. 2000 orifices are used with 1000orifices in injection plane. Transient simulations are carried out, with time step of 0.009s and total time of 0.27m. The fig (2) shows the temperature v/s radial height is given below. Maximum temperature reached is lowest for the case of 0.4m position of second injection plane and it is highest for the case of 3m. The 1m and 2m cases are showing almost same maximum temperature. The minimum temperature attained is almost same for the cases 3m and 0.4m.

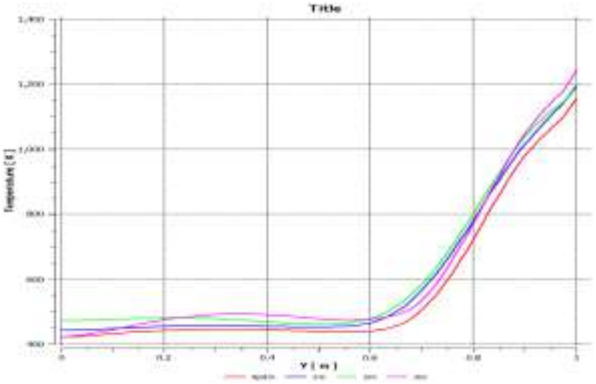


Fig. 2: Temperature variation along the central vertical plane for different injection planes

In case (3), the effect of no. of nozzles is analysed with 500, 1000 and 2000 nozzles. This simulation is carried with a fixed orifice diameter of 3mm and staggered distribution of the orifices in two planes. The time break up, time step and total time for different no. of orifices are shown in the table no.(3).

Table No. 3: Time break up, time step and total time for different no. of orifices

Number of orifices	Time for break up in sec	Time step in sec	Total time in sec
500	0.002	0.003	0.3
1000	0.00412	0.005	0.3
2000	0.00824	0.009	0.27

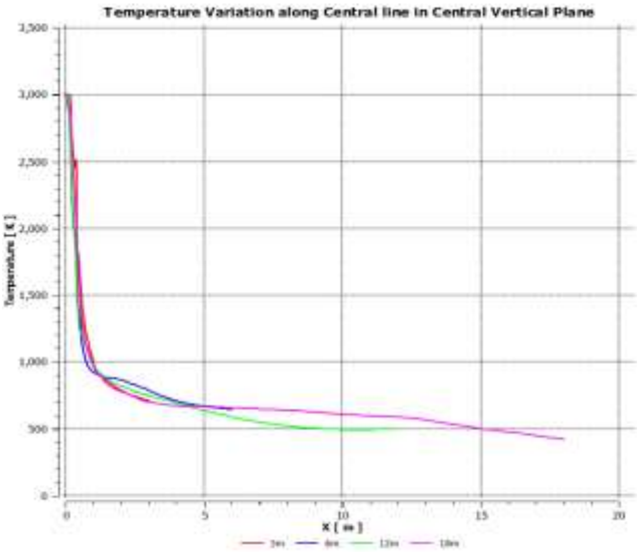
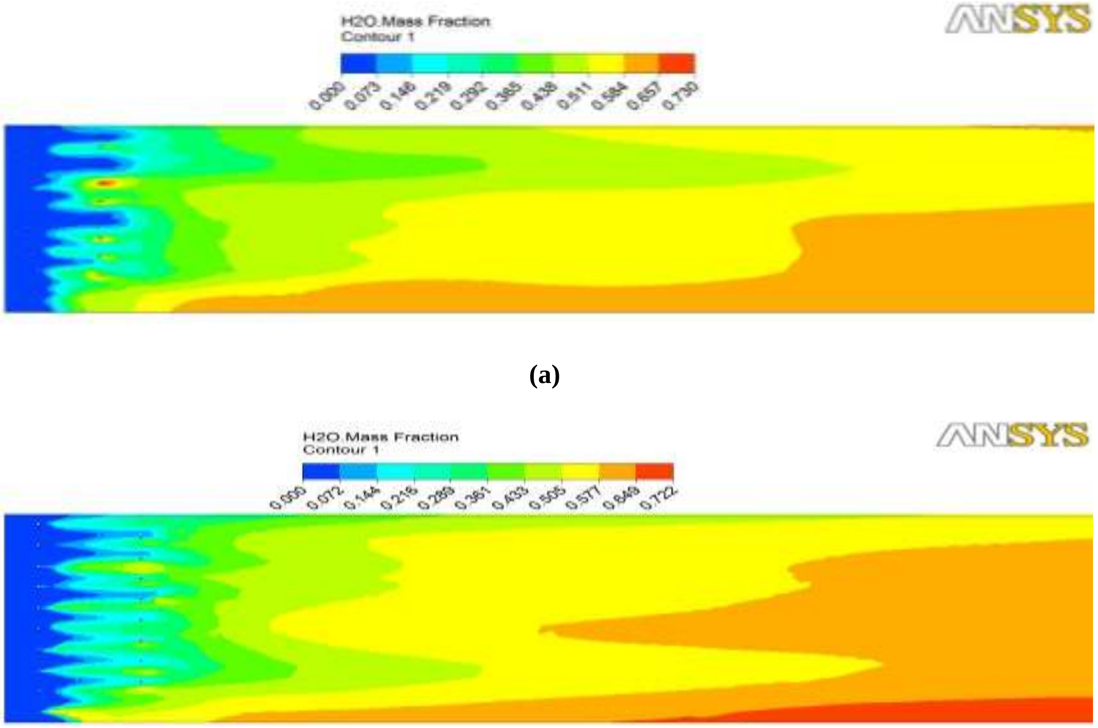


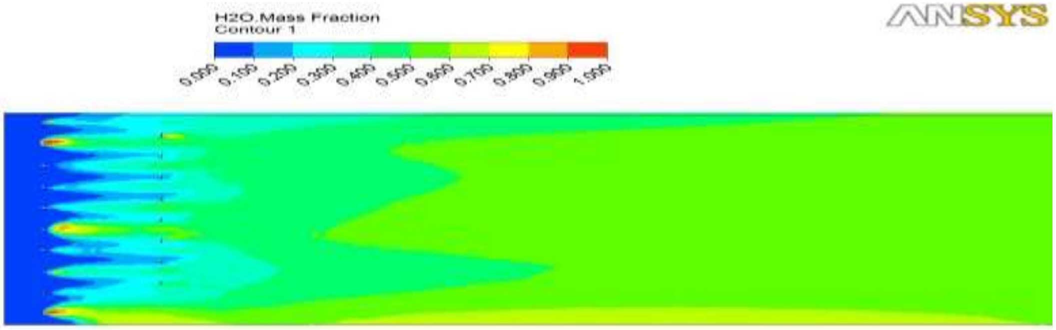
Fig. 3: Temperature variation along the central vertical plane for different number of orifices

In the case (4), the effect of the injector diameter is analyzed with fixed mass flow rate of 101.322kg, 1000 orifices in two injection planes (at 0.1m, 0.4m) for the full length of 18m. The temperature profiles are shown for 1mm, 2mm & 3mm diameter nozzles. Table no. (4) Shows the exit temperature corresponding to nozzle diameter.



(a)

(b)



(c)

Fig.5: temperature pattern with (a) 1mm (b) 2mm (c) 3mm diameter nozzles

Table No.4. Exit temperature vs nozzle diameter

INJ DIA	No. of nozzles in each plane	Area averaged temperature in K	% of un evaporated water
1	500	683.24	-ve accumulation
3	500	515.286	11.4
5	500	636.405	5.14

CONCLUSION

Computational simulations presented in this paper reveals that the spray cooler with water sprays is effectively cool the hot exhaust gas to a sufficiently low temperature before releasing into the environment. From the simulations, it is understood that the parameters are interdependent on each other and highlighted the need for optimization. The simulation results concluded that the spray cooler with 6m length,3mm orifice diameter, 1000 nozzles in two injection planes at 0.1m & 0.4m from the entrance of the cooler is optimum configuration for the present application. Based on the above results, with minimum cost, best performance of the spray cooler can be extracted by doing optimization.

REFERENCES

[1] IS Habib (1973) “The rapid cooling of a hot gas discharge by liquid sprays”.J.Appl. Sci. Res. 28,62-72.
[2] ManikandaKumaran R, et al (2008) “Performance characteristics of a gas cooler in a high altitude test facility”. Proc. Int. Conf. on Aerospace Sci. & Technol., 26-28 June 2008, Bangalore, India.
[3] Sankaran S, et al (2002) “CFD analysis for simulated altitude testing of rocket motors”. Can. Aero. Space J. 48,153-161.
[4] J.S. Markus, “CFD Analysis of Gas Cooling”, Spray Analysis and Research Services”, May, 2007, pp : 1-15.
[5] H. Montazeri, “Evaporative cooling by water spray systems: CFD simulation, experimental validation and sensitivity analysis”, Building and environment, March 23, 2014, pp: 1-21
[6] Charles Ortloff, Spray cooling Heat Transfer-Test and CFD Analysis”, Electronics Cooling, June, 2012, pp: 1-5.