

CFD SIMULATION COLD FLOW

by Fajri Vidian

Submission date: 14-Apr-2023 06:14AM (UTC+0700)

Submission ID: 2063889238

File name: IRJET.pdf (1,003.66K)

Word count: 1334

Character count: 7003

CFD SIMULATION COLD FLOW INSIDE BOILER: THE EFFECT OF BOILER HEIGHT ON TEMPERATURE DISTRIBUTION

Fajri Vidian¹, Permadi Waskito¹

¹Department of Mechanical Engineering, Faculty of Engineering, Universities Sriwijaya, Jalan Raya Palembang-Prabumulih km 32, Indralaya, Ogan Ilir, Sumatera Selatan, 30662, Indonesia

Abstract - Temperature distribution in boiler has an important role in an operation. The temperature distribution in the superheater zone or the top of the boiler must be controlled to prevent over heat. In this study a cold flow simulation was carried out using air to investigate the effect of temperature changes in the boiler. Simulation was done by varying the height of the boilers each of 7 m, 8 m, 9 m at the temperature of air inlet of 900 °C. The simulation results show a decrease on temperature distribution in the boiler due to the increase on the height of boiler. The final outlet temperature of the boiler for each of height were 370.3 °C, 345.3 °C, 336.3 °C.

Key Words: CFD, Cold Flow, Boiler, Temperature

1. INTRODUCTION

Boiler is one of the most important parts of a steam power plant. The temperature distribution in the boiler is very important in producing good performance of the boiler. The temperature distribution at the superheater zone should not be too high because it can damage the tubes in the superheater (1). Cold flow simulation is a simulation method that does not involve a chemical reaction. This condition could facilitate to predict the initial influence of a variable on boiler performance parameters. Several studies have been conducted using cold flow simulation. Vs naveen et al. [2] have simulated the flue gas flow in the duct of the heat waste recovery plant, simulations were carried out to analyze and improve flow distribution for increasing boiler efficiency. Rohith et al. [3] have conducted a cold flow simulation on internal combustion engine to see the effect of moving the crank shaft on the velocity distribution in the cylinder. Fajri et al. (4) have conducted a cold flow simulation on an external recirculation updraft gasifier, simulation was carried out to obtain the effect of ejector geometry and nozzle exit position on recirculation flow. Ferreira et al. [5] have simulated on isothermal gas flow in a kraft recovery boiler to investigate effect of air inlet port position on optimal operation of boiler. Florin et al. (6) have simulated air-methane mixed flow in the swirl burner to obtain optimum condition of combustion on the front of burner. S Bharat et al. [7] have conducted a simulation on the swirl combustor gas turbine, simulation was conducted to observe the effect of vane angle on swirl flow. Artit et al. (8) have done an isothermal simulation on combustor vortex to investigate swirl flow indication using the turbulence model of the standard k- ϵ , RNG k- ϵ , SST k- ϵ and RSM k- ϵ . In this

study was simulated cold flow inside boiler to investigate the effect of boiler height on temperature distribution.

2. METHODOLOGY

Simulations were carried out on the 3D of industrial boilers by varying the height of X as shown in Figure 1. The height of X was varied each of 7 m, 8 m, 9 m respectively. The air inlet speed was used at 80 m/s with a temperature of 900 °C [9]. The amount of the duct of air inlet and outlet were 6 and 1 respectively. The 3D calculation domain is shown in Figure 2. The mesh generation as shows in Figure 3, was used to calculate the fluid flow and energy equation in domain of calculation. Numerical calculations were performed using finite element method that executed using Autodesk CFD Simulation.

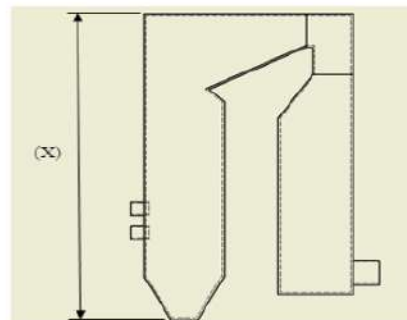


Figure 1. The position of height variation (X)

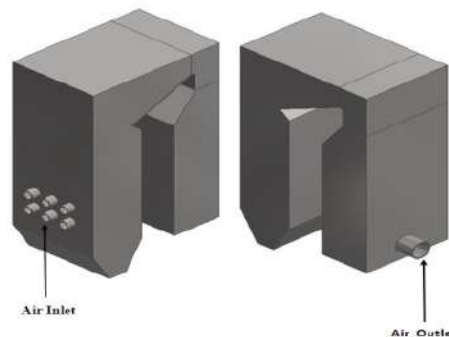


Figure 2. The 3D of Calculation Domain

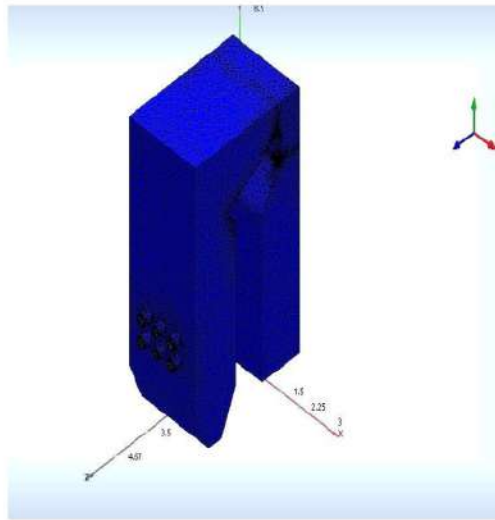


Figure 3. The Meshing of Calculation Domain

3. RESULTS AND DISCUSION

Figure 4 shows the effect of boiler height on the temperature distribution in the boiler. The increase of boiler height tended to decrease the temperature distribution in the boiler that was identified by decreasing the yellow contour, especially at the top of the boiler. The temperature distribution contour at the top of the boiler in the x-z direction is shown in Figure 5. The temperature distribution along the center of the top of boiler is shown in Figure 6. The final temperature at outlet of boiler at height of 7 m, 8 m, 9 m were 370.3 °C, 345.3 °C, 336.3 °C, respectively.

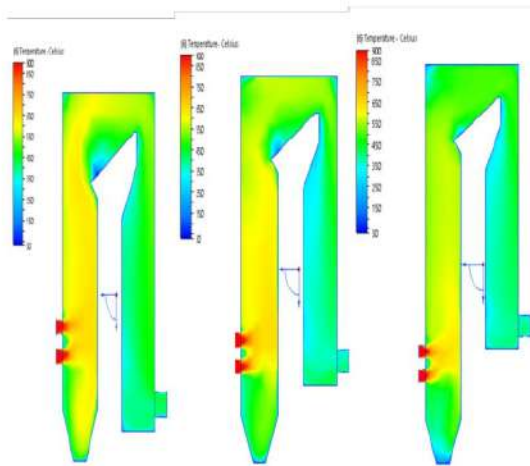


Figure 4. Temperature Distribution inside Boiler : (a) 7 m; (b) 8 m; (c) 9 m.

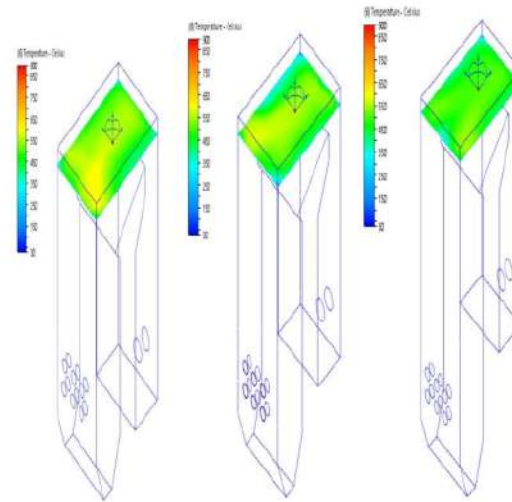


Figure 5. Temperature Distribution at The Top of Boiler (Superheater Zone): (a) 7 m; (b) 8 m; (c) 9 m.

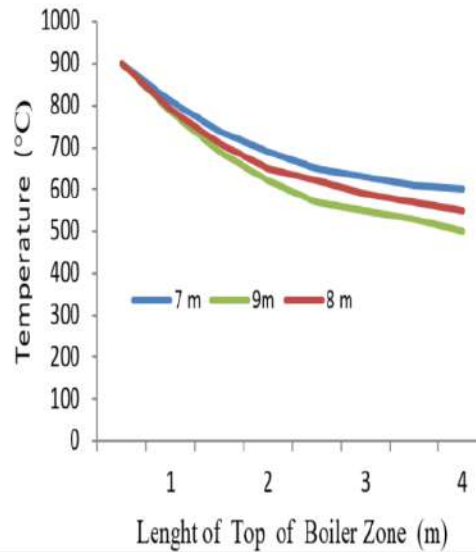


Figure 6. Temperature Distribution along of Top of Boiler

The temperature distribution along the boiler from inlet to outlet for each of height of 7 m, 8 m, 9 m are shown in Figure 7-9. The decrease of temperature in the boiler was represented at point of height of 4 m. Boiler height of 7 m, 8 m, 9 m had temperature at point of 4 m were 610 °C, 601 °C, 585 °C respectively.

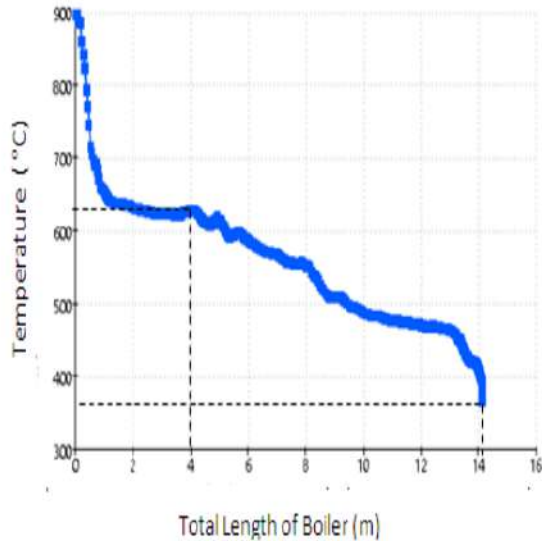


Figure 7. Temperature Distribution along of Total Length of Boiler at x = 7 m.

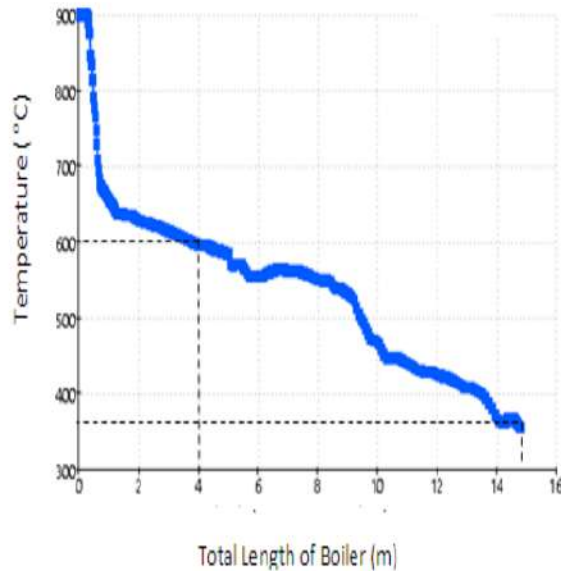


Figure 8. Temperature Distribution along of Total Length of Boiler at x = 8 m

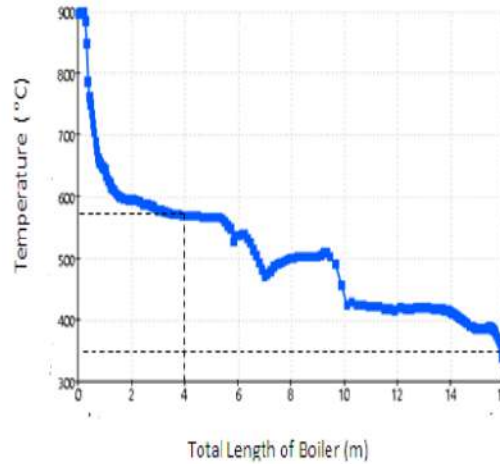


Figure 9. Temperature Distribution along of Total Length of Boiler at x = 9 m

4. CONCLUSION

The simulation results showed an increasing of boiler height of 7 m, 8 m, 9 m respectively decreased on air temperature distribution inside the boiler. The temperature distribution in the top boiler (superheater zone) decreased with an increasing of boiler height. The temperature of air at outlet boiler for each of height were 370.3 °C, 345.3 °C, 336.3 °C.

REFERENCES

- [1] Raja Saripalli, Ting Wang, Benjamin Day, Simulation of Combustion and Thermal Flow in An Industrial Boiler, Proceedings of 27th Industrial Energy Technology Conference, New Orleans, Louisiana, (2005).
- [2] VS Naveen J, Prof. A. Roma Krishna, CFD Simulation of Flue Gas Ducting in Waste Heat Recovery Plant, International Journal of Engineering Sciences & Research Technology, 6(10), 220-231, (2017).
- [3] Rohith. S, Dr. G .V. Naveen Prakash, Cold Flow Simulation in an IC Engine, International Research Journal of Engineering and Technology (IRJET), 02 (07), 82 -87, (2015).
- [4] Fajri Vidiana, Adi Surjosatyo, Yulianto Sulisty Nugroho, Computational Fluid Dynamics Analysis of External Recirculation Flow at Updraft Gasifier Using Ejector, AIP Conference Proceedings 1440, 936, (2012).
- [5] DJO. Ferreira, M. Cardoso, S.W. Park, Gas Flow Analysis in Kraft Recovery Boiler, Fuel Processing Technology ,91, 789-798, (2010).

- [6] Florin Bode, Victor Hodor, Corina Giurgea, Numerical Investigation on A Swirl Burner with Internal Flue Gas Recirculation, Scientific Bulletin of the "Politehnical University of Timisoara" Transactions on Mechanics Tom 52 (66), Fascicola 3, (2007).
- [7] S Bharat Krishna, Prof V Ganesan, CFD Analysis of Flow through Vane Swirlers, IE (I) Journal.MC, 86, (2005).
- [8] Artit Ridluan, Smith Eiamsa-ard, Pongjet Promvonge, Numerical simulation of 3D turbulent isothermal flow in a vortex combustor, International Communications in Heat and Mass Transfer, 34 (7), 860-869, (2007).
- [9] Marko Huttunen, Juho Peltola, Sirpa Kallio, Lassi Karvonen, Timo Niemi, Ville Ylä-Outinen. Analysis of the processes in fluidized bed boiler furnaces during load changes, Energy Procedia 120, 580-587, (2017).

CFD SIMULATION COLD FLOW

ORIGINALITY REPORT

16%

SIMILARITY INDEX

12%

INTERNET SOURCES

11%

PUBLICATIONS

10%

STUDENT PAPERS

MATCH ALL SOURCES (ONLY SELECTED SOURCE PRINTED)

6%

★ Submitted to Stefan cel Mare University of
Suceava

Student Paper

Exclude quotes Off

Exclude bibliography On

Exclude matches < 1%