ECHAHID HAMMA LAKHDAR UNIVERSITY - EL-OUED
Under the Supervision of the DGRSDT and in collaboration with the CRTI
International PluridisciplinaryPhDMeeting (IPPM'20)
1st Edition, February23-26, 2020
Theme: Energy

Effect of baffles dimension on the combustion reaction inside an industrial sulphur burner

Souhir Echi*, Abdallah Bouabidi*, Raffaella Gerboni**,Zied Driss*

*Laboratory of Electro-Mechanic Systems (LASEM), National School of Engineers of Sfax (ENIS), University of Sfax (US), B.P. 1173, Road Soukra km 3.5, 3038 Sfax, TUNISIA
**DipartimentoEnergia - Politecnico di Torino (POLITO), CorsoDucadegli Abruzzi, 24 - 10129 Torino ITALY
E-mail: echi.souhir@yahoo.com, bouabidi_abdallah@yahoo.fr, raffaella.gerboni@polito.it, zied.driss@enis.tn

Abstract

The sulphur boiler furnace used in phosphate industries usually contains three baffles. Their role is to guide the fluid flow inside the combustion chamber. So, they are useful to enhance a complete reaction. The present work shows a numerically investigation of the effect of increasing the baffles dimension on the combustion reaction that occurs inside a sulphur burner. The simulations are performed using the commercial software Ansys Fluent 16.2. A non-premixed combustion model is carried out. The fluid characteristics such as temperature, velocity and mixture fraction are presented.

Keywords: industrial, turbulence, combustion, baffles, simulation

1. Introduction

Nowadays, researchers focus their interest on the optimization of industrial applications. Their aim is to reduce the emissions produced by combustion processes. In fact, the incomplete combustion reaction inside burners can be very harmful for human health and for our environment. In this context, many previous studies used CFD simulations to study combustion processes [1-17]. The studied industrial sulphur burner is used in phosphate industries. Brown et al. [18-21] developed a similar process with one baffle and with six spray nozzles of molten sulphur. Their results are in good agreement with experimental results. They studies also the effect of spray nozzles orientation on the performance of the burner. Chidambaram [22] reported the thermal damage inside a combustion chamber with various evidences from visual inspections. Many repair solutions are proposed. This illustrates that CFD tools can improve knowledge about the internal reaction mechanism that occurs inside a fuel burner. It makes the optimization study more feasible. In our case, we evaluated the effect of baffles dimension on the combustion reaction inside a sulphur boiler furnace.

2. Numerical approach

The Ansys Fluent software is used to develop the numerical simulation of the fluid flow behavior inside the sulphur boilerfurnace. This equipment contains four air inlets encountering the four sulphur inlets and two secondary air inlets located on the top. The combustion chamber contains three baffles that guide the fluid until it will be evacuated by the outlet. A combustion reaction will occur between the injected liquid sulphur and the entered dry air. The burner geometry is presented in figure 1 and it is identical to an industrial equipment. The dimensions are detailed in a previous work.

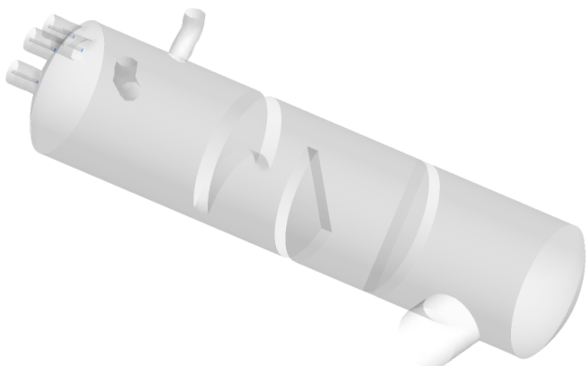


Fig 1.Sulphur boiler geometry

Figure 2 presents a mesh convergence study that is investigated in order to choose the adequate mesh. Different grid sizes are performed and reported. The elements size are 304 277, 582 473, 991 111 and 1 135 095. The fourth mesh is chosen in function of computations cost.

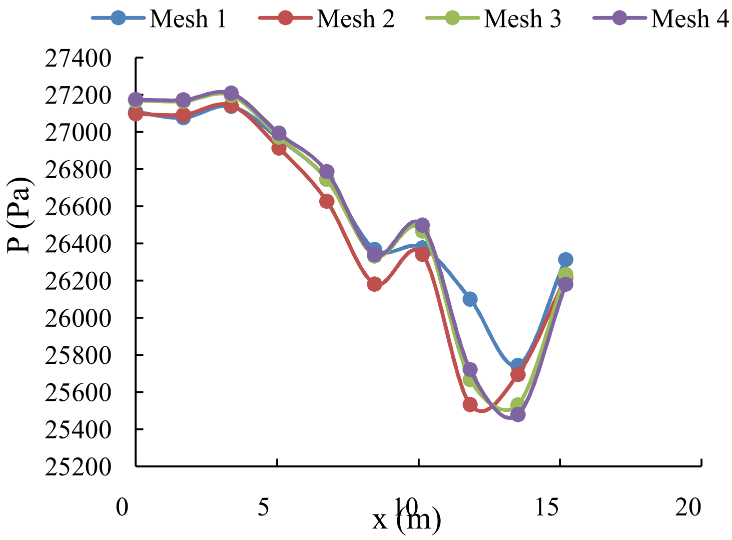


Fig 2. Mesh convergence

This is a non-premixed combustion model. It is very essential to choose the adequate combustion mechanism in the species model.In order to examine the numerical model validity, the numerical results are compared to data collected from the industrial. The data are measured in the outlet zone. A good agreement was shown.

For the baffles dimension modification, we present the following geometries as shown in figure 3. The width of the baffles is increased, the different parameters are presented in Table1.

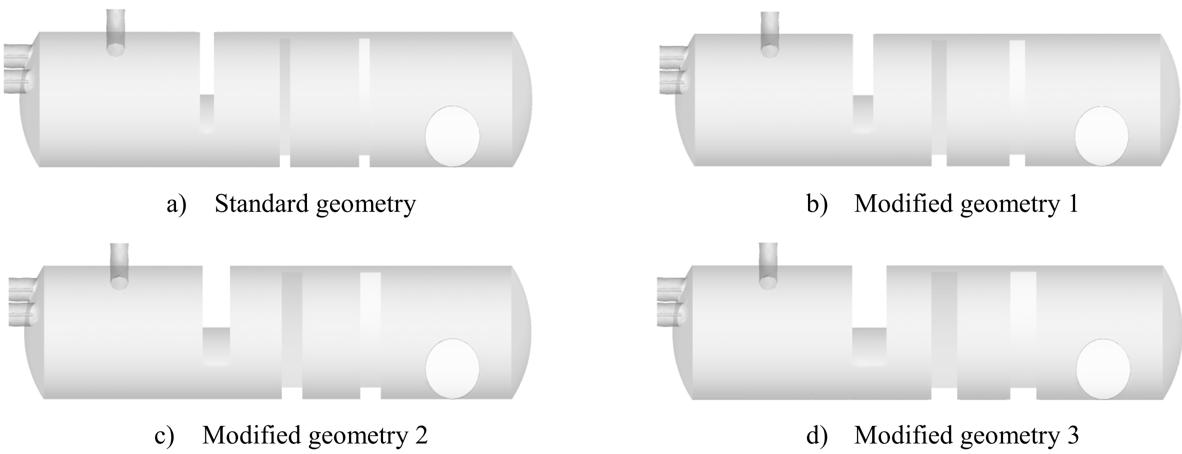


Fig 3. Geometry modifications

Table 1. Modified width dimensions

	Baffle width (mm)
Standard geometry	445
Modified geometry 1	667.5
Modified geometry 2	890
Modified geometry 3	1335

3. Results and Discussion

For numerical results we choose a reference plane in order to present the different distributions. It is located in the (xz) at z= -0.47m.

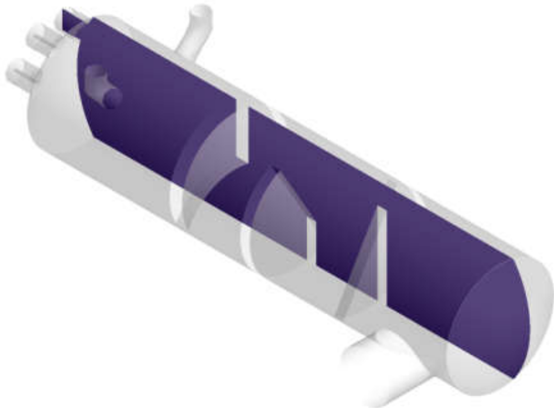
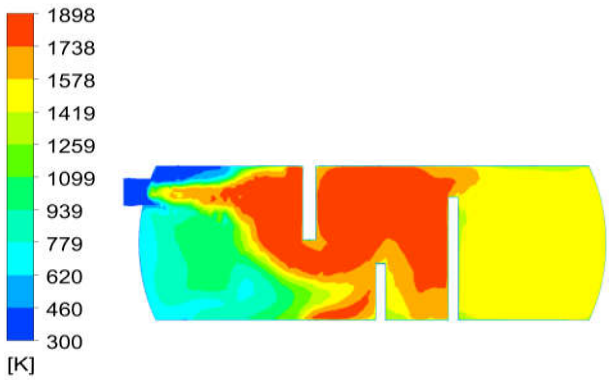


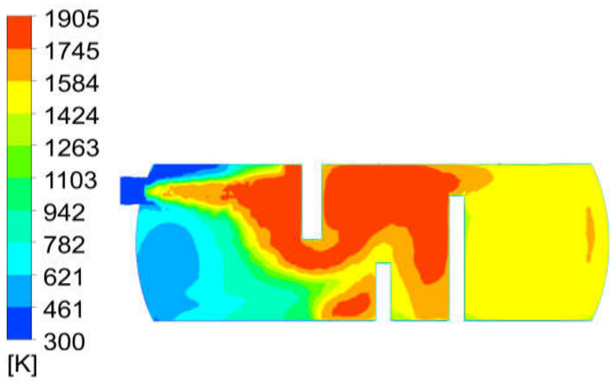
Fig 4. Reference plane

3.1. Temperature distribution

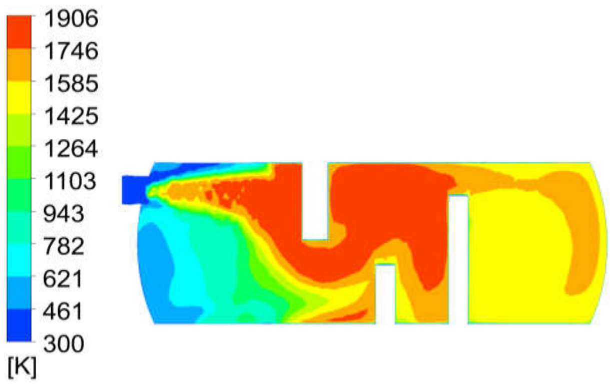
Figure 5 illustrates the 3D temperature distribution inside the modified sulfur boiler designs. It is observed that there is a vast red zone that presents the increase of temperature in the combustion chamber. In this area, the combustion reaction between the liquid sulphur and dry air occurs. It is very important to identify the fluid flow behavior then to optimize the design. In order to improve this industrial equipment productivity it is essential to have a complete combustion of sulphur. It is clear that the maximum temperature value is reached near the baffles zone. In fact, modifying the width of these baffles can affect the total reaction.



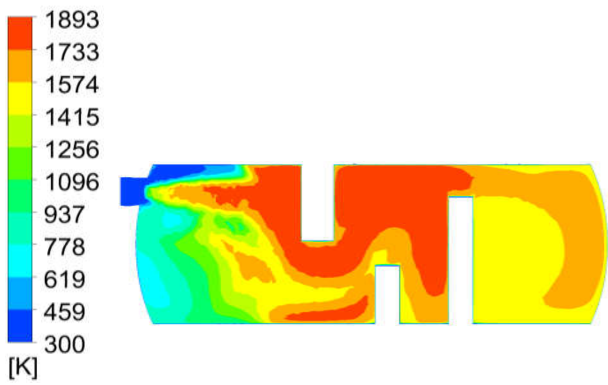
a) Standard geometry



b) Modified geometry 1



c) Modified geometry 2

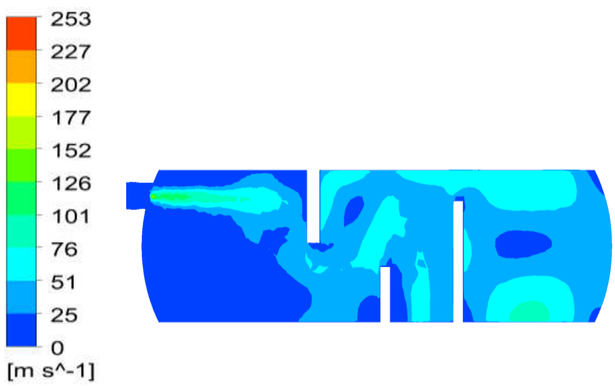


d) Modified geometry 3

Fig 5. Temperature distribution in 2D plane

3.2. Velocity distribution

Figure 6 presents the velocity magnitude distribution inside the modified sulfur boiler geometries. It is observed that the sulphur is injected with a high velocity compared to air velocity inlet. It is clear that the velocity distribution decrease when we increase the dimension of the baffles. In order to have a complete combustion, it is important to decrease the fluid flow velocity. So all the entered species should be well mixed. The velocity distribution could explain the fluid flow trajectories inside this burner.



a) Standard geometry

b) Modified geometry 1

c) Modified geometry 2

d) Modified geometry 3

Fig 6. Velocity distribution in 2D plane

3.3. Mixture fraction distribution

Figure 7 illustrates the mixture fraction distribution inside the modified sulfur boiler geometries in comparison with the standard geometry. It is obvious that the sulphur is injected in horizontal orientation. When the sulphur reaches the zone of the entered air, a combustion reaction takes place. This mixture produces the sulphur dioxide. So, in the baffles zone the sulphur disappears. It is seen that the mixture fraction appears with important value near the sulfur inlets.The numerical results show that the boiler design affects the mixture fraction distribution.

a) Standard geometry

b) Modified geometry 1

c) Modified geometry 2

d) Modified geometry 3

Fig 7. Mixture fraction distribution in 2D plane

4. Conclusion

In this paper, a numerical model focused on the non-premixed sulphurcombustion reaction was developed using the Ansys Fluent software. It was used to study the fluid flow inside an industrial sulfur boiler. We presented the modified geometriesin order to improve efficiency of the sulfur boiler. The modification of the boiler design was examined by means of CFD. The numerical simulation results revealed the fluid flow behavior inside these new designs. This study could be a useful tool to study the efficiency analysis and design optimization purpose.

References

1. AdamczykW. P., KozolubP., KlimanekA., BialeckiR. A., AndrzejczykM., &KlajnyM. (2015). Numerical simulations of the industrial circulating fluidized bed boiler under air- and oxy-fuel combustion.*Applied Thermal Engineering*(vol. 87, pp. 127-136.
2. Mahmoodi B., Hosseini S. H., Ahmadi G., &Raj A. (2017). CFD simulation of reactor furnace of sulfur recovery units by considering kinetics of acid gas (H2S and CO2) destruction. *Applied Thermal Engineering*, 123:699–710.
3. MadejskiP. (2018). Numerical study of a large-scale pulverized coal-fired boiler operation using CFD modeling based on the probability density function method. *Applied Thermal Engineering*,doi: https://doi.org/10.1016/j.applthermaleng.2018.09.004.
4. PerroneD., CastiglioneT., KlimanekA., MorroneP., &AmelioM. (2018). Numerical simulations on Oxy-MILD combustion of pulverized coal in an industrial boiler. *Fuel Processing Technology* (vol. 181, pp. 361–374).
5. YangM., ShenY.Y., XuH.T., ZhaoM., ShenS.W., &HuangK. (2015). Numerical investigation of the nonlinear flow characteristics in an ultra-supercritical utility boiler furnace. *Applied Thermal Engineering* (vol. 88, pp. 237-247).
6. AsotaniT., YamashitaT., TominagaH., UesugiY., ItayaY., &MoriS. (2008). Prediction of ignition behavior in a tangentially fired pulverized coal boiler using CFD. *Fuel* (vol. 87 pp. 482–490).
7. TangG., WuB., JohnsonK., KirkA., FuD., &ZhouC. Q. (2016). Numerical study of a tangentially fired boiler for reducing steam tube overheating. *Applied Thermal Engineering* (vol. 102, pp. 261–271).
8. Secco S. D., Juan O., Louisy M. L., Lucas J. Y., Plion P., &Porcheron L. (2015).Using a genetic algorithm and CFD to identify low NOx configurations in an industrial boiler. *Fuel*, 158:672–683.
9. AthanasiosN., NikolaosN., NikolaosM., PanagiotisG., &KakarasE. (2015). Optimization of a log wood boiler through CFD simulation methods. *Fuel Processing Technology*, 137:75–92.
10. ParkH. Y., BaekS. H., KimY. J., KimT. H., KangD. S., &KimD. W. (2013). Numerical and experimental investigations on the gas temperature deviation in a large scale, advanced low NOx, tangentially fired pulverized coal boiler. *Fuel*(vol. 104 pp. 641–646).
11. Nieckele A. O., Naccache M. F., &Gomes M. S. P (2011). Combustion performance of an aluminum melting furnace operating with natural gas and liquid fuel. *Applied Thermal Engineering*, 31:841-851.
12. MadejskiP. (2018). Numerical study of a large-scale pulverized coal-fired boiler operation using CFD modeling based on the probability density function method. *Applied Thermal Engineering*, doi: https://doi.org/10.1016/j.applthermaleng.2018.09.004.
13. DugumA., &HanjalićK. (2018). Numerical simulation of coal-air mixture flow in a real double-swirl burner and implications on combustion anomalies in a utility boiler. *Energy*; doi: 10.1016/j.energy.2018.12.121.
14. YangM., ShenY.Y., XuH.T., ZhaoM., ShenS.W., &HuangK. (2015). Numerical investigation of the nonlinear flow characteristics in an ultra-supercritical utility boiler furnace.*Applied Thermal Engineering*(vol. 88, pp. 237-247).
15. Tomanovic´ I., Beloševic´ S., Crnomarkovic´ N., Milic´evic´A., &Tucakovic´D. (2019). Numerical modeling of in-furnace sulfur removal by sorbent injection during pulverized lignite combustion. *International Journal of Heat and Mass Transfer* (Vol. 128, pp. 98–114).
16. Wang X., Liu X., Li D., Zhang Y., &Xu M. (2015). Effect of steam and sulfur dioxide on sulfur trioxide formation during oxy-fuel combustion. *International Journal of Greenhouse Gas Control*, 43:1–9.
17. Centeno-González F. O., Silva Lora E. E., Villa Nova H. F., Mendes Neto L. J., Martínez Reyes A. M., Ratner A., &Ghamari M (2017). CFD modeling of combustion of sugarcane bagasse in an industrial boiler. *Fuel* (vol. 193 pp. 31–38).
18. BrownK., BadeK., & SchickR. (2017). Computational model of large-capacity molten sulphur combustion spray efficacy and process efficiency.*The Journal of the Southern African Institute of Mining and Metallurgy* (volume 117, November 2017. Page 1035-1043), http://dx.doi.org/10.17159/2411-9717/2017/v117n11a8.
19. Brown K., Kalata W., &Schick R. (2014).Optimization of SO2 Scrubber using CFD Modeling. *Procedia Engineering*,83 (2014) 170 – 180,doi: 10.1016/j.proeng.2014.09.035.
20. Brown K., Kalata W., & Schick R. (2014). Spray Injectors within Large Capacity Molten Sulfur Combustion: Fluid- Structure Interaction Study. *2nd International Symposium on Innovation and Technology in the Phosphate Industry,[SYMPHOS 2013]*.
21. Brown K., Bade K., & Schick R. (2013). Computational Model of Large Capacity Molten Sulfur Combustion Spray Efficacy and Process Efficiency. *ILASS Americas, 25th Annual Conference on Liquid Atomization and Spray Systems, Pittsburgh, PA.*
22. Chidambaram S.(2018). Thermal damage of sulfur processed chamber under Clausoperating reaction conditions—A case study. Construction Materials (8, 517–529).
23. Echi S., Bouabidi A., Driss Z., Abid M. S. (2019). CFD simulation and optimization of industrial boiler. *Energy*,doi: https://doi.org/10.1016/j.energy.2018.12.006.