Journal of Farayandno – Vol. 18 – No. 82 (2023): pp. 5-23

National Iranian Oil Refining and Distribution Company (NIORDC)

Journal of Farayandno

Research Paper

Investigating the Medium Pressure Flare Performance of South Pars Gas Refinery in Design and Operational Conditions Using CFD Simulation

Mohammad Irani [1*], Yaghob Behjat [1]

[1] Research Institute of Petroleum Industry (RIPI)

Received: 25 April 2023 **Accepted:** 30 July 2023

1. ABSTRACT

In this article, the CFD simulation of the medium pressure flare of South Pars 5th refinery has been carried out in industrial dimensions in order to investigate the effect of the flare tip geometry on the flare combustion characteristics. CFD model was developed. In order to avoid numerical errors proper meshing was done. The results of CFD simulation show that in the operational conditions, the increase in the temperature of the Flare type body due to the impact of the flame is negligible, so its performance is suitable with the current amount of sweeping gas. Therefore, it is not possible to further reduce the gas sent to the MP flare because the flame may be drawn into the flare network, the temperature of the flare tip surface and the wind shield may increase, which in the long run will cause the flare tip surface to be destroyed.

Keywords: Combustion, flame, MP flare, CFD.

2. INTRODUCTION

Flaring is widely used in refinery, petrochemical and chemical industries to eliminate undesirable combustible gases by burning in an open flame environment. The purpose of making flares is to receive the unwanted gases sent by the industrial unit and burn these gases in order to prevent them from entering the environment. In the industry, there are factors such as power cuts, changes in the input to the units, utilization exceeding the design capacity. From production units, improper maintenance and repair, deviation from correct methods and operating instructions, human errors, etc. are among the factors that cause process changes and pressure increase in tanks, towers and other industrial equipment Flaring of accompanying gases is one of the methods that is mostly done in technical dimensions in order to increase the safety of equipment and prevent potential risks such as explosions and other possible risks due to the increase in pressure and the flaming of high-pressure gases during crude oil extraction.

Many experimental and simulation studies have been conducted on flares. Huang et al investigated the stability and flame behavior of a propane combustion jet in a laboratory manner and presented relations for the movement path of combustion jets in a cross flow [1]. Bourguignon and colleagues presented a method to measure the combustion efficiency of the flame in the cross flow [2]. Johnson et al., by conducting about penetration flames in the path of fluid flow, showed that wind flow and fuel energy content are directly effective on combustion efficiency [3]. The results of Kostiuk and Johnson's research showed that flares have a high efficiency in still air and increasing the wind speed causes a decrease in efficiency [4]. Castchiera and Edgar, by CFD simulation, investigated the effect of air and steam injection on the performance of flares in a laboratory scale with a non-premixed flame [5]. Their results showed that the use of very high ratios of steam to fuel and air to fuel causes a significant decrease in flare combustion efficiency and unburnt hydrocarbon production. According to the topics raised in many sources, it can be concluded that the research done on flares using CFD modeling is mostly limited to lab-scale flares and little research has been done on industrial flares,

* iranim@ripi.ir

Please Cite This Article Using:
Irani, M., Behjat, Y., "Investigating the Medium Pressure Flare Performance of South Pars Gas Refinery in Design and Operational Conditions Using CFD Simulation", Journal of Farayandno – Vol. 18 – No. 82, pp. 5-23, In Persian, (2023).

which can be Due to the wide time span and the size of the flares, the high turbulence of the flow, the complex kinetics and the significant effect of the turbulence on the kinetics of combustion reactions.

Conservation equations

The mass conservation, momentum, energy and species, can be expressed as:

Mass: $\nabla.(\vec{v}\rho\ ) = 0$ (1)

Momentum: $\nabla.(\rho\vec{v}\,\vec{v}) = -\nabla P + \nabla.\left[\mu\left(\nabla\vec{v} + \nabla\vec{v}^{T}\right)\right] + \rho g + S$ (2)

Energy: $\nabla.(\vec{v}(\rho\ H\ + P)) + \nabla.(\sum_{i=1}^{n} h_i\, j_i) = -\nabla.(q) + S_R$ (3)

Species: $\nabla.(\vec{v}C_i - \mathrm{D_i}\nabla C_i) = R_i$ (4)

Where, ρ represents mixture density, $\vec{v}$ is velocity vector, H and hi are total enthalpy and enthalpy of species, respectively. P is the static pressure and Ci stands for concentration of chemical species.

3. RESULTS AND DISCUSSION

Figure 1 shows the temperature profile and the shape of the flame formed in the MP flare in the design conditions. As it is clear in the figure, due to the very high-volume flow of gases sent to the flare in the design mode, a very large flame is formed. So that the height of the formed flame is more than 25 meters and it is out of the considered calculation range.

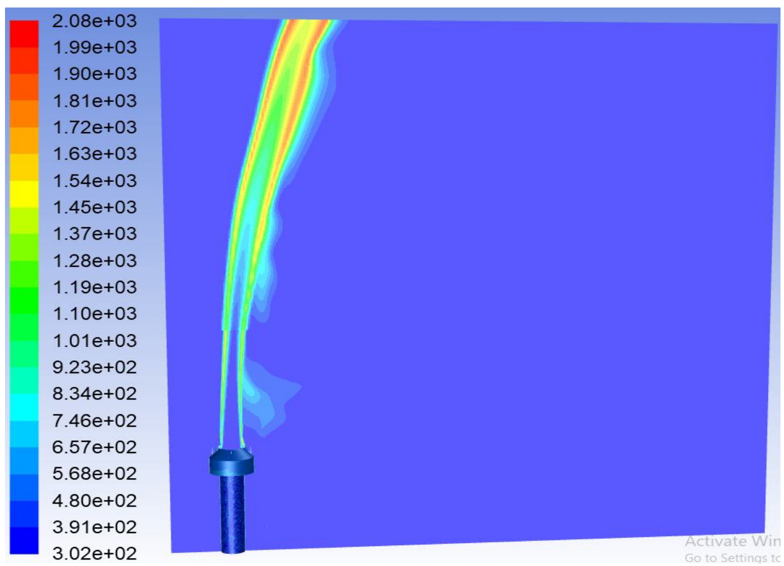

Figure 1. Temperature profile and flame height formed in MP flare (design conditions)

Also, the temperature contour in the area of the wind shield and the opening of the flare tip is shown in Figure 2. As can be seen, there is no flow return inside the flare network and the temperature of the wind shield and the temperature of the flare tip opening has not increased, and the possibility of the flame returning inside the flare network is eliminated. Therefore, it can be concluded that the performance of the MP flare is suitable in the design conditions and there is no need to redesign it in the functional design conditions.

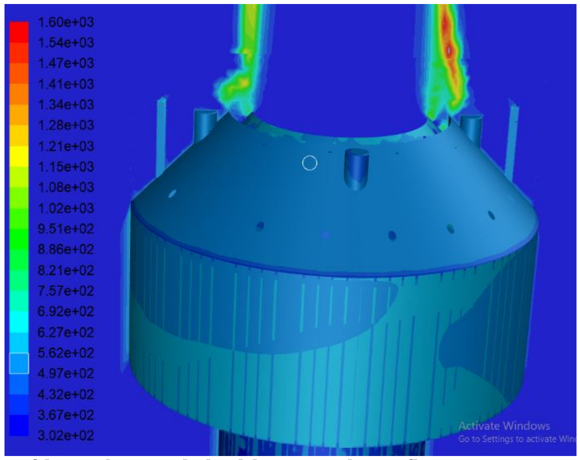

Figure 2. Temperature profile in the wind shield area and MP flare tip opening at operating conditions.

4. CONCLUSION

In this research, the CFD simulation of the refinery's MP flare was carried out on an industrial scale and in operational and functional conditions. In the performed simulations, the effect of all related transfer phenomena including (transfer of momentum, heat, mass, radiation, turbulence and chemical reactions) as well as environmental conditions including wind speed have been considered. Considering the complexity of the tip flarer and the effect of each of the geometric details on the performance and combustion efficiency in the tip flarer, all the geometric details of the tip flarer have been considered based on the design drawings. So that the drawn geometry does not have any simplification. By creating a suitable computing network, better convergence was achieved in solving equations. So that around the flame stabilizer, steam injection nozzles, Gas Seal and the internal parts of the flare tip, the meshes were chosen smaller. On the other hand, in the other points of the computational domain and away from the tip flare, regular gridding has been used to reduce numerical errors. The results of the CFD simulation including the temperature profile (flame shape) and the velocity of the combustion gases in the MP flare in operational conditions show that the increase in the body temperature of the flare type due to the impact of the flame is small, so its performance is suitable with the current amount of sweeping gas. . Therefore, the further reduction of the gas sent to the MP flare is not justified because the flame may be drawn into the flare network or due to the very low height of the flame and the ambient wind speed, the surface temperature of the flare tip and wind shield may increase, which in the long run time will cause the destruction of the surface of the flare tip and its total replacement.

5. REFERENCES

[1] R. F. Huang and J. M. Chang, "The stability and visualized flame and flow structures of a combusting jet in cross flow," Combust. Flame, vol. 98, no. 3, pp. 267–278, Aug. 1994.

[2] E. Bourguignon, M. R. Johnson, and L. W. Kostiuk, "The use of a closed-loop wind tunnel for measuring the combustion efficiency of flames in a cross flow," Combust. Flame, vol. 119, no. 3, pp. 319–334, Nov. 1999.

[3] M. R. JOHNSON, D. J. WILSON, and L. W. KOSTIUK, "A FUEL STRIPPING MECHANISM FOR WAKE-STABILIZED JET DIFFUSION FLAMES IN CROSSFLOW," Combust. Sci. Technol., vol. 169, no. 1, pp. 155–174, Aug. 2001.

[4] L. Kostiuk, M. Johnson, and G. Thomas, "University of Alberta Flare Research Project Final Report November 1996–September 2004", 2004.

[5] D. Castiñeira and T. F. Edgar, "CFD for Simulation of Steam-Assisted and Air-Assisted Flare Combustion Systems," Energy & Fuels, vol. 20, no. 3, pp. 1044–1056, May 2006.