Int. J. Advanced Design and Manufacturing Technology, 2024, Vol. 17, No. 3, pp. 25-30

DOI: 10.30486/ADMT.2024.873807 *ISSN: 2252-0406* *https://admt.isfahan.iau.ir*

Upgrading Existing CRU Heaters for Energy Saving by Adding Convection Section

Abbas Behzadi *
Esfahan Oil Refining Co.Esafahan
PhD candidate, Department of Mechanical Engineering,
University of Kashan, Kashan
E-mail: Behzadi@eorc.ir
*Corresponding author

Aliakbar Abbassianarani, Ali Arefmanesh
Department of Mechanical Engineering,
University of Kashan, Kashan
E-mail: Abbasian@kashanu.ac.ir

Received: 12 January 2024, Revised: 9 March 2024, Accepted: 25 August 2024

Abstract: Industrial heaters play a pivotal role in various processes in industries and consume significant energy resources. Increasing the energy efficiency of these heaters is of great importance in the current energy-saving scenario. This study improves existing heaters by introducing an additional convective section, using the power of computational fluid dynamics to optimize heat transfer and improve overall efficiency. This paper presents a comprehensive study on upgrading existing heaters by incorporating an additional convection section, facilitated by a thermal and computational study. The main focus is on increasing the efficiency of conventional heaters by optimizing the heat transfer process through the newly added convection section. This paper describes the methodology used to analyze, model, and simulate the reinforced system, presents the results generated by CFD, and discusses the implications of the findings on the overall efficiency of the heaters. In this regard, to study and design the optimal mode of energy saving and complying with the conditions of optimal operational performance, various plans are examined and the best method is selected for heater optimization.

Keywords: Adding Convection Section, Computational Fluid Dynamics, Efficiency Improvement, Upgrading of Existing Heaters

Biographical notes: Abbas Behzadi received his MSc in Mechanical Engineering in 2000 from Amirkabir University of Technology, Tehran, Iran. He has 24 years' experience in the oil industry, Esfahan Refining Co., and has different local certificates regarding mechanical and oil issues. He is a PhD candidate from the University of Kashan and his current research interest includes energy saving and boiler & heater optimization and retrofit. **Aliakbar Abbasianiarani & Ali Arefmanesh,** accordingly, are full Professor & Associate of Mechanical engineering at the University of Kashan, Iran. Their current research focuses on fluid mechanics, thermodynamics and heat transfer.

Research paper

1 INTRODUCTION

Due to the very high temperature of the exhaust gases from the stack of two sets of heaters in the catalytic reformer unit of Isfahan Refinery, including eight cylindrical heaters, the present research was carried out with the aim of optimizing the heaters to recover energy from flue gases and reduce the temperature of the exhaust gases. In the current situation, these gases enter the atmosphere with a temperature of about 1000 °C, so it is possible to optimize the existing heaters by using valid standards and related software. In this research, taking into account the operational conditions of the heaters and in order to increase energy efficiency and reduce environmental pollutants, various options have been investigated, and a convection section is selected and added to retrofit and optimize the existing heaters. By using these studies, the gases from combustion can be used to preheat boiler water (from 117°C to 240°C). With the aim of heat recovery, this issue will increase the efficiency of heaters, save energy, and reduce environmental pollutants.

2 PROCESS DESCRIPTION

The basic design of the research has been such that the combustion gases from four heaters 251 to 254 enter the common duct and exit the common stack with a temperature of about 1000 degrees Celsius, which wastes a lot of energy. The current research aims to use the energy of these gases in order to heat the high-pressure feed water of boilers from the temperature of 117°C to 240°C and reduce the temperature of the exhaust gas from the stack to below 200°C, ("Fig. 1").

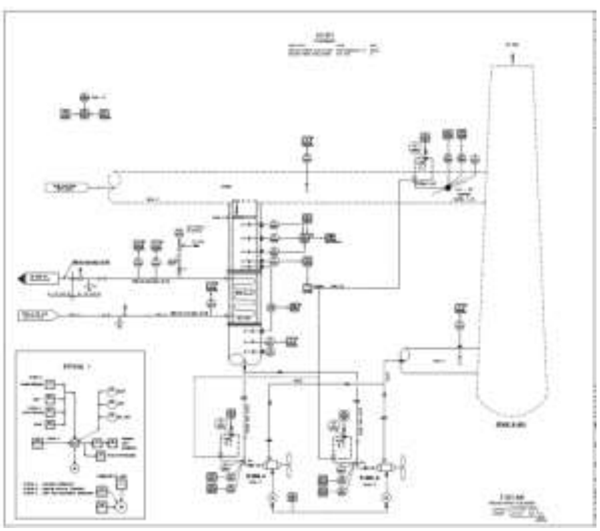

Fig. 1 Process design schematic [1].

The heaters of the catalytic conversion unit produce the feed of light fuel production units such as gasoline, and for this reason, they are important, and their failure will affect the production of light fuel in the country. In this regard, it is very important to observe the pressure of the outlet duct of the heaters to maintain the functioning of the heaters and also to maintain the flame conditions of the burners. In order to maintain the efficiency of heaters, the pressure of 10-25 mm of water in the exhaust gas duct of the heaters should be observed. In this regard, the design of the new convective part should be done considering these conditions.

3 REMOVAL OF THE DAMPER FROM THE COMMON DUCT

The use of a guillotine or damper on the common duct and Inlet guide vane at the inlet of the suction fans to adjust the required suction of the heaters is commonly used in the design of such systems, which is Acceptance of the common design standards, but considering malfunction of burners, the unfavorable experiences and the bad effect on the performance of the heaters in the use of existing dampers for high temperatures, the removal of the damper from the common duct has been investigated and suitable solutions are presented to ensure that the heaters run properly safe.

The operating conditions of the existing heaters are simulated in "Fig. 2" by considering the natural suction of air in the burners. As can be seen, the natural suction of the stack and the Stack effect are created as a result of the temperature gradient and the volumetric mass difference and are applied through the stack length.

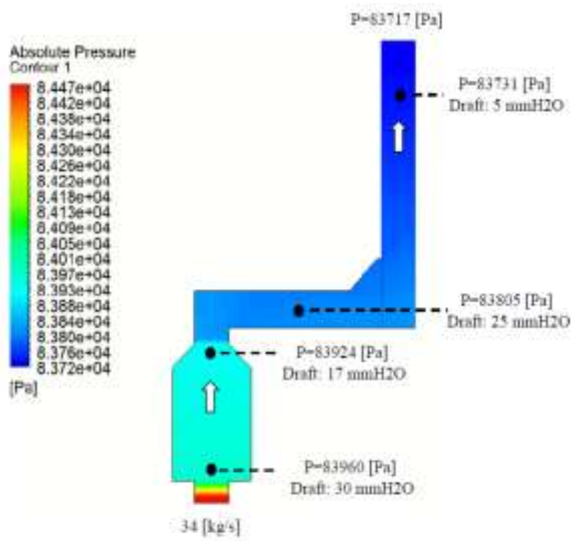

Fig. 2 Existing heaters natural draft.

4 OPTIMUM DESIGN FOR CONVECTION SECTION

Process data for the thermal calculation of a new convection section usually includes various parameters and inputs that are necessary to design an effective and efficient heat transfer system. Operating conditions include the desired temperature range, hot and cold fluid flow rates, and specific application or process requirements. Some of the key process data elements involved in the thermal calculation for a new convection section are as follows, ("Table 1").

Table 1 Process design data

Description	Value		UOM
Flue Gas Flow Rate	34.4 (124000)		Kg/s (kg/hr)
Flue Gas Temperature	900		°C
Flue Gas Density	0.35		Kg/m^3
Flue Gas Pressure in Common Duct	-10~-25		mmH2O
Flue Gas Composition	N2	71.4	Mole%
	H2O	16.7	
	O2	2.49	
	Ar	0.89	
	CO2	8.52	
BFW Inlet Temperature	*117*		°C
BFW Outlet Temperature	240		°C
BFW Inlet Pressure	60		barg
Number of Fans	2 x 50%		-

Considering the 45-year operation of the heaters, as well as the dependence of gasoline production on their performance and the lack of sufficient space, the design of the convection section has faced various problems. In this context, according to the available limited space, the design of the economizer should be done in such a way that the maximum heat exchange between the combustion gases and the feed water of the boilers takes place at the same time, and also the pressure drop should be such that the suction required by the burners is provided and there should be a negative effect on the functioning of the heaters. After reviewing the basic data documents and specifications of the heaters and taking into account the installation time of the equipment and the problems of running out of service for the heaters, in this research, various plans have been examined and studied. The establishment of the convection section on top of the existing common duct and the placement of fans on top of it and the use of a separate short stack on top of the fan to discharge the combustion gases into the atmosphere are considered the best design considering all the technical and operational aspects, ("Fig. 3").

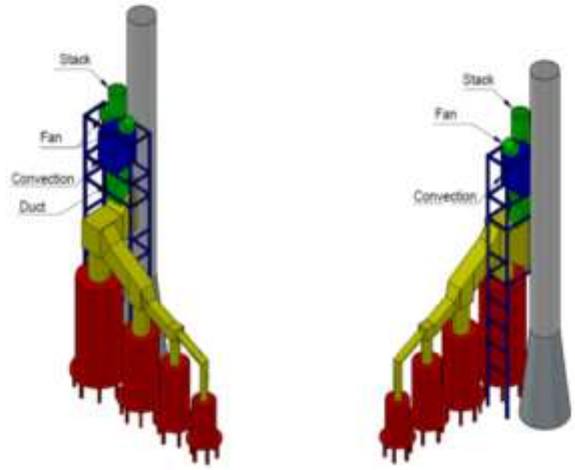

Fig. 3 Predominant schematic plan.

In this design, the gases from combustion are directed into the economizer before being connected to the stack. Then, in order to compensate for the pressure drop in the convection section, these gases enter two fans with 50% of the total capacity and are directed to the atmosphere through a separate stack.

Continuity, momentum, and energy Equations can be expressed in the form of conservative Equations for the general variable ϕ in the form of the following Equations [2].

$$div(\rho\phi\vec{v}) = div(\Gamma\nabla\phi) + S_\phi \quad (1)$$

These Equations can be as Equation (2) for each control volume or cell with a two-dimensional triangular cell in the computational domain [2].

$$\int_S n.(\rho\phi\vec{v})\, dA = \int_S n.(\Gamma\nabla\phi)\, dA + \int_{CV} S_\phi dV \quad (2)$$

In the simulation of flows that undergo separation and eddy formation, it is important to ensure an accurate simulation of the fluid field, to ensure the appropriate mesh size near the walls, respectively the layers around the walls. The height of the first grid cell of the wall is calculated using the following relations [3-4].

$$Re_x = \frac{\rho\, V\, l}{\mu} \quad (3)$$

$$C_f = \frac{0.026}{Re_x^{\frac{1}{7}}} \quad (4)$$

$$\tau_{wall} = \frac{1}{2} C_f\, \rho\, V^2 \quad (5)$$

$$u^* = \sqrt{\frac{\tau_{wall}}{\rho}} \quad (6)$$

$$H_{1st} = \frac{y^* \mu}{u^* \rho} \quad (7)$$

To calculate the height of the first wall from the grid, operating conditions at 70% load are considered and by placing the values, the height of the first cell will be equal to 0.5 mm for y*=5. In order to better detect the formation of eddies due to sudden expansion from the side of the channel, we use the K-Omega SST turbulence model in the Fluent software, which can properly cover y* less than 5 [5].

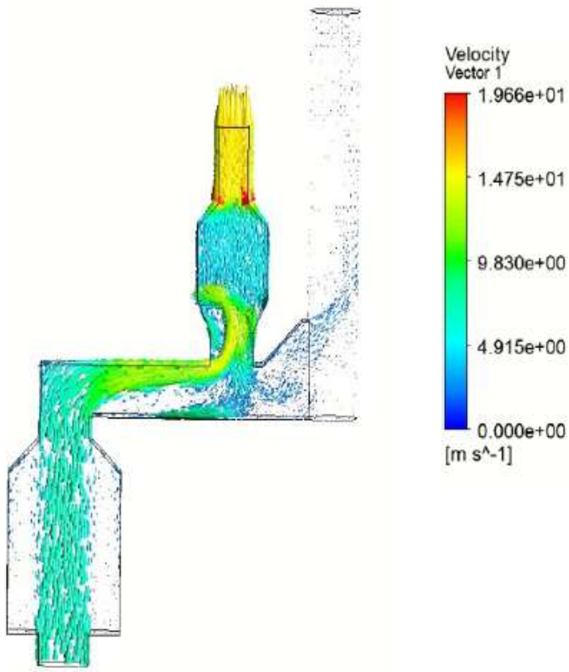

Fig. 4 Fan suction fluid speed at 37 mm of water.

After the investigations, it was determined that the following three situations would occur in this plan:

1- The suction pressure of the fan at the point of branching should be lower than the natural suction pressure of the existing stack due to the stack phenomenon: in this case, a significant percentage of the gas flow from combustion is directed to the atmosphere without passing through the convection section and exchanging heat with the water entering the boilers, which will lead to not achieving the goals of the plan.

2- The suction pressure of the fan at the branching point should be equal to the natural suction pressure of the existing stack due to the stack phenomenon: in this case, about 50% of the gas flow from combustion is directed to the atmosphere without passing through the convection section and exchanging heat with the feed water of the boilers. Therefore, 50% of the combustion gases will reach a temperature of 200°C after passing through the economizer, and the remaining 50% that will enter the stack directly will not have a noticeable temperature change.

3- The suction pressure of the fan at the branching point should be higher than the natural suction pressure of the existing stack due to the stack phenomenon: in this case, the fan suction can be adjusted in such a way that a significant percentage of the flow of combustion gases passes through the convection and exchange section. The heat is directed to the atmosphere with the feed water of the boilers, which will lead to the achievement of the goals of the project. The important point in this plan is to find the optimal performance range of the fan so that the minimum flow through the main stack is achieved simultaneously with the stability of the flames. In "Figs. 4 and 5", the results of fluid pressure and velocity analysis in the fan suction mode equal to 37 mm of water are presented.

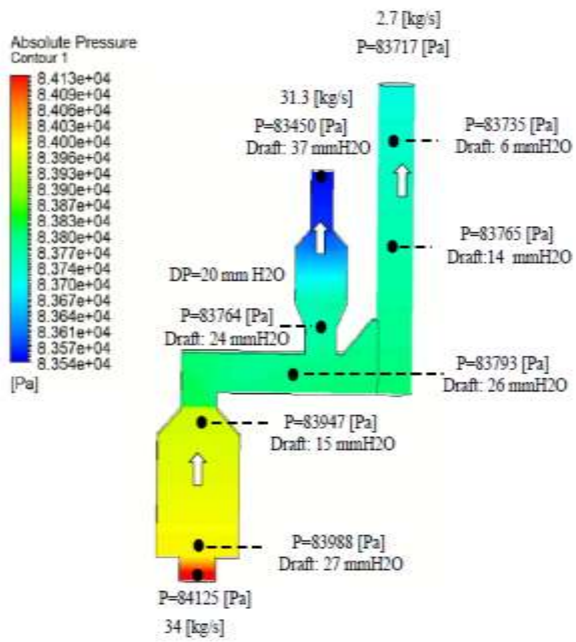

Fig. 5 Fan suction fluid pressure at 37 mm of water.

The speed and fluid pressure conditions in the suction of the fan equal to 32 mm of water are as follows, ("Fig. 6"). As can be seen, reducing the fan suction by 5 mm of water will lead to a reduction in the flow through the convection by 7 kg/s, which will only reduce the efficiency of the convection system and will not change the operation conditions of the heaters. In other words, weak suction by the fan causes some of the combustion gas to escape from the main stack, and strong suction leads air to enter the convection, and the risk in heater operation is minimal. In this regard, the CFD studies conducted show that about 8% of the exhaust gas escapes from the stack path (2.7 kg/s), which can be reduced to less than 5% with manual control. Considering the obstacles and limitations of the existing space and geometry, this design will have the least changes on the existing infrastructure and will have the most use for diversity in the use of the convective part.

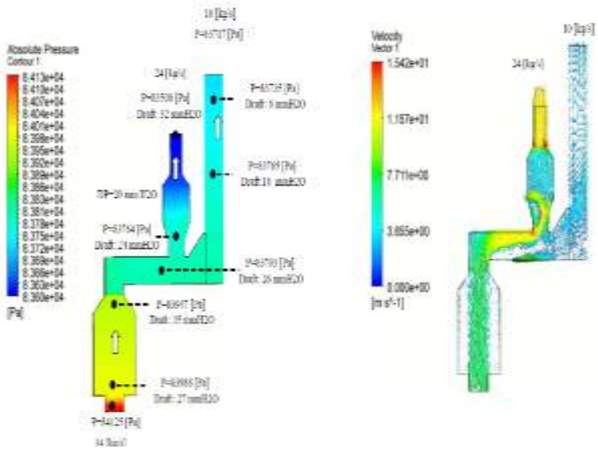

Fig. 6 Fluid pressure and velocity conditions in the fan suction at 32 mm of water.

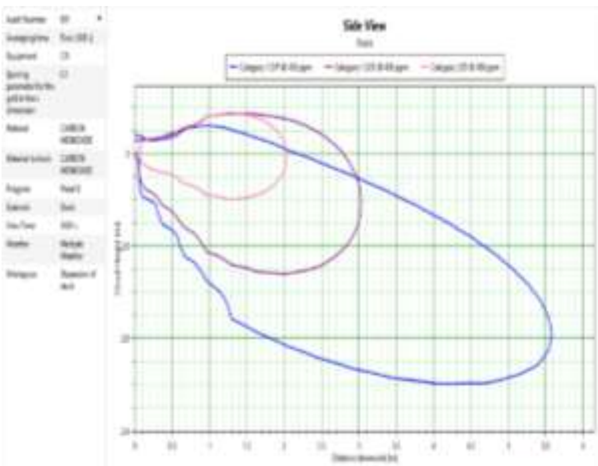

Fig. 7 The distribution of carbon monoxide gas 436 ppm, equivalent to 500 mg/Nm3 [6].

5 DISPERSION STUDY

In this research, due to the importance of the spread of polluting and environmentally damaging gases, the studies and modeling of the distribution of gases coming out of the new stack have been studied. As it is clear from these results, none of the toxic compounds of the outgoing gas stack reach the operating range of the access platforms and according to the height of the stack, it is diluted in the space above the stack.

In this work, the limit of emission of air pollutants in refinery and petrochemical industries has been considered according to the Clean Air Law. The first-class standard is applied to new units (maximum 15 years) and units whose establishment is under the rules and regulations for the establishment of production, industrial, and mineral units, the subject of notification No. 26869/100/97 dated 7/9/1397 of the Environmental Protection Organization to be biocontradictory. In this research, the studies have been done based on the first-class unit, and the height of the stack above the height of the equipment installation has been considered to be 3 meters. Figures 7 and 8 show the distribution of carbon monoxide and sulfur dioxide gases in the conditions of this research.

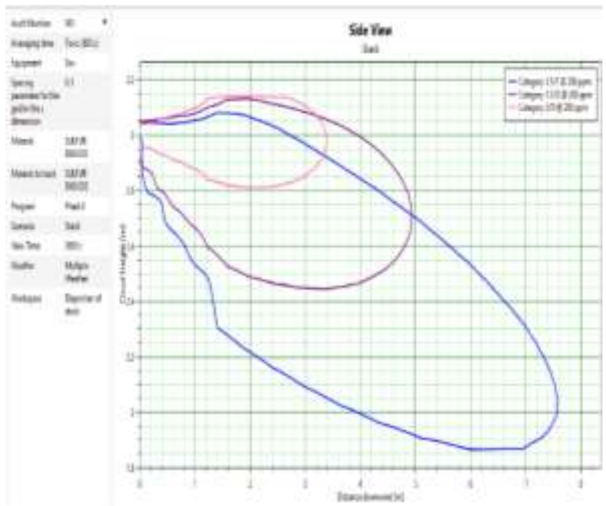

Fig. 8 Dispersion of sulfur dioxide gas 288 ppm equivalent to 752 mg/Nm3 [6].

Table 2 Convection section technical data

TUBE SPECIFICATION		
Design pressure	barg	72
Design temperature	°C	324
No. of tubes	-	360
No. of rows of tubes (banks 1,2&3)	-	4/3/8
No. of flow passes	-	4
Effective tube length	mm	4000
Tube OD	mm	114.3
Average tube wall thickness	mm	8.56
Tube material specification	-	ASTM A106 Gr B
Fin height	mm	15/21
Fin thickness	mm	1.5

6 MECHANICAL DESIGN SPECIFICATION

Designing a convective section for heaters to save energy requires careful consideration of various factors to maximize heat transfer efficiency and minimize heat loss. The technical data used in the construction of the convection section, including thermal conductivity, corrosion resistance, and specific heat, are necessary to evaluate the heat transfer characteristics. Information on pressure drop limitations, which can affect fluid flow and system efficiency, is considered during the calculation [7], (Table 2).

7 CONCLUSION

In this research, the design of the convection cross-section has been done with the help of thermal calculations and computational dynamic modeling. The project continues with the construction and installation of practical equipment in the operational unit, and its results are examined and confirmed in practice with the performance test. By equipping the common duct of the heaters to the convection section, important achievements including the following in the field of improving the heater performance, reduction of fuel consumption and optimal energy consumption, and reduction of environmental pollution will be achieved:

- Increasing thermal efficiency by at least ten percent for each set of heaters
- Reduction of gas fuel consumption by 90 thousand tons per year for heaters about 15 percent of fuel gas reduction for heaters
- Savings of 8.4 million dollars from the cost of supplying fuel gas
- Reducing smoke emissions to the environment by 3.3 million tons for one year for all heaters
- Reducing carbon dioxide emissions to the environment by 500,000 tons for one year for all heaters.
- Payout time to return the investment of the project for five months

ACKNOWLEDGMENTS

The author is grateful to the respected managers and colleagues of Isfahan Oil Refining Holding for providing facilities and technical assistance.

REFERENCES

[1] CRU Project Documentation- P& ID Drawings, 2023.

[2] Abbasian Arani, A. A., Arefmanesh, A., and Ehteram, H., Numerical Optimization of Obstructed High Temperature Heat Exchanger for Recovery from The Flue Gases by Considering Ash Fouling Characteristics, International Journal of Numerical Methods for Heat & Fluid Flow, Vol. 30, No. 5, 2020, pp. 2273-2303.

[3] Jagad, P. I., Puranik, B. P., and Date, A. W., A Finite Volume Procedure for Fluid Flow, Heat Transfer and Solid-Body Stress Analysis, International Journal for Computational Methods in Engineering Science and Mechanics, Vol. 19, No. 3, 2018, pp. 171-184.

[4] Velten, K., Lubitz, W., and Hopf, A., Simulation of Airflow Within Horticulture High-Tunnel Greenhouses Using Open-Source CFD Software (Doctoral Dissertation, PhD Thesis, 02, 2018.

[5] Lacombe, F., Pelletier, D., and Garon, A., Compatible Wall Functions and Adaptive Remeshing for the K-Omega SST Model, In AIAA Scitech 2019 Forum, 2019, pp. 2329.

[6] CRU Project HAZOP and SILL Study, 2023.

[7] ASME PTC 4, Fired Steam Generators Performance Test Codes, 2014, https://pdfcoffee.com/asme-ptc-4-2013-pdf-free.html.