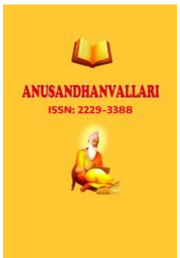


Anusandhanvallari
Vol 2024, No.1
February 2024
ISSN 2229-3388

Thermal–Fluid Optimization of Stack Architecture for Enhanced Efficiency and Emission Control in Incinerator-Fired Heaters

Sangewar Ravi Kumar
Professor, Affiliation Address: Mandsaur University, Mandsaur, Madhya Pradesh, India
Email: sangewarravikumar@gmail.com

Abstract

In this paper, a novel integration of an incinerator and a fired heater system is proposed to enhance overall heat recovery and emission performance in oil and gas field operations. The hot flue gases from the incinerator are directed beneath the convection section of the fired heater to maximize heat utilization. Effective mixing between the incinerator and heater flue gases at the convection section inlet is critical to achieving uniform temperature distribution and optimal thermal efficiency. Additionally, the stack design must simultaneously satisfy the contrasting draught requirements of the incinerator, which operates under forced draught, and the fired heater, which relies on natural draught. Computational Fluid Dynamics (CFD) simulations were employed to evaluate the flow characteristics, temperature uniformity, and draught performance of the integrated stack system. The initial CFD analysis identified design limitations, prompting geometric modifications to improve gas mixing and flow stability. The optimized configuration demonstrated satisfactory thermal and flow characteristics, ensuring both efficient heat recovery and emission control. Implementing these improvements at the design stage prevented potential operational challenges and significant cost and schedule impacts during plant commissioning.

Keywords: Stack Design Optimization, Heat Recovery CFD analysis, Emission Control, Forced and Natural Draught, Stack Design, Thermal Efficiency, Degree of Mixing, Heat Recovery, Turbulence, Oil & Gas Application

NOMENCLATURE.

ε	Turbulence dissipation rate
h	Enthalpy
κ	Turbulence kinetic energy
λ	Fluid thermal conductivity coefficient
m	Mass flow rate
μ	Molecular viscosity
μ_{eff}	Effective viscosity accounting for turbulence
P	Pressure
P_k	Turbulent kinetic energy production due to viscous forces
ρ	Fluid density

S_M	Source momentum
S_E	Source energy
T	Absolute temperature
ui	Velocity component in i –direction
x_i	Indicates i –direction

Introduction

Fired heaters are used for process fluid heating in Refinery, Petrochemical & Fertiliser units. Fired heater consists of radiant and convection section wherein heating coils are placed for heat recovery from hot flue gases. Hot flue gases are generated from the firing in burners that are placed in radiant section. Cold flue gases from convection section are routed to atmosphere via stack placed usually at top of convection section. In natural draught fired heaters, draught for flue gases flow is created by stack alone through chimney effect. Incinerator is widely used to dispose of various types of chemical and refinery wastes. Waste gases are burned in a chamber using burner. Usually high temperature more than 950°C is maintained in the chamber to destroy/incinerate harmful chemical components. Large amount of high temperature gases generated during Incineration are usually let out to atmosphere in sites where no water available for steam generation. In a unique and first of its kind of energy optimization, hot flue gases of Incinerator have been routed below convection section of Fired heater for enhanced heat recovery in an ongoing Oil & Gas field development projects. Hot flue gases of Incinerator are routed through a 'Transfer Duct' connecting to Fired Heater. Combined flue gases from Incinerator & Fired Heater after heat recovery are routed to a common stack placed at top of convection section of the Fired Heater. Bypass duct has been provided on or heater not in operation. Butterfly type dampers are placed in Transfer Duct & Bypass Duct to regulate quantity of flow between Heater & Incinerator. Hot oil Therminol 66 is heated in the Fired Heater from 138°C to 165°C. Heat in the hot oil is rejected in Amine Reboiler in other area of the Plant. The complete hot oil Heater – Incinerator system has been shown in Figure 1.

Table 1: SALIENT DESIGN FEATURES FIRED HEATER

Sr no.	Parameters	Design values
1.	Common stack inside diameter, m	1.8
2.	Stack tip inside diameter, m	0.9
3.	Transfer duct inside diameter, m	1.43
4.	Bypass duct inside diameter, m	1.43
5.	Stack height from the grade, m	46.5
6.	Incinerator exhaust flue gas temperature, °C	982
7.	Fired heater radiant section exit flue gas temperature, °C	805

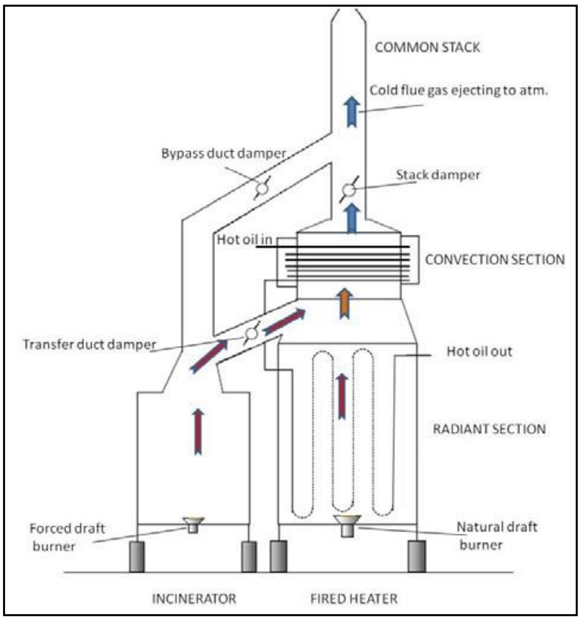
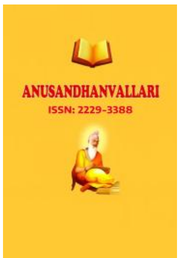


Figure 1: INTEGRATED INCINERATOR – FIRED HEATER SYSTEM

Forced draught fans are used to draw ambient air for combustion in the Incinerator. Usually as per vendor design practice, Incinerator operates under positive pressure however technically there is no strict requirement for Incinerator to operate under positive or negative pressure. Fired Heater furnace section is always designed to operate under slight negative pressure (-2.5 MMWC (g)) as per API 560 guidelines, [1]. Design of stack is very critical in this system as it is controlling pressure in Incinerator and desired negative pressure in the Heater furnace. The height and diameter of the stack are selected such that required pressure draught of (-ve) 2.5 MMWC (g) is maintained at the heater arch section. Heat transfer areas (tube surfaces) are calculated during design of Fired heater considering uniform average temperature that can only be achieved during ideal mixing of the two streams. Hence proper mixing of hot flue gases from Incinerator at 980°C and from heater at 805°C are required below heat recovery coils for optimum heat transfer and safety of tube material. Salient design feature of the system is shown in Table 1. Neither supplier of Incinerator nor supplier of Fired Heater had any experience of such kind of integration of Heater & Incinerator. Hence it is very much necessary to carry of design verification of such system.

Literature Search And Design Verification Methodology

As mentioned above design verification of stack diameter & height and degree mixing of flue gases below convection section are required to get design confidence of the proposed system. There are correlations available for calculation of gas draught in the duct/stack of Fired heater [1] & related systems [2]. However, combination of Incinerator (high exhaust temperature) and Fired Heater through a common convection & stack has not been mentioned anywhere. No literature could be found on guidelines of flue gas velocity to be used for proper mixing of Incinerator and Heater flue gases. Velocity misdistribution & Temperature stratification is created below heat transfer coil which affects the degree of heat transfer in the system. Also, due to 3-D geometry, it was decided to estimate velocity & temperature profile after mixing by modeling of flow & heat transfer in the existing domain using computational technique. Prediction of pressure draught and flow mixing in the system can be very well done using suitable flow analysis software/technique.



Since past references of such integrated system were not available, Computation Fluid Dynamics (CFD) technique has been utilized for verification of the design.

Cfd Modeling Approach

Objective of the CFD simulation is to find out flue gas velocity and temperature profile at inlet of convection section tube bundles and to verify adequacy of stack design to maintain required draught at heater arch. 3-dimensional domain has been considered for the analysis. Combustion in Incinerator and Fired heater are not modeled. Hot Flue gases generated after complete combustion in Incinerator& Fired Heater are taken as input for CFD analysis. CFD governing equations, assumptions, meshing & boundary condition details are described below.

Governing Equations

The governing conservation equations of fluid flow represent mathematical statements of the conservation laws of mass (continuity), momentum and energy in steady states as given below.

Continuity equation:

$$\frac{\partial}{\partial x_j}(\rho u_j) = 0 \qquad (1)$$

Momentum equation:

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + S_M \qquad (2)$$

Energy equation:

$$\frac{\partial}{\partial x_j}(\rho u_i h) = \frac{\partial}{\partial x_j}\left(\lambda \frac{\partial T}{\partial x_j}\right) - U_j \frac{\partial P}{\partial x_j} + U_i \frac{\partial \tau_y}{\partial x_j} + S_E \qquad (3)$$

$$\mu_{eff} = \mu + \mu_l \qquad (4)$$

Where SM & SE are momentum source and energy source respectively. μ is molecular viscosity & μt is turbulent viscosity.

In this study, the k–ε model was used as it has been widely applied in practical parametric studies with reasonable accuracy. The turbulence kinetic energy k (and its rate of dissipation ε is defined in the following transport equations:

$$\frac{\partial}{\partial x_j}(\rho \kappa u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_l}{\sigma_\varepsilon}\right)\frac{\partial \aleph}{\partial x_j}\right] + P_x - \rho\varepsilon \qquad (5)$$

$$\frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_l}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{\varepsilon}{k}\left(C_{El}P_k - C_{g2}\rho\varepsilon\right) \qquad (6)$$

$$\mu_l = \rho C_\mu \frac{k^2}{\varepsilon} \qquad (7)$$

$$P_x = \mu_l\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\left(3\mu_l \frac{\partial u_k}{\partial x_k} + \rho k\right) \qquad (8)$$

Above equations has been solved using ANSYS CFX software. Default turbulent model coefficients available in ANSYS CFX [3] are used in all calculations; i.e., Cε1 = 1.44, Cε2 = 1.92, Cμ = 0.09, σk =1.0, and σe= 1.3.

Geometry & Mesh Details

CFD model has been built as shown in Figure 2. The model was meshed with around 2.24 million mixed cells (majority hexahedral cells). Mesh density correlates well with the size of the model which will provide good results.

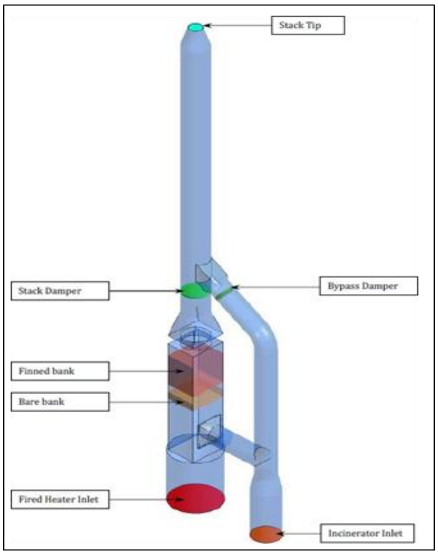


Figure 2: CFD MODEL OF INCINERATOR-HEATER SYSTEM

Boundary Conditions

Table 2 show boundary conditions considered for analysis. Boundary conditions for both case (case 1 & case 12) are shown in the table. Bypass in Case 12-HF is more than 10%, as there is excess heat available in Incinerator flue gases than required in convection section of Fired Heater. Multi- component mixture has been specified to model Incinerator & Heater exit flue gases which comprises of H2O, O2, N2, CO2 gases. Radiation heat transfer has been invoked to take care of high temperature flue gases heat transfer to tubes & refractory surfaces.

Table 2: BOUNDARY CONDITIONS

Sr. No.	**Description**	**Design Case 1-HMB**	**Design Case 12-HF**
1.	Domain Reference Pressure	97.3 kPa	97.3 kPa
2.	Fired Heater Inlet	m = 2.15215 kg/s T = 805 ^{0}C	m= 1.216742 kg/s T = 653 ^{0}C
3.	Incinerator Inlet	m= 3.6387743 kg/s T = 982 ^{0}C	m = 6.678536 kg/s T = 982 ^{0}C
.4	% Flow bypass in Bypass duct	10.0 %	13.2 %
5.	Domain Walls	Thin, smooth, adiabatic walls with no slip	Thin, smooth, adiabatic walls with no slip

Analysis Results

Operating cases Case 1-HMB & Case 12-HF have been solved using ANSYS CFX software. First, the operating cases have been analyzed without any changes in the design (Base Design). Looking at the results of Base design, certain modification in the design were carried out which have been presented below in Modified Design Result.

Base Design Results

For base case simulation, geometry & design parameters were taken same as shown in Table-1. Streamline plot of velocity & temperature are shown in Figure 3. Plots of stack tip portion have been not shown as it is not important for analysis of flue gas mixing which is happening below convection section of the heater. One can see flue gas temperature variation from 982°C to 270.6°C which shows that heat transfer phenomenon through tube bundles have been modeled properly. Velocity profile at inlet of convection section (seen from bottom) has been shown in Figure 4. One can see that more mass of flue gases are getting accumulated at opposite end of 'Transfer duct'. Bare tube bank has a function of flow correction device and very beneficial at correcting flow mal-distribution at this location because of which the flow is more uniform at the entrance of fin tube bundle. One can see that uniform velocity profile in Figure 5. Similar plots have been obtained for Case 12-HF also which are not shown here. It was seen that results are not much different for both the cases.

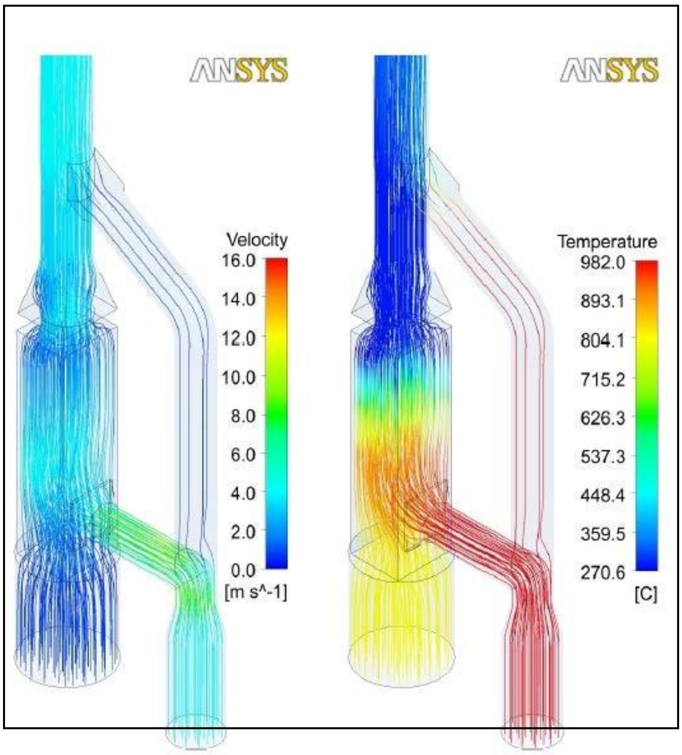


Figure 3: STREAMLINE PLOTS OF VELOCITY & TEMPERATURE FOR BASE DESIGN, CASE 1-HMB

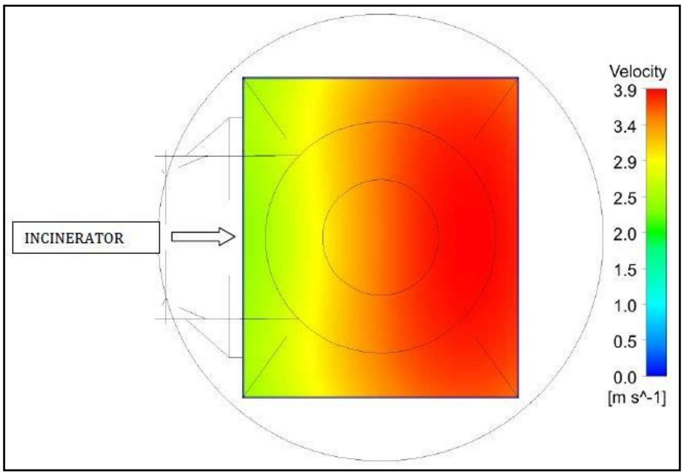


Figure 4: VELOCITY PROFILE AT BARE TUBE BANK ENTRANCE FOR BASE DESIGN, CASE 1-HMB

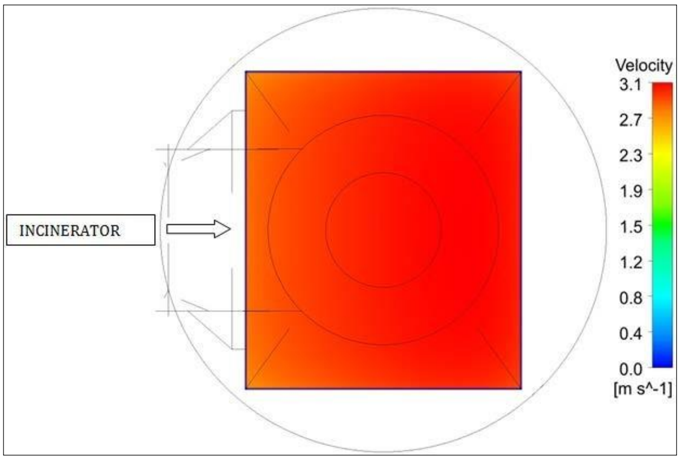


Figure 5: VELOCITY PROFILE AT FIN TUBE BANK ENTRANCE FOR BASE DESIGN, CASE 1-HMB

The velocity profile at entrance of convection section for both cases is shown in Table 3. RMS deviation of velocity up to 15% at entrance of heat exchangers is generally acceptable [4]. Additionally, one can see from Table 3 that velocity misdistribution reduces drastically at entrance to first row of finned tube. In bare tube bank also velocity deviation reduces after first row of tube. Hence one can see that the deviation of velocity after mixing of Incinerator & Heater flue gases is within acceptable limit. Also, one can see that the deviations are not much different in Case 12-HF from Case 1-HMB. Hence increase in flow is not causing much difference in the velocity deviation; in fact it is helping due to high turbulence. Due to relatively smooth profile at convection inlet, the temperature distribution after mixing is expected to be uniform as can be seen in Figure 6. The temperature variation shown in Figure 6 is within ±28°C over its average value of 907°C. Temperature variation further reduces and the temperature profile becomes more uniform at fin tube bank inlet shown in Figure 7. The temperature variation shown in Figure 7 is within ±4°C over its average value of 771°C.

Highest flue gas flow case (Case 12-HF) is governing case for pressure draught study as frictional forces are high in this case due to high velocity. Pressure and temperature profile in Case 12-HF has been shown in Figure 8. Buoyancy effects are dependent on temperature. High temperature causes higher buoyancy and more draught in the system. It has been seen in Case 12-HF that the increase in frictional pressure drops is much more than the gain in pressure due to buoyancy. The pressure at arch of Fired heater as estimated in Case 12-HF is 11.8MMWC (g) (116 Pa) which is much higher than desired value of -2.5 MMWC (g).

Table 3: VELOCITY PROFILE AT INLET OF CONVECTION SECTION

Location	**Mean velocity, m/s**		**% RMS deviation of Mean velocity**	
	Case 1-HMB	Case 12-HF	Case 1-HMB	Case 12-HF
At entry to Bare tube bank	3.32	4.53	15.6	15
At entry to 2nd row of Bare tube bank	3.18	4.32	12.6	11.2

At entry to Finned tube bank	2.97	4.06	2.8	2.3

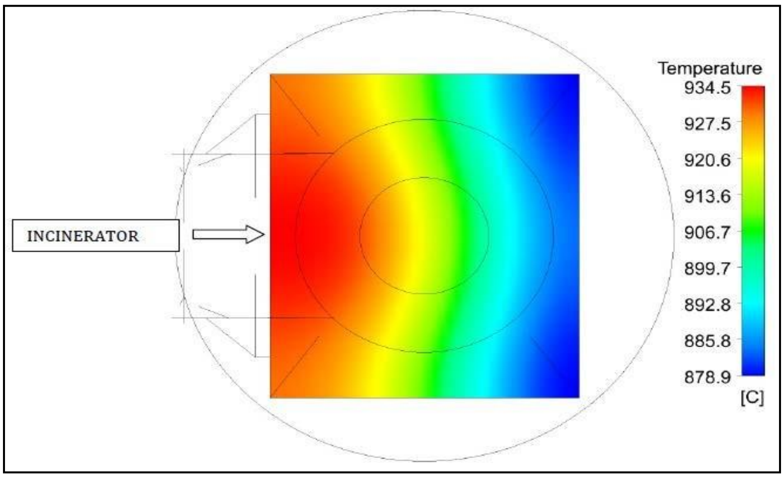


Figure 6: TEMPERATURE PROFILE AFTER MIXING BELOW CONVECTION SECTION FOR BASE DESIGN, CASE 1-HMB

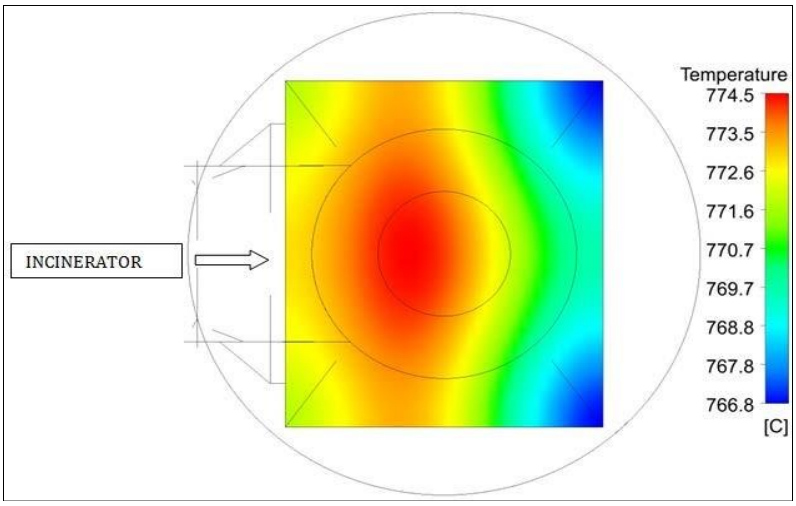


Figure 7: TEMPERATURE PROFILE AT FIN BANK ENTRANCE FOR BASE DESIGN, CASE 1-HMB

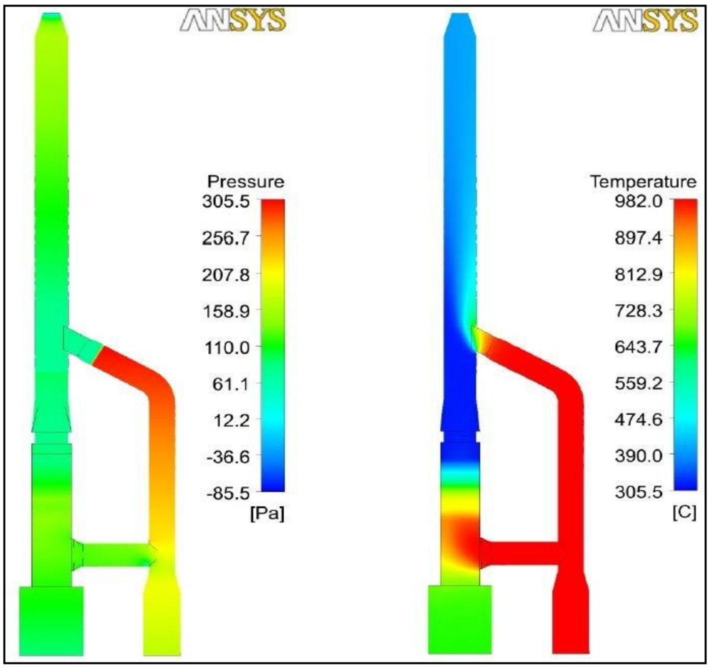


Figure 8: PRESSURE (DRAUGHT) & TEMPERATURE PROFILE IN THE HEATER-INCINERATOR FOR BASE DESIGN, CASE 12-HF

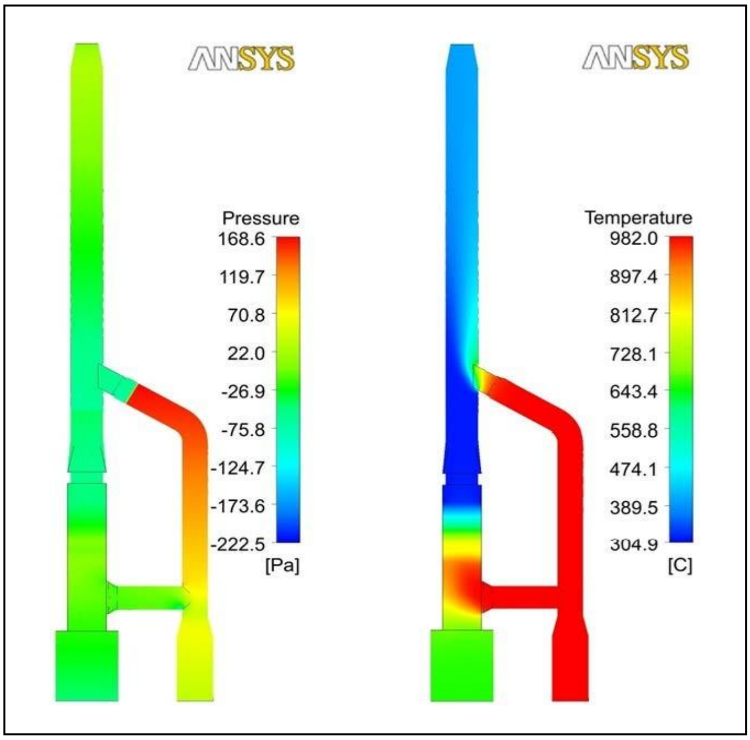


Figure 9: PRESSURE (DRAUGHT) & TEMPERATURE PROFILE IN THE HEATER-INCINERATOR SYSTEM FOR MODIFIED DESIGN, CASE 12-HF

Modified Design Results

Detailed analysis of the results revealed that there is a huge pressure drop at stack exit because of high exit velocity. One of the options of increasing draught in the system is by increasing height of the stack. This option was ruled out due to huge changes in the structural design which would increase cost of equipment. Other option of reducing pressure drop is to increase stack tip diameter which will reduce stack exit velocity & hence reduce exit losses. Based on past data, it was decided to target stack exit velocity to around 12 m/s. Hence stack tip diameter was increased to 1270 mm. Rest of all design parameters remained same as shown in Table 1. Pressure & temperature profile in the system for modified dimensions have been shown in Figure 9. One can see the pressure drop at stack tip has come down drastically.

Estimated pressure at Heater arch now is -2.13 MMWC (g) (21 Pa) which is very close to required value. Stack exit velocity has drop down to a reasonable value 12.4 m/s. Pressure at Incinerator arch section is +6.3 MMWC(g) (62 Pa). Positive pressure in the Incinerator is desired as per design practice followed by Incinerator vendor. Stack tip diameter change has not caused any significant change in the velocity & temperature distribution at entrance to convection section of the heater and hence related plots and tables for Case 1-HMB are not shown for modified design.

Conclusion

Integrated Fired Heater-Incinerator system has been modeled using CFD technique. Mixing of hot flue gases from Incinerator and Fired heater and their resultant flue gas misdistribution below tube banks have been studied. The misdistribution of flue gases below convection section of Fired Heater is found to be within acceptable range. Flue gases flow through common stack has also been studied using CFD. It was found that existing stack design is not appropriate to take care of frictional pressure drop in the system. Stack tip diameter has been proposed to be increased from 900 mm to 1270 mm to take care of pressure draught requirements in the system. CFD analysis of modified design shows acceptable pressure draught in the system.

References

[1] ANSI/API Standard 560, *Fired Heaters for General Refinery Service*, 4th ed., August 2007.
[2] V. Ganapathy, *Industrial Boilers and Heat Recovery Steam Generators*, New York: Marcel Dekker, 2003.
[3] ANSYS CFX, *ANSYS CFX Version 14.0 User's Guide*.
[4] API Recommended Practice 534, *Heat Recovery Steam Generators*, 2nd ed., February 2007.
[5] M. J. Bagajewicz, "New robust approach for the globally optimal design of fired heaters," *Chemical Engineering Research and Design*, vol. 197, pp. 434–448, 2023.
[6] A. Garg, "CFD simulations to improve the performance of fired heaters," 2023. [Online]. Available: https://ashutosh-garg.squarespace.com/s/FIS-CFDSimulationCasestudySept2023.
[7] B. Faqihi and F. A. Ghaith, "A novel design of heat recovery system from exhaust stacks' silencers in a simple cycle gas turbine," *International Communications in Heat and Mass Transfer*, vol. 148, 2023.
[8] H. Zhang, "CFD-DEM simulations of municipal solid waste gasification in a pilot demonstration facility," *Science of The Total Environment*, vol. 876, p. 162745, 2023.
[9] M. Wang, "CFD–DEM simulation of heat transfer and reaction characteristics in municipal solid waste pyrolysis," *Processes*, vol. 12, no. 2, p. 390, 2024.
[10] M. J. Bagajewicz, "Standards for fired heater design: An assessment based on computational modelling," *Chemical Engineering Research and Design*, vol. 197, pp. 434–448, 2023.
[11] H. Sarakikya, "Design and computational fluid dynamics modeling for a municipal solid waste incineration process," *Open Journal of Fluid Dynamics*, vol. 11, no. 4, pp. 177–191, 2021.

[12] M. Tutar, "Optimized CFD modelling and validation of radiation section of an industrial top-fired steam methane reforming furnace," *arXiv preprint arXiv:2501.08660*, 2025.
[13] P. Schwarzmayr, "Packed bed thermal energy storage for waste heat recovery in the iron and steel industry: An experimental study on powder hold-up and pressure drop," *arXiv preprint arXiv:2307.01585*, 2023.
[14] A. Islas, "Biomass dust explosions: CFD simulations and venting experiments in a 1 m^3 silo," *arXiv preprint arXiv:2307.04717*, 2023.