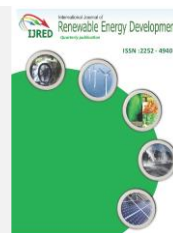




Contents list available at CBIORE journal website

International Journal of Renewable Energy Development

Journal homepage: <https://ijred.cbiore.id>



Research Article

Optimization and management of flare gases through modification of knock-out drum HP flares by 4R approach based on 3E structures

Ali Ahmadzadeh¹, Alireza Noorpoor^{*1}, Gholamreza Nabi Bidhendi¹

Faculty of Environment, School of Engineering, University of Tehran, Tehran- Iran

Abstract. The goal of this study is the Optimization and Management of Flare Gases through the Modification of Knock-out Drum HP Flares. The optimization of the K.O.D. is to create a shell around it and inject water steam into the shell, so that a uniform temperature distribution has done inside the drum, so freezing does not occur, and liquid that drops inside the burner, does not burn. The result of the simulations showed that in the drainage part of the drum, humidity associated with inlet gas freezes upon entering it after pressure and temperature drop suddenly. In the drainage part of the drum and the entrance of water steam with a temperature of 438 K and relative pressure of 550,000 Pa, the freezing of the coating part of it is eliminated. Finally, the water steam with liquid water caused by the heat transfer between the steam, and the bottoms of the drum is out from its drainage part. In the following, two issues were examined; First, simulating the drum to prove the insufficient power of the electric heater at the entrance of the drum. Second, simulating the drum with its surrounding cover in order to eliminate possible freezing. As the result, this work simulated and optimized the K.O.D. flare system to reduce valuable and toxic gas which burned in the flare system and caused environmental, economic, and social effects. This modelling optimized 8 points to add optimum heat flux and used a water steam jacket to prevent the formation of a freezing zone. The optimum zone around the bottom of K.O.D. steam injected this zone and observed no ice formation occurred in this zone. The steam jacket creates uniform heating by using this design and steam injection to the outer wall of the drum. For many reasons, the implementation of this project will reduce smoke and flare pollution: Inhibition of freezing in the liquid outlet of the K.O.D., the liquid level inside the drum remains constant and prevents the transfer of liquid droplets associated with the exhaust gas to the flare.

Keywords: Simulation, CFD, Optimization, Knock out Drum, Steam Jacket, Ethylene Glycol



@ The author(s). Published by CBIORE. This is an open access article under the CC BY-SA license (<http://creativecommons.org/licenses/by-sa/4.0/>).

Received: 17th July 2023; Revised: 6th Oct 2023; Accepted: 2nd Dec 2023; Available online: 22nd Dec 2023

1. Introduction

World population growth increased greenhouse gas emissions (Rezakazemi *et al.*, 2017; Younas *et al.*, 2020). Fossil fuel combustion is generally caused global warming (Ahmadipouya *et al.*, 2021; Monjezi *et al.*, 2021; Pazani *et al.*, 2022). CO₂ emissions can be reduced by capturing and storing greenhouse gases (Sohaib *et al.*, 2020; Swati *et al.*, 2020). Chemical reduction of CO₂, such as catalytic hydrogenation, is an effective method to reduce its concentration in the atmosphere (Arabi Shamsabadi *et al.*, 2021; Rajabloo *et al.*, 2022). Preventing of the burning unwanted gas with the implementation of a no-flare design will significantly reduce the emissions. It is expected that new environmental policies that increase public awareness not be allowed gas flaring to cause changes in refinery equipment and processing (Bjorndalen *et al.*, 2005). The flare system is commonly used by chemical processing facilities, to burn unwanted gas generated by the refining process (Hajilary *et al.*, 2020). Essential equipment in degassing stations in the oil production field is Knock-out drum (K.O.D.) separators (Ali *et al.*, 2020). Figure 1 shows K.O.D. (red line) that the gases from the other process units pass through the K.O.D. to remove liquid drops from vapors that best efficiently burn vapors in the flare tips.

This fuel gas consumed in refinery processes like methane and hydrogen, specifically methane, is used to produce other valuable products. (Rahimpour and Alizadehesari, 2008). Burners with low NO_x must separate fuel gas from its impurities of liquid droplets that cause burner trouble (Platvoet and Baukal, 2013; Sazhin *et al.*, 2006). Feed into a K.O.D. is the common equipment to remove this liquid droplet from the fuel gas sent to the flare stack (Jekel *et al.*, 2001). Due to high maintenance costs and damage of flare burners using K.O.D. to remove droplets from feed (Quadro and Guarieiro, 2016; Zadakbar *et al.*, 2008). Computational Fluid Dynamics (CFD) is currently an essential interdisciplinary tool (Mavriplis, 2012). CFD applied a set of Navier-Stokes equations to scale down industrial equipment to experimental for reduced experimental cost and implementation to real projects. (Patankar, 2018; Vedovoto *et al.*, 2015).

Reaching a large amount of harmful flaring gases into the atmosphere will cause economic costs and energy loss. For optimization of the flaring system, a steam jacket is used to prevent ice formation in K.O.D., because it makes the obstacle for valuable liquid out from the top of drum and it is the key role in the economic industrial applications. The steam jet system is new environmentally friendly technology that reduces flare emissions and the loss of salable liquid petroleum products to the stack system. New vapor units are used for low-temperature waste heat for obstacle freezing in a K.O.D. system. Eventually,

* Corresponding author
Email : noorpoor@ut.ac.ir (A. Noorpoor)

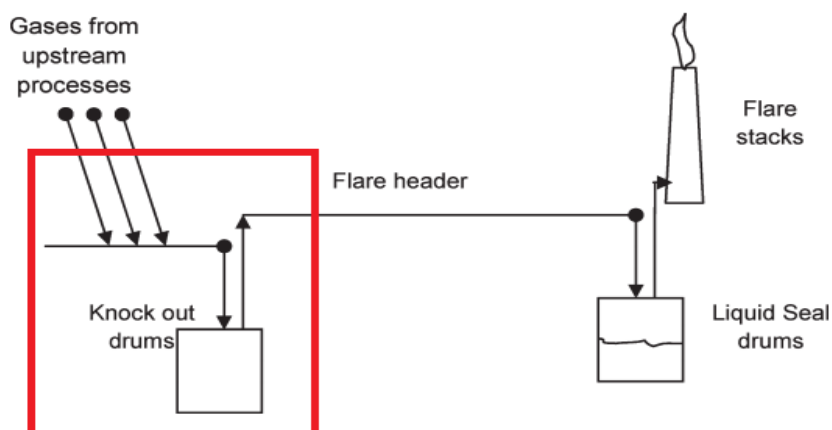


Fig 1 Basic scheme of a flare process system (Yadav et al., 2012).

bottom freezing should be avoided resulting in a too-high-pressure drop and undesired plant shutdown. In this case, a well-designed steam jacket could save the economics and energy for refinery companies. To prevent poor performance of K.O.D. two scenarios were studied in this paper. Firstly, modeling the steam injection at eight points in K.O.D. and secondly, using Ethylene Glycol (EG) in the drainage to prevent ice formation.

The purpose of this study is to find the appropriate and optimal implementation method to control the gases coming out of the burners, monitor and predict the sustainable energy production system based on the intelligent management of existing resources, more profitability, and prevent environmental damage with an exact look at the requirements of feed supply security and sustainable development in the study area.

2. Simulation and Modeling

This section describes the model geometry and simulation setup used in the present study. The modeling process consists of three steps: optimization of composition out of the drum, CFD modeling of drum represented ice formation in its bottom and CFD modeling of the drum with steam jacket

2.1. Process Simulation Using ASPEN PLUS

In this study, exact calculations were made using the NRTL thermodynamic equation, which is suitable for polar environments in the presence of H_2O and H_2S , executed in Aspen PLUS., with a wide range of temperature, pressure, applicability, and its extensive binary coefficients database. Adiabatic vessels are considered for modeling the system (Quattordio et al., 2019).

For the simulation of a unit, operational data is used, which includes temperature, pressure, flow rate, and percentage composition of materials in the process. Checking the temperature of other units that have an output towards the flare unit obtained a temperature of 298 K. To find accurate information for inlet composition, K.O.D. used the ASPEN PLUS for simulation, relied on information obtained from the articles,

and selected the information closer to the real conditions. Calibration by the engineering unit and portable equipment was used to obtain an inlet flow rate of 5.8 kg/s, which was used for simulation. To achieve the composition of the output from the drum, simulations perform through ASPEN PLUS.

2.2. CFD Simulation Using ANSYS- Fluent

The steps for the numerical simulation process are; discretizing control volumes, discretizing the governing equations for each individual grid, linearizing discretized equations, and solving the equation system iteratively, where iteration consists of updating fluid properties for each cell, updating solved momentum equations to the velocity field, solving the other variables such as turbulence using the previously updated values, checking the convergence of the equation set (Castiñeira Areas, 2006), and considering the velocities obtained in Step b) the pressure correction equation is solved to obtain continuity is satisfied. Fluid properties for each cell are updated.

These steps continued until the converge criteria are met. Then, simulation results should be analyzed and validated with practical data. ANSYS FLUENT was used in this study, and the commercial CFD package and the geometry were created with DESIGN MODELER. Using a Pressure-based solver with double precision and the standard k- ϵ model for turbulence. The pressure-velocity coupling used Pressure-Implicit with Splitting of Operators (PISO) algorithm (Patankar, 2018). Discretization of gradients for cell faces was computed using the Green-Gauss Cell-based method. The pressure discretization is used pressure staggering option (PRESTO) (Singh, 2013).

2.3. ANSYS- Fluent

ANSYS-Fluent is a powerful computational fluid dynamics (CFD) software that is widely used for simulating and modeling fluid flow and heat transfer in various industries, including automotive, aerospace, chemical, and energy. The simulation process begins with pre-processing, where the model geometry is imported, meshed, and prepared for the simulation. This step also involves defining the fluid properties, boundary

Table 1

The criteria for quality of mesh K.O.D.

	Number of Mesh	Skewness _{avg}	Aspect Ratio _{max}	Orthogonal Quality _{max}
Without steam jacket	395143	0.20	10	0.91
With steam jacket	846977	0.21	10	0.99

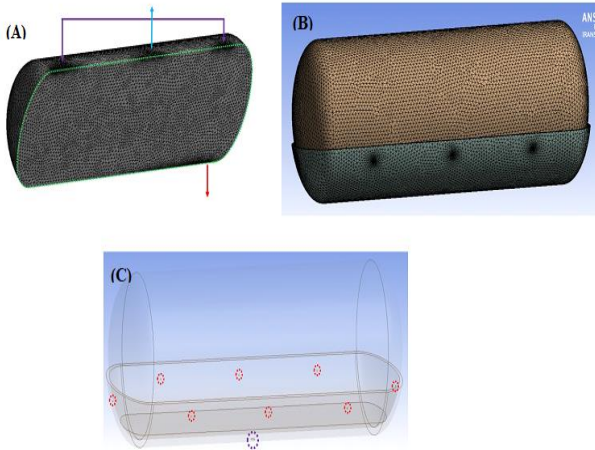


Fig 2 Schematic representation of meshing of K.O.D. A) Without the steam jacket. B) With the steam jacket. C) Cover the lower part of the drum and the location of the inlets (red) and the steam outlet (purple).

conditions, and solver settings. Next step is to solve the equations. ANSYS-Fluent solves the governing fluid flow and heat transfer equations numerically using various solution methods, such as the finite volume method. Once the equations are solved, the simulation is run to obtain the desired results. After the simulation run is complete, post-processing is performed to analyze and visualize the results. It offers a wide range of tools and techniques to plot and interpret data, including velocity vectors, pressure contours, and temperature. ANSYS-Fluent parameters are: Geometry (shape, size, and arrangement of the components), Boundary conditions (inlet velocities, pressures, temperatures, and wall conditions such as no-slip or applied heat flux), Physical properties (fluid density, viscosity, and thermal conductivity), Meshing (structured, unstructured, and hybrid meshing methods), and Solver settings to control the accuracy and convergence of the simulation.

ANSYS Fluent sophisticated CFD software packages predict multiphase behavior by solving momentum equations. The K.O.D. is a three-dimensional (3D) transient CFD model a feed of 5.8 kg/s a diameter of 5.2 m and a length of 14.2 m designed. A graphical design of the K.O.D. is shown in Figure 2a. The boundary conditions are zero velocity at the wall, an internal pressure of 550000 Pa, with atmospheric outlet pressure. Using practical and economic analysis is the aspect ratio of a K.O.D. determined in a reasonable range. In this study, the K.O.D. aspect ratio was selected as 3 for pressure 0-1700000 Pa, according to Walas for multiphase separators (Laleh *et al.*, 2012).

In the first stage, an attempt is made to simulate the multiphase flow of the input fluid mixture and the temperature drop due to the pressure drop resulting from the expansion of the input fluid to the drum, according to its actual conditions. In the second step, according to the decrease in temperature observed in the previous simulation, first, the freezing of humidity with the incoming fluid mixture and finally, applying heat flux to the wall of the drum to eliminate the freezing should be simulated. The heat flux required to eliminate is obtained through trial and error.

The input boundary condition is used for the Initialize part, and the Residual Monitors part is considered the default. Drum geometry meshing without and with steam jacket built by Design Modeler software with simplification is shown in Figure 2. The quality criteria for this meshing with and without a steam jacket

are given in Table 1, which has an acceptable quality for modeling. For eliminated the freezing of the coating part of the drum, to simplify its simulation has been assumed that there is only air and ice in it, and freezing is located in the lower part of the drum. To simulate the operational conditions at the boundary between air and freezing, a negative temperature flux is used, which shows the amount of temperature drop that the gas mixture gives to the associated humidity due to the pressure drop. The cover of 8 different points in the lower part of the drum injected water vapor with a temperature of 438 K, which is eliminated by the transfer of heat between the freezing in the lower part of it and the steam. The thickness and material of the drum shell are considered according to the operating conditions. Figure 2C shows a schematic of the cover in the lower part of the drum and the location of the inlets and the steam outlet. The cover's diameter compared to the drum's diameter has been increased by 0.5 m. The diameter of the steam inlets is 0.05 m and the outlet diameter is 0.2 m. The input flow rate is considered 1.4 kg/s, and the output boundary condition is the output pressure.

3. Numerical Simulation of Turbulence and Multiphase Flow

In this section, the basic equations used for the numerical simulation of turbulence are highlighted to better understand the dynamics of turbulence and multiphase flow. Meanwhile, the principles and techniques used in the numerical analysis of the PISO solver used in the modeling (Afolabi, 2012) are described.

3.1. Navier Stokes Governing Equations

In turbulent flow with the incompressible, isothermal Newtonian flow with a velocity field $\vec{V} = (u_r, u_\theta, u_z)$, only the continuity and momentum equations need to be considered. The continuity equation for an incompressible fluid in cylindrical coordinates, (r, θ, z) is (Tu *et al.*, 2018):

$$\frac{1}{r} \frac{\partial}{\partial r}(ru_r) + \frac{1}{r} \frac{\partial}{\partial \theta}(u_\theta) + \frac{\partial}{\partial z}(u_z) = 0 \quad (1)$$

Equations of motion at constant dynamic viscosity, μ , and density, ρ , for an incompressible fluid, are:

$$\begin{aligned} \rho \left[\frac{Du_r}{Dt} - \frac{u_\theta^2}{r} \right] &= -\frac{\partial p}{\partial r} + f_r + \mu \left[\nabla^2 u_r - \frac{u_r}{r^2} - \frac{2}{r^2} \frac{\partial u_\theta}{\partial \theta} \right] \\ \rho \left[\frac{Du_\theta}{Dt} - \frac{u_\theta u_r}{r} \right] &= -\frac{1}{r} \frac{\partial p}{\partial \theta} + f_\theta \\ &+ \mu \left[\nabla^2 u_\theta - \frac{u_\theta}{r^2} - \frac{2}{r^2} \frac{\partial u_r}{\partial \theta} \right] \\ \rho \frac{Du_z}{Dt} &= -\frac{\partial p}{\partial z} + f_z + \mu \nabla^2 u_z \end{aligned} \quad (2)$$

Where p is the pressure, f_r, f_θ, f_z , are the body force components, and for an incompressible, Newtonian fluid are:

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + u_r \frac{\partial}{\partial r} + \frac{u_\theta}{r} \frac{\partial}{\partial \theta} + u_z \frac{\partial}{\partial z} \quad (3)$$

$$\nabla^2 = \frac{\partial^2}{\partial r^2} + \frac{1}{r} \frac{\partial}{\partial r} + \frac{1}{r^2} \frac{\partial^2}{\partial \theta^2} + \frac{\partial^2}{\partial z^2}$$

$$\sigma_{rr} = -p + 2\mu \frac{\partial u_r}{\partial r} \quad (4)$$

Table 2
Compositions of materials out from K.O.D.

Component	Feed flow rate (kg/s)	Vapor outlet flow rate (kg/s)
H ₂ O	0.00599313	0.00599313
N ₂	0.2073170	0.2073170
CO ₂	0.2573387	0.2573387
H ₂ S	0.0629695	0.0629695
C ₁	2.823731	2.823731
C ₂	1.214702	1.214702
C ₃	0.6876640	0.6876640
IC ₄	0.1363923	0.1363923
NC ₄	0.1967797	0.1967797
IC ₅	0.0738765	0.0738765
NC ₅	0.0529894	0.0529894
C ₆	0.0478543	0.0478543
C ₇	0.0279499	0.0279499
C ₈	0.0157850	0.0157850
C ₉	0.0219901	0.0219901

$$\begin{aligned}
 \sigma_{zz} &= -p + 2\mu \frac{\partial u_z}{\partial z} \\
 \sigma_{r\theta} &= \mu \left(\frac{1}{r} \frac{\partial u_r}{\partial \theta} + \frac{\partial u_\theta}{\partial r} - \frac{u_\theta}{r} \right) \\
 \sigma_{r\theta} &= \mu \left(\frac{1}{r} \frac{\partial u_z}{\partial \theta} + \frac{\partial u_\theta}{\partial z} \right) \\
 \sigma_{\theta\theta} &= -p + 2\mu \left(\frac{1}{r} \frac{\partial u_\theta}{\partial \theta} + \frac{u_r}{r} \right) \\
 \sigma_{r\theta} &= \mu \left(\frac{\partial u_r}{\partial z} + \frac{\partial u_z}{\partial r} \right)
 \end{aligned}
 \quad
 U_p = \frac{\int_A |\Psi - \Psi_{avg}| dA}{\int_A dA} \quad \Psi_{avg} = \frac{\int_A \Psi_{avg} dA}{\int_A dA} \quad (5)$$

Heat flux calculation refers to the measurement of the rate at which heat is transferred through a surface.

Heat Flux (q) = Rate of Heat Transfer (Q) / Surface Area (A)

$$Q = k * A * \Delta T / d$$

Where: Q is the rate of heat transfer, k the thermal conductivity of the material, A the surface area of heat transfer, ΔT the temperature difference between the hot and cold sides, d is the thickness of the material through which heat is being transferred.

In the analytical part of this study, the importance of ambient temperature fluctuations with changes in temperature boundary conditions and changes in feed composition, which can significantly affect the heat transfer process and properties such as specific heat capacity and thermal conductivity and the resulting heat flux, is considered.

4. Results and discussion

4.1. Analysis of data from ASPEN PLUS simulation

A refinery flare gas with a flow rate of 5.8 kg/s and composition of 0.04 %-mass of hydrogen sulfide, 2 %-mass of hydrogen dioxide and 97.96 mass. % of hydrocarbons were used in this study. Table 2 shows the compositions of materials out from K.O.D. Due to the lack of operational information about the output flows of the drum, it was not possible to compare the simulation results with them, so relying on the empirical knowledge of the industry managers, which included checking the amount of liquid flow and gas flow and composition and temperature out from K.O.D.

3.2. PISO Method

The PISO method is an improved version of the SIMPLE algorithm. To improve the efficiency of the required iterative method, three complementary corrections, momentum correction and neighbour correction, and skewness correction are applied by the PISO method. Issa (Issa, 1986) noted that the main idea of the PISO algorithm is to bring the repeated pressure correction equation. This iterative process is called "momentum correction" or "neighbour correction" and the updated velocities better satisfy the continuity and momentum equations, hence, rapidly moving to convergence (FLUENT, 2014). An internal iterative technique, similar to neighbour correction, is applied by the PISO method for highly skewed meshes to reduce the impact of cell skewness on the quality of the updated velocities (Ferziger and Peric, 1996).

For expression of non-uniformity of the flow in the software using Equation 5 which A is the surface area, U_p uniformity index, and Ψ parameters such as concentration, pressure, or temperature whose distribution is considered. If U_p tends to zero, the current distribution is uniform; if it tends to one, it is non-uniform. In the drum model, the value is equal to 0.64.

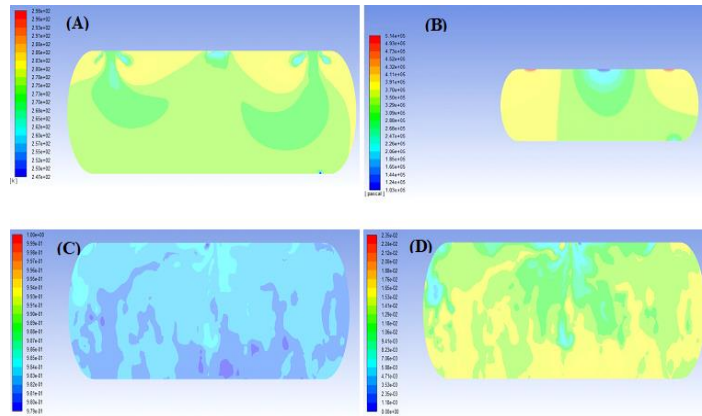


Fig 3. Schematic representation A) temperature (K), B) pressure (Pa), C) gas phase volume fraction, and D) liquid phase volume fraction inside of K.O.D.

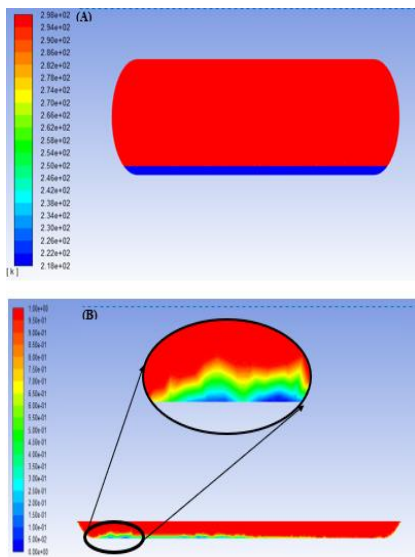


Fig 4 Schematic representation A) liquid fractions in the freezing zone; B) temperature (K) contour in the freezing model which -10000 w/m^2 heat flux is given to the bottom of drums.

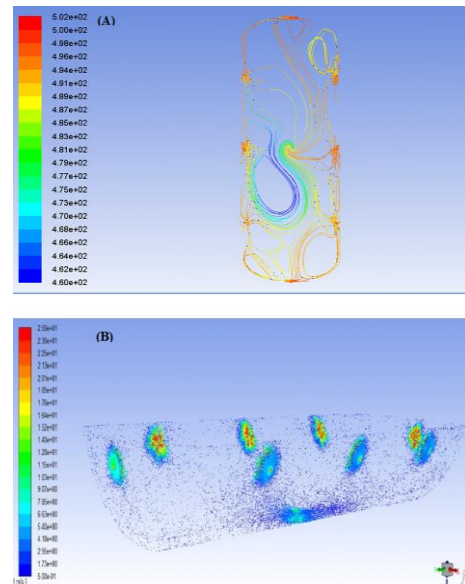


Fig 5. Schematic representation of water vapor A) flow path and temperature (K) (top view) and B) inlet and outlet velocity vectors.

4.2. Model 1-K.O.D.

In this model, the temperature reduction due to pressure drop is considered. This phenomenon is called the Joule-Thomson expansion phenomenon. This phenomenon examines how a gas's temperature drops during adiabatic expansion. The temperature drops as the gas enters and expands in the drum. Figure 3A shows the non-uniform temperature distribution.

Figure 3B shows the temperature and pressure contour, as it is clear that the drum input feed has a pressure of 550000 Pa and decreases early when it enters to it, and finally, it has atmospheric pressure at the exit. As a result of pressure reduction, the two phases of liquid and gas are separated from each other and the gas moves towards the upper outlet, and the liquid moves towards the lower outlet. Figure 3C shows snapshots at different times of contours of gas volume fraction, where the red and blue color indicated volume fraction of gas are 1 and 0 (i.e., the volume fraction of condensate is 1.0). The volume fraction of condensate at the gas outlet is 0 showing that the K.O.D. separated condensate droplets from gas entirely. Gas

funnel formation at the liquid outlet is also clearly shown in Figure 3D.

4.3. Model 2- proof freezing is located in the lower part of K.O.D.

As seen in Figure 4A, the liquid fraction reached zero in the lower part of the drum, and the water freezes in this part simultaneously. According to the previous step, when the temperature is reduced to 484 K, and according to the freezing temperature of the water, it can be concluded that the moisture along with the feed is frozen. In operational conditions, the heater heated the space inside the drum and prevented the temperature decrease to this degree, but the freezing of the lower part of the drum has been observed in the refinery. It can be concluded that the heat flux that the heater gives to the space inside the drum is not enough, and the temperature drop due to the feed pressure drop is more than the heat flux of the heater. In the simulation in Figure 4B, -10000 W/m^2 heat flux is given to the shell of drums to observe freezing.

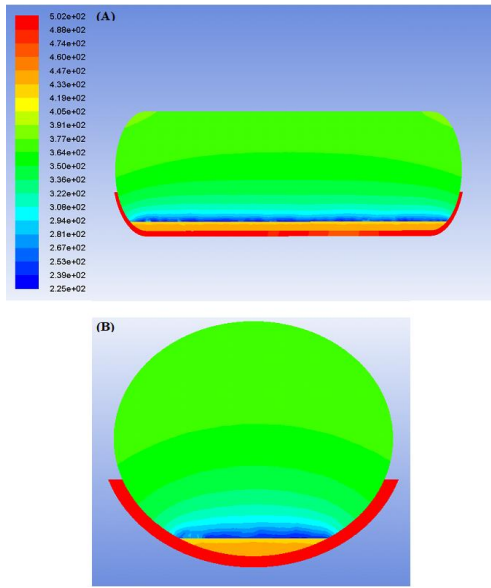


Fig 6 Schematic representation of temperature contour A) bottom of the drum and B) transverse view.

4.4. Model 3-K.O.D. with steam jacket

The track path of steam around the jacket shows in Figure 5A when steam enters through 8 points to the jacket, the path of steam flow and temperature. Figure 5B presents the input and output velocity vectors to the cover.

The temperature contour indicates that the heat transfer from the side of the steam cover causes the liquid inside the drum to heat up, and the virtual wall causes the temperature of the liquid inside the drum to decrease. The color of the contour represented different temperatures. Figures 6A and 6B show the

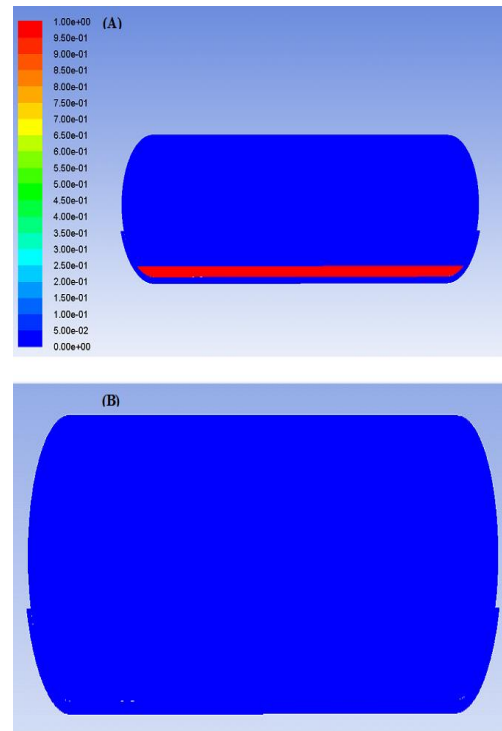


Fig 8 Schematic representation of A) liquid and B) solid fraction inside the drum.

temperature contour in the front and the transverse plane of the drum.

In Figures 7A and 7B, the contour of the water vapor and liquid phase in the cover is shown; due to the heat transfer between the water vapor and the liquid inside the drum, the temperature of the water vapor decreases and changes into liquid water. Figure 8 shows the location of the liquid phase and solid phase inside the drum to prevent ice formation. According to the simulation results, there is no freezing inside the drum, and it can be concluded that if a heater with the specifications duty into the designed cover, with the heat transfer between the steam and the fluid inside the drum, prevented freezing in the lower part of the drum.

4.5. Economic analysis

This section reports the saved capital costs after installing the steam jacket system Installation of the steam jacket for K.O.D. an d measurement of gas burn-off at tips of flare is the main cost. The main components burned out in the system are shown in Table 3; the higher cost is due to the burned-out valuable gases. The combustion of 122,707,785.6 m³ /year methane in HP (High Pressure) flare causes nearly 10,430,162\$/year economic cost (Table 3). This K.O.D. with steam jacket provides solutions for problems caused by flared gases in the petroleum industry: 1. Minimize environmental emissions in line with the green environment program instructions. 2. Recover valuable products such as LPG (Liquefied Petroleum Gas) from flared gases.

For economic savings in refineries using qualify studied to prevent the water from freezing in the drum by adding EG. If this goal is achieved, the amount of gases sent to the flare will be reduced. For the feasibility of this study, Figure 9 represents the EG injection unit where the liquid out of the flash drum has low H₂S. To do this, the required EG should be sent to the drum by creating a flow line from the EG storage tank (102-T-101 A/B)

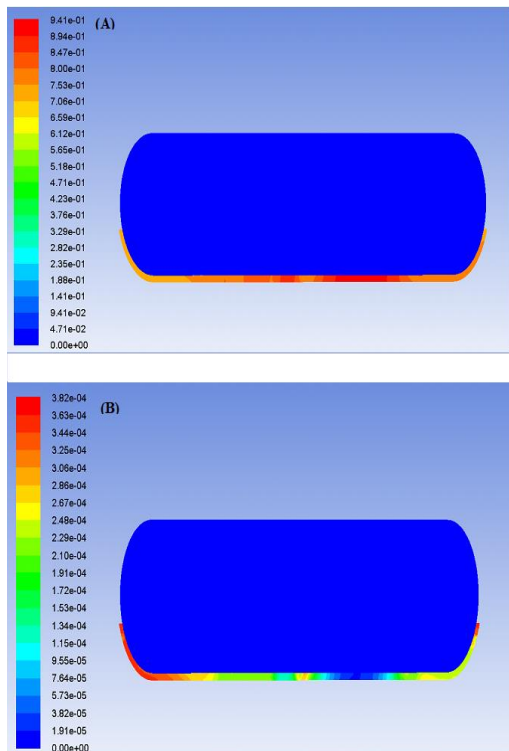


Fig 7 Schematic representation of A) vapor inside the cover and B) liquid inside the jacket.

Table 3
Volume and economic value of burning in HP flare.

	Daily	Monthly	Yearly
Amount of Methane (m ³)	340854.96	10225648.8	122707785.6
Value of Methane (\$)	28972.6716	869180.148	10430161.78
Amount of Ethane (m ³)	104949.576	3148487.28	37781847.36
Value of Ethane (\$)	25187.89824	755636.9472	9067643.366
Amount of LPG (L)	167558.5008	5026755.024	60321060.29
Value of LPG (\$)	103886.2705	3116588.115	37399057.38
Amount of Condensate (kg)	20422.56528	612676.9584	7352123.501
Value of Condensate (\$)	18090.30833	542709.2498	6512510.997
Total cost for 1 phase (\$)	176137.1487	5284114.46	63409373.52
Total cost for 23 phases (\$)	1734021.076	52020632.28	624247587.3

and then injected with the EG by a pump. Then that liquid can be sent to the off-spec condensate storage section and after the settling time and dewatering and then sent to the condensate stabilization section, the purification operation is performed again. and in addition to gas condensate (C₅⁺), it separates different hydrocarbons and is used as gas for petrochemical and downstream industries. For injection, EG in high H₂S amount changed the process to prevent freezing zone in a drum that in supplementary information described process flow diagram (PFD) and piping and instrument diagram (P&ID) for practical studied

5. Innovative aspects, and main research findings

The study focused on the optimization and management of flare gases using a 4R (Reduce, Reuse, Recycle, and Recover) procedure based on 3E (Energy, Economy, and Environment) structures. Specifically, the study looked into the simulation and optimization of the knock-out drum high-pressure flares.

The knock-out drum is an essential component of the flare system that removes liquid droplets and solid particles from the flared gases before they are released into the atmosphere. However, the traditional operation of high-pressure flares can result in significant energy waste and environmental pollution.

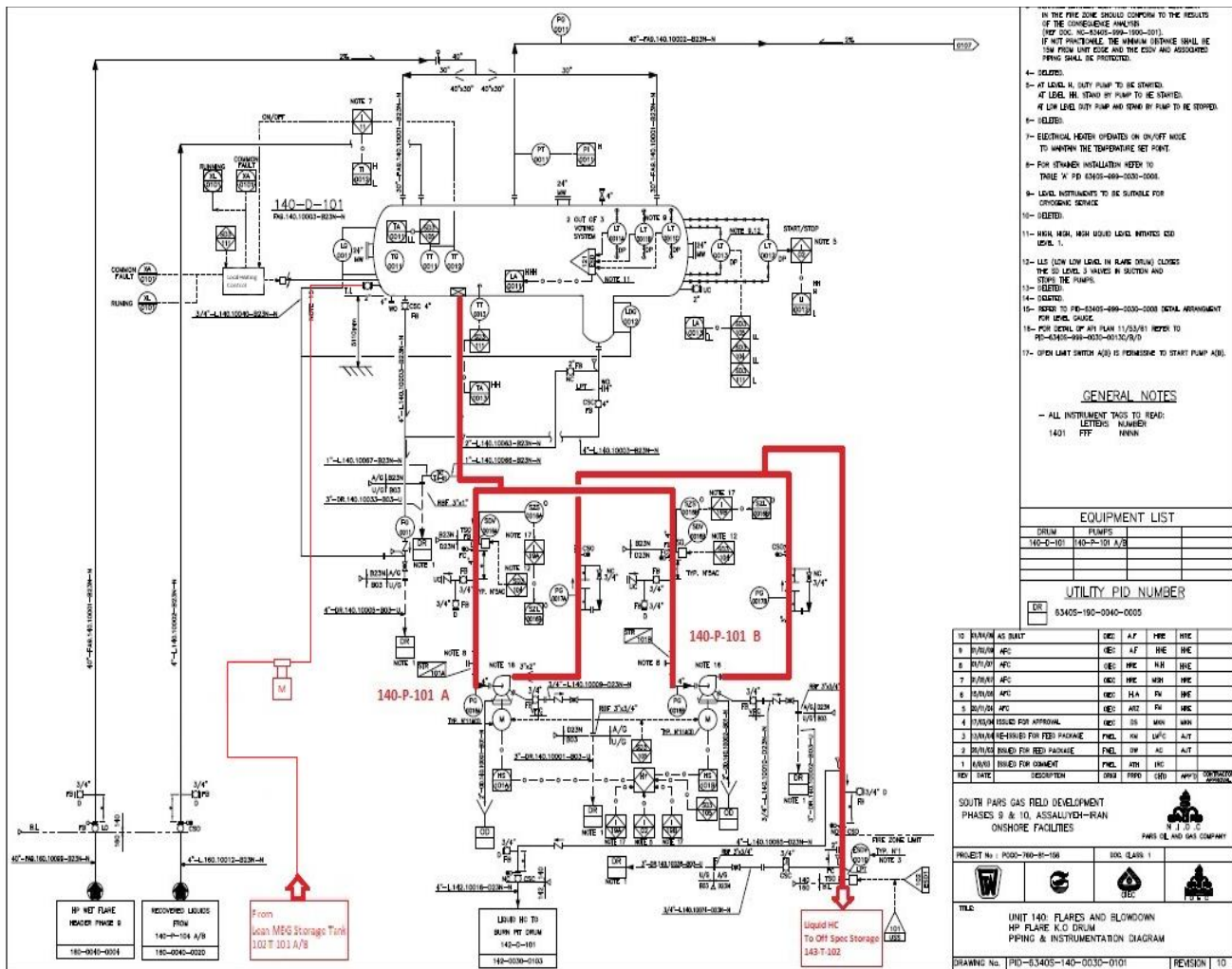


Fig 9. 140-D-101 drum flow and equipment position, which changes have been made for EG injection (Red).

To address these issues, the study proposed a 4R procedure that aimed to reduce the amount of flare gases generated, reuse the recovered gases for heating purposes, recycle the condensate formed during gas cooling, and recover the waste heat from the flare system for energy generation.

The optimized knock-out drum high-pressure flares showed higher energy efficiency, reduced emissions, and improved overall performance compared to traditional flare systems. Considering that freezing is caused by the drop in the temperature of the gas mixture that is given to the humidity, therefore, it was tried to replace the gas mixture with the help of negative heat flux, which causes the drop in the temperature of the humidity and leads to its freezing. So, two issues were investigated including: 1- Simulation of the drum to prove the insufficient power of the electric heater in the drum; In this simulation, the complete geometry of the heater has been designed and gridded with a detailed mechanical map according to the heater model used in the industry, and it was used to solve the problem by using simplifications in the drum model, 2- Simulating the drum and its surrounding cover to eliminate freezing; in this section, the heater is completely removed and the drum was designed according to the mechanical drawings received from the company (size, thickness, and material).

Therefore, based on the findings of the study the recommendations that are presented are; (1)- Increasing the efficiency of the gas purification process. Off gas and tail gas created from gas processing facilities are burned along with other additional gases by flare chimney and gas burner. Currently, there are several technologies available that have higher process performance and produce less process waste gas. The efficiency of the sulphur recovery unit (SRU), which recovers the sulphur contained in the H₂S acid gases (separated from the raw gas in the sweetening unit), is deeply dependent on the amount of tail gas with very high concentrations of H₂S. A low-efficiency or worn-out SRU unit can be one of the largest sources of H₂S emissions depending on operational conditions. Using a high-efficiency SRU unit or adding a tail gas purification unit in the process is an effective way to reduce tail gas generation, (2)-Using combustion equipment with high efficiency. This technology is especially effective for large-scale combustion equipment, such as steam boilers, gas turbines, thermal furnaces, etc. Reducing fuel consumption and protecting the environment can be achieved through the use of efficient combustion equipment. This method should be considered for the installation of new combustion equipment in the refinery, and (3)- Reduction of volatile gases (modification of existing equipment). Volatile gases that are unintentionally released from equipment and facilities or leak include a significant share of widespread leaks in operational units. Volatile gas mainly includes volatile organic compounds (VOC) as well as BTX (benzene, toluene, and xylene) present in raw materials, process fluids, and intermediate and final products of the unit. The source of the emission of these gases is process equipment, pipe connections, valves, pumps, compressors, storage tanks, product loading equipment, etc.

6. Conclusion

This work simulated and optimized the K.O.D. flare system to reduce valuable and toxic gas which burned in the flare system and caused environmental, economic, and social effects. From other units sent, flow to K.O.D. separated the gas and liquid phase so that it prevented entering liquid to the flare tip and caused partial combustion. When flow enters K.O.D., the temperature reduces, resulting in a pressure drop, and the bottom of K.O.D. is frozen. This simulation represented this

result. The contour of temperature represented to reduced 520 K and compared to the water freezing point; therefore, ice formation in the bottom of K.O.D. in this simulation used practical operation data added -1000 W/m² hear flux to observe the freezing zone. This modeling optimized 8 points to add optimum heat flux and used a water steam jacket to prevent the formation of a freezing zone. The optimum zone around the bottom of K.O.D. steam injected this zone and observed no ice formation occurred in this zone.

The steam jacket creates uniform heating by using this design and steam injection to the outer wall of the drum. For many reasons, the implementation of this project will reduce smoke and flare pollution: Inhibition of freezing in the liquid outlet of the K.O.D., the liquid level inside the drum remains constant and prevents the transfer of liquid droplets associated with the exhaust gas to the flare.

6.1. Limitations and challenges of the Study

Various limitations and challenges of this study are the limited data availability related to flare gas emissions and their characteristics such as variable flare gas compositions and flow rates, implementing the 4R approach and optimizing flare gas management requires technical expertise and resources such as specialized skills and knowledge in process modelling, control systems, compliance with regulatory requirements while optimizing the flare gas management poses a challenge and can influence the feasibility of certain modifications, implementing modifications to optimize flare gas management may involve significant costs, including the installation of additional equipment, upgrading infrastructure, and implementing new control systems, ensuring that the modified system maintains safety standards and mitigates potential risks requires careful evaluation and consideration during the study ,and coordinating modifications with ongoing operations can be challenging, requiring careful planning and coordination to minimize disruption and downtime.

6.2. Recommendations for future research

Future research can consider these issues; investigating the feasibility of integrating advanced technologies and process modifications to optimize the reduction of flare gas emissions, exploring the use of innovative technologies such as flare gas recovery systems, fuel gas systems, or flare gas diversion options to mitigate environmental impacts and improve resource efficiency, conducting a comprehensive assessment of the economic and environmental impacts associated with implementing the proposed modifications to the knock-out drum high-pressure flares, quantifying the cost savings and greenhouse gas emission reductions achieved through the adoption of the 4R approach and comparing them to the traditional flare gas management practices, examining the potential scalability and applicability of the proposed modifications to knock-out drum high-pressure flares in different industrial sectors or geographical regions, and identifying the key factors influencing the implementation of the 4R approach and its effectiveness in various contexts, such as oil refineries, petrochemical plants, or natural gas processing facilities.

Nomenclature

A	Surface Area (m ²)
C ₁	Methane
C ₂	Ethane
C ₃	Propane

C ₆	Hexane
C ₇	Heptane
C ₈	octane
C ₉	Nonane
CPU	Central Processing Unit
CO ₂	Carbon Dioxide
D	Diameter (m)
EG	Ethylene Glycol
IC ₄	Isobutane
IC ₅	Isopentane
H ₂ O	Water
H ₂ S	Hydrogen Sulfide
HP	High Pressure
K	Kelvin
k _g	Kilogram
K.O.D	Knock out Drum
L	Length (m)
LPG	Liquefied Petroleum Gas
f _r , f _θ , f _z	Body force Components in the (r, θ, z)
m	Meter
N ₂	Nitrogen
NC ₄	Normal butane
NC ₅	Normal pentane
NO _x	Nitrogen oxides
PISO	Pressure-Implicit with Splitting of Operators
PRESTO	PRESSure Staggered Option
Pa	Pascal
RANS	Reynolds-Average Navier-Stokes
T	Temperature (K)
t	Time (s)
U _p	Uniformity Index
s	Second
SIMPLE	Semi-Implicit Method for Pressure Linked Equations
r	Spatial coordinate
θ	Spatial coordinate
z	Spatial coordinate
μ	Viscosity (N.s /m ²)
ρ	Density (kg/m ³)
ν	Kinetic viscosity (m ² /s)
σ _{ij}	Shear tension (N/m ²)
Ψ	Parameter such as Concentration, Pressure or Temperature

References

- Afolabi, E.A., 2012. Experimental investigation and CFD simulation of multiphase flow in a three phase pipe separator. Newcastle University. <https://theses.ncl.ac.uk/jspui/handle/10443/1756>
- Ahmadipouya, S., Ahmadijokani, F., Molavi, H., Rezakazemi, M., Arjmand, M., 2021. CO₂/CH₄ separation by mixed-matrix membranes holding functionalized NH₂-MIL-101(Al) nanoparticles: Effect of amino-silane functionalization. *Chemical Engineering Research and Design* 176, 49-59. <https://doi.org/10.1016/j.cherd.2021.09.011>
- Ali, Y.M., Saleh, S.N., Ayash, W.A., Ghani, S.Z., Salih, S.A., 2020. Design and CFD Simulation of Knockout Drum. *Journal of Petroleum Research and Studies* 10, 181-198. <https://doi.org/10.52716/jprs.v10i4.377>
- Arabi Shamsabadi, A., Rezakazemi, M., Seidi, F., Riazi, H., Aminabhavi, T., Soroush, M., 2021. Next generation polymers of intrinsic microporosity with tunable moieties for ultrahigh permeation and precise molecular CO₂ separation. *Progress in Energy and Combustion Science* 84, 100903. <https://doi.org/10.1016/j.pecs.2021.100903>
- Castiñeira Areas, D., 2006. A computational fluid dynamics simulation model for flare analysis and control. Thesis. University of Texas
- Fegade, U., Inamuddin, I., Altalhi, T., Ahamed, M.I., Kanchi, S., 2021. Effective adsorption of Fuchsin dye on FeZnOAC: kinetic, isotherm, double-layer modelling and reusability study. *International Journal of Environmental Analytical Chemistry*, 103(16), 3954-3970. <https://doi.org/10.1080/03067319.2021.1917559>
- Ferziger, J., Peric, M., 1996. Computational Methods for Fluid Dynamics Springer-Verlag Berlin Heidelberg New York. https://www.researchgate.net/profile/Ghassan-Smaism/post/FEM_mesh_generator/attachment/59d655b479197b80779acc79/AS%3A526929528274950%401502641197635/download/111+ferziger+peric+2002+computational+methods+for+fluid+dynamics+.pdf
- FLUENT, A., 2014. 6.3, 2006, FLUENT 6.3 User's Guide, Fluent. Inc., Lebanon, NH..
- Hajilary, N., Rezakazemi, M., Shahi, A., 2020. CO₂ emission reduction by zero flaring startup in gas refinery. *Materials Science for Energy Technologies* 3, 218-224. <https://doi.org/10.1016/j.mset.2019.10.013>
- Issa, R.I., 1986. Solution of the implicitly discretised fluid flow equations by operator-splitting. *Journal of computational physics* 62, 40-65. [https://doi.org/10.1016/0021-9991\(86\)90099-9](https://doi.org/10.1016/0021-9991(86)90099-9)
- Jethave, G., Fegade, U., Siddiqui, M.F.A., Ahamed, M.I., Suryawanshi, K.E. 2023. Adsorption of hexamethyl pararosaniline chloride dye on MgO-PbFe₂O₄: Experimental study and statistical physics modeling via double-layer model. *International Journal of Environmental Analytical Chemistry*, 103, 15. <https://doi.org/10.1002/kin.21600>
- Laleh, A.P., Svrcek, W.Y., Monnery, W.D., 2012. Design and CFD studies of multiphase separators—a review. *The Canadian Journal of Chemical Engineering* 90, 1547-1561. <https://doi.org/10.1002/cjce.20665>
- Mavriplis, C., 2012. Interdisciplinary CFD. *International Journal of Computational Fluid Dynamics*, 26(6-8), 333-335. <https://doi.org/10.1080/10618562.2012.739353>
- Monjezi, A.H., Mesbah, M., Rezakazemi, M., Younas, M., 2021. Prediction bubble point pressure for CO₂/CH₄ gas mixtures in ionic liquids using intelligent approaches. *Emergent Materials* 4, 565-578. <https://doi.org/10.1007/s42247-020-00148-4>
- Patankar, S.V., 2018. Numerical heat transfer and fluid flow. CRC press. <https://doi.org/10.1201/9781482234213>
- Pazani, F., Salehi Maleh, M., Shariatifar, M., Jalaly, M., Sadzadeh, M., Rezakazemi, M., 2022. Engineered graphene-based mixed matrix membranes to boost CO₂ separation performance: Latest developments and future prospects. *Renewable and Sustainable Energy Reviews* 160, 112294. <https://doi.org/10.1016/j.rser.2022.112294>
- Platvoet, E., Baukal, C., 2013. Process burners 101. *Chemical Engineering Progress* 109, 35-39. <https://www.aiche.org/sites/default/files/cep/20130835.pdf>
- Quadro, E., Guarieiro, L., 2016. Enhanced performance of knock drums by feed distributor designed with CFD assistance.. *Latin American Applied Research* 46, 161-166. https://www.laar.plapiqui.edu.ar/OJS/public/site/volumens/index/artic_v4604/Vol46_04_161.pdf
- Quattordio, J., Groenendijk, S., Kedzierski, R., Göebel, H., 2019. Optimal design of multi-stage depressurization systems using dynamic modelling. *Computer Aided Chemical Engineering*. 733-738. <https://doi.org/10.1016/B978-0-12-818634-3.50123-5>
- Rahimpour, M.R., Alizadehhesari, K., 2008. A Novel Fluidized-Bed Membrane Dual-Type Reactor Concept for Methanol Synthesis. *Chemical Engineering & Technology: Industrial Chemistry-Plant Equipment-Process Engineering-Biotechnology* 31, 1775-1789. <https://doi.org/10.1002/ceat.200800375>
- Rajabloo, T., De Ceuninck, W., Van Wortswinkel, L., Rezakazemi, M., Aminabhavi, T., 2022. Environmental management of industrial decarbonization with focus on chemical sectors: A review. *Journal of Environmental Management* 302, 114055. <https://doi.org/10.1016/j.jenvman.2021.114055>
- Rezakazemi, M., Heydari, I., Zhang, Z., 2017. Hybrid systems: Combining membrane and absorption technologies leads to more efficient acid gases (CO₂ and H₂S) removal from natural gas. *Journal of CO₂ Utilization* 18, 362-369. <https://doi.org/10.1016/j.jcou.2017.02.006>
- Sazhin, S., Kristyadi, T., Abdelghaffar, W., Heikal, M., 2006. Models for fuel droplet heating and evaporation: comparative analysis. *Fuel* 85, 1613-1630. <https://doi.org/10.1016/j.fuel.2006.02.012>
- Sheikh, M., Riasat Harami, H., Rezakazemi, M., Valderrama, C., Cortina, J., 2023. Efficient NH₃-N recovery from municipal wastewaters via membrane hybrid systems: Nutrient-Energy-Water (NEW) nexus in the circular economy. *Chemical Engineering Journal*. 465, 142876. <https://doi.org/10.1016/j.cej.2023.142876>
- Singh, K.D., 2013. Computational fluid dynamics modeling of industrial flares. Lamar University-Beaumont.
- Sohaib, Q., Muhammad, A., Younas, M., Rezakazemi, M., 2020. Modeling pre-combustion CO₂ capture with tubular membrane contactor using ionic liquids at elevated temperatures. *Separation*

- and *Purification Technology* 241, 116677. <https://doi.org/10.1016/j.seppur.2020.116677>
- Swati, I.K., Sohaib, Q., Cao, S., Younas, M., Liu, D., Gui, J., Rezakazemi, M., 2020. Protic/aprotic ionic liquids for effective CO₂ separation using supported ionic liquid membrane. *Chemosphere*, 128894. <https://doi.org/10.1016/j.chemosphere.2020.128894>
- Tu, J., Yeoh, G.H., Liu, C., 2018. Computational fluid dynamics: a practical approach. Butterworth-Heinemann.
- Vedovoto, J.M., Serfaty, R., Silveira Neto, A.D., 2015. Mathematical and numerical modeling of turbulent flows. *Anais da Academia Brasileira de Ciências* 87, 1195-1232. <https://doi.org/10.1590/0001-3765201520140510>
- Yadav, R., Parekh, A., Gupta, A., Das, A.K., 2012. Multiphase CFD Simulation of Flow Inside Liquid Seal Drums of the Refinery Flare System. *Industrial & engineering chemistry research* 51, 1073-1082. <https://doi.org/10.1021/ie202683h>
- Younas, M., Rezakazemi, M., Daud, M., Wazir, M.B., Ahmad, S., Ullah, N., Inamuddin, Ramakrishna, S., 2020. Recent progress and remaining challenges in post-combustion CO₂ capture using metal-organic frameworks (MOFs). *Progress in Energy and Combustion Science* 80, 100849. <https://doi.org/10.1016/j.pecs.2020.100849>
- Zadakbar, O., Vatani, A., Karimpour, K., Zadakbar, A., 2008. Flaring Minimization to Reduce Fuel Consumption-2 Case Studies in Tabriz Petroleum Refinery and Khangiran Gas Refinery, The 2008 Spring National Meeting.



© 2024. The Author(s). This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution-ShareAlike 4.0 (CC BY-SA) International License (<http://creativecommons.org/licenses/by-sa/4.0/>)