Gas Processing Journal

Vol. 8, No. 1, 2020, pp.37 - 48
http://gpj.ui.ac.ir

DOI: http://dx.doi.org/10.22108/gpj.2020.118993.1061

Simulation and CFD Analyses of Condensing Shell and Tube Heat Exchanger for CHP Package Application

Mousa Meratizaman [1*], Habibollah Kazemi Afracoti [2], Mohsen Shomali [3]

[1] Khaje Nasiriddin Tusi University of Technology [PhD], Tehran, Iran
[2] Master of Science in Mechanical Engineering, Mazandaran University of Technology, Mazandaran, Iran
[3] Master of Science in Mechanical Engineering, Shahrood University of Technology, Shahrood, Iran

Received: 2019-09-02 Revised: 2020-01-24 Accepted: 2020-01-27

Abstract: Simulation and Computational Fluid Dynamic (CFD) analyses of condensing shell and tube heat exchanger is subjected in this article. The condensing model using a User Defined Function (UDF) through Ansys Fluent 18.2 is developed. Also, the heat transfer between the shell and tube bundles is considered too. The subjected heat exchanger included seven tubes with a length of 600 mm and a shell with a diameter of 90 mm. The effect of different types of baffles and the distance between them is investigated in this article. All the calculations were done in three different mass flow rates. Finally, the distribution of heat flow is depicted for each stage of the considered heat exchanger. Results show that for the specific geometry and constant distance between baffles, increment in the mass flow rate leads to increasing the heat transfer coefficient. In addition, it shows that the formation of dew droplets and condensed water is higher in the first geometry due to the maximum heat transfer by hot flue gas.

keywords: Condensing Shell and Tube Heat Exchanger, UDF functions, Ansys Fluent 18.2, Baffles, Heat transfer coefficient

Introduction

By reducing energy reserves and increasing consumption in the industries, we will have to optimize energy consumption. In this sector, heat exchangers played an important role in heat transfer in the process. The most common and most widely used type of heat exchangers used in the industry is shell & tube heat exchangers designed and constructed for various applications in various sizes.

With increasing energy costs worldwide, manufacturers have always sought to improve the quality of operation and use of proper control equipment in the process. One of the most important of which is to expand the use of condensing shell and tube heat exchanger. In this study, a condensing shell and a heat exchanger are simulated using Ansys Fluent software.

After selecting the appropriate mesh for analysis (CFD), the calculation of the heat transfer coefficient and the outlet temperature in the fluid between the tube and shell boundary are calculated numerically in different baffle forms with the specified flow rates. As mentioned, one of the ways to improve efficiency is to use condensed heat exchangers. The heat of hot gases from the combustion of fuels contains steam with latent heat. Recovering this heat can increase efficiency by about 10 to 12%. When the smoke reaches the lowest dew point, the steam condenses and produces low pH acid with sulfur or carbon dioxide present in the smoke. Therefore, the choice of materials for corrosive parts should be appropriate. Aluminum and stainless steel are commonly used at high temperatures. In the lower temperature parts of heat exchanger components, plastics such as UPVC are used which have an impact on the final price. The generated condensate in a heat exchanger has to be discharged into the sewer. In this manner, heat exchangers are manufactured in the lowest technical size. For

* Corresponding Author.
Authors' Email Address: [1] M. Meratizaman (m.meratizaman1364@gmail.com), [2] H. Kazemi Afracoti (habibollahkazemi69@gmail.com)
[3] M. Shomali (mohsen.shomaly@gmail.com),
ISSN (Online): 2345-4172, ISSN (Print): 2322-3251

removing the exhausted gas from narrow channels, an induced fan is used in the condensing heat exchangers.

Because the gases usually cooled below 100 °C, they do not have any other driving force for the natural outflow of the chimney.

In the same research, Browne and Bansal [1] carried out an overview of the condensation heat transfer on a series of horizontal tubes bundles. The application of transmission models including the effects of steam in the pipe bundle is still a difficult reason for predicting the velocity distribution through the tube bundles. The results show that: The rate of cooling of the fluid has a significant effect on the overall heat transfer coefficient for the advanced levels of the pipes, but it has the slightest effect on the smooth tubes. Osaka et al [2], experimentally investigated condensation heat transfers on horizontal stainless-steel tubes by using actual flue gas from a natural gas heat exchanger. This test is based on various proportions of the flue gas and a wide range of wall temperature. By reducing the wall temperature, the wall surface is covered with a thin layer of liquid. The condensation heat transfer was well predicted with a simple analogy correlation in the high-wall-temperature region. In the low-wall-temperature region, less than 30 °C, the total heat transfer was higher than that predicted by the analogy correlation. Sun et al [3], experimentally studied a condensation heat transfer of wet flue gases. The heat transfer performance of the vapor-gas mixture with vapor condensation was discussed. The results showed that when water vapor concentration is high in the wet flue gas containing So_2, plastic (e.g, PTFE) heat exchangers can be used to recover sensible and latent heat and to avoid acid corrosion.

The flue gas temperature drops and the thermal efficiency is raised. In this method, the temperature of the exhaust gases is reduced and the thermal efficiency increases. The condensation of water vapor in wet gases dramatically increases the transmission of heat. In experimental mode, the heat transfer coefficient is twice as large as the transmission of heat in a single-phase mode. Also, the annular thin film condensation of water vapor in wet flue gas flowing through a vertical tube was studied theoretically and experimentally by Peng et al [4].

Particularly the discussion of the effects of condensation of low water vapor content (10 to 20 volume fractions) was carried out in the heat transfer of vertical displacement loads. The convection heat transfer was enhanced by the condensation of the condensable gas existing in the wet flue gas. This experiment also showed that wall temperature is an important factor affecting the condensation rate and the fog formation in the wet flue gas. Shee, et al [5].discussed the investigation of the performance of compact heat exchanger for latent heat recovery from exhaust flue gases. Characteristics of the heat transfer and the pressure of the heat exchanger, the inside of the tube and the fin of a heat exchanger is studied. Experimental results show that the Colburn factor and friction coefficient for wet air is larger than dry air. It has also been determined that the coefficient f and j for moist air increases by increasing the concentration of steam.

In the next step, Lee et al [6] examined the design factors for the heat exchanger and the boiler using a simplified model from an experimental heat exchanger. Characteristics of Specifications of each heat exchanger component (e.g., an upper heat exchanger (UHE) and lower heat exchanger (LHE); coil heat exchanger (CHE); baffles) were investigated using a model apparatus, and the comprehensive performance of the pilot gas boiler was examined experimentally.

When the optimal heat exchangers were designed, the heat output of the boiler was about 90%. Compared to a typical Bunsen-type boiler, the heating efficiency is improved by about 10%. Goel [7], of the University of Pennsylvania, USA, reports the possibility of recovering heat and water from gas by using a condensing heat exchanger. The effects on water efficiency, general health, and annual costs were investigated and analyzed for five different methods.

The effect of different design parameters such as pipe length and width of the converter pipe and their diameter were investigated. Five different thermal arrangements were evaluated to identify the design of a heat exchanger, which would increase the high condensation and the heat transfer velocity. Both the fluid flow of the flue gas and the cooling water stream, when it passes through the heat exchanger, eliminates the pressure and, therefore, the cooling water produces a higher pressure, resulting in high operating costs.

Changing tube diameter has significant effects on the total heat transfer coefficient and condensation efficiency. Wang et al [8], On the basis of the analysis of the mechanism of increasing the heat transfer in the inner tubes, suggested a new type of symmetric tubes with left and right fins. The type of tubes has the potential to increase the heat transfer. Experimental results show that excess air coefficient, the cooling water flow rate, the water inlet temperature, and the R_e number have prominent effects on the convection-condensation heat transfer. The left right symmetric internally finned tube can effectively decrease the radial temperature gradient reduce the thickness of the non-condensable gas layer and significantly strengthen the water vapor condensation. Lane et al [9], also, a numerical study has been carried out to investigate the heat and mass transfer characteristics of a condensing combustion flue gas in a cross-flow transport membrane tube bundle.

The tube wall is made of a porous material that is able to extract condensate liquid from the flue gas. The flue gas investigated consists of one condensable water vapor (H_2o).) and three non-condensable gases ($N_2 . Co_2 . O_2$). A simplified multi-species simple transport model is designed to transfer the mass and heat of combustion fuels. The condensation-evaporation process was simulated as a two-step chemical reaction. The numerical study was conducted within ranges of Reynolds number of $1 \cdot 0 \times 10^3 - 7 \times 10^4$ based on hydraulic diameter of flue gas channel, and $6 \cdot 4 \times 10^0 - 3 \cdot 3 \times 10^2$ based on the inner diameter of the water tube.

As the Reynolds number of the flue gas increases, the evaporation rate of the heat and mass decreases. Precise results on temperature, mass fraction, enthalpy and shell fraction coefficients are also presented. As a result, the overall performance of the pipes in the heat recovery from water is high when the chimney gas is lower in Reynolds numbers. In another step, Soheil Soleimani et al [10], from the University of Florida studied numerically the effects of different condensation heat exchanger performance.

The effects of inhospitable gases on condensation or condensation rates along with the number of pipe spacing (TMCs) in the direction of transverse and longitudinal tubes were investigated. Numerical simulation is performed using Ansys Fluent commercial software and condensation and mass transfer are defined using specific functions.

The results show that the increase in the number of TMC tubes, when the input flow rate is constant, increases both condensation levels and the average temperature has opposite effects on the density, both of which should be considered in industrial applications. In this study, the effects of the longitudinal and transverse steps of TMC pipe bundles based on heat exchangers were studied numerically.

Investigations showed that the heat transfer rate increases with increasing the amount of intake steam vapor, in addition to increasing the longitudinal and transverse steps of the reverse effect on the amount of heat transfer from the flue gas to the cooling water inside the TMC. In addition, the results showed that the heat transfer rate increased with increasing the amount of mass fraction of the vapor of the incoming water.

In addition, increasing the longitudinal and transverse steps has a negative effect on the amount of heat transfer from the flue gas to the cool water inside the pipes. Emerging and low-temperature water recycling using a ceramic membrane in a heat exchanger is a thriving technology that can be used to increase the efficiency of boilers and gas or coal boilers in various industrial processes and conventional plants.

The wall of a tube is a TMC-based heat exchanger capable of extracting pure water from the flue gas in the presence of other incompressible gases by means of a Nano-prussic state. Soheil Soleimani et al [11], In this paper, a numerical study was conducted to study the effects of longitudinal and transverse steps of TMC pipes on the performance of cross- flow heat exchangers. A multi-functional heat transfer model has been used to study the properties of heat transfer and the mass of a chimney gas in two symmetric pipes. Different steps (0.4-0.6) inches and longitudinal steps (0.4-0.8) have been used.

Numerical results show that the effect of transition steps on the output parameters is more evident. The results also show that longitudinal and transverse steps are one of the key parameters in the recovery of water and heat in a TMC pipe bundle and can be used to obtain an optimal state for maximizing the performance of cross- flow heat exchangers.

Terhan and Kumakley [12], studied on a heating system using 60 MW of natural gas to recover heat from the flue gas. They used One-dimensional mode (FDM) to calculate the chimney gas. The design was carried out on a gas condenser with an outer diameter of 34 mm and a thickness of 12 mm in the form of u-shaped tubes. Based on the calculated results, the chimney gas temperature can be reduced to 40 ° C by a gas- fired condenser of 80 square meters.

After the combustion of natural gas, the output chimney gas contains a lot of water vapor. The hidden water vapor heat in the flue gas is 10 to 11% lower than the natural gas heat content, respectively. That is, if the condensed heat from the flue gas (gas) is improved, energy efficiency can increase significantly. To investigate the compression flow patterns and the mechanisms for heat transfer and mass in different flow regimes, a horizontal shell- side condenser has recently been tested for a multifunctional condensation by Gu and et al [13].

The horizontal condenser is made up of 36 pipes with alternative modes and tubes. Steam and air mixture was also used as a test fluid. This paper provides empirical results on shell-shaped condensation patterns with transfer properties. The results are as follows: Condensation flow patterns and convective properties of a series of horizontal pipe bundles are significantly different from the horizontal condensation tubes. On the other hand, parameter F is presented to determine the flow patterns of condensation flow based on laboratory data and studies.

This function expresses the shear forces, the gravitational forces, and the tensile force of the fluid. By increasing the flow of non-condensing gases, the coefficient of condensation heat transfer of flow patterns decreases, because a high concentration of incompressible gases can reduce the portion of the condensation pressure of the steam and reduce the temperature of the liquid-vapor.

Most industrial systems include heat transfer facilities with non-design conditions during periods of their working cycle. Study and performance for off-design situations is an important part of any industrial system that ensures the safety and performance of the system for a wide range of operating conditions. In this context, Lane et ai [14] examined the numerical simulation of a heat exchanger shell and a TMC pipe for off-design operating conditions.

Modeling multi-component multiphase model and UDF functions and commercial flue, the density model, heat and mass transfer are modeled in a heat exchanger. In this study, the effects of various parameters such as incoming cooling temperature, flue gas mass flow rate and cool water on the overall heat transfer and condensation rate in the TMC tube heat exchanger have been numerically investigated. Changes in boundary conditions have been shown to indicate that heat transfer and condensation rates increase with increasing flue gas flow rate. The results also showed that increasing the temperature of cold water reduces the overall heat transfer and condensation rate.

another study, pipes with a ceramic body shell as a condenser are investigated to recover the heat transfer from smoke from the chimney to heat the water by Wang et al [15]. The effects of operating parameters such as fluid flow rate (water-gas), flue gas temperature and cool water, as well as flue gas humidity, are in process performance for the mass and heat transfer along the studied shell. In particular, the overall transmission coefficient is also calculated.

Increasing the flow rate of water or reducing the cooling temperature can effectively improve the efficiency and transfer of heat and mass. Increasing the inlet gas temperature can increase and improve the flow of water and heat, but does not increase the transmission efficiency. The increase in the humidity of the flue gas can significantly increase the amount of water and heat transfer and the total coefficient of transfer, but it has a small effect on water and heat.

Fin wall models (wall fins) with different shapes have many effects on the gas flow have a chimney that affects the performance of the plasma converter. In this paper, a numerical model of heat exchangers based on CFD theory and characteristics of heat and flow transmission from chimney gas was studied using CFD and experimental methods.

Cao [16] proposed the optimal shape of the Finn wall for the chimney gas duct. The results show that traditional circular fines reduce the effects of contact time and the effects of the transfer of heat between the chimney and the Fin. When used with elliptical fins, the retardation location and the effects of heat transfer zones are up to 7.6% and the mean transmittance of the radiation is increased by 12.1%. In the end, an appropriate formula for the transfer coefficient of convection and Nusselt number is obtained in optimal conditions.

Yin et al [17] obtained a three-dimensional numerical simulation of the VOF method, carried out from the condensate flow of water vapor, with the presence and presence of uncompromising interference of gases inside a 1 mm mini-tube. A UDF function (defined functions) has been applied to phase changes in a circular flow pattern. The temperature of the interface is assumed to be the saturation temperature of the water vapor. The effect of input speed, wall temperature, saturation temperature and super-heater temperature for pure water vapor concentration has been investigated.

The air is initially selected as an uncompromising gas, and then the volume of input is changing from 0.5-0.3 percent. In addition, the effect of incompressible gases on the condensation has been investigated. The results of this study can be extracted from the following way: The pure steam transfer coefficient is almost independent of the input velocity and super heater temperature, but with high saturation, temperatures increase the temperature difference between the saturation and the high wall temperatures. The steam quality is linearly reduced from input to output. When the condensation is mixed with an uncompromising gas, the mass transfer rate along the joint is severely reduced, resulting in higher vapor quality at the outlet and a significant drop in the heat transfer coefficient.

Chen et al. [18] conducted an experimental study on porous ceramic membranes for the removal of water and heat from flue gases under different conditions. Experimental results showed that the recovered water and heat flux increase as the feed gas flux rises, but a higher water recovery rate is achieved in a lower feed gas flow rate and the heat efficiency decreases for the saturated feed gas. When the feed gas flow rate grows, the promoting effect of the growth of temperature on the water recovery is more obvious. And the heat recovery efficiency increases with the growth of gas temperature within the feed gas temperature range (50–70 °C). The higher the relative humidity, the greater the amount of recovered water and heat are at the same temperature. Water recovery rate increases and reaches the peak at the condition of RH = 100% as the relative humidity rises, but the heat recovery efficiency is otherwise. The change of cooling water flux has little influence on the performance of the membrane but its rise can promote heat recovery. The amount of recovered water and heat, the water recovery rate and heat recovery efficiency decreases with the increase of inlet temperature of cooling water. However, due to the capillary condensation, the membrane can still maintain the amount of recovered water of about 0.5 kg/(m^2.h) and the water recovery rate of about 20–30%, when the temperature of cooling water increases to 65 °C (the water dew point is 60 °C).

In this research, 27 different configurations for the heat exchanger are compared to determine the optimum configuration. These configurations are various in terms of baffle shapes, the number of baffles and the flux flow rate.

Material and Methods

In this paper, a shell and tube heat exchanger condensing are used to check the heat transfer coefficient and temperature at the boundary between pipe and shell fluid, which are specified in different shapes of baffles with different mass. Some of the geometric parameters presented in Table 1 are water and flue gases as working fluid inside the tube and the shell side. Here, the effects of different baffles on different fluxes on the heat transfer coefficient and the water outlet temperature of the converter are investigated, and by reducing the distance between the baffles, we examine its effects. The heat exchanger with a baffle is shown in Fig. 1.

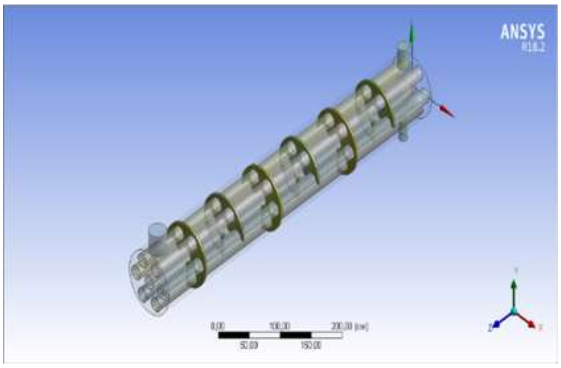


Figure 1. Shell and tube heat exchanger with 6 segments Baffles

Table 1. Geometric Parameters of the Design

Shell diameter	90 mm
External tube diameter	20 mm
Step and tube geometry	30 mm triangle
Number of tubes	7
The length of the heat exchanger	600 mm
Shell input temperature	450 Kelvin
Number of baffles	6-8-10
Distance between baffle centers	86-61-52 mm

Governing Equations

According to the following assumptions, the governing equations for this model of experiment are presented:

1- Uncontinental, Continuous, and Newtonian Fluid.

2- The effects of gravity are negligible and thermal radiation is ignored.

3- The assumption is that the problem is constant, meaning that time-dependent parameters are eliminated from the equation.

4- Due to the low spacing between buffer and shell, as well as pipes and baffles, the flow of leaks between them is not considered.

Mass survival equations:

$$\nabla.\left(\rho\vec{V}\right)=0$$

$$\nabla.\left(\rho u\vec{V}\right)=-\frac{\partial \mathrm{P}}{\partial x}+\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z}$$

$$\nabla.\left(\rho v\vec{V}\right)=-\frac{\partial P}{\partial y}+\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{zy}}{\partial z}+\rho g$$

$$\nabla.\left(\rho w\vec{V}\right)=-\frac{\partial P}{\partial z}+\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}$$

$$\nabla.\left(\rho e\vec{V}\right)=-P\nabla.\vec{V}+\nabla.\left(k\nabla T\right)+q+\Phi$$

$$\Phi=\mu 2\left[\left[\left(\frac{\partial u}{\partial x}\right)^2+\left(\frac{\partial v}{\partial y}\right)^2+\left(\frac{\partial w}{\partial z}\right)^2\right]+\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)^2+\right.$$

$$\left.\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)^2+\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)^2\right]+\lambda\left(\nabla\vec{V}\right)^2$$

The two-order accurate volume of fluid (VOF) method which is a free-surface modeling technique is used in this work [17]:

$$\varphi=\sum_{1}^{n}\alpha_q\varphi_q \; with \; \varphi=\rho.k.\mu$$

$$\varphi=\frac{1}{\rho}\sum_{1}^{n}\alpha_q\rho_q\varphi_q \; with \; \varphi=c_p$$

$$\nabla\cdot\left(\alpha_q\rho_q \underset{v}{\rightarrow}{}_q\right)=\dot{m}$$

$$\nabla\cdot\left(\rho \underset{v}{\rightarrow}\underset{v}{\rightarrow}\right)=-\overline{\nabla p}+\overline{\nabla}\cdot[\mu(\overline{\nabla}\vec{v}+\overline{\nabla}\vec{v}^T)]\;+\rho\vec{g}+\vec{F}$$

$$\overline{\nabla}\cdot\left(\vec{v}\left(\rho c_p T+p\right)\right)=\overline{\nabla}\cdot(k\overline{\nabla}T)+L_H\dot{m}$$

$$F=\sum_{ij.i<j}\sigma_{ij}\frac{\alpha_i\rho_i k_j\overline{\nabla}\alpha_j+\alpha_j\rho_j k_i\overline{\nabla}\alpha_i}{(\rho_i+\rho_j)}$$

$$\rho\frac{DV}{Dt}=\rho f-\nabla P+\mu\nabla^2 V$$

To simulate mean flow characteristics for turbulent flow conditions $k-\varepsilon$model is applied which consist of transport equations [9]:

$$K=\frac{1}{2}\overline{u_i'u_i'}\varepsilon=\left(\frac{\mu}{\rho}\right)\overline{u_{i.j}'u_{i.J}'}$$

$$\mu_t\propto\rho u_l\delta_l$$

$$u_l\propto\sqrt{k}$$

$$\delta_l\propto\frac{\sqrt{k^3}}{\varepsilon}$$

$$\rho\frac{\partial k}{\partial t}+\rho u_j k_{.j}=\left(\mu+\frac{\mu_t}{\sigma_k}k_{.j}\right)_{.j}+G+B-\rho\varepsilon\rho\frac{\partial\varepsilon}{\partial t}$$

$$+\rho u_j\varepsilon_{.j}=\left(\mu+\frac{\mu_i}{\sigma_\varepsilon}\varepsilon_{.j}\right)_{.j}+C_1\frac{\varepsilon}{k}G$$

$$+C_1(1-C_3)\frac{\varepsilon}{k}B-C_2\rho\frac{\varepsilon^2}{k}$$

Boundary Conditions

To simplify numerical simulation, some of the essential features of the process are assumed to be.

1. The thermal properties of the shell side are constant.
2. The flow of fluid and heat transfer in a turbulent process and is stable.
3. The leakage between the pipe and the baffle, as well as between the baffle and the shell, is not considered.

The value of the flow rate and the desired temperature at the input of the heat exchanger is determined. The input temperature to the shell is 450 K and the nozzle size is set to zero. The specification of the input speed is assumed to be uniform and the non-slip condition is considered for all levels.

Figure 2. Modified mesh corrosion mode

The thermal flux for the outer boundary of the shell is set to zero, assuming that the external wall of the shell is completely adiabatic. In this geometry, a two-phase mode is used, and the fluid in the shell is generated from combustion and fluid in the pure water pipe for the heating of the plant. In this model, mesh geometric is used Ansys software to produce the mesh and the shape is fitted with an automatic method. To obtain temperature and heat transfer coefficient, the hexahedral method has been used for mesh elements, which has the advantage of less volume and time.

For more precision and control over nodes in the heat exchanger, we split the model into parts. The simulation was carried out in Ansys Fluent 18.2. In the fluent for solving, the selected baseline state and the absolute velocity formula are determined with a steady-state for simulation. In this model, the energy and viscous mode calculations are considered as well as the RNG Caesar. Fluid is water in the tube and flue gas in the shell. Simple mode is determined for gravity and pressure. In the discrete gradient region, it is based on the minimum square cells. Quick site design for momentum states, finite volume methods for energy equations was used. In control, the pressure is 0.1, density 1, momentum 0.2, kinetic energy 1, as well as energy and turbulence viscosity 1. Initial initialization is standard.

Results and Discussion

To confirm the CFD modeling in the referenced article [18], the correctness of a complete model for fluid flow in a condenser heat exchanger is shown by the results with the data in the research.

Verification

As a first step in the present study, a three-dimensional numerical simulation of the constant was performed by finite volume method for condensing water vapor in a mini-horizontal tube, despite incompressible gases. A function defined for phase changes is used. For the input boundary conditions, the speed distribution for single-phase simulation is used.

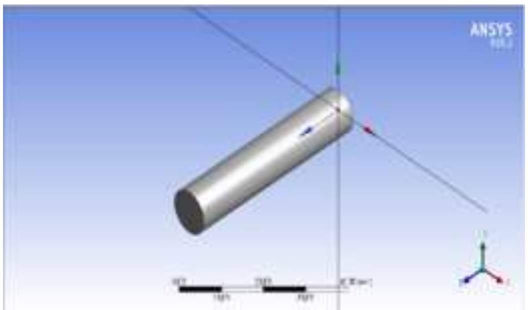
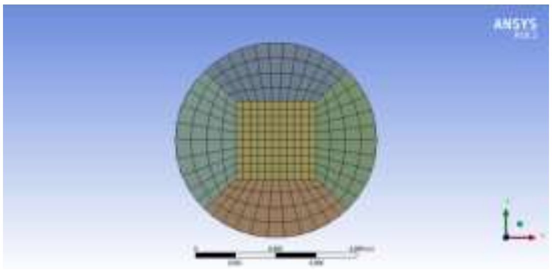


Figure 3. The miniature tube related to the projection geometry

Under operating conditions one atmosphere, the saturation temperature of steam is 373 K, when the wall is isothermal at uniform temperature. The speed and pressure algorithms have been used in simple mode while presto (Pressure-Taggering-Option) has been used in the simulation of pressure interpolation.

In order to obtain a liquid-liquid interface, the mesh size near the wall in which the condensate liquid film is produced is much smaller than the vapor area. Meanwhile, mesh size is relatively axial in comparison with the radial direction. Seven mesh sets representing a number of cells are examined. The results show that the grid mesh is more important in a radial direction.

As a result, the network type is selected from the Hexahedral mode.



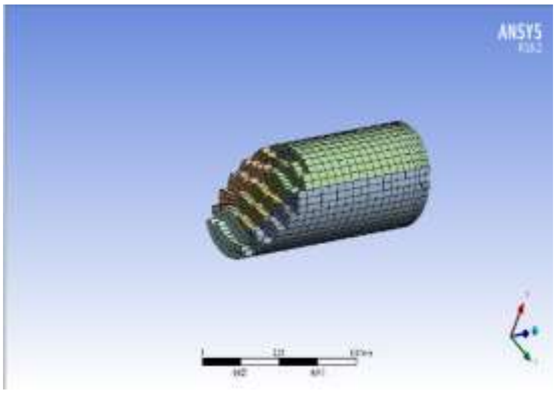


Figure 4. Output pipe mesh in Ansys 18.2

Here, the effects of the difference of saturation vapor temperature and wall temperature were studied in heat transfer performance. The saturation temperature ranges from 373 K to 343 K, while the input speed remains at 10 m / s. The difference between the wall temperature and the saturated vapor is about 5 to 20 Kelvin.

Table 2. The percentage difference between the reference and the results CFD

$T_s - T_w$ K	Heat transfer coefficient $w \cdot m^{-2} \cdot k^{-1}$		Difference percentage Error(%)
	Reference results	CFD results	
5	3.00E+03	2936.188	-2.28569
10	2.78E+03	2793.3	0.447916
15	2.67E+03	2652.465	-0.61325
20	2.62E+03	2589.684	-0.97795

Table 2 shows the percentage difference between the reference and the CFD results. From Fig. 5, it can be concluded that the saturation temperature has an extraordinary effect on the heat transfer coefficient of condensation.

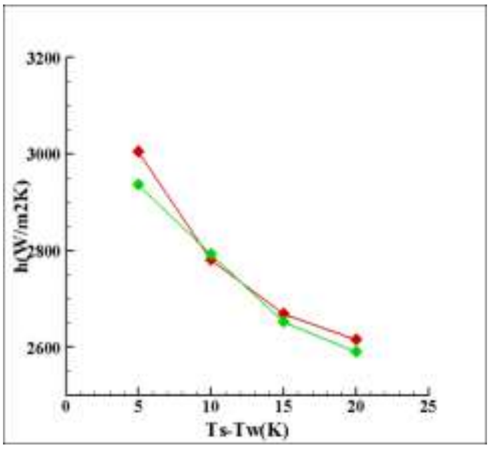
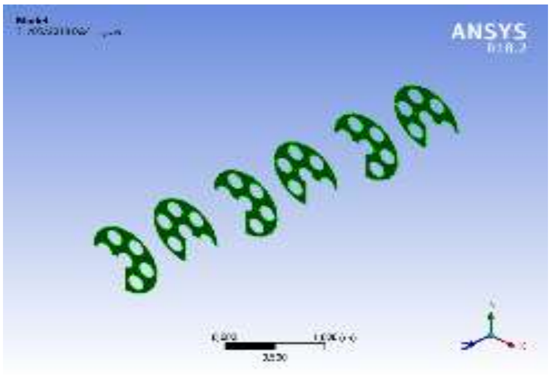


Figure 5. Condensation heat transfer coefficient graph based on the difference in saturation and wall temperature

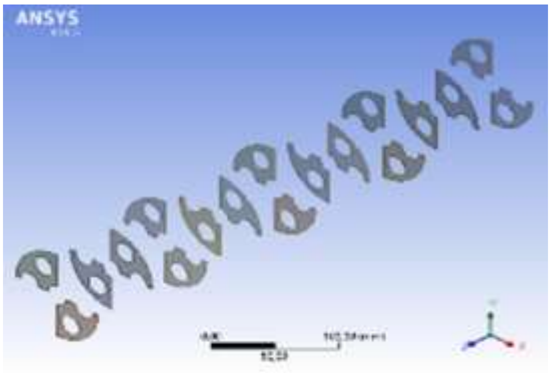
In comparison with the saturation temperature of 373 K, the reduction of the coefficient of the average heat transfer from 28.9%, 58.7%, and 84.1%, respectively, is due to saturation temperatures of 343, 353 and 363 K. It may be due to molecular activity and weakening the relationship between the vapor and liquid phase at low temperature and low pressure. In addition, the amount of heat transfer coefficient decreases as the saturation temperature decreases. As shown in Fig. 5, the reference miniature tube and the CFD results were checked for different saturation and wall temperatures, and the output temperature and pressure were verified by the research [18]. This chart shows that it is in good alignment with the reference. We checked this mode for three different masses.

Results

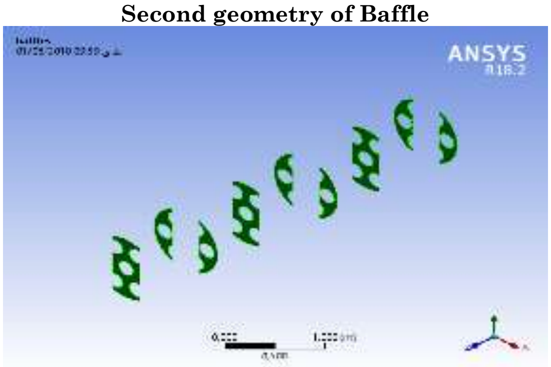
In this study, a shell and tube heat exchanger condenser model has been modeled in sufficient detail to solve the flow and temperature field. In this solution, the two-phase method is used and the heat exchanger has a flow, the fluid is inside the pure water pipe, and there is between the outer wall of the tube and the flame generated from the combustion shell. The inlet temperature of the smoke gas is 450 Kelvin and the input water temperature is 310.9 Kelvin. In Figure 6, the various baffles used in this study are shown.



First geometry of Baffle



Second geometry of Baffle



Third Baffle Geometry

Figure 6. Different forms of baffles used in research geometry

Table 3 shows the characteristics of the fluid and the input flow. The results of simulation of CFD, heat transfer and pressure drop in a condenser heat exchanger have been obtained. The results are obtained by changing the spacing of the baffles between 6, 8, 10 in three different baffles for the flux flow rate of 0.024-0.034-0.044 kg / s.

Table 3. Specifications of water and gas from combustion

Combustion gas	Input temperature	450 Kelvin
	Flow rate	0.024-0.034-0.044
	Gas combinations	$Co_2 - N_2 - O_2 - H_{2O}$
Water fluid	Input temperature	310.9 K
	Flow rate	0.325
The outer wall of the shell		Adiabatic
The inner tube genus		Aluminum

Effect of gap between baffles at outlet temperature and heat transfer coefficient:

In this section, the extent of their effect is discussed by changing the distance between the baffles for the condenser heat exchanger. In Fig. 7, flow lines and vector of converter velocity can be seen in a flow of 0.024 kg / s. As it is evident from the figure, in the number of higher baffles, the flue gas of the pipes with tubes is better due to the number of additional barriers, and it improves heat transfer. Of course, these additional barriers cause turbulence and flowing back.

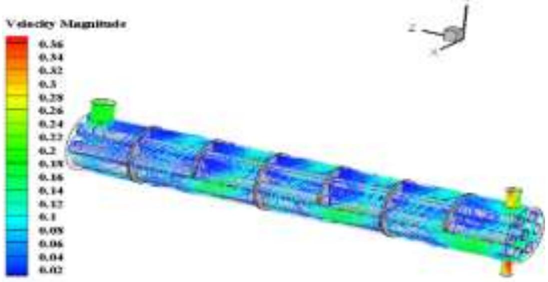


Figure 7. The velocity vector in the heat exchanger with 6 baffles

Fig. 8 shows the effect of the number of baffles on the transfer heat transfer coefficient in a steady-state discharge of 0.034 kg / s. As it is known, in the first figure of a baffle in the same number of six, the coefficient of h is greater, but by reducing the distance between the plates, the first figure provides a better displacement coefficient. The reason for this is that the two and three shapes, due to the design, have a lower barrier to flow and lower heat transfer. However, the first figure, because the flow path of the fluid is smaller and spiral, it causes more time to flue gas the pipes into the pipes and increase the heat transfer.

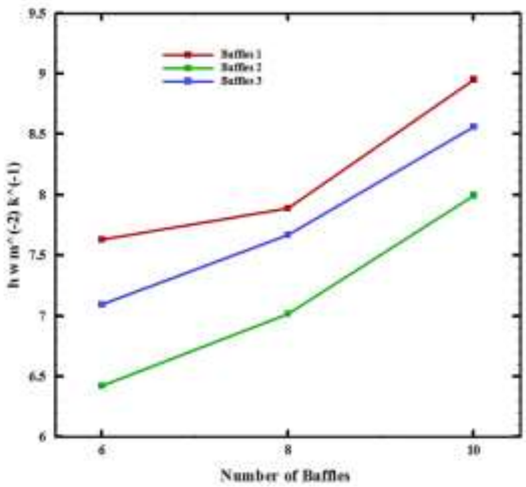


Figure 8. Heat transfer coefficient diagram based on the distance between the baffle plates

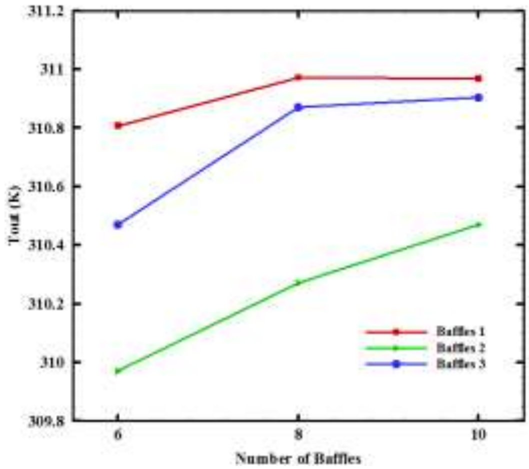


Figure 9. Output temperature diagram based on different baffle geometries

As shown in the diagram above, as in the previous state, in the first-order baffle, the outlet temperature is greater than the water's fluid. If we look at the green lines in the diagram that are related to the second type of the retaining plates, we see that both the heat transfer coefficient and the outlet temperature have the least amount compared to the other two. The reason for this is that this type of Baffles has a lower contact surface than the rest of the state.

Flux variations and its Effects on the Outlet Temperature, h

In this section, by changing the flux flow rate of the combustion gases and keeping the cold-water input at 0.325 kg / s constant, we will discuss the effects of this component on the temperature of the water outlet and the heat transfer coefficient. First, the contour of the phase fraction of the water property is depicted in each figure of the geometry of eight baffles with a discharge of 0.024 kg / s in the figure below.

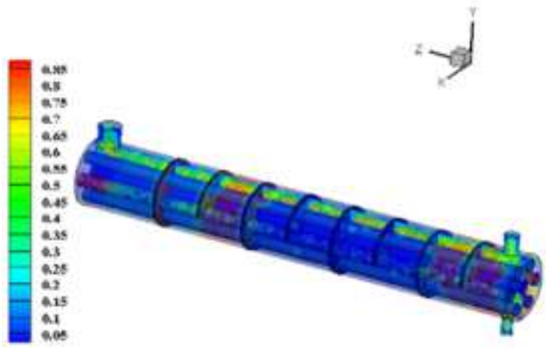


Figure 10. Contour cutting from the center of the fuzzy fraction H_2O

As shown in Fig. 10, the formation of dew droplets, or the water produced from the condensate, is higher in the first figure due to the higher heat transfer and the higher heat that loses the flue gas and, respectively, in the third and second geometries, due to lower heat transfer with droplets dew is less. Here we can conclude that the first Baffle geometry can have a higher heat transfer than the two mentioned above. Of course, we will continue to use the other charts.

In Fig. 11 we have examined the temperature changes in terms of different flow rates. In 0.024 kg / s, for the number of 6 baffles, the output temperature in the first type of the retaining plates is still higher than the previous ones. With an increase of 0.034 kg / s, the flow temperature of the water output in the first and third types is baffles.

At the end, when the flow rate reaches 0.044 kg / s, the first and third type baffles at the exhaust temperature are approximately equal to one, which in turn has the effect of increasing the discharge due to the change in the geometry of the retaining plates in the water's corona. An increase in temperature in higher flow rate is more significant since the vortex formation is much more in lower flow velocity. This vortex interrupts the heat transfer process, and consequently, outlet temperature increase is not significant for lower mass flow rate.

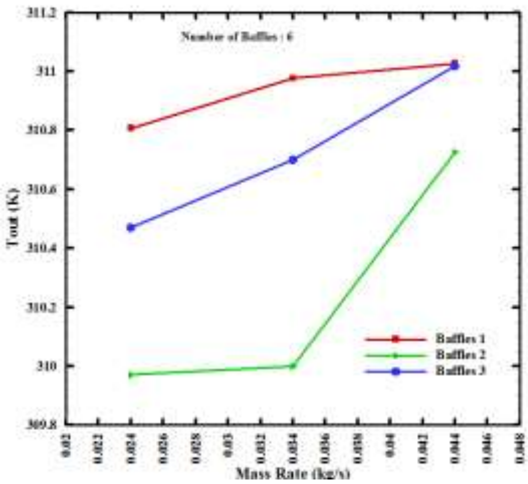


Figure 11. Chart of variations in the temperature of the output on different flow rates

Table 4. The Temperature of the Output - Flow rate0.024

Baffles	The Temperature of the Output		
	87 mm	61 mm	52 mm
1	310.8073 K	310.9818 K	310.968 K
2	310.97088 K	310.9713 K	310.97 K
3	310.971 K	310.9714 K	310.397 K

As seen in the previous figure, the temperature has fluctuated in terms of discharge, but now we want to discuss the effects of the variation of the discharge on the heat transfer coefficient in different baffles. In Fig. 12, we see how the transfer coefficient h changes in different flows with different types of baffles. It can also be concluded that the contact surface of the smoke with the tube is less than that of the baffle due to the passageway, which is the main reason for increasing the heat transfer coefficient. (The first step is to see the trend increase). Similar to Fig 11, the influence of high vortex formation in a lower flow rate is shown, too. Therefore, it should be concluded that the maximum heat transfer highly depends on the flow rate, the shape and the number of baffles.

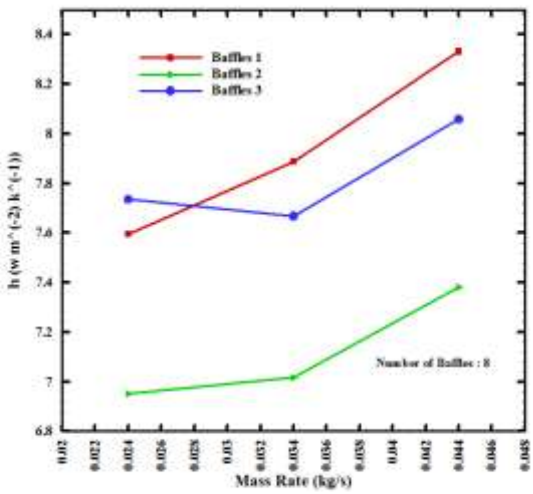


Figure 12. Changes in heat transfer coefficient in different flow rates

Conclusion

In this project, the heat transfer coefficient and the temperature of the fluid flow in a shell and tube condensing heat exchanger with turbulent flow and in different geometries of segments of the baffle are examined. Results are reported for various geometric parameters such as changing the spacing of the baffles as well as different flow rates. The results obtained from this study can be summarized as follows:

1. In the first geometry of baffle and in a flow and a constant distance between the retaining plates, as compared to the other modes, the output temperature increases due to the higher contact surface.

2. Reducing the distance between the center of the baffles causes more turbulence inflow. This will increase the output temperature.

3. Increase the amount of flow from 0.024 to 0.034 and then 0.044 kg/s in a constant distance of the baffle, causes the increment for coefficient.

4. The formation of dew droplets, or the water produced from the condensate, is higher in the first figure due to higher heat transfer and higher heat losses, which is lower in third and second geometries, respectively. The main reason for this event is lower heat transfer with dewdrops. This means that the first Baffle geometry can have a higher heat transfer than the two mentioned above.

Offers

To continue the research, the following options are suggested:

The use of nanoparticles in heat exchangers will increase the heat transfer rate. Therefore, the study on the use of these particles and their concentration and optimal sex are of great importance. Experimental study of heat transfers and flow in shell heat exchangers and condensation tubes with the mentioned baffles to verify the numerical studies performed in this study and available resources seems necessary. It is recommended to use a porous medium to eliminate the reverse flow in the path and to increase the turmoil to add the transfer of the heat exchanger.

Reference

Brownea M. W, Bansala P.K, (1999), An overview of condensation heat transfer on horizontal tube bundles, Applied Thermal Engineering, 19, 565-594.

Osakabe M, Ishida K, Yagi K, Itoh T, Ohmasa K, (2001), Condensation heat transfer on tubes in actual flue gas , Heat Transfer Asian Research 30, 139–151.

Jia L, Peng X. F, Sun J.D, Chen T. B, (2001), An Experimental Study on Vapor Condensation of Wet Flue Gas in a Plastic Heat Exchanger, Heat Trans Asian Res 30(7), 571-580.

Jia L, Peng X. F, Yan Y, Sun J. D, Li X. P, Effects of water vapor condensation on the convection heat transfer of wet flue gas in a vertical tube, International Journal of Heat and Mass Transfer 44, 4257-4265.

Shi X, Che D, Agnew B, Gao J, (2011) An investigation of the performance of compact heat exchanger for latent heat recovery from exhaust flue gases, International Journal of Heat and Mass Transfer 54, 606–615.

Lee S, Kum S. M, Lee C. E, (2011), Performances of a heat exchanger and pilot boiler for the development of a condensing gas boiler, Energy,36, 3945-3951.

Goel N, (2012), Design and Performance Analyses of Condensing Heat Exchangers for Recovering Water and Waste Heat from Flue Gas, A Thesis of Lehigh University, Paper 1164.

Wang Y, Zhao Q, Zhou Q, Kang Z, Tao W, (2013), Experimental and numerical studies on actual flue gas condensation heat transfer in a left–right symmetric internally finned tube, International Journal of Heat and Mass Transfer, 64, 10–20.

Lin C. X, Wang D, Bao A, (2013), Numerical modeling and simulation of condensation heat transfer of a flue gas in a bundle of transport membrane tubes, International Journal of Heat and Mass Transfer, 60, 41–50.

Soleimani S, Lin C. X, Wang D, (2015), Numerical Modeling of Industrial Scale Transport Membrane Condenser Based Heat Exchanger for Flue Gas Waste Heat and Water Recovery, IMECE November 13-19, Houston, Texas, 52324.

Soleimani S, Lin C. X, Wang D, (2016), Numerical Investigation of Heat Transfer and Condensation Rate in Two-Stage Transport Membrane Condenser Heat Exchanger Units, HT July, 10-14, Washington, DC, USA, 7291.

Terhan M, Comakli K, (2016), Design and economic analysis of a flue gas condenser to recovery latent heat from exhaust flue gas, Applied Thermal Engineering, 100, 1007-1015.

Gu H, Chen Q, Zhang Z, Guo H, (2016), Study of Condensation Flow Patterns and Heat Transfer Characteristic on a Horizontal Tube Bundle, IMECE November 11-17, Phoenix, Arizona, USA.

Lin C. X, Soheil Soleimani S, Wang D, Ghasemi E, (2017), Off-Design Modeling of Shell and Tube Transport Membranes Condenser Heat

EXxchangers, IMECE November 3-9, Tampa, Florida, USA-72495.

Zhao S, Yan S, Wang D. K, Wei Y, Qi H, Wu T, Feron P. H. M, (2017), Simultaneous heat and water recovery from flue gas by membrane condensation: Experimental investigation, Applied Thermal Engineering 113, 843–850.

Cao W, You X, (2017), Effects of Wall Fins Patterns on the Flue Gas Performance of Condensing Heat Exchanger, Science Direct Procedia Engineering 205, 2281–2288.

Yin Z, Guo Y, Sunden B, Wang Q, Zeng M, (2015), Numerical Simulation of Laminar Film Condensation in a Horizontal Mini-tube with and Without Non-Condensable Gas by the VOF Method, An International Journal of Computation and Methodology, 68:9, 958-977.

Chen H, Zhou Y, Cao S, Li X, Su X, An L, Gao D, (2017), Heat exchange and water recovery experiments of flue gas with using Nano-porous ceramic membranes, Applied Thermal Engineering, 110, 686-694.