European Online Journal of Natural and Social Sciences 2014;
Vol.3, No.3 Special Issue on Environmental, Agricultural, and Energy Science
ISSN 1805-3602
www.european-science.com

# Investigating the Causes of Corrosion in Heat Exchanger of Waste Incinerator System of Fajr Petrochemical and Providing Suggestions for Changing the Structure of Heat Exchanger

**Naser Shakeri Asl[1*], Taleb Zarei[1], Soroush Zarin Abadi[2]**
[1]Department of Chemical Engineering, Sirjan Science and Research branch, Islamic Azad University, Sirjan, Iran; [2]Islamic Azad University, Region 6, Khuzestan, Iran
*E-mail: naseer.8680@yahoo.com

**Abstract**

Optimization of operational performance of heat exchanger of waste incinerator systems from the temperature point of view in order to reduce corrosion is considerably important. At Fajr Petrochemicals, equipped with a biological waste incinerator system, some difficulties have arisen including corrosion in the equipment such as the heat exchanger and filter, which is the main factor of this corrosion rests in the heat exchanger. Therefore, the modeling of heat exchanger was investigated to find the causes of condensation formation and acid production. The modeling was done based on Computational Fluid Dynamics (CFD), and it was determined that, due to entry of hot gases to the heat exchanger and instantaneous cooling of them, humidity and condensation are formed in some parts of the heat exchanger and will cause the formation of acids and corrosion in it. In this study, a developed heat exchanger model is suggested so that the cooling of hot entry gases would be done in a mild manner, and also it would be possible to have a temperature control over outlet gases from the Heat Exchanger, and after the operational construction of the suggested model, and installation of it in the system, the previous difficulties were no longer.

**Keywords:** Waste Incinerator, heat exchanger, Computational Fluid Dynamics, Biological Sludge

## Introduction

Generally, any process that can burn solid wastes or reduce their weight, and transform them to less hazardous materials, is called waste incinerator. Fajr Petrochemicals, in order to realize the goals of managing the wastes, installed a waste incinerator system to incinerate the biological sludge from the wastewater. After the installation, a few difficulties were raised, including formation of condensation in the heat exchanger and formation of acid and corrosion in not only the heat system, but the related equipment. Although there have been extensive studies done on heat exchanger of waste incinerator systems, but while investigating the data, it can be said that the large part of resulted knowledge related to corrosion causes is, in fact, from the ash-flow of the heat exchanger. Therefore, in this study, by investigating the heat performance of heat exchanger using computational fluid dynamics, we tried to analyze the current difficulties. The endeavor towards improving productivity using a better design, can immensely help reduce energy consumption. But, considering the high costs of industrial tests and laboratory activities, more everyday turn to modeling methods. Among all this, in recent years, the use of computational fluid dynamic tools has gained considerable attention in various fields of chemical engineering.

Modeling using computational fluid dynamics has superior advantages compared to laboratory methods, which some of the most considerable of them are the rather low costs, higher speed, the capability of simulating semi-real and ideal situations and obtaining general data resulted from modeling.

In this project, in addition to investigating the hydrodynamics of gas flow speed, the cooling process of hot gases in the heat exchanger has been analyzed too, and a developed model of the heat exchanger has been suggested in order to better gain better thermal control. In the simulations, commercial programs of Gambit for geometrical drawings and networking, and Fluent for defining the physics of the matter, implementing border situations, and analyzing the results have been utilized, both being products of Ansys Company.

**Description of Dual Fuel Waste-Burning Furnace Process**

This system, considering its special design, has two main entrance feeds including waste liquid and waste sludge, which it has the ability to burn them solely or as a combination of both. The sludge initially enters a device called rotary furnace and starts to burn. 95% of the compounds exit from the other side of the device as combustion gases and 5% exit as ash. Combustion gases exiting from the rotary furnace enter the middle part of the stationary furnace so that the combustion process can be done fully. Inside the furnace the combustion operation is done in a vacuum of 150 pa (Pascal) so that the temperature rises up to 900 c. In this part, the operation of combustion completion of combustion gases in attempted. The ash from total combustion in the furnace, is transferred by helical transference systems to the ash tank, and gases from external combustion from the furnace, after passing through the heat exchanger, reach the temperature of 180 c, and furthermore, they are guided to the so-called Cyclone system for removing the heavier particles. Also, the gases from the combustion are guided to the device from the top side in order to enter the filter. In normal situations, the gases from combustion are guided towards the chimneys after passing through the filter in order to remove the particulate matters close to 3 microns. This is all while the analyzer system measures the percent of pollutant gases including: No, $No_2$, $O_2$, $H_2S$, NOX, CO, $CO_2$ right after their exit from the filter, and in the case of them exceeding the environmental standards, it instantly and automatically activates the pollutant compound neutralizing system by injecting powders, in which Calk powder in measured amounts is injected into gases from external combustion coming from the heat exchanger before their entry into silicon. Additional calk powder, following the components produced from neutralizing such as carbonated salts and solphate etc., will also be separated from the gases from combustion in the silicon and filter stage. The neutralizing operation take one second to be accomplished.

**The Required Equipment for Waste Incinerator System**

This system includes storage tanks, rotary furnace, stationary furnace, heat exchanger, calk injection system, cyclone, filter, and chimney

**Heat Exchanger**

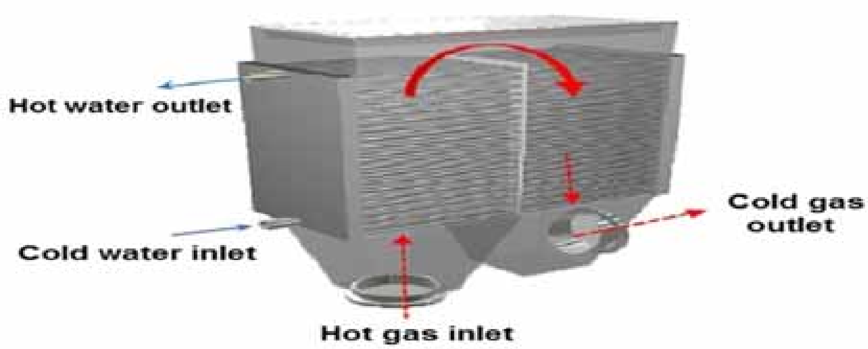


**Figure 1. The output and input view of water and gas from the heat exchanger**

**Investigating the Current Problems of Waste Incinerator System**

This system, considering its special situation, has quite a few process problems, some of them being the corrosion condition in heat exchanger section and its filtration system, and also the combustion gas pipe lines. After investigating the various parts of the system, it was viewed that the corrosion was due to formation of condensation in the heat exchanger section. Therefore, after the entry of combustion gases to the heat exchanger for cooling, their temperature goes from 900 ° C to nearly 200 ° C, which in this section due to the cooling of combustion gases, we will face a change of phase as condensation and formation of acidic liquids. This liquid has a pH lower than 1 and is acidic, and it mostly consists of Chloride. It is notable that this condensate liquid is exposed quite a lot during cold seasons of the year, and this causes serious corrosion in the heat exchanger and other components of this waste incinerator. It could also be noted that the material of this heat exchanger and gas transference pipelines leading to the chimney is stainless steel 316. In other investigations done on the internal parts of the heat exchanger, it was witnessed that other factors are involved in internal corrosion of the heat exchanger; small pieces of ash, enter the Heat Exchanger in order to cool the gases simultaneous with the exit of combustion gases, and due to numerous and rapid collision with one-inch thick pipes containing the Heat Exchanger water they cause small holes to appear on the internal pipes, which this corrosion can be mainly seen in the initial part of these pipes, and on the other hand this matter is a significant contributor to causing corrosion in the heat exchanger. The matter of passage of particulate was also investigated and it was witnessed that after long periods of time and numerous collisions of particulate dust with the pipes and causing of numerous holes on them, some of the pipes, due to the lessening thickness and the running water in one-inch pipes are used in order to cool the emission gases during the operation and leakage along with the emission gases and are mixed and by absorbing the chloride in these gases become acidic and therefore there will be double corrosions in the heat exchanger. Hence, in order to repair these pipes, a large amount of time and considerable expenses are required, also with the need to bring the system out of service for an extensive amount of time.

**Figure 2. Corrosion and Suspended Particulate Matter on the Internal Pipes of the Heat Exchanger**

As it can be seen, one of the key points in the occurrence of processing problems in this system is the waste incinerator of the heat exchanger. Therefore, thorough investigations must be done in this field, and this matter is analyzed using fluid dynamics methods of the heat exchanger, and after the needed designing on the speed of hot emission gases and the amount of heat transfer have been done in GAMBIT and FLUENT software, the data are analyzed and another suggestion for a heat exchanger to cool the combustion gases is reached.

**System Modeling Using Fluid Dynamics and Solution Method**

Computational Fluid Dynamics, consists of analyzing the systems including fluid flow, heat transfer and associated phenomena such as chemical reactions, based on computer simulation that includes a large range of industrial and non-industrial functions. CFD can be defined as using the computer to obtain information on the fluid flow in various situations. CFD includes various technologies such as mathematics, computer, engineering and physics, that the sum of them are used in order to create the fluid flow simulation tools. Such simulations are used in various fields of science and engineering, but will be fruitful if only their results be a trustable flowing fluid simulation. The quality of this simulation depends on the simulated matter, utilized software and the user skill.

**Equations of Turbulent Flow**

Flow field can be described by mass conservation equations, momentum, and energy. With respect to the boundary conditions, the resulting flow design is determined by equation solving of Navier-Stokes and energy or other scalar equations. There are various choices in order to predict the specialties of turbulence flow. In this study, the RNG turbulent flow has been utilized.

**A Review of Other Studies**

***A Review of Simulations Related to Heat Exchanger***

Hay Hun et al (2013) attempted studies related to investigating the formation of rust resulting from the passage of ash through the heat exchanger. In this investigation, they suggested a numerical model in order to predict the development of ash particles; this study that was done using the FLUENT software, and investigated the impact of six parameters including particle diameter, pipe direction, geometric shape of the pipes, pipe formation, flow speed and the speed of sediment on the sediment ration and the heat exchange performance, and it was determined that particle precipitation, initially, starts in the field of flow stagnation and is slowly gathered in the circulation areas and with the increase in particle diameter, ash sediment is made. Therefore, after many investigations and using GAMBIT and FLUENT software and modeling a part of the heat exchanger in two dimensional mode and investigating the sediment ratio on the suggested pipes, made certain suggestions in order to overcome these obstacles; in the first stage with increasing the speed of the flow to the allowed limit the formation of sediment can be prevented, and in the next stage, making changes in the formation of pipes, due to changing the doldrums prevents the formation of sediment, and in the third stage, utilization of oval tubes can effectively reduce the level of sediment.

Roman Weber et al (2011) attempted studies on predicting ash tendencies in the boiler using CFD. In this investigation, predicting the time of particles burning, heat and predicting distribution measurement of the oxidation rate resulting from ash flying close to the sediment level by computational fluid dynamics. The presence of ash and sediment on the pipes causes a reduction in the heat transfer rate and also corrosion. Therefore, they used modeling ash sediment using the RNG turbulence model, Reynolds number for the mean residence time and fluctuation speed and the static pressure of ash residues utilizing the k-ε Lagrangian model. And after the investigations done on the causes of ash sediment on steam pipes, the best setup to install the pipe banks was expressed to be vertical and facing down-side with a certain size, so that the time of stay for the ash residues be reduced. After this attempt, it was realized that sediment in second and third rows of heat exchange pipes has decreased.

Bohosle et al (2013) composed studies related to investigating the methods of hydraulic designing of heat exchangers in waste incinerator systems. This study, which was done on important parameters for hydraulic thermal design of heat exchangers, reached a new form of heat exchangers that are gradually developed. In this investigation, it was determined that the important parameters for choosing thermal exchangers are sediment, level of heat exchange, pressure drop, flow speed, surface temperature and geometry surface, and that they must be analyzed and investigated, so that the best choice can be reached. And since sediment makes up for a large portion of operational problems of heat exchangers, the sediment must be eliminated. Based on the attempted studies, sediment is divided into various categories; the amount of sediment precipitation, particle sediment, chemical reaction sediment, corrosion sediment, freezing and crystallization precipitation, which in most heating operations more than one kind of a sedimentation happen simultaneously. In the case of using finned tubes for heat exchangers, the pressure is increased, and with the placement of fins the level of heat exchange is increased and solid particles gather on the pipes. And, this will force us to bring the waste incinerator system out of service every time in order to clean it. Therefore, a suggestion was made to use simple and straight pipes, and due to the high temperature of exchangers, U pipes, so that the problem of sediment would be minimized. The mentioned matter was investigated using CFD and the precipitation speed on the pipes was analyzed. Following this study, it was announced that using the CFD method we can realize the flow pattern inside the heat exchanger in order to access the flow uniformity during equipment design, and this will optimize sedimentation in the future.


**Figure 3. Locating the Convertor in Fajr Petrochemical Waste Incinerator System**

**Defining Type A and B Exchanger**

Two types of systems have been utilized in this study. In the first system, which we call as (A), it was built based on the information related to the current heat exchanger and in the waste incinerator division by the company who produced the system, and in the second simulation the suggested design, which we name as (B) was simulated. It must be noted that this naming process is only done in order to better understand the results of each simulation; and any other name can be used.

**Modeling**

In order to model gas flow through heat exchanger pipes using CFD, not only we need mass conservation equations and momentum, but based on the flow, we would need completing models such as: turbulence model, momentum transfer equations. In here, the complete model is presented in two-dimensionally for hydrodynamics of heat transfer. The following hypothesis have been implemented in order to reach the dominant equations and solve them:

- Simulation has been done two-dimensionally for both states of A and B exchanger.
- The physical attributes of fluids have been considered as fixed.
- No chemical reactions take place in the system.
- Gas and liquid flow enter the system monotonically.

And phenomena such as input and output flow fluctuations are ignored.

**Boundary Conditions for A and B Exchanger**

In the present simulation the following boundary conditions have been utilized:

- In the entry of hot combustion gases to the heat exchanger, the speed and temperature is determined; the entry speed to the exchanger is 4m/s and the entry temperature is 900 c and the entry gas density in 15 c is $0.86^{0}kg/m^{3}$ and the entry gas intensity is 6000 $nm^{3}/h$.
- In the parapets the free boundary conditions are used for the gases.
- Heat transfer for cold water pipes is considered as a coefficient of fixed heat transfer.
- In the entry of flows to the water pipes and also gas entries and exits to the heat exchanger, certain values have been determined for speed and temperature.

**Two Dimensional Modeling Method of A Exchanger**

As it can be seen in Figure 1, the cooling of hot gases is done by transferring the heat by cold water pipes. It is important since the goal of modeling is to cool the gases after passing through each row of horizontal pipes, being 10 of them. Therefore, the cooling of hot gases in the exchanger is considered two-dimensionally.

**Geometric Characteristics and Two-Dimensional Drawing Method**

As seen in Figure 4, heat exchanger A was considered two-dimensionally, in a way that due to identicalness of heat exchange in internal pipes in each row and each sector, and considering the existence of 10 one-inch horizontal pipe and also the arrangement of 20 rows vertically, and also the same at the exit, the considered situation is as follows; hot gases enter the exchanger from below and after encountering the cold water pipes and exchanging heat each time, exit the first part of the exchanger and enter the second, in which all the gases move downward.

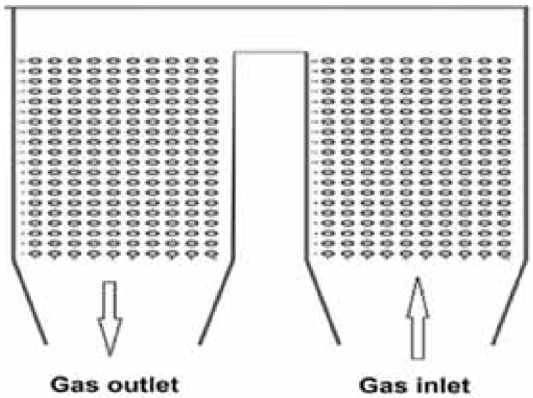


**Figure 4. Two-Dimensional View of Exchanger A**

**Table 1. Geometrical Characteristics of Exchanger A**

| | |
|---|---|
| Total length of the two parts of the Heat Exchanger (Meters) | 1 |
| Total width of the Heat Exchanger (Meters) | 0/5 |
| Total height of the Heat Exchanger (Meters) | ½ |
| Pipes Diameter (Meters) | 0/0127 |
| Exchanger Wall Thickness (Meters) | 0/05 |
| The Number of one meter pipes | 200 |

**Reticulation Methods**

In order to create geometry and reticulate geometric models, the Ansys Gambit 2.4.6 standard software was used. For reticulation, a Quad-dimensional and Tri-dimensional hybrid network was utilized. Also, in order to increase the accuracy of problem solving and preventing unneeded increases in computational costs, in a distance close to pipes, the network is finer and smaller and in far distances larger. The reason of this is that the main concern is pointed towards heat transfer in the parts of water carrying pipes. The total amount of the nodes is 55711. Figure 5 shows a view of the meshing.

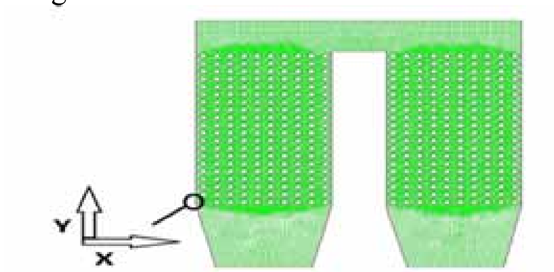


**Figure 5. System Geometry and A Reticulation Type in order to Choose Points X and Y for Exchanger A**

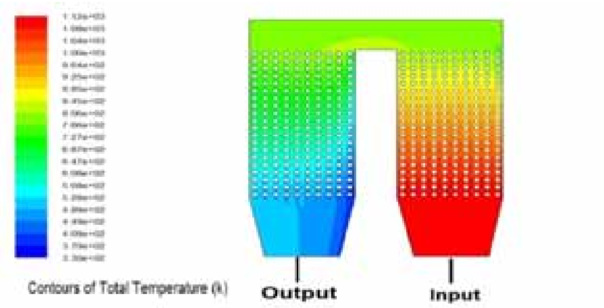


**Figure 6. Two-Dimensional Type Profile of Heat Transfer Solved in FLUENT with Respect to Exchanger A**

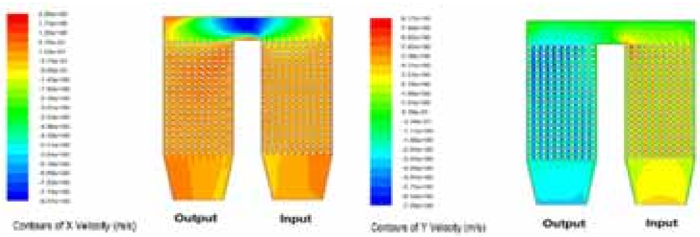


**Figure 7. Two-Dimensional Type Profile of Speed in Two Directions of X and Y Axis**

As it can be witnessed in figure 6, hot gases entering the heat exchanger to the amount of 800c are considered equal to 1073K, and the temperature of cold water entering the exchanger was considered 30c in normal situations, and the water exiting it as 85c. As the hot gases enter the heat exchanger, and after the collision of these gases with the pipes carrying cold water, heat exchange initiates in the first rows, such that we witness the highest amount of heat transfer in the first part of the exchanger. It is due to the dividing wall of the exchanger in the initial part and the high amount of heat exchange in this section, we would face various problems; the first being the change of direction of gas in a few parts of the heat exchanger, as shown in (figure 8).

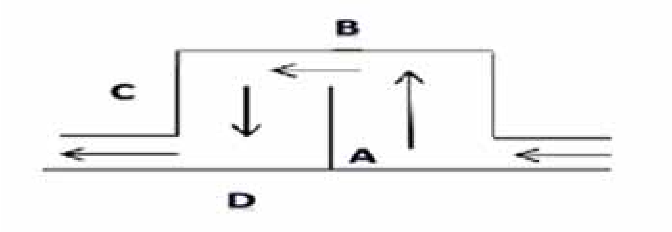


**Figure 8. The Entry State of Gas in Exchanger A and Wall Collision**

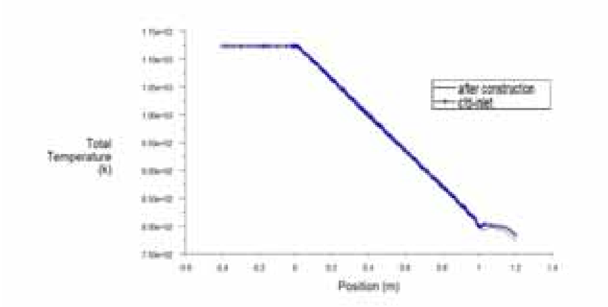


**Figure 9. The Distribution of Temperature in the First Part of Exchanger A Two-Dimensionally**

And as it can be witnessed in (figure 9), in the modeling of Exchanger A system which was installed in the waste incinerator system, the simulation done by CFD completely corresponded to the real information with function temperatures of the current heat exchanger, and this part of the diagram is related to the first part of heat exchanger A. In (figure 10), too, which is related to the second part of heat exchanger A, the simulation performance of heat exchanger can be witnessed by addressing the heat exchanger's information. The temperature drop in each part of the heat exchanger causes a formation of condensation in the second part of the exchanger in cold seasons, and this condensation of water will create an acidic condensation by being combined with some external gases, and eventually cause corrosion in the exchanger.

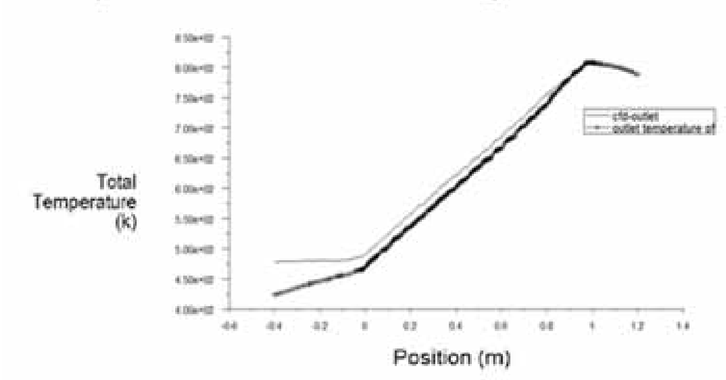


**Figure 10. Heat Distribution in the Second Part of Exchanger: A Two-Dimensional Model**

It can be seen in figure 10 that in the second part of heat exchanger, the temperature of emissions reaches nearly up to 130 to 150 c, and this causes condensation in the second part of the exchanger. And yet, on one hand, there can be no temperature control on this exchanger, since we have an inlet and outlet water to water carrying pipes, and this is one of the other problems in the heat exchanger. In the next figure, we can see the charts related to speed in inlet and outlet of the exchanger. In figure 11, the inlet speed is 4m/s at the entry, which considering that the exchanger pipes are aligned in both parts, the simulated chart is quite correspondent to the operational information taken from the site; and all this shows the properness of the simulation done by CFD. In the diagram (figure 11), the speed is leading up in the direction of field Y positive, and in the second part it is facing downwards in the direction of field Y negative, such that the speed profile in the direction of Y can be seen in figure 7.

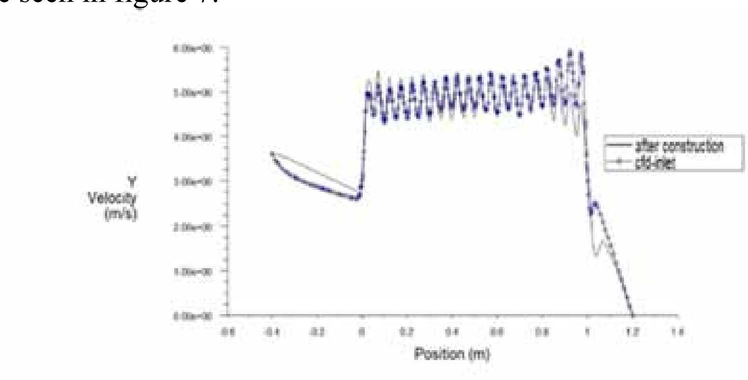


**Figure 11. Speed Distribution in the First Part of Exchanger: A Two-Dimensional model**

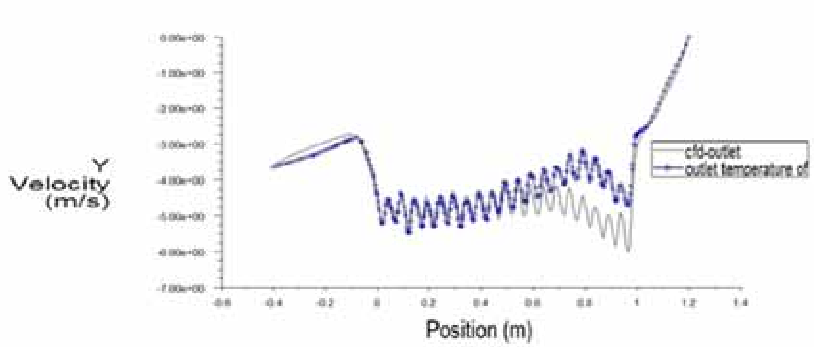


**Figure 12. Speed Distribution in the Second Part of Exchanger: A Two-Dimensional model**

**Investigating the Empirical Results of Simulating Heat Exchanger Type A**

Gas movement in figure 8 is fixed after exiting the furnace, and after entering the exchanger, it collides with the facing wall (A), then after colliding with the upper barrier (B) it changes direction and collides with barrier (C), and ultimately it exits after colliding with barrier (D), so that the gas movement exits the exchanger path after changing four directions, and this causes the emergence of sediment on the pipes, and this sediment is from the particles passed through the eliminatory rotary furnace and sludge incinerator and also the fixed furnace due to air intake. It can be seen in figure 8, that this problem is due to the changing direction of hot gases in various parts of exchanger, and also as it was mentioned and it can be seen in figure 6, the heat transfer in the first part of the exchanger is considerable, and this causes the formation of humidity and condensation and phase change due to the momentary cooling of hot gases especially in cold seasons. Also, this formed humidity and condensation is combined with the chloride of hot outlet gases and form a highly corrosive acidic liquid, causing extreme corrosion in higher parts of the heat exchanger. In addition, even due to the enormous heat waste in the path leading to the chimney, an extensive corrosion in seen in the internal body of the filter and the chimney; this is due to the fact that the temperature of outlet gas is considerably lower than the dew point of the corrosive acids. Therefore, considering that there is Sox in furnace outlet gases, and saturation critical temperature of $So_3$, one of the subsets of Sox, is 130 c, leading to the performance in temperatures lower than 130 c in the chimney outlet, and is not permitted due to the path lines leading to the chimney and are not insulated. Also, high thermal waste in this path increases the chance of formation of acidic condensation. Finally, while based on temperature records, utilization of outlet temperature of the converter has always been recorded as lower than this temperature especially in cold seasons such as winter. Considering that even a very little amount of $So_3$ (Nearly 0.15 vppm) in final outlet gases along with a low percent of steam present in combustion gases (6.5%) reach the dew point in 95c and form acid, which will definitely form acidic condensation and line and equipment corrosion in the path of outlet gases leading to the chimney, and unfortunately, based on the evidences such an event has taken place. It is notable that based on scientific resources and traditional methods of neutralization and prevention of emission of pollutants and corrosives in the environment, the proper operational temperature before filtration is higher than +300c, since acidic gases' absorbents function better in high temperatures, and also the rising of temperature up to +300c will prevent the condensation of this gases and therefore no corrosion will take place. It must be noted that operation in the temperature range of 300c to 400c will create an optimal situation for Nox control with the method of SCR (Selective Catalytic Reduction). In addition, the occurrence of the operation in a temperature higher than 300c will prevent the formation of De novo dioxin.

Therefore, it can be concluded that the most critical part in a furnace system is its heat exchanger. After investigating the exchanger, evidence including temperature fluctuations in different seasons of the year, and the frequency of corrosion and puncture of the walls and exchanger pipes and their outlet lines all prove the existence of glitches in the performance of the exchanger. These glitches can be divided into several categories:

- The high amount of the cooling water in the exchanger and also it not being limited by increasing and decreasing changes in the environment temperature in different seasons of the year.
- The spraying of water into the exchanger due to punctures in one or many of the pipes inside the exchanger and increasing the amount of acidic condensation formation and causing extensive corrosion.
- Choosing the service water as cooling water, which considering the high chloride in the material of the pipes which are made of steel, is a strong corrosive element, and considering the high temperature of the pipes, the speed of the corrosion and vulnerability of the pipes will be increased.

All the mentioned factors cause a great thermal waste and form condensation droplets on the exchanger walls and also condensation film on the pipe sets of the exchanger and outlet gas line wall, and consequently walls and pipes of the exchanger is corroded. Of course, the corrosion is followed by the puncture of one or some of the exchanger pipes, spraying of water through punctured pipes inside the exchanger, and this will create a water sprinkler combined with acidic gases that increase the amount of acidic condensation and consequently speed up the corrosion process. The combination of these acids with ash particles on the move, causes a destructive and progressive and extensive corrosion in the pipe, and till its exit, the chimney. As seen in figure 7, changes in speed show the axis of gas flow in the two-dimensional mode, since particles, too, due to changes in gas direction and reductions and increases of pressure and speed inside the exchanger cause surface corrosion on the pipes, which can be witnessed in figure 2. This greatly assists internal corrosion of the exchanger.

**Proposing a Plan for Changing the Type of Utilized Heat Exchanger**

After investigating the mentioned matter in order to deal with the corrosion problem of heat exchanger A and also obtaining a better potential for thermal control, various models were analyzed till a certain model was directly proposed; it was called exchanger type B and was investigated using CFD method. The following are the results of the simulation.

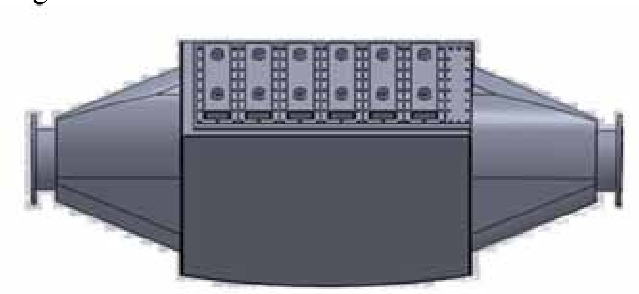

**Figure 13. General View of the Proposed Plan of the New Heat Exchanger**

The heat exchanger plan is as follows: it includes 6 feeders (Drawers), which each includes 40 one inch pipes made of A310 steel due to their high thermal resistance, in the upper part, there is a panel box and in the lower part, there is another from which 20 first pipes liquid enters and exits through the other 20 pipes, that 6 feeders (drawers) are attached to each other vertically on the gas flow. But the best situation is that the pipes be made in the shape of U in the lower part of the feeder.

- Considering the high temperature and the relatively acidic environment, pipes made of Stainless Steel TP A310 are selected, and also the boxes are from the same material.
- The exchangers are sided through the curst vertically and installed on the gas flow in a serial fashion.

The exchanger crust is made of carbon steel A285 that its walls are insulated by refractory materials able to withstand high temperature and with a thickness of 7 cm, which will prevent the heat from penetrating the wall and being wasted.

**Geometric Characteristics and Two-Dimensional Drawing Method of the Heat Exchanger**

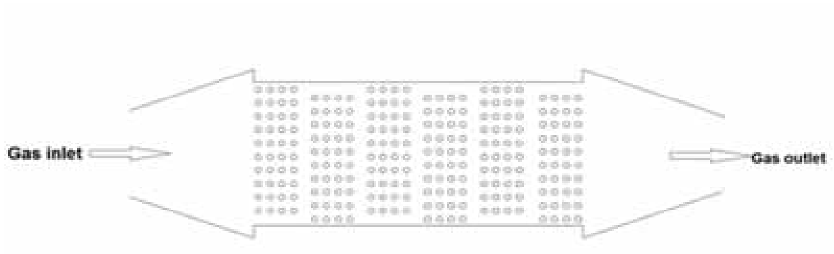


**Figure 14. The considered two-dimensional mode for the type B exchanger for drawing in the GAMBIT software**

As can be seen in figures 13 and 14, the type B heat exchanger was considered in two-dimensional mode, such that due to the presence of 6 feeders which are perpendicular to the gas flow, in each section of the exchanger water separately enters these feeders and then goes out. Therefore, in the considered mode, hot gases enter from one side of the exchanger and after collision with pipes perpendicular to the carrier stream of cold water and performing the heat exchange process in each section, its temperature will drop after colliding with 6 steps and finally goes out. It's worth mentioning that the walls inside the shell were coated using super alloy materials with high temperature tolerance and a thickness of about 7 cm, which prevents the penetration of heat to the outside.

**Table 2. Geometric properties of the type B heat exchanger**

| | |
|---|---|
| Length of the heat exchanger (m) | 2.6 |
| Width of the heat exchanger (m) | 0.8 |
| Height of the heat exchanger (m) | 1.2 |
| The diameter of pipes (m) | 0.0127 |
| Exchanger wall thickness (m) | 0.9 |
| Number of Pipes | 240 |

**The Reticulation method of the type B exchanger**

Geometry and meshing of the B geometric model was carried out similar to the A model. The total number of nodes is 78911. Figure 15, shows a view of the meshing.

***The results of simulation of heat exchanger type B***

The obtained results from CDF simulation of type B heat exchanger are presented as follows.

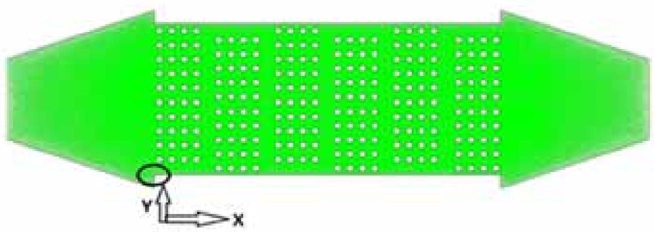


**Figure 15. Directions based on which X and Y points are chosen, for the type B exchanger**

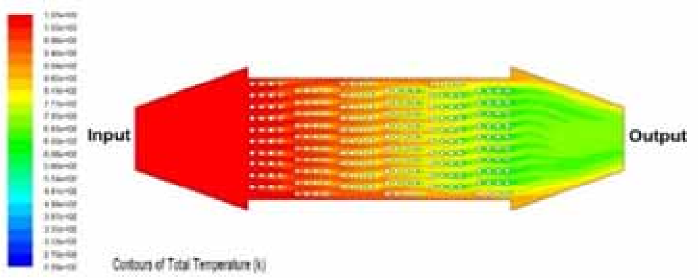


**Figure 16. The two-dimensional mode profile of the solved heat exchange for the type B exchanger**

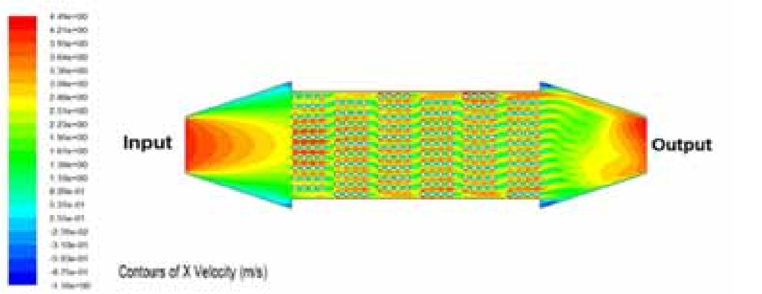


**Figure 17. Velocity profile in the direction of X in input and output section of the type B heat exchanger**

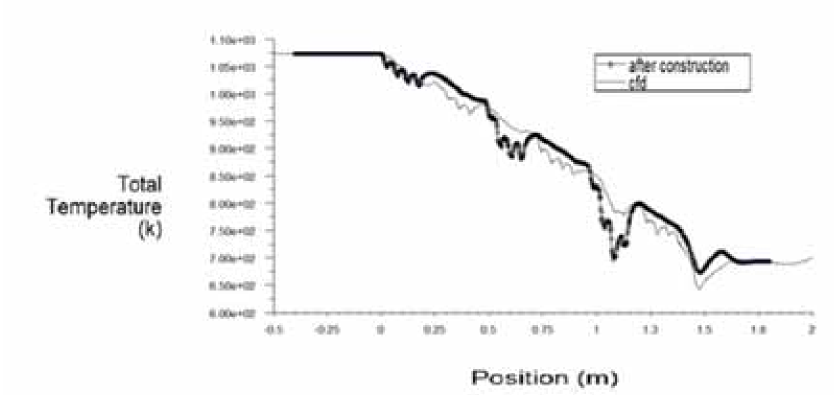


**Figure 18. Temperature distribution in the type B exchanger; two-dimensional, CFD mode**

As can be seen in figures 16 and 18, hot gas inlet temperature to the heat exchanger was considered $900°c$, which is equal to 1173 K, and the temperature of the outlet gas was considered $350°c$, which is equal to 623 K. Also, the gas inlet velocity to the exchanger was 4 m/s. The velocity profile can be seen in Figure 17. Therefore, considering the gas collisions with the walls of pipes and applying enough retention time, and simultaneously solving momentum and energy equations in FLUENT Software, we reached the required temperature condition. On the other hand, considering that we can uniformly distribute the inlet water to each of the feeders with low temperature, temperature control can be applied in all seasons, such that it is possible to divide the inlet water between several entries. For example, we can jointly consider two or three feeders; these feeders can deliver the cold inlet water to a section and the warm outlet water to the entry of the second section, which can be performed by placing a few valves in the inlet sections of feeders. Given the fact that in the previous model, because of the generated condensate in the exchanger, the inner wall of the exchanger was damaged, therefore it was needed to make exchanger shells from Carbon Steel A285 and coat its walls with refractory material with high temperature tolerance with a thickness of about 7 cm in order to prevent the penetration of heat to the outside.

**Conclusion**

In this study, first we named the two-dimensional model of the existing heat exchanger, A; it was simulated and evaluated using Computational Fluid Dynamics, and then hydrodynamic results for the temperature and velocity were analyzed.

Simulations conducted on the type A exchanger showed that it was unable to control the temperature and form acidic condensation, which led to corrosion of the heat exchanger.

Finally, it was concluded that structure of the heat exchanger have to change. Therefore, after evaluation of various designs using Computational Fluid Dynamics Software and analyzing different two-dimensional temperature, pressure, and velocity profiles, we reached a developed design which we subsequently called B. After building and installing the new heat exchanger in waste incineration systems, the temperature control problem was solved and we were able to successfully avoid the formation of acidic condensate. It should be noted that, type B, in addition to controlling the temperature of the outlet gases, is able to prevent abrasion and corrosion of pipes.

**References**

Bayern Brad, R. et al. (2010). Transfer Phenomena. Translated by Mouhamad Reza Afzali. Academic book publishing.

Fajr Petrochemical (2011). Technical documents regarding the newly installed hybrid waste incineration furnace. Publication of Fajr Petrochemical Company.

Halvayi, M. (2012). Biological and Environmental Science Life website. October 11.

Ibrahim Zadeh, M. (2011). CFD Tickner Modeling. MS Thesis in Chemical Engineering. Shahid Bahonar University of Kerman. Department of Chemical Engineering.

Niroomand, M. (2012). Investigating the Efficiency of Screening Tray Using CFD. MS Thesis in Chemical Engineering. Sistan and Bluchestan University. Department of Chemical Engineering.