Thesis for the Degree of Master of Engineering

Numerical Analysis for Optimization of LNG Vaporizer

by

MOHAN KUMAR DEY

Department of Interdisciplinary Program of Biomedical Mechanical & Electrical Engineering, the Graduate School, Pukyong National University

Busan, Korea.

February 2017

Numerical Analysis for Optimization of LNG Vaporizer

Advisor: Professor Yeon Won Lee

by

MOHAN KUMAR DEY

A thesis submitted in partial fulfillment of the requirements for the degree of Master of Engineering

In the Department of Interdisciplinary Program of Biomedical Mechanical & Electrical Engineering,

The Graduate School,

Pukyong National University

February 2017

Numerical Analysis for Optimization of LNG Vaporizer

A thesis

by

MOHAN KUMAR DEY

Approved as to styles and contents by:

ILL YEONG LEE

(Chairman)

KYOUNG JOON KIM

(Member)

YEON WON LEE

(Member)

February 2017

Contents

Numerical Analysis for Optimization of LNG Vaporizer

MOHAN KUMAR DEY

Interdisciplinary Program of Biomedical

Mechanical & Electrical Engineering

Pukyong National University

ABSTRACT

Liquefied Natural Gas (LNG) is usually heated from cryogenic state by a heat exchanger before supply to consumers, a process generally called heating. A heat exchanger is a device which is used for transferring heat from one fluid to another fluid with the two fluids temperature difference. In this study, the cryogenic state LNG is passed through the tube side and heated by the hot Glycol-water on the shell side. To increase the efficiency of the system and reducing energy consumption, it is suggested that maximum heat gets extracted from hot Glycol-water and transferred to LNG stream.

Hence, the present study compares different shell and tube sides arrangement system in order to check whether the assumed design satisfies all requirements or not. The shell side outlet temperature, tube side outlet temperature, pressure drop, re-circulation near the baffles, optimal mass flow rate, the optimal baffles position and percentage of baffles cut for the given heat exchanger geometry are determined from simulation.

The simulation results show that staggered tube arrangement with eight number of baffles (30% BC) case has a comparatively good result to other cases. The numerical analysis is carried out by ANSYS 16.2 package CFX.

Nomenclature

S_T	Transverse pitch [m]
S_L	Longitudinal pitch [m]
D	Tube diameter [m]
A_1	Transverse plane [m^2]
A_2	Diagonal plane [m^2]
$\dot{m}$	Mass flow rate $\frac{kg}{s}$
U	Overall heat transfer coefficients [$\frac{W}{m^2.^{o}C}$]
K	kinetic energy [$\frac{m^2}{s^2}$]
ε	Turbulence dissipation rate [$\frac{m^2}{s^3}$]
μ	Viscosity $\frac{kg}{m-s}$
u	Flow velocity component in x direction [m/s]
v	Flow velocity component in y direction [m/s]
w	Flow velocity component in z direction [m/s]

1. Introduction

1.1. Background of the study

The LNG (Liquefied Natural Gas) is the cleanest burning fossil fuel which is colorless, odorless, and shapeless. The LNG fuel has many advantages so the demand for LNG fuel is increasing all over the world to compare the conventional fuel [1]. It has a higher thermal efficiency and lower energy consumption. For so many positive reasons LNG consumption growing all over the world. According to the economist reports of anonymous oil and water about Asia Pacific, United States has no longer the only country to import LNG country like South Korea, Japan and China also import LNG. LNG used in power plant, internal combustion engine and ships propulsion in many ship around the world since 2011 [2]. Liquefied Natural Gas (LNG) is the product of raw natural gas. The maximum raw natural gas resource is the remote area. It is very difficult to transfer natural gas from resource to users by the pipeline most times it is impractical. So, natural gas is received as liquefied natural gas in a cryogenic state which temperature is approximate -162 ^{O}C and shrinking its volume by 600 times [3-5].

Figure 1 shows pressure verses temperature of LNG. The LNG become subcooled at point D by the pressurizing of HP pump then it’s capable of absorbing heat and reach the point E and increasing more pressure it’s heated more [6]. Heating process has done from hot water heat rejects to LNG by the shell and tube heat exchanger. Shell and tube heat exchanger is a device which is used for transferring heat from one fluid to another [7]

Shell and Tube Heat exchanger is the branch of recuperator's type heat exchanger. Recuperators is a closed type heat exchanger used to recover heat from different sources - such as wastes, exhaust gases and liquids - and transfer them to other mediums for reuse. Many closed type heat exchangers are used in the industry. Popular among them is the "shell and tube type heat exchanger".

There are two types of shell and tube type heat exchangers that can be classified according to the flow directions. They are (i) counter flow shell and tube type heat exchanger, where the hot and cold water flows in the opposite direction to each other and (ii) parallel flow shell and tube type heat exchanger, where the hot and cold water flows in the same direction.

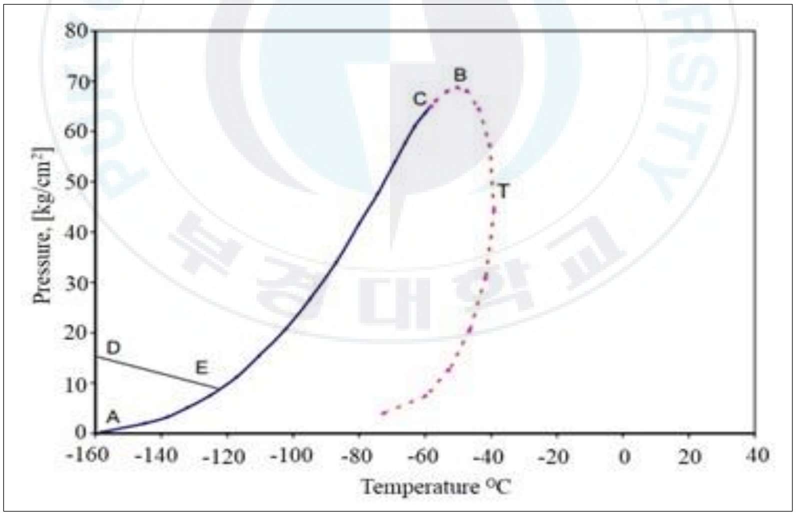

Fig. 1: Pressure verses Temperature in LNG Storage Tank

To get maximum heat extracted from hot fluid it's necessary to analysis the area of heat transfer, pressure drops in the shell and tube heat exchanger properly [8]. The amount of heat transferred from the water to natural gas depends on two things: a) Contact area between tube and shell side fluid. b) Contact time of small elements of tube side fluid with shell side fluid, for which heat transfer take place. To get maximum efficiency possible, both contact area, as well as contact time, are needed to be increased. a) Contact area can be increased by increasing the number of tubes. b) Contact time can be increased by increasing the length of the tubes and introducing a system of baffles. The concept contact time is studied by design a numerical model for two type's tube arrangement system- 1) staggered and 2) aligned tube arrangement system with different number of baffles and baffles cut percentage.

1.2. Purpose

A number of shell side and tube side flow arrangement are used in shell and tube heat exchangers depending on heat transfer, pressure drop, cost and predicting the performance of an existing exchanger operating under prescribed conditions. However, the purposes of this study are optimized the mass flow rate, inlet temperature, tube arrangement, baffles number and baffles size on the heat exchanger.

2. LNG Heat Exchanger

2.1. Segments in the LNG Heating process

2.1.1. Global Heating Capacity:

Since 2000, the number of LNG importing countries has tripled and heating capacity has more than doubled [9]. A wide range of LNG supply options, flexible shipping strategies, the growth of the spot market and floating heating technology have allowed new countries to become LNG importers.

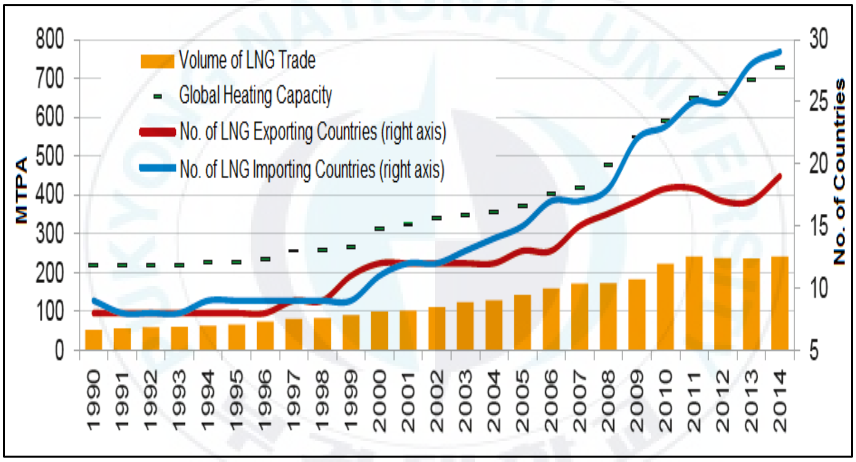

Figure 2: LNG Trade Volumes from 1990 to 2014

More than 75% of the world’s heating capacity is located within the leading five countries in this respect- Japan holds 30% of global heating capacity; USA, 20%, South Korea, 12%; Spain, 8%, and the UK, 6% and total heating capacity in operation presently is approximately 780 mtpa at the yearend 2012.

2.1.2. LNG Pumping Equipment:

The NBOG (Natural Boil-off Gas) is compressed by the HP compressor and condensed which is using a part of send out cold LNG. HP pumps send LNG at high pressure where LNG regains its gas state again (The boiling temperature of LNG is -162^{O}C at a pressure of one atmospheric) and delivered to the heat exchanger. LNG is pressurized before being heating as it is more efficient to heat the LNG. After the heating process, additional compression is not needed because the LNG leaves that stage with sufficient pressure to be transported.

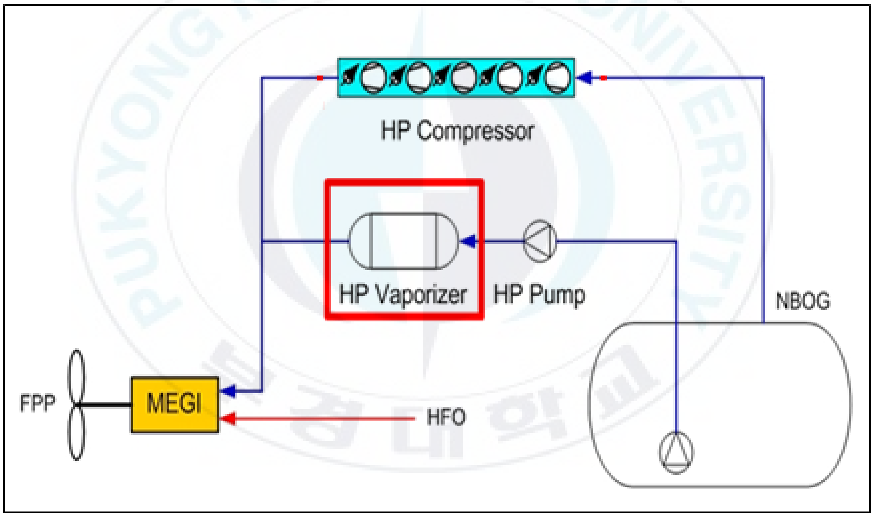

Figure 3: ME-GI Engine Fuel Supply System

2.1.3. LNG Tank

There are three main types of storages tanks:

- Above ground tanks with simple containment: the inner tank, typically nickel steel, contains liquid and vapor; and the outer tank, usually carbon steel, holds insulations but would not hold liquid if inner tank was breached.
- Above ground tanks with double containment: the outer tank, generally concrete would contain liquid in case of breach of inner tank, but vapor would escape. If the outer tank also holds vapor then it is a full containment tank.
- Buried or semi-buried (in-ground) tanks: more expensive than ground tanks, are used when environmental or aesthetic considerations are paramount.

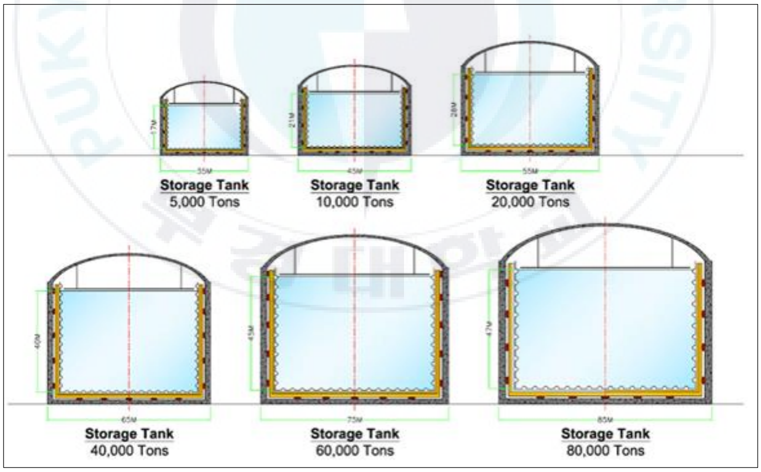

Figure 4: LNG Storage Tank

Tanks may be filled from the top or the bottom to ensure that different qualities and densities of LNG are blended to avoid rollover stratification phenomenon; if this occurs, pressures inside the storage tank may rise to excessive levels.

2.1.4. LNG Heating Process:

The cryogenic state LNG is heating by the shell and tube heat exchanger. The Glycol-Water mixture is heated by the some external sources like boiler, engine exhaust etc., and supply to the shell side on the heat exchanger while the HP pump, pumping the natural-boil off gas from the LNG storage tank and supply to the tube side on the heat exchanger at high pressure. The heating process is shown in figure.5.

Glycol- Water solution is used in the shell side of the heat exchanger for heating the LNG because it is considered to be more temperature consistent [10]. This will increase the efficiency of the system thereby reducing energy requirements. The number of tubes, surface area of the tubes, tube design and the distance between baffles are needed to be calculated to achieve desired temperature changes.

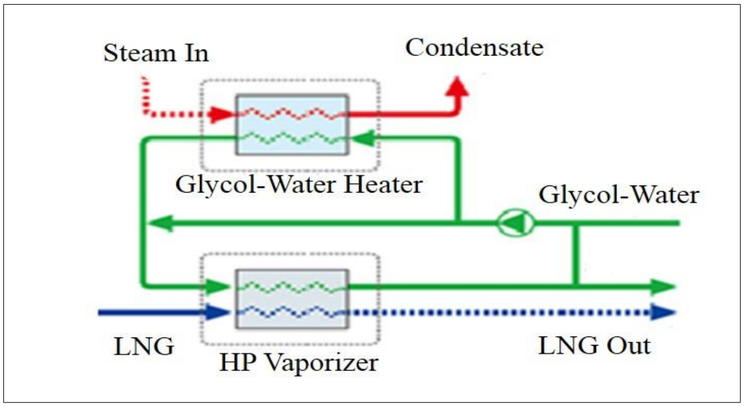

Figure 5: Process Configuration

2.2. Modeling of LNG STHE

Heat exchanger is a device which transfers heat from one fluid to the other and more specifically heating the cold fluid. This LNG heat exchanger is one shell and two tube pass U type counter flow shell and tube heat exchanger.

Figure 6: LNG STHE

Numerical analysis the heat exchanger is considered in two domains -1) Shell domain and 2) Tube domain. The shell domain is consist of shell cover, baffles plates and header plates while tube domain only tubes.

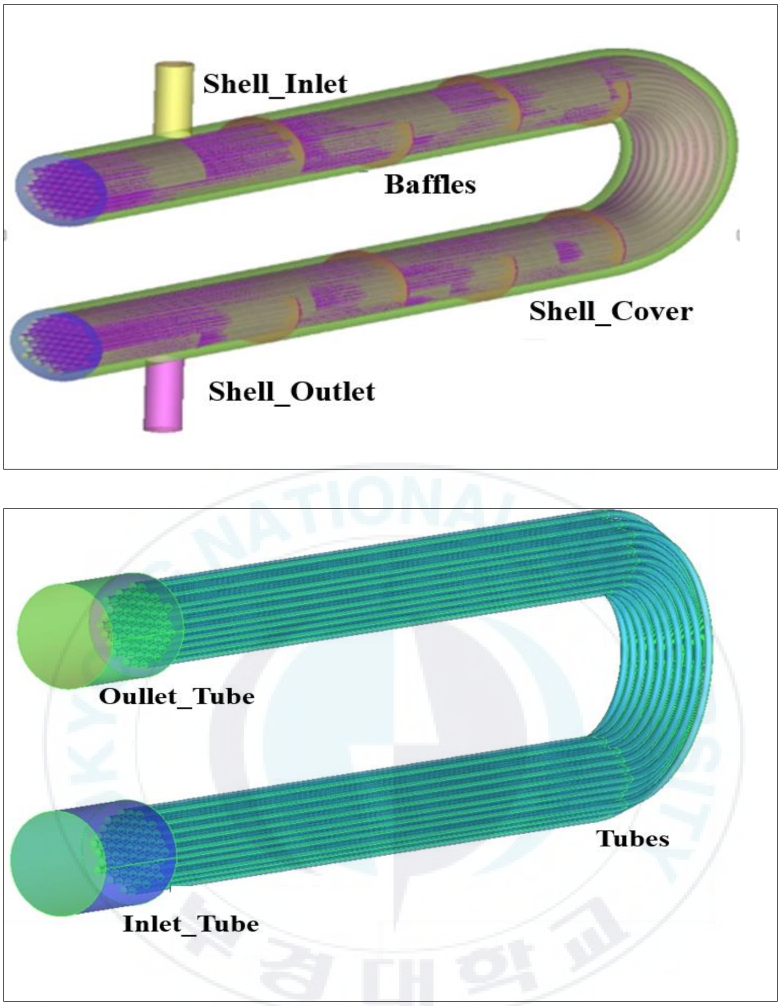

Figure 7: Shell and Tube Domain

2.2.1. Tubes Bank Modeling

2.2.1.1. Staggered and Aligned tubes arrangement in a Bank :

Figure 8 shows two types of tube arrangement in a bank (a) Staggered (b) Aligned. Tube banks are commonly employed design elements in heat exchangers. The hot fluid move over the tube in tubes bank, while cold fluid passes through the tubes therefore temperature differences between two sides (shell side and Tube side), so heat transfer occurs in that place.

The staggered configuration horizontal direction all tubes are in line where vertical direction tubes are in offset one after one row. The aligned configuration all vertical and horizontal tubes are in line and square arrangement.

The distance between two vertical tubes is transverse pitch S_T (16.5) and two conjugate vertical Colum distance is longitudinal pitch S_L (19.05). To determine the diagonal pitch S_D which is between two different row centers is used Pythagoras theorem. The shell has diameter 202.7 mm and it contains 63 tubes having outer diameter 12.7 mm individually with 2.108 mm thickness.

The staggered arrangement flow will be going through area A_1 but also have to go through area A_2 which are indicate in the figure that makes different in calculation. For staggered arrangement V_{max} could be through transverse plane A_1 or diagonal plane A_2, Whichever is smaller. However for aligned tube arrangement, Vmax always occurs at transverse plane A_1.

In this model the tube rows of a bank staggered alignment in the direction of the mas flow $\dot{m}$ and temperature T in the figure.8.

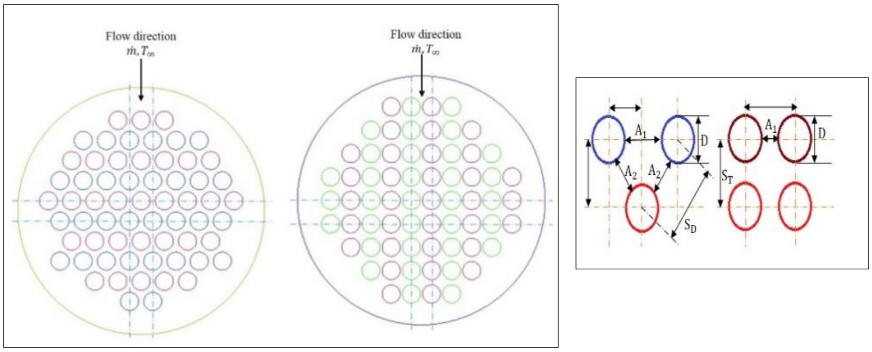

Figure 8: Staggered and Aligned tube arrangement in a tube bank.

2.2.2. Baffles Modeling

2.2.3. Perpendicular and Parallel Cut Baffles Modeling

Shell cover is covering the total shell side fluid and baffles are welded on the inside shell cover to enhance the mixing and turbulence of the shell side fluid stream. In order to allow the fluid to flow backwards and forwards across the tubes, there are a number of different baffles types and baffles cut system, which support the tubes and promote flow across the tubes. In this study the horizontal and perpendicular baffles cut arrangement are analyzed by numerically.

Baffles are used in the heat exchanger for reducing the contact time in shell fluid. However, it can be increased the pressure drop of the fluid. In figures 9 show the perpendicular (horizontal) and parallel (perpendicular) baffle-cut for vertical shell inlet axis. Baffles cut is the height of the segment that is cut in each baffle to permit the shell side fluid to flow across the baffle. This is expressed as a percentage of the shell inside diameter

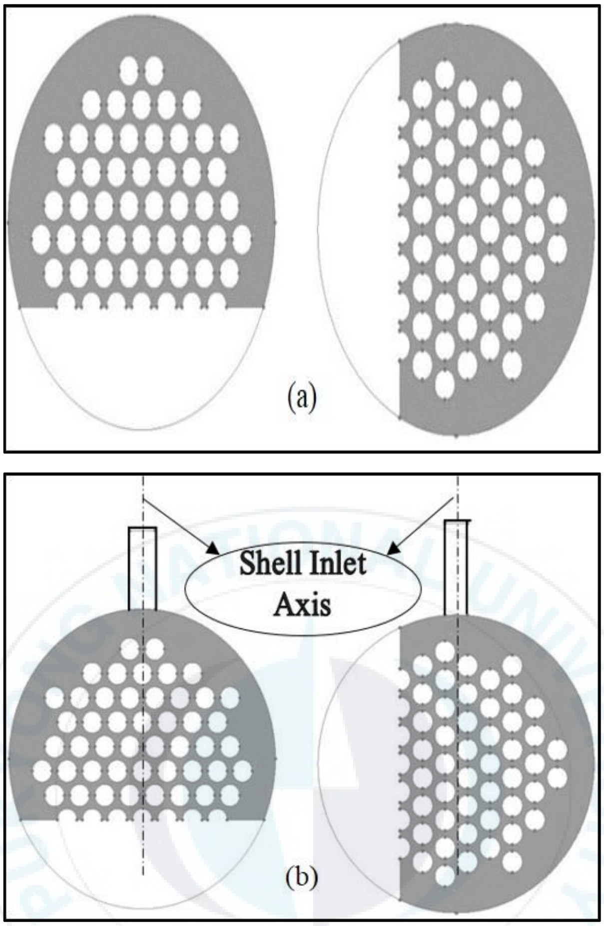

Figure 9: (a) Horizontal and Perpendicular cut Figure: (b) Perpendicular cut baffle and Parallel cut baffle.

3. Numerical Analysis

3.1. Basic Equation:

Continuity Equation:

In fluid dynamics, the continuity equation is an expression of conservation of mass for a steady and incompressible flow the continuity equation is

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

Momentum Equation:

The Naiver Stokes equations are the basic governing equations for a fluid flow. It is obtained by the applying Newton's second law of motion to a fluid element and is called the momentum equation.

x component of the momentum equation:

$$\rho\,[u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}] = \frac{\partial(-p+\tau_{xx})}{\partial x} + \frac{\partial(\tau_{yx})}{\partial y} + \frac{\partial(\tau_{zx})}{\partial z}\ ;$$

y component of the momentum equation:

$$\rho[u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}] = \frac{\partial(\tau_{xy})}{\partial x} + \frac{\partial(-p+\tau_{yy})}{\partial y} + \frac{\partial(\tau_{zy})}{\partial z}\ ;$$

z component of the momentum equation:

$$\rho[u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}] = \frac{\partial(\tau_{xz})}{\partial x} + \frac{\partial(\tau_{yz})}{\partial y} + \frac{\partial(-p+\tau_{zz})}{\partial z}\ ;$$

Thermal Energy Equation:

Thermal energy is responsible for the temperature of the system, the thermal energy equation is

$$\nabla(\rho hU) = \nabla(k\ grad\ T) + \frac{\partial(u\tau_{xx})}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(u\tau_{zx})}{\partial z} + \frac{\partial(v\tau_{xy})}{\partial x} + \frac{\partial(v\tau_{yy})}{\partial y} + \frac{\partial(v\tau_{zy})}{\partial z} + \frac{\partial(w\tau_{xz})}{\partial x} + \frac{\partial(w\tau_{yz})}{\partial y} + \frac{\partial(w\tau_{zz})}{\partial z}$$

3.2. K- Epsilon Model ($\boldsymbol{K-\varepsilon}$):

K-epsilon turbulence model is the most common model which has been applied in most general CFD codes to simulate mean flow characteristics for turbulent flow conditions. It has proven to be stable and numerically robust and has a well-established regime of predictive capability.

The first two- equations model for predicting the behavior of turbulent flows was proposed in 1942 by A.N. Kolmogorov later the standard $k-\varepsilon$ model has developed Launder and Spalding in 1974 which minimizing unknowns and presenting a set of equations which can be applied to a large number of turbulent applications. The first transported variable determines the energy in the turbulence and is called turbulent kinetic energy (K), unit is $\frac{m^2}{s^2}$. The second transported variable is the turbulent dissipation (ε) which determines the rate of dissipation of the turbulent kinetic energy and unit is $\frac{m^2}{s^3}$.

The standard model uses the following transport equations used for K:

$$\nabla(\rho KU) = \nabla\left[\frac{\mu_t}{\sigma_k} grad\ K\right] + 2\mu_t E_{ij}.E_{ij} - \rho\varepsilon$$

The standard model uses the following transport equations used for ε:

$$\nabla(\rho\varepsilon U) = \nabla\left[\frac{\mu_t}{\sigma_\varepsilon} grad\ \varepsilon\right] + C_{1\varepsilon}\frac{\varepsilon}{k} 2\mu_t E_{ij}.E_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

In words the equations are:

Transport of k or ε by convection	=	Transport of k or ε by diffusion	+	Rate of production of k or ε	-	Rate of destruction of k or ε

Here,

$K - \varepsilon$ to define velocity scale ϑ and length scale ℓ respectively,

$$\vartheta = k^{1/2} \quad \ell = \frac{k^{3/2}}{\varepsilon}$$

Turbulence eddy viscosity:

$$\mu_t = C\rho\vartheta\ell = \rho C_\mu \frac{k^2}{\varepsilon}$$

The constant of these equation are

$$C_\mu = 0.09 \quad \sigma_k = 1.00 \quad \sigma_\varepsilon = 1.30 \quad C_{1\varepsilon} = 1.44 \quad C_{2\varepsilon} = 1.92$$

3.3. Numerical Model:

In numerical simulation the whole heat exchanger model are considered. The modeling and meshing are done by the ICEM CFD and CATIA V5 respectively.

Tubes meshing are done by the unstructured tetra-mixed Robust (Octree) volume mesh method due to growth the boundary layer near the wall, prism meshing also generated in this modeling. The shell side meshing is generated as the same way of tube side meshing except prism mashing layer. The Shell and Tube domains are connected by interface model. This is often useful for connecting static domains together with non-matching grids. So, prism meshes on tube side connected to tetrahedral meshes on the shell side by GGI mesh connecting method.

The shell side hot water transfer the heat through the interface material to the tube side LNG. Normally, in tube materials is used as aluminum at shell and tube heat exchanger but LNG vaporizer the stainless steel is used for handling the cryogenic state of fluid. Because at low temperature conditions tube material change to its atomic bond and it becomes brittle material so keep the hardness of material, the stainless steel tube is used in LNG heat exchanger.

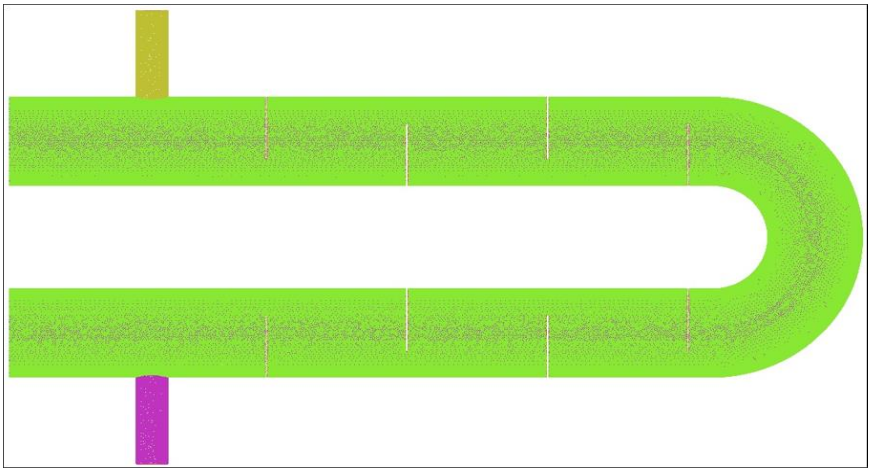

Figure 10: Meshing of Shell Domain

Figure 11: Meshing of Tube Domain

3.4. Computational Conditions:

Table1: Conditions for Model:

Feature	Details
Analysis Type	Steady State
Turbulence Model	K-Epsilon
Heat Transfer Model	Thermal Energy
Domain Type	Double Domain
Boundary Conditions at Shell Domain	Inlet: Mass Flow [Kg/s], Temperature [^{O}C]
	Outlet: Static Pressure [pa]
Interface Model & Material	Domain Interface & Thin Material (Steel)
Boundary Conditions at Tube Domain	Inlet: Mass Flow [Kg/s], Temperature [^{O}C]
	Outlet: Static Pressure [pa]

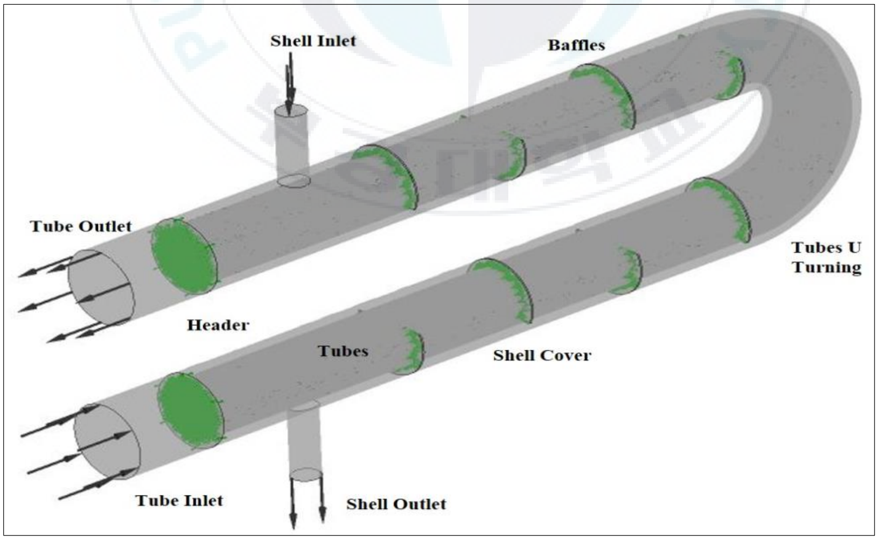

Figure 12: Computational Model of STHE

ANSYS CFX is useful for studying fluid flow, heat transfer, chemical reactions etc by solving mathematical equations with the help of numerical analysis this resolves the entire system in small cells and apply governing equations on these discrete elements to find numerical solutions regarding pressure distribution, temperature fields [16]. This software can also build a virtual prototype of the system or device before can be apply to real-world physics to the model, and the software will provide with image and data, which predict the performance of that design.

ANSYS CFX pre-processor, the fluid domain is defined. The details of the domain created with the corresponding fluid-fluid interface is provided in Table 1. The flow in this study is turbulent, the boundary conditions are specified in ANSYS CFX pre-processor and then the file is exported to the ANSYS CFX solver. The same procedure is adopted for every case studies.

4. Results and Discussion

Table 2: Shell and Tube Outlet Temperature for Staggered and Aligned Tube with Perpendicular and Parallel Cut Baffles Arrangement.

Mass Flow (kg/hr)	Baffles Number	Staggered Tube Arrangement				Aligned Tube Arrangement			
		Perpendicular Cut Baffles		Parallel Cut Baffles		Perpendicular Cut Baffles		Parallel Cut Baffles	
		Shell Out ^0C	Tube Out ^0C	Shell Out ^0C	Tube Out ^0C	Shell Out ^0C	Tube Out ^0C	Shell Out ^0C	Tube Out ^0C
12500	6	64.55	40.09	64.38	39.62	64.35	41.39	64.11	40.86
	8	64.6	39.02	64.04	42.56	64.45	39.36	64.11	40.13
	10	64.08	44.57	64.01	43.63	64.05	45.07	63.96	44.44
	12	64.46	41.12	64.76	43.19	64.06	44.58	63.98	45.23
17500	6	68.7	44.64	68.63	44.21	68.63	45.84	68.51	45.31
	8	68.6	44.27	68.54	46.57	69.65	43.54	68.53	44.45
	10	68.37	48.19	68.36	47.39	68.46	48.69	68.39	48.4
	12	68.72	45.36	68.25	48.5	70.15	45.42	68.39	46.25
22500	6	71.19	47.57	71.11	47.17	71.04	48.45	70.99	48.35
	8	71.1	46.47	70.92	49.26	70.25	47.87	70.84	47.77
	10	71.03	50.53	70.9	49.88	70.39	51.06	70.92	50.04
	12	71.16	47.79	70.82	50.7	71.45	49.4	70.84	49.88

From the CFD simulation results, for fixed shell inlet temperature, shell and tube side outlet temperature values for varying Perpendicular and Parallel cut baffles with different shell side inlet fluid flow rates are provided in table.

According to the data table, the lowest shell and tube outlet temperature found for the mass flow rate at 12500 kg/hr and highest shell and tube outlet temperature found for the mass flow rate at 22500 kg/hr.

However, the inlet shell mass flow rate at 17500 kg/hr with staggered tube perpendicular cut baffles and aligned tube parallel cut baffles provide a good agreement with desired outlet temperature.

4.1. Overall Heat Transfer Coefficient Analysis with Baffles number:

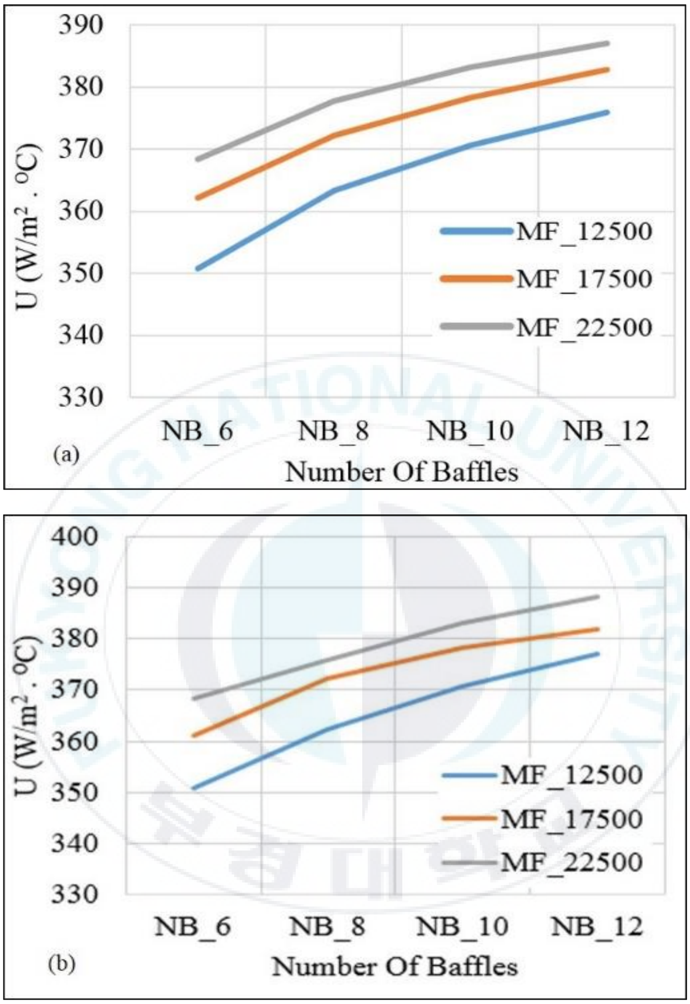

Figure 13: Number of Baffles effect on overall heat transfer co-efficient (a) Staggered (b) Aligned tube arrangement.

It can be observed that from the following curve, the overall heat transfer co-efficient will be increased because as the baffles number is increased at both staggered and aligned tube arrangement. The baffles number will be increased as the baffle spacing is decreased. So the contact time is increased between the shell and tube side fluids which leads to increase the overall heat transfer.

4.2. Variation of Pressure Drop with mass flow rate

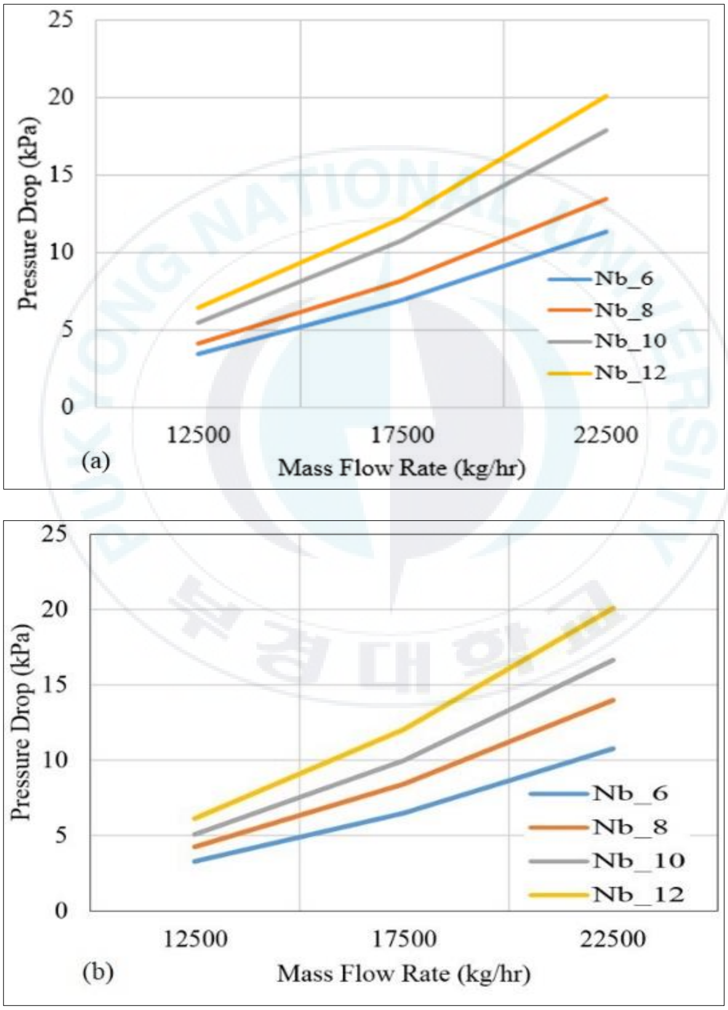

Figure 14: Pressure drop at shell side (a) staggered tube perpendicular cut baffles (b) staggered tube parallel cut baffles

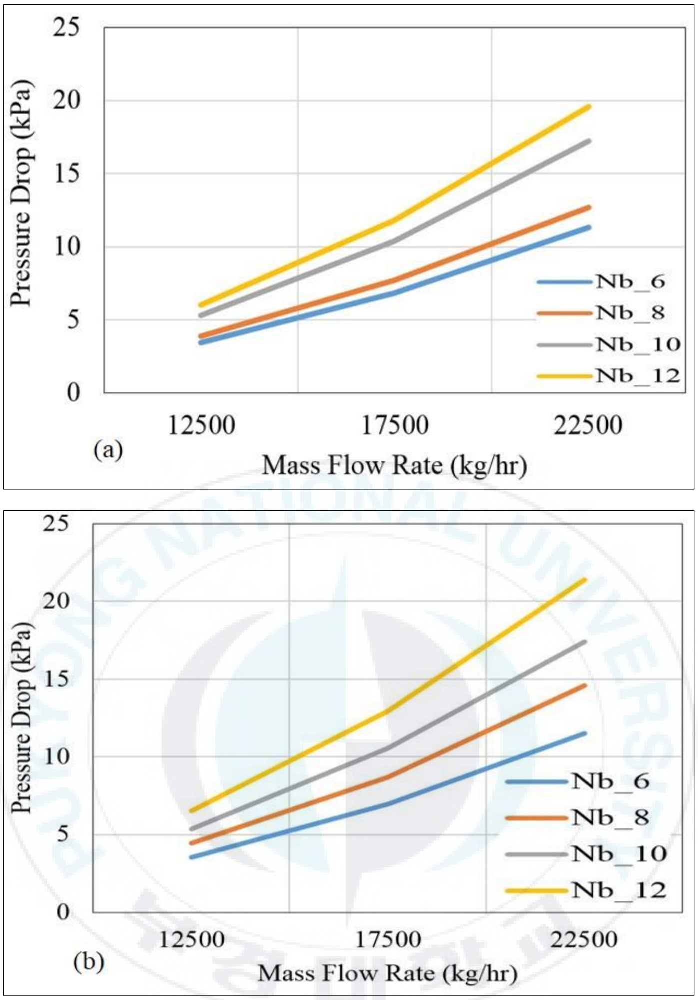

Figure 15: Pressure drop at shell side (a) aligned tube perpendicular cut baffles (b) aligned tube parallel cut baffles

Pressure drop increases linearly with respect to the mass flow rate as well as with number of baffles increases. From all the pressure drop figures, the highest pressure drop is founded at aligned tube parallel cut baffles while the lowest pressure drop at aligned tube perpendicular cut baffles.

To compare the staggered tube pressure drop, the lowest pressure drop is founded at staggered tube parallel cut baffles.

4.3. Analysis of Velocity and Streamline on Shell Side with Different Number of Baffles:

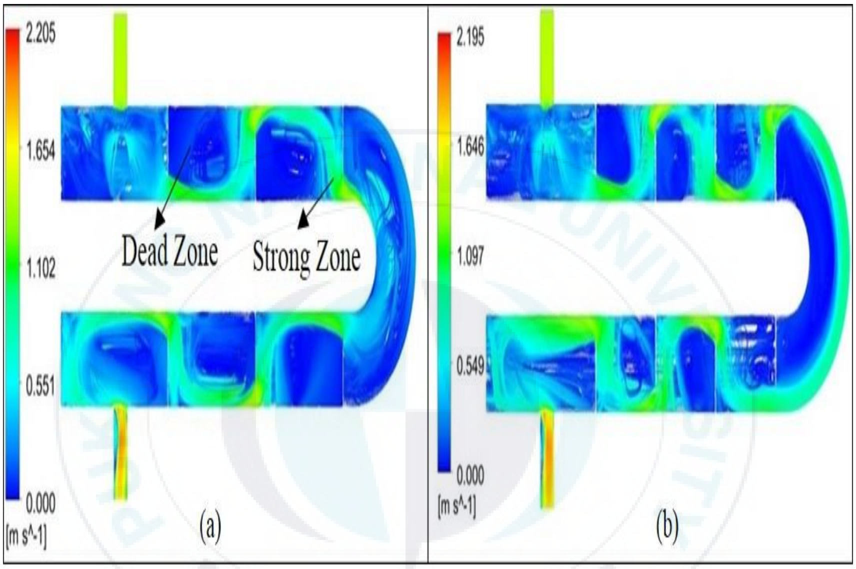

Figure 16: Velocity and Streamline in Shell Domain for perpendicular (a) 6 Baffles (b)8 baffles

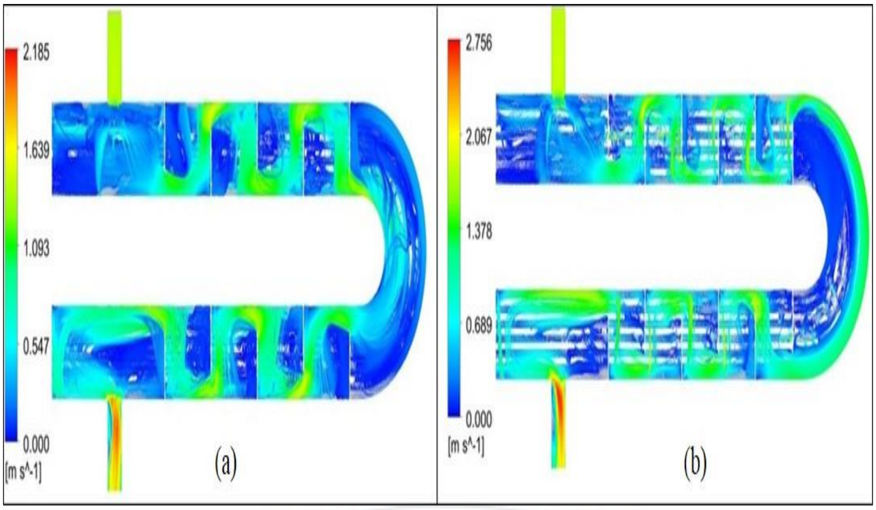

Figure 17: Velocity and Streamline in Shell Domain for perpendicular (a) 10 baffles (b) 12 baffles

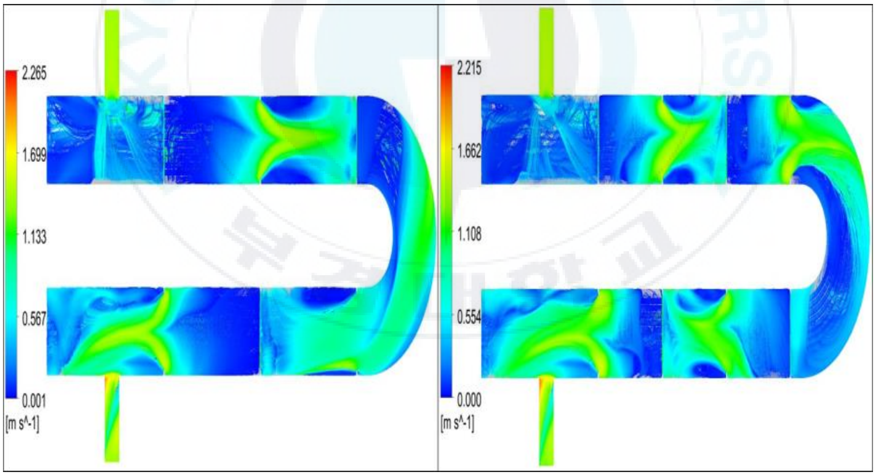

Figure 18: Velocity and Streamline in Shell Domain for parallel (a) 6 baffles (b) 8 baffles

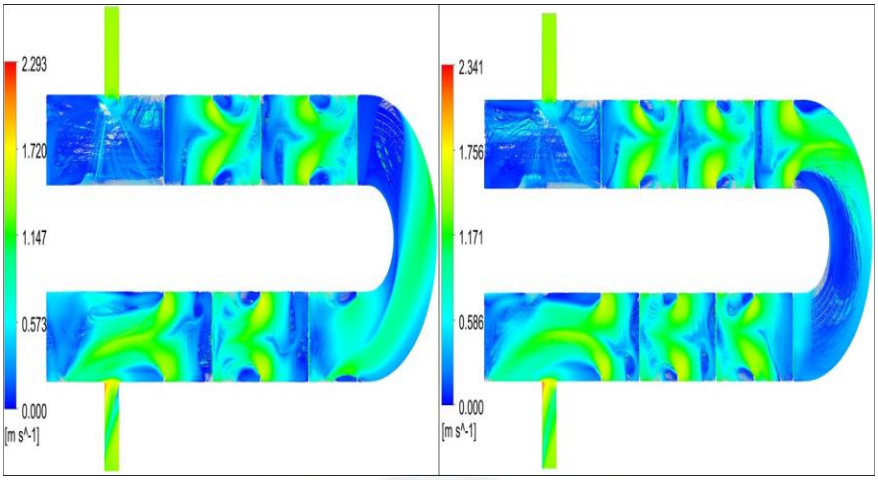

Figure 19: Velocity and Streamline in Shell Domain for parallel (a) 10 baffles (b) 12 baffles

Figures 16 indicate the two zones – (i) Strong Zones (gap between shell cover and baffles) (ii) Dead Zones. Strong has high velocity while the dead zone is almost zero velocity.

The dead zones have low heat transfer coefficients, because they can not transfer exchange the heat with main flow stream. They active zones can exchange the heat with main flow steam quickly.

4.4. Temperature Distribution on Tubes Wall

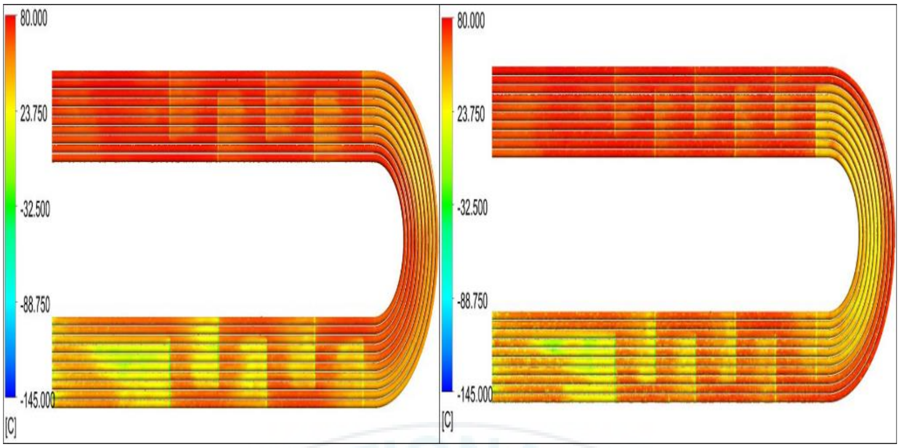

Figure 20: Temperature distribution on staggered tube arrangement for shell side (a) 6 baffles (b) 8 baffles

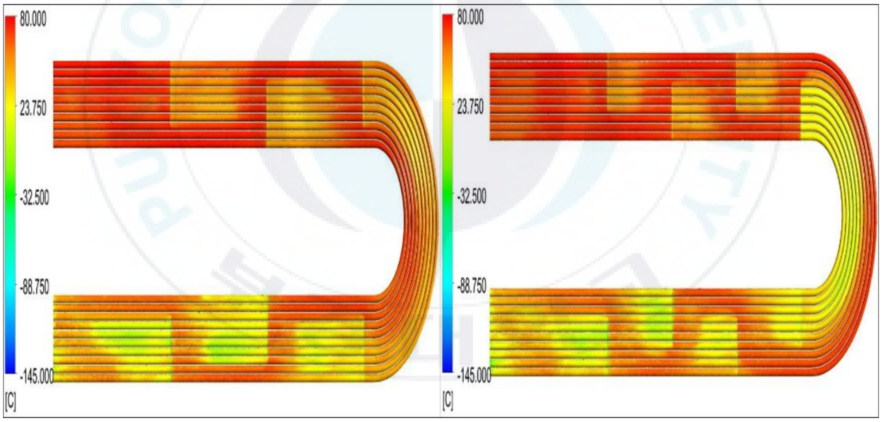

Figure 21: Temperature distribution on staggered tube arrangement for shell side (a) 10 baffles (b) 12 baffles

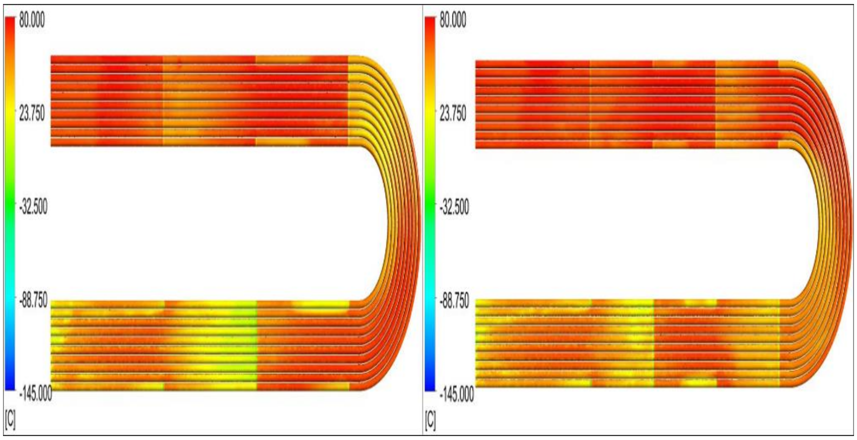

Figure 22: Temperature distribution on aligned tube arrangement for shell side (a) 6 baffles (b) 8 baffles

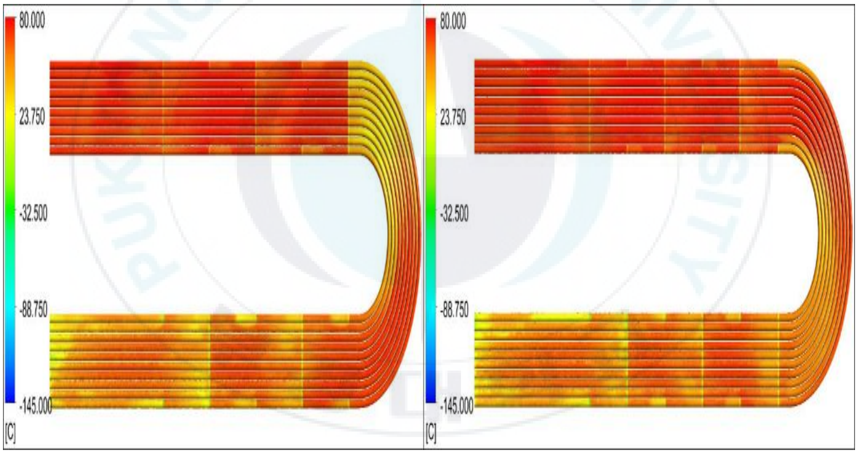

Figure 23: Temperature distribution on aligned tube arrangement for shell side (a) 10 baffles (b) 12 baffles

In figures 22 to 23, the tubes wall temperature of both staggered and aligned tube arrangement system is increased due to the increases of baffles number. Compared the both tube arrangement system, the highest and lowest temperature found at 10 and 6 number of baffles arrangement system respectively.

4.5. Baffles Cut Performance Analysis

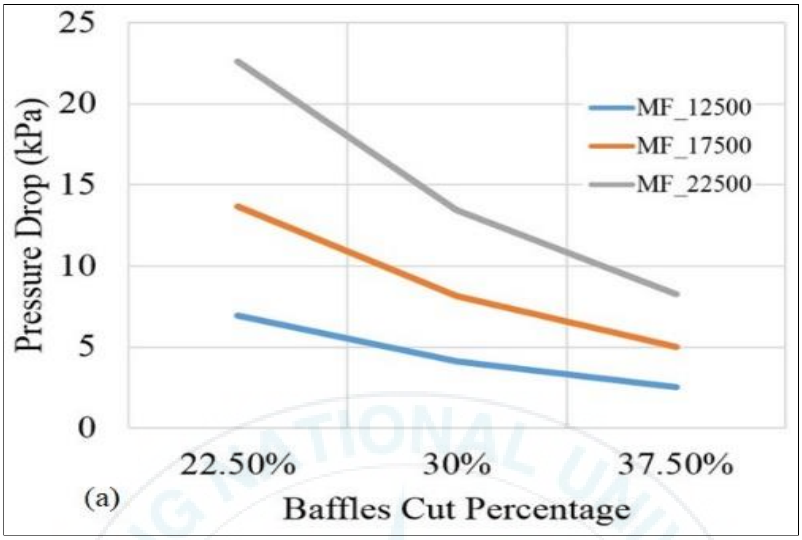

Figure 24: Baffles Cut Percentage Vs Shell side pressure drop

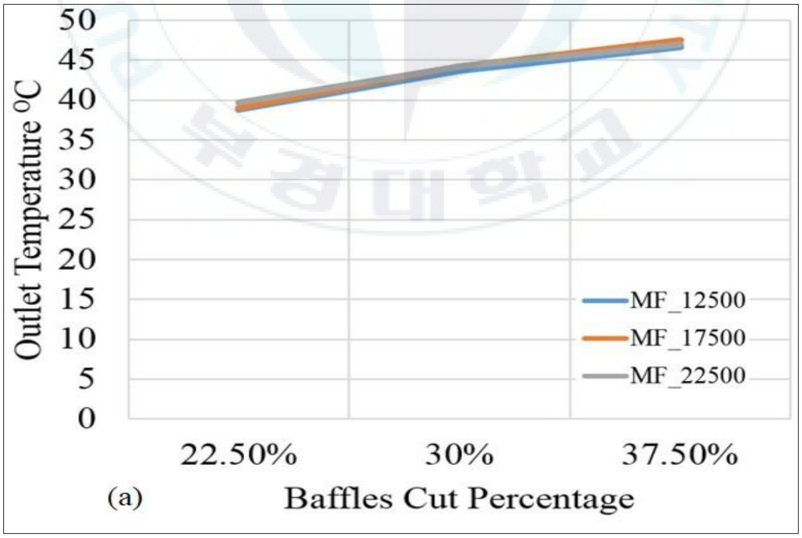

Figure 25: Baffles Cut Percentage Vs Tube side outlet temperature.

The pressure drop and outlet temperature analysis at shell side of the heat exchanger for changing the baffles size. To observe the results, when the baffles cut percentage increase the pressure drop is linearly decreased at shell side. During the temperature analysis the baffle cut 30% results has good agreements compare the require temperature of this analysis work.

5. Conclusions:

The STHE is modeled in detail to resolve the flow and temperature fields.

(a) Three different mass flow rate and three different inlet temperature conditions are considered.

- 22000 kg/hr glycol water inlet ……… 90 ^{O}C glycol water inlet
- 17500 kg/hr glycol water inlet ……… 80 ^{O}C glycol water inlet
- 12000 kg/hr glycol water inlet ……… 70 ^{O}C glycol water inlet

Among them, only 17500 kg/hr and 80 ^{O}C can heat the LNG at 45 ^{O}C

(b) The two types of tube arrangement system are studied at same conditions. The staggered tube arrangement shows better performance compared to the aligned tube arrangement system.

(c) For increasing the contact time between two side fluids, the simulations are carried out by the different baffles number (6, 8, 10 and 12) with perpendicular and parallel cut baffles system. The staggered tube with perpendicular 8 number of baffles cut has good agreement with manually calculated results.

(d) Three different baffles cut 22.5%, 30%, and 37.5% are analyzed by the staggered tube with perpendicular 8 number of baffles cut. The results show, for 30% baffle cut obtains the desired temperature at tube outlet position.

References:

1 Kumar S, kawon HT, Choi KH, Lim WS, Cho JH, Cho KJ, et al, LNG: "An eco-friendly cryogenic fuel for sustainable development". Apple Energy 2011; 88(12); 4264-73.

2 Wursig G. Technical outlook and future development of LNG as ship fuel. In: Proceeding of Naples Shipping Week 2014;20(2):226-37

3 Uwitonze H, Han S, Jangryeok C, Hwang KS. Design process of LNG heavy hydrocarbons fractional: low LNG temperature recovery. Chem Eng Procss Intesif 2014;85:187-65.

4 Nakaiwa M, Akiya T, Owa M, Tanaka Y. Evaluation of an energy supply system with air separation. Energy Converts Manag 1996:37(3):295-301.

5 TUO H, Li Y, Tan H. Combined cycle of air separation and natural gas liquefaction. Huagong Xuebao/J Chem Ind Eng China 2008;59 (10): 2498-504.

6 Man-Jin Jung, Joseph H. Cho, ph.D., and Wungsang Ryu. LNG Terminal design feedback from operator's practical improvements.

7 P.N.Sapkal, P.R.B., M.J.Sable, S.B.Barve To optimise air preheater design for better performance. in new aspects of fluid mechanics, heat transfer and environment. 2010. Taipei: World Scientific and Engineering.

8 Sandeep K. Patel, Professor Alkesh M. Mavani. Shell and tube heat exchanger thermal design with optimization of mass flow rate and baffle spacing. Patel et al, International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974.

International Gas Union – World LNG Report in 2015.

9 Jobil Joy, Kanchan Chowdhury. Study of intermediate fluid selection in indirect contact ambient air heater based LNG regasification systems in receiving terminals. ICEC 26- ICMC 2016.

10 Eldhose Lomy, Nidheesh P. Optimization of Shell and Tube Heat Exchanger for Tube Arrangements. IJIRSET, ISSN : 2319-8753.

11 Vindhya Vasiny Prasad Dubey, Raj Rajat Verman, Piyush Shanker. Performance Analysis os Shell and Tube Type Heat Exchanger under the Effect of Varried Operating Conditions. IOSR_JMCE, ISSN : 2278-1684. May-Jun.2014.

12 Swarup S Deshpande, Shreeniket A Hinge. Design and Performance Study of Shell and Tube Heat Exchanger with Single Segmental Baffle Having Perpendicular and Parallel-Cut Orientation. IJERT, ISSN : 2278-0181 vol. 3 Issue 11, November-2014

13 Zeinab Sayed Abdel-Rehim, Heat Transfer Charateristics and Fluid Flow past Staggered Flat-Tube Bank Using CFD. World Academy of Science, Engineering and Technology. Vol :7, No : 10,2013.

14 J.P Holman: Heat Transfer 10th edition McGraw-Hill Book Co, New York, NY (10020), pp. 521-567

15 Frank P. Incopera, David P. Dewitt, Theodore L. Bergman and Adrienne S. Lavine: Fundamental of Heat and Mass Transfer 7th edition John Wiley & Sons, Inc.,111 River Street, Hoboken,NJ ,pp 705-748

16 Khairun Hasmadi Othman, CFD Simulation of Heat Transfer In Shell and Tube Heat Exchanger, University Malaysia Pahang April 2009.