Journal of Mechanical Engineering Research and Developments
ISSN: 1024-1752
CODEN: JERDFO
Vol.43, No.5, pp. 332-345
Published Year 2020

Three Dimensional Study of Baffles Effect on Heat Transfer in Shell and Tube Heat Exchanger

Muna S. Kassim†, Ahmed Oleiwi‡, Dhamyaa Saad Khudhur‡†, Laith Jaafer Habeeb‡‡*

†Al-Mustansiriyah University, College of Engineering, Iraq-Baghdad,
‡University of Warith Alanbiyaa, College of Engineering, Iraq-Karbala,
‡†Al-Mustansiriyah University, College of Engineering, Iraq-Baghdad,
‡‡University of Technology, Training and Workshops Center, Bagdad, Iraq

* Corresponding Author Email: laithhabeeb1974@gmail.com

ABSTRACT: This work is based designing and performance evaluation of a shell and tube heat exchanger using ANSYS package. Heat exchanger possesses a many usages of different activities. In this research one of the shell and tube heat exchangers is taking the consideration. At the current design process, the water is used both shell and tube sides. The modeling of the heat exchanger without and with baffle plates was carried out to study their effect on the device performance. The work domain is to calculate the distribution of temperature into both of shell and tube side, and to compute the rate of transfer as well as the effectiveness of both configurations. The flow and temperature fields inside the shell and tube are resolved by a FLUENT. It is shown that using baffles can significantly improve the device performance and increase the total heat transfer rate. the heat exchanger effectiveness is increased from 0.68 to 0.76 after using the baffle plates around the tubes. Plate baffle increased a turbulence of a shell fluid and minimized tube-to-tube temperature variances and thermal stresses by reason of the cross flow

KEYWORDS: shell and tube heat exchanger, CFD, overall thermal performance.

INTRODUCTION

In the heat exchanger energy is transported from one to another fluid over a solid surface via conduction and convection. as mentioned, heat exchangers are used in different industries such as nuclear reactors, refrigeration system, power plants, chemical processing, automotive industrial, systems of recovering of heat, industries of food, and system of air conditioning. (STHE) shell and tube heat exchanger include tubes pack surrounded in the cylindrical shell one fluid move through a tubes and second fluid passes between shells and tube. Shell and tube heat exchangers have main devices as the shell and shell-side nozzles, tubes, the tube side channels and the nozzles, the channel covers, baffles, tube sheets, and pass divider etc. More normally utilized STHE have great ratios of surface area of heat transfer to volume to give high efficiency of transfer of heat in comparison to the others. Heat exchangers size plays a significant role for in cost optimization. Likewise, heat exchangers effectiveness and efficiency are considering important parameter during selection of industrial heat exchangers. Many ways of heat transfer improvement had been studied over many years so as to achieve best cost with good efficiency [1].A baffle element of STHX displays so important roles, for example the shell side fluid distributing and packs of help tube. A STHX can be classified to three groups according to shell side fluid flow path: the helical flow, the longitudinal flow and the transverse flow. The heat transfer characteristics in the STHX shell side vary in accordance to of unlike flow, which have a cumbersme influence on the heat exchangers work [1].**G. SesiBhusana Rao and S. Suresh Babu**[2]performed the numerical study of the tube and shell heat exchanger through different glycerin features. When concentration of glycerin is 20%, will supplied extreme rates of transfer of heat and the lesser drops of pressure that is what conclude from that study. **ShuiJi** et al. [3] introduced numerical investigation on double shell-pass shell-and-tube heat exchanger with continuous helical baffles. A numerical results indication that a shell-side factors of transfer of heat of a new heat exchanger around (14–25) % and (12–17) % more than the single shell-pass STHXCH and STHXSG, respectively; a dropping of pressure of shell-side of a modern heat exchanger was slightly smaller than of STHXSG and approximately (29–35) % superior than of the STHXCH single shell-pass. **Yingshuang Wang** et al. [4] performed experimental study of the tube and shell heat exchanger with then oval type baffles. The work of procedure for the two heat exchangers are comparing too. Beneath the same cases, a total work of a new design is (20–30) % more effective than that of a heat exchanger of segmental baffle that as results suggest. **HalilBayram** et al. [5] introduced numerical study of an influence of alteration of spaces of baffle on a thermal performance of tube and shell heat exchanger. The alteration of space with the space pattern of centered baffle can be proposed an alternate shell side structure design compared to an identical space scheme of baffle. The objective of this study is to improve a STHX without and with baffles and find the baffles influence on the heat exchanger work, which is what obtained from numerical results.

RESEARCH METHODOLOGY

Computational fluid dynamics (CFD) begins to study a system with a desired geometry construction and the net for design considerations. In general, geometry has been making simpler for studies of computational fluid dynamics. The meshing is the division of a field for the minor sizes where calculations have been excuted via iterative ways assistance. The design begins by the description of the primary provisions and a border for the supremacy, then total system designing.

The GEOMETRY

An ANSYS design unit of workbench has been used to compose heat exchanger as shown in figure (1) through (3).

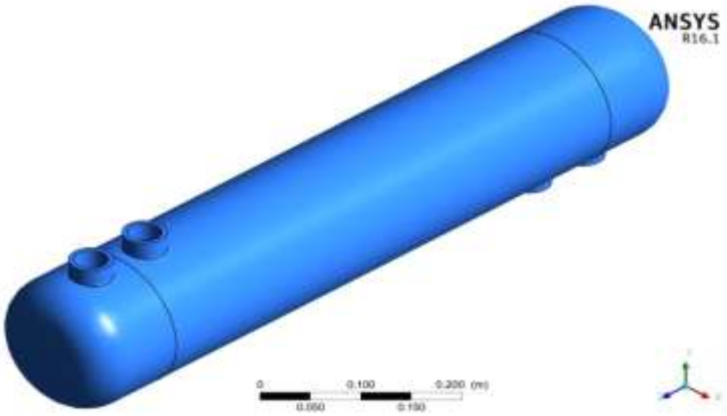

Figure 1. Heat exchanger.

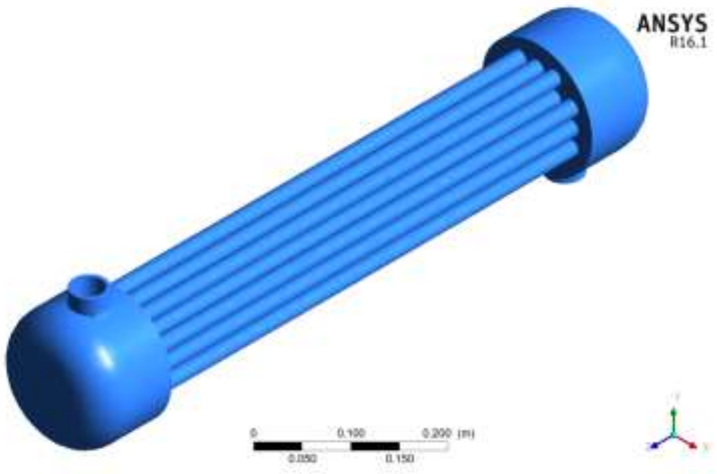

Figure 2. Heat exchanger tubes

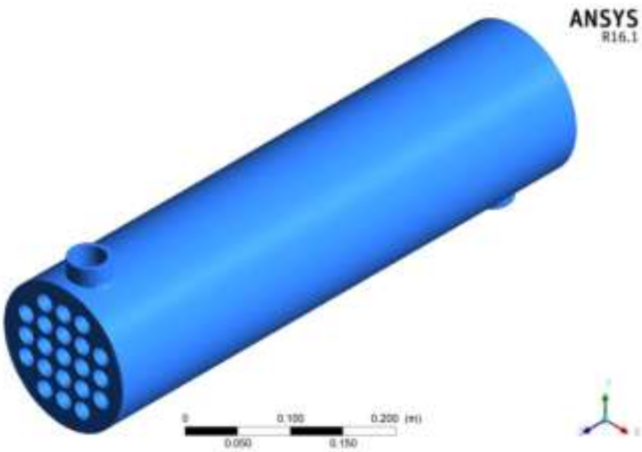

Figure 3. Heat exchanger shell.

The dimensions of the heat exchanger are shown in figure (4) and listed below.

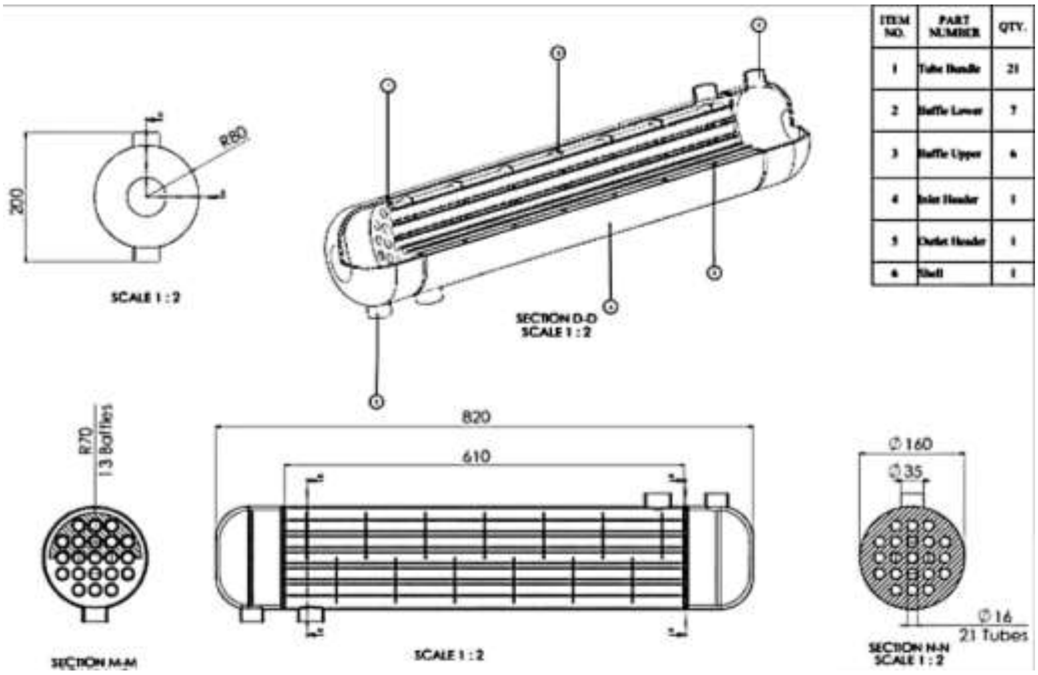

Figure 4: Dimensions of the shell and tube heat exchanger.

The specification of heat exchanger is given in table (1)

Table 1. Heat exchanger specifications

No of tubes = 21	Tubes length = 610 mm	Thickness = 2 mm
Shell length = 820 mm	Tube diameter = 16 mm	No of upper baffles = 7
No of lower baffles = 6	Shell diameter = 160 mm	

MESHING

Mesh includes mixture of cells such as (Hexahedral and Tetra cells) possessing together faces of quadrilateral and triangular at the borders. An optimum grid size of a mesh has been chosen by testing different grid size. The mesh of the tubes side and shell side are shown in figure (4) to through figure (6)

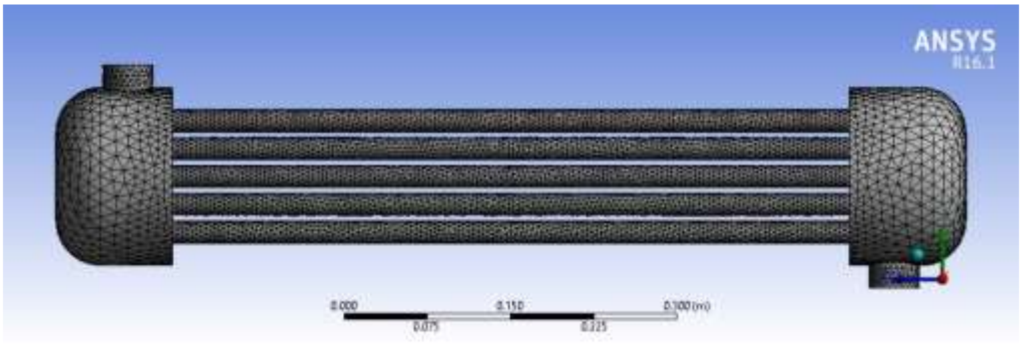

Figure 4. Tube side mesh (without baffles).

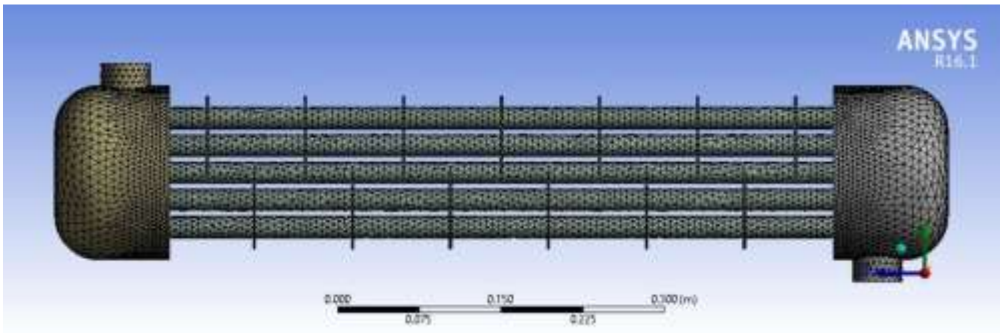

Figure 5. Tube side mesh (with baffles).

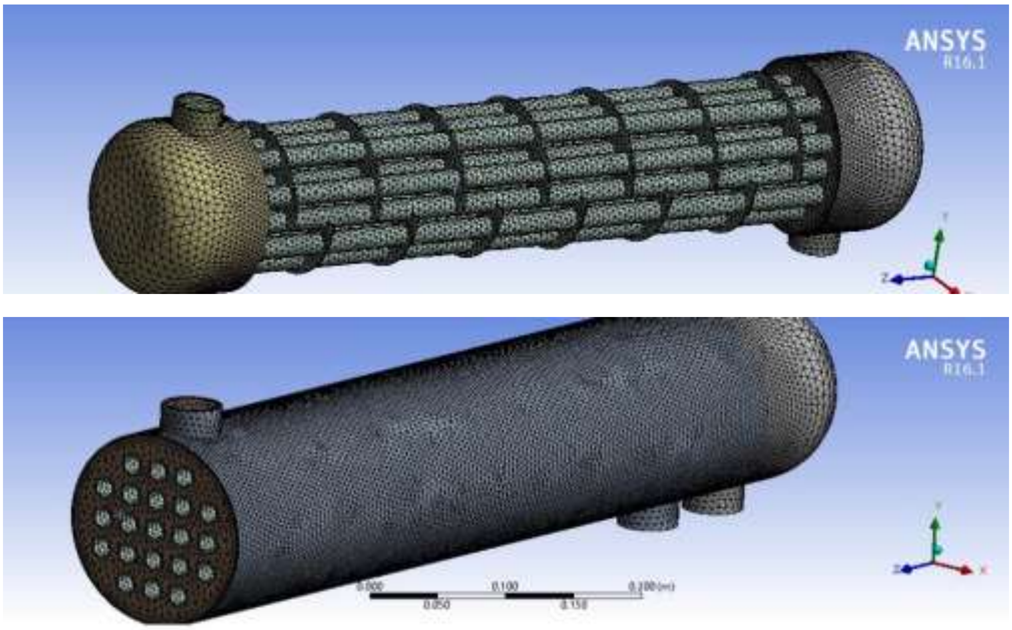

Figure 6. Mesh for the shell and tube side.

NAMED SELECTION

A variable surfaces of a solid have been named according to require exits and entrances for exiting and entering of fluids as shown in figure (7). Insulation surfaces named for the outer wall.

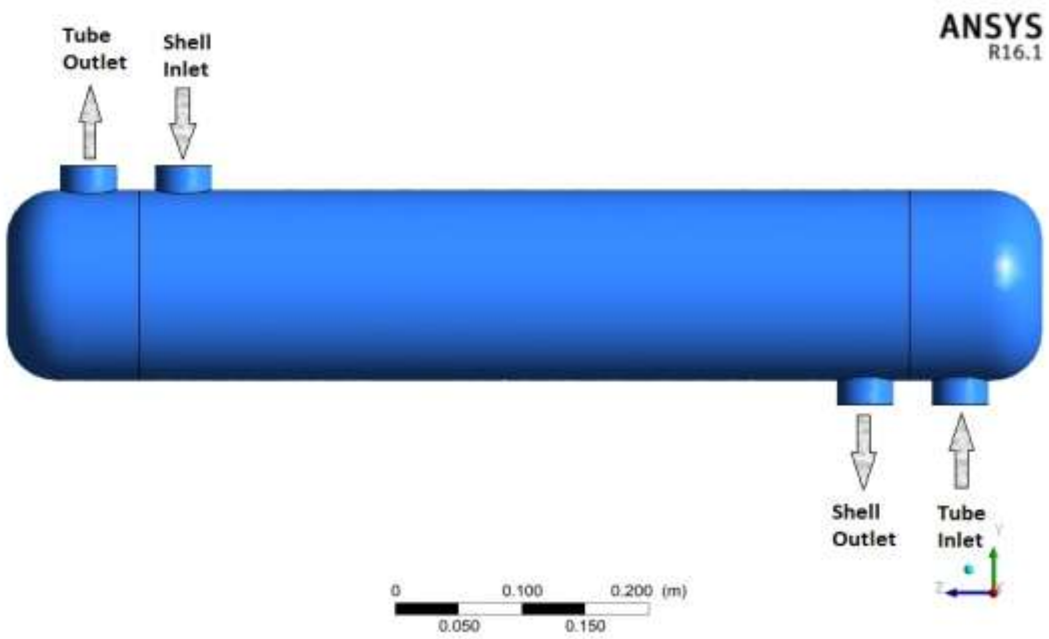

Figure 7. Locations of device inlets and outlets

PROBLEM SETUP

The mesh is tested and quality was gotten. Analysis type was altered according to type of pressure based. A formularization of velocity was altered to absolute and steady state for time. Gravity was known for example $y = -9.81\ m.s^{-2}$;The energy was fixed to on place. The viscid typical was a selection of as design of "k-ε (two mathematical equations).

Basic assumptions:

a) Neglected the outer wall thickness in order to simplify the numerical solution.
b) Heat transfer conditions were assumed steady state.
c) Constant fluid property by incompressible fluid.
d) Neglected the natural convention and Radiation.
e) The heat transfer between the two fluids was considered conjugate.

The governing equations

The equation of governing for the energy, momentum, and the continuity to the turbulent flow in the present heat exchanger was termed in the next parts [6]:

The Continuity Equation

$$\frac{\partial wp}{\partial z} + \frac{\partial up}{\partial x} + \frac{\partial \rho}{\partial t} + \frac{\partial vp}{\partial y} = 0$$

The continuity equation for steady state, incompressible flow can be written as:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \qquad (1)$$

Momentum Equation

$$\frac{\partial}{\partial x_j}\left(\rho u_i u_j\right) = -\frac{\partial P}{\partial x_i} + \mu \frac{\partial}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) + \frac{\partial}{\partial x_j}\left(-\rho \underline{u'_i u'_j}\right) \qquad (2)$$

Where:

$\left(-\rho\,\underline{u'_i u'_j}\right)$ Represent a stress term of Reynold number which is related through the gradients of local velocity and the turbulent viscosity (μ_t).

$$\left(-\rho\,\underline{u'_i u'_j}\right) = \mu_t\left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right] \tag{3}$$

ENERGY EQUATION

$$\rho c_p \frac{\partial u_i T}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\lambda \frac{\partial T}{\partial x_i} - \rho\,\underline{u'_i T'}\right) \tag{4}$$

THE EQUATION OF TRANSPORT FOR THE TYPICAL OF RNG K-E

A k-εmodel obtainable via **FLUENT** contain the realizable models, the renormalization collection of (RNG), and standard. In a current research, a model of RNG k-ε had been utilized, as its provides properties make it dependable and truer for the broader type of flow:

- A design of RNG possess the extra term in it is mathematical equation that rises the exactitude for quickly stressed streams.
- Swirl influence on the turbulence was involved in model of RNG, improving exactitude for the swirling streams.
- A theory of RNG affords the methodical formulation for the turbulent Prandtl numbers, whereas the other models utilized fixed standards for it.

$$\frac{\partial}{\partial x_i}(\rho u_i k) + \frac{\partial}{\partial t}(k\rho) = \frac{\partial}{\partial x_j}\left(\mu_{eff}\alpha_k \frac{\partial k}{\partial x_j}\right) + G_b + G_k - \varepsilon\rho \tag{5}$$

And

$$\frac{\partial}{\partial t}(\varepsilon\rho) + \frac{\partial}{\partial x_i}(\rho u_i \varepsilon) = \frac{\partial}{\partial x_j}\left(\mu_{eff}\alpha_\varepsilon \frac{\partial \varepsilon}{\partial x_i}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}(G_b C_{3\varepsilon} + G_K) - \rho C_{2\varepsilon}\frac{\varepsilon^2}{k} - R_\varepsilon \tag{6}$$

α_ε and α_k represent constants of equation and have value of 1.393 .Whereas$G_K$characterizes the turbulence kinetic energy generation due to gradients of mean velocity and can termed such as following:

$$G_K = \mu_t S^2 \tag{7}$$

Where:

$$S = \sqrt{\frac{1}{2}\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right)} \tag{8}$$

G_b: The turbulence kinetic energy generation due to resilience, it can be calculated as:

$$G_b = g_i \beta \frac{u_t}{Pr_t}\frac{\partial T}{\partial x_i} \tag{9}$$

A turbulent viscosity μ_t is computed by combining k and **ε** as followsε:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{10}$$

The extra term R_ε in ε equation is written as:

$$R_\varepsilon = \frac{C_\mu \rho \eta^3\left(1 - \frac{\eta}{\eta_0}\right)}{1 + \beta + \eta^3}\frac{\varepsilon^2}{k} \tag{11}$$

Where:

$$\eta = \frac{S}{\varepsilon} \tag{12}$$

$\beta = 12*10^{-3}$, $\eta_0 = 4.380$

The model has values of constants as:

$G_{2\varepsilon} = 1.680, C_{\mu} = 845 * 10^{-4}, C_{1\varepsilon} = 1.420$

BOUNDARY CONDITIONS

According to the desired model, the provisions of boundary have been utilized. The conditions were entrance of mass flow as inlet and exiting of pressure as outlet. There are two exits and two entrances in this is heat exchanger of the shell and tube. Walls were separately definite with specific settings of border. For other walls not any condition of mistake is measured. Except the walls of shell, other walls are fixed to zero condition of heat flux. A table (2) shows the boundary conditions information.

Table 2. Boundary Conditions details

	Boundary conditions	**The Rate of Mass Flow**	**Turbulent Kinetic Energy**	**Turbulent Rate of Dissipation**	**Temperature**
Tube inlet	Inlet Mass-flow	0.01 kg/s	$1*10^{-2}$ $m^2.s^{-2}$	$1*10^{-1}$ $m^2.s^{-3}$	350 K
Tube outlet	outlet of Pressure	-	-	-	-
Shell inlet	Mass-flow inlet	0.02 kg/s	$1*10^{-2}$ $m^2.s^{-2}$	$1*10^{-1}$ $m^2.s^{-3}$	300 K
Shell outlet	outlet of Pressure	-	-	-	-

SOLUTION MODELS

A solution models are selected as follows:

- Simple for the scheme
- Least square cell based to the gradient
- Standard for the pressure
- 2^{nd} order upwind for the momentum
- 2^{nd} order reversing for the turbulent kinetic energy
- 2^{nd} order reversing for the turbulent rate of dissipation

INITIALIZATION AND SOLUTION CONTROL

The parameters for beneath relaxation parameters are;

- The density = 1 $kg.m^{-3}$
- The momentum = 0.70 $kg\text{-}m.s^{-1}$
- The body forces = 1 $kg.m\text{-}^{2}s^{-2}$
- The pressure = 0.30 Pa
- The turbulent kinetic = 0.80 $m^2.s^{-2}$

CONVERGENCE ESTIMATIONN

It is tried to have a better convergence through a model of simulation and therefore standards is put firm in order to obtain true outcome. Table (3) shows the variables of residuals as follows:

Table 3. Variable of Residuals

Variable of Residuals	Residual
z-component for velocity	$0.1*10^{-5}$
y-component for velocity	$0.1*10^{-5}$
x- component for velocity	$0.1*10^{-5}$
The energy	$0.1*10^{-8}$
Turbulent kinetic	$0.1*10^{-4}$
The specific dissipation energy/ dissipation energy	$0.1*10^{-4}$
The continuity	$0.1*10^{-5}$

RUN CALCULATION

The number of repetition is set to five hundred, the solution has been calculated and numerous contours, vectors and plots are gotten.

RESULTS AND DISCUSSIONS

The obtained results are characterized by different outline of temperature and velocity vectors as shown next. Figure (10) and figure (11) show the temperature outline plots for a shell side fluid in addition to the tube side fluid of heat exchanger without baffles. It is clearly visible that the hot water in the tubes is being cooled by the cold water in the shell.

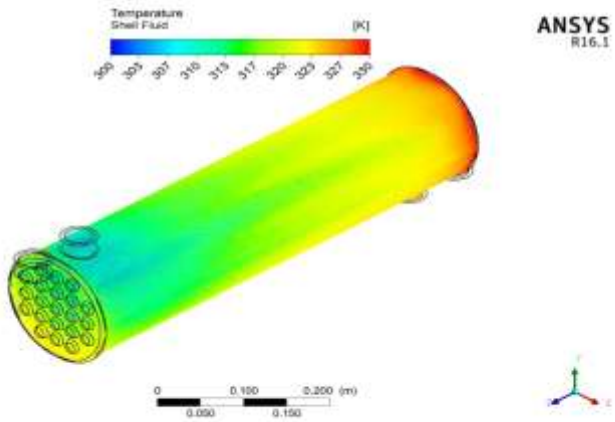

Figure 10. Three-dimensional temperature contours of the shell side fluid (without baffles).

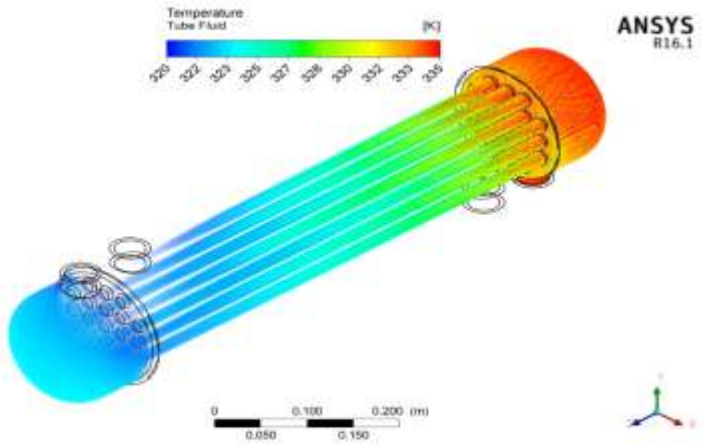

Figure 11. Three-dimensional temperature contours of the tube side fluid (without baffles).

However, the temperature distribution can be better visualized when depicting the 2-D contour plot along the center of the whole facility as displayed in figure (12). A hot water enters a tube inlet at temperature of about 350 K, while the cooled fluid enters a shell inlet on about 300 K. A hot fluid temperature continuously decreases as it flows inside the tube bundles until it leaves the device at a temperature of about 316 K.

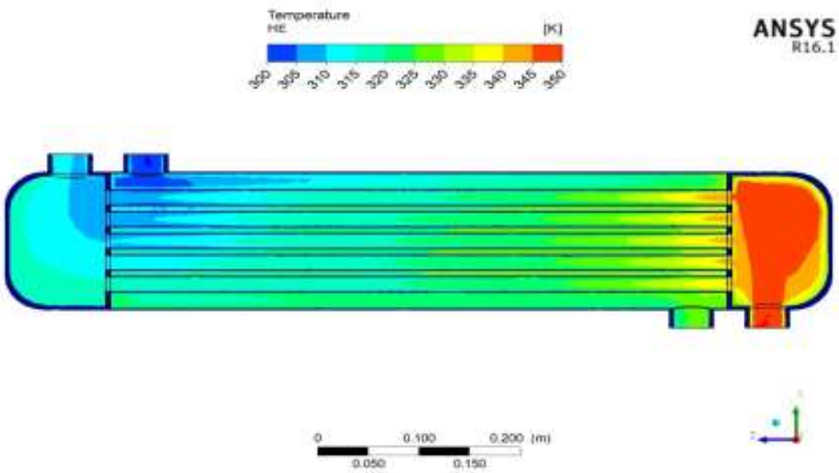

Figure 12. Two-dimensional temperature contours (without baffles)

Figure (13) shows the fluid velocity streamlines inside the shell of a heat exchanger without baffles. It can be observed that a fluid is moving in straight direction along the tubes until existing the device from the shell outlet.

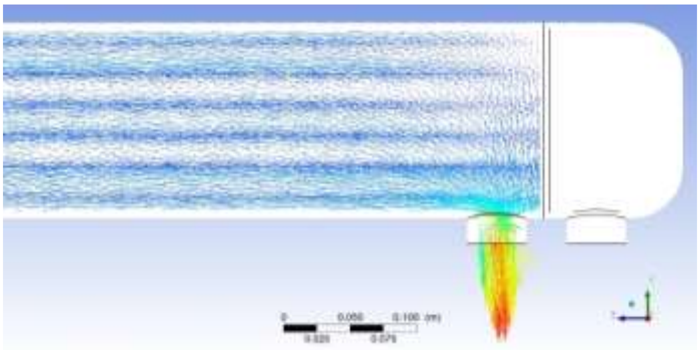

Figure 13. Fluid streamlines inside the shell (without baffles)

Similar trend can be seen when depicting the 3-D contour plots of the heat exchanger with baffles as shown in figures (14) and (15).

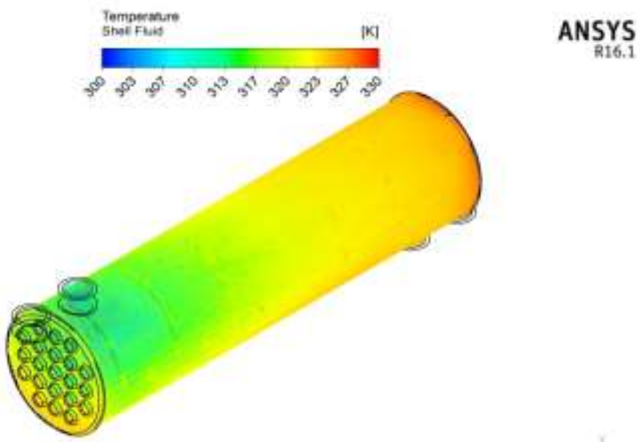

Figure 14. Three-dimensional temperature contours of the shell side fluid (with baffles).

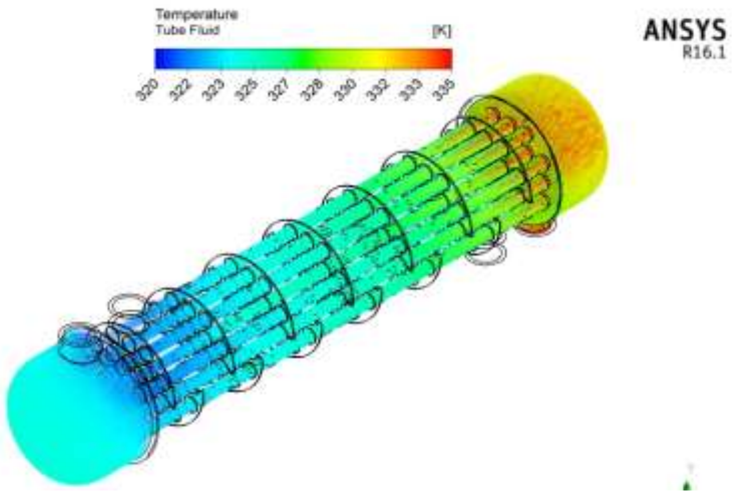

Figure 15. Three-dimensional temperature contours of the tube side fluid (with baffles).

It is also visible that the hot water in the tubes is being cooled by the cold water in the shell. However, when examining the 2-D contour plot along the center of the whole facility as displayed in a figure (16), a hot water enters a tube inlet at temperature of about 350 K, while its temperatures continuously decreases as it flows inside the tube bundles until it leaves the facility at a temperature of about 312 K. This value is lower when compared to the heat exchanger without baffles. The exact temperature values recorded from the CFD results are presented in table (4) and table (5).

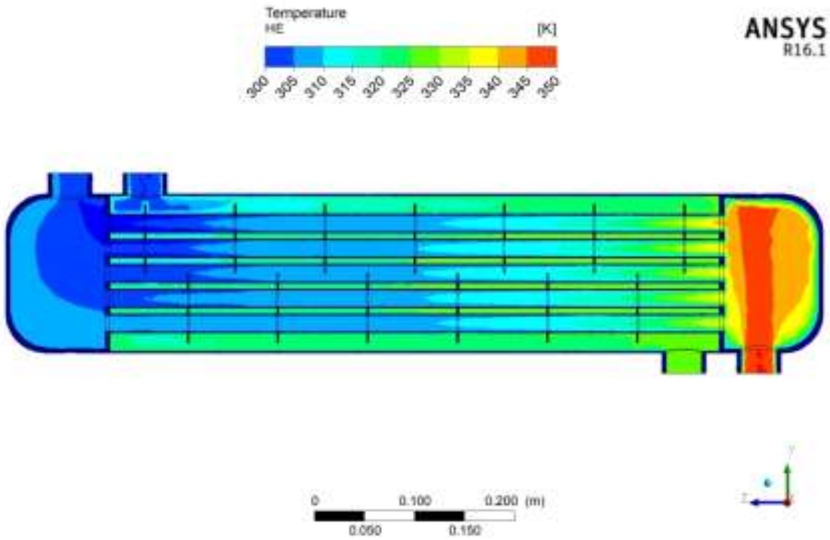

Figure 16. Two-dimensional temperature contours (with baffles).

Figure (17) shows the fluid velocity streamlines inside the shell of the heat exchanger with baffles. It can be seen that the baffles have caused the fluid to move upwards and downwards around the plates while generating vortices inside the shell of the device.

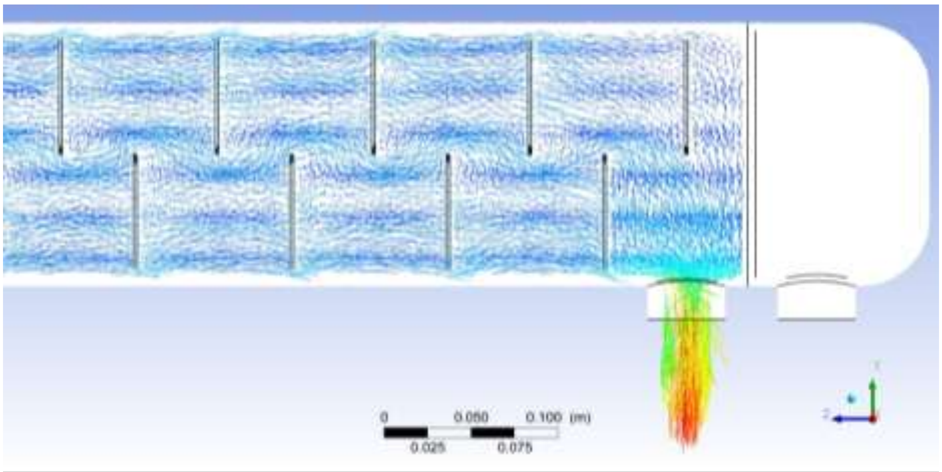

Figure 17. Fluid streamlines in the shell (with baffles).

Effectiveness of Heat Exchanger

The heat release and effectiveness of the shell and tube heat exchangers may be obtained from the overall energy balance [7]

$q = \dot{m} Cp \, \Delta T$

Where,

Cp = specific heat = 4182 (J. $kg^{-1}.k^{-1}$)

$\dot{m}$ = rate of mass flow (kg.s^{-1})

ΔT = difference of temperature (K)

For heat exchanger without baffles, inlet and outlet temperatures recorded from the CFD results can be presented in table (4).

Table 4. Inlet and outlet temperatures (Without Baffles)

-	Mass Flow Rate	Inlet Temperature	Outlet Temperature
Tube Hot Fluid	0.01 kg/s	350 K	316 K
Shell Cold Fluid	0.02 kg/s	300 K	317 K

From table (3):

Entering of Hot water T_{hi} = 350 K

Exiting of Hot water T_{ho} = 316 K

Flow rate of inlet mass of the hot water in Tube = 0.01 kg/s

Thus,

$q = \dot{m}_h Cp_h (T_{hi} - T_{ho}) = (0.01)(4182)(350 - 316) = 1422 \text{ w}$

Maximum heat transfer rate can be estimated from:

$q_{max} = (\dot{m} Cp)_{min} (T_{hi} - T_{ci}) = (0.01)(4182)(350 - 300) = 2091 \text{ w}$

Accordingly the effectiveness of a heat exchanger (without baffles) is found as of:

$$\varepsilon = \frac{q}{q_{max}} = \frac{1422}{2091} = 0.68$$

Similarly, but for heat exchanger with baffles, inlet and outlet temperatures recorded from the CFD results can be presented in table (4).

Table 4. Inlet and outlet temperatures (With Baffles)

-	Mass Flow Rate	Inlet Temperature	Outlet Temperature
Tube Hot Fluid	0.01 kg/s	350 K	312 K
Shell Cold Fluid	0.02 kg/s	300 K	319 K

From table (4):

Entering of Hot water T_{hi} = 350 K

Exiting of Hot water T_{ho} = 312 K

Flow rate of inlet mass of the hot water in Tube = 0.01 kg/s

Thus,

$$q = \dot{m}_h Cp_h (T_{hi} - T_{ho}) = (0.01)(4182)(350 - 312) = 1589 \text{ w}$$

Maximum heat transfer rate can be estimated from:

$$q_{max} = (\dot{m} Cp)_{min} (T_{hi} - T_{ci}) = (0.01)(4182)(350 - 300) = 2091 \text{ w}$$

Accordingly, the effectiveness of the heat exchanger (without baffles) is found from:

$$\varepsilon = \frac{q}{q_{max}} = \frac{1589}{2091} = 0.76$$

CONCLUSION

After carrying out all the investigation for shell and tube heat exchanger, the next observations have been obtained:

1. Results showed that fluent analysis is an effective tool to study the heat transfer phenomenon in a shell and tube heat exchanger, and decrease consuming time for theoretical performace.
2. The use of baffles has shown an enhancement in a heat exchanger performance because their facility toward assisting the maximum heat transfer (because high-tube-side heat transfer coefficient) in order to give pressure drop in the minimum space.
3. Effectiveness of a heat exchanger is increased from 0.68 to 0.76 after using a baffle plates around the tubes (inside the shell) of the facility.
4. Plate baffle increased a turbulence of a shell fluid and minimized tube-to-tube temperature variances and thermal stresses due to the cross flow.

RECOMMENDATIONS

1. Amount of heat transfer could be improved by changing a number, length, diameter of tubes.
2. Through varying baffles' number and shape, transfer of heat could be improved.
3. By varying the material of tubes heat transfer rate can be enhanced.

ACKNOWLEDGEMENTS

The authors would like toward express their deep thanks and appreciation to the support as long as by (Mustansiriyah University) to complete the present work.

REFERENCES

[1] S. Liu, and M. Sakr, "A Comprehensive Review on Passive Heat Transfer Enhancements in Pipe Exchangers", *Renewable and Sustainable Energy Reviews*, Vol. 19, Pp. 64-81, 2013.

[2] G.S.B. Raoand, and S.S. Babu, "Numerical Analysis of Shell and Tube Heat Exchanger with Different Proportions of Glycerine", International Journal and Magazine of Engineering Technology, Management and Research, Vol. 5, 2018.

[3] S. Ji, W. Du, P. Wang, and L. Cheng, "Numerical Investigation on Double Shell-Pass Shell-and-TubeHeat Exchanger with Continuous Helical Baffles", Institute of Thermal Science & Technology, Shandong University, Jinan 250061, China, 2011.

[4] Y. Wang, Z. Liu, S. Huang, W. Liu and W. Li, "Experimentalinvestigation of shell-and-tube heat exchanger with a new type of baffles", *Heat Mass Transfer*, Vol. 47, Pp. 833–839, 2011.

[5] H. Bayram, and G. Sevilgen,"Numerical Investigation of the Effect of Variable Baffle Spacing on the Thermal Performance of a Shell and Tube Heat Exchanger", *Energies*, Vol. 10, Pp. 1156, 2017. doi:10.3390/en10081156.

[6] ANSYS FLUENT Theory Guide, Ansys Inc., South Pointe 275 Technology Drive Canonsburg, 2011.

[7] F.P. Incropera, D.P. DeWitt, T.L. Bergman, and A.S. Lavine, "Fundamentals of Heat and Mass Transfer", *Seventh Edition, John Wiley & Sons, New York*, 2011.

List of symbols

English symbols

Symbol	Description	Units
C_p	Specific heat at constant pressure	J/kg.˚C
$C_{1\varepsilon}, C_{2\varepsilon}, C_{\mu}$	Turbulence constant	
D	Diameter	m
m	Mass	kg
$\dot{m}$	Mass flow rate	Kg/s
Q	Heat transfer rate	W
T	Temperature	˚C
u	Velocity component in X direction	m/s
v	Velocity component in Y direction	m/s
w	Velocity component in Z direction	m/s
Greek Symbol		
Symbol	**Title**	**Units**
ρ	Fluid Density	kg/m^3
μ	Dynamic viscosity	kg/m.s
μ_t	Turbulent viscosity	$N.s/m^2$
ε	Turbulent dissipation rate	m^2/s^3
η	Thermal performance factor	

Subscripts	
Symbol	**Title**
h	hot
i	inlet
max	maximum
min	minimum
c	cold
CFD	Computational Fluid Dynamics