University of Baghdad
College of Engineering
JE
JOURNAL OF ENGINEERING

**Journal of Engineering**
journal homepage: www.joe.uobaghdad.edu.iq
Number 11 Volume 25 November 2019

***Chemical, Petroleum and Environmental Engineering***

# Analysis of Shell and Double Concentric Tube Heat Exchanger Using CFD Application

**Basma Abbas Abdulmajeed***
Professor
College of Engineering - University of Baghdad
Email: basma1957@yahoo.com

**Hawraa Riyadh Jawad**
M Sc student
College of Engineering - University of Baghdad
Email: eng.hawraa90@gmail.com

## ABSTRACT

This study focuses on CFD analysis in the field of the shell and double concentric tube heat exchanger. A commercial CFD package was used to resolve the flow and temperature fields inside the shell and tubes of the heat exchanger used. Simulations by CFD are performed for the single shell and double concentric tube.

This heat exchanger included 16 tubes and 20 baffles. The shell had a length of 1.18 m and its diameter was 220 mm. Solid Works 2014, ANSYS 15.0 software was used to analyze the fields of flow and temperature inside the shell and the tubes. The RNG k-ε model was used and it provided good results. Coarse and fine meshes were investigated, showing that aspect ratio has no significant effect. 14 million elements were used in the mesh. A comparison was made between the profiles of temperature and velocity for the experimental and results of the model and it had an acceptable adaptation.

**Key Words:** Temperature transfer, RNG k-ε model, baffle, Shell and double concentric tube heat exchanger, CFD

# تطبيق ديناميكا الموائع الحسابية على المبادل الحراري ذو القشرة والانابيب المتداخلة المتمركزة

**بسمة عباس عبد المجيد**
استاذ
كلية الهندسة – جامعة بغداد

**حوراء رياض جواد**
طالبة ماجستير
كلية الهندسة – جامعة بغداد

## الخلاصة

ان هذه الدراسة تركز على تحليل ديناميكا الموائع الحسابية في مجال مبادل حراري. يتم حل مجالات التدفق ودرجة الحرارة داخل القشرة والانابيب باستخدام حزمة ديناميكا الموائع الحسابية التجارية. يتم اجراء مجموعة من محاكاة ديناميكا الموائع الحسابية لقشرة واحدة وانابيب متحدة المركز مزدوجة مع عدد من المصدات وجريان مضطرب .وقد تم التحقيق في المبادل الحراري ذو القشرة والانابيب المتداخلة المتمركزة من قبل البرمجيات ديناميكا الموائع الحسابية . ويشمل هذا المبادل الحراري 16 انبوب و 20 مصدة داخل القشرة مع طول 1.18 م وقطر 220 ملم . تم تحليل مجالات التدفق ودرجة الحرارة داخل القشرة والانابيب باستخدام برمجيات ( SOLID WORKS 2014 وANSYS15.0). من خلال التحقيقات التي اجريت على نماذج الاضطراب, تم استخدام نموذج RNG k-ε الذي يوفر نتائج افضل. تم فحص اثنين من الشبكات الشبكة الخشنة والشبكة الناعمة, وقد تبين ان نسبة العرض الى الارتفاع ليس لها تأثير كبير. وبالتالي ,تم استخدام الشبكة الناعمة التي تحتوي

*Corresponding author

Peer review under the responsibility of University of Baghdad.
https://doi.org/10.31026/j.eng.2019.11.02

Article received: 22/1/2018
Article accepted: 26/2/2018
Article published: 1/11/2019

على 14 مليون عنصر . تم مقارنة جوانب من درجة الحرارة والسرعة مع النتائج العملية وكانت النسبة مقبولة. وتبين ان الجريان بوجود المصدات لا يبقى متوازي بالانبوب. ونتيجة لذلك يتحسن مستوى نقل الحرارة وبالتالي يزيد من انتقال الحرارة.

**الكلمات الرئيسية**: نقل درجة الحرارة, نموذج K-ε RNG, مصدات, مبادل حراري ذو القشرة والانابيب المتحدة المتمركزة, ديناميكا الموائع الحسابية.

## 1. INTRODUCTION

Any simulation for a process in the industry is done by manufacturing a small prototype then, and this prototype is subjected to the same boundary conditions that may be encountered in the original part. This process is somewhat expensive, and repeating the manufacturing process gives a long time. Computational Fluid Dynamics (CFD), takes this action instead of the prototype. Analyzing the problems with heat transfer and fluid flow can be accomplished by CFD. It includes three stages. These represent the necessary fundamentals of any numerical simulation process.

Kern's method and Bell-Delaware's method are the most commonly used correlations for the design of shell and tube heat exchanger.

The turbulent flow includes a wide range of scales for the length, velocity and time. To solve all of them gives high simulation costs. Therefore, turbulence models have been designed and developed with the Navier-Stokes Equations. CFD models of turbulence are available in the software. These include the Large Eddy Simulation (LES) and Reynolds Average Navier-Stokes (RANS). Several models of RANS exist that depend on the feature of flow, for example, Standard k-ε model, k-ε- RNG model, and Reynolds Stress Model (RSM).

The objective of the present work is to simulate the 3D geometry for counterflow heat exchanger with using hot oil inside the inner tube and shell and cooling water in the annuals tubes by using computational fluid dynamic (ANSYS-FLUENT 15.0).

## 2. LITERATURE REVIEW

### 2.1 Numerical Investigations

**Hilde et al.,** in 2003, modeled a three-dimensional tube exchanger numerically in CFD. The results of heat transfer coefficient and friction factor were compared with established correlations. The second part of this study showed the ability of CFD to model a prototype configuration of a tube in tube exchanger. This ability decreased time and cost. A comparison is made between the numerical data and the analytical predictions and experimental results.

**Uday and Satish, 2005**, investigated a theoretical model for a shell-side pressure drop. This model includes a pressure drop effect in inlet and outlet nozzles. Also, the losses in the segments created by baffles were studied. The results showed that for Reynolds numbers ranging from $10^3$ to $10^5$ correspond to the experimental results for different configurations of heat exchangers.

**Ender and Ilker**, **2010**, investigated numerically the effects of the baffle spacing, baffle cut, and shell diameter on the heat transfer coefficient and pressure drop of a shell-and-tube heat exchanger. CFD simulation is performed for a heat exchanger with a single shell and single tube pass with a variable number of baffles and turbulent flow. A comparison was made between the CFD results of heat transfer coefficient, outlet temperature, and pressure drop with that of the Bell-Delaware method. It is surprising that the differences between Bell-Delaware and CFD predictions of the total heat transfer rate are below 2% for most of the cases. That confirms the well-deserved trust that Bell-Delaware method gained in the heat exchanger industry and shows the power of CFD technique as a heat exchanger design tool.

### 2.2 Experimental and Numerical Investigations

**Huang, et al.**, **1996,** studied heat transfer improvements and fluid flow modeling. He especially focused on the algebraic terms of the resistance that is locally distributed and the coefficients of volumetric heat transfer. An experimental investigation and local flow field numerical simulations of tube bundles were followed.

A CFD analysis of a tube heat exchanger was done by **Kumar, et al.**, **2003**. A close agreement was found between the results obtained from the CFD simulation and that of the experiments. **Ozden and Tari, 2010,** performed a CFD analysis on the shell side, showing that Kern method yielded a very high percentage of error.

**Anshul, et al.**, **2015**, performed a theoretical and experimental calculation of heat transfer numerically and they resolved the flow and temperature fields by using CFD package (ANSYS FLUENT). They were the CFD turbulence models used for investigation are k-epsilon, SST, Eddy Viscosity and Laminar model, and the boundary conditions for the computational domain are derived out of the experimental results where they also used experimental investigation for comparison purpose. They found out that the k-epsilon model gives the best model to predict the flow parameters, heat transfer coefficient and behavior of the present case of STHE. Reasonable agreement is found between the simulation and experimental data.

### 2.3 Experimental Investigations

**Schlunder**, **1974,** presented an investigation concerned with the design, performance, and development of all kinds of heat exchanging equipment. The goal of his research was to predict a design and performance data of heat exchanging equipment, and as a result, this can be applied to investigate the engineering problems under defined conditions, thus, enabling to find out the ruling phenomena and laws.

**Yusur, 1997**, investigated a step by step method for the thermal-hydraulic design of a shell and tube condenser. He presented a design procedure based on the Silver method in which the baffle spacings are considered one by one, then each baffle spacing is subdivided into several steps selected by the designer, and the output conditions of each step are taken as the input to the next step.

**Baadache, 2010**, studied the thermal and hydraulic design for the new type of heat exchanger called (shell -and -double concentric tube heat exchanger) and its performance depending on the inner tube diameter of the heat exchanger.

In **Basma and Fadhil, 2015,** published a study concerning the design of shell-and-double concentric tube heat exchangers. They studied both the design and performance calculations of the heat exchanger.

The design was conducted according to Kern method with volumetric flow rates of 3.6 $m^3$/h and 7.63 $m^3$/h for the hot oil and water respectively. They studied: temperature, the flow rate of hot oil, and that of cold water, and the pressure drop.

The efficiency of the heat exchanger was increased using γ- $Al_2O_3$/water nanofluid as a cold stream in the shell and double concentric tube heat exchanger. A hot stream of basis oil was used counter-currently. The results showed that as nanofluid concentrations increased, each of the overall heat transfer coefficient and the Nusselt number increased. **Basma and Noor, 2017.**

## 3. WORK DESCRIPTION

The continuity equation, the energy equation, and the momentum equation controlled the flow. The transfer of mass, energy, and momentum happens by convective flow in addition to the molecular distribution and eddies. Control volume regulated all equations.

- **Continuity Equation**

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

- **Momentum Equation**:

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial P}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)\text{ in the x\_dir} \tag{2}$$

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial P}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)\text{ in the y\_dir.} \tag{3}$$

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial P}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)\text{ in the z-dir} \tag{4}$$

- **Energy Equation**

$$\rho c_v\left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}\right)=K\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right)+\emptyset \tag{5}$$

$$\emptyset=\mu\left[2\left(\frac{\partial u}{\partial x}\right)^2+2\left(\frac{\partial v}{\partial y}\right)^2+2\left(\frac{\partial w}{\partial z}\right)^2+\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)^2+\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right)^2+\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)^2\right]-\frac{2}{3}\left[\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right]^2 \tag{6}$$

## 4. METHODOLOGY

The work uses the CFD analysis for the counterflow shell and double concentric tube heat exchanger, in order to evaluate the effect of temperature rise and pressure drop along the length of the tubes and the shell. The hot oil enters the shell side and inner tube fluid, while cold water flows in the annuals tube side. The hot fluid in the inner tubes and shell transfer heat to the cold fluid (water) that is flowing through the annuals tubes**.**

### 4.1 Computational Domain

The computational domain of the present work is represented by the following:
1- The inlet and outlet for hot oil which flows inside single pass single carbon steel tubes and a shell side.
2- The inlet and outlet for cooling water which flows inside annuals between concentric tubes.

The following assumptions are used:

1- Steady-state conditions.
2- Adiabatic process.
3- No heat is generated.
4- No phase change.
5- Constant properties of the fluids.
6- No radiation effects.
7- No conduction in axial directions.

## 4.2 Computational Modeling

The first step of computational modeling is Geometry Modeling. It requires the geometric parameters of the model. The basic approaches to using CFD are, according to **Gurbir and Hemant**, **2014**, and **Mohammad**, **2014**:

### 1. Geometry:

The SOLID WORK 2014 design module was used to build the heat exchanger geometry in 3D form. The geometry is shown in **Fig.1**.

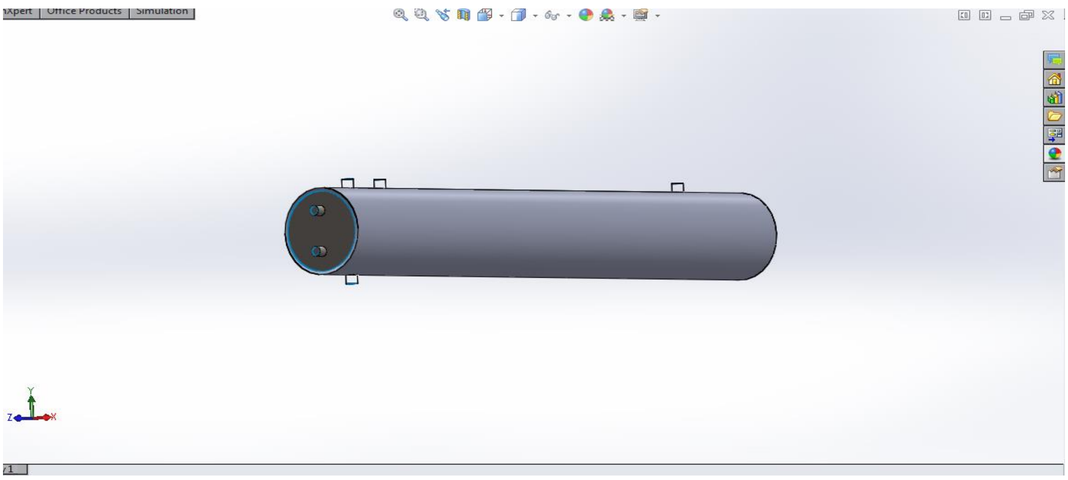
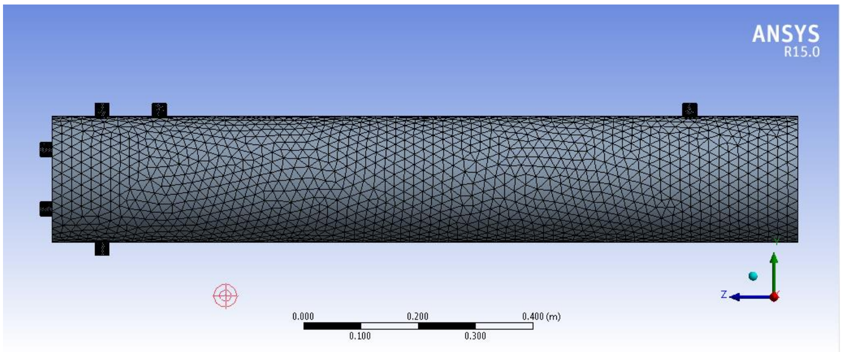


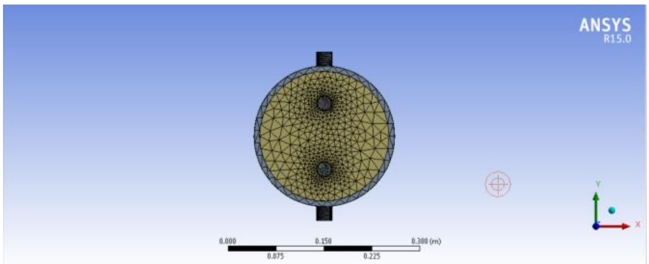
**Figure 1.** The geometry of the test section.

### 2. Mesh:

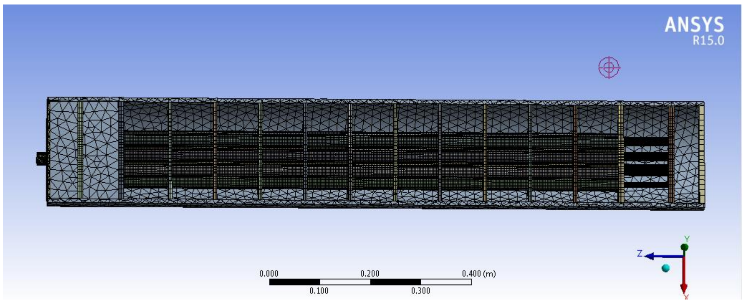
Mesh generation is a very important step of the pre-processing stage because it fits the limits of the computational domain. The irregular mesh was adopted because of the complex geometry used. The mesh generation for the present work is shown in **Fig 2.a,b,c.**




(a)




(b)




(c)

**Figure 2.** a,b,c. Mesh generation of the present work geometry.

## 3. Solver:

### 1. Problem Setup

The mesh is checked, and the analysis type is changed to Pressure Based type while the velocity formulation is changed to absolute and time to steady state.

### 2. Models

Energy is set to ON position. Viscous model is selected as “RNG k-ε model.

### 3. Materials

The create/edit option is clicked to add water-liquid and forty stock oil and carbon steel to the list of fluid and solid respectively from the fluent database.

**4. Cell zone conditions:**

The parts are assigned as water, forty stock, and carbon steel as per fluid/solid components.

**5. Boundary conditions:**

The desired mass flow rate and temperature values are assigned to the inlet nozzle of the heat exchanger. The hot oil and cold water inlet temperatures are set to 348 K and 293 K respectively. At the outlet nozzle, zero gauge pressure is assigned, to obtain the relative pressure drop between inlet and outlet. A uniform velocity profile is assumed at the inlet. No slip condition, and zero heat flux boundary condition is assigned to the outer shell wall. This is done by assuming the shell is perfectly insulated outside.

**6. Run Calculation:**

The number of iteration is set to 106250 and the solution is calculated and various contours, vectors, and plots are obtained. The contours of temperature and velocity are shown in **Figures 3** and **4**, respectively.

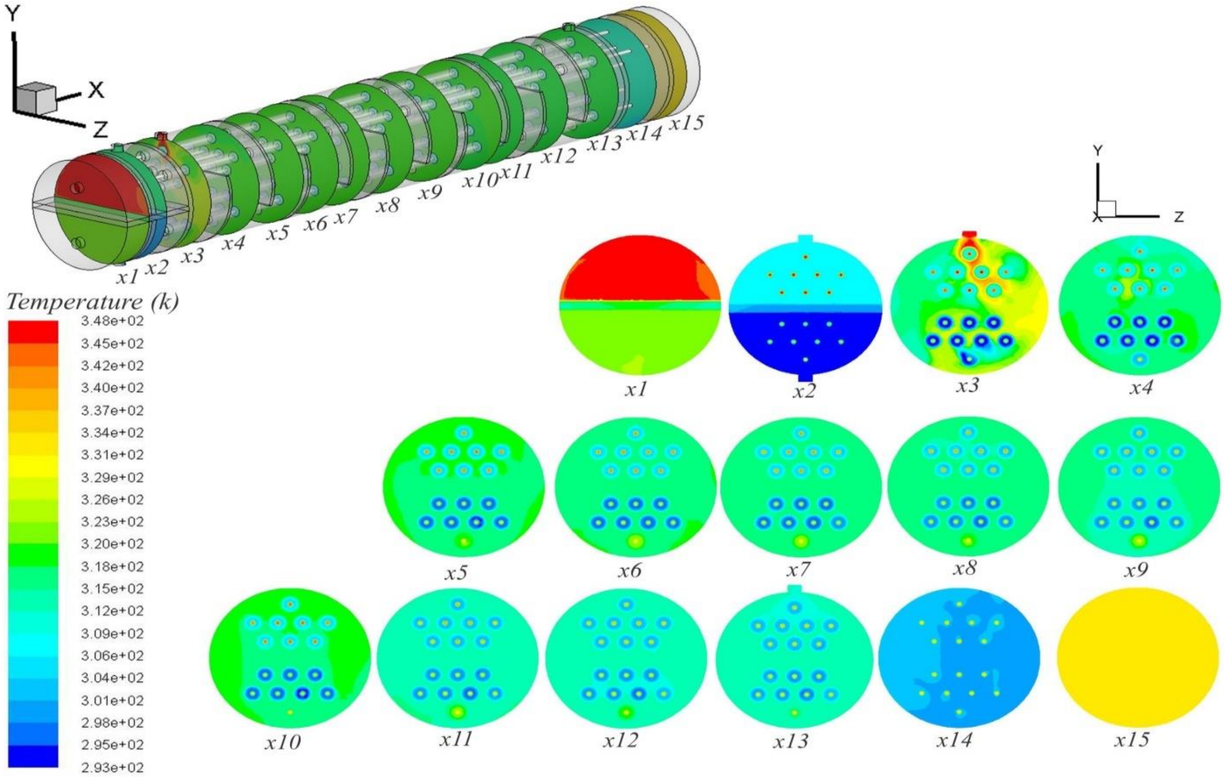



**Figure 3.** Drawing contour of temperature in the symmetrical plane.

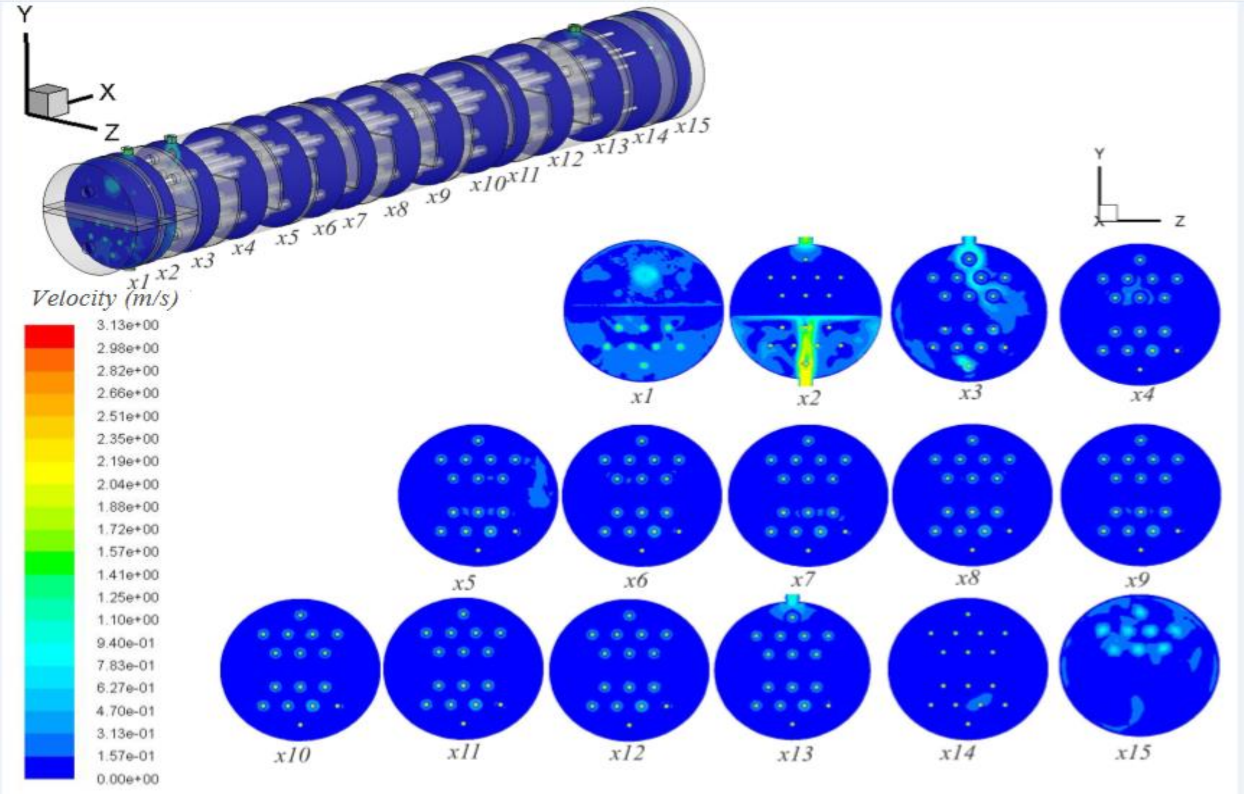



**Figure 4.** Drawing contour of velocity in the symmetrical plane.

**Table 1.** and **Fig 5.** illustrate the comparison between the results from numerical simulation and experimental work. The inlet temperature of the hot oil is constant 75$^{o}$C but the outlet temperature is varied with the flow rate which is ranged between 20 – 45, while the flow rate of cold water is 40 l/min.

**Table 1.** Comparison between the numerical simulation and experimental work.

| The outlet temperature of hot oil ($^{o}$C) | | The flow rate of oil (l/min) |
|---|---|---|
| **Experiment** | **CFD** | |
| 42 | 41 | 20 |
| 41 | 40 | 25 |
| 39 | 38 | 30 |
| 37 | 36 | 35 |
| 36 | 35 | 40 |
| 34 | 33 | 45 |

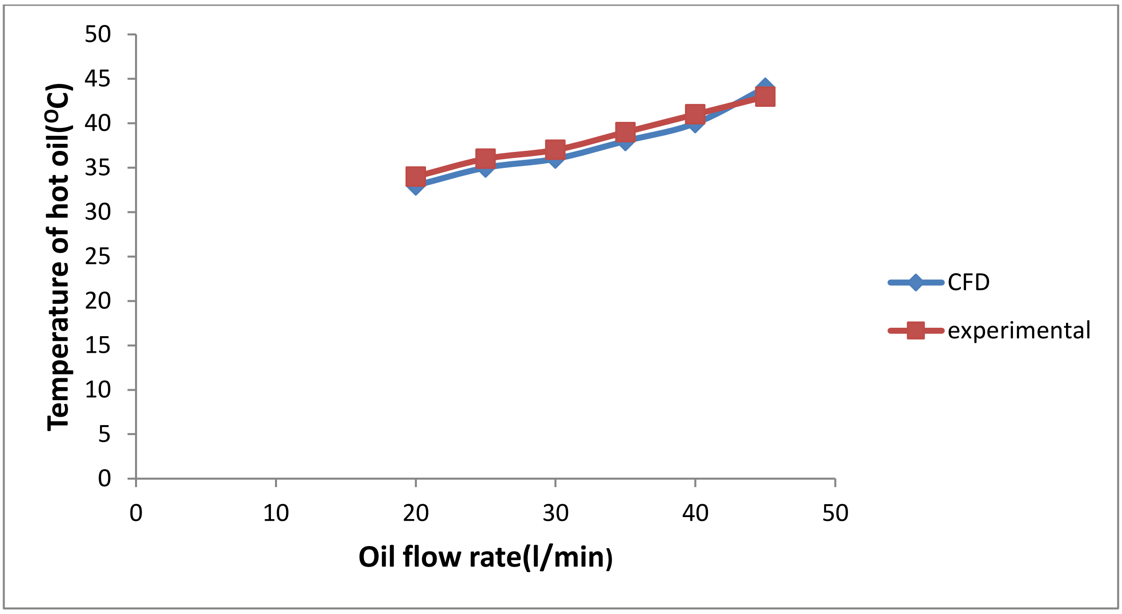



**Figure 5.** The outlet temperature of the hot oil at different flow rates of oil. Oil inlet temperature 75°C and water flow rate of 40 l/min.

## 6. CONCLUSIONS

CFD provides a cost-effective alternative, speedy solution and eliminates the need for prototype. The literature review focus on the analysis of various parameters which influence the performance of the STHE. It has been observed that computational modeling is one of the efficient techniques to study this type of heat elements. The parameters like tube and shell diameter, number of tubes, pitch, and baffle angles are the important one to be worked upon. A detailed analysis using the CFD simulation will be worthy to be carried out.

The present study provides a CFD analysis for counterflow heat exchanger with the smooth tube. The following conclusions can be detailed:

1- Good agreement is attained between the experimental and numerical results with a maximum deviation of (+3.65%).
2- Ansys Fluent is good CFD program to simulate the heat transfer cases.
3- Numerical investigation of a double concentric tube heat exchanger under the steady-state condition is carried out using finite volume method to describe the thermal behavior of the heat exchange between the three fluids along the length of the heat exchanger. The analysis is first carried out to show the validation of the mathematical model.

### Sample of Calculations

The heat exchanger serves for cooling a flow of oil (forty stock) $Q_1 = 2.7\ m^3/h$ of $T_{i1} = 100°C$ to $T_{o1} = 50°C$ with water flowing in the tubes of $T_{i2} = 20°C$ to $T_{o2} = 25°C$.

The thermo-physical properties of the oil for an average temperature of 75°C are as follows:

**Table 2.** Thermo-physical properties of the oil.

| | |
|---|---|
| Density | 822 kg/m$^3$ |
| Specific heat | 2135 J/(kg.k) |
| Thermal conductivity | 0.1299 W/(m.k) |
| Viscosity | 3.97×10$^{-4}$ Pa.s |

The thermo-physical properties of water for an average temperature of 25°C are as follows:

**Table 3.** Thermo-physical properties of water.

| | |
|---|---|
| Density | 1000 kg/m$^3$ |
| Specific heat | 4180 J/(kg.k) |
| Thermal conductivity | 0.607 W/(m.k) |
| Viscosity | 8.9×10$^{-4}$ Pa.s |

The heat exchanger is constituted of a bundle of $N_t$ = 16 steel tubes of thermal conductivity $K_w$ = 50W/ (m.k), of diameters inside/outside ($D_2/D_1$) of 20/25 mm, in the normal triangular pitch p = 31.25 mm.
The heat exchanger has two passes. The shell has a diameter $D_s$ = 203 mm and possesses baffles of thickness δ = 6 mm spaced by a distance B = 60 mm. The free section left with baffles is of 25%.
To determine the length of the tube:
Mass flow rate $m_1$ of the oil is:

$$m_1 = \frac{2.7 \times 822}{3600} = 0.616 kg/s \tag{7}$$

The exchanged heat flux is:

$$q = m_1 Cp_1 (T_{i1} - T_{o1}) = 65758 W \tag{8}$$

Mass flow rate $m_2$ of the water is:

$$m_2 = \frac{q}{Cp_2 (T_{o2} - T_{i2})} = 1.57 kg/s \tag{9}$$

The volumetric flow rate of water is:

$$Q = 5.65 m^3/h \tag{10}$$

For counter flows the logarithmic mean temperature difference is calculated as:

$$LMTD = \frac{(100-25)-(50-20)}{\ln\frac{(100-25)}{(50-20)}} = 49.12^{o}C \tag{11}$$

The values of temperature ratio are:

$$R = \frac{(100-50)}{(25-20)} = 10 \tag{12}$$

$$S = \frac{(25-20)}{(100-20)} = 0.0625 \tag{13}$$

The corrective factor F of the logarithmic mean temperature difference, corresponding to the calculated values of R and S is:

F = 0.95 (14)

The cross-sectional area of the tube is:

$$A_{c3} = \frac{3.14}{4}(0.006)^2 = 0.000028 m^2 \tag{15}$$

The velocity of the water in tubes is:

$$u_3 = 1.67 m/s \tag{16}$$

The calculation of the Reynolds number and the Prandtl number:

$$\mathrm{Re}_3 = \frac{822 \times 1.67 \times 0.006}{3.97 \times 10^{-4}} = 20746 \tag{17}$$

$$\mathrm{Pr}_3 = 6.52 \tag{18}$$

By using the Colburn Equation, the Nusselt number is:

$$Nu_3 = 0.023(20746)^{0.8}(6.52)^{0.33} = 121 \tag{19}$$

The heat transfer coefficient from the Equation below is:

The shell equivalent diameter for triangular pitch is:

$$D_e = \frac{1.10}{0.025}(\left(31.25\times10^{-3}\right)^2 - 0.917(0.025)^2) = 0.0176 m^2 \tag{20}$$

The bundle crossflow area is:

$$a_s = \frac{0.203\times0.06\times0.00625}{31.25\times10^{-3}} = 0.0024 m^2 \tag{21}$$

The shell side mass flow rate is calculated from Equation as:

$$G_s = \frac{0.308}{0.0024} = 128kg/m^2.s \tag{22}$$

The heat transfer coefficient in the shell side is calculated as:

$$h_1 = 572W/\left(m^2K\right) \tag{23}$$

The flow cross-sectional area of the annulus passages is calculated as :

$$A_{c2} = \frac{3.14}{4}\left((0.02)^2 - (0.01)^2\right) = 2.35 \times 10^{-4} m^2 \tag{24}$$

The velocity of water in annulus flow passages is:

$$u_2 = \frac{2.12}{1000 \times 2.35 \times 10^{-4} \times 8} = 0.84m/s \tag{25}$$

The equivalent diameter of the annulus is calculated as:

$$d_h = 0.02 - 0.01 = 0.01m \tag{26}$$

By using the Colburn equation, the Nusselt number is

$$Nu_2 = 0.023(9438)^{0.8}(6.13)^{0.33} = 63 \tag{27}$$

The heat transfer coefficient is:

$$h_2 = \frac{63 \times 0.607}{0.01} = 3824W/\left(m^2K\right) \tag{28}$$

**Overall heat transfer coefficient:**

The first overall heat transfer coefficient ($U_{12}$) between (the fluid in the shell side and fluid in the annulus passage) is calculated as:

$$U_{12} = \frac{1}{\frac{0.020}{0.025 \times 572} + \frac{0.02}{2 \times 50}\ln\frac{0.025}{0.02} + \frac{1}{3824}} = 588W/\left(m^2.K\right) \tag{29}$$

The second overall heat transfer coefficient $U_{23}$ between (the fluid in the annulus passage and the fluid in the inner tube side) is calculated as:

$$U_{23} = \frac{1}{\frac{0.006}{0.01 \times 3824} + \frac{0.006}{2 \times 50}\ln\frac{0.01}{0.006} + \frac{1}{2620}} = 1757W/\left(m^2.K\right) \tag{30}$$

**Inner Tubes Side Pressure Drop Calculation**

The inner tube side pressure drop is calculated as:

$$\Delta P_3 = \left(4 \times 0.026 \times \frac{1.08 \times 2}{0.006} + 4 \times 2\right) \frac{822 \times (1.67)^2}{2} = 26kPa \tag{31}$$

**Shell Side Pressure Drop Calculations**

The hydraulic diameter of the shell is calculated as:

$$d_{hs} = 0.025\left(\frac{3.46}{3.14}\left(\frac{31.25 \times 10^{-3}}{0.025}\right)^2 - 1\right) = 0.018m^2 \tag{32}$$

The pressure drop is calculated as:

$$\Delta P_1 = \frac{0.34 \times (128)^2 (18+1) \times 0.203}{2 \times 822 \times 0.89 \times 0.0176} = 0.83kPa \tag{33}$$

**Annulus Side Pressure Drop Calculation**

$$\Delta P_2 = \left(4 \times 0.032 \times \frac{1.08 \times 2}{0.01} + 4 \times 2\right) \frac{1000 \times (0.84)^2}{2} = 6.3kPa \tag{34}$$

**The total power expenditure:**

The total power expenditure of the new heat exchanger is calculated as :

$$P_T = \frac{0.308 \times 0.83}{822} + \frac{1.67 \times 6.3}{1000} + \frac{0.308 \times 26}{822} = 0.0205W \tag{35}$$

**The mass of heat exchanger:**

The mass of shell and double concentric tubes heat exchanger is calculated as :

$M_{sdct}$ = 63.9 kg (36)

## 7.REFERENCES


- Ansys Fluent User's Guide, 2015, *Ansys Inc., South Pointe 275 Technology Drive Canonsburg.*

- Anshul J., K. K. Jain, and Sudarshan P., 2015, *Comparative Study of Different CFD Models to Evaluate Heat Transfer and Flow Parameters in STHE*, International Journal of Engineering Sciences & Research Technology, ISSN: 2277-9655, pp. 536-547.

- Bougriou, C., Baadache, K., 2010, *Shell-and-Double Concentric Tube Heat Exchangers*, Heat and Mass Transfer, 46. PP.315-322.

- Basma, A. and Fadhil, A., 2015, *Shell and Double Concentric Tube Heat Exchanger Calculations and Analysis*, Journal of Engineering, Number 1, Volume 21 January, pp.62-82.

- Basma, A., and Noor, S., 2017, *Study and Analysis of Concentric Shell and Double Tube Heat Exchanger Using γ- Al2O3 Nanofluid*, Journal of Engineering, Number 9, Volume 23 September, pp.50-62.

- Dilip S.P., Ravindrasinh R.P., and Vipul M.P., 2015, *CFD Analysis of Shell and Tube Heat Exchangers –A review*, International Research Journal of Engineering and Technology (IRJET), Volume: 02 Issue: 09, ISSN: 2395 -0056, pp.2231-2235.

- Ender O., Ilker T., 2010, *Shell side CFD Analysis of a Small Shell-and-tube Heat Exchanger*, Energy Conversion and Management, Vol. 51, No. 5, pp. 1004-1014.

- G.A. Quadir, Irfan A. B., and N.J. Salman, 2014, *Numerical investigation of the performance of a triple concentric pipe heat exchanger*, International Journal of Heat and Mass Transfer 75 (2014), PP.165-172.

- Hilde V.D., Jaco D., and Josua P.M., 2003, *Validation of A CFD Model of a Three Dimensional tube – in – Tube heat exchanger*, Third International Conference on CFD in the Minerals and Process Industries CSIRO, Melbourne, Australia.

- Huang, L.Y., Karayiannis, T.G., Mathews, R.D., 1996 *CFD Model of Fluid Flow and Heat Transfer in a Shell and Tube Heat exchanger*, The Phoenics Jor. of CFD and its Application, Vol.9, No.2, pp.181-209.

- Kumar, V., et al., 2003, *CFD Analysis of Cross Flow Air to Air Tube Type Heat Exchange*, PHOENICS 10th International User Conference.

- Mahmoud G.Y., Ahmed A.A. et al., 2014, *On The Computations of Thermal Behaviour of Shell and Tube Heat Exchanger*, AIAA Propulsion and Energy Forum and Expositions, Cleveland, Ohio, Cleveland Medical Mart & Convention Center.

- Mohammad R. S., Tooraj Y., and Mostafa K. M., 2014, *Numerical Analysis of Shell and Tube Heat Exchanger with Simple Baffle By CFD*, Indian Journal Scientific Research. Vol.7 , No.1, ISSN: 0976-2876 (Print) , ISSN: 2250-0138(Online), pp.1334-1345.

- Ozden, E. and Tari, I., 2010, *Shell Side CFD Analysis of a Small Shell and- Tube Heat Exchanger*, Energy Conversion and Management, Vol. 51, pp. 1004-1014.

- Schlΰnder, E.U., 1974, *Application of Heat Transfer Theory to Heat Exchanger Design*, Heat Exchangers Design and Theory, Scripta Book Company.

- Uday C. K., and Satish C., 2006, *Modeling for shell-side pressure drop for liquid flow in the shell-and-tube heat exchanger,* International Journal of Heat and Mass Transfer, Vol.49, No.2, pp. 601–610.

- Yusur, A. N., 1997, *Step by Step Method for Thermal-Hydraulic Design of Single and Multipass Condensers*, A Thesis for Master of Science in Mechanical Engineering, Al-Nahrain University.


**NOMENCLATURE**

$A_c$ = cross sectional area of the tube, $m^2$
$a_s$ = cross flow area at the shell, $m^2$
B = baffle spacing, m
Cp = specific heat, J/(kg K)
D = diameter, m
De= equivalent diameter on the shell-side, m
Ds = shell inside diameter, m
d = inner tube diameter, m
dhs = hydraulic diameter of the shell, m
dh = hydraulic diameter of the annulus, m
F = corrective factor
f = friction factor
Gs = shell side mass velocity, kg/ m2.s
h = heat transfer coefficient, W/ m2.K
k = thermal conductivity, W/ m K
U = overall heat transfer coefficient, W/ m2.K
L = length of tube, m
M = mass, kg
m = mass flow rate, kg/s
Nb = number of baffles
Nt = total number of tubes
Ntp = number of tubes per pass
p = tube pitch, m
PT = total power expenditure, W
$\Delta p$ = pressure drop, Pa
Q = volumetric flow rate, m3/h
q = heat transfer rate, W
R = dimensionless temperature ratio
S = dimensionless temperature ratio
Sa = exchange surface, m2
T = temperature, ˚C
u = fluid velocity, m/s

**GREEK SYMBOLS**

$\delta$ thickness, m
$\mu$ dynamic viscosity, Pas
$\rho$ density, $kg/m^3$

## SUBSCRIPTS

1 Hot oil (shell side), outer
2 Water, inner
3 Hot oil (inner tube)
12 Shell and annulus
23 Annulus and inner tube
s Shell
h Hydraulic
i Inlet
io Hot oil
o Outlet
st Shell-and-tube heat exchanger
sdct Shell-and-double concentric-tube heat exchanger
w wall

## ABBREVIATIONS

| | |
|---|---|
| RANS | Reynolds Average Navier- Stokes |
| CFD | Computational Fluid Dynamics |
| LMTD | log-mean temperature difference |
| LES | large Eddy Simulation |