Modelling of the single staggered wire and tube heat exchanger

Article *in* International Journal of Applied Engineering Research · January 2016

4 authors, including:

I Made Arsana
Universitas Negeri Surabaya

Ali Altway
Institute Technology of Sepuluh Nopember (ITS)

International Journal of Applied Engineering Research ISSN 0973-4562 Volume 11, Number 8 (2016) pp 5591-5599

Modelling Of The Single Staggered Wire And Tube Heat Exchanger

I Made Arsana
Department of Chemical Engineering,
Sepuluh Nopember Institute of Technology (ITS),
Surabaya, East Java, Indonesia.

Kusno Budhikardjono
Professor, Department of Chemical Engineering,
Sepuluh Nopember Institute of Technology (ITS),
Surabaya, East Java, Indonesia.

Susianto
Department of Chemical Engineering,
Sepuluh Nopember Institute of Technology (ITS),
Surabaya, East Java, Indonesia.

***Ali Altway,**
Professor, Department of Chemical Engineering,
Sepuluh Nopember Institute of Technology (ITS),
Surabaya, East Java, Indonesia.
Corresponding Author,

ABSTRACT
Generally, a wire and tube heat exchanger uses the array of inline (symmetry) wires between both its sides, but this research used the array of staggered wires so that it decreases the resistance of convection heat transfer. The purpose of this research is to perform the modelling of single staggered wire and tube heat exchanger. Simulation used a finite element method with a MATLAB program assistance, which a heat exchanger is divided into some element units. To support the result of the modelling processed by MATLAB program, then the simulation analysis with CFD was done to perform the convection current dynamics on the outside surface of the heat exchanger. The model developed to present the heat transfer correlation on single staggered wire and tube heat exchanger was valid. The validation conducted by the approach of element temperature parameter on 9 points thermocouple with the error was less than 5% between the experimental and modelling results. From CFD analysis, it was obtained that the speed distribution of convection fluid current was averagely smoother and higher in the staggered heat exchanger, and it finally makes the convection heat transfer better.

Keywords: Single staggered wire and tube heat exchanger; heat exchanger capacity; modelling; small refrigerator; computational fluid dynamics (CFD)

INTRODUCTION

A wire and tube heat exchanger consists of a coiled tube, with wires welded on the two sides in normal direction to the tube. This heat exchanger has been widely used to dissipate heat from hot fluid which flows through a tube as a condenser of a small air refrigeration system (refrigerator) or applied only as fluid cooler that flows in tube without a changing phase [1]. Generally, a wire and tube heat exchanger uses the array of inline (symmetry) wires between both its sides, but in this research used the array of staggered wires so that it decreases the resistance of convection heat transfer.

Some researches about wire and tube heat exchangers explained as follows. First, it was Witzel and Fontaine (1957 a,b) who study about heat exchanger characteristics on wire and tube condenser (1957, a) and about wire and tube condenser design (1957, b) which produced the equation of empirical Nusselt as the function of Grashoft number, that is $Nu = 0.4724\ (Gr)^{0.2215}$ [2, 3]. Then the similar research performed by Cyphers et al. (1959) about heat transfer characteristics on wire and tube heat exchanger [4], and Collicott et al. (1963) which studied about free convection and radiation heat transfer from wire and tube heat exchangers [5]. The following contribution performed by Tanda and Tagliafico (1997) about free convection heat transfer from wire and tube heat exchanger [6]. To achieve that purpose, it was studied experimentally by using wire and tube heat exchanger with water used as a working fluid in tubes.

Tagliafico and Tanda (1997), also expanded the previous study with a radiation heat transfer. To evaluate the coefficient of free convection heat transfer from heat exchanger's surface, it was used a semi empirical equation. But for radiation heat transfer, they developed a therotical model to calculate the average heat transfer coefficient according to radiation. The developed model was validated by experimental data from 8 wire and tube heat exchangers in different geometry.

In terms of wire and tube heat exchanger, Lee et al. (2000) also studied the determination of airside heat transfer coefficient [7]. The difference is heat exchanger which becomes an object using tube bundles applied on a bigger system. The examination's sample was using single layer from tube bundles, then examined in a wind tunnel in forced convection condition and used electric heater as a heater resource to heat up the tube.

Quadir et al. (2002) also performed modelling with numerical analysis towards wire and tube condensor by using refrigerant as working fluid and operated in normal condition (free convection) [8]. The modelling used an element unit method and modelled 8 various surrounding temperatures and refrigerant flow rates to examine the influence on heat exchanger performance. The developed model was validated by the experiment data which used shell and tube heat exchanger. They assumed coefficient of free convection heat transfer of 10 W/m^2K.

Bansal dan Chin (2003) developed a computer model for wire and tube heat condenser in free convection condition [9]. This study used the same method as Tagliafico and Tanda (1997) to get a free convection and radiation heat transfer coefficient. Then the model's validation performed by comparing the result of modelling with total heat load from condensor. It obtained that heat load deviation is ± 10%.

The last study was performed by Pradeep Kumara (2011) which is a simulation of domestic condenser, that is NST 200 type, by using the model developed by Tagliafico and Tanda which aimed to increase heat transfer [10].

Some researches above mostly used or modelled the wire and tube heat exchanger with inline wire design (inline arrangement), on the other hand this research has done a new study with uninline wire arrangement or single staggered configuration between both sides of heat exchanger.

Wire and Tube Heat Exchanger

Heat exchanger is a supporting instrument of production which functions to conduct heat energy transfer from a flowing fluid to another flowing fluid. Type and size of this heat exchanger varies depend on the needs. One of these instruments is a wire and tube heat exchanger.

As its name, this heat exchanger consists of horizontal tubes coiled in vertical arrangement, with wires welded on both sides in normal direction to the tubes [1]. The following figure is the configuration of wire and tube heat exchanger with the array of inline wire on both sides of heat exchanger.

On the other hand the fins's array of wire and tube heat exchanger has another configuration, which uses the array of staggered wires between both sides of heat exchanger as shown in figure 2. By the array of staggered wires, there is no face-to-face position of wire in a small distance. It makes the convection heat transfer performs better because the thermal boundary layer from both sides of wires are not mergered [11].

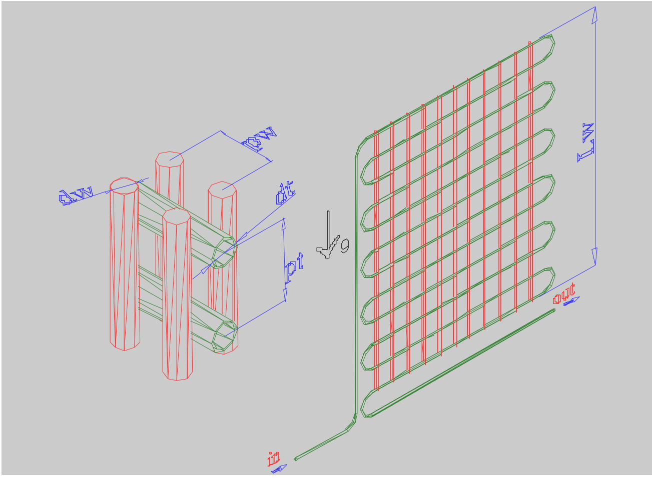


Figure 1: Wire and tube heat exchanger (inline wire)

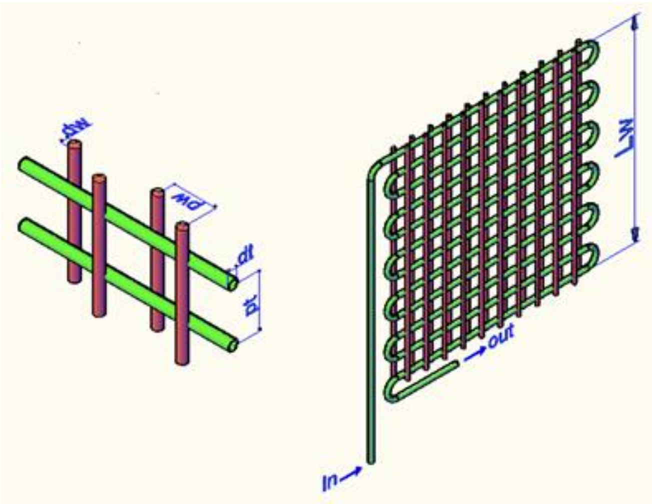


Figure 2: Configuration of single staggered wire and tube heat exchanger.

CFD Based Simulation (Computational Fluid Dynamics)

CFD, Computational Fluid Dynamics, defined as a computation technology which enables to study the dynamics of flowing substances [12]. Besides that, CFD can also be defined as a science discipline which studies how to predict fluid flow, heat transfer, chemical reaction, and other phenomenons by solving the mathematical equation (mathematical model). Basically, the equation of fluid developed and analyzed based on the partial differential equation which presents the physical phenomenons on the flow. The differential equation based on:

- Mathematical model (partial differential equation) especially Navier Stokes's equation
- Numerical method (technics of solution and discretization), and
- Software, tools (solvers, tools pre and post processing)

A Software of CFD gives help to its user to perform fluid flow simulation, heat transfer, mass transfer, moving things, multiphase flow, physical-chemistry reaction, interaction between fluid and structure, and also acoustic system just by modelling in computer. Generally, there are three phases which must be done when performing simulation of CFD, which are preprocessing, solving and post processing steps [13].

RESEARCH METHOD

Modelling

The simulation used the element finite method with the assist of a MATLAB program, which a heat exchanger element is divided into some element units [14]. Each modelling element consists of a tube as long as the pitch wire (pw) and a wire as a fin as long as the pitch tube (pt) with the entrance and exit flows of every element, as the following figure 3.

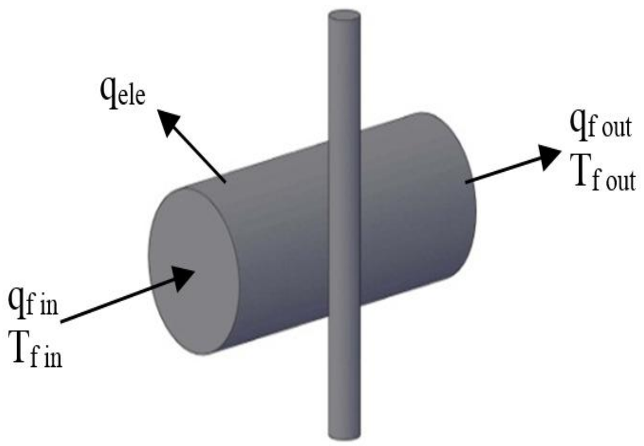


Figure 3: One element of heat exchanger in the finite element method

The heat transferred by every element is defined as Q_{ele}, then the number of all Q_{ele} is the total heat transferred by heat exchanger (Q_{total}). Validation is performed by comparing the element surface temperature of the model with the experiment results, which the average element temperature (T_{ex}) calculated by this following equation:

$$T_{ex} = \frac{T_{to} + GP.\eta_W (T_{to} - T_\infty) + GP.T_\infty}{(1 + GP)} \quad .1$$

Where:

T_{ex} = average element temperature

T_{to} = outside tube temperature

GP = geometry parameter

η_W = wire efficiency

T_∞ = outside temperature (enviroment)

By developing a new geometry parameter (*GP*) based on the geometry characteristics of the single staggered wire and tube heat exchanger which only has a wire, the equation will be:

$$GP = \left(\frac{P_t}{d_{to}}\right)\left(\frac{dw}{pw}\right)$$

Experiment Set-up

After the result of model program was out, then the next step was examined its validation by comparing it with the experiment data which using series of experimental tools with the same operation condition, fluid characteristic and geometry configuration with the model program. To perform the experiment, it needs experimental tools which can be seen on the following figure:

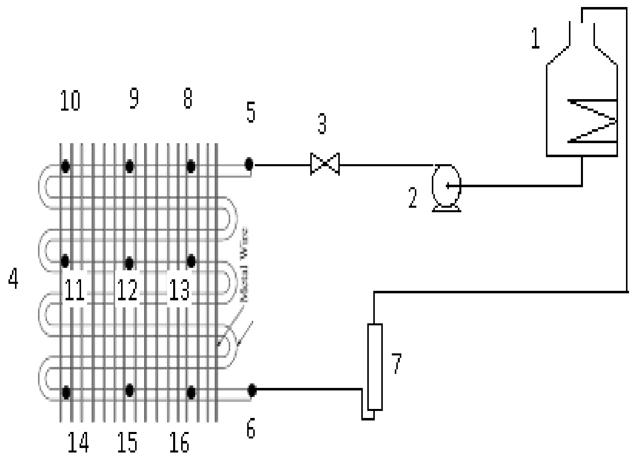


Figure 4: The experiment set-up 1 - Head fluid tank; 2 - Pump; 3 - Valve; 4 - Heat exchanger; 5 - Input temperature thermocouple; 6 - Output temperature thermocouple; 7 - Flowmeter; 8 - 16 Thermocouple wire and tube.

Specifications of Heat Exchanger and Operations Condition

In this case, the model used was validated by the result of experiment performed by using a set of single staggered wire and tube heat exchanger with the following specifications:

- Height of heat exchanger (H) : 445 mm
- Diameter of tube outside (dto) : 4.8 mm
- Diameter of tube inside (dti) : 3.2 mm
- Diameter of wire (dw) : 1.2 mm
- Distance between wires (pw) : 7. 14, 21 mm,
- Number of tube : 12 coils
- Width of heat exchanger (W) : 431 mm
- Distance between wires : 40 mm

Operating condition:

- Fluid flow rate : 0.006 kg/s
- Entrance fluid temperature : 60^0C dan 70^0C
- Environment temperature : 30^0 C

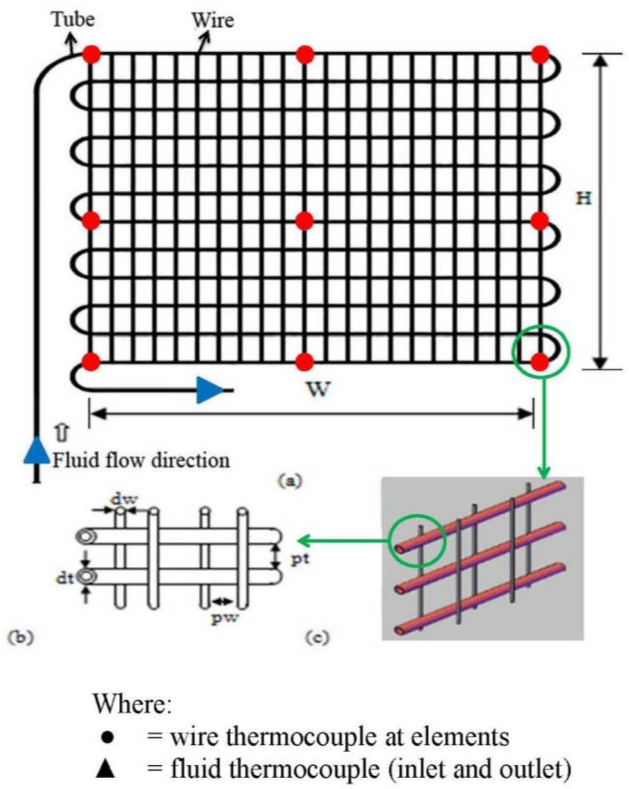


Figure 5: Heat exchanger geometry for validition

Model Validation

Validation performed by comparing the temperature of experiment result at the certain elements with the temperature of calculation at the same elements. For the heat exchanger with pw = 7 mm, the elements's model are at elements no: 1, 96, 283, 378, 474, 567, 6661, 756, $T_{out.}$ Then for pw= 14 mm, elements's model are at elements no: 1, 49, 96, 144, 192, 241, 288, 336, 384, T_{out}. But for pw = 21 mm, the elements are at elements no: 1, 34, 66, 99, 132, 198, 231, 264, T_{out}. The condition of calculation is equated as the condition of experiment.

Simulation Tool With CFD

This research process also used simulation of Computational Fluid Dynamics (CFD). Turbulent model used is equation of Reynold-Average Navier-Stokes (RANS). Air current on the surface of heat exchanger is turbulent and viscous. Turbulent model of RANS k-omega SST (Shear-Stress Transport) used in this simulation and also used turbulent model of standard k-omega as a standard of comparison. Turbulent model of k-omega SST has a high stability in numerical calculation and accurate flow prediction on adverse pressure gradient in boundary layer area [11]. Any equations used as basic in mass and heat transfer modelling on surface of wire and tube heat exchanger are: energy, mass conservation, and momentum.

In this case, model used will be validated by the experiment result performed it self using a set of heat exchanger with the following dimentions:

- Height of heat Exchanger (H) : 445 mm
- Diameter of tube outside (dto) : 4.8 mm
- Diameter of tube inside (dti) : 3.2 mm
- Diameter of wire (dw) : 1.2 mm

Whereas parameter of operating conditions as follows:

- Fluid flow rate : 0.006 kg/s
- Entrance fluid temperature : 60^0C dan 70^0C
- Environment temperature : 30^0 C

RESULTS AND DISCUSSION

Developing of Model

This study developed a model of a single staggered wire and tube heat exchanger by performing the following steps:

1) Determination of one unit modelling element.

One element used in this model is a tube as long as pitch wire, and a wire as a fin as long as pitch tube (pt) with entrance heat flow and with each element, as the following figure:

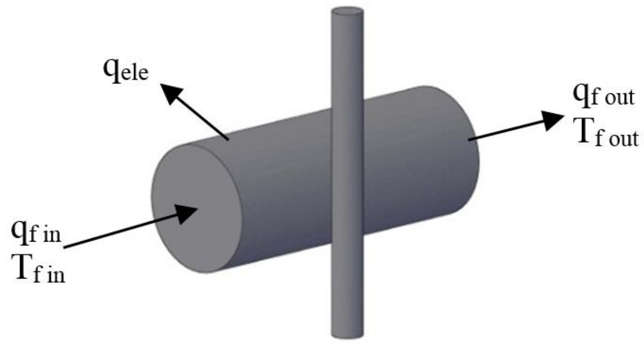


Figure 6: One unit element in single staggered wire and tube heat exchanger

2) Analyzing one unit element using finite element,, it obtained energy balance as the following figure:

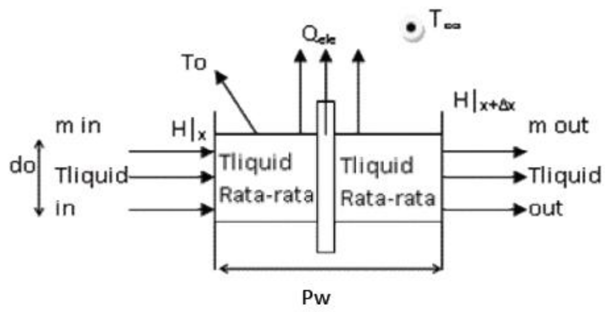


Figure 7: One unit element on finite element method

3) Making flowchart in purpose to evaluate the capacity of heat exchanger and convection heat transfer coefficient.

4) Writing programming language of MATLAB.

In this phase, this study succeeded to develop MATLAB program which can be applied to calculate temperature and total heat transfer capacity based on the number of heat transfer capacity of all unit element [14].

5) Running the program to perform the model simulation in purpose to get the distribution of heat exchanger surface temperature. In this step, the simulation was done by geometry factor variation, which is the distance between wire are 7 mm, 14 mm, and 21 mm.

For example, the result of simulation for heat exchanger model with pw = 7 mm at entrance fluid temperature of 60°C, so it obtained the distribution of element calculation temperature as described on the following figure.

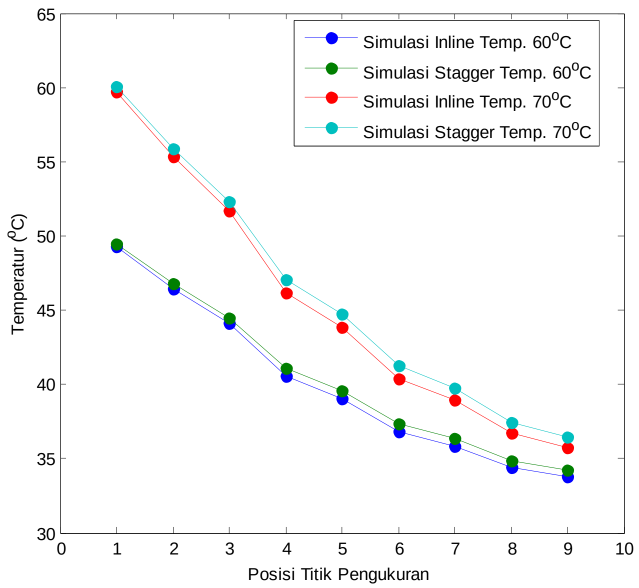


Figure 8: Temperature distribution on certain elements of heat exchanger based on simulation

6) Performing experiment to validate the developed model.

Model Validation with The Results of Experiment

Validation performed by comparing the temperature of experiment result at the certain elements with the temperature of calculation at the same elements. For the heat exchanger with pw = 7 mm, the elements's model are at elements no: 1, 96, 283, 378, 474, 567, 6661, 756, T_{out}. Then for pw= 14 mm, elements's model are at elements no: 1, 49, 96, 144, 192, 241, 288, 336, 384, T_{out}. But for pw = 21 mm, the elements are at elements no: 1, 34, 66, 99, 132, 198, 231, 264, T_{out}. The condition of calculation is equated as the condition of experiment.

As an initial step in validation, initial trial was performed towards distribution condition of heat exchanger's surface temperature. The following figure is the observation result of mapping infrared thermography:

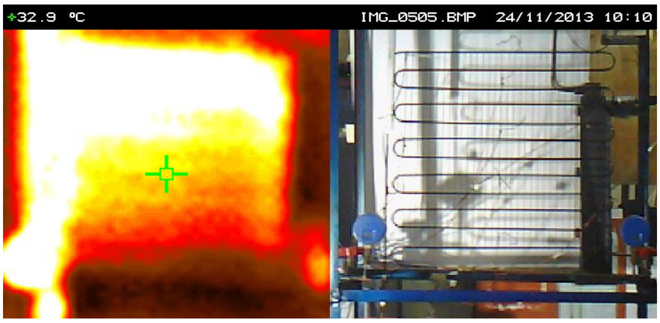


Figure 9: Mapping of surface temperature distribution of heat exchanger using infrared thermography

This figure above defines the result of infrared thermography photo which shows surface hot area of heat exchanger after being flown by hot fluid (hot oil). White area is a part of surface with high temperature area, because this area is a part of entrance fluid to heat exchanger which the fluid temperature is relatively high. Then the yellowish area which has medium temperature is a middle part of overall heat exchanger surface area. Meanwhile the part of red color is the area which has low temperature because fluid leaves the heat exchanger in this part. By this early description, it could be obtained that temperature is decrease in parallel to the release heat which performed by the array of wires which serve as fins, so then the cold fluid leaves the heat exchanger.

Then it was validated by the result of CFD visualization, and the result described in the following figure:

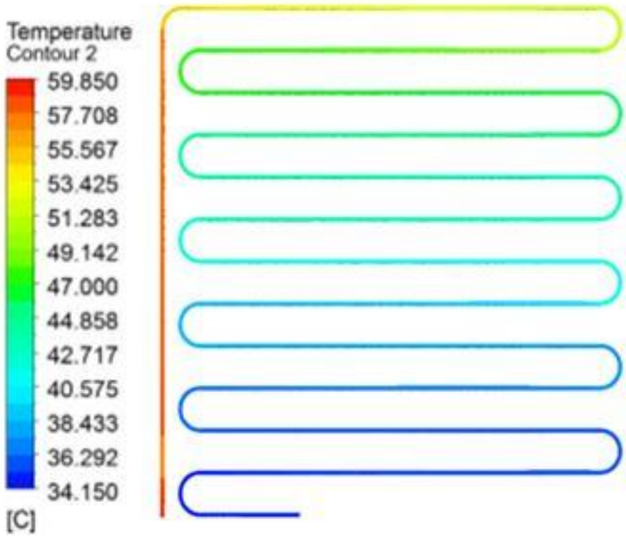


Figure 10: The result of CFD simulation on fluid temperature profile in tubes

Based on figure above, it can be seen that the behavior of fluid temperature in tube decreases since entering inlet and then leaving outlet. The entrance temperature was in 60°C (visualized by red yellowish) then it decreased in the first track with the visualization of yellowish colour, then temperature was released as long as the following tube track (visualized by the change from green to blue) and finally comes out from outlet with low temperature about of 34°C (visualized by dark blue colour). The result of this CFD visualization validated the result of data sampling with infrared thermography, which the temperature is really decreasing as long as tube track because of the heat release by the array of tube and wire.

Then the trend of decreasing temperature in each track is described on the following figure:

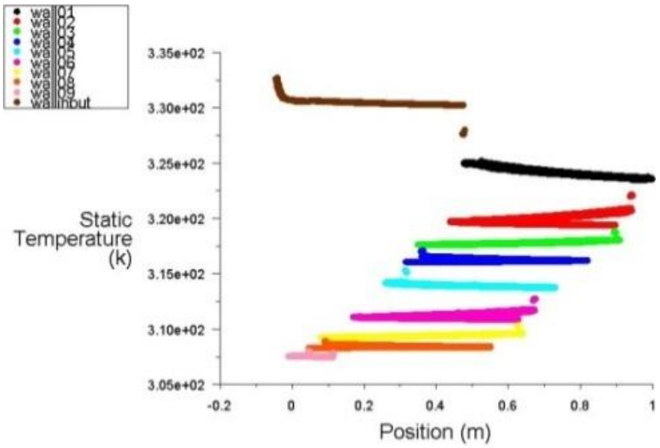


Figure 11: Temperature drop profile on internal flow of heat exchanger's tube

The figure above shows the trend of decreasing temperature in each track of the whole track (coils) of heat exchanger's tube which is in amount of 12 tracks. The hot fluid entered vertical track at first in temperature about 60°C (visualized by brown), then the temperature dropped to the first track of horizontal tube (visualized by black), and then fluid flow entered track by track so that it came out from the last track in low temperature (visualized by yellow reddish). On some track passed by fluid, there was the most temperature drop which has the biggest gradient, on the second track.This case occurs because on the second track all fins (wires) has functioned to release heat, which there are only half fins on the first track so that the temperature drop occurs less.

After inclination of temperature characteristics or temperature behavior on internal flow of heat exchanger generally identified, then the measurement of surface temperature distribution is performed accurately by using thermocouple at each element point of heat exchanger. The following table describes the temperature of element points (9 measurement points) for heat exchanger with pw (pitch wire) = 7 mm at entrance fluid temperature in amount of 60°C.

Based on the observation result data at 9 thermocouple points located at the elements above, then it was identified the difference with its modelling result data. The analysis result shows the error percentage between the observation and modelling result data is 1.36%. And the temperature profile of each point based on the observation and modelling can be presented in the following figure.

Table I: Distribution of element temperature at heat exchanger with pw= 7 mm

Number of Elements	*Experimental Temp.*	*Calculation Temp.*
1	49.94649	49.425
96	46.20797	46.78
189	44.43979	44.51
283	42.98776	41.093
378	40.53906	39.608
474	37.79227	37.355
567	36.10708	36.385
661	35.28887	34.8972
756	34.29777	34.255

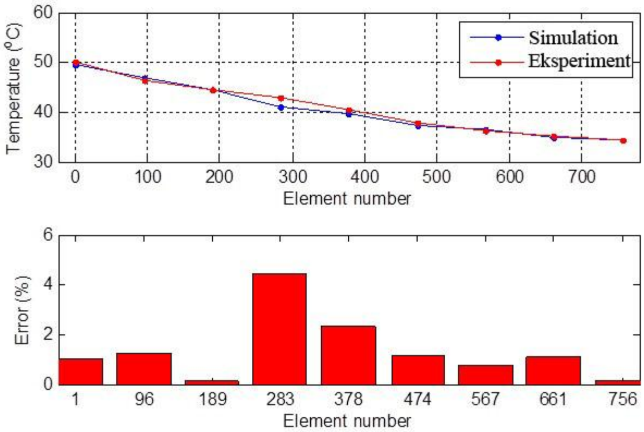


Figure 12: Temperature validation graphic on each element for heat exchanger with pw = 7 mm

On the other hand, for heat exchanger with pw = 14 mm obtained that element temperature distribution for 9 points thermocouple on those elements can be descibed on the following table. These datas taken on entrance fluid temperature of 60°C and surrounding temperature of 30°C.

Table II: Element temperature distribution on heat exchanger with pw = 14 mm

Number of Elements	*Experimental Temp.*	*Calculation Temp.*
1	48,75064	49,6487
34	46,00827	47,218
66	42,55736	45,104
99	42,35471	41,84
132	38,34888	40,395
166	36,25591	38,155
198	34,60221	37,17
231	33,52144	35,6335
264	32,14177	34,957

Based on the analysis obtained that the error between modelling and experiment result was 4.96%. The following figure shows temperature profile on each element point based on the experiment and modelling result.

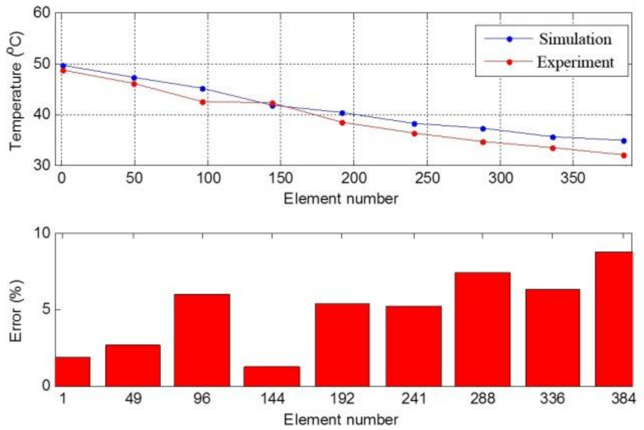


Figure 13: Temperature validation graphic on each element for heat exchanger with pw = 14 mm

Based on both data verification between the experiment and modelling result data, it was identified that its error percentage is relatively small which is lower than 5%. It means that the model developed is considered as good enough and accurate. On the other hand, the previous research performed by Basal and Chin (2003) which used heat load validation method got deviation in amount of 10%. It means that the result of this new research validation is accurate.

The Influence of Heat Exchanger Geometry Towards Heat Exchanger Capacity

The influence of diameter wire (dw) and pitch wire (pw) towards how much heat transferred will be presented on the following table:

Table III: The influence of pw and dw towards Q_{tot}

dw/pw	***0.005***	***0.007***	***0.008***	***0.009***	***0.01***	***0.011***	***0.012***
0.0008	120.86	120.2507	119.7495	119.3572	119.3819	119.055	118.3943
0.0009	121.513	121.0994	120.6373	120.2519	120.2756	119.9214	119.2145
0.001	121.791	121.8046	121.4058	121.0528	120.5808	120.7312	119.9919
0.0011	122.296	122.3841	121.0586	121.7564	121.3069	121.4778	120.7202
0.0012	122.808	122.8627	122.6061	122.3646	121.9526	122.157	121.3947
0.0013	123.356	123.2683	123.064	122.884	122.5189	122.767	122.0125
0.0014	123.959	123.6277	123.4509	123.3249	123.0101	123.3083	122.5717
0.0015	124.624	123.965	123.786	123.6999	123.4329	123.7839	123.0733

Based on table 3 obtained that when the condition of optimization factor is maximum [15], the heat transferred is only 119.9214 watt obtained on pw = 0.011 m and dw =0.0009 m. Table 3 also shows that the smaller the distance between wires, the bigger Q_{total}, and the increase of diameter will also makes Q_{total} increases. However when pw < = dw, the system will be considered as tube which its fin has the form of plate, and it means Q_{total} will be lower.

The following graphic defines the ability of releasing heat from each heat exchanger.

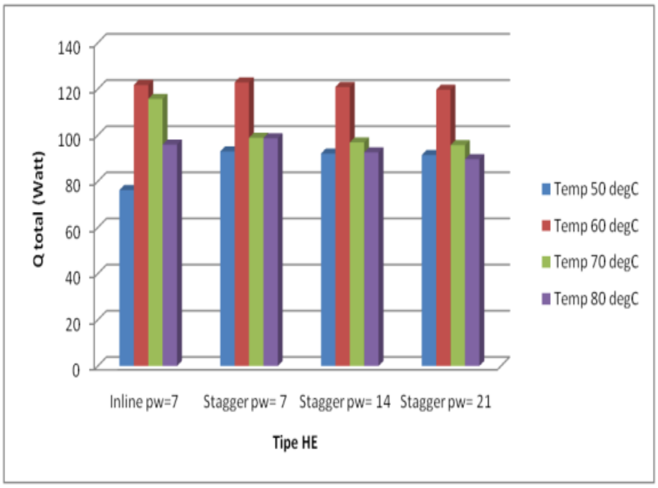


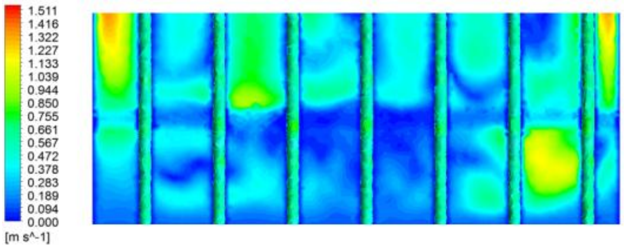
Figure 14: The influence of heat exchanger geometry to heat transfer rate.

From the graphic above can be seen the rate of total heat transfer released by each heat exchanger. It is seen that heat exchanger pw = 7 mm inline (basis) releases the highest heat rate because this heat exchanger has the biggest heat transfer surface. Then it is continuously followed by heat exchanger pw = 7 mm staggered, heat exchanger pw = 14 mm staggered and the last heat exchanger pw = 21 mm staggered are suitable for total surface area of each heat exchanger. The interesting thing which can be seen from the figure above is the heat exchanger pw = 7 staggered with less number of wires but it can release heat that is not much different from basis heat exchanger which has much more number of wires. This case means that heat exchanger pw = 7 staggered has a high optimization factor because by having the ability of releasing big enough heat with material mass which is low enough.

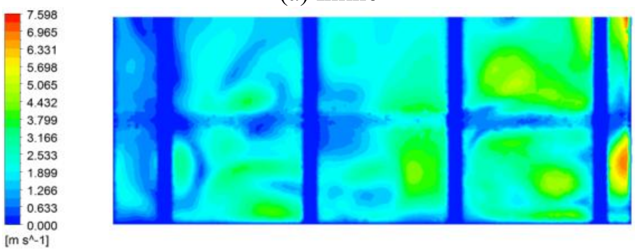
By comparing these two results, it can be identified that changing wire geometry into the array of single staggered gives much more optimum condition. It occurs because practically the staggered design enables to reduce a wire into one for each element. Reducing one wire doesn't only reduce the construction material mass but also influences the convection current becomes bigger, because it is not blocked by geometry wall as explained by the following phenomenons.

The figure above shows that air current speed distribution on the array of wire between inline and staggered heat exchanger, which current speed distribution on staggered heat exchanger is smoother and faster. This smooth current distribution is due to the heat flux of staggered heat exchanger is bigger than inline, however convection current speed is faster because of no blocking by geometry wall on inline heat exchanger. Stagnant current condition occurs on area under the tube's wall because of blocking current by existing two face-to-face inline wires.

Furthermore, when heat transfer phenomenon on wire and tube heat exchanger are viewed on cross sectional surface of wire and tube element, the visualization will be described as the following figures:

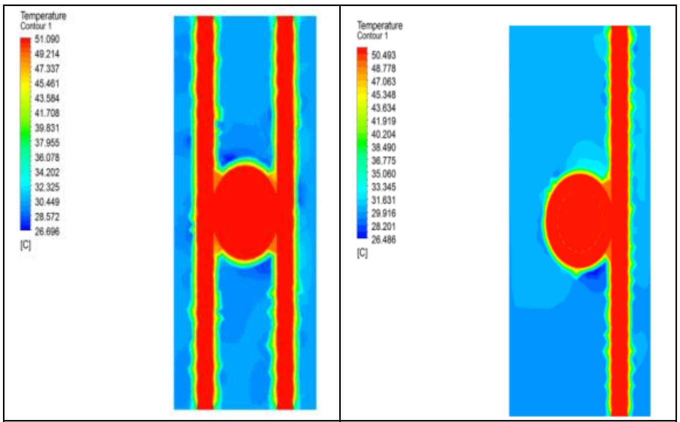


(a) Inline



(b) Staggered

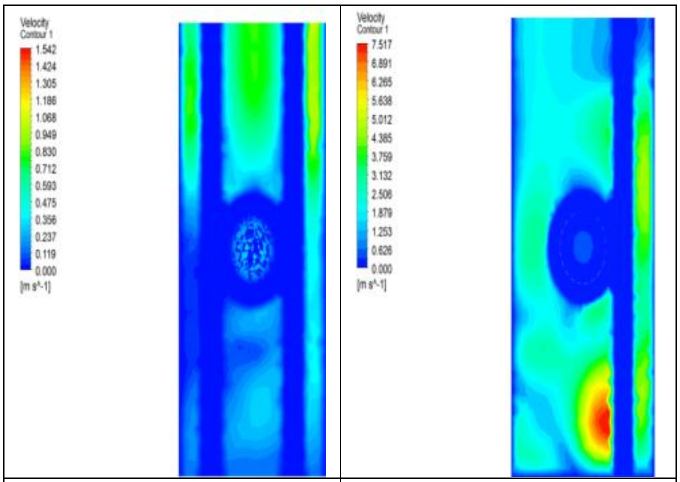
Figure 15: Air speed distribution on longitudinal wire element surface of heat exchanger



(a) (b)

Figure 16: Temperature distribution around cross sectional surface on the wire and tube element of heat exchanger; (a) inline pw= 7, (b) staggered pw= 7

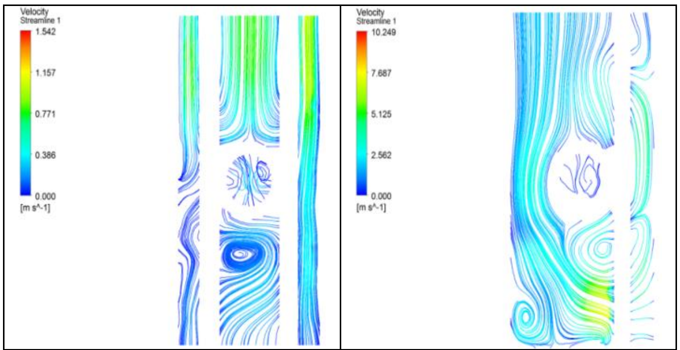
The figure above shows that on the area around staggered wire, there is a higher temperature distribution around 31°C. It is because heat flux of staggered heat exchanger is bigger than these of inline heat exchanger. However, there is a lower temperature distribution around 30°C on inline heat exchanger. Then when the air current speed distribution is viewed around cross sectional surface on the wire and tube element of heat exchanger, the visualization will be described on the following figure.



(a) (b)

Figure 17: Air curremt speed distribution around cross sectional surface on the wire and tube element of heat exchanger; (a) inline pw= 7, (b) staggered pw= 7

This figure shows that the current speed distribution of convection fluid is smoother and higher averagely in the staggered heat exchanger. And also by average, convection current speed is about 0.5 m/s, even the area which is near the wire has a higher value. This case occurs because geometric factor and heat flux are better on staggered heat exchanger. On the other hand, it shows the different thing that inline heat exchanger has a current speed distribution about 0.2 m/s. Besides it is not smooth and also makes a circular current because of the influence of inline wire geometry. This condition will cause the convection current to be less so that convection heat transfer becomes lower.
The phenomenon above is then clearly shown by the following figures of streamline fluid current speed.

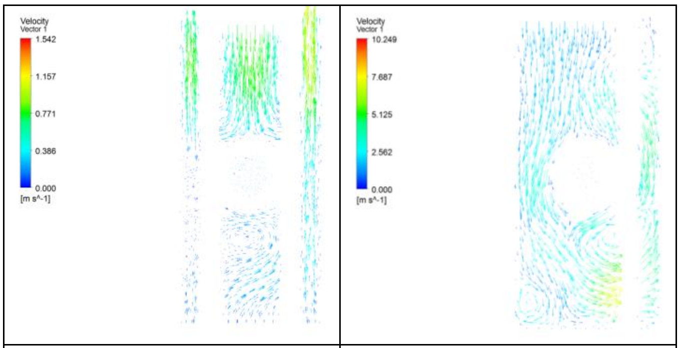


(a) (b)

Figure 18: Streamline air current speed around cross sectional surface on the wire and tube element of heat exchanger; (a) inline pw= 7, (b) staggered pw= 7

The figure above shows that field or streamline current becomes more on staggered heat exchanger. On the other hand, on inline heat exchanger, there is a circular current in low speed especially on wall area under the array of tube which makes convection current become smaller. It is because of the influence of inline heat exchanger geometry which is the area between two wires that makes the current tend to be stagnant. The different case occurs on staggered heat exchanger, which has a small current restriction so that the current dominantly becomes a streamline.

Furthermore, in order to clarify the streamline current, it can be seen on this visualization of current speed vector:



(a) (b)

Figure 19: Air current speed vector around cross sectional surface on the element wire and tube element of heat exchanger; (a) inline pw= 7, (b) staggered pw= 7

The figure above shows the air speed vector around the surface of staggered wire and tube heat exchanger happens in front side of wire by downwards and circulates in a high speed on the below of tube area and finally it makes vector directs to upwards on the back side of wire. It indicates that convection current on staggered heat exchanger becomes better. However on the inline heat exchanger, the vector of convection current directs to downwards on the area between two inline wires, then going up on the outside of both wires. But it has blocking current because of two face-to-face geometry wires. Then the current under the tube position has a collision of the current vector in low speed, so that this case will damage the movement of convection current.

CONCLUSIONS

Based on data of the research results and analysis performed, it can be concluded some important things dealing with the modelling of single staggered "wire and tube heat exchanger", which are:

1) The model developed to present heat transfer correlation at single staggered wire and tube heat exchanger is one heat exchanger element, which consists of one tube as long as pitch wire and one wire as long as pitch tube, was valid. Validation performed with the approach of element temperature parameter at 9 points thermocouple with the error is less than 5% between the experiment and modelling resuts.

2) From the result of CFD analysis obtained that by average, the speed distribution of convection fluid current was smoother and higher on the single staggered wire and tube heat exchanger so that it makes the convection heat transfer better.

ACKNOWLEDGMENT

This research was funded by a research grant through the Higher Education Department of Indonesia, 2014. The authors express their sincere thanks for the financial support.

REFERENCES

[1] Tagliafico, L., and Tanda, G.: Radiation and natural convection heat transfer from wire-and-tube heat exchangers in refrigeration appliances, *International Journal of Refrigeration*, 1997, 20 (7), pp. 461-469.

[2] Witzell, O.W. dan Fontaine, W.E.: What are the heat transfer characteristics of wire and tub condensers?, *Journal of Refrigerating Engineering*, 1957, 65, pp. 33-37.

[3] Witzell, O.W. dan Fontaine, W.E.: Design of Wire and Tube Condensers, *Journal of Refrigerating Engineering*, 1957, 65, pp. 41 - 44.

[4] Cyphers, J. A., Cess, R. D., Somers, E. V.: Heat Transfer Character of Wire and Tube Heat Exchangers, 1959.

[5] Collicott, H. E., Fontaine, W. E., Witzell, O.W.: Radiation and Free Convection Heat Transfer from Wire and Tube Heat Exchangers, *ASHRAE Journal,* 1963, 5, pp.79-83.

[6] Tanda, G, dan Tagliafico, L.: Free Convection Heat Transfer From Wire and Tube Heat Exchangers, *Journal of Heat Transfer*, 1997, 119, pp. 370 - 372.

[7] Lee, T.H., Yun, J.Y., Lee, J.S., Park, J.J., and Lee, K.W.: Determination of airside heat transfer coefficient on wire-on-tube type heat exchanger, *International Journal of Heat and Mass Transfer*, 2001, 44, pp. 1767-1776.

[8] Quadir G.A., Krishnan G.M.,Seetharamu K.N: Modeling of wire-on-tube heat exchangers using finite element method", *Finite Element in Analysis and Design*, 2002, 38, pp. 417-434.

[9] Bansal, P.K., and Chin, T.C.: Modelling and optimisation of wire-and-tube condenser, *International Journal of Refrigeration*, 2003, 26, pp. 601-613.

[10] Kumara, Pradeep: Optimization Of Performance And Assessment Of Material Cost Of The Refrigerator Condenser, *Master Of Science Thesis*, KTH School of Industrial Engineering and Management, 2011.

[11] Incropera, Frank P., DeWitt, David P.: *Fundamentals of Heat and Mass transfer*, John Wiley & Sons, New York, 1990.

[12] Tuakia, Firman.: *Dasar-dasar CFD Menggunakan Fluent*, Informatika Bandung, Bandung, 2008.

[13] Blazek, J.: *Computational Fluid Dynamics: Principles and Applications*. Elsevier Science Ltd: United Kingdom, 2001.

[14] Kwon Young W., Bang Hyochoong.: *The Finite element methode using MATLAB,* 3rd edition, CRC Press, New York, 1997.

[15] Arsana, I, M.: *Optimasi Penukar Panas Jenis Pembuluh Dan Kawat Berseling Tunggal*, *Laporan Penelitian Disertasi Doktor Universitas Negeri Surabaya*, Surabaya, 2015.