Nahrain University, College of Engineering Journal (NUCEJ) Vol.16 No.1, 2013 pp.20 - 29

Experimental and numerical investigation of heat transfer and flow characteristics for tractor heat exchanger (radiator)

Kadhim F. nasir
Mechanical engineering

Abstract

An experimental and numerical study was carried out to investigate heat transfer and flow characteristics through the tractor heat exchanger (radiator). The engine of the tractor is Diesel (compression ignition) engine type. The experimental study used heat exchanger (radiator) manufactured from copper instead of the original heat exchanger of the tractor, which is manufactured from iron. Both heat exchangers consist of circular cross sectional tubes and plate fins. The temperatures was measured inside the cooling room of engine for both cases at the same time and the same loads. The results of the experimental investigation showed that the heat exchanger is manufactured from copper provided temperatures less than the heat exchanger manufactured from iron, that means heat rejected by copper heat exchanger is more than heat rejected by iron heat exchanger. The use of copper heat exchanger is better than iron heat exchanger for cooling the tractor engine. In the numerical study used model of copper heat exchanger to analyzed by CFD method at different air velocity from (0.5 m/sec) to (7 m/sec) for laminar air flow. The object of this study was enhancement the heat transfer process from the cooling fluid of heat exchanger. The results of the numerical study shows that heat transfer rate increase with increasing air flow during heat exchanger (radiator). Good agreement between experimental and numerical results is obtained.

Key words: heat exchanger, cooling, fluid, temperature, copper, iron, tube, fins

Introduction

A heat exchanger is a device that is used for transfer thermal energy between two or more fluids, between a solid surface and a fluid, or between solid particulates and a fluid, at differing temperatures and in thermal contact, usually without external heat and work interactions. The fluids may be pure or mixtures. Typical applications involve heating or cooling of a fluid stream of concern, evaporation or condensation of a single or multi component fluid stream, and heat recovery or heat rejection from a system. In other applications, the objective may be to sterilize, pasteurize, fractionate, distill, concentrate, crystallize, or control process fluid. In some heat exchangers, the fluids exchanging heat are in direct contact. In other heat exchangers, heat transfer between fluids takes place through a separating wall or into and out of a wall in a transfer manner. [shah(1995)]

In heat exchangers the temperature of each fluid changes as it passes through the exchangers, and hence the temperature of the dividing wall between the fluids also changes along the length of the exchanger. Examples of heat exchangers include intercooler and preheaters, condensers and boilers in steam plant, condensers and evaporators in refrigeration units, regenerators, automobile radiators oil coolers of heat engine, milk chiller of pasteurizing plant, and several other industrial processes. [Rajput (2007)]

Type of heat exchangers: in order to meet the widely varying applications, several types of heat exchangers have been developed which are classified on the basis of nature of heat exchange process, relative direction of fluid motion, design and constructional features, and physical state of fluids. The basis of nature of heat exchange process, are classified as direct contact heat exchangers and indirect contact heat exchangers, in direct contact or open heat exchanger the exchange of heat takes place by direct mixing of hot and cold fluids and transfer of heat and mass takes place simultaneously, in indirect contact heat exchangers the heat transfer between two fluids could be carried out by transmission through wall which separates the two fluids. According to the relative directions of two fluid streams the heat exchangers are classified into three categories: parallel flow , counter flow, and cross flow, in the parallel flow exchanger, the two fluid streams (hot and cold) travel in the same direction. The two streams enter at one end and leave at the other end. The flow arrangement of the fluid streams in case of parallel flow heat exchangers are shown in figure.(1a). In a counter heat exchanger, the two fluids flow in opposite directions. The hot and cold fluids enter at the opposite ends. The flow arrangement for such a heat exchanger are shown schematically in figure.(1b).

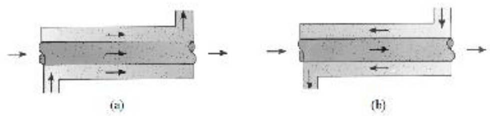


Figure (1) parallel flow heat exchangers [Mills, A.F.(1992)]

In cross flow heat exchangers, the two fluids (hot and cold) cross one another in space, usually at right angels, figure(2) shows a schematic diagram of common arrangement of cross flow heat exchangers.

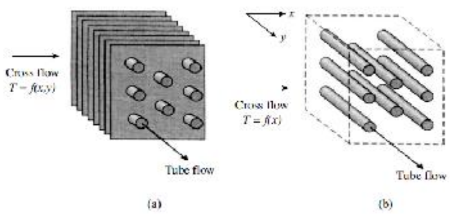


Figure (2) cross flow heat exchangers [Mills, A. F(1992)]

On the basis of design and constructional features are classified as under: concentric tubes, shell and tube, multiple shell and tube passes, and compact heat exchangers. In concentric tubes type, two concentric tubes are used, each carrying one of the fluids. This direction of flow may be parallel or counter, the effectiveness of the heat exchanger is increased by using swirling flow. In shell and tube type of heat exchanger one of the fluids flow through a bundle of tubes enclosed by a shell. The other fluid is forced through the shell and it flows over the outside surface of the tubes as shown in figure (3). For both type of radiators we used the same fan and water pump in the engine.

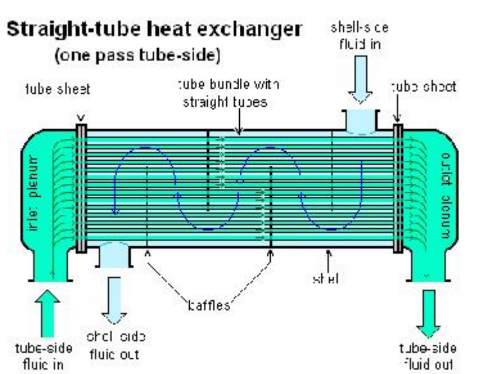


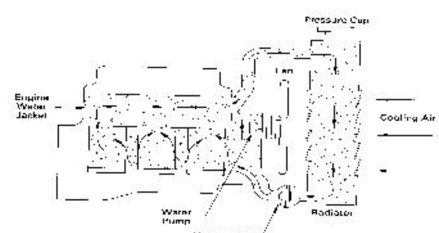
Figure (3) Shell and tube heat exchangers [Mills, A. F.(1992)]

Multiple shell and tube passes are used for enhancing the overall heat transfer, compact heat exchangers are special purpose heat exchangers and have a very large transfer surface area per unit volume of the exchanger. Heat exchangers depending upon the physical state of fluids are classified as condensers and evaporators. [Rajput (2007)]

Radiator

Radiator is a liquid to air heat exchanger of honeycomb construction used to remove heat from the engine coolant after the engine has been cooled. The radiator is usually mounted in front of the engine in the flow of air as the automobile moves forward. An engine driven fan is often used to increase air flow through the radiator.
[Pulkrabek(1996)]

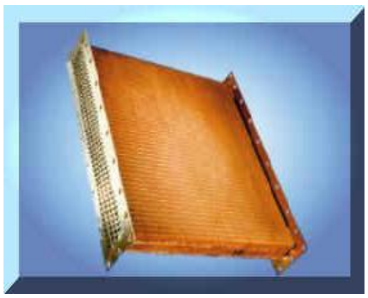
The tubes sometimes have a type of fin inserted into them called a turbulator, which increases the turbulence of the fluid flowing through the tubes. If the fluid flowed very smoothly through the tubes, only the fluid actually touching the tubes would be cooled directly. The amount of heat transferred to the tubes from the fluid running through them depends on the difference in temperature between the tube and the fluid touching it. So if the fluid that is in contact with the tube cools down quickly, less heat will be transferred. By creating turbulence inside the tube, all of the fluid mixes together, keeping the temperature of the fluid touching the tubes up so that more heat can be extracted, and all of the fluid inside the tube is used effectively.
Radiators usually have a tank on each side, and inside the tank is a transmission cooler. In the picture above, you can see the inlet and outlet where the oil from the transmission enters the cooler. The transmission cooler is like a radiator within a radiator, except instead of exchanging heat with the air, the oil exchanges heat with the coolant in the radiator.[Karim Nice (2009)]



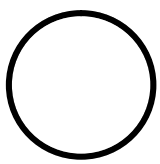
Figure(4) Radiator of liquid cooled engine[LIUEDAHL (1979)]

Experiment model

In this work an automobile radiator was designed and built from copper for compression ignition (CI) engine , instead of iron. This radiator is a cross flow unmixed heat exchanger. The geometry of this radiator consists of a circular cross section area of tubes, and plate fins as shown in figure (4). We tests this radiator experimentally using the water as cooling fluid for several engine loads for long time and measured the temperature of the cooling fluid in the engine cooling room. The old radiator which manufactured from the iron had a circular cross section of tubes used for the same purpose and using the same cooling fluid for the same loads and time in the same engine.



(a)



(b)

Figure(5) (a) the manufactured radiator, (b) the cross section of new copper tubes

,

Mathematical analysis and governing equations

Heat exchangers usually operate for long periods of time with no change in their operating condition. Therefore, they can be modeled as steady- flow devices. As such, the mass flow rate of each fluid remains constant. Also, the fluid streams experience little or no change in their velocities and elevations. And thus the kinetic and potential energy changes are negligible. The specific heat of fluid, in general, changes with temperature. But , in a specified temperature ranges, it can be treated as a constant at some average values with little loss in accuracy. Axial heat conduction along the tube is usually insignificant and can be considered negligible. Finally, the outer surface of the heat exchanger is assumed to be perfectly insulated, so that there is no heat loss to the surrounding medium, and any heat transfer occurs between the two fluids only.[[Cengel (2002)]

The heat exchanger model and performance parameters used in characterizing heat transfer and pressure. The model heat exchanger for this project is presented, and information about the heat exchanger, fin-and-tube efficiency, pressure drop, and the dimensionless groups used in the calculation process are presented.

The governing equations are the continuity, Navier-Stokes for momentum, energy, and scalar transport equations for steady-state flow, and can be written as follows:

Continuity equation:

$$\frac{\partial(\circ u_i)}{\partial x_i} = 0. \qquad 1$$

Momentum equation:

$$\frac{\partial}{\partial x_i}(\circ u_i u_j) = \frac{\partial}{\partial x_i}\left(\ll \frac{\partial u_j}{\partial x_i}\right) - \frac{\partial p}{\partial x_j}. \qquad 2$$

Energy equation:

$$\frac{\partial}{\partial x_i}(\circ u_i T) = \frac{\partial}{\partial x_i}\left(\frac{k}{C_p}\frac{\partial u_j}{\partial x_i}\right). \qquad 3$$

General transport equation (for scalars): 4

$$\frac{\partial(\circ u_i ¤)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\Gamma_{¤} \frac{\partial ¤}{\partial x_i}\right] + S_{¤} -$$

The general equations 1-3 are used in the CFD computations to calculate the flow field for both thermal and fluid (air) dynamics, solving for heat transfer and pressure drop.

There are limitations in using fin-and-tube heat exchangers. Normally one side must be a gas or liquid with a low heat transfer coefficient . They are difficult to mechanically clean, requiring non-corrosive clean fluids. Temperature and pressure limits are lower than some other types due to brazing or mechanical expansion when joining the fins to the tubes (though pressure can be high on the tube side).[Rohsenow et. al, 1998].

Fins are commonly made of aluminium, while tubes are made of copper. Typical geometry: fin thickness 0.11 to 0.13 mm, tubes with outside diameter 10 mm, transverse pitch 25 mm, longitudinal pitch 22 mm, and fin density of 6 to 16 fins per inch. (However, smaller tube spacing and tube diameters are becoming more widespread). The geometry of the heat exchanger in this project is close to the typical geometrical ranges listed above. [Baggio and Fornasieri, 1994] .

A diagram of the studied model is shown in Figure (5) shows the fine used in this work, and consists of the air flow area between two fins of plain fin geometry and around the surfaces of two rows of tubes.

Figure (5)

The overall heat transfer rate $\dot{Q}$ can be found in terms of the heat capacity rates ($\dot{m}C_p$) and temperature differences between the inflow and outflow of the hot or cold side:

$$\dot{Q}=(\dot{m}C_p)_h(T_{h,in}-T_{h,out})=(\dot{m}C_p)_c(T_{c,out}-T_{c,in}) \quad 5$$

$$\dot{Q}=UA\Delta T_m \quad 6$$

where ΔT_m refers to the mean temperature difference.

Efficiency determinations are used to find the over all heat transfer coefficient *U* for design and analysis of heat exchangers. The three methods most used for determining this are: ε-NTU, P-NTU, and LMTD. They are briefly described below. The ε-NTU method is described in more detail, since this is the method used for calculating efficiency in this work.

ε-NTU Method: The efficiency factor **ε** is a function of NTU, C* (ratio of minimum heat capacity rate to maximum heat capacity rate) , and flow arrangement.

P-NTU Method: The efficiency factor P is a function of NTU, R (ratio of temperature difference on one side to temperature difference on the other), and flow arrangement. This is similar to the ε-NTU method, but uses R instead of C*.

LMTD Method: The efficiency factor F is a function of P, R, and flow arrangement, and is a ratio of actual mean temperature difference to the log-mean temperature difference (LMTD).

The ε-NTU method is the one used in the research article for validation. The calculations were based on unmixed-unmixed cross-flow configuration, and expressed as [Kays and London, 1998] [Wang et al., 2006]:

$$\varepsilon = 1-\exp\frac{NTU^{0.22}}{C^*[\exp(-C^*NTU^{0.78})-1]} \quad 7$$

$$C^*=\frac{C_{\min}}{C_{\max}}=\frac{(\dot{m}C_p)_{air}}{(\dot{m}C_p)_{water}} \quad 8$$

$$\varepsilon=\frac{\dot{Q}_{avg}}{\dot{Q}_{\max}}=\frac{\dot{Q}_{avg:(\dot{Q}_{wtr}+\dot{Q}_{air})/2}}{\dot{m}_{wtr}C_{p,wtr}(T_{wtr,in}-T_{air,in})} \quad 9$$

$$NTU=UA/C_{\min} \quad 10$$

NTU, or “Number of Transfer Units”**,** is the ratio of *UA* (overall conductance) to min. heat capacity rate C_{min}. *NTU* determines the ‘thermal size’ of a heat exchanger. The overall heat transfer resistance is determined with the following equation:

$$\frac{1}{UA}=\frac{1}{\eta_0 h_o A_o}+\frac{\delta_w}{k_w A_w}+\frac{1}{h_i A_i} \quad 11$$

To find the water-side heat transfer coefficient h_i, the Gnielinski correlation is used [Gnielinski, 1976] [Wang et al., 2006]:

$$h_i=\left(\frac{k}{D}\right)_i\frac{(\mathrm{Re}_i-1000)\Pr(f_i/2)}{1+12.7\sqrt{f_i/2}(\Pr^{2/3}-1)} \quad 12$$

$$f_i=(1.58\ln(\mathrm{Re}_{D_i})-3.28)^{-2} \quad 13$$

The fin surface effectiveness η_0 is the ratio of actual heat transfer to the maximum possible heat transfer which could occur if the base and fin are both at the same temperature. This is described in the following equation set, where A_0 = A_{fin} + A_{base} and the actual fin efficiency η is calculated using the Schmidt approximation for staggered plate-fin geometry [Schmidt, 1949] [Wang et al., 2006]:

$$\eta_o=1-\frac{A_f}{A_o}(1-\eta) \quad 14$$

$$\eta=\frac{\tanh(mr\phi)}{mr\phi} \quad 15$$

$$m=\sqrt{\frac{2h_o}{k_f t}} \quad 16$$

$$\phi=\left(\frac{R_{eq}}{r}\right)\left[1+0.35\ln(R_{eq}/r)\right] \quad 17$$

$$\frac{R_{eq}}{r} = 1.27 \frac{X_M}{r} \left(\frac{X_L}{X_M} - 0.3 \right)^{1/2} \qquad 18$$

When the equations are solved and the air-side heat-transfer coefficient h_o has been calculated, the Colburn *j*-factor can be calculated by first solving for the Nusselt and Reynolds numbers, then using the calculated values in the final calculation for the *j*-factor. As shown previously in the following equations, the Nusselt number, Reynolds number, and *j*-factor are calculated [Rohsenow et. al, 1998] [Wang et al., 2006]

$$Nu = \frac{h_o}{k / D_h} \qquad 19$$

The Nusselt number is based on the hydraulic diameter D_h. There are different calculations for this available in the literature. The hydraulic diameter in this study is the ratio of the 4 times the minimum free flow air-side area to the wetted perimeter (ratio of air-side surface area to heat exchanger length), and is given by the following expression [Fornasieri and Mattarolo, 1991]:

$$D_h = \frac{4(F_p - t)(P_t - D_c)P_l}{2(P_l P_t - \pi D_c^2 / 4) + \pi D_c (F_p - t)} \qquad 20$$

$$\mathrm{Re} = \frac{\rho \cdot V \cdot D_c}{\mu} \qquad 21$$

where *V* is the minimum free-flow air velocity (in the minimum flow cross-section of the tube row), and is calculated:

$$V = V \cdot \left\{ \frac{P_t \cdot F_p}{P_t \cdot F_p - D_c \cdot F_p - t(P_t - D_c)} \right\} \qquad 22$$

$$j = \frac{Nu}{Re_{D_c} \cdot Pr^{1/3}} \qquad 23$$

The Prandtl number *Pr* is the ratio of a fluid's momentum diffusivity to thermal diffusivity.

$$\mathrm{Pr} = \frac{\nu}{\alpha} = \frac{\mu C_p}{k} \qquad 24$$

The Fanning friction factor is the ratio of wall shear stress to the flow kinetic energy. It is related to pressure drop in tube-and-fin heat exchangers as: The Fanning friction factor is the ratio of wall shear stress to the flow kinetic energy. It is related to pressure drop in tube-and-fin heat exchangers as: The Fanning friction factor is the ratio of wall shear stress to the flow kinetic energy. It is related to pressure drop in tube-and-fin heat exchangers as [Rohsenow et. al, 1998] [Wang et al., 2006]:

$$f = \frac{A_c}{A_o} \frac{\rho_m}{\rho_{in}} \left[\frac{2\rho_{in} \Delta p}{G^2} - (K_c + 1 - \sigma^2) - 2\left(\frac{\rho_{in}}{\rho_{out}} - 1 \right) + (1 - \sigma^2 - K_e) \frac{\rho_{in}}{\rho_{out}} \right] \qquad 25$$

Pressure Drop

The pressure drop determines the amount of pumping power needed to run a heat exchanger. It is therefore important to characterize the pressure drop for design. This section describes how the pressure drop relates to the pumping power, followed by a description of what causes the pressure drop and finally the pressure drop equations for tube-and-fin heat exchangers are presented.

Pumping power P is often seen as an important design constraint because the pressure drop in a heat exchanger (along with associated pressure drops in the inlet/outlet headers, nozzles, ducts, etc.) is proportional to the amount of fluid pumping power needed for the heat exchanger to function, as given by the following expression:

$$P = \frac{\dot{m} \Delta p}{\rho} \qquad 26$$

The overall pressure drop consists of two parts: (1) the pressure drop in the heat exchanger core, and (2) the pressure drop from associated devices the fluid flows through before and after the heat exchanger core (i.e. inlet/outlet manifolds, nozzles, valves, fittings, ducts, etc.).

The actual calculation for pressure drop depends on the specific type of heat exchanger being studied. For fin-and-tube heat exchangers, the pressure drop equation is given in as [Wang et al., 2006]:

$$\frac{\Delta p}{p_{in}} = \frac{G^2}{2g_c} \cdot \frac{1}{\rho_{in} p_{in}} \left[(K_c + 1 - \sigma^2) + 2\left(\frac{\rho_{in}}{\rho_{out}} - 1 \right) + f \frac{A}{A_c} \frac{\rho_{in}}{\rho_m} - (1 - \sigma^2 - K_e) \frac{\rho_{in}}{\rho_{out}} \right] \qquad 27$$

However, the entrance and exit loss effects K_c and K_e become zero when flow is normal to the tube banks or through wire matrix surfaces, resulting in the following equation:

$$\frac{\Delta p}{p_{in}} = \frac{G^2}{2g_c} \cdot \frac{\neg_1}{p_{in}} \left[(1 + \pm^2)\left(\frac{\neg_2}{\neg_1} - 1 \right) + f\,\frac{A}{A_c}\frac{\neg_m}{\neg_1} \right] \quad 28$$

G: $(u*\rho)$. G is the mass velocity entering the core based on minimum free-flow area.

g_c: A gravitational constant (equals 1 when working with SI units).

$\neg_1$: This is the specific volume (m3/kg) at inlet temperature.

$\neg_2$: This is the specific volume (m3/kg) at outlet temperature.

$\neg_m$: This is the average specific volume $(\neg_1 + \neg_2)/2$.

σ : Sigma represents the ratio of minimum free-flow area to frontal area.

A_c : Flow cross-sectional area.

Results

The CFD tools required for carrying out a simulation in order to solve a problem .Figure(6) shows the Numerical mesh of the system, figure (7) shows the Contours and vector plot for *V* velocity field at inlet air flow of 0.5 m/s. Figure (8) shows the Contours and vector plot for *V* velocity field at inlet air flow of 7 m/s. Figure (9) shows the air velocity with Reynolds number, the Reynolds number will be increasing with increasing the air velocity between the heat exchanger tubes due to accelerates of air flow. Figure (10) shows the air velocity with the pressure drop and the computation results compares with experimental results, the pressure drop will be increasing with increasing the air velocity due to accelerates of air flow. Figure (11) the Air velocity with Prandtl number, the Prandtl number values will be increasing with increasing the air velocity due increasing in specific heats of air flow. Figure (12) shows the Nusselt number, the Nusselt number values will be increasing with increasing in the air velocity. Figure (13) shows the Air velocity with air side heat transfer coefficient, air side heat transfer coefficient will be increasing with increasing the air velocity. Figure (14) shows the Air velocity with fin efficiency, fin efficiency will be decreasing with increasing the air velocity due to increasing in air side heat transfer coefficient. Figure (15) shows the Air velocity with fin surface effectiveness, fin surface effectiveness will be decreasing with increasing the air velocity due to decreasing in fin efficiency . Figure (16) shows the experiments results for iron and copper radiators, which presented the relation between the temperature of the cooling fluid in the engine cooling room and the time, this figure gives the experimental results compares between the iron and the copper heat exchangers that used in this test, we found a difference in temperature of cooling fluid in the engine cooling room at the same time of work, the temperatures of cooling fluid in the cooling engine room which used copper heat exchanger is lower than the temperatures in the cooling engine room that used iron heat exchanger. Figure (17) shows the Air velocity with heat transfer rate, the heat transfer rate will be increasing with increasing the air velocity. Figure (18) shows the Air velocity with friction factor and the computation results compares with experimental results, the friction factor will be decreasing with increasing the air velocity .

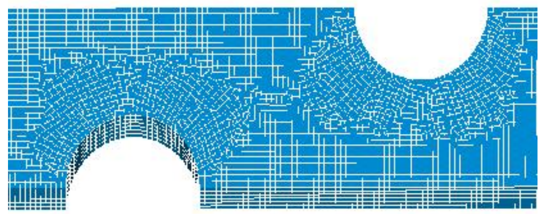
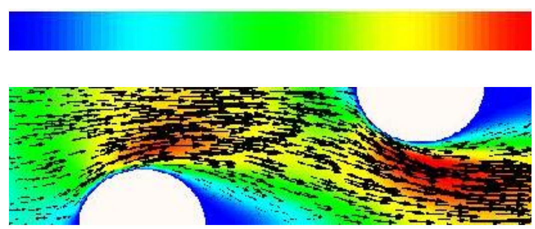


Figure (6): CFD computational domain for heat exchanger simulation .



Figure(7). Contours and vector plot for *V* velocity field at inlet air flow 0.5 m/s.

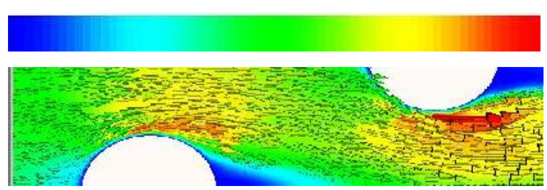


Figure (8) Contours and vector plot for *V* velocity field at inlet air flow 7 m/s.

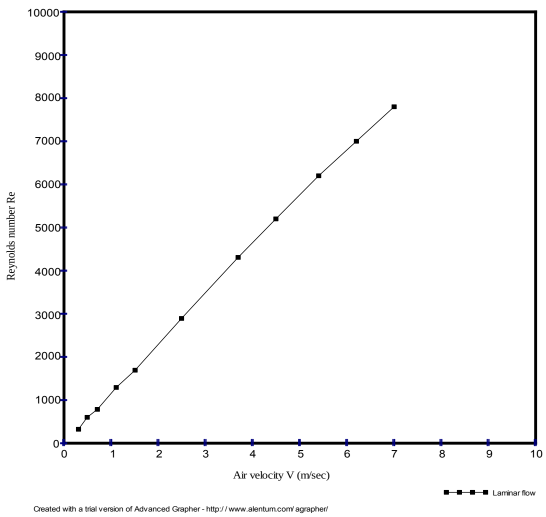
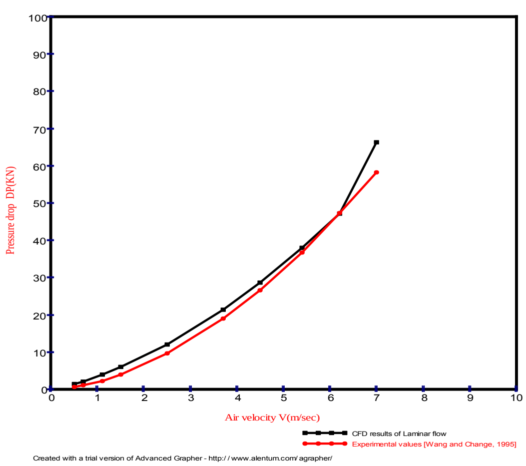
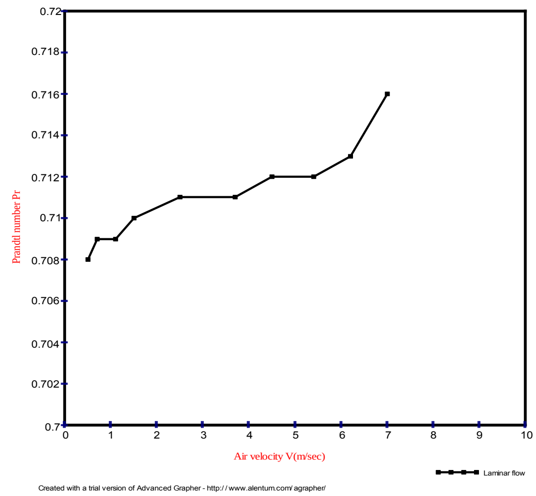


Figure (9) air velocity with Reynolds number.



Fig(10) Air velocity with pressure drop.



Figure(11) Air velocity with Prandtl number.

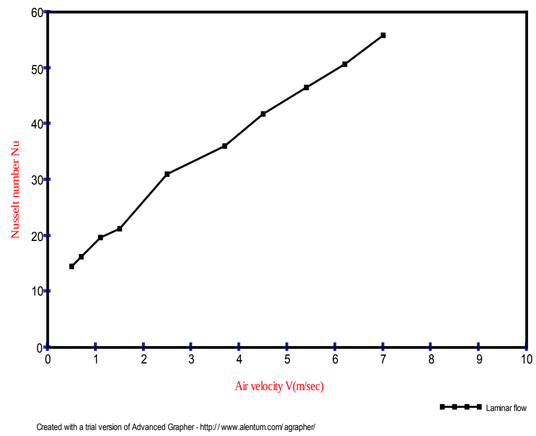


Figure (12) Air velocity with Nusselt number.

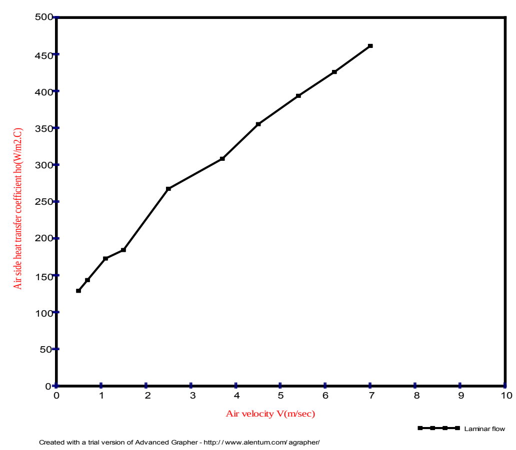


Figure (13) Air velocity with air side heat transfer coefficient

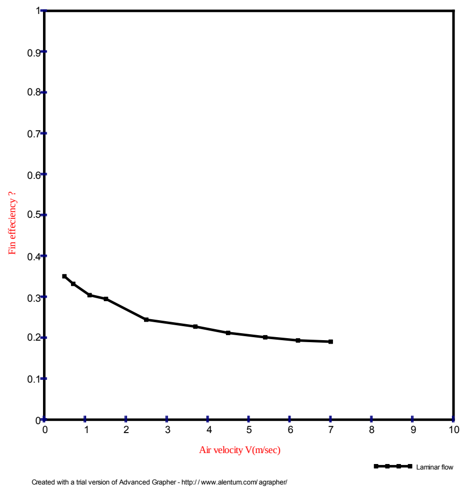


Figure (14) Air velocity with fin efficiency

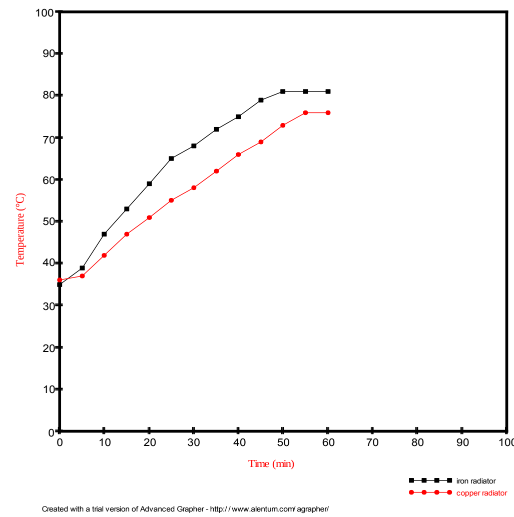


Figure (16) Experiments results for iron and copper radiators.

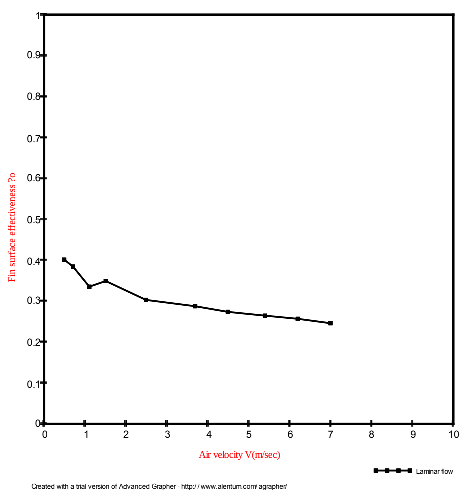


Figure (15) Air velocity with fin surface effectiveness.

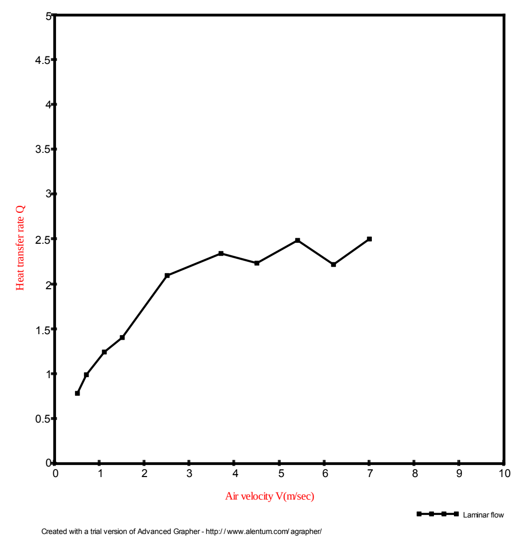


Figure (17) Air velocity with heat transfer rate.

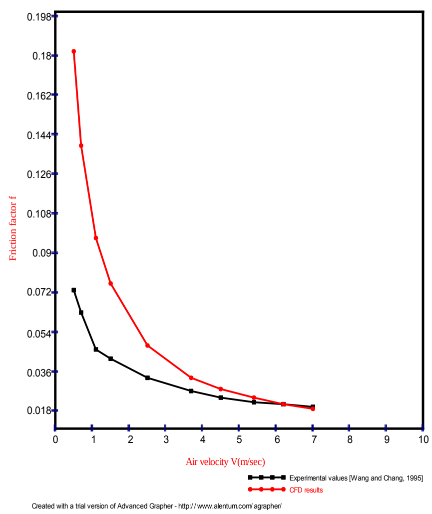


Figure (18) Air velocity with friction factor

.

Conclusion

The most important results can be summarized as follows:

- The heat exchanger which manufactured from the copper best than the heat exchanger that manufactured from the iron.
- The heat transferred from the copper heat exchanger more than the heat transferred from the iron heat exchanger.
- The heat transfer rate is increasing with increasing the air flow.
- The air side heat transfer coefficient increasing with increasing the air velocity.
- The weight of the copper heat exchanger less than the weight of iron heat exchanger
- Good agreement between experimental numerical results.

Nomenclature

A: Area
Abase: Area of fin base
Af Area of fin
Aw Area of wall
Cp Specific heat at constant pressure
Cmax maximum specific heat
Cmin Minimum specific heat
Cpwtr: water specific heat
D: Diameter
Dh: Hydraulic diameter
F: Friction factor
G: Mass velocity
h: Heat transfer coefficient
ho: Out side heat transfer coefficient
hi: Inside heat transfer coefficient
j: j-factor
K: Thermal conductivity
m: Mass flow rate
mwtr: Water mass flow rate
Ntu: Number of transfer units
Nu: Nusselt number
P: Pressure
Pr: Prandtl number
Q: Heat transfer rate
Qavg: Average heat transfer rate
Qmax: Maximum heat transfer rate
Re: Reynolds number
T: Temperature
Th: High temperature
Tl: Low temperature
Tair: Air temperature
Twtr: Water temperature
U: Overall heat transfer coefficient
u: Velocity
V: Air velocity

References:

1-R. K. Shah* and D. R Sekulib, Heat exchangers. *University of Kentucky* (1995)

2- R. K. Rajput., Heat exchangers. (2007)

3- Mills, A. F., Heat Transfer, Irwin, (1992)

4- Willard W. Pulkrabek. Engineering Fundamentals of the Internal Combustion Engine. *University of Wisconsin-· .. Platteville*

5- Karim Nice. HowStuffWorks How Car Cooling Systems Work (2009)

6- LIUEDAHL,J. B., W. M. CARLETON, P. K TURNQUIST, and D. W. SMITH,Tractors and Their Power Units. New Yok: Wiley, (1979)

7_ Cegale**.,** Fundamentals of Heat transfer,2nd edition (2002)

8- Rohsenow, Warren; Hartnett, James P.; Cho, Young **I.** **"**Handbook of Heat Transfer", 3rd Edition, McGraw-Hill Companies, Inc., New York (1998).

9- Baggio, P.; Fornasieri, E. **"**Air-side heat transfer and flow friction: theoretical aspects", in *Recent* *developments in finned tube heat exchangers.* Energy Technology (1994) pp. 91-159

10- Kays, W.M.; London, A.L. **"**Compact Heat Exchangers", Sub-edition 3, Krieger Publishing Company, New York (1998).

11- Wang, Chi-Chuan; Chang, Yu-Juei; Hsieh, Yi-Chung; Lin, Yur-Tsai**.** "Sensible heat and friction characteristics of plate fin-and-tube heat exchangers having plane fins", *International Journal of Refrigeration*, Vol. 19, No. 4 (1996) pp. 223-230

12- Gnielinski, V. "New Equation for heat and mass transfer in turbulent pipe and channel flow", *International Chemical Engineering*, (1976) 359-368.

13- Schmidt, Th. E**.** "Heat transfer calculations for extended surfaces", *Refrigeration Engineering*, April (1949) pp. 351-357.

14- Fornasieri, E.; Mattarolo, L. "Air-side heat transfer and pressure loss in finned tube heat exchangers: state of art", *Proceedings of the European Conference on Finned Tube Heat Exchangers*, Lyon, France, (April 1991).

التحليل العددي والتجريبي لانتقال الحرارة ودراسة الجريان لمبادل حراري في جرار زراعي

م.م. كاظم فاضل ناصر

الخلاصة

تم أجراء دراسة عملية ونظرية لفحص خواص الجريان وانتقال الحرارة خلال المبادل الحراري للساحبة. **محرك الساحبة** من نوع الديزل (**الاشتعال بالضغط**) . **الدراسة** العملية تضمنت استخدام مبادل حراري مصنوع من النحاس بدلا من المبادل الحراري الاصلي للساحبة والمصنوع من الحديد. كلا المبادلان الحراريان يحتويان على أنابيب دائرية المقطع لها نفس القطر ومن صفائح زعنفية . تم قياس درجات الحرارة داخل غرفة التبريد في المحرك في الحالتين وبأوقات وأحمال متساوية . كانت النتيجة للفحص العملي أن أستخدام ألمبادل الحراري المصنوع من النحاس حقق درجات حرارة اقل من أستخدام ألمبادل الحراري المصنوع من الحديد يعني أن كمية الحرارة المطروحة باستخدام المبادل الحراري المصنوع من النحاس اكبر من كمية الحرارة المطروحة من استخدام المبادل الحراري المصنوع من الحديد وبذلك يحقق المبادل النحاسي افضلية في الاستخدام لتبريد محرك الساحبة. الدراسة النظرية تضمنت نموذج للمبادل الحراري النحاسي تم تحليل هذا النموذج بطريقة (**سي اف دي**) عند سرع جريان للهواء مختلفة ابتداءا **من (٠.٥ م/ثا) الى (٧ م/ثا)** للجريان الطباقي للهواء. تهدف الدراسة الى تحسين عملية انتقال الحرارة عند مرور مائع التبريد في خلال المبادل الحراري. النتائج النظرية أظهرت ان معدل انتقال الحرارة يزداد بازدياد سرعة الهواء المار خلال المبادل الحراري.كما اظهرت النتائج تطابق بين الحالة العملية والحالة النظرية .