Optimizing Thermal Efficiency in HVAC Systems

Elena Vasquez, Rohan Kumar
National Institute of Energy Research, New York, USA

Abstract**—**In this paper comprehensive studies have been carried out for the design optimization of a waste heat recovery system for effectively utilizing the domestic air conditioner heat energy for producing hot water. Numerical studies have been carried for the geometry optimization of a waste heat recovery system for domestic air conditioners. Numerical computations have been carried out using a validated 2d pressure based, unsteady, 2nd-order implicit, SST k-ω turbulence model. In the numerical study, a fully implicit finite volume scheme of the compressible, Reynolds-Averaged, NavierStokes equations is employed. At identical inflow and boundary conditions various geometries were tried and effort has been taken for proposing the best design criteria. Several combinations of pipe line shapes viz., straight and spiral with different number of coils for the radiator have been attempted and accordingly the design criteria has been proposed for the waste heat recovery system design. We have concluded that, within the given envelope, the geometry optimization is a meaningful objective for getting better performance of waste heat recovery system for air conditioners

Keywords**—**Air-conditioning system, Energy conversion system, Hot water production from waste heat, Waste heat recovery system.

I. Introduction

THE lucrative design of any waste heat recovery system is a challenging task and it has been an active research topic for the last few decades. Recovering waste heat using compact heat exchangers is a straightforward and easy way to boost the energy efficiency of a plant. The literature review reveals that the International Energy Agency (IEA) predicts world energy demand to increase by 45% over the next two decades. They also predict the supply of fossil fuels will not be able to meet this demand, even when taking new, undiscovered fields into account. There are many alternative ways to battle the energy challenge [1]-[10]. An effective way to increase energy efficiency is to recover waste heat.

Heat pipe heat exchanger (HPHX) is an excellent device used for heat recovery in air conditioning systems. Among the many outstanding advantages of using the heat pipe as a heat transmission device are constructional simplicity, exceptional flexibility, accessibility to control and ability to transport heat at high rate over considerable distance with extremely small temperature drop [3]. A comprehensive review on the application of horizontal heat pipe heat exchangers for air conditioning in tropical climates was conducted by [4]. Di Liu et al. [5] designed and tested a looped separate heat pipe as a waste heat recovery facility for the air-conditioning exhaust system and performed a parametric analysis to investigate the effects of the length of the evaporator, vapor temperature and power throughput on the critical values of the upper and lower boundaries. Waste heat found in the exhaust gas of various processes or even from the exhaust stream of a conditioning unit can be used to preheat the incoming gas. This is one of the basic methods for recovery of waste heat. Many steel making plants use this process as an economic method to increase the production of the plant with lower fuel demand. There are many different commercial recovery units for the transferring of energy from hot medium space to lower one. A waste heat recovery unit (WHRU) is an energy recovery heat exchanger that recovers heat from hot streams with potential high energy content, such as hot flue gases from a diesel generator or steam from cooling towers or even waste water from different cooling processes such as in steel cooling.

In air conditioning system, HPHX can be used for exchange of heat between fresh outdoor air and conditioned return air (heat recovery application) and also enhancing the dehumidification capability of cooling coil as well as reheat savings. Nevertheless, the second application of dehumidification with reheat is beneficial in situation wherein reheating is necessary for maintaining required indoor air conditions.

Yi Xiaowen and W.L. Lee [6] conducted an experimental study on the performance of a domestic water-cooled airconditioner (WAC) using tube-in-tube helical heat exchanger for preheating of domestic hot water. The experimental results showed that the cooling coefficient of performance (COP) of the WAC improves with the inclusion of the heat recovery option by a minimum of 12.3%. This can be further improved to 20.6% by an increase in tap water flow rate. A mathematical model relating the overall heat transfer coefficient to the outer pipe diameter was established which provides a convenient way of optimizing the design of the helical heat exchanger. Air conditioning units are designed to remove heat from interior spaces and reject it to the ambient (outside) air. Heat rejection may occur directly to the air, as in the case of most conventional air source units, or to water circulating from a cooling tower. The circulating water eventually rejects the heat to the ambient air in the cooling tower. While this heat is of a "low grade variety," it still represents wasted energy. From an energy conservation standpoint, it would be desirable to reclaim this heat in a usable form. The best and most obvious form of heat recovery is for heating water. This paper focuses on the geometry optimization of waste heat recovery system from air conditioners for domestic applications.

II. Experimental Method

As a first step, for the design of a waste heat energy recovery system, an experiment has been carried out for recovering waste heat rejected by a typical air conditioning system for heating the water for domestic applications for getting bench mark solution for numerical model validation. Fig. 1 shows the experimental set up [10]. During the test the normal water is filled inside the radiator and the duct is used as a connector between condenser and radiator then the hot air passes into the radiator. We observed that water at an inflow temperature 26ºC and flow rate of 26 Kg/s raises its temperature to 42ºC after 10 minutes while passing it through a spiral type radiator having average skin temperature 50ºC. This could be the surrogate for water heater and it fulfills almost all the domestic applications of hot water. The knowledge gained from this experiment lead to say that more parametric studies are required for the geometry optimization of a waste heat energy recovery system. Therefore, instead of proceeding with empirical design technique in the second phase we have carried out numerical studies, which are discussed in the subsequent session.

Fig. 1 Experimental set up for the waste heat energy recovery system

III. Numerical Method of Solution

Numerical computations have been carried out using a validated 2d pressure based, unsteady, 2nd-order implicit, SST k-ω turbulence model. In the numerical study, a fully implicit finite volume scheme of the compressible, ReynoldsAveraged, Navier-Stokes equations is employed.

The SST *k*–ω model has a similar form to the standard *k*–ω model:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left(\Gamma_k\frac{\partial k}{\partial x_j}\right)+\tilde{G}_k-Y_k+S_k \quad (1)$$

$$\frac{\partial(\rho \omega)}{\partial t}+\frac{\partial(\rho \omega u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left(\Gamma_\omega\frac{\partial \omega}{\partial x_j}\right)+G_\omega-Y_\omega+S_\omega \quad (2)$$

In these equations, $\tilde{G}_k$ represents the generation of turbulence kinetic energy due to mean velocity gradients, calculated from G_k and is given by,

$$\tilde{G}_k=\min(G_k,10\rho\beta^{*}k\omega) \quad (3)$$

G_ω represents the generation of ω, calculated as described for the standard k-ω and is given by,

$$G_\omega=\alpha\frac{\omega}{k}G_k \quad (4)$$

Γ_k and Γ_ω represent the effective diffusivity of k and ω, respectively, which are calculated as described below. Y_k and Y_ω represent the dissipation of k and ω due to turbulence. D_ω represents the cross-diffusion term, calculated as described below. S_k and S_ω are user-defined source terms.

The model uses a control volume based technique to convert the governing equations to algebraic equations. The grid system in the computational domain is selected after a detailed grid refinement exercises. The grids are clustered near the solid walls using suitable stretching functions. The geometric variables and material properties are known *a priori*. Initial wall temperature, inlet temperature and constant mass flow rate are specified. Water is selected as the working fluid. At the solid walls a no slip boundary condition is imposed. In all the cases CFL was selected as 200. Fig. 2 shows the physical model of a radiator having seven aluminum coils with a total length of 4.64 m having 1 cm diameter.

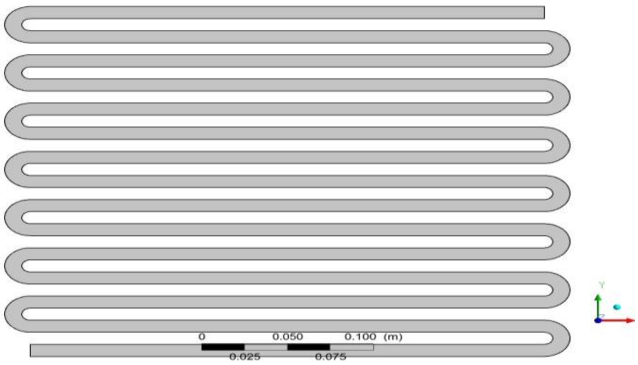


Fig. 2 Physical model of seven coiled radiator pipe

IV. Results and Discussion

Although the theory of fluid flow is reasonably well understood, theoretical solutions are obtained only for a few simple cases such as fully developed laminar flow in a circular pipe. Therefore, we must rely on experimental results and empirical relations for most fluid flow problems rather than closed-form analytical solutions. However, of late numerical results are giving better solutions for meeting the industrial requirements. Note that the fluid velocity in a pipe changes from zero at the surface because of the no-slip condition to a maximum at the pipe center. In fluid flow, it is convenient to work with an average velocity V_{avg}, which remains constant in incompressible flow when the cross-sectional area of the pipe is constant. The average velocity in heating and cooling applications may change somewhat because of changes in density with temperature. But, in practice, we evaluate the fluid properties at some average temperature and treat them as constants. Note that the timeaveraged differential equation for energy in a given flow field is linear in the temperature if we consider the fluid properties of temperature. Thus, the concept of a heat transfer coefficient arises such that the heat transfer rate from a wall is given by:

$$q^{\cdot} = h\,(T_w - T_r) \quad (5)$$

where, the heat transfer coefficient, *h*, is only a function of the flow field. T_w is the wall temperature and T_r, the recovery or adiabatic wall temperature. The above is also true of the boundary layer energy equation, which is a particular case of the general energy equation. When fluids encounter solid boundaries, the fluid in contact with the wall is at rest and viscous effects thus retard a layer in the vicinity of the wall.

It is known that the transition from laminar to turbulent flow depends on the geometry, surface roughness, flow velocity, surface temperature, and type of fluid, among other things. After exhaustive experiments it has been found out by Osborne Reynolds (1880) that the flow regime depends mainly on the ratio of inertial forces to viscous forces in the fluid, which is popularly known as Reynolds number. It certainly is desirable to have precise values of Reynolds numbers for laminar, transitional, and turbulent flows, but this is not the case in practice. It turns out that the transition from laminar to turbulent flow also depends on the degree of disturbance of the flow by surface roughness, pipe vibrations, and fluctuations in the flow. For large Reynolds numbers based on distance from the leading edge, the viscous layers are thin compared to this length. Therefore selection of radiator geometry is important in the light of its flow field features.

At large Reynolds numbers, the inertial forces, which are proportional to the fluid density and the square of the fluid velocity, are large relative to the viscous forces, and thus the viscous forces cannot prevent the random and rapid fluctuations of the fluid. At small or moderate Reynolds numbers, however, the viscous forces are large enough to suppress these fluctuations and to keep the fluid *in line*. Thus the flow is turbulent in the first case and laminar in the second. The Reynolds number at which the flow becomes turbulent is called the critical Reynolds number, Re_{cr}. The value of the critical Reynolds number is different for different geometries and flow conditions. This merits further investigation on geometry optimization of pipe lines for waste heat recovery system design. Therefore in this paper an attempt has been made for geometry optimization waste heat recovery unit.

Note that when the wall is at a different temperature to the fluid, there is a small region where the temperature varies. These regions are the *velocity* and *thermal boundary layers*, which are important for predicting the heat transfer between the wall and the fluid. Many earlier researchers carried out separate analyses of these regions from the bulk fluid flow. In these cases the pressure variation normal to the wall is neglected and the pressure is given by the free stream features. Note that these types of analyses will not give the accurate solutions for many industrial problems because the boundary layer equations for steady incompressible laminar flow in two dimensions are approximated.

Most flows encountered in engineering practice are turbulent, and thus it is important to understand how turbulence affects wall shear stress. However, turbulent flow is a complex mechanism dominated by fluctuations, and despite tremendous amounts of work done in this area by researchers, the theory of turbulent flow remains largely undeveloped. Therefore, we must rely on experiments and the empirical or semi-empirical correlations developed for various situations. Turbulent flow is characterized by random and a rapid fluctuation of swirling regions of fluid, called eddies, throughout the flow. These fluctuations provide an additional mechanism for momentum and energy transfer. In laminar flow, fluid particles flow in an orderly manner along path lines, and momentum and energy are transferred across streamlines by molecular diffusion. In turbulent flow, the swirling eddies transport mass, momentum, and energy to other regions of flow much more rapidly than molecular diffusion, greatly enhancing mass, momentum, and heat transfer. As a result, turbulent flow is associated with much higher values of friction, heat transfer, and mass transfer coefficients. Even when the average flow is steady, the eddy motion in turbulent flow causes significant fluctuations in the values of velocity, temperature, pressure, and even density (in compressible flow). Also, the friction between the fluid particles in a pipe does cause a slight rise in fluid temperature as a result of the mechanical energy being converted to sensible thermal energy. But this temperature rise due to frictional heating is usually too small to warrant any consideration in calculations and thus is disregarded in most of the water pipe flow problems. The primary consequence of friction in fluid flow is pressure drop, and thus any significant temperature change in the fluid is due to heat transfer. Therefore, a suitable numerical model is inevitable solving these types of problems.

Turbulent flow is characterized by the energy transport by turbulent eddies which is more intensive than the molecular transport in laminar flows. Heat transfer coefficient and the Nusselt number are therefore greater in turbulent flows. Note that Nusselt number doesn't depend upon the length of pipe in turbulent flows significantly (unlike the case of laminar flows characterized by rapid decrease of Nusselt number with the length L). Also earlier studies reveal that Nusselt number doesn't depend upon the shape of cross section in the turbulent flow regime (it is possible to use the same correlations for elliptical, rectangular cross sections using the concept of equivalent diameter – this cannot be done in laminar flows). However, in the advent of CFD coupled with various unsteady heat transfer models, it is possible to predict the transient surface heat flux and the fluid flow temperature in highly turbulent conditions in any types of geometry and we have attempted it accordingly in this study.

In this paper numerical computations have been carried out using a validated 2d pressure based, unsteady SST k-ω turbulence model. In the numerical study, a fully implicit finite volume scheme of the compressible, ReynoldsAveraged, Navier-Stokes equations is employed. The code has been validated with the in-house experimental results [10] and found excellent agreement. Numerical studies have been carried out with same inflow and boundary conditions but with different geometries of the radiator, viz., straight pipe and spiral with different total length. Fig. 3

shows the grid system in the computational domain corresponding to Fig. 2.

In the first phase of this study numerical simulations have been carried out for predicting the near wall and axial temperature of water with the experimental inflow conditions. Fig. 4 shows the comparison of the axial temperature variations and the near wall temperature variations of water in a straight pipe having the same length as that of a radiator shown in Fig. 2.

Fig. 3 Grid system in the computation domain (Uniform quad / tri mesh with number of elements 49788, number of nodes 54701 for a seven coiled radiator pipe)

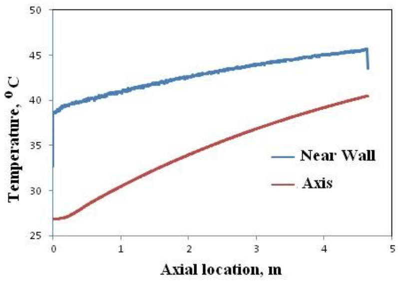


Fig. 4 Comparing the axial temperature and the near wall temperature variations of water in a straight pipe having the same length as that of a radiator shown in Fig. 2

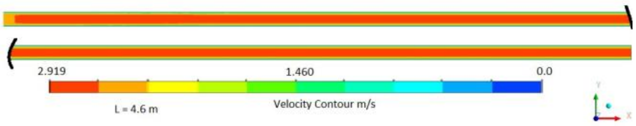


Fig. 5 Velocity contour for single radiator pipe at time 10 mins

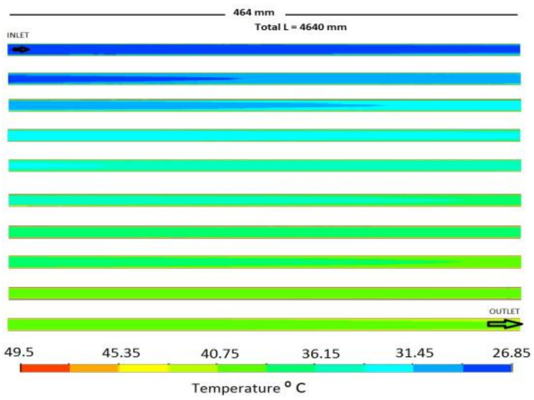


Fig. 6 Temperature contour for single pipe flow case shown with continuous sections (L= 4.64 m, time = 10 mins)

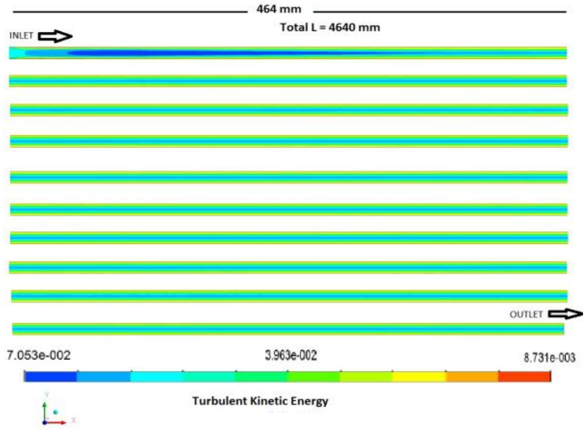


Fig. 7 Turbulence kinetic energy for single pipe flow case

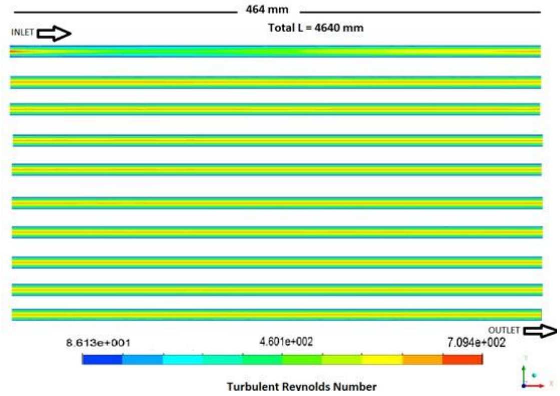


Fig. 8 Turbulent Reynolds number for single pipe flow case

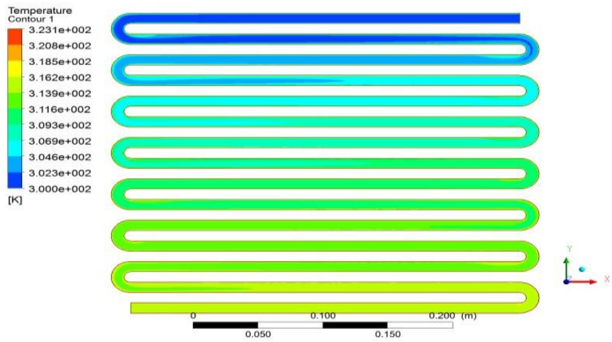


Fig. 9 Temperature contour for seven Coiled Radiator pipe at a time of 10 mins; Total Length = 4.64 m

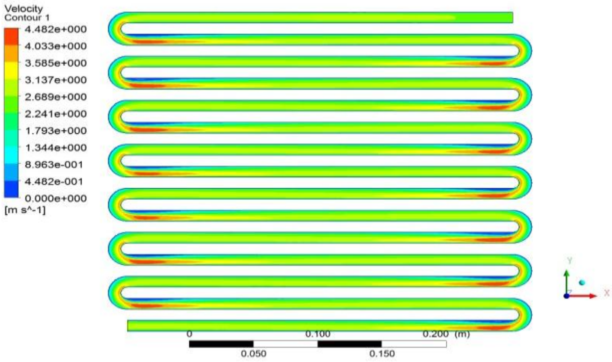


Fig. 10 Velocity contour for seven coiled radiator pipe at time 10 mins; Total Length = 4.64 m

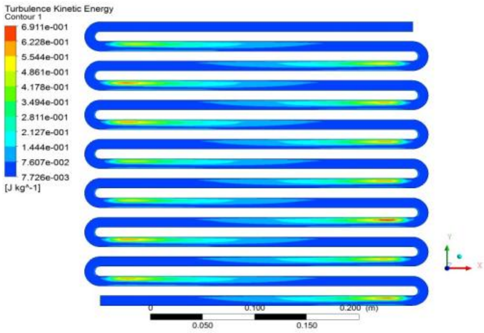


Fig. 11 Turbulence kinetic energy for seven coiled radiator

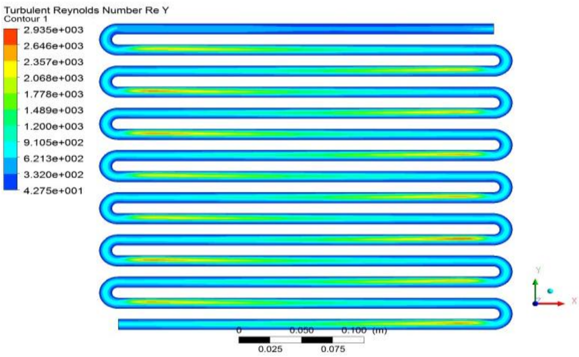


Fig. 12 Turbulent Reynolds number for seven coiled radiator

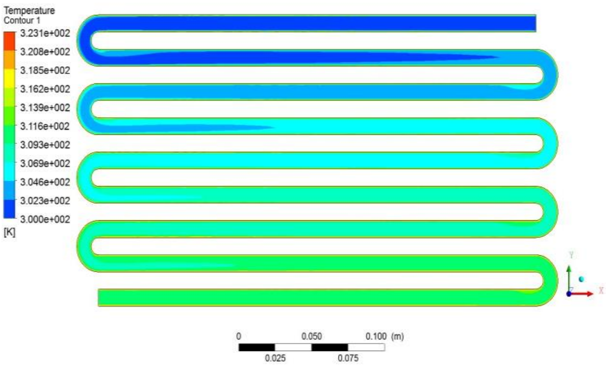


Fig. 13 Temperature contour of four coiled radiator (L = 2.78 m)

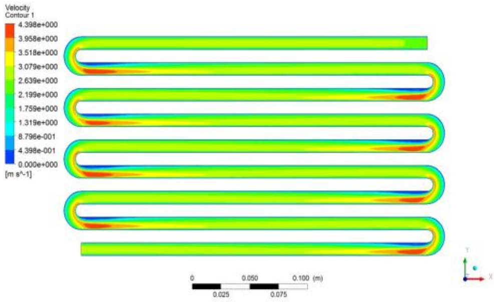


Fig. 14 Velocity contour for four coiled radiator

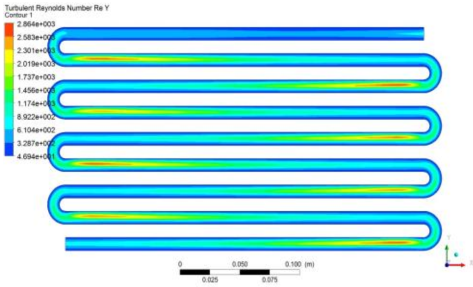


Fig. 15 Turbulent Reynolds number for four coiled radiator

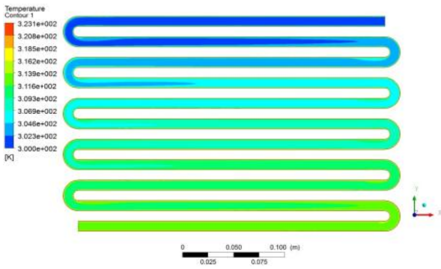


Fig. 16 Temperature contour for five coiled radiator at 10 mins. (Length = 3.4 m)

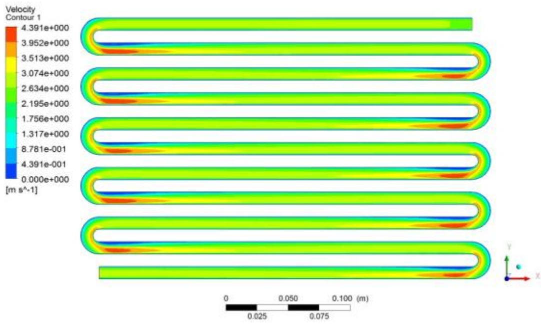


Fig. 17 Velocity contour for five coiled radiator at 10 mins

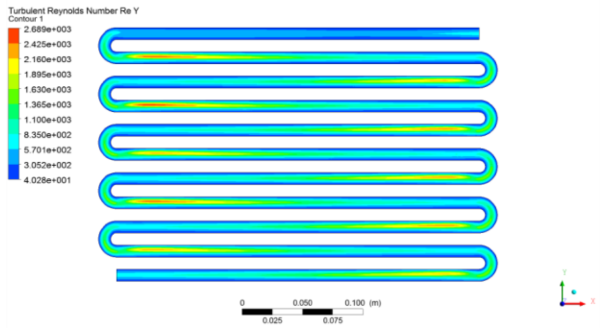


Fig. 18 Turbulent Reynolds number (Re Y) contour for five coiled radiator. at 10 mins. (Length = 3.4 m)

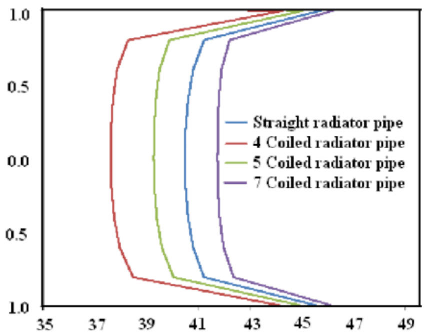
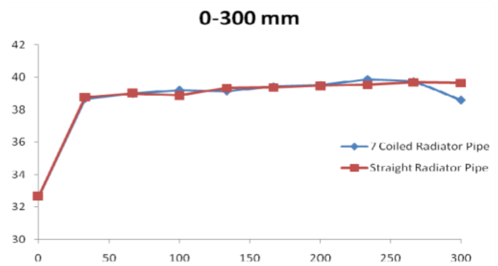
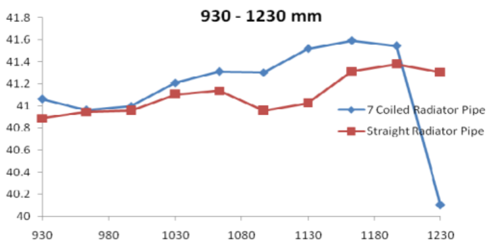


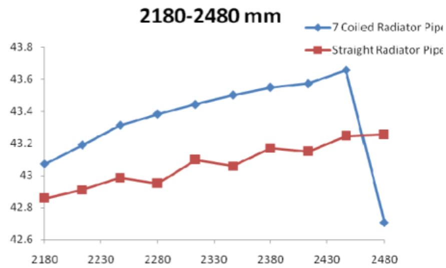
Fig. 19 Comparison of the exit radial temperature of four different radiators



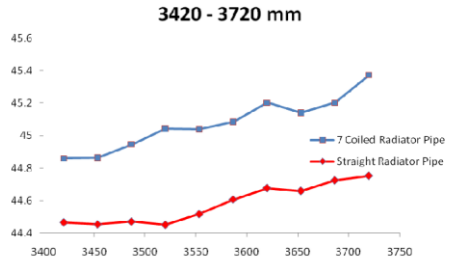
(a) Entry length (0-0.3m)



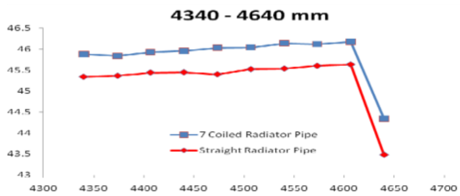
(b) Length (0.9-1.2 m)



(c) Length (2.18-2.48 m)



(d) Length (3.4-3.72 m)



(e) Length (4.3 – 4.64 m)

Fig. 20 (a)-(e) Comparison of fluid temperature at the near wall for two different radiator cases with segregation establishing that a case with coil shape will give more fluid temperature than straight pipe

Figs. 5-8 demonstrate the internal flow features of water viz., velocity contours, temperature contours, turbulence kinetic energy and turbulent Reynolds number for a case with straight pipe radiator having total length 4.64 m at time 10 mins of flowing. Figs. 9-12 show the temperature contour, velocity contour, turbulence kinetic energy, and turbulent Reynolds number for seven coiled radiator. Figs. 13-18 depict the temperature contour, velocity contour and turbulent Reynolds number for four and five coiled radiators respectively. Fig. 19 shows the comparison of the exit radial temperature of four different radiators. It is evident from this figure that a coiled radiator with highest number of coil is giving the highest outlet temperature while comparing to the other three cases. Figs. 20 (a)(e) show the sectional comparison of fluid temperature at the near wall for two different radiator cases, with same length but with different geometry, corroborating that a case with coil shape will give more fluid temperature than straight pipe.

V. Conclusion

In this paper at identical inflow and boundary conditions various geometries were tried and effort has been taken for proposing the best design criteria for radiators. Through the parametric analytical studies we have conjectured that water flow rate, temperature difference between incoming water and radiator skin temperature, material, geometry of the water pipe viz., length, diameter, and wall thickness are having bearing on the lucrative design of a waste heat recovery system for air conditioners. Several combinations of pipe line shapes viz., straight and spiral with different number of coils for the radiator have been attempted and accordingly the design criteria has been proposed for the waste heat recovery system design. We have proved conclusively through our numerical studies that within the given envelope, the geometry optimization is a meaningful objective for getting better performance of waste heat recovery systems for air conditioners. This study is a pointer towards for devising lucrative waste heat energy recovery systems for industrial applications.

REFERENCES

[1] T. S. Jadhav, M. M. Lele, "Theoretical Energy Saving Analysis of Air Conditioning System Using Heat Pipe Heat Exchanger for Indian Climatic Zones", Engineering Science and Technology, an International Journal, May 2015.
[2] D. A. Reay, E & F. N. Span, Heat Recovery Systems, 1979.
[3] S.W. Chi, Heat Pipe Theory and Practice: A Sourcebook Hemisphere Publishing Corporation (1976), p. 1.
[4] Y.H. Yau, M. Ahmadzadehtalatapeh, A Review on the Application of Horizontal Heat Pipe Heat Exchangers in Air Conditioning Systems in the Tropics, Appl. Therm. Eng., 30 (2010), pp. 77–84.
[5] Di Liu, Guang-Fa Tang, Fu-Yun Zhao, Han-Qing Wang, Modeling and Experimental Investigation of Looped Separate Heat Pipe as Waste Heat Recovery Facility, Appl. Therm. Eng., 26 (2006), pp. 2433–2441.
[6] Yi Xiaowen, , W.L. Lee, "The Use of Helical Heat Exchanger for Heat Recovery Domestic Water-Cooled Air-Conditioners," Energy Conversion and Management, Volume 50, Issue 2, February 2009, Pages 240–246.
[7] Z.S.Lu, R.Z. Wang, "Performance Improvement by Mass-Heat Recovery of an Innovative Adsorption Air-Conditioner Driven by 50-80^{o} C Hot Water, Applied Thermal Engineering, Applied Thermal Engineering 55 (2013) 113e120.
[8] Jung-Yang San, Chih-Hsiang Hsu, Shih-Hao Chen, Heat Transfer Characteristics of a Helical Heat Exchanger, Science Direct, Applied Thermal Engineering, Applied Thermal Engineering 39 (2012) 114e120.
[9] M.S. Soylemez "On the Thermo Economical Optimization of Heat Pipe Heat Exchanger HPHE for Waste Heat Recovery" Energy Conversion and Management, Vol. 44, (2003)2509–2517.
[10] Arunkumar, S., Ganesh, R., Panthalaraja Kumaran.K, Design and Fabrication of Waste Energy Recovery System, Mini Project Report, Department of Mechanical Engineering, KCT, Coimbatore-49, India, Nov. 2014.