DESIGN, FABRICATION AND ANALYSIS OF AIR COOLED HEAT EXCHANGER FOR EXHAUST GAS HEAT RECOVERY FROM IC ENGINE

Praveen Math
School of Mechanical Engineering
REVA University, Bangalore, INDIA

Abstract- The increasingly worldwide problem regarding rapid economy development and a relative shortage of energy, the internal combustion engine exhaust waste heat and environmental pollution has been more emphasized heavily recently. Out of the total heat supplied to the engine in the form of fuel, approximately, 30 to 40% is converted into useful mechanical work; the remaining heat is expelled to the environment through exhaust gases and engine cooling systems. If we can reduce this figure by 10% also then it will be a substantial contribution. A set of CFD simulations is performed for a single shell and tube bundle and is compared with the experimental results. The results are found to be sensitive to turbulence model and wall treatment method. It is found that there are regions of low Reynolds number in the core of heat exchanger shell. Thus, SST model provides better results as compared to other models. Significant fraction of total shell side pressure drop is found at inlet and outlet regions.

Keywords: Shell-and-Tube Heat exchanger, Waste heat recovery from engine, SST turbulence model, CFD simulation.

I. INTRODUCTION

Large quantity of hot flue gases is generated from internal combustion engine etc. If same of this waste heat could be recovered, a considerable amount of primary fuel could be saved. It is depends upon mass flow rate of exhaust gas and temperature of exhaust gas. The internal combustion engine energy lost in waste gases cannot be fully recovered. However, much of the heat could be recovered and losses be minimized by adopting certain measures. There are different methods of the exhaust gas heat recovery namely for space heating, refrigeration and power generation. The mass flow rate of exhaust gas is the function of the engine size and speed, hence larger the engine size and higher the speed the exhaust gas heat is larger. So heat recovery system will be beneficial to the large engines comparatively to smaller engines. The heat recovery from exhaust gas and conversion in to mechanical power is possible with the help of Rankine, Stirling and Brayton thermodynamic cycles, vapour absorption cycle. These cycles are proved for low temperature heat conversion in to the useful power. Engine exhaust heat recovery is considered to be one of the most effective means and it has become a research hotspot recently. For example, Doyle and Patel have designed a device for recovering exhaust gas heat based on Rankine cycle on a truck engine. The commissioning experiment of 450 kilometers showed that this device could save fuel consumption by 12.5%. Cummins Company has also done some research on waste heat recovery on truck engines, and the results showed that engine thermal efficiency could improve by 5.4% through exhaust heat recovery. James C. Conklin and James P. Szybist have designed a six-stroke internal combustion engine cycle with water injection for in-cylinder exhaust heat recovery which has the potential to significantly improve the engine efficiency and fuel economy. R. Saidur et al Rankine bottoming cycle technique to maximize energy efficiency, reduce fuel consumption and green house gas emissions. Recovering engine waste heat can be achieved via numerous methods. The heat can either be reused within the same process or transferred to another thermal, electrical, or mechanical process. Hau xuewjun et al has studied the analysis of exhaust gas waste heat recovery and pollution processing for diesel engine. They analyzed total effect of waste heat on pollution or environment. Waste heat can be utilized for some useful works and it is reduces pollution. The diesel engine exhaust gas waste heat recovery rate increase with increasing diesel engine exhaust gas emission rate.

The increasing fuel costs and diminishing petroleum supplies are forcing governments and industries to increase the power efficiency of engines. A cursory look at the internal combustion engine heat balance indicates that the input energy is divided into roughly three equal parts: energy converted to useful work, energy transferred to coolant and energy lost with the exhaust gases. There are several technologies for recovering this energy on a internal combustion engine, whereas the dominating ones are: Waste heat can utilized for heating purpose, power generation purpose, refrigeration purpose, etc.floating tube bundle (where the tube plates are not welded to the outer shell) is available.

Shell and tube heat exchanger design is normally based on correlations, among these; the Kern method and Bell-Delaware method are the most commonly used correlations. Kern method is mostly used for the preliminary design and provides conservative results. Whereas, the Bell-Delaware method is more accurate method and can provide detailed results. It can predict and estimate pressure drop and heat transfer coefficient with better accuracy. The Bell-Delaware method is actually the rating method and it can suggest the weaknesses in the shell side deign but it cannot indicate where these weaknesses are. Thus in order to figure out these problems, flow distribution must be understood. For this reason, several analytical, experimental and numerical studies have been carried out. Most of this research was concentrated on the certain aspects of the shell and tube heat exchanger design. These correlations are developed for baffled shell and tube heat exchangers generally. Our studies aims at studying simple un-baffled heat exchanger, which is more similar to the double pipe heat exchangers. Almost no studies is found for an un-baffled shell and tube heat exchanger. Thus general correlations of heat transfer and pressure drop for straight pipes can be useful to get an idea of the design. Generally there has been lot of work done on heat transfer and pressure drop in heat exchangers. Pressure drop in a heat exchanger can be divided in three parts. Mainly it occurs due to fanning friction along the pipe. In addition to this it also occurs due to geometrical changes in the flow i.e. contraction and expansion at inlet and outlet of heat exchanger. Handbook of hydraulic resistance provides the correlations for the pressure losses in these three regions separately by introducing the pressure loss coefficients.

II MATHEMATICAL BACKGROUND AND CFD

2.1 Flow Calculation

The flow is governed by the continuity equation, the energy equation and Navier-Stokes momentum equations. Transport of mass, energy and momentum occur through convective flow and diffusion of molecules and turbulent eddies. All equations are set up over a control volume Where i; j; k = 1; 2; 3 correspond to the three dimensions.

Continuity Equation

The continuity equation describes the conservation of mass and is written as in equation.

$$\frac{\partial \rho}{\partial t}+\frac{\partial \rho U_1}{\partial x_1}+\frac{\partial \rho U_2}{\partial x_2}+\frac{\partial \rho U_3}{\partial x_3}=0$$

or 2.1

$$\frac{\partial \rho}{\partial t}+\frac{\partial \rho U_i}{\partial x_i}=0, i=1,2,3$$

Equation 2.1 defines the rate of increase of mass in a control volume as equal to the amount through its faces. Whereas, for constant density continuity equation is reduced to

$$\frac{\partial U_i}{\partial x_i}=0, i=1,2,3$$

Momentum Equations (Navier-Stokes Equations)

The momentum balance, also known as the Navier-Stokes equations, follows Newton’s second law: The change in momentum in all directions equals the sum of forces acting in those directions. There are two different kinds of forces acting on a finite volume element, surface forces and body forces. Surface forces include pressure and viscous forces and body forces include gravity, centrifugal and electro-magnetic forces.

The momentum equation in tensor notation for a Newtonian fluid can be written as in equation 2.2

$$\frac{\partial U_i}{\partial t}+U_j\frac{\partial U_i}{\partial x_j}=-\frac{1}{\rho}\frac{\partial P}{\partial x_i}+\nu\frac{\partial}{\partial x_j}(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i})+g_i \qquad 2.2$$

In addition to gravity, there can be further external sources that may effect the acceleration of fluid e.g. electrical and magnetic fields. Strictly it is the momentum equations that form the Navier-Stokes equations but sometimes the continuity and momentum equations together are called the Navier-Stokes equations. The Navier-Stokes equations are limited to macroscopic conditions. The continuity equation is difficult to solve numerically. In CFD programs, the continuity equation is often combined with momentum equation to form Poisson equation 2.3. For constant density and viscosity the new equation can be written as below.

$$\frac{\partial}{\partial x_i}(\frac{\partial P}{\partial x_i})=-\frac{\partial}{\partial x_i}(\frac{\partial(\rho U_i U_j)}{\partial x_j}) \qquad 2.3$$

Energy Equation

Energy is present in many forms in flow i.e. as kinetic energy due to the mass and velocity of the fluid, as thermal energy, and as chemically bounded energy. Thus the total energy can be defined as the sum of all these energies.

$$h = h_m + h_T + h_C + \Phi \tag{2.4}$$

$$\begin{array}{ll} h_m = \frac{1}{2}\rho U_i U_i & \text{Kinetic energy} \\ h_T = \sum_n m_n \int_{T_{ref}}^{T} C_{p,n} dT & \text{Thermal energy} \\ h_C = \sum_n m_n h_n & \text{Chemical energy} \\ \Phi = g_i x_i & \text{Potential energy} \end{array}$$

In the above equations mn and Cpn are the mass fraction and specific heat for species n. The transport equation for total energy can be written by the help of above equations. The coupling between energy equations and momentum equations is very weak for incompressible flows, thus equations for kinetic and thermal energies can be written separately. The chemical energy is not included because there was no species transport involved in this project.

The transport equation for kinetic energy can be written as under

$$\frac{\partial(h_m)}{\partial t} = -U_j \frac{\partial(h_m)}{\partial x_j} + P\frac{\partial U_i}{\partial x_i} - \frac{\partial(PU_i)}{\partial x_i} - \frac{\partial}{\partial x_j}(\tau_{ij}U_i) - \tau_{ij}\frac{\partial U_i}{\partial x_j + \rho g U_i} \tag{2.5}$$

The last term in the equation 2.5 is the work done by the gravity force. Similarly, a balance for heat can be formulated generally by simply adding the source terms from the kinetic energy equation.

$$\frac{\partial(\rho C_p T)}{\partial t} = -U_j \frac{\partial(\rho C_p T)}{\partial x_j} + k_{eff}\frac{\partial^2 T}{\partial x_j x_j} - P\frac{\partial U_j}{\partial x_j} + \tau_{kj}\frac{\partial U_k}{\partial x_j} \tag{2.6}$$

The term on left side of the equation is accumulation term. The first on the right is convection term, second is the conduction, third expansion and last is dissipation term. Here the terms in the equation for transformation between thermal and kinetic energy, i.e. expansion and dissipation occur as source terms.

Turbulence Modeling

Definition of Turbulence

Turbulent flows have some characteristic properties which distinct them from laminar flows. The motions of the fluid in a turbulent flow are irregular and chaotic due to random movements by the fluid. The flow has a wide range of length, velocity and time scales. Turbulence is a three dimensional diffusive transport of mass, momentum and energy through the turbulent eddies that result in faster mixing rates. Energy has to be constantly supplied or the turbulent eddies will decay and the flow will become laminar, the kinetic energy becomes internal energy. Turbulence arises due to the instability in the flow. This happens when the viscous dampening of the velocity fluctuations is slower than the convective transport, i.e. the fluid element can rotate before it comes in contact with wall that stops the rotation. For high Reynolds numbers the velocity fluctuations cannot be dampened by the viscous forces and the flow becomes turbulent. Turbulent flows contain a wide range of length, velocity and time scales and solving all of them makes the costs of simulations large. Therefore, several turbulence models have been developed with different degrees of resolution. All turbulence models have made approximations simplifying the Navier-Stokes equations. There are several turbulence models available in CFD-soft wares including the Large Eddy Simulation (LES) and Reynolds Average Navier- Stokes (RANS). There are several RANS models available depending on the characteristic of flow, e.g., Standard ke" model, kw" RNG model, Realizable ke", kw! and RSM (Reynolds Stress Model) models.

Turbulence Model

The RANS models assume that the variables can be divided into a mean and fluctuating part. The pressure and velocity are then expressed as

$$U_i = \langle U_i \rangle + u_i$$

$$P_i = \langle P_i \rangle + p_i$$

where the average velocity is defined as

$$\langle U_i \rangle = \frac{1}{2T} \int_{-T}^{T} U_i dt$$

The decomposition of velocity and pressure inserted into Navier-Stokes equations give

$$\frac{\partial \langle U_i \rangle}{\partial t} + \langle U_i \rangle \frac{\partial \langle U_i \rangle}{\partial x_j} = -\frac{1}{\rho} \frac{\partial}{\partial x_j} \left\{ \langle P \rangle \delta_{ij} + \mu \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \rho \langle u_i u_j \rangle \right\} \quad 2.7$$

The last term is called the Reynolds stresses and describes the velocity fluctuations caused by turbulence. This term needs to be modeled to close this equation. The Reynolds averaged stress models use the Boussinesq approximation which is based on the assumption that the Reynolds stresses are proportional to mean velocity gradient. The Boussinesq approximation assumes that the eddies behave like the molecules, that the turbulence is isotropic and that the stress and strain are in local equilibrium. These assumptions cannot be made for certain flows, e.g., the highly swirling flows having a large degree of anisotropic turbulence and then inaccurate results are obtained. The Boussinesq approximation allows the Reynolds stresses to be modeled using a turbulent viscosity which is analogous to the molecular viscosity. Thus above equation becomes,

$$\frac{\partial \langle U_i \rangle}{\partial t} + \langle U_i \rangle \frac{\partial \langle U_i \rangle}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \langle P \rangle}{\partial x_i} - \frac{2}{3} \frac{\partial k}{\partial x_i} + \frac{\partial}{\partial x_j} \left[(\nu + \nu_T) \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] \quad 2.8$$

The use of RANS models requires that two additional transport equations, for the turbulence kinetic energy, k, and the turbulence dissipation rate, or the specific dissipation rate, are solved.

Two-Equations Models

Different turbulence models can be classified on the basis of number of extra equations used to close the set of equations. There are zero, one and two equations models which are commonly employed for turbulence modeling. Zero equation model makes a simple assumption of constant viscosity (Prandtl's mixing length model). Whereas one equation model assumes that viscosity is related to history effects of turbulence by relating to time average kinetic energy. Similarly, two equation model uses two equations to close the set of equations. These two equations can model turbulent velocity or turbulent length scales. There are many variables which can be modeled for example vorticity scale,frequency scale,time scale and dissipation rate. Among these variables, dissipation rate " is the most commonly used variable. This model is named with respect to the variables being modeled. For example k e "model, as it models k (Turbulent kinetic energy) and k e "(Turbulent energy dissipation rate). Another, important turbulence model is k w model. It models k (Turbulent kinetic energy) and w (Specific dissipation rate). These models have become now common in industrial use. These provide significant amount of reliability as they use two variables to close the set of equations.

K e Models

The first transported variable is turbulent kinetic energy, k. The second transported variable in this case is the turbulent dissipation. Their respective modeled transport equations are as under,

For k,

$$\frac{\partial k}{\partial t} + \langle U_j \rangle \frac{\partial k}{\partial x_j} = \nu_T \left[\left(\frac{\partial \langle U_i \rangle}{\partial x_j} + \frac{\partial \langle U_j \rangle}{\partial x_i} \right) \frac{\partial \langle U_i \rangle}{\partial x_j} \right] - \varepsilon + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]$$

And for ε

$$\underset{1}{\frac{\partial \varepsilon}{\partial t}} \; \underset{2}{+ \langle U_j \rangle \frac{\partial \varepsilon}{\partial x_j}} = \underset{3}{C_{\varepsilon 1} \nu_T \frac{\varepsilon}{k} \left[\left(\frac{\partial \langle U_i \rangle}{\partial x_j} + \frac{\partial \langle U_j \rangle}{\partial x_i} \right) \frac{\partial \langle U_i \rangle}{\partial x_j} \right]} \; \underset{4}{+ C_{\varepsilon 2} \frac{\varepsilon^2}{k}} \; \underset{5}{+ \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_T}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right]}$$

The physical interpretation of the e equation is,

1. Accumulation of e
2. Convection of e by the mean velocity
3. Production of e
4. Dissipation of e
5. Diffusion of e

The standard k e model does not always give good results. There are some flows which cannot be predicted accurately, such as streamline curvature, swirling flows and axis-symmetrical jets. The inaccuracies stem from underlying Boussinesq hypothesis which imposes isotropy and the way it models the dissipation equation. This model was derived and tuned for high Reynolds numbers. This implies that it is suited for flows where the turbulence is nearly isotropic and to flows where energy cascade proceeds in local equilibrium with respect to generation. Furthermore, the model parameters in k e model are a compromise to give a best performance for wide range of different flows. Due to these weaknesses in k e model, several variants are derived for overcoming some of its short comings. Realizable k e model is one of them and is described here.

Realizable k e Model

The realizable k e model differs from the standard k e model in that it features a reliability constraint on the predicted stress tensor, thereby giving the name of realizable k e model. The difference comes from correction of the k-equation where the normal stress can become negative in the standard k e model for flows with large strain rate. This can be seen in the normal components of the Reynold stress tensor.

$$\langle u_i u_i \rangle = \sum_i \langle u_i^2 \rangle = \frac{2}{3}k - 2\nu_T \frac{\partial \langle U_i \rangle}{\partial x_j} \qquad 2.9$$

Note that (ui ui), must be larger than zero by definition since it is a sum of squares. However, equation 2.9 implies that if strain is sufficiently large, normal stresses become negative. The realizable k e model uses a variable Cm so that this will never occur. In fact, Cm is no longer constant, instead it is a function of the local state of flow to ensure that the normal stresses are positive under all flow conditions, i.e. to ensure that normal stresses are positive under all flow conditions.

III MODELLING AND EXPERIMENTAL SETUP

Geometry

Heat exchanger geometry is built in the ANSYS workbench design module. Geometry is simplified by considering the plane symmetry and is cut half vertically. It is a parallel heat exchanger, and the tube side is built with 11 separate inlets comprising of 10 complete tubes and with. The shell outlet length is also increased to facilitate the modeling program to avoid the reverse flow condition.

Experiment Calculations for set up :

Flow rates of both streams:

A 150 cc 2 stroke petrol engine is taken for the analysis. To calculate the mass flow rate of the exhaust, following relation is considered.

Assuming 3500 rpm (a variation of 1000rpm to 3500rpm can be achieved, designing heat exchanger for maximum speed).

Mass flow rate = Volume * Density * speed

$$= 150 \text{ m3} * 1.3 \text{ kg/m3} * \frac{3500}{106} = 0.6875 \text{ kg/s.}$$

Approx. 0.7 kg/s of exhaust flow is considered for the design.

Cold air mass flow rate is also approximated to 0.7 kg/s. Since the cold air is heated and then used in the engine, same flow rate is considered for the design.

Inlet and outlet temperatures of both streams.

Exhaust gas temperature = Inlet : 250 C

Outlet is contended to 30 C

Inlet air temperature = Inlet : 25 C

Outlet: 120 C (Inlet air temperature is contended to 120 C, due to knocking problem with higher inlet temperature).

Operating pressure of both streams: Atmospheric pressures are considered for both the fluids, and density of fluid does not vary with increase in pressure.

Allowable pressure drop for both streams. : 0.1 kg/cm2 for gases.

Fig.1: Experimental set up of heat exchanger

The dimensions of the geometry are also given in the Table.1 below

Sl. No.	Parameter	Dimensions
1	Tube diameter	18 mm
2	Tube length	500 mm
3	Total number of tubes	10
4	Shell Width	200 mm
5	Shell Length	500 mm

The exhaust of 2-stroke engine is connected to shell side inlet and the fresh air is connected to tube side of the heat exchanger. Temperature points are monitored at inlet and outlets of both the streams.

Computational set up:

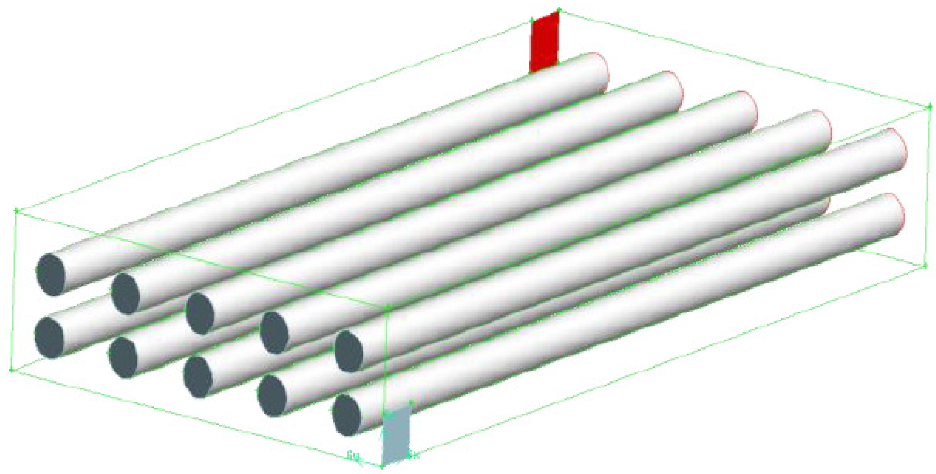

Fig.2: model of heat exchanger

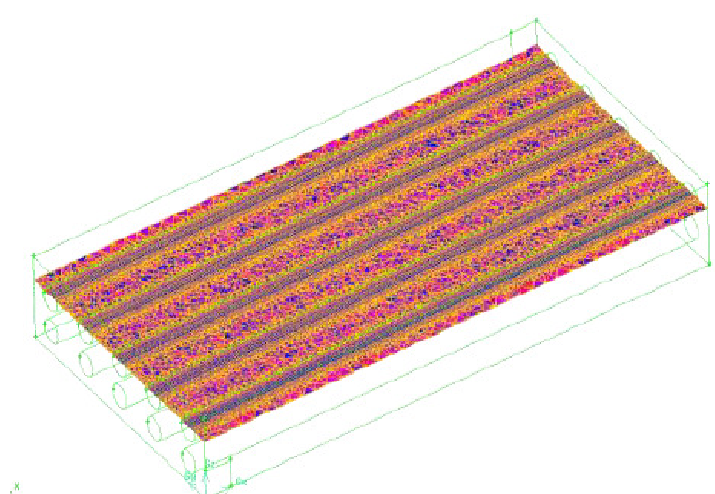

Fig 3: model of heat exchanger

RESULTS AND DISCUSSIONS

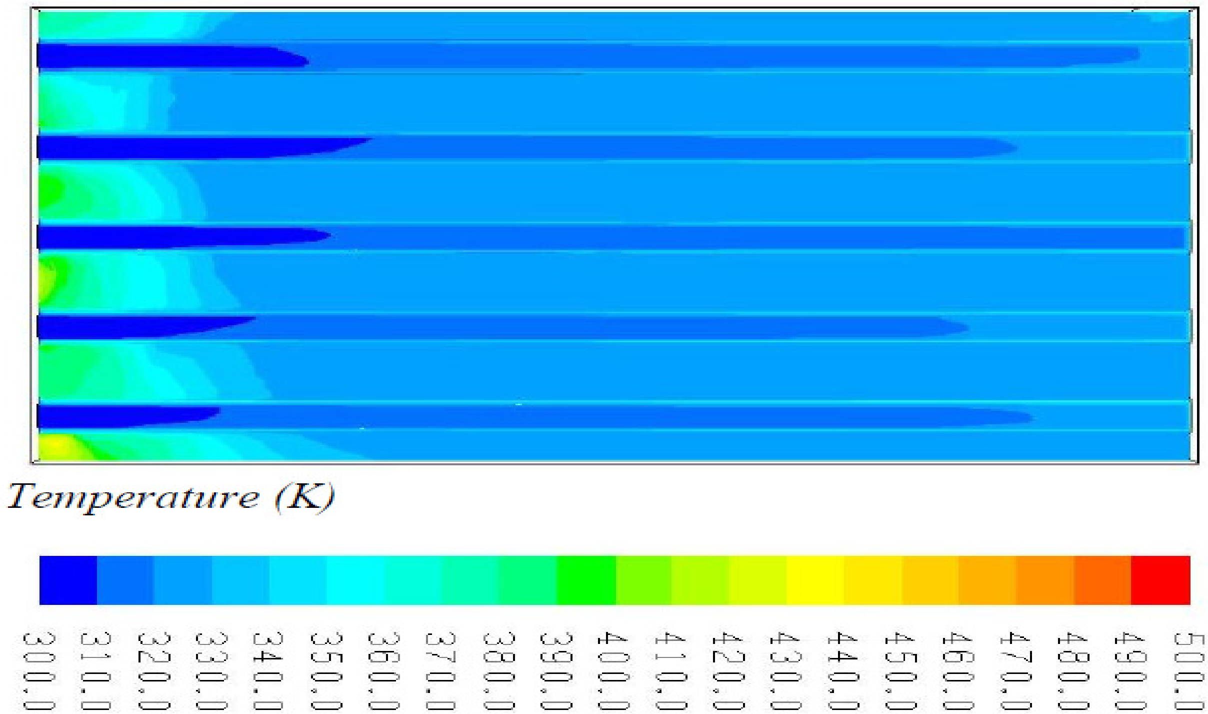

Fig.4 Temperature distribution

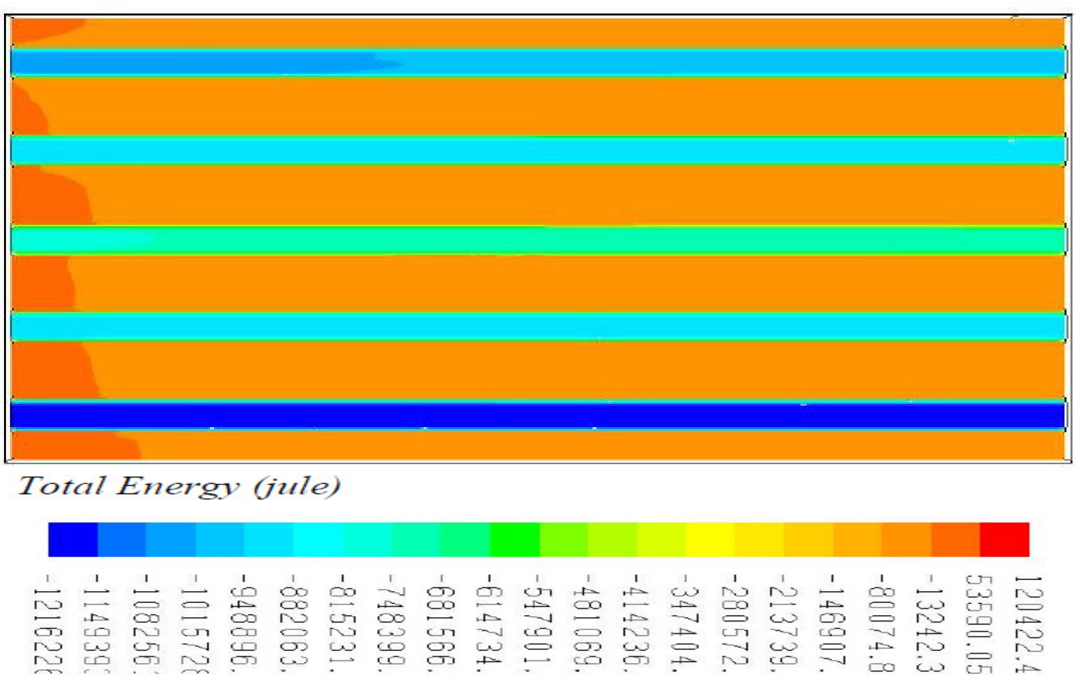

Fig. 5: Total Energy distribution

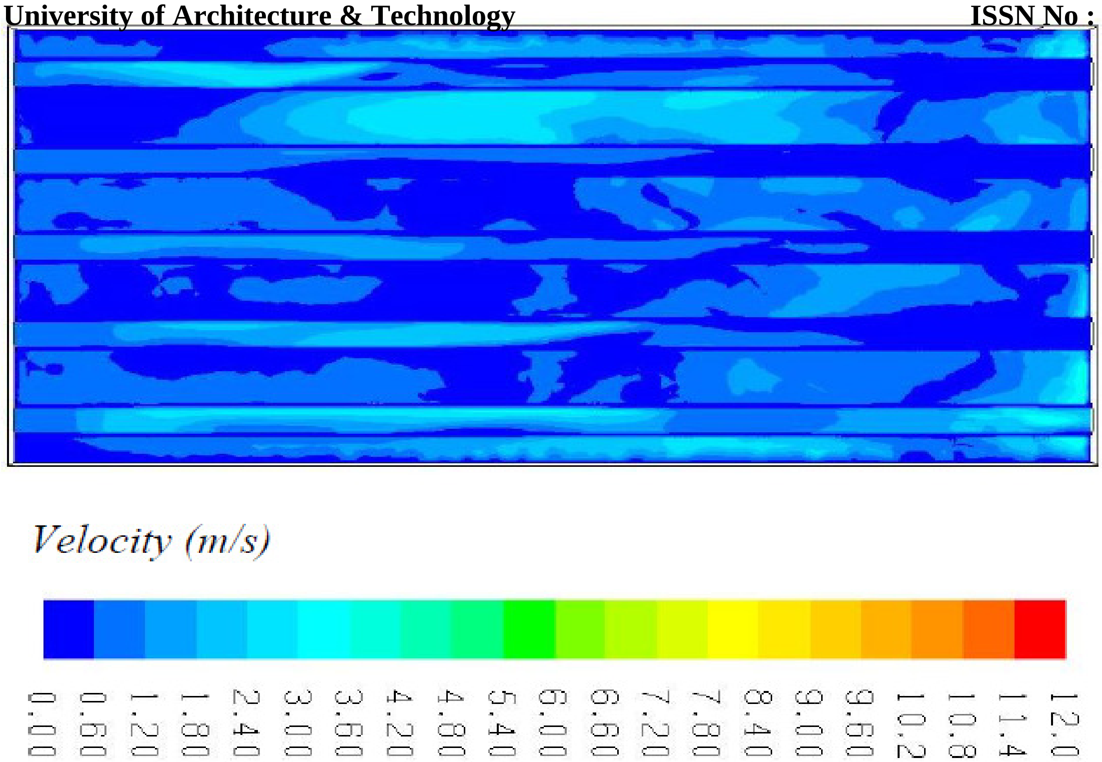

Fig.6: Velocity Distribution

Experimental Results

Sl. No		Temperature in deg C			
	Flow rate (kg/s	T1	T2	T3	T4
1	0.5	198	56	30	61
2	0.48	190	53	30	60
3	0.47	188	53	30	57
4	0.45	180	49	30	52

The results obtained during preliminary test and main tests are presented here. Along with this, an estimation of amount of waste heat recovered and the effectiveness is also done for both parallel and counter flow arrangements.

Figure 4 shows temperatures of fuel and exhaust gases at different engine loadings for simple concentric tube type heat exchanger. From the graph it is clear that as load is increased, temperature of fuel increased from 79°C to 133°C. Exhaust gas temperature also increased simultaneously from 255°C to 503°C. When load is increased beyond 4 kg exhaust gas temperature increased abnormally resulting in uncontrolled combustion of the engine.
The reason for increase in exhaust gas temperature along with fuel temperature could be due to extended combustion process till the end of exhaust stroke. For this reason, the injection timing of fuel is monitored by

which it came to know that injection timing is at TDC. Therefore it was decided to advance injection timing to 23° BTDC which is a standard for given diesel engine.

Figure 5 shows temperature variation of preheated fuel with parallel and counter flow arrangement. The preheated fuel temperature at maximum load for parallel flow found to be 73°C and for counter flow is 90°C respectively. ince the objective is to determine waste heat recovery from the engine. Waste heat recovered is the ratio of heat absorbed by fuel (Qf) in the heat exchanger to the heat available in the exhaust gas (Qe) at the given load conditions. In figure 6, normal condition indicates heat available in the exhaust gas (Qe) and red lines indicates heat absorbed by the fuel for both parallel and counter flow arrangements. It is observed that at 4kg and 6 kg load heat absorbed by fuel is higher for counter flow heat exchanger.

Figure 7 below shows heat recovered in percentages for both parallel and counter flow arrangements. It is found that waste heat recovered for counter flow is more than that for parallel flow at same loading. This is mainly due to higher fuel temperature achieved in counter flow heat exchanger for the same engine loading than parallel flow type. 75% heat is recovered in counter flow as against 40.63% in parallel flow type for 6 kg load at 1500 rpm.

REFERENCES

[1] A. Ouadha, Y. EI-Gotni, (2013), Integration of an ammonia-water absorption refrigeration system with a marine diesel engine: A thermodynamic study, Procedia computer science, vol.19,754-761.

[2] S. Bux, A. Tiwari, (2014), Vapour absorption based automobile air conditioning using exhaust waste heat of diesel engine through plate and frame heat exchanger, VSRD international journal of Mechanical, Civil, Automobile and production engineering, vol.4,3,pp. 25-35

[3] K. AlQdah, (2011), Performance and evaluation of aqua ammonia auto air conditioner system using exhaust waste energy, Energy procedia, vol.6, pp. 467-476.

[4] S. Manojprabhakar, R. Ravindranath, R. Vinothkumar, A. Selvakumar, K. Visagavel, (2014), International journal of research in engineering and technology, vol.3,no.11,299-304.

[5] S. Lakshmi Sowjanya, (2013), Thermal analysis of a car air conditioning system based on an absorption refrigeration cycle using energy from exhaust gas of an internal combustion engine, Advanced engineering and applied sciences: An international Journal, vol.3no.4, pp.47-53.

[6] A. Pathania, D. Mahto, (2012), Recovery of engine waste heat for utilization in air conditioning system in an automobile: An investigation, Global journal of researches in engineering Mechanical and Mechanics engineering, vol.12,no.1.

[7] G. Vicatos, J. Gryzagoridis, S. Wang, (2008), A car air-conditioning system based on an absorption refrigeration cycle using energy from exhaust gas of an internal combustion engine, Journal of energy in Southern Africa, vol. 19,no.4,pp.6-11.

[8] S. Mathapati, M.Gupta, S. Dalimbkar, (2014), A study on automobile air-conditioning based on absorption refrigeration system using exhaust heat of a vehicle, International Journal Of Engineering Research And General Science, vol. 2, no.4, pp.80-86.

[9] S. Maurya, S.Awasthi, S.Siddiqui, (2014), A cooling system for an automobile based on vapour absorption refrigeration cycle using waste heat of an engine, International Journal of engineering research and applications, vol.4, no.3, 441-444.

[10] C.Keinath, J. Delahanty, S. Garimella ,(2012), M. Garrabrant, Diesel engine waste-heat driven ammonia-water absorption system for space-conditioning applications, International Refrigeration And Air Conditioning Conference, paper 1318.
[11] R. Singh, S. Vats, N. Baghel, R. Kumar, (2014), Integration of water-lithium bromide absorption refrigeration system with diesel engine: A thermodynamic study, International conference of advanced research and innovation, pp.150-154.
[12] K. AlQdah, S. Alsaqoor, A. AI-Jarrah, (2011), Design and fabrication of auto air conditioner generator utilizing exhaust waste energy from a diesel engine, International journal of thermal and environmental engineering, vol.3,no.2,87-93.
[13] V. Krishnadasan, N. Mohammed Sajid, (2015), International journal of advanced research trends in engineering and technology, vol.2,no10, pp.868-872.
[14] A. Bangotra, A. Mahajan, (2012), Design analysis of 3 TR aqua ammonia vapour absorption refrigeration system, International journal of engineering research and technology, vol.1,no.8.