HEAT TRANSFER ANALYSIS IN LIGHT PASSENGER CAR RADIATOR WITH VARIOUS GEOMETRICAL CONFIGURATIONS

P.Natarajan[1]

[1]*Assistant Professor, Dept. of Mechanical Engineering, Government College of Engineering, Bargur, India.*

---***---

Abstract-*Now day's uses of Automobile are becoming high due to transportation. In Which radiator can dissipate maximum amount of heat for any given space, this has lead to the increased demand on the power packed radiators. The research work is focused on the Design and analysis of various aspects of single-phase convective heat transfer. The Design and analysis of the effects of various geometrical configurations on the Performance of a staggered type fin and tube radiator is presented. The method is demonstrated on fins-tube as elements for the heat transfer, but it can in principle be applied also to other geometrical configurations forms. The study is conducted to investigate the effects of a tube performance of the radiator having various geometrical configurations. Samples of fin and tube heat exchanger with transverse tube pitch of 13.87mm, tube outside diameter of 6.35mm and longitudinal tube pitch of 12mm are analysed in FLUENT 6.3. The radiator is made from aluminium fin and tube. Hot water is used for the tube side while atmospheric air is used as a working fluid in the air side. The results are presented as contours of Velocity, Pressure and Temperature with the fin length of the Radiator.*

***Key Words*: Staggered type pin and tube, transverse tube pitch, longitudinal tube pitch, FLUENT 6.3., aluminium plate.**

1. INTRODUCTION

Radiators are installed in automobiles to remove heat from engine. When driving a car, the engine produces intense heat which must be dissipated otherwise the engine will overheat.

Somchai Wongwises et al [1] An experimental study is conducted to investigate the effects of a fin pitch and number of tube rows on the air side performance of fin and tube heat exchangers having herringbone wavy fin configuration at various thickness. A total of 10 samples of fin and tube heat exchanger with a tube outside diameter of 9.53mm, transverse tube pitch of 25.4mm and longitudinal tube pitch of 19.05mm, having various fin pitches.The heat exchangers are made from

aluminium plate finned, copper tube. Ambient air is used as a working fluid in the air side while hot water is used for the tube side. The experimental results revealed that the fin pitch has an insignificant effect on the heat transfer characteristic.

M. S. Sohal et al [2] reported the circular tubes were removed instead of oval tubes and adding Winglets in the fins. This results in increase of heat transfer coefficient due to addition of winglets.

Ke-Wei Song et al [3] studied Winglets mounted on surfaces of the fin which increases heat transfer instead of tube bank fin heat exchanger.

J. He et al [4] experimentally investigated plain-fin round-tube heat exchanger in wind-tunnel testing. Winglet pairs are placed at an angle of 10 deg and 30 deg compared with pair 30 deg angle of attack produced better performance.

Jae dong Chung et al [5] reported numerical analysis is conducted on the rectangular winglet pair with plate heat exchanger to examine the louver fins and vortex generators combined effects. The test was conducted various angle of attack from in this 30 deg angle gave the better performance.

K.M.Kwak et al [6] investigated longitudinal vortices mounted air-cooled condensers. Result reveled that in –line arrangement of fin-tube have better thermal performance than staggered arrangement.

2. PROBLEM DEFINITION

Aerodynamic front end styling with narrower openings are decreasing the space available for circulation of cooling air.

These conditions demand a better understanding of the cooling fluid flow direction and thermal performance of the radiator to analysis various geometry configurations using CFD.

3. DESIGN AND ANALYSIS

The radiator of a commercially existing vehicle is chosen for the analysis to bring in the practicality to the study. The details of the geometry of the radiator were obtained by the process of reverse engineering. The dimensions of individual components of the radiator were measured using suitable measuring instruments then used to generate the Gambit model.

Discrediting the fluid domain, simulation of the fluid flow with heat transfer at steady state and post processing the results and drawing suitable conclusions.

3.1 RADIATOR SPECIFICATIONS

Radiator type	= staggered
Number of tube rows (N)	= 25
Number of tube columns	= 2
Radiator material	= Aluminium
Tube outside diameter (d_o)	= 6.35mm
Tube inside diameter (d_i)	=5.35mm
Tube thickness (T_t)	= 0.5mm
Longitudinal tube Pitch (S_L)	=12 mm
Transverse tube pitch (S_T)	=13.87 mm
Diagonal tube pitch (S_D)	= 13.87mm
Fin thickness (F_t)	= 0.3mm
Fin spacing (F_S)	= 1.60 mm
Fin pitch (F_P)	= 1.90 mm
Length of radiator (L)	= 300 mm
Depth of radiator (D)	= 28 mm
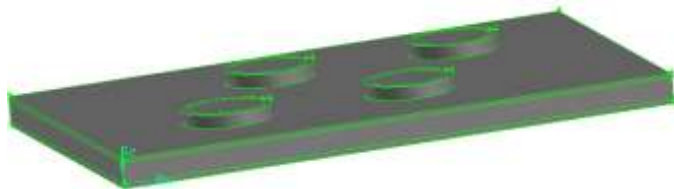
Fig- 3.1: GAMBIT Model

3.2 INLET CONDITIONS

Inlet temperature of air (30 °c)

Air density (ρ)	= 1.165 kg/m³
Air viscosity (μ)	= 18.63 × 10 $^{-6}$ Ns / m²
Thermal Diffusivity (α)	= 22.861 × 10^{-6} m²/s
Prandtl number	= 0.701
Specific heat (C_p)	= 1.005 KJ/Kg K
Thermal conductivity of air (k) = 0.02675 W / m K	

Inlet temperature of water (90 °c)

Water density (ρ)	= 967.5 kg/m³
Water viscosity (μ)	= 0.315× 10 $^{-3}$ Ns / m²
Thermal Diffusivity (α)	= 0.16585 × 10^{-6} m²/s
Prandtl number	= 1.98
Specific heat (C_p)	= 4.2055 KJ/Kg K
Thermal conductivity of air (k) = 0.67455 W / m K	

Aluminium Properties

Aluminium density (ρ)   = 2700 kg/m³

Specific heat (C_p)   = 0.871 KJ/Kg K

Thermal conductivity of Al (k) = 202.4 W / m K

Hydraulic diameter of fin for air side

Hydraulic diameter = 4A / p   (5.5)

Where,

A = Cross Sectional Area, P = Perimeter

= 4× 41.8×1.6 / 2 (41.8 + 1.6)

= 267.52 / 86.8

= 3.08 mm

Hydraulic diameter of tube for water side

Hydraulic diameter   = 4A / p   (5.6)

Where,

A =Cross Sectional Area, P = Perimeter

= (4×π / 4 × d_i^2) / π d_i

= 5.35 mm

Re = ρ V d / μ (5.7)

V_1 = (1000×18.63 × 10 $^{-6}$)/1.165×0.00308

= 5.1920 m/s

Where,

ρ = 1.165 Kg/m³

Re = 1000

d = 3.08 mm

μ - 18.63 × 10 $^{-6}$ Ns / m²

3.3 COMPUTATIONAL DOMAIN AND BOUNDARY CONDITIONS

The model for current study, consist of the computational domain of dimensions are 41.8 mm length, 4.2 mm width, and 28 mm height. Sketch of the computational domain is as per fig 3.3(a). The cooling fin base area is the same for all models. The inlet velocity of air V various ($V_{1=}$ 5.1920 m/s, 19.145 m/s, 15.576 m/s, 20.768 m/s, 25. 968 m/s) and the inlet temperature T_i is assumed uniform through the entire boundary equivalent to Reynolds Number of 1000 to 5000 respectively at inlet temperature of T_{in} = 303 K (Pr = 0.7). Water of the tube are smooth, and have the inlet velocity of water V ($V_{1=}$ 5.1920 m/s) and the inlet temperature T_{in} are assumed uniform through the entire boundary. For all cases, the incompressible hot fluid is water at 363 K. The shape chosen are circular, oval tubes and the fin spacing on the heat transfer is 1.6mm.
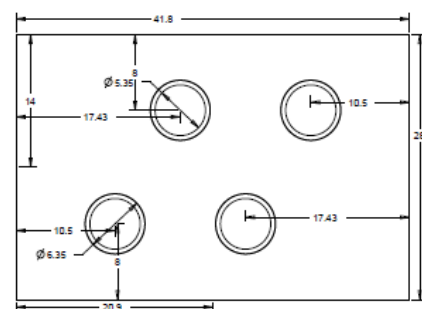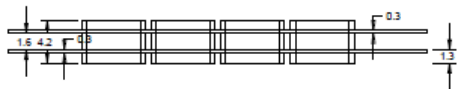
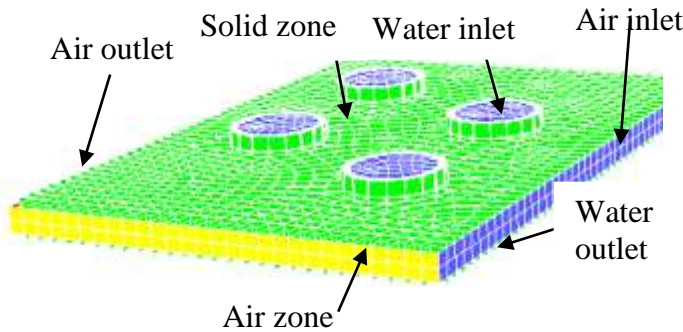**Fig- 3.3(a):** Schematic of a computational domain
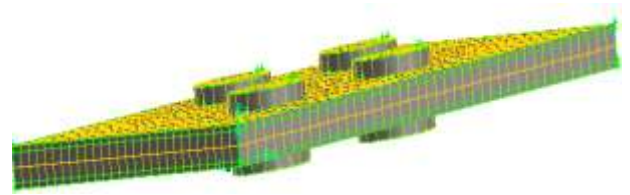

Fig- 3.3(b): Computational domain

The computational domain was as per fig-3.3(a) and fig-3.3(b), shows a schematic and Gambit model for the computational analysis respectively. The model having a width, length and height. The domain consists of tube and fin defined as air and water.

The geometric similarity between the rows of tube and fin helps us in limiting the computational domain to a four tube and adjoining fin arrangement. Hence, the fluid domain is created for a double fin and tube assembly and design analysis is carried out. The fluid domain includes the air flow volume and the coolant flow volume. The problem is solved as a conjugate heat transfer requiring the thickness of the tube and fin also to be modeled.

Fig- 3.3(c): Mesh model from GAMBIT

In Gambit model the fin spacing is air domain, ___ is hot water domain, tube thickness and fin ___ckness are discredited with same mesh density in accordance to the physics of fluid flow and heat transfer. The grid independence study of the simulation is carried out to arrive at the minimum number of elements required to maintain the required stability and accuracy in the computation. The finalized element count and the related aspects are listed below .The meshed geometry for a four tubes and double fin assembly of the radiator as shown in fig 3.3(c).

Element count in tube-fin assembly

Number of Hexahedral cells	= 8904
Number of quadrilateral cells	= 33678
Total Number of Cells	= 42582
Skewness	= 0.97%

4. RESULTS AND DISCUSSION

4.1 Contours of velocity magnitude

Design and Analysis of a single elliptical cylinder with a minor to major axes ratio of 1, 0.69, 0.47 , 0.909 and 1.1 in a flow of air having Reynolds numbers of $1000 < Re_d < 5000$ with angles of attack 0 °. Re_d is the Reynolds number based on the shape of circular tube and Various oval tube conditions. A different velocity field around the circular tube staggered arrangement could be observed compared with that of the oval tube staggered arrangement.Whereas the flow through the staggered arrangement periodically separates and joins again. Further, because in the staggered arrangement each tube is located in the adjacent of the previous pins, the flow separated from the first tube row not hits at second rows tubes.So each tube makes separate boundary layer.
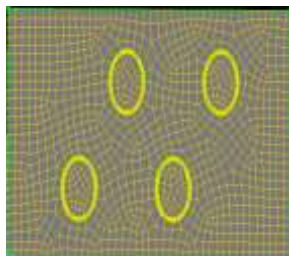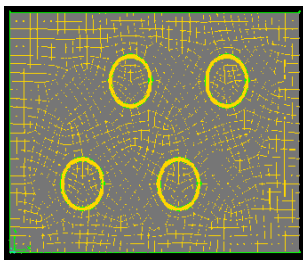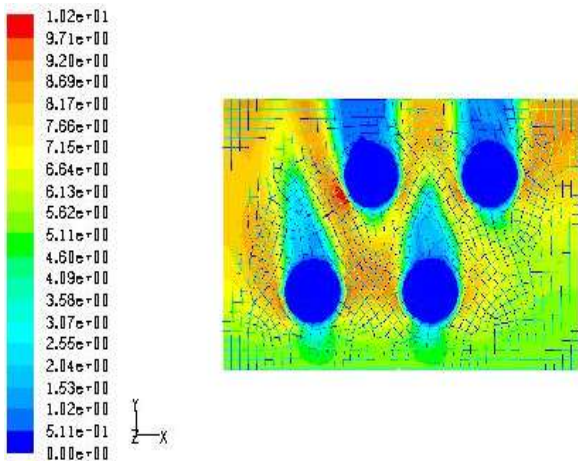
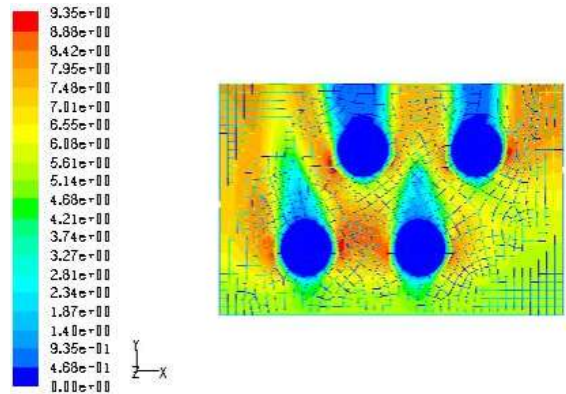**Fig-(4.1a):** Circular tube (Ratio=1)
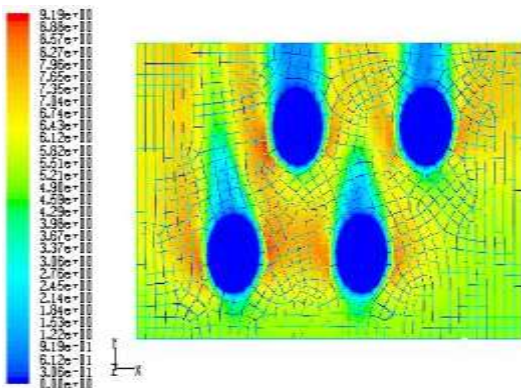

Fig-(4.1b): Oval tube (Ratio=0.69)

Fig-(4.1c): Oval tube-I (Ratio=0.47)

Fig-(4.1d): Oval tube-II (Ratio=0.909)
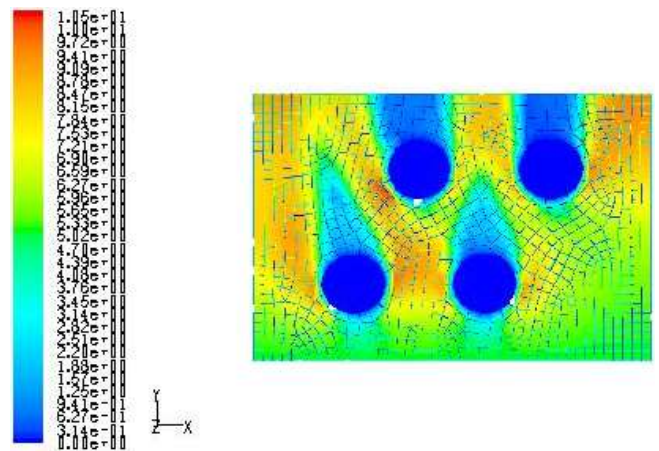
Fig-(4.1e): Oval tube-III (Ratio=1.1)
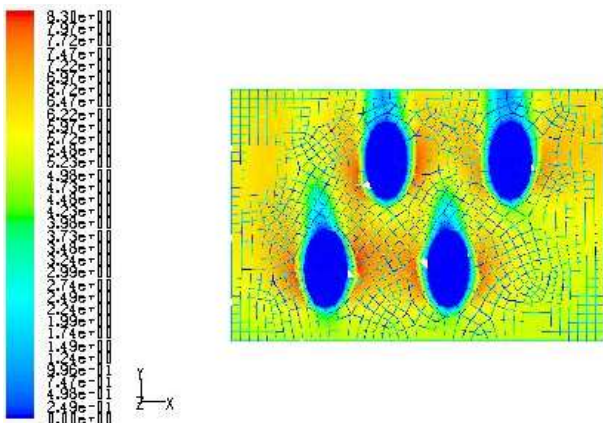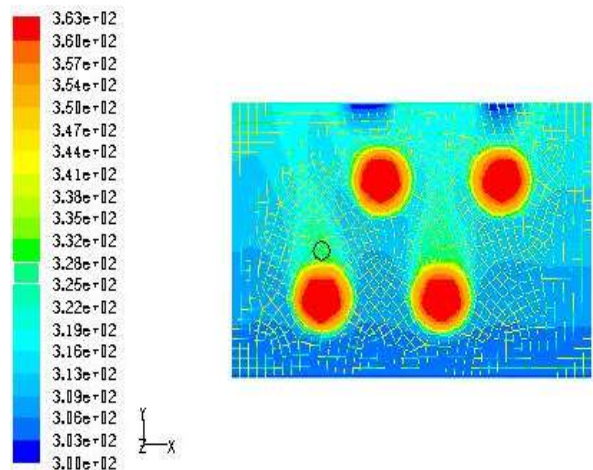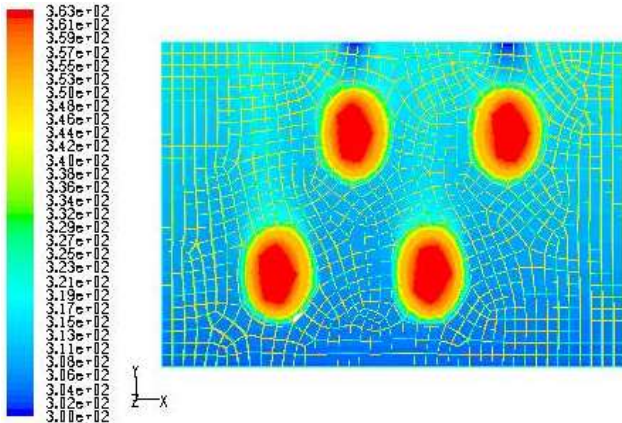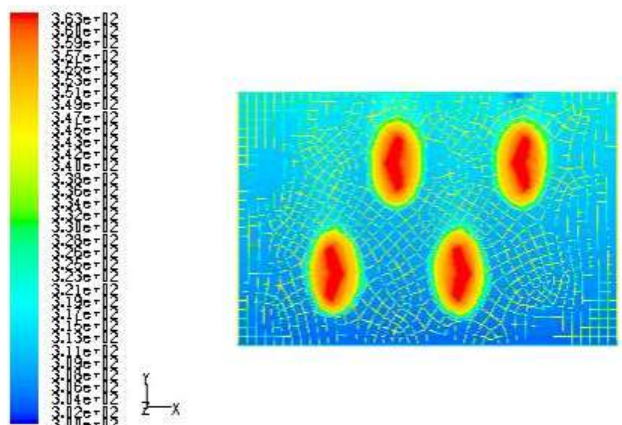
Fig- 4.1: Contours of velocity magnitude

4.2 Contours of Static Temperature

Fig-(4.2a): Circular tube (Ratio=1)

Fig-(4.2b): Oval tube (Ratio=0.69)

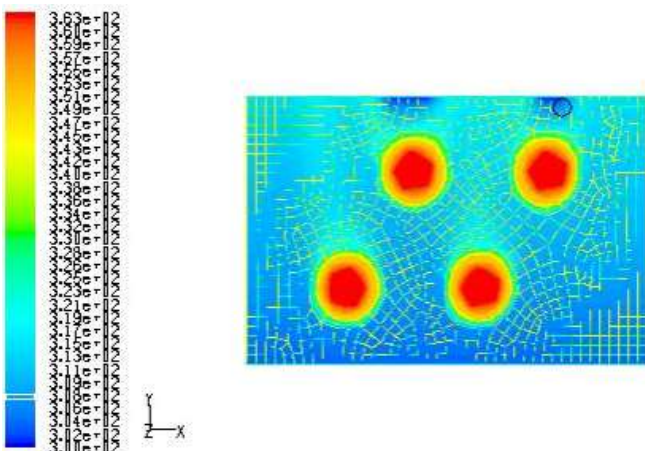**Fig-(4.2c):** Oval tube-I (Ratio=0.47)

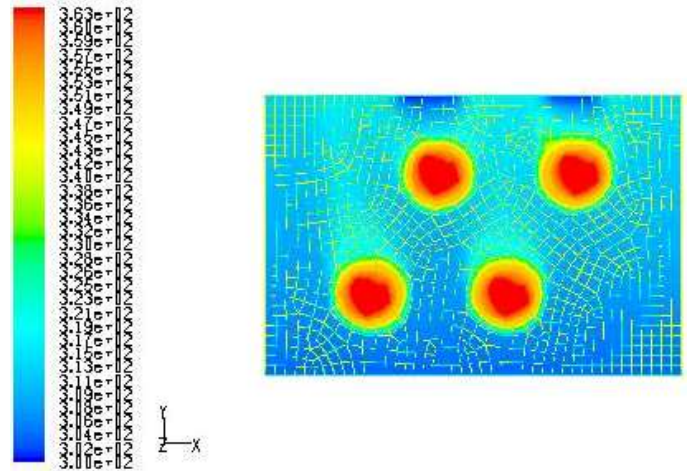**Fig-(4.2d):** Oval tube-II (Ratio=0.909)

Fig-(4.2e): Oval tube-III (Ratio=1.1)

Fig- 4.2: Contours of static temperature

The thermal characteristics of the convective heat transfer from fin-tube can be observed on the temperature field in fluid and solid parts of the computation domain. The complete temperature changes in the computation domain. The presented temperature fields, as expected, shows that the thermal boundary layer develops around tubes, but thickest thermal boundary layers around the tubes when more amount of heat is absorbed in air by convection. whereas for the subsequent rows, quite small temperature differences between the tubes and the air were observed .Here it should be mentioned that the intensive reduction of the temperature difference between the tubes and the surrounding air in addition to the fin cross-section and surrounding velocity field depends also on the fin material chosen. The temperature difference between circular tubes and oval tubes occur but circular tubes shows higher temperature values than oval tubes.
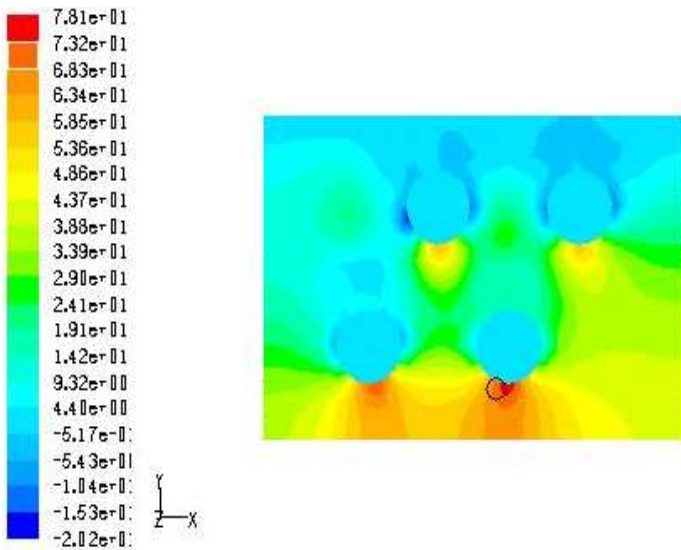
4.3 Contours of static pressure
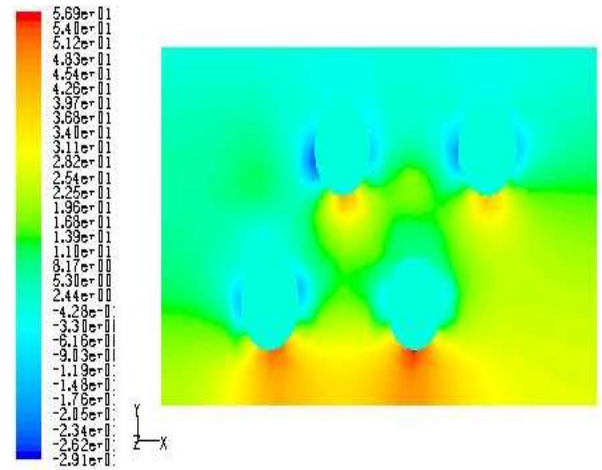

Fig-(4.3a): Circular tube (Ratio=1)
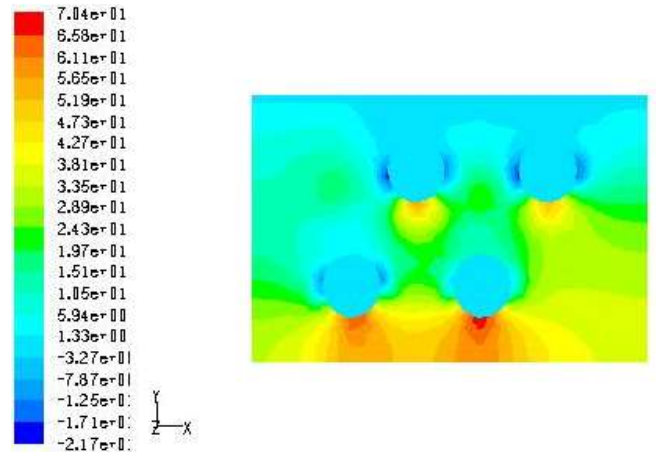

Fig-(4.3b): Oval tube (Ratio=0.69)

Fig-(4.3c): Oval tube-I (Ratio=0.47)
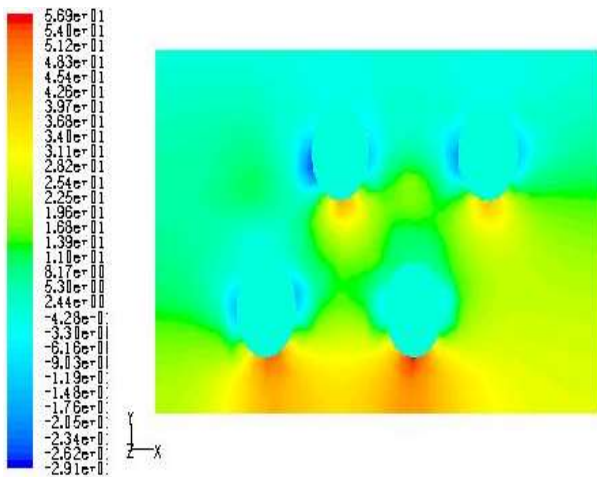

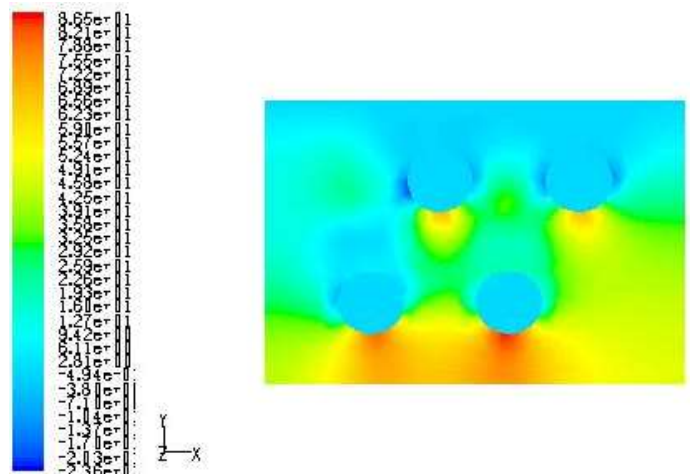**Fig-(4.3d):** Oval tube-II (Ratio=0.909)

Fig-(4.3e): Oval tube-III (Ratio=1.1)

Fig- 4.3: Contours of static pressure

The flow across a cylinder when flow field can be divided into two regions. A boundary layer region near the surface and inviscid region away from the surface. The pressure gradient along the surface of the cylinder is not zero, and in fact this pressure gradient is responsible for the development of a separated flow region on the back side of the cylinder. The separation of flow affects the drag force on a curved surface to a great extent.

5. PERFORMANCE EVALUATION

The outlet temperature performance of existing circular tube and various geometrical configurations are oval tube, oval tube-I, oval tube-II, oval tube-III. Inlet of fin side air velocity and air temperature are applied based on the hydraulic diameter of the fin areas. Outlet temperature of oval tube-III have higher ranging value than existing circular tube values.
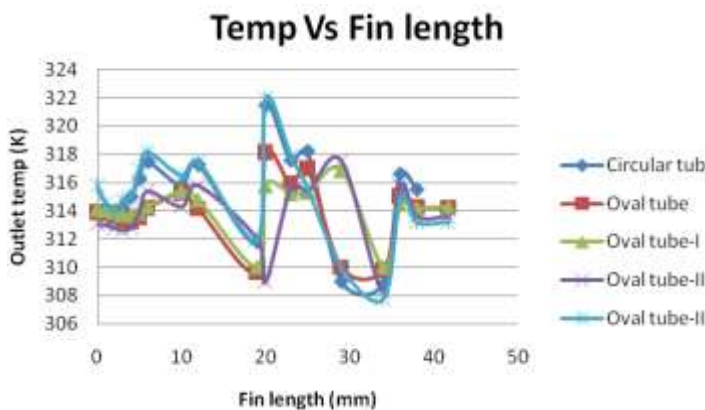

Fig- 5.**1:** Outlet temp Vs fin length

The fig-5.1 shows outlet temperature of various geometrical configurations for entire fin length. The air entre to the atmospheric conditions parallel to major axis when increase major axis value then decrease the out let temperature range. So change the major to minor axis ratio = 1.1 when atmospheric air entre parallel to the minor axis then increase outlet temperature values.

The inlet pressure drop performance of existing circular tube and various geometrical configurations are oval tube, oval tube-I, oval tube-II, oval tube-III. Inlet of fin side air velocity and air temperature are applied based on the hydraulic diameter of the fin areas. Pressure drop of oval tube-III higher ranging value than existing circular tube values.
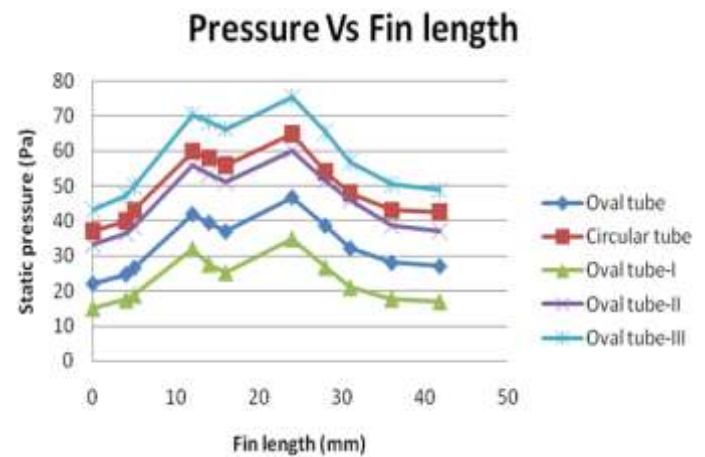

Fig- 5.2: Pressure Vs fin length

The fig-5.2 shows pressure drop of various geometrical configurations for entire fin length. The air entre to the atmospheric conditions parallel to major axis when increase major axis value then decrease the pressure drop range. So change the major to minor axis ratio = 1.1 when atmospheric air entre parallel to the minor axis then increase pressure drop values.

The outlet velocity performance of existing circular tube and various geometrical configurations are oval tube, oval tube-I, oval tube-II, oval tube-III. Inlet of fin side air velocity and air temperature are applied based on the hydraulic diameter of the fin areas. Outlet velocity of oval tube-III higher ranging value than existing circular tube values.
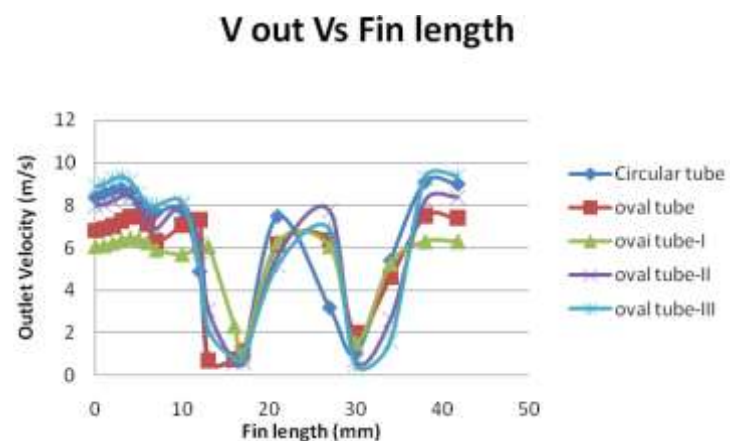

Fig- 5.3: Outlet velocity Vs fin length

The fig- 5.3 shows outlet velocity of various geometrical configurations for entire fin length. The air entre to the atmospheric conditions parallel to major axis when increase major axis values then decrease the outlet velocity range. So change the major to minor axis

ratio = 1.1 when atmospheric air entre parallel to the minor axis then increase outlet velocity values.

6. CONCLUSIONS

The fluid flow and heat transfer analysis of a staggered type tube-fin arrangement of an automotive radiator is successfully carried out by using numerical simulation built in commercial software FLUENT. The variations of the pressure, temperature and Velocity in the direction of coolant flow and air flow are presented and analysed.

The outlet temperature and maximum velocity for circular and oval tube-III are nearly same other oval tube, oval tube-I, oval tube-II are less than oval tube-III.

Pressure is high for oval tube-III than circular tube.

REFERENCE

[1]   Somchai wongwises, and yutasak chokeman, "Effect of fin pitch and number of tube rows on the air side performance of a herringbone wavy fin and tube heat exchangers,"2002 Elsevier Ltd.

[2]   M. S. Sohal, and J. E. O'Brien,"Improving air-cooled condenser performance using winglets and oval tubes in a geothermal power plant,"Geothermal Resources Council Transactions,2001, Vol. 25, pp.1-7.

[3]   Ke-Wei Song, Liang-Bi Wang, Ju-Fang Fan, Yong-Heng Zhang, and Song Liu,"Numerical study of heat transfer enhancement of finned flat tube bank fin with vortex generators mounted on both surfaces of the fin", 2007,Heat and Mass Transfer, Vol. 44, pp.959-967.

[4]   J. He, L. Liu, and A. M. Jacobi,"Air-Side Heat-Transfer Enhancement by a New Winglet-Type Vortex Generator Array in a Plain-Fin Round-Tube Heat Exchanger",2010 Journal of Heat Transfer, Vol. 132, pp.1-9.

[5]   Jae dong chung, Byung kyu park, and Joon sik lee,"The combined effects of angle of attack and louver angle of a winglet pair on heat transfer enhancement", International Journal of enhanced Heat Transfer,2003, Vol.10, pp.31-43.

[6]   K.M. Kwak, K. Torii, and K. Nishino, "Heat transfer and   flow characteristics of fin-tube bundles with and without winglet-type vortex generators", 2002, Springer,Vol.33, pp.696-702.