International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249-8958 (Online), Volume-1 Issue-6, August 2012

Effect of Variation in Pitch of Tube on Heat Transfer Rate in Automobile Radiator by CED Analysis

P. K. Trivedi, N. B.Vasava

Abstract— ***The demand for more powerful engines in smaller hood spaces has created a problem of insufficient rates of heat dissipation in automotive radiators. This has lead to the increased demand on the power packed radiators, which can dissipate maximum amount of heat for any given space. The geometry of the finned-tube heat exchanger is an intricate one and there are no analytical optimization schemes available to optimize their design, while experimental trial and error is far too time consuming. The radiator designs at present depend on the empirical methods, wherein existing experimental data is used as the thumb rules for the design process. However, for any preliminary design the performance of the radiator can be accessed through Computational Fluid Dynamics (CFD) in prior to the fabrication and testing. In this thesis, first of all solid modeling of heat exchanger in Solid works is prepared and then this solid model is transferred to ANSYS Workbench mesh module for meshing. After completing meshing, this meshed model is transferred to ANSYS CFX for CFD Analysis. Once CFD Analysis is completed with ANSYS CFX, all the flow parameters like heat transfer rate, temperature contour etc. is identified. After getting all the flow parameter it is possible to examine how the heat transfer rate of radiator can be enhanced. For that purpose one geometrical parameter e.g. pitch of tube is varied. As a result of this parametric study, the effect of pitch of tube for best configured radiator for optimum performance is suggested.***

Keywords: CFD, Heat transfer, Modeling, Pitch, radiator design, Simulation, etc.

I. INTRODUCTION

It is known that heat transfer increases as the surface area of the radiator assembly is increased. But due to the customer demand of small car having inside better space, the manufacturers of commercial vehicles are facing a substantial increase of heat release into the cooling system. This leads to change the geometry by modifying the arrangement of tubes in automobile radiator to increase the surface area for better heat transfer. The modification in arrangement of tubes in radiator is carried out by studying the effect of pitch of tube by CFD analysis using CFX.

II. EXPERIMENTAL HEAT TRANSFER CALCULATION

Radiator is considered as a Shell and Tube Type Heat Exchanger and Overall Dimensional Experimental Radiator are as under.

Shell Side Data:-

Media: - Air
Temperature: - 35°C
Inlet Velocity: - 30 Kmph (Vehicle Speed)
Outlet Pressure: - 1.01325 bar

Manuscript received on August, 2012.
P. K. Trivedi,
N. B.Vasava,
.

Tube Side Data:-

Diameter of Tube: - 7 mm
No. of Tubes: - 29
Media: - Water + Ethanol (50%)
Temperature (Engine):- 95 °C
Inlet Velocity: - 2m/s
Outlet Pressure: - 1.01325 bar

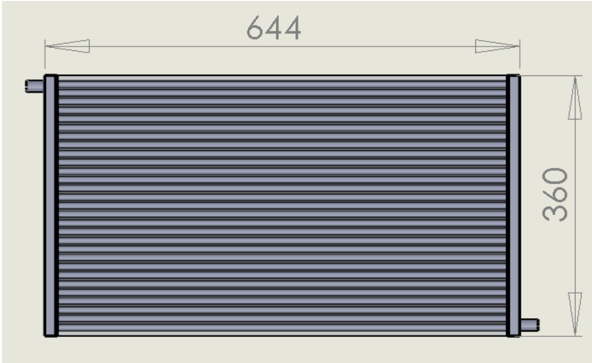

Figure (1): Experimental heat exchanger

Table 1: Experimental Results Given by Company

Sr No.	Velocity of Car Kmph	Velocity of Car m/s	Engine Temperature Tube Side Inlet	Tube Side Outlet Temperature (Experimental)	Shell Side inlet temperature	Shell Side Outlet temperature (Experimental)
1	30	8.333333333	95	87.12	35	60.52
2	40	11.11111111	95	86.92	35	62.15
3	50	13.88888889	95	86.52	35	63.52
4	60	16.66666667	95	86.14	35	63.89
5	70	19.4444444 4	95	85.95	35	64.52
6	80	22.22222222	95	85.14	35	65.27
7	90	25	95	84.96	35	66.29
8	100	27.77777778	95	84.52	35	68.26

Mass Flow Rate, Heat Transfer Rate and Overall Heat transfer Co-efficient are calculated as per its respective equations e.g. $m = A * V * \rho$, $Q = m * C_p * \Delta T$ and

$$U = \frac{Q}{A * (T_{outlet} - T_{mean})}$$

Published By:
Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP)

Table 2: Experimental Result summary

Sr No	Velocity of Car Kmph	Engine Temperature Tube Side Inlet	Tube Side Outlet Temperature	Shell Side inlet temperature	Shell Side Outlet temperature	Mass of Air (m)	Heat Transfer Rate
1	30	95	87.12	35	60.52	2.013	53.3696625
2	40	95	86.92	35	62.15	2.684	76.4727964
3	50	95	86.52	35	63.52	3.355	98.200179
4	60	95	86.14	35	63.89	4.026	121.7438244
5	70	95	85.95	35	64.52	4.697	143.267894
6	80	95	85.14	35	65.27	5.368	169.5901504
7	90	95	84.96	35	66.29	6.039	195.9124068
8	100	95	84.52	35	68.26	6.71	231.980133

III. CFD ANALYSIS

3.1 Modeling of Radiator

After performing simple calculation, the modeling has been performed on the Solid works 2009 version and then after the analysis work has been performed on the ANSYS12.0 version.

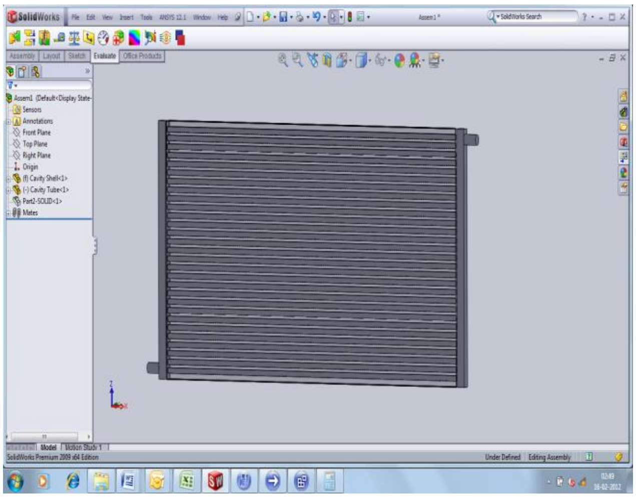

Figure (2): CAD model of radiator

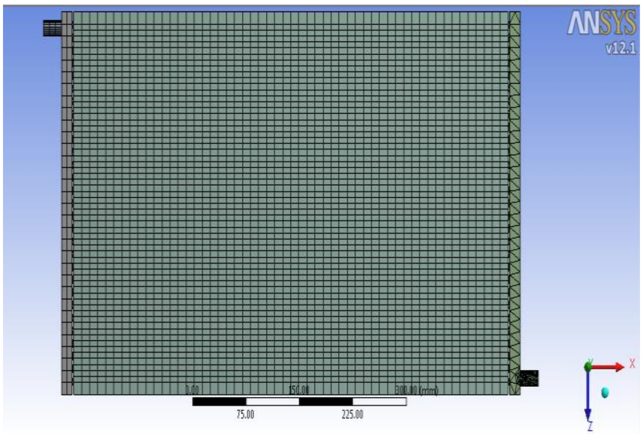

Figure (3): Meshed model of radiator

3.2 CFD analysis

The Cavity Pattern method is used for CFD Analysis of radiator in this study. In cavity model, there are basically two Domains. D-1:-Water with addition of glycols & D- 2:- Air Domain The input data and boundary conditions are chosen from the study of Changhua Lin and Jeffrey Saunders [5]. The properties of air and coolant were defined for standard conditions and kept constant throughout the analysis.

3.3 Results of Analysis

3.3.1 Tube Side Results:-

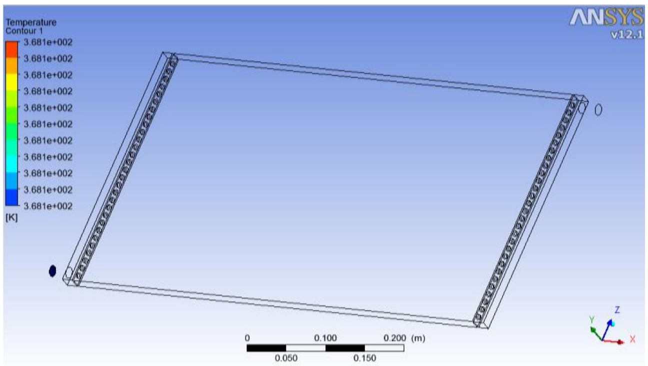

Figure (4): Inlet Temperature:-368*F (95* C)

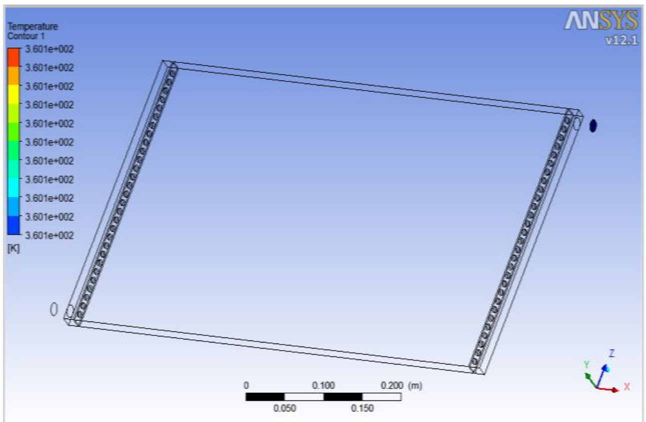

Figure (5): Outside Temperature:-359.94*(86.94*C)

3.3.2 Shell Side Results:-

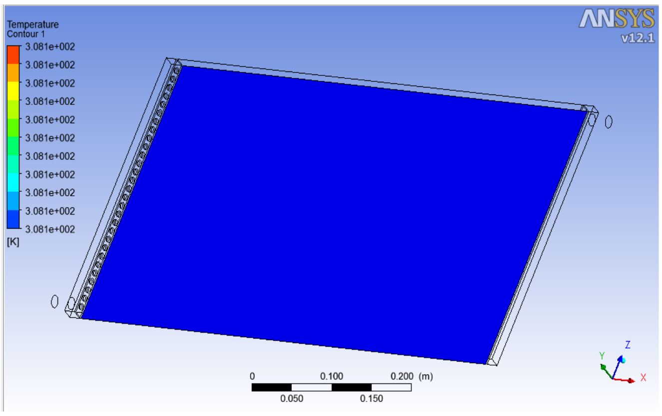

Figure (6): Inlet Temperature:-308*F (35* C)

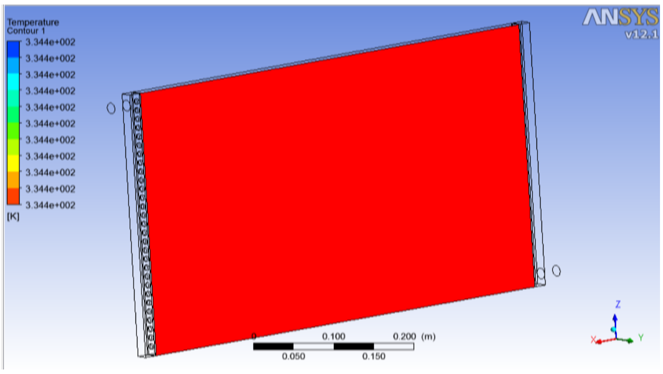

Figure (7): Outside Temperature:-334.25*F (61.25* C

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

As Per above Procedure, We have done 8 iteration for different Velocity and inlet temperature configuration which results are as below.

Table 3: CFD Result summary

Sr No.	Velocity of Car Kmph	Engine Temperature Tube Side Inlet	Tube Side Outlet Temperature	Shell Side inlet temperature	Shell Side Outlet temperature	Mass of Air(m)	Heat Transfer Coefficient	Thermal Efficiency
1	30	95	86.94	35	61.25	2.013	53.3696625	42.857143
2	40	95	86.652	35	63.21	2.684	76.4727964	44.629014
3	50	95	86.24	35	63.98	3.355	98.200179	45.295405
4	60	95	85.96	35	64.94	4.026	121.7438244	46.104096
5	70	95	85.79	35	65.2	4.697	143.267894	46.319018
6	80	95	84.15	35	66.28	5.368	169.5901504	47.193724
7	90	95	83.86	35	67.12	6.039	195.9124068	47.854589
8	100	95	83.15	35	69.23	6.71	231.980133	49.443883

3.4 CFD Validation

To validate the CFD results, comparisons were drawn between obtained results and received experimental data which is given below.

Table 4: Comparison of Experimental results and CFD Results

Sr No	Velocity of Car Kmph	Engine Temperature Tube Side Inlet	Tube Side Outlet Temperature (Experimental)	Tube Side Outlet Temperature	Shell Side inlet temperature	Shell Side Outlet temperature (Experimental)	Shell Side Outlet temperature	Percentage of variation Tube Side Temperature	Percentage of variation Shell Side Temperature
1	30	95	87.12	86.94	35	60.52	61.25	0.2066	1.2062
2	40	95	86.92	86.652	35	62.15	63.21	0.3083	1.6954
3	50	95	86.52	86.24	35	63.52	63.98	0.3236	0.7241
4	60	95	86.14	85.96	35	63.89	64.94	0.2089	1.6434
5	70	95	85.95	85.79	35	64.52	65.2	0.1861	1.0539
6	80	95	85.14	84.15	35	65.27	66.28	1.1745	1.5474
7	90	95	84.96	83.86	35	66.29	67.12	1.2947	1.2520
8	100	95	84.52	83.15	35	68.26	69.23	1.6209	1.4210

IV. CFD ANALYSIS BY VARYING PITCH OF TUBE.

4.1 Modeling of modified radiator

The existing cad model of this radiator is replaced by making modification in it. In this model the existing Pitch of Tube is 12 mm and Dia of Tube is 7 mm. The Pitch of Tube is decreased and taking 10 mm. Now, the analysis is carried out by modified radiator.

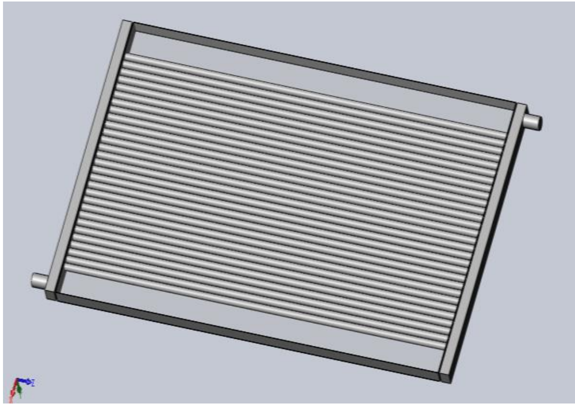

Figure (8): Modified Radiator Model (Pitch)

4.2 Results of Analysis:-

4.2.1 Tube Side Results:

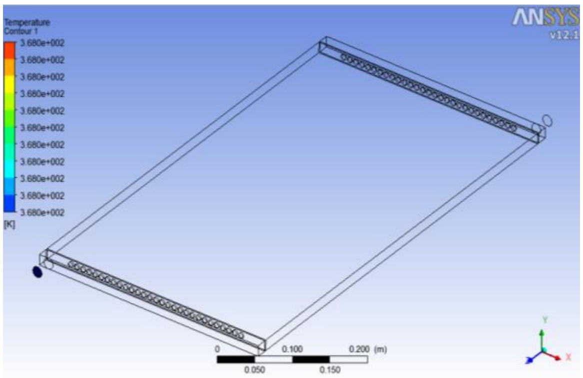

Figure (9): Inlet temperature contour

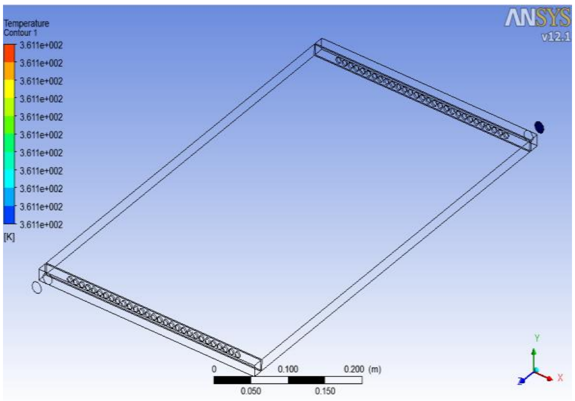

Figure (10): Outlet Temperature Contour

4.2.2 Shell Side Results:

*Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)*

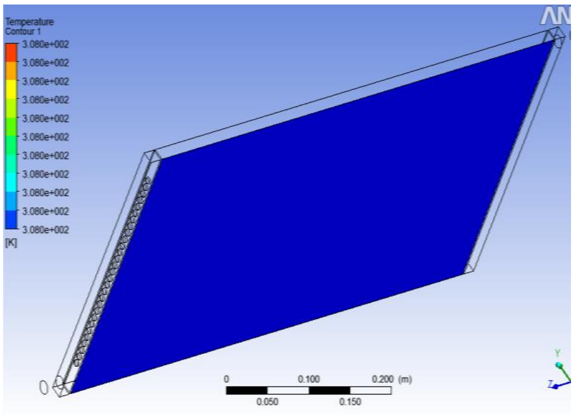

Figure (11): Inlet Temperature contour

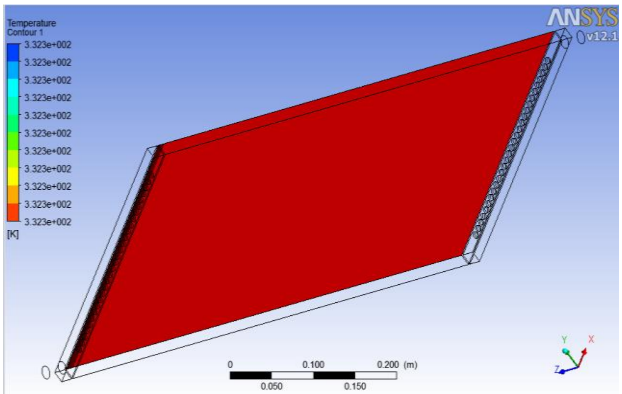

Figure (12): Outlet Temperature contour

As same as above procedure, the another trial is made by increasing Pitch from 12 mm to 14 mm using ANSYS CFX keeping the boundary condition as same as taken.

4.2.3 Result table showing effect of pitch

Table 5: Pitch effect result table

Sr No.	Pitch of tube	Engine Temperature Tube Side Inlet	Tube Side Outlet Temperature	Shell Side inlet temperature	Shell Side Outlet temperature	Heat Transfer Coefficient	Thermal Efficiency
1	12	95	86.94	35	61.25	53.369	42.85714
2	10	95	88.1	35	59.3	49.405	40.97807
3	14	95	85.14	35	60.12	51.072	41.7831

V. CONCLUSIONS:-

The fluid flow and heat transfer analysis of an automotive radiator is successfully carried out using numerical simulation built in commercial software ANSYS 12.1. Above Results Shows that as the pitch of tube is either decreased or increased, the heat transfer rate decreases. So we can say that optimum efficiency is coming at the pitch of 12 mm. The study forms a foundation for the fluid flow analysis of an automotive radiator. With the computational time and resources available, the results obtained were found to be satisfactory. However, to account for the variation of the inlet conditions with time as in practical cases, transient analysis can be done.

REFERENCES

[1]. J.P.Holman, 2002, Heat transfer, Tata-McGraw-Hill Publications.
[2]. ANSYS 12.1 User Guidelines.
[3]. Hucho, W.H., Aeodyanamics of Road Vehicles, 4th Edition, SAE International, 1998.
[4]. Hucho, W.H., Aeodyanamics of Road Vehicles, 4th Edition, SAE International, 1998.
[5]. Changhua Lin and Jeffrey Saunders, 2000, "The Effect Of Changes in Ambient and Coolant Radiator Inlet Temperatures and Coolant Flow rate on Specific Dissipation", SAE Technical Papers, 2000-01-0579.
[6]. Wikipedia, the free encyclopedia, "Radiator", Wikipedia@ Wikimedia Foundation, Inc, 2006.
[7]. Incropera, F.P.; and DeWitt, D.P. (2002). Fundamentals Of heat and mass Transfer. (5th Ed.), Wiley, New York.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)