International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 4, April - 2013

Effect of Percentage of Coolant and Water Mixture for Cooling In Automobile Radiator Using CFD Analysis.

Gita R. Rohit
Post Graduate Student,
Mechanical Engineering Department,
Shri S'ad Vidmandal Institute of Engg. & Technology, Bharuch

Dipak C. Gosai
Associate Professor,
Mechanical Engineering Department,
Shri S'ad Vidmandal Institute of Engg. & Technology, Bharuch

ABSTRACT

The thermal performance of an automotive radiator plays an important role in the performance of an automobile's cooling system and all other associated systems. For a number of years, this component has suffered from little attention with very little changing in its manufacturing cost, operation and geometry. As opposed to the old tubular heat exchangers currently form the backbone of today's process industry with their advanced performance reading levels tubular heat exchangers can only dream of. In this Paper, we perform first solid modeling of radiator in Solid works 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 complied with ANSYS CFX. We would be able to find all flow parameters like heat transfer rate, temperature contour etc. After getting CFD Result and experimental data we can say that both results are nearest result. So, after analysis says that Percentage of Ethylonal Glycol increases as a coolant temperature is decreased.

KEY WORDS: Radiator, temperature, Domain, Ethanol, Coolant.

INTRODUCTION ABOUT CFD

The equations of fluid mechanics which have been known for over a century are solvable only for a limited no. of flows. The known solutions are extremely useful in understanding fluid flow but rarely used directly in engg analysis or design. CFD makes it possible to evaluate velocity, pressure, temperature, and species concentration of fluid flow throughout a solution domain, allowing the design to be optimized prior to the prototype phase.

Availability of fast and digital computer makes techniques popular among engineering community. Solutions of the equations of fluid mechanics on computer has become so important that it now occupies the attention of a perhaps a third of all researchers in fluid mechanics and the proportion is still is increasing. This field is known as computational fluid dynamics. At the core of the CFD modeling is a three-dimensional flow solver that is powerful, efficient, and easily extended to custom engineering applications. In designing a new mixing device, injection grid or just a simple gas diverter or a distribution device, design engineers need to ensure adequate geometry, pressure loss, and residence time would be available. More importantly, to run the plant efficiently and economically, operators and plant engineers need to know and be able to set the optimum parameters.

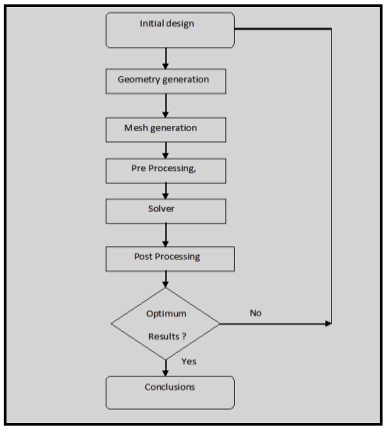


Fig:1 CFD Procedures steps chart

PROCEDURE OF CFD ANALYSIS FOR RADIATOR

Technical Specification and Dimensional Details

Radiator is considered as a Shell and Tube Type Heat Exchanger.

Overall Dimensional Drawing of Experimental Radiator.

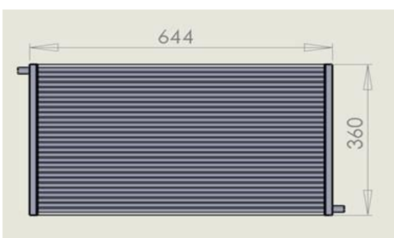


Fig:2 Overall Dimensions of Radiator

Shell Side Data:-

Media:- Air

Temperature:- 35°C

Inlet Velocity: - 30 Kmph (Vehicle Speed)

Outlet Pressure:- 1.01325 bar

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

Define Porous Domain for Fins.

Domain Type: - Porous

Domain Material: - Air Ideal Gas

Domain Motion: - Stationary

Volume Porosity: - 0.8

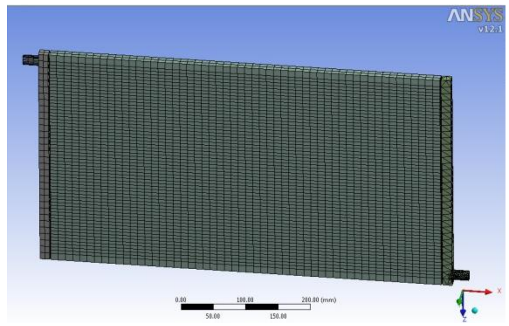


Fig:3 Meshed Radiator

Define Porous Domain for Fins.

Domain Type: - Porous

Domain Material: - Air Ideal Gas

Domain Motion: - Stationary

Volume Porosity: - 0.8

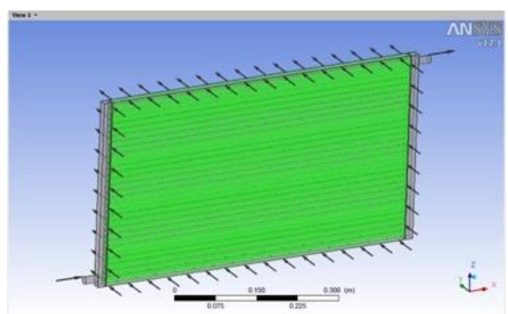


Fig:4 (Domain -1 Porus domain)

Define Domain for Water + Ethanol.

Domain Type:- Fluid

Domain Material: - Water + Ethanol

Domain Motion:- Stationary

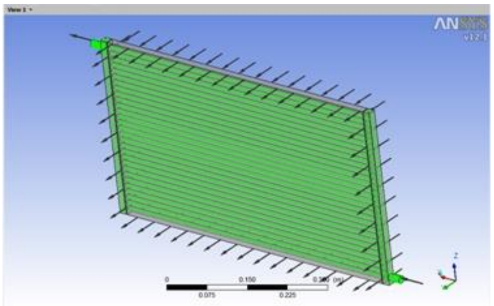


Fig:5(Domain -2 Water+Ethonal)

Result Analysis

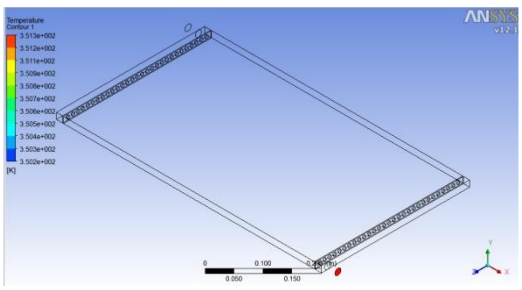


Fig:6 Result Analysis

EXPERIMENTAL DATA

Experimental Results

Shell Side inlet temperature-35

Engine Temperature Tube Side Inlet-90

Case :1 Velocity of Car :50 Kmph=13.88m/s

Case :2Velocity of Car:30 Kmph=8.33m/s

SR No.	Velocity of Car m/s	Tube Side Outlet Temperature (Experimental)	Shell Side Outlet temperature (Experimental)
1	13.88	79.12	63.52
2	8.33	77.12	61.25

Case1 :Velocity 50 m/s

Heat Transfer Calculation:-

Surface Area of Shell is

A = h*l

= 0.36*0.61

=0.2196 m^2

Mass Flow Rate is

$m = A * V * \rho$

= 0.2196 * 13.88 * 1.1

= 3.35 Kg/s

Heat Transfer Rate is

$Q = m * C_p * \Delta T$

= 3.35 * 1.0044* (63.52-35)

= 98.20 W

Overall Heat transfer Co-efficient is

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

$$= \frac{98.20}{0.2196*(63.52-35)}$$

= 15.8 W/ m^2 K

Case 2: Velocity:30 m/s

Heat Transfer Calculation:-

Surface Area of Shell is

A = h*l

= 0.36*0.61

= 0.2196 m^2

Mass Flow Rate is

$m = A * V * \rho$

$= 0.2196 * 8.33 * 1.1$

$= 2.0122$ Kg/s

Heat Transfer Rate is

$Q = m * C_p * \Delta T$

$= 2.0122 * 1.0044 * (61.25-35)$

$= 53.0527$ W

Overall Heat transfer Co-efficient is

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

$$= \frac{53.05}{0.2196*(61.25-35)}$$

$= 9.20$ W/ m^2 K

SR No.	Velocity of Car m/s	Heat Transfer Coefficient
1	13.88	98.200179
2	8.33	53.0527

Table 1 Experimental result table (Heat transfer co-efficient)

EFFECT OF RADIATOR TUBE SIDE FLUID ON RADIATOR PERFORMANCE

Modification 1

Shell Side:-

Fluid:- Air

Inlet Temperature: - 35°C

Inlet Velocity: - 30 Kmph

Outlet Pressure: - 1.03125bar

Tube Side:-

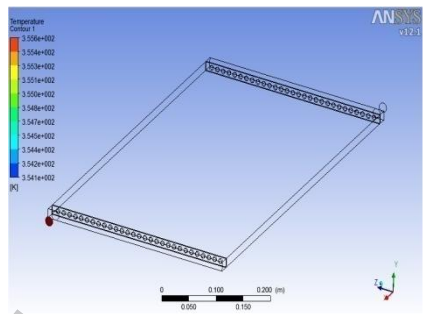
Fluid:- Water + 70% Ethanol

Inlet Temperature: - 95°C

Inlet Velocity: - 2 m/s

Outlet Pressure: - 1.01325 bar

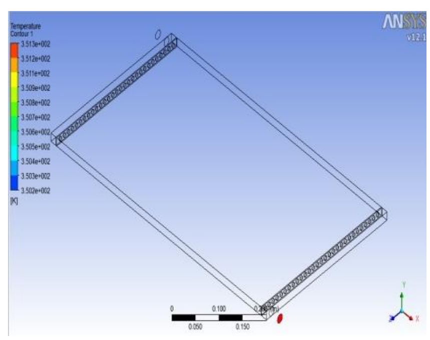
Results of Modification 1



Modification 2

Fluid: (Water + 50% Ethanol)

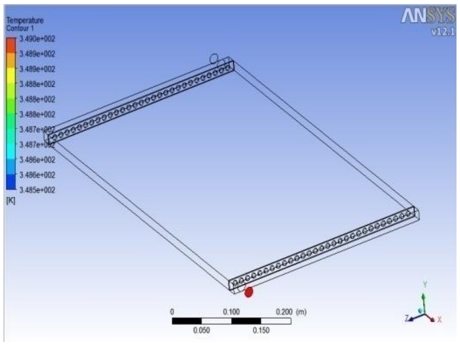
Results of Modification 2



Modification 3

Fluid:- Water + 30% Ethanol

Results of Modification 3

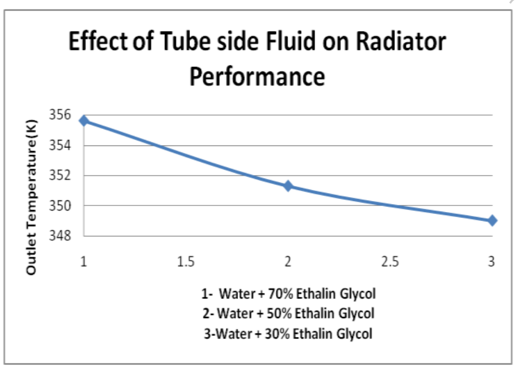


RESULT ANALYSIS

Comparison of Experimental Results with CFD Analysis Results:-

Sr No.	Experimental Results	CFD Analysis Results	% Variation
1	77.12	78.3	1.18

Table 2 Result comparison table



CONCLUSION

- CFD Analysis Results Fairly matches with the experimental Results which show that CFD Analysis is a good tool for avoiding costly and time consuming experimental work.
- Through experimental data that increases velocity ,heat transfers also increases.
- Percentage of Ethylonal Glycol increases as a coolant temperature is decreased.

REFERENCES

1. Salvio Chako,Dr.Biswadip Shome and Vinod Kumar "Numerical Simulation For Improving Radiator Efficiency", Engineering Automation Group, Tata Technologies Limited, Engineering research Center, Tata Motors Limited, India, 2003.
2. Mohd Yusoff Sulaiman, Shamsul Bahari Azraai, Wan Mokhtar "CFD Modelling of Air Flow Distribution from a Fan", University Teknical Malaysia Melaka, Malaysia, 2009.
3. A.Witry, M.H. Al-Hajeri and Ali A. Bondac, 2003, "CFD analysis of fluid flow and heat transfer in patterned roll bonded aluminium radiator", 3rd International conference on CFD, CSIRO, Melbourne, Australia, pp. 12-19.
4. S.N Sridhara,S.R.Shankalp and V.Umesh Babu "CFD Analysis of fluid flow and heat transfer in a single tube-fin arrangement of an automotive Radiator" International Conference on Mechanical Engineering 2005(ICME2005)28-30December 2005,Dhaka,Bangladesh.
5 C. Oliet, A.Oliva,J.Castro, " Parametric studies on automotive radiators" Applied Thermal Engineering 27 (2007) 2033–2043.