OPEN ACCESS

International
Journal
Of Modern Engineering Research (IJMER)

Performance Analysis of an After Cooler Used In High Power Engine Using CFD

M. Safraaj Salaamer[1], Dr. K. S. Amirthagadeswaran[2], R. K. Barathraj[3]

1 P.G.Scholar, Department of Mechanical Engineering, Government College of Technology, Coimbatore, India.
2 Professor, Department of Mechanical Engineering, Government College of Technology, Coimbatore, India.
3 P.G.Scholar, Department of Mechanical Engineering, Government College of Technology, Coimbatore, India.

Abstract: *After Cooler is a Heat Exchanger which is mainly used to cool air by using the coolant water based upon the principles of Heat transfer. Since considerable amount of charged air is been cooled in the After Cooler it is also called as Charged Air Cooler. This action of cooling is converse to the cooling in terms of heat absorbing and releasing fluids, found in other heat exchangers like radiators etc. As the demand grows the heat transfer required will also be increasing proportionately, which will surely result in some modifications in the currently used After Cooler. This study deals with the examination of the performance of the after cooler using CFD which will be the basement for optimization process in the examined software model to meet the growing demands.*

Keywords: *After Cooler, Growing Demands, CFD.*

I. Introduction

In India the most affordable mode of transport is the rail route transport. Due to its economical advantage its demands are also growing day by day, which will result in some required modifications in the sub systems of the rail. Among the various sub system of the rail, engine will have major effects due to the growing demands. In this Study a sub part of the engine called After Cooler is taken into consideration and its performance in the actual present conditions are studied using CFD. By using that examined software model, further optimization studies can be made on it to see how the After Cooler can be modified to meet the growing demands. This way of study will be very effective on the grounds of economy and time consumption. In this study the fin- tube arrangement of the After Cooler alone is taken for analysis in CFD kernel. This is done to complete the analysis with the system in hand. This is based on the reason that mesh count will be increasing as the number of components are increasing during analysis, which will require higher configuration system like work stations to complete the analysis. Moreover with reference to the knowledge gained about the fin-tube heat exchanger, the majority of heat transfer takes place in the fin-tube assembly. So it will be worthy to analyze the fin-tube assembly alone. In this study the fin-tube assembly is modeled using Solidworks 13, meshed using Hypermesh 10 and solved using Fluent 14.5, a component software package of the ANSYS 14.5. In this work the performance of the given After Cooler is analysed using Fluent 14.5 and this work is concluded by saying how optimization process can be carried out in the software model.

II. METHODOLOGY

As mentioned in the Introduction section, this work is carried out using software, so software analysis methodology is been adopted. In this section we are going to see how the software analysis is been carried out. The first step in the analysis is to model the fin-tube assembly using designing software, in this work Solidworks 13 is used to design and assemble the fin-tube assembly. Here in Solidworks the fin is separately modeled by using its dimensions length 680 mm, breath 145 mm and thickness 0.127 mm with 30 holes of 6.35 mm along the length wise, and 6 holes of same dimension along the breath wise. Similarly the tube is also separately modeled using Solidworks 13 for the dimensions of diameter 6.35 mm, thickness 0.508 mm and height of 444.40 mm and finally the assembling of the fin-tube arrangement is done for the specification of 240 fins occupying the entire vertical height of the tube with the distance between the fins as 1.6667 mm.

Fig 1 The Modeled Tube

This is designed by using circle sketch and extrudes command in the Solid Works 13 Kernel.

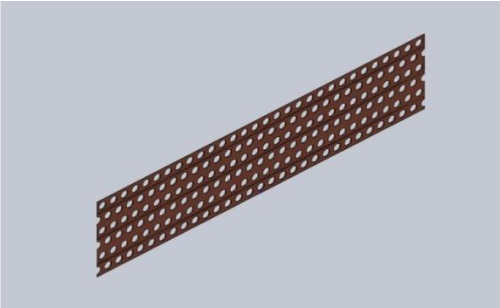

Fig 2 The Modeled Fin.

This is designed by using rectangular, triangular sketch and extrudes, extrude cut command in the Solid works 13 Kernel.

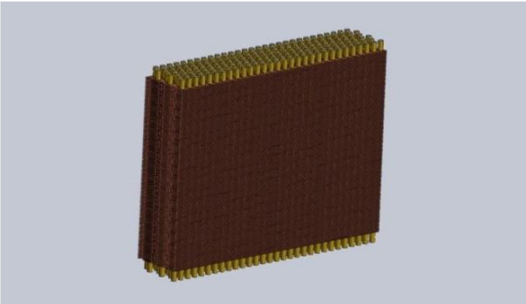

Fig 3 Fin-Tube Assembly

The assembling of the fin-tube arrangement is done by using the insert and pattering command in the Solidworks 13 kernel. To the point during the analysis the tandem arrangement of the fin-tube assembly is taken into consideration.

Next step in the Software analysis methodology is the meshing or pre-processing. In this study the Hypermesh 10 is used for meshing. Here in this the fin and tube are separately meshed by using 3D Hexahedral mesh and then they are assembled. Here in the meshing the mid section of the fin and the tube are taken and they are meshed by using 2D Hexahedral mesh and they are extruded as 3D Hexahedral mesh which is mainly done to reduce the time in meshing and also to achieve good quality mesh.

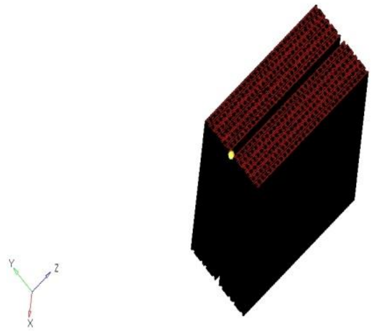

Fig 4 Air Volume Mesh

During this meshing the air side boundary conditions like air inlet (Perpendicular to Fin-Tube assembly) and air outlet (Behind the air inlet section) are also specified.

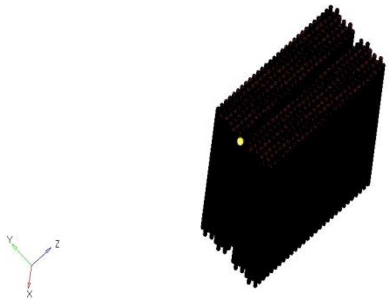

Fig 5 Water Volume Mesh

During this water volume mesh generation the water side boundary conditions like water inlet (Top side of the tube) and water outlet (Bottom side of the tube) are also specified.

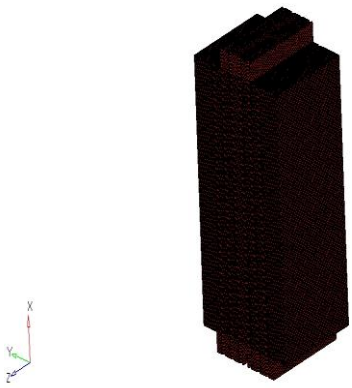

Fig 6 Combined Air & Water Volume Mesh

From the above meshed model we can infer a slightly extruded portion on both top and bottom side which is nothing but the meshing provisions provided to achieve four coolant passes.

The final step in this software methodology is the solving. Since in this After Cooler both the mass as well as the energy transfer takes place, so it would be more worthy that we adopt a Control Volume Software for analysis. Fluent is Control Volume software so it is adopted. More specifically Fluent 14.5 is adopted for analysis. Before analysis all input conditions are provided namely tube and fin materials as red brass and copper respectively, the tube and fin fluids as the water entering at 5.4 kg/s with the inlet temperature 79.5 Celsius and the air at 3.3375 kg/s with the inlet temperature of 177 Celsius. And to the point Standard K-Epsilon model is considered for demonstrating the turbulence.

To the point due to some system constrains half of the tandem fin-tube arrangement is taken and its input conditions are proportionately reduced and then the analysis process is carried out by ensuring that the Cross Counter flow is Demonstrated in the Four Coolant Pass condition.

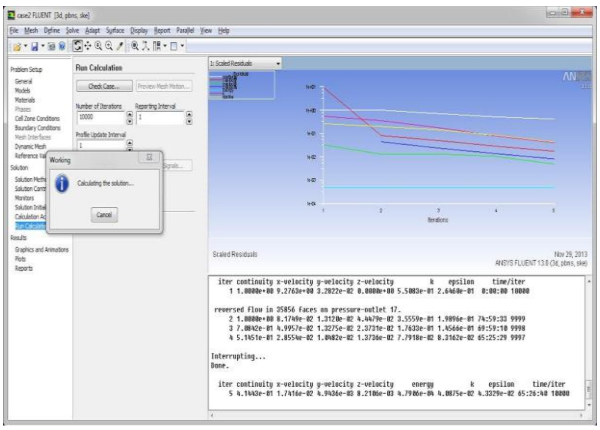

Fig 7 Screen Shot Fluent GUI during Analysis

III. Results and Discussion

As per the input conditions provided the iterations are preceded until all the curves indicating the iterations process are attaining the stable converging form. After that the Plots from the Kernel is taken and the results are studied. Since we are only dealing with the Thermal studies it is enough that we take only the temperature contour plot for our discussion.

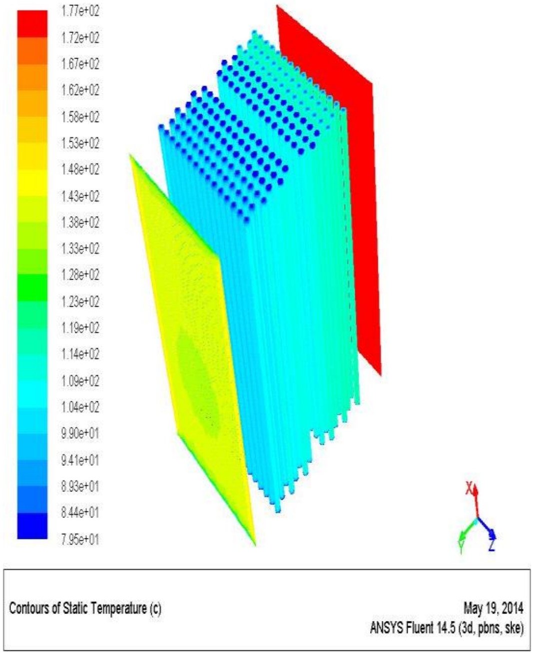

Fig 8 Temperature Contour for Air Side.

Form the Temperature Contour of the air side the outlet temperature of air is inferred as 145 Celsius. So this can be taken as the reference value and we can precede the optimization process with various combinations. Similarly the same can be done in water side also.

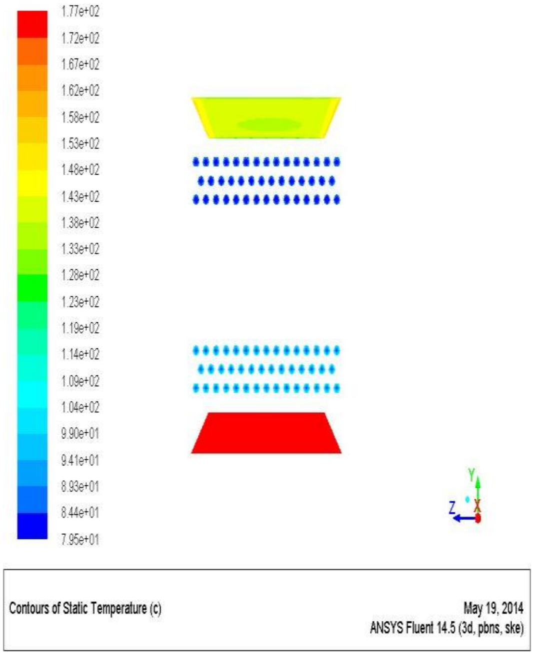

Fig 9 Temperature Contour for Water side.

From the Above Temperature Contour it is inferred that the outlet temperature of water is 87 Celsius. So this will remain the reference value for further optimization studies.

IV. Conclusion

From the CFD analysis of the After Cooler we have got the reference temperature of water and air outlet. So we can how proceed to the optimization process by using various combinations of the factors affecting the heat transfer rate of the fin-tube arrangement like material and geometry of the fin and tube assembly, mass flow rate of the fluids, prevailing conditions of the set up, Special arrangements to enhance the heat transfer rate etc can be used and the optimized model can be found by again doing all the above specified analysis for the various combinations provided by the optimization technique adopted. Finally the model with more reduced temperature on the air side and with considerable amount of increase in temperature in water side can be declared as the optimized model which can satisfy the growing demands.

REFERENCES

[1] G.Senthilkumar1,S.Ramachandran2,M.Purusothaman "Indigenous Development of Automobile Radiator using CFD" in 2010.
[2] D.Ganga Charyulu1,Gajendra Singh2,J.K.Sharam3 "Performance evaluation of a radiator in a disel engine-a case study" in 1999.
[3] A.Wirly,M.H.Al-Hajeri,Ali A Bondok "Thermal Performance of Automobile Plate Radiator" in 2005.
[4] D.K.Tafi and J.Ali "Fin-Tube Junction effects on the flow and heat transfer in flat tube corrugated Heat Exchangers" in 2002.