ISSN (Online) : 2454-4159
International Journal of Advanced Research in Science Management and Technology
Volume 1, Issue 5, October 2015

Experimental Investigation of Radiator System for a Stationary C.I. Engine

Mr. Iqbal Mansuri[1], Prof.V. H. Patil[2], Prof.Dr.A.A.Patil[3]

PG Student, Dept. Of Mechanical Engg., Godavari College of Engineering,Jalgaon, Maharashtra, India[1]

Head & Associate Professor,Of Mechanical Engg, Godavari College of Engineering, Jalgaon, Maharashtra, India[2]

Associate Professor,Of Mechanical Engg, Godavari College of Engineering, Jalgaon, Maharashtra, India[3]

***ABSTRACT*—** The effectiveness of the vehicle cooling framework emphatically relies on upon the wind current through the radiator centre. The course through the radiator centre thusly relies on upon different boards that are in the region of the radiator and these incorporate the radiator spread, grille, front inward board, cowl, floor, and so on. An unmistakable comprehension of the stream design inside the radiator spread is crucial for enhancing the radiator spread shape to expand the move through the radiator centre, in this way expanding the warm proficiency of the radiator.

The idea of condenser, fan, and radiator force train cooling module (CFRM) was further assessed by means of three-dimensional computational liquid elements (CFD) thinks about in the present paper for vehicle out of gear conditions. The investigation demonstrates that the CFRM arrangement was more inclined to the issue of front-end air re-flow as contrasted and the traditional condenser, radiator, and fan force train cooling module (CRFM). The upgraded front-end air re-flow prompts a higher air temperature going through the condenser. The higher air temperature, left unchanged, could render the vehicle cooling (AC) unit inadequate. The investigation likewise demonstrates that the front-end air re-course can be diminished with an included fixing between the CFRM bundle and the front of the vehicle, making the CFRM bundle worthy at the vehicle unmoving conditions.

KEYWORDS – Vehicle Cooling, Radiator centre, CFRM bundle,CFD,AC.

I. INTRODUCTION

In this paper, CFD empowered enhancement of wind stream dispersion inside the radiator spread is examined. Beginning from a CFD arrangement of the pattern plan that was accepted against indoor test information, a progression of advancement cases were executed to touch base at the ideal setup of the radiator spread. The wind current appropriation inside the radiator spread and the course through the radiator center was checked to give a quantitative premise to the advancement process.

The Cooling done by the radiator in the high stacked applications like in trucks and other overwhelming vehicles, assumes an essential part to choose the motor warm proficiency and outflows leaving motors. CFD Study is useful to comprehend the cooling instrument of the framework and to test the diverse design of radiator for powerful cooling as appeared in the accompanying figure 1. The CFD investigation was directed utilizing the business programming

FLUENT™, while the Surface and volume cross section were produced utilizing ANSA™ and TGRID™,respectively.Use of ANSA™ for surface lattice was instrumental in diminishing the CFD process duration CFD investigation of the standard plan that was approved against test information demonstrated that critical locales of re-coursing stream existed inside the radiator spread. This distribution decreased the course through the radiator centre, prompting a development of hot air stashes near the radiator surface and resulting debasement of radiator warm productivity. The CFD empowered streamlining prompted radiator spread arrangement that dispensed with these distribution districts and expanded the move through the radiator centre by 34%. It is foreseen that this increment in radiator centre stream would altogether expand the radiator warm effectively.

II. LITERATURE SURVEY

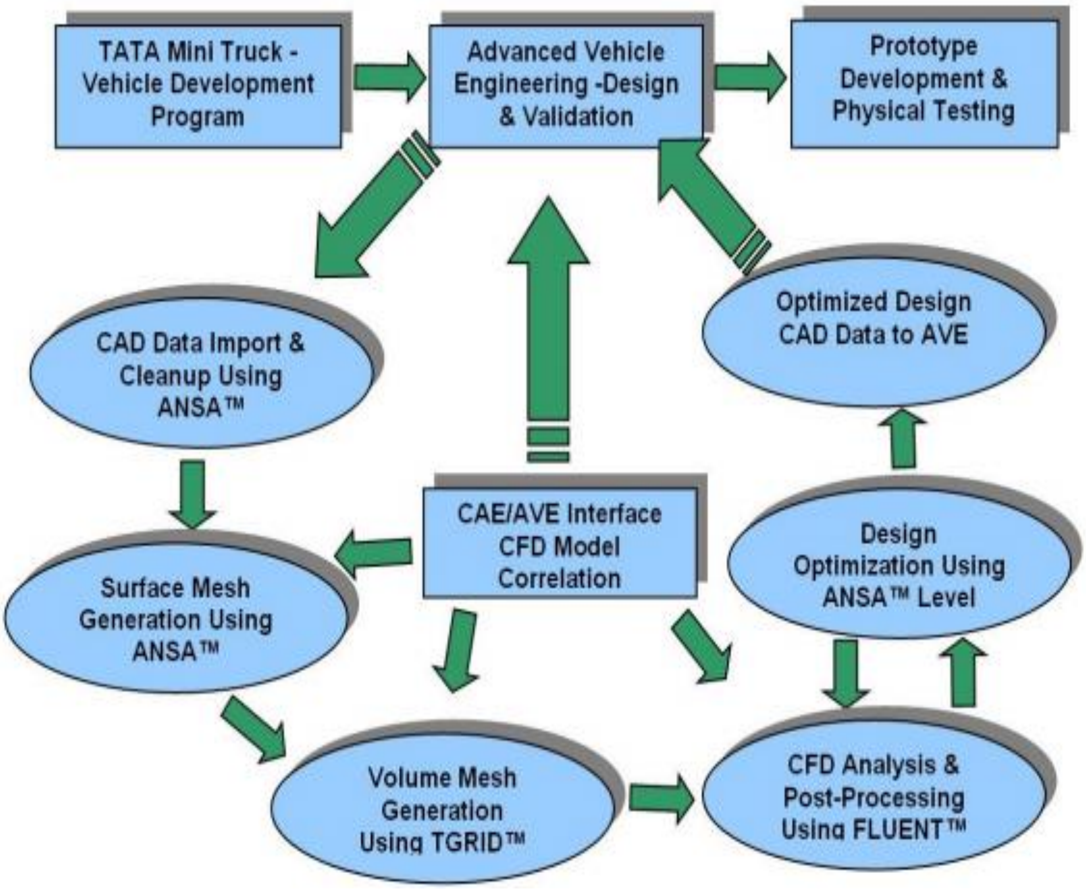

Fig.1 Process for the Vehicle development program of the Radiator

The wind current dissemination inside the radiator spread is intricate because of the mind boggling state of the spread. The wind stream through the motor compartment experiences the accompanying parts: grille, radiator, fan and cover leaving the compartment through the floor openings. Each segment has an impact on the course through the primary radiator; some of them direct others by implication. As the stream and the communications are mind boggling and the utilization of CFD is moderately new, every connection must be considered, measured and re-enacted independently.

The Numerical experimentation of the work conveyed out included figuring out of a business car radiator for the equired liquid area,

discretising the liquid area, recreation of the liquid stream furthermore, warmth exchange at unfaltering state and post handling the results and making suitable determinations. The radiator of a financially existing vehicle is decided for the examination to get the reasonableness to the

study. The points of interest of the geometry of the radiator were gotten by the procedure of figuring out. The measurements of individual parts of the radiator

were measured utilizing suitable measuring instruments (see Table 1). The estimations got were utilized to create the CAD model in CATIA V5 R9 (see Fig. 1).

Table 1: Specifications of the radiator chosen

Radiator type	Cross flow, single row core, forced air cooled radiator
In hose barb (inlet)	6.81mm
Out hose barb (outlet)	6.81mm
Core rows	35
Core tube	1.84mm
Fin density	1.5mm/fin
Core dimensions (L x H x W)	510mm x 395mm x 40mm
Tube and fin material	Aluminium
End tube material	Nylon 6,6

Discretisation of the liquid area the geometric closeness between the lines of tube what's more, blade helps us in constraining the computational area to a single tube and connecting blade game plan as appeared in Fig. 2. Thus, the liquid area is made for a solitary blade what's more, tube get together and numerical investigation is done. Fig 2. Liquid area of the single tube blade game plan the liquid area incorporates the wind current volume and the coolant stream volume. The issue is illuminated as a conjugate warmth exchange requiring the thickness of the tube what's more, balance likewise to be demonstrated. The surfaces noticeable all around space, coolant area, tube thickness and balance thickness are discretised with differing network thickness in agreement to the material science of liquid stream what's more, warmth exchange. Denser cross section size is utilized at basic volumes in the liquid stream and warmth exchange area. The framework freedom investigation of the recreation is done to touch base at the base number of components required to keep up the required security and exactness in the calculation. The settled component tally and the related perspectives are recorded in Table 1. The fit geometry for a single tube and blade is then reached out to the whole 32 balance get together of the radiator.

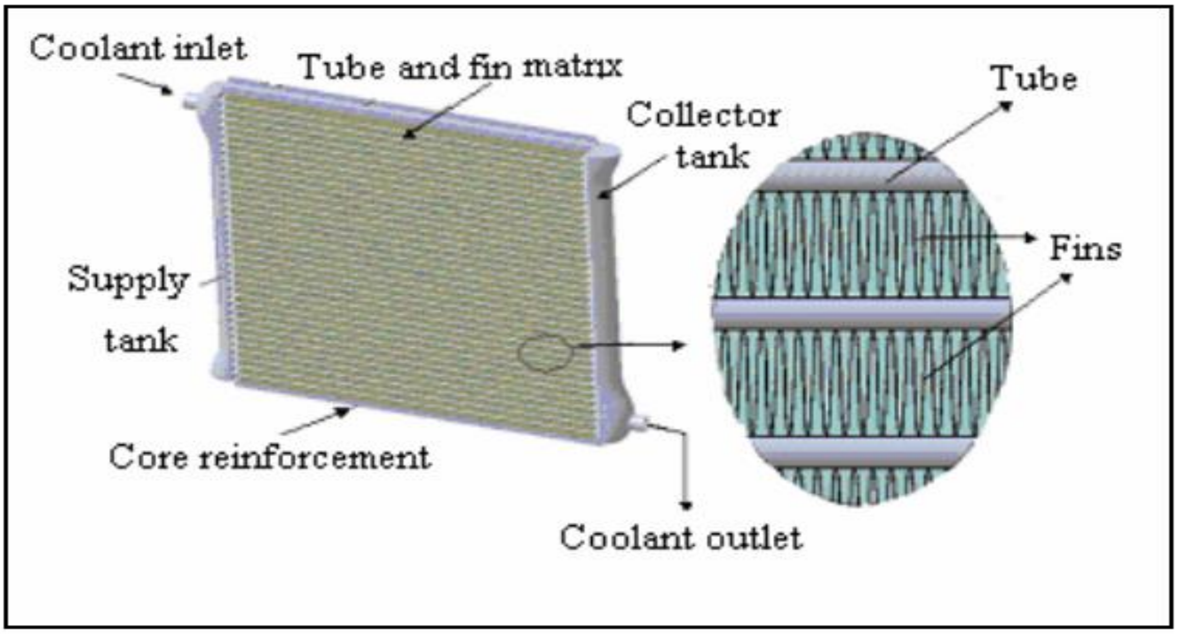

Fig 2. CATIA model of Radiator

III. PROBLEM DEFINITION

CFD Model description:

The above analysis will be performed using CFD as design tool and optimization for the different configurations. The model include

1. Steady Turbulent Flow
2. Incompressible Fluid
3. Working Medium is Dry Air
4. Standard Physical Properties Corresponding to 34 °C
5. Turbulence Modeled by High Reynolds Number k-e Model
6. Standard Model Constants
7. Wall Functions Employed at Walls
8. Second Order Upwind Differencing Scheme for All Variables
9. SIMPLE Algorithm for Pressure – Velocity Coupling
10. Simplified Fan Model
11. Fan Suction Simulated by Pressure Jump

ISSN (Online) : 2454-4159
International Journal of Advanced Research in Science Management and Technology
Volume 1, Issue 5, October 2015

12. Interior Details of Radiator Core Neglected
13. Pressure Drop/Flow Resistance Simulated by Porous Media
14. Radiator Heat Generation Neglected
15. The radiator cover walls are assumed to be thermally insulated and thus adiabatic.

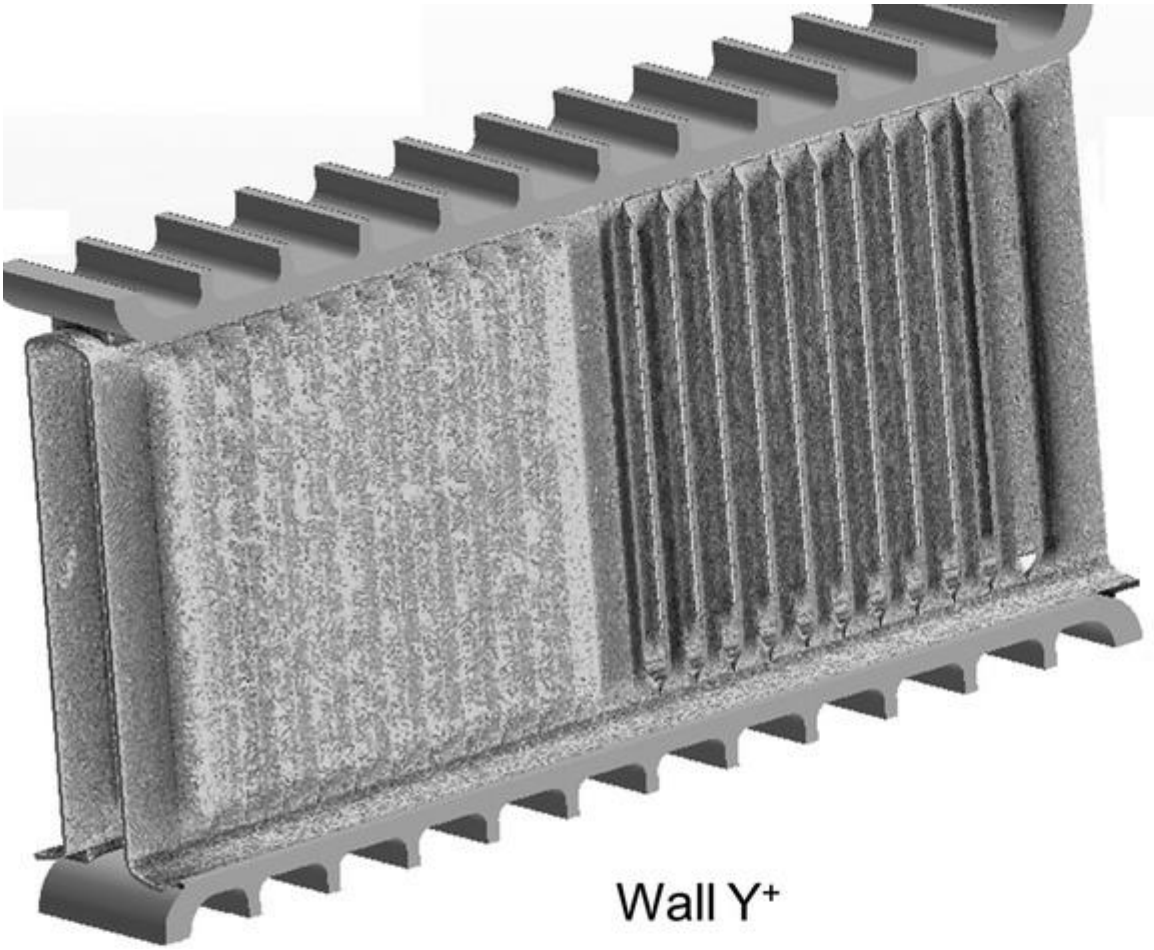

Fig. 2 Basic Configuration of the Radiator

IV. PROPOSED SOLUTION

CFD based optimization of airflow distribution inside the radiator cover will be discussed. CFD solution of the baseline design will be presented. Forced convection will be considered as the cooling mode Optimization cases were executed to arrive at the optimum configuration of the radiator cover. The airflow distribution inside the radiator cover and the flow through the radiator core was monitored to provide a quantitative basis for the optimization process.

Temperatures on the particular dissemination is talked about. The air delta speed (Vai) is decided to be 4.4m/s air at surrounding temperature [4,5]. With reference to the particulars of the coolant pump utilized for the vehicle picked, coolant delta speed (Vci) is 0.0063 kg/s in each tube. The properties of air and coolant were characterized for standard conditions and kept consistent all through the examination. The solver decided for the examination is isolated, certain, 3D, consistent state solver in FLUENT 6.1. The isolated solver has been utilized for incompressible and somewhat compressible streams by numerous agents [3,4] what's more, has been appeared to delineate the outcomes with better precision. It unravels the vitality and stream mathematical statements consecutively. The progression, force and vitality mathematical statements (comparisons not appeared in this paper for their sweeping statement) of liquid stream are unraveled during the time spent getting temperature profiles. The isolated methodology fathoms for a solitary variable field (e.g., weight, p) by considering all phones in the meantime. It then unravels for the following variable field by again considering all phones at the same time, thus on [6]. This methodology is well recorded in writing and ordinarily embraced by the researchers in complex issues, for example, conjugate warmth move managed in this paper. Standard k-ε model is picked to represent turbulent stream. Strength, economy, and sensible precision for an extensive variety of turbulent streams also, warmth exchange recreations are the explanations behind picking this model

V. EXPECTED RESULTS

CFD analysis has reduced the cost, time in design and development of radiator as compared to conventional methods. It also reduces the need of prototype during design process while we do iterations to get optimized design. Now we need to make prototype of the optimized design only for physical testing. In past years we have seen very increasing trend in the use of CFD in many fields over worldwide and in India also. Its effect is reaching in Indian universities at very hopeful rate and scholars are showing very interest in it. Today in India over 500 scholars are doing work regarding CFD and numbers are increasing. However the need of high processing capabilities computer and generation of codes is still a big problem in India. Students may have good grip on the required mathematics and fluid mechanics concepts but they lack knowledge regarding computer languages to generate codes. So we feel before increasing scope in CFD at graduation course, first we should focus on computer programming course in engineering colleges.

VI. CONCLUSION

The liquid stream and warmth exchange examination of a solitary tube-balance course of action of a car radiator is effectively completed utilizing numerical reenactment assembled in business programming FLUENT. The varieties in the weight, temperature and speed toward coolant stream and wind current are exhibited and investigated. It is watched that the temperature of coolant drops by 6 K and a weight drop of 52.3 Pa in the coolant. The air that assimilates the warmth because of constrained convection picks up an in increment in temperature by 9.5 K. The study shapes an establishment for the liquid stream investigation of a car radiator. With the computational time and assets accessible, the outcomes acquired were observed to be tasteful. In any case, a proceeded with study in different perspectives towards a superior outline of the radiator is proposed as demonstrated as follows.

1. To represent the variety of the delta conditions with time as in useful cases, transient examination should be possible.
2. Optimizing the estimations of the stream rates and the measurements of the radiator for a given force generating so as to rate of the vehicle, CFD cerements of the radiator for a given force generating so as to rate of the vehicle, CFD codes.

REFERENCES

[1] Zhigang Yang, Jeffrey Bozeman and Fred Z. Shen, James A. Acre, "CFRM Concept at Vehicle Idle Conditions", SAE-2003-01-0613.

[2] Yang, Z., Bozeman, J., Shen, F.Z., Turner, D., Vemuri, S., and Acre, J., "CFRM concept for vehicle thermal systems", SAE-01-1207, 2002.

[3] ANSA version 11.x User's Guide, BETA CAE Systems S.A., November 2002

[4] Sridhar Maddipatla, Coupling of CFD and Shape Optimization for Radiator Design.

[5] Fluent 6.1 User's Guide, Fluent Inc 2003-01-25. [6] Tgrid 3.6.8 Documentation, Fluent Inc 2003-01-25.

[6] Hilde Van Der Vyer, Jaco Dirker and Jousoa P Meyer, 2003, "Validation of a CFD model of a three dimensional tube-in-tube heat exchanger", Third International Conference on CFD in the Minerals and Process Industry, CSIRO, Melborne, Australia. pp. 25-32.

[7] 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, Melborne, Australia, pp. 12-19.

[8] J A Chen, D F Wang and L Z Zheng, 2001, "Experimental study of operating performance of a tube-and-fin radiator for vehicles", Proceedings of Institution of Mechanical Engineers, Republic of China, 215: pp. 2-8.

[9] Sridhar Maddipatla, 2001, "Coupling of CFD and shape optimization for radiator design", Oakland University. Ph.D. thesis.

[10] Changhua Lin and Jeffrey Saunders, 2000, "The Effect of Changes in Ambient and Coolant Radiator Inlet Temperatures and Coolant Flowrate on Specific Dissipation", SAE Technical Papers, 2000-01-0579.

[11] Fluent company, 2000, FLUENT Help manual. 7. J.P.Holman, 2002, Heat transfer, Tata-McGraw-Hill Publications.