Numerical Simulation for Improving Radiator Efficiency by Air Flow Optimization

Salvio Chacko[1], Dr. Biswadip Shome[1], and Vinod Kumar[1]
A.K. Agarwal [2], D.R. Katkar [2]
[1]Engineering Automation Group, Tata Technologies Limited, Pune, India
[2]Engineering Research Center, Tata Motors Limited, Pune, India

ABSTRACT

The efficiency of the vehicle cooling system strongly depends on the air flow through the radiator core. The flow through the radiator core in turn depends on other panels that are in the vicinity of the radiator and these include the radiator cover, grille, front inner panel, cowl, floor, etc. A clear understanding of the flow pattern inside the radiator cover is essential for optimizing the radiator cover shape to increase the flow through the radiator core, thereby increasing the thermal efficiency of the radiator.

In this paper, CFD enabled optimization of airflow distribution inside the radiator cover is discussed. Starting from a CFD solution of the baseline design that was validated against indoor test data, a series of 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.

The CFD analysis was conducted using the commercial software FLUENT™, while the surface and volume mesh were generated using ANSA™ and TGRID™, respectively. Use of ANSA™ for surface meshing was instrumental in reducing the CFD cycle time.

CFD analysis of the baseline design that was validated against test data showed that significant regions of re-circulating flow existed inside the radiator cover. This recirculation reduced the flow through the radiator core, leading to a build-up of hot air pockets close to the radiator surface and subsequent degradation of radiator thermal efficiency. The CFD enabled optimization led to radiator cover configuration that eliminated these recirculation regions and increased the flow through the radiator core by 34%. It is anticipated that this increase in radiator core flow would significantly increase the radiator thermal efficiency.

Keywords: Radiator, Shape Optimization, Numerical Simulations, CFD

1. INTRODUCTION

Increasing demands on engine power and performance to meet ever increasing customer requirements to meet increased load carrying capacity for trucks have necessitated to improve the heat management system of the vehicle. Manufacturers of commercial vehicles are facing a substantial increase of heat release into the cooling system. The main sources for this increase are: more stringent emissions leading to new combustion technologies and increased power of the engines. The total increase in cooling requirement may be up to 20% over the current level. At the same time the noise levels have to decrease and the fuel economy have to be improved. This forces the manufacturers to think about new concepts and optimized efficiency of the cooling system.

A critical requirement of vehicle design is adequate airflow through the radiator core to ensure adequate engine cooling under all operating conditions. Therefore, it is important to optimize the airflow through a radiator in the design and development stage of the vehicle engine cooling system. The final evaluation of the system is usually based on prototype building and on-road testing. The goal in any vehicle development program is to reduce the concept-to-production cycle time and the number of prototype vehicles and still meet the design targets. Given the high cost of building the prototypes, it is becoming increasingly necessary to develop a set of validated numerical simulation tools and techniques that could reduce prototype testing, thereby leading to reduction in vehicle design cost and cycle time. Achieving this, within the time and cost constraints of a modern vehicle development program, places increasing reliance on Computational Fluid Dynamics (CFD) techniques. In this study, a rigorously validated CFD model was used to optimize the airflow distribution inside the radiator enclosure to enhance the cooling performance of the radiator.

2. NUMERICAL MODELS

An advanced in-depth study was conducted for the cooling system performance of TATA Mini truck. The goal was to develop CFD techniques that would enable optimization of the flow through the cooling system early during the product design cycle. A representative process for the vehicle development program for TATA Mini Truck, which encompasses both testing and CFD analysis is as shown in the Figure - 2.1.

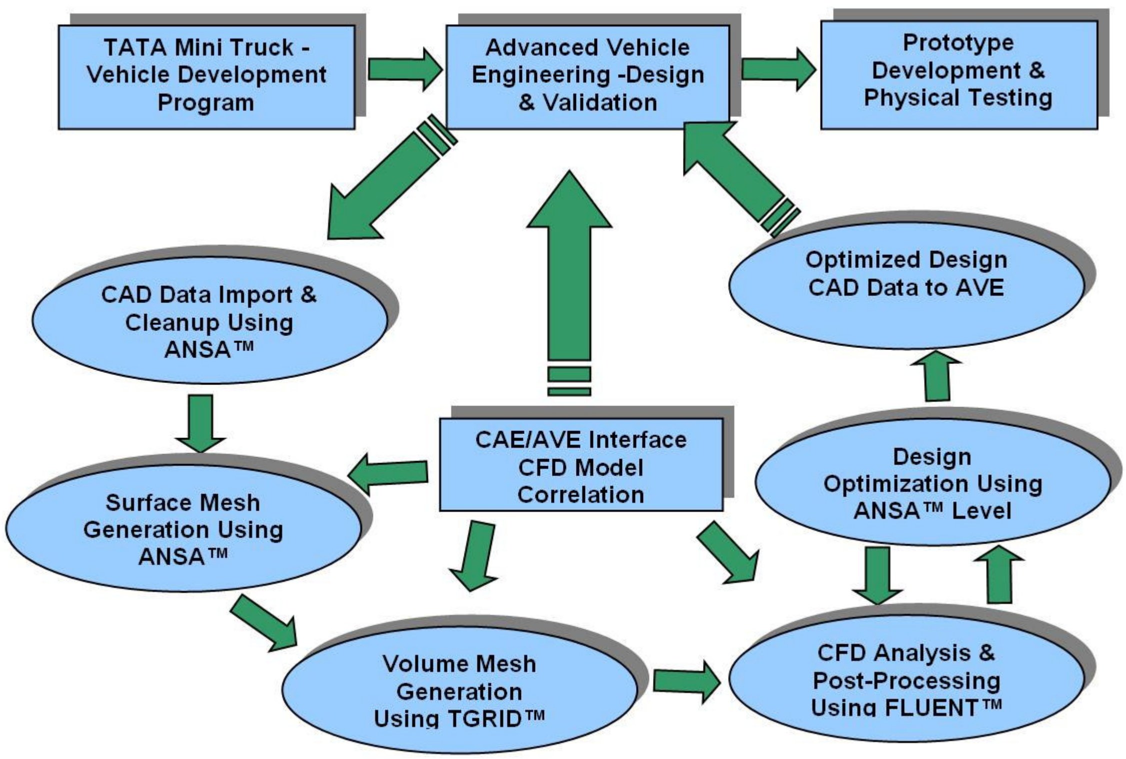


Figure - 2.1: Process for the Vehicle Development Program of TATA Mini Truck

2.1 OBJECTIVE

The objective of this study is to optimize the airflow distribution inside the radiator cover - front inner panel of TATA Mini Truck in order to increase the thermal efficiency of the radiator and that of the engine. Starting from the CFD model of the baseline model that was extensively validated against test data, CFD enabled optimization of the radiator cover shape was carried out to arrive at an optimum configuration that minimizes recirculation regions inside the cover and maximized the airflow through the radiator core.

2.2 PHYSICAL MODEL SETUP

The TATA Mini truck radiator assembly consists of the radiator located behind the grill with a fan mounted downstream of the radiator. The radiator-fan assembly sits inside the radiator cover. The fan draws air from the outside ambient through the grill and subsequently draws it through the radiator where it picks up the heat from the radiator, and finally the air is forced out through the floor-facing opening of the radiator cover.

The airflow distribution inside the radiator cover is complex due to the complex shape of the cover. The airflow through the engine compartment goes through the following components: grille, radiator, fan and shroud leaving the compartment through the floor openings. Every component has an influence on the flow through the main radiator; some of them direct others indirectly. As the flow and the interactions are complex and the use of CFD is relatively new, every relation has to be studied, measured and simulated separately.

Experimental Setup Conditions

- ✓ Vehicle at Rest
- ✓ Ambient Pressure & Temperature Corresponding to Test Condition
  - ➢ Pressure = 101325 Pa (Atmospheric)
  - ➢ Temperature = 34 ℃ (Room Temperature During Testing)
- ✓ Rated Fan Flow rate

The experimental program was done on a suitably modified radiator test facility, which was capable of dealing with light & large truck radiators alike. Figure - 2.2 shows the schematic of the radiator Assembly. The instrumentation was in addition to the test rig. On this facility the microprobes & anemometers were installed in the radiator, which measured flow rates at the radiator locations as shown in the Figure - 2.11. Against every location indicated by the bubble, the flow rates were measured, tabulated and compared with the data from CFD.

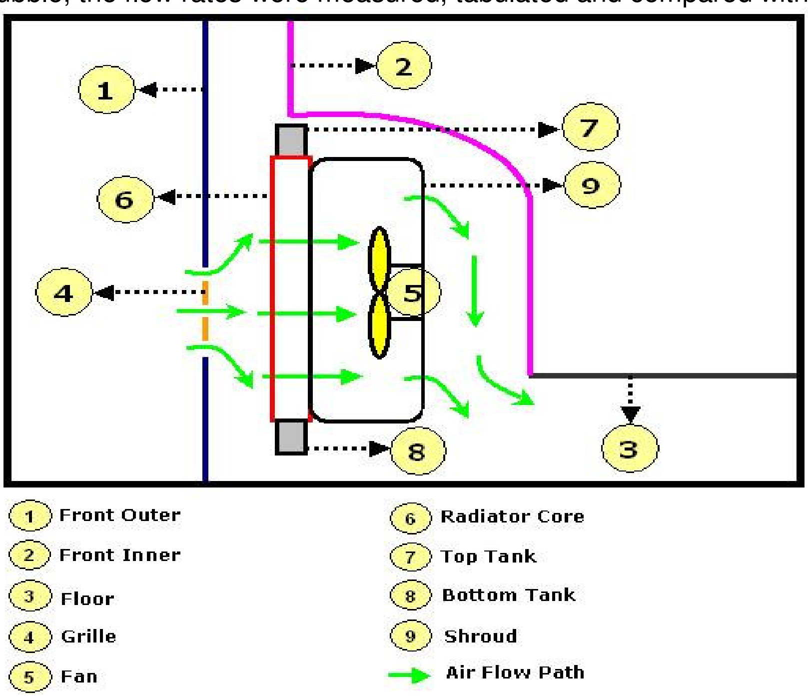


Figure - 2.2 Schematic of Radiator Assembly.

2.3 Numerical Model Setup

The commercial CFD Code Fluent**™** was used along with TGRID**™** & ANSA**™** as preprocessor for solving the flow physics. The computational Domain had to be represented as a finite volume model. The preprocessing activity involved Cleaning up and surface meshing of the CAD data, which was done in ANSA™. The process is defined as shown in Figure-2.1 the Road map for vehicle development program of a TATA Mini Truck.

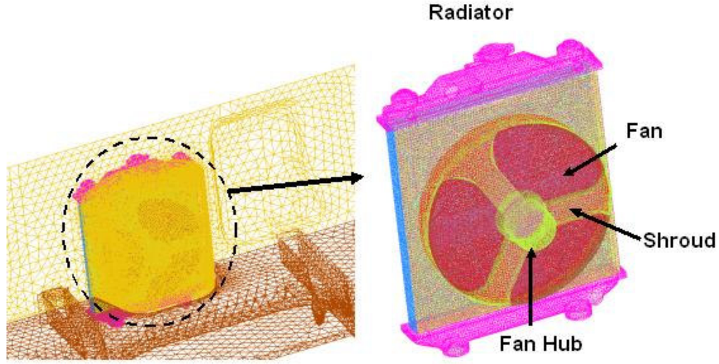


Figure – 2.3 Surface Mesh Generated By ANSA

The use of tools like Auto Meshing & Morphing had helped us to reduce the cycle time for building the CFD Model in ANSA™. During the design optimization phase, a total of 4 iterations were done in quick time after the model had been validated with test data.

The surface Mesh data had to be taken into TGRID for the grid generation. After the model was build, the problem setup process essentially consists of importing the TGRID™ mesh into FLUENT™, putting appropriate scaling factors for scaling the mesh/geometry, and setting up the appropriate turbulence model, fluid properties, solver controls, convergence monitors, etc.

All simulations were performed using the Standard variant of the k-ε turbulence model. For stability of the solution the linear upwind scheme was used in the beginning, once stabilized a second order upwind differencing scheme was used.

2.4 CFD Model Assumptions:

- ✓ Steady Turbulent Flow
- ✓ Incompressible Fluid
- ✓ Working Medium is Dry Air
  - ➢ Standard Physical Properties Corresponding to 34 °C
- ✓ Turbulence Modeled by High Reynolds Number k-e Model
  - ➢ Standard Model Constants
  - ➢ Wall Functions Employed at Walls
- ✓ Second Order Upwind Differencing Scheme for All Variables
- ✓ SIMPLE Algorithm for Pressure – Velocity Coupling
- ✓ Simplified Fan Model
  - ➢ Fan Suction Simulated by Pressure Jump
- ✓ Interior Details of Radiator Core Neglected
  - ➢ Pressure Drop/Flow Resistance Simulated by Porous Media
  - ➢ Radiator Heat Generation Neglected
- ✓ The radiator cover walls are assumed to be thermally insulated and thus adiabatic.

Model Setup

A Segregated solver was used with an implicit formulation. The problem was solved for steady state with absolute velocity formulation and cell based gradient option with porous medium formulation being physical velocity.

Material Property Setup

Since the problem involves air at 34°C, the material properties were calculated to correspond to air at 34°C. As the flow was incompressible, the density was set as constant.
These air properties are calculated as using the following equations:

Density

$$\rho = \frac{P}{(287.0856)\,(T + 273.15)}$$

Where ρ is air density in Kg/m^3, P is 101325 Pa (atmospheric pressure), and T is air temperature in °C.

Viscosity

$$\mu = 1.8402 \times 10^{-5} \left(\frac{T + 273.15}{298} \right)^{0.78}$$

Where μ is air viscosity in Kg/m-s and T is air temperature in C.

Inlet and Outlet Boundary Conditions

Inlet - Mass Flow Inlet
Outlet - Pressure Outlet

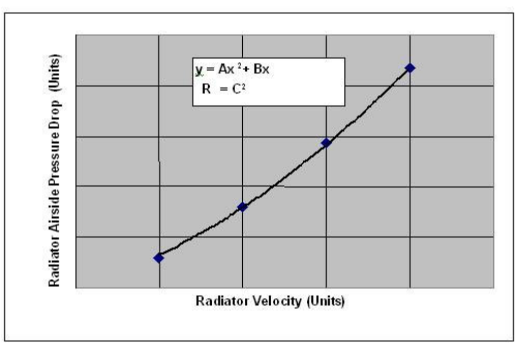
Porous Medium Setup

The radiator was modeled as porous medium to account for the pressure drop inside the radiator. Refer Figure - 2.4.

Calculation of Porous Media Constants

The step for calculating the viscous and inertial resistance constants are:

- Tabulate Pressure Drop in Pa and Velocity in m/s
- Plot Pressure Drop (ΔP) as function of Velocity (U)
- Fit a curve such that $\Delta P = AU^2 + BU$
- Viscous Resistance Constant $= \frac{B}{\mu}$, Inertial Resistance Constant $= \frac{2A}{\rho}$



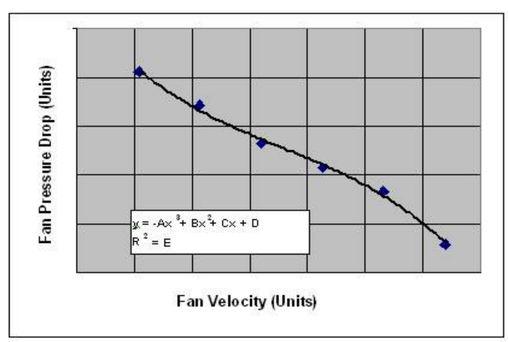


Figure – 2.4 Pressure Drop characteristics for Radiator & Fan

Fan Boundary Conditions
Pressure Jump = Polynomial (3rd Order),
ΔP = C1 + C2 U + C3 U^2 + C4 U^3, Here ΔP is pressure rise and U is fan velocity

Steps for Generating Polynomial for Fan Pressure Jump

- Tabulate Fan Velocity (in m/s) and Fan Pressure Rise (in Pa)

- Plot Pressure Rise (ΔP) as a function of Fan velocity (U)
- Curve Fit such that $\Delta P = C1 + C2\ U + C3\ U^2 + C4\ U^3$

The boundary conditions are set, the solutions are initialized, and the residuals are set, monitored to check for convergence. A good practice is also to check for mass balance. Computing the difference between the inlet and outlet mass flow rates can perform a mass-balance check. The difference in mass flows should be less than 1.0E-6.

Base Case Observations

On post processing the CFD Results for the Base Case the following were the observations. Refer Figure – 2.5.

- ✓ **Presence of Re-circulation Regions**
  - In the space between the radiator & front inner panel, there were high velocity air moving back into the around the radiator package thus reducing the radiator's efficiency.
- ✓ **Presence of Leakage paths**
  - Leakage paths around the radiator, which would reduce the radiator's efficiency was identified. In one of the subsequent optimization cases performed on fluent, additional sealing strips were added around the radiator pack and effect simulated in Fluent.

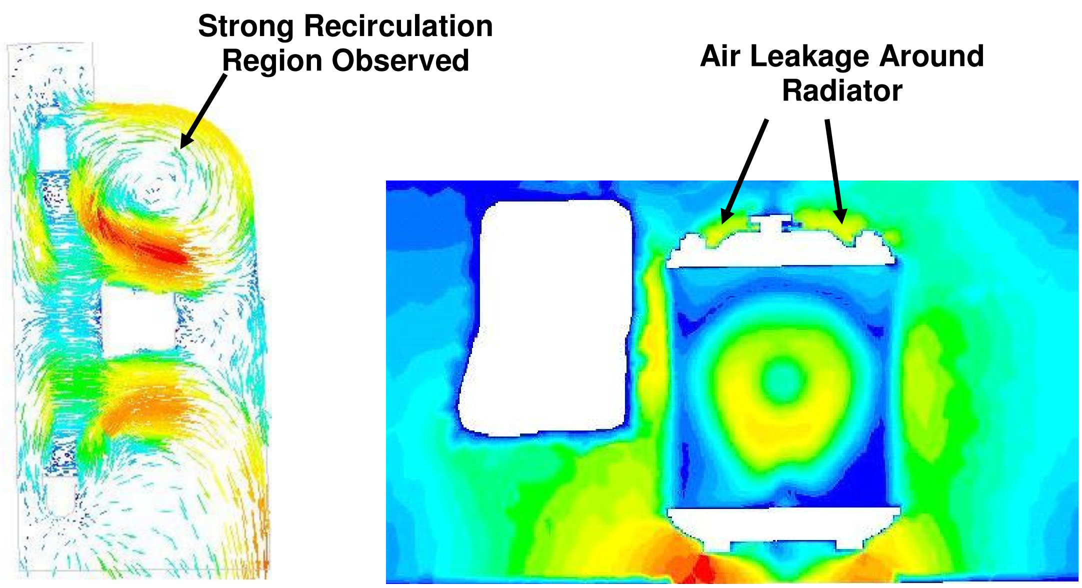


Figure - 2.5 Base Case Air Flow Distributions across Radiator Cover Center Line

Based on the following observations four case models as tabulated below, were build for Flow optimization. The First three cases dealt with streamlining the front inner panel to increase flow through the radiator core and reducing the re-circulation zones where-as the optimized shape of front inner panel and sealing strips around the radiator pack in fourth case reduced the re-circulation zones as well as the leakage paths.

Case Number	Description
CASE 1	Radiator Cover Streamlining
CASE 2	Radiator Cover Streamlining
CASE 3	Radiator Cover Streamlining
CASE 4	Radiator Cover Streamlining & Sealing strips for reducing Leakage

Case Wise Conclusions

CASE 1

Optimized radiator cover case 1 Conclusions. Refer Figure – 2.6.

- Flow Through Radiator Core Improved by 26%
- Re-circulation Region Inside Radiator Cover Significantly Reduced

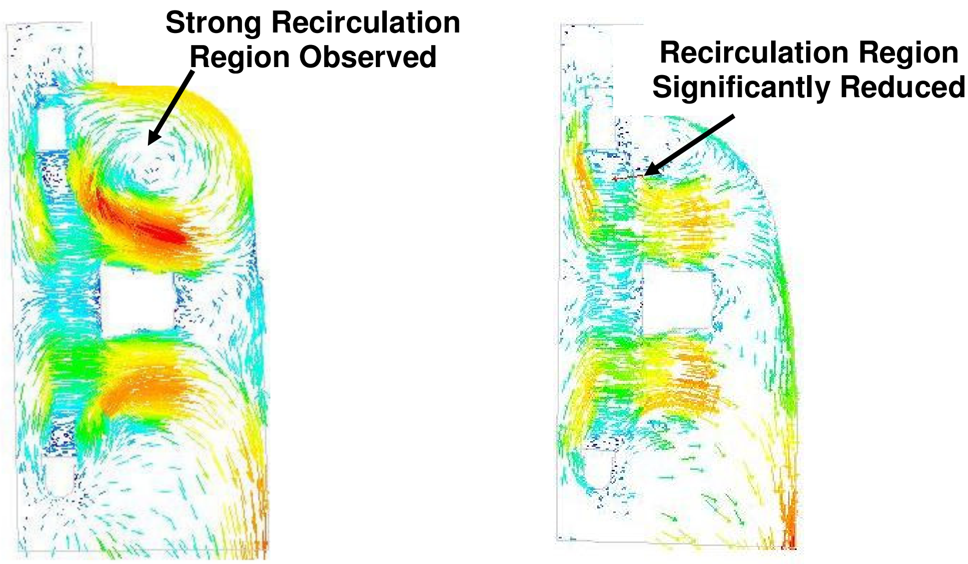


Figure - 2.6 Base Case Versus Case 1 - Air Flow across Radiator Cover Center Line

CASE 2

Optimized radiator cover case 2 Conclusions. Refer Figure – 2.7.

- No Improvement in Flow Through Radiator Core
- Small Re-circulation Region Inside Radiator Cover Still Present

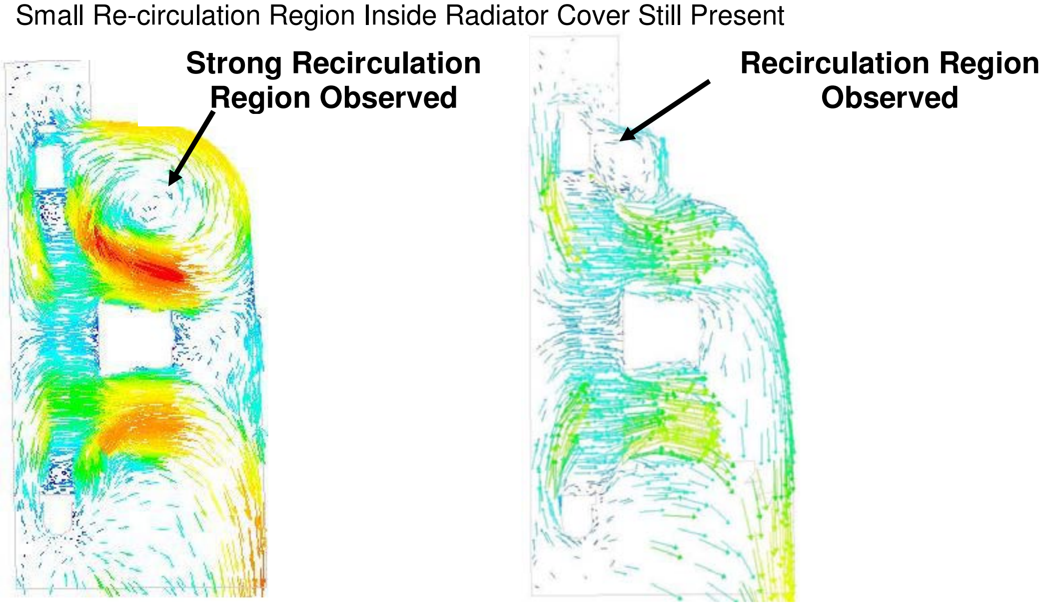


Figure - 2.7 Base Case Versus Case 2 - Air Flow across Radiator Cover Center Line

CASE 3

Optimized radiator cover case 3 Conclusions. Refer Figure – 2.8.

- Flow Through Radiator Core Improved by 32%
- Re-circulation Inside Radiator Cover Significantly Reduced/Eliminated

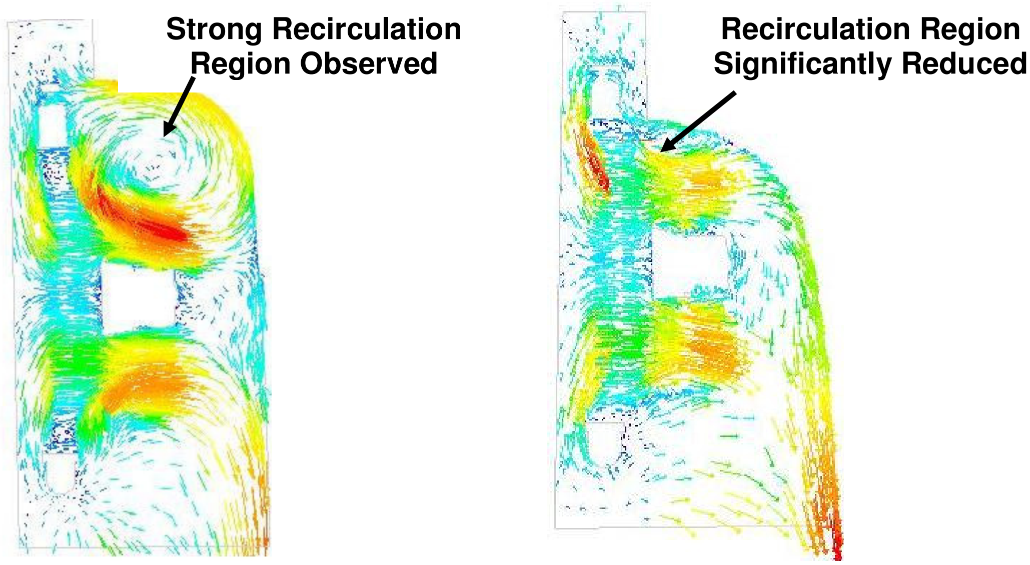


Figure - 2.8 Base Case Versus Case3 - Air Flow across Radiator Cover Center Line

CASE 4

Optimized radiator cover case 4 Conclusions. Refer Figure – 2.9, 2.10, 2.11, 2.12.

- Flow through Radiator Core Improved by 34%
- Re-circulation Inside Radiator Cover Almost Eliminated
- Leakage paths around the radiator Eliminated

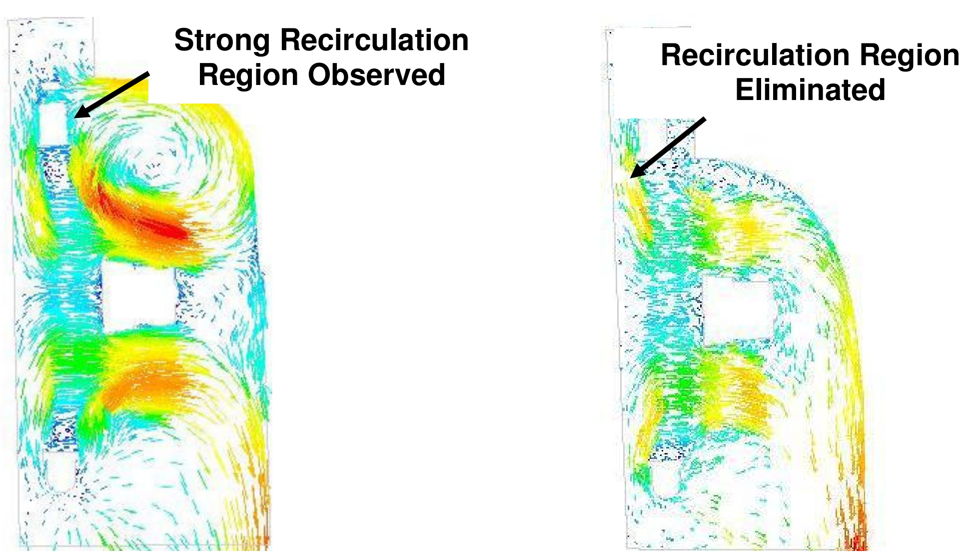


Figure - 2.9 Base Case Versus Case4 - Air Flow across Radiator Cover Center Line

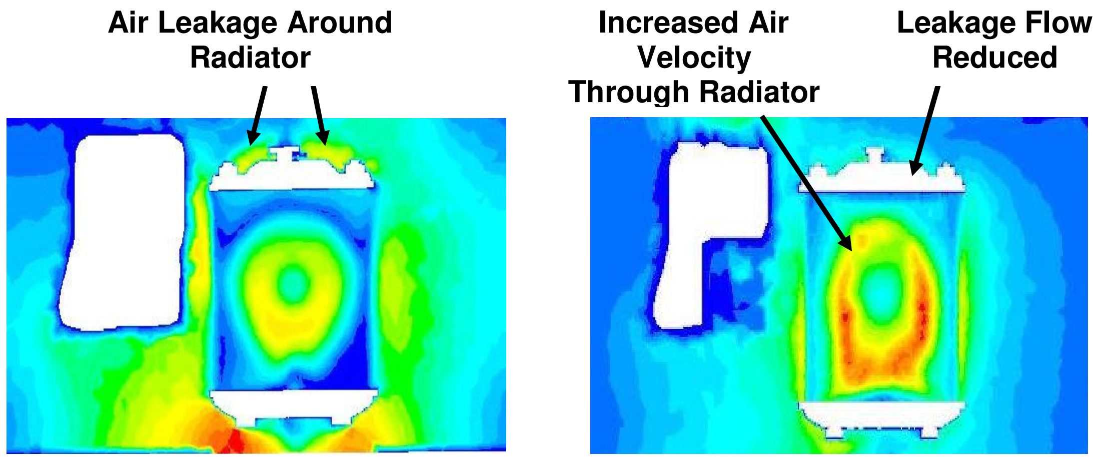


Figure - 2.10 Base Case Versus Case4 - Air Flow across Radiator face

Figure - 2.11 Velocity Measured Locations on Radiator Face

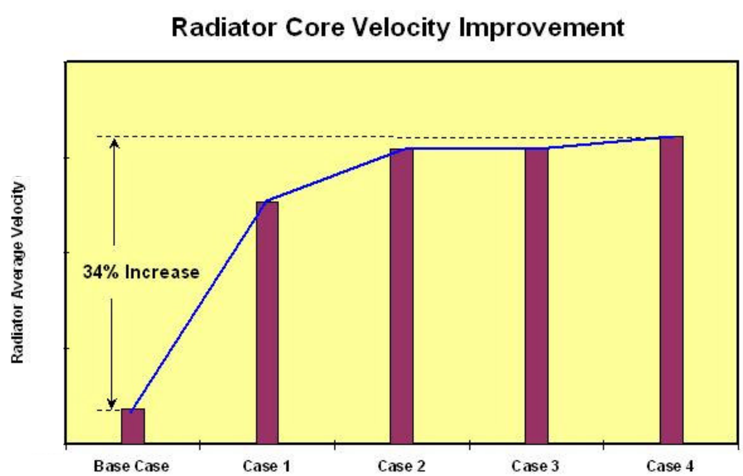


Figure - 2.12 Percentage Improvement Graph for Various cases

3.0 CONCLUSION

CFD enabled optimization of TATA Mini Truck radiator cover was carried out. Starting from a validation solution of the baseline design, optimization of the radiator cover was carried out using four design iterations. The analysis of the final optimized configuration shows elimination of recirculation zones and an increase of flow through the radiator core by 34%.

This study shows that a complex three-dimensional CFD simulation of the flow through the front-end of a truck is possible under a variety of conditions. The accompanying validations have shown that the results are accurate for realistic operating conditions. This opens the door to the possibility to optimize the cooling system design early in the development. The next step is to incorporate heat transfer effects in the radiator and a tighter coupling of the CFD simulation with heat transfer models.

ACKNOWLEDGEMENTS

The authors would like to thank the senior leadership at Tata Technologies Ltd. and Tata Motors Ltd. for giving permission to publish this paper. Special thanks to the testing and validation team at Tata Motors Ltd., for providing the performance test data results for CFD validation purposes.

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 .