CFD Prediction to Optimize Front End Cooling Module of a Passenger Vehicle

Biswadip Shome
Tata Technologies Limited

Vinod Kumar
Tata Technologies Limited

S. V. Ranganath Kumar
Tata Technologies Limited

Gyan Arora
Tata Technologies Limited

CFD PREDICTION TO OPTIMIZE FRONT END COOLING MODULE OF A PASSENGER VEHICLE

Dr. Biswadip Shome*, Vinod Kumar[2], S.V. Ranganath Kumar[2], Dr. G. Arora[2]

[*]Tata Technologies Limited, 25 Pune Infotech Park, Hinjawadi, Pune, India 411057
Email: biswadip.shome@tatatechnologies.com
Phone: +91 20 5652 9216, Fax: +91 20 5652 9140

[2]Climate Control Group, Engineering Research Center, Tata Motors Limited, Pune, India 411018

ABSTRACT

Front end cooling module which consists of condenser, intercooler, radiator and fan is an important part of vehicle design as it affects radiator thermal performance. The current study describes front end cooling module optimization using CFD methodology on a passenger vehicle. The CFD model is validated by comparing predicted radiator coolant inlet temperature with experimental results and the agreement is within 2 to 3°C. The validated model shows that the heat rejected by the condenser results in a 7°C increase in radiator coolant inlet temperature. This assessment was used to resize and relocate the condenser to minimize its influence on the radiator thermal performance. The optimized condenser configuration resulted in a 3°C reduction in radiator coolant inlet temperature.

Key Words: CRFM, CFD, Radiator, Vehicle Air Conditioner

1. INTRODUCTION

In recent years, increased engine thermal load and compact engine packaging requirements has led to significant challenges for vehicle designer to provide innovative thermal management and packaging solutions. Such solutions are being increasingly required in early design stages where prototypes are not readily available. In such scenarios, Computational Fluid Dynamics (CFD) models are being increasingly used to characterize and optimize the thermal performance of front end cooling module. An important part of the front end cooling module design is to ensure that the radiator has adequate heat removal capacity over a wide range of operating conditions. In particular, the adverse effect of condenser heat rejection on the thermal performance of the radiator needs assessment and if required, optimization of condenser size and location is carried out to prevent overheating of engine.

There has been some work in using CFD models for optimization of the front end cooling module. The work by Yang et al. (2002) has used CFD based models to evaluate condenser-fan-radiator module (CFRM) against the conventional condenser-radiator-fan module (CRFM) at a fixed vehicle velocity. Their work although later extended to vehicle idle condition (Yang et al., 2004) is limited to a fixed vehicle velocity and engine load condition and does not present any information on the radiator coolant side thermal performance. The other works include the study by Lee and Hong (2000) that used a validated CFD model to modify the front end opening for optimizing the engine cooling performance. Ecer et al. (1996) has used a similar analysis approach to predict the radiator coolant inlet temperature. However, both these studies focused on a single operating condition.

Even though there is a wide body of knowledge in application of CFD simulation to predict under-hood thermal performance and flow-thermal performance of cooling modules, limited published efforts are evident in using these simulations to predict the front end cooling module performance over a wide range of vehicle operating conditions. In particular, no published efforts are visible that have used CFD simulation to assess the effect of condenser size and location on radiator thermal performance.

In this study, a CFD simulation model has been used to assess the effect of condenser size and location on the radiator performance of a passenger. Starting from CAD data, a detailed model of the vehicle under-hood which includes the front end cooling module has been built. The model has been validated by comparing the predicted flow velocities in the vicinity of the radiator with experimental values. Subsequently, the predicted radiator coolant inlet temperatures have been validated by comparing them with experimental measurements. The validated CFD model has been used to resize and move the main condenser downward such that the heat rejected by the condenser on to the radiator is reduced resulting in a reduction in radiator coolant inlet temperature. Such use of validated CFD models has led to significant cost savings by reducing prototype testing and has also led to reduction of design cycle time by as much as 6 months. The methodology described in this paper can be adopted for new generation vehicle development programs in early design phase.

2. METHODOLOGY

In this study, a CFD model of a passenger vehicle has been built using the commercial CFD software Fluent (Fluent 6.2 Users Guide, 2005). The front end cooling module consisted of a condenser-radiator-fan module (CRFM) configuration. The fan module consisted of twin electric fans that run at constant speed and pull air through the cooling module. The CFD analysis carried out in this study is a vehicle level flow-thermal analysis in which the front end, under-hood, and underbody was included and analyzed as an integrated system. The major advantage in such vehicle level analysis is that ambiguities in setting up boundary conditions for different vehicle sub-systems, such as under-hood, under-body, and front-end module are eliminated, as these sub-system level boundary conditions are rarely known with accuracy.

2.1 System Modeling

Due to complex nature of the vehicle geometry, particularly its under-hood engine compartment, a tetrahedral mesh consisting of around 2 million cells is constructed. The under-hood engine compartment components included in this study consists of all major components such as the engine block, manifolds, flywheel, battery, sub-frames, boosters, and alternator. In addition to these components, the under-hood compartment also includes the condenser, radiator, fan, and intercooler. The front-end components modeled include the front-grill, the logo, the air deflectors, and sealing strips around the radiator and in-between the condenser-radiator assembly. The under-body module modeled in this study includes the wheel-arches, the bottom shield, axles, and tires. To accurately capture the incoming flow through the front end, the vehicle outer panel comprising of the side panels up to the B-pillar, the wind shield, roof, and the front bumper with the cutouts are included. The rear portion of the vehicle beyond the B-pillar is not included in this study as including it would have significant increased the model size with out contributing much to the prediction accuracy. To facilitate application of boundary condition, the model is placed inside a virtual wind tunnel box measuring 6m x 12m x 12m (height x width x length) to simulate a typical wind tunnel facility. The baseline front end cooling module that included the condenser-radiator-fan module and the intercooler is shown in Figure 1.

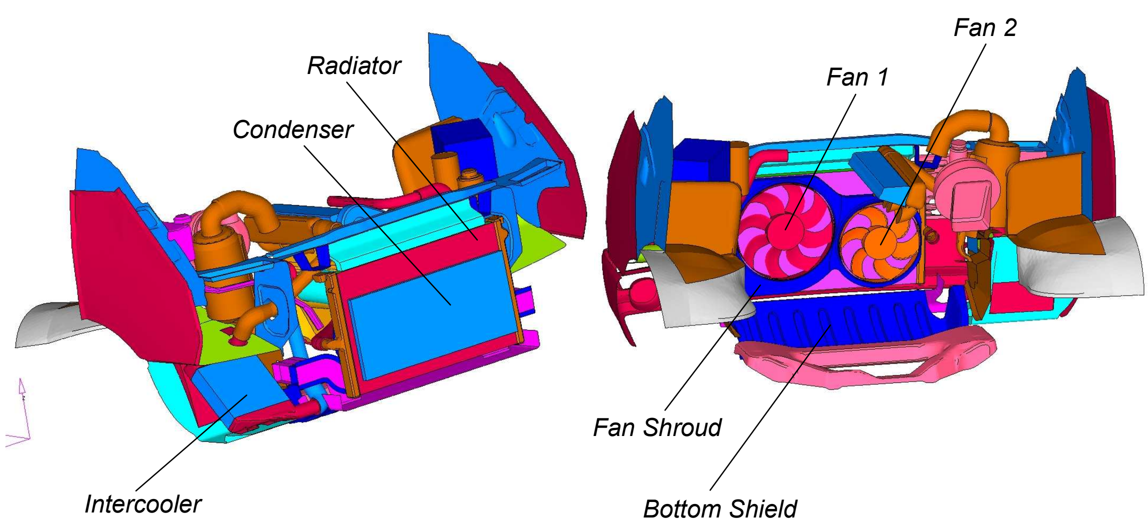

Figure 1: Baseline Configuration of Front End Cooling Module.

In this study, the effect of the condenser, radiator, and intercooler on the flow and thermal field is simulated. The thermo-hydraulic performance characteristics of these heat exchangers are typically represented by the pressure drop and heat rejection characteristics which are made available by the heat exchanger manufacturers. The pressure drop through these heat exchangers caused due to the flow resistance offered by them is simulated using a porous media approach (Fluent, 2005). The viscous and inertial component of the pressure drop is determined by a curve-fit of the measured pressure drop-air velocity data for the heat exchanger of interest. The heat rejection of the condenser and the intercooler is simulated by treating them as volumetric heat sources.

For radiator heat rejection, a different methodology is adopted as calculating the radiator coolant side temperature is of primary interest in this study. The methodology adopted is similar to that of Ecer et al. (1996) wherein the radiator core is divided into 48 x 45 macroscopic cells. Each macroscopic cell has equal number of computational cells. The heat rejected by the radiator coolant side is then calculated as:

$$q_{cell} = \varepsilon \left\lfloor \dot{m} c_p \right\rfloor_{air} \left\lfloor T_{coolant,\, in} - T_{cell} \right\rfloor \tag{1}$$

$$q_{macro} = \sum_{All\ Cells} q_{cell} \tag{2}$$

$$q_{total} = \sum_{All\ Macros} q_{macro} \tag{3}$$

where q_{cell} is the heat rejected by the computational cells, q_{macro} is the heat rejected by the macroscopic cells, q_{total} is the total heat rejected by the radiator coolant side, ε is the radiator effectiveness, and T_{cell} is the airside temperature of each computational cell. The radiator coolant temperature is calculated by performing a local cell-by-cell energy balance between the air-side and the coolant-side. The radiator fans are simulated using the Multiple Reference Frame (MRF) method as it accounts for the swirling flow induced by the rotating fans (Krus, 1998).

2.2 Boundary Conditions

For vehicle running condition, the incoming airflow is simulated by applying a constant velocity (equal to vehicle velocity of interest) boundary condition to the front face of the virtual wind tunnel domain. No-slip boundary condition is applied to the bottom face of the domain to represent the road surface. On the side and the top faces of the domain, a zero-shear boundary condition is applied to prevent boundary layer growth. On the rear face of the domain, a zero gradient along the flow direction is applied. For vehicle idle condition, a constant pressure equal to atmospheric pressure is applied at both front and rear faces of the computational domain, whereas the boundary condition at other faces of the computational domain remained unchanged from that used for vehicle running condition.

2.3 Assumptions

The model with the above boundary conditions is simulated using the following assumptions

(1) steady turbulent incompressible air flow
(2) turbulence modeled using the RNG k-ε model (Yakhot and Orszag, 1986) with non-equilibrium wall functions (Kim and Choudhury, 1995).
(3) inlet turbulence intensity of 5% and a turbulent to molecular viscosity ratio of 10.
(4) incoming air is at ambient temperature with temperature-dependent properties
(5) metal components included in the study are thermally adiabatic.

2.4 Simulation and Analysis

The conservation equations of mass, momentum, energy, and turbulence quantities for the above model were solved in a multiple CPU cluster till convergence norms were satisfied. Using the converged thermal and flow field, the radiator coolant inlet temperature was calculated using equations (1) to (3) and the velocity and temperatures at regions of interest was analyzed.

3. RESULTS

The CFD model developed in this study has been validated against experimental measurements in a planned multi-step approach. Two validation exercises have been conducted for the baseline design configuration to build confidence in the CFD model prediction quality.

3.1 Flow Validation for Baseline Configuration

The first validation exercise has been conducted in vehicle stationary and engine idle condition with both fans in operation. The predicted air velocities at distance of 50mm in front of the grill are compared against measured values at slow and fast fan speed settings. The comparison of the average predicted and measured velocity is shown in Table 1. As seen in Table 1, the predicted and measured air velocities for slow and fast fan speed condition compare to within 10%.

Table 1: Comparison of CFD Predicted Air Velocities with Measurements at a Distance of 50mm from Grill for Vehicle Stationary and Engine Idle Condition.

Fan Speed Settings	Fan 1 Speed (RPM)	Fan 2 Speed (RPM)	Measured Air Velocity (m/s)	Predicted Air Velocity (m/s)
Slow	1320	1920	2.1	1.9
Fast	2955	2735	3.4	3.4

3.2 Radiator Coolant Inlet Temperature Validation for Baseline Configuration

The second validation exercise has been conducted at vehicle running condition for a range of engine load conditions and vehicle speed. The predicted coolant inlet temperature has been compared against the experimental value for air conditioner (AC) on and off conditions and is shown in Table 2. As seen in Table 2, the CFD predictions and experimental data of radiator coolant inlet temperature compare well and are within 2 to 3 °C.

Table 2: Comparison of CFD Predicted Radiator Coolant Inlet Temperature with Experimental Data.

Test Point	AC	Vehicle Speed (km/hr)	Load Condition	Heat to Coolant (kW)	Radiator Coolant Inlet Temperature (°C)	
					Experimental Data	CFD Prediction
1	ON	25	1st Gear, 12% Gradient	35.4	109	109
2	OFF	25	1st Gear, 12% Gradient	35.4	94	96
3	ON	25	2nd Gear, Open Throttle	39.6	98	99
4	OFF	25	2nd Gear, Open Throttle	39.6	93	92
5	ON	20	1st Gear, 16% Gradient	43.6	112	111
6	OFF	20	1st Gear, 16% Gradient	43.6	93	95
7	ON	40	2nd Gear, Open Throttle	46.0	108	108
8	OFF	40	2nd Gear, Open Throttle	46.0	98	101

The velocity and temperature distribution in the radiator is shown in Figure 2 for a typical operating condition of 1st gear with 12% gradient, 25 km/hr vehicle speed with AC on.

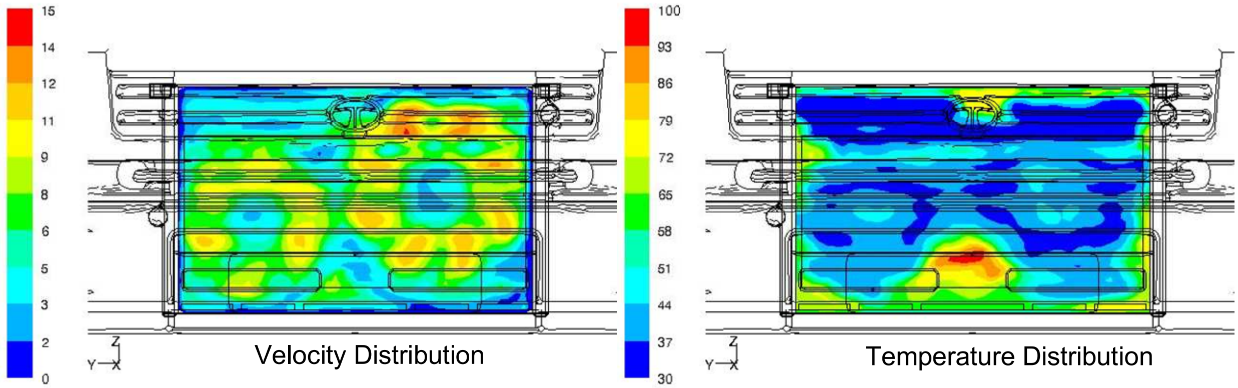

Figure 2: CFD Predicted Velocity and Temperature Distribution in Radiator for 1st Gear, 12% Gradient, and 25 km/hr Vehicle Speed with AC On.

As seen in Figure 2, low velocity regions in the radiator core correlates very well with the high air side temperatures, as expected. The results show that the heat rejected by the condenser adversely influences the radiator coolant inlet temperature, with the coolant inlet temperature increasing by as much as 7°C, when the air conditioner is on.

3.3 Modified Configuration

This adverse effect of condenser heat rejection on the radiator coolant inlet temperature has led us to optimize the front end cooling module. The optimization consisted of reducing the main condenser height by 100mm, the length by 10mm with a resulting reduction of condenser core frontal area by around 33%. In addition, the main condenser was relocated downward and an auxiliary condenser with fan was added to the side of the engine compartment as shown in Figure 3.

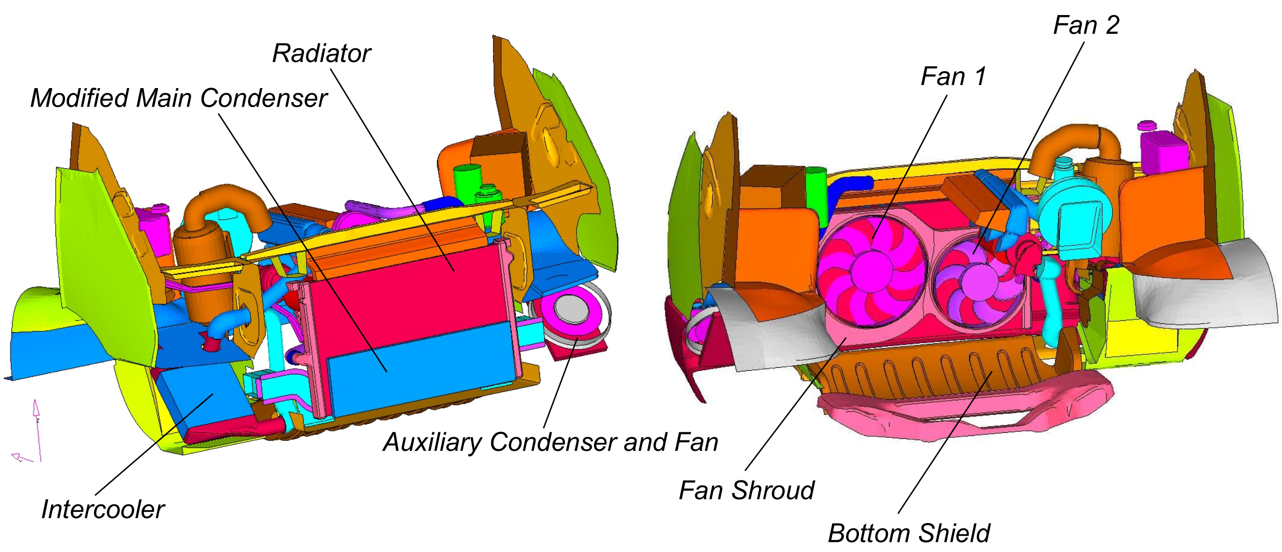

Figure 3: Modified Configuration of Front End Cooling Module.

The auxiliary condenser has been sized to compensate for the lost main condenser capacity and it was placed in a different air flow path, such that the heat rejected by it does not affect the thermal performance of radiator. The CFD prediction results of this modified configuration with AC on are shown in Table 3.

Table 3: Comparison of CFD Predicted Radiator Coolant Inlet Temperature between Baseline and Modified Configuration with AC On.

Test Point	AC	Vehicle Speed (km/hr)	Load Condition	Heat to Coolant (kW)	Radiator Coolant Inlet Temperature (°C)	
					Baseline	Modified
1	ON	25	1st Gear, 12% Gradient	35.4	109	106
2	ON	25	2nd Gear, Open Throttle	39.6	98	96
3	ON	20	1st Gear, 16% Gradient	43.6	112	109
4	ON	40	2nd Gear, Open Throttle	46.0	108	105

For AC on condition, the modified configuration has reduced the radiator coolant inlet temperature by around 3°C without compromising the air conditioning capacity. A comparison of the radiator core temperature between the baseline and the modified configuration is shown in Figure 4 for a typical operating condition of 1st gear, 12% gradient, and 25 km/hr with AC on. Figure 4 shows that the high temperature zone in radiator core caused by the heat rejected from the condenser is reduced in the modified configuration.

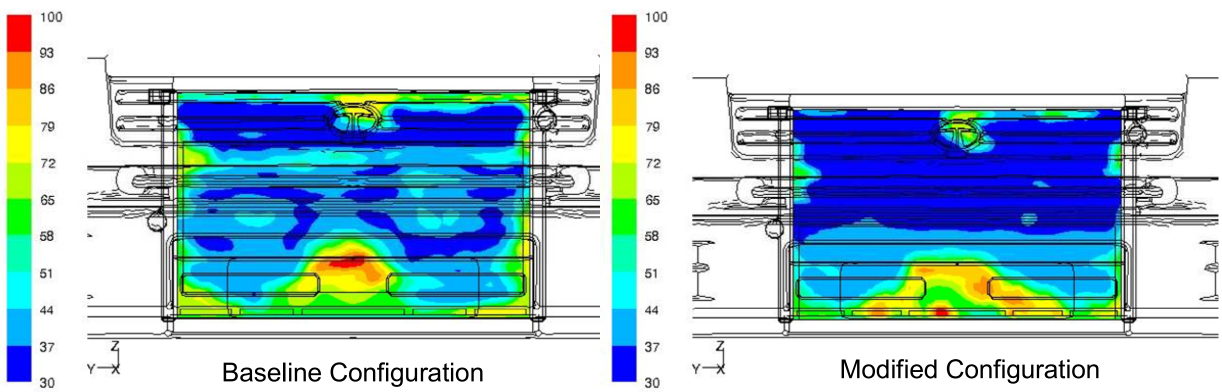

Figure 4: CFD Predicted Temperature Distribution in Radiator for Baseline and Modified Design for 1st Gear, 12% Gradient, and 25 km/hr Vehicle Speed with AC On.

4. CONCLUSIONS

A vehicle level CFD model for simulating and optimization the front end cooling module of a passenger vehicle has been developed. Comparison of the model predictions with experimental data shows a high degree of agreement. The major conclusions drawn from this study are:

- In the baseline design, heat rejected by the condenser adversely influences the radiator thermal performance, with the radiator coolant inlet temperature increasing by as much as 7°C, when the air-conditioner is on.
- Optimization of the condenser by reducing its size by 33% and relocating it downward to cover a lesser part of the radiator frontal area minimizes its influence on the radiator coolant inlet temperature. For AC on condition, the modified front end module reduces the coolant inlet temperature by around 3 °C.

- To compensate for the lost capacity in main condenser caused by its size reduction, an auxiliary condenser with fan has been added in the side of the engine compartment.

REFERENCES

Ecer, A., Toksoy, C., Rubek, V., Hall, R., Gezmisoglu, G., Pagliarulo, V., Caruso, S., and Azzali, J., 1996, Air Flow and Heat Transfer Analysis of an Automotive Engine Radiator to Calculate Air-to-Boil Temperature, *SAE International Congress and Exposition*, Detroit, U.S.A.

Fluent 6.2 User's Guide, 2005, Fluent Incorporated, New Lebanon, U.S.A.

Kim, S.E. and D. Choudhury, 1995, A Near-Wall Treatment Using Wall Functions Sensitized to Pressure Gradient, *ASME FED Vol. 217, Separated and Complex Flows,* p. 13-21.

Krus, H.W., 1998, An Evaluation of the Simulated Flow through the Engine Cooling system of a Truck, *Warmemanagement Tagung*, Essen, Germany.

Lee, Y.L. and Hong, Y.T., 2000, Analysis of Engine Cooling Including Flow Non-uniformity Over a Radiator, *Int. J. Vehicle Design*, Vol. 24, No. 1, p. 121-135.

Yakhot, V. and Orszag, S.A., 1986, Renormalization group analysis of turbulence, *J. Sci. Comput.*, Vol.1, p. 3-23.

Yang, Z., Bozeman, J., Shen, F.Z, Turner, D., and Vemuri, S., 2002, CFRM Concept for Vehicle Thermal System, *SAE Paper 2002-01-1207*.

Yang, Z., Bozeman, J., and Shen, F.Z., 2004, CFD for Flow Rate and Air Circulation at Vehicle Idle Conditions, *SAE Paper 2004-HX21*.

ACKNOWLEDGEMENTS

The authors would like to thank the climate control and vehicle testing group for their support and allowing the authors to publish this paper.