Numerical Prediction of Vehicle Front End Module Effects on Engine Cooling System Performance

Neelakandan K, Goutham Sagar M, Tushar Damodare, Pooja Nemade

Abstract— A numerical study to investigate the effects of passenger vehicle front end module on engine cooling system performance has been conducted. Since the front end module has a direct impact on vehicle drag and vehicle cooling performance, it plays an important role in vehicle design. The current study investigates the effects of grill area, horn blockage, condenser, and fan capacity on engine cooling performance. A 1-dimensional CFD modeling methodology is used with different sub-systems such as coolant, engine oil circuit, intercooler, condenser and other components such as thermostats, radiators, fans and pump etc. using GT-SUITE. A pre-processing tool called COOL3D which is part of GT-SUITE is used to build the 3D underhood of vehicles using a component-by-component build approach, and therefore allowing inclusion of much more details than a usual 1D simulation model. GT-SUITE object-based code helps us to build reliable cooling systems and optimize the front end module for various operating conditions and reduces our effort on real time testing. With above considerations and methodologies we significantly improved the engine cooling performance.

Keywords — Automotive cooling system, Front end module, Underhood, 1D CFD simulation, GT Suite

I. Introduction

An engine provides mechanical energy from an air/fuel mixture with efficiency between 20 and 45%. The rest flows as kinetic and heat energy in exhaust gases and as heat energy through metallic bodies due to the friction [3]. In this context, the cooling system must allow the engine to give its best performance, ensure the durability of this performance and ensure engine reliability by guaranteeing an acceptable level of thermo-mechanical stresses in any point of the engine. Average heat balance of a diesel engine can reach up to 40% efficiency and direct injection gasoline engine can now reach 30% efficiency with heat losses between 18% and 20% [4].

The liquid cooling system is also used to ensure passengers car heating, to regulate engine oil temperature, to regulate automatic transmission oil temperature and to cool EGR. In some particular cases, it can also be used to limit alternator temperature, to heat up the throttle body; to cool down power assisted steering, to extract energy from exhaust system, to cool down turbo bearing. Consequently, the number of critical situations increases, as well as control and monitoring difficulties or interferences between different requirements.

Considering the above criteria, it is necessary to select an optimum cooling pack to ensure better performance of the vehicle. In this current study, we carried out set of steady state simulations to predict the effects of Condenser, Front top and bottom apertures, fan and other underhood components.

II. Methodology

Thermal protection is increasingly important in the development process of passenger cars. Tightly packaged engine compartments and strongly increased engine power demand extensive testing and analysis. Traditionally thermal protection is tested in climate wind tunnels and road tests. Reducing time to market and high costs for prototypes result in the need for building the first prototype very close to its optimum design. The introduction of numerical methods allows the optimization of cooling requirements as well as thermal analysis of temperature sensitive components in a very early stage of the vehicle development process [3].

GT-SUITE is a fully transient commercial simulation software with multi-domain applications including flow, thermal, mechanical, electrical, magnetic, and controls - provides flexibility to model entire cooling system as 1D pipe network and quasi 3D underhood airflow modeling.

A. *1D CFD modeling*

The flow model involves the solution of the Navier-Stokes equations [3], namely the conservation of continuity, momentum and energy equations. These equations are solved in one dimension, which means that all quantities are averages across the flow direction. We used implicit time integration methods to solve the primary variables such as mass flow, pressure and total enthalpy. The conservation equations solved are shown below:

Manuscript published on 30 December 2014.
***** Correspondence Author (s)
Neelakandan K, PT-Cooling, TATA Technologies Ltd, Pune, India
Goutham Sagar M, PT-Cooling, TATA Technologies Ltd, Pune, India
Tushar Damodare, PT-Cooling, TATA Technologies Ltd, Pune, India
Pooja Nemade, PT-Cooling, TATA Technologies Ltd, Pune, India

Continuity:

$$\frac{dm}{dt} = \sum_{Boundaries} \dot{m}$$

Momentum:

$$\frac{d\dot{m}}{dt} = \frac{dpA + \sum_{Boundaries} (\dot{m}u) - 4C_f \frac{\rho u|u|}{2}\frac{dxA}{D} - K_p\left(\frac{1}{2}\rho u|u|\right)A}{dx}$$

Enthalpy:

$$\frac{d(pHV)}{dt} = \sum_{Boundaries} (\dot{m}H) + V\frac{dp}{dt} - hA_s(T_{Fluid} - T_{Wall})$$

B. Heat Transfer calculation

Colburn analogy is used to predict the heat transfer coefficient from fluids inside of pipes and flow split to their wall, the heat transfer coefficient is calculated at every time steps from the fluid velocity, the thermo-physical properties and the wall surface roughness. The heat transfer coefficient of smooth pipes is calculated using the Colburn analogy as below:

$$h_g = \left(\frac{1}{2}\right) K_f \rho U_{eff}\, C_p \Pr\left(\frac{-2}{3}\right)$$

Heat transfer is calculated between the fluid stream and the heat exchanger structure. This heat transfer calculation includes the effects of the wall thermal capacitance, the conductivity of the material; the temperature of the structure is calculated from a balance of the heat transfer rates between the structure and the two fluids using the following equation:

$$\frac{dT_{Wall}}{dT} = \frac{Q_m + Q_s}{\rho V C_p} = \frac{\left[hA\Delta T - \frac{2kA\Delta T_{Wall}}{t}\right]_M + \left[hA\Delta T - \frac{2kA\Delta T_{Wall}}{t}\right]_S}{\rho V C_p}$$

B. Underhood Consideration

Underhood components are modeled in quasi 3D methods with consider air flow boundaries in GT Suite-Cool3D, Size and positions of heat exchanger components are measured from CAD data and modeled in Cool3D. Baseline and modified front modules are studied with four different vehicle operating conditions of (1) high speed, (2) high gradient and low speed, (3) low gradient and high speed, and (4) idle conditions. Corresponding engine operating boundary conditions are calculated by suitable assumptions of vehicle data, ambient temperature and charge air cooler inlet temperature etc., and the effect of underhood dimensional changes on engine cooling system performance was studied.

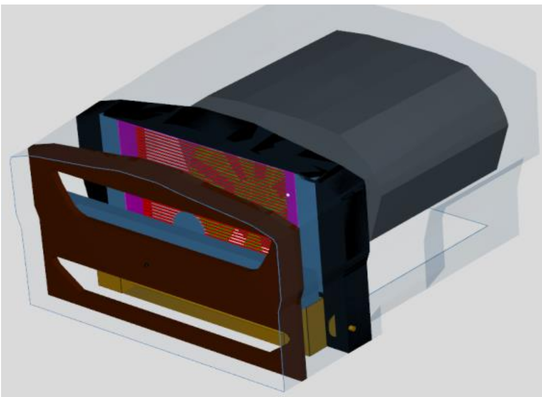


Figure1: Baseline Condenser

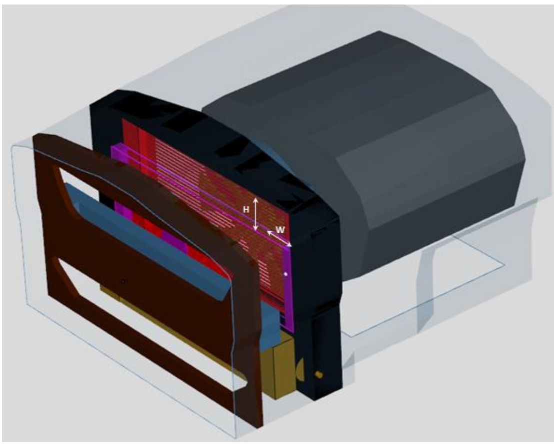


Figure2: Modified (reduced) Condenser

III. HEAT EXCHANGER MODELING

Heat exchangers are modeling both fluid circuits a lumped 1D master-slave type of heat exchanger except condenser, Heat exchangers contains the heat exchanger geometry, performance data, and pressure drop data for both fluids etc. Performance and geometry data is to be entered for a particular heat exchanger as it was tested values [3].

Nusselt number correlation method is used to define heat transfer performance data of a heat exchanger as a function of flow rate, pressure, and temperature. Heat transfer rates can be calculated for each of the experimental data points. Heat transfer rates at an arbitrary flow rate for each fluid are calculated using heat transfer convection coefficients defined by a Nusselt number correlation of the form:

$$Nu = C \operatorname{Re}^m \Pr^{\left(\frac{1}{3}\right)}$$

Where:

$$Nu = \frac{hL}{K}; \quad \operatorname{Re} = \frac{\rho UL}{\mu}; \quad \Pr = \frac{\mu C_p}{K}$$

It used to determine the correlation coefficient, C, and exponent, m, for both sides of the master-slave heat exchanger using a linear regression analysis of the experimental data. The temperature difference above is an overall difference between the temperature of the fluid and the wall temperature, which is averaged over the heat exchanger.

IV. RESULS AND DISCUSSION

The initial performance is calculated with baseline module which has full face 280 x 400 x 20 mm condenser. Due to this large area heat addition, the cooling airflow preheats and dramatically reduces the radiator performance. In order to reduce condenser effect on cooling airflow we reduced original condenser height and increased width by 100 mm and 100 mm, respectively and we studied the effects on radiator outlet temperatures. Figure 1 and Figure 2 shows the cooling pack configurations of baseline and modified condenser underhood. Figure 3 and Figure 4 show that the modified condenser reduces the wall surface temperature and results in an improved the radiator outlet temperatures.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

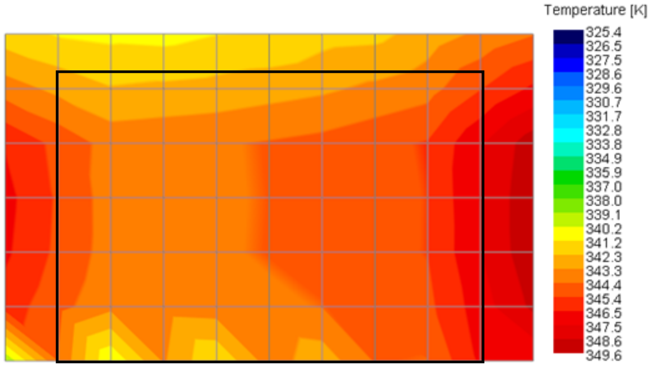


Figure3: Radiator wall temperature (Baseline condenser)

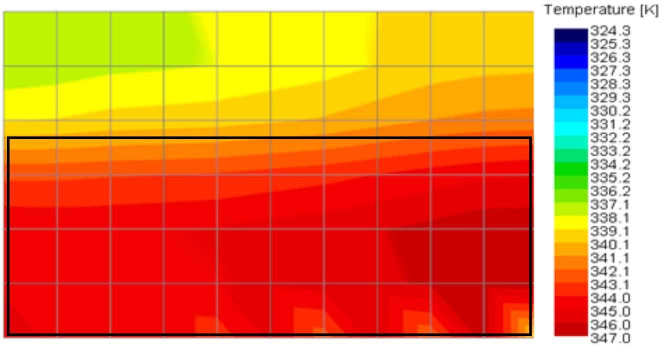


Figure4: Radiator wall temperature (Reduced condenser)

Due to the reduction of condenser size, increased air velocity is observed over radiator external surface. Figure 5 and Figure 6 show the variations in air velocity on radiator surface for high speed and idle test conditions.

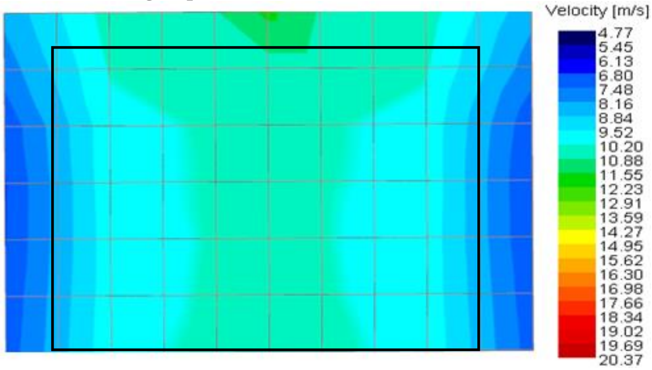


Figure5: Air velocity at Radiator (HS-Baseline condenser)

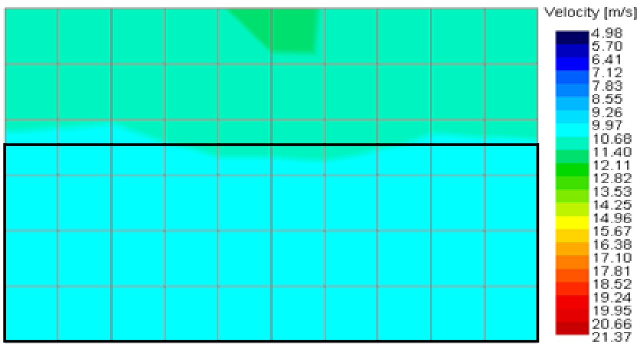


Figure6: Air velocity at Radiator (HS-Reduced condenser)

There is signifiant air velocity improvement observed in case of both high speed and Idle test conditions.

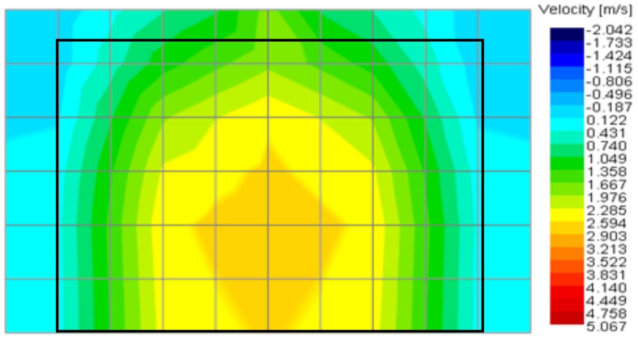


Figure7: Air velocity at Radiator (Idle-Baseline condenser)

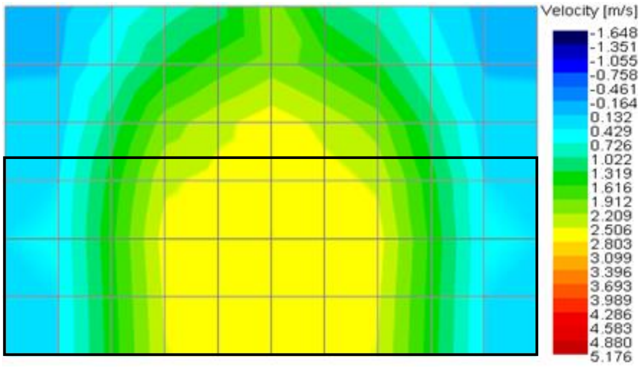


Figure8: Air velocity at Radiator (Idle-Reduced condenser)

Due to the improvement in flow of ram air and reduced heat addition of modified condenser, radiator effectiveness is improved and thus, the outlet temperature of the radiator is reduced signifiantly, Figure 9 shows the reduction of radiator outlet temperatures.The medium and low speed cases outlet temperature has signifinat improved around 4°C and in high speed cases only 2°C variations are observed due to its air recirculation and upheat effects.

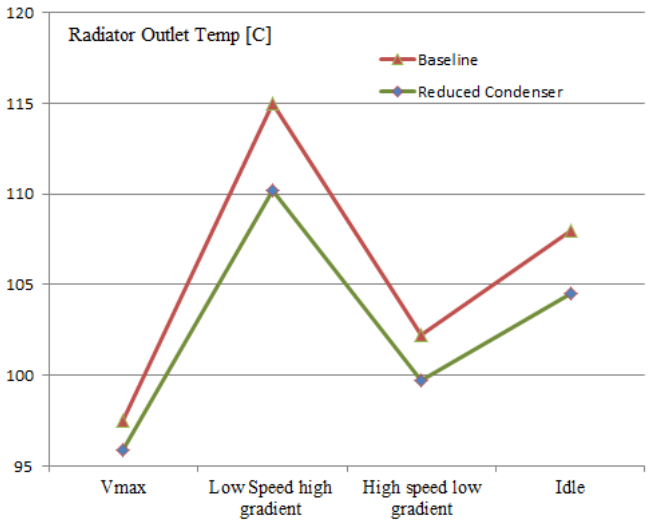


Figure9: Radiator out temperature (Baseline & Modified condenser)

We also studied the effects of horn block location on radiator performance. Figure 10 shows that moving horn block just below the bumper can improve the airflow rate to the underhood, thus, increasing radiator outlet temperature in low speed cases around 2°C.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

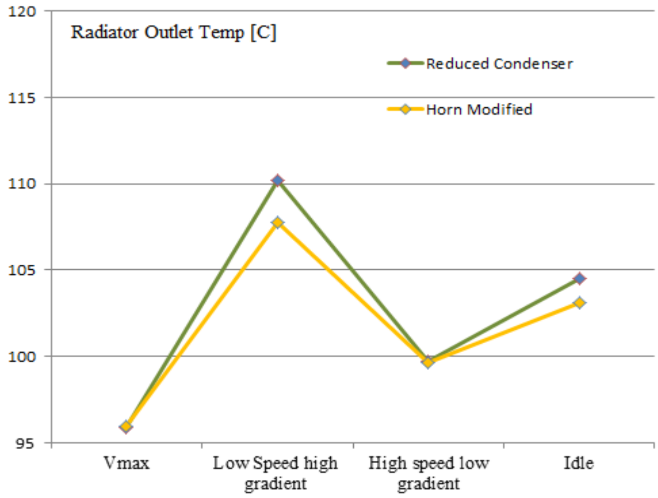
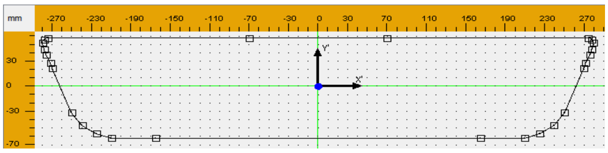


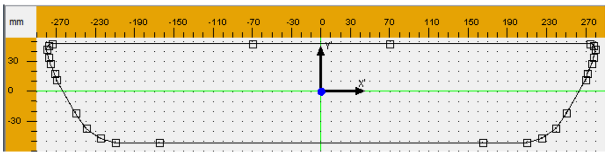
Figure10: Radiator out temperature (Modified condenser & modified horn location)

In order to get improved cooling performance and reduced vehicle drag, it is necessary to study characteristics of the front end opening. The modified condenser with updated horn block module is studied with different front grill profiles (areas), which are shown in Figure 11. We studied three different profiles A, B and C having total area of 62113.0 mm², 51713 mm² and 39988 mm², respectively. The respective grills' average pressure drops are calculated and directly imposed by the power law coefficient and we considered the same vehicle operating points for these studies. Due to the change in front grill areas, the total cooling airflow changes resulting in significant impact on the radiator cooling performance.

Profile A: 62113.0 mm²



Profile B: 51713.0 mm²



Profile C: 39988.0 mm²

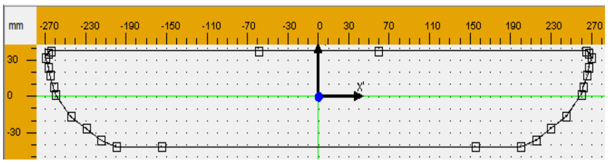


Figure11: Front grill profile

Consider profile A and C, The large and small areas of grills have significant changes on airflow values as compared to those in case of profile B. Below, Figure 10 and Figure 11 show the variations of cooling airflow rates through the radiators and their effect on radiator outlet temperatures. Profile A has higher cooling airflow rates because of higher opening area and so it shows significant reduction in radiator outlet temperatures in comparison to profile B and profile C. Because of smaller area of profile C, it is not able to meet the target airflow for few cases as a result of which, the outlet temperature of radiator reached the maximum desired temperature limit of 110 °C.

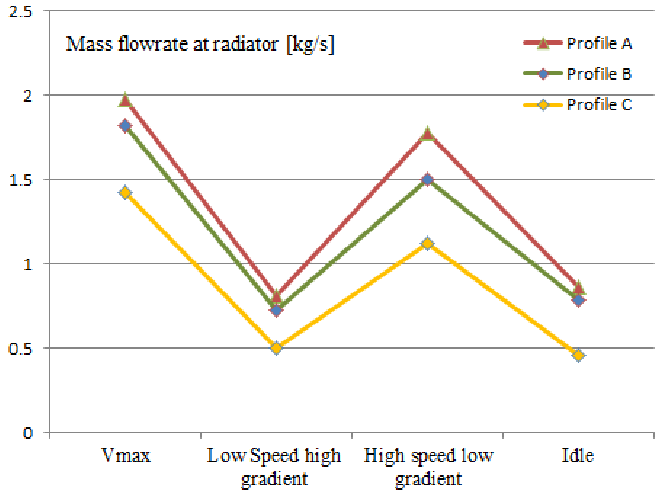


Figure10: Cooling airflow rate at Radiator

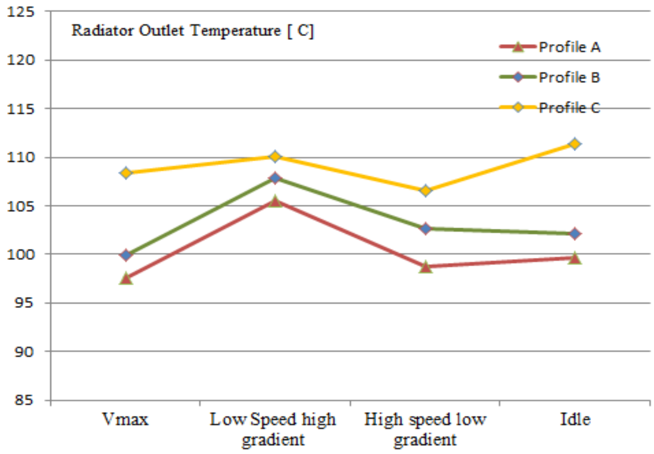


Figure11: Radiator outlet temperature

Even though profile A provides better cooling at the radiator side, its wider opening area significantly increases the vehicle drag, and so we choose profile B for further study relating to the effects of fan capacity on cooling performance.

We also studied the effect on fan capacity on cooling airflow. All above studies were carried out with 650W fan, We replaced it with a 400W fan on the module and studied its effects for the same vehicle operating points. We assumed the constant fan speed 2400 rpm for entire simulations procedure,

Due to reduction in fan capacity, cooling air flow significantly decreases at the radiator for idle and low speed cases. It has less impact on cooling air flow in high speed cases. Figure 12 shows the reduction in radiator outlet temperature due to reduction in the fan cooling airflow.

We can see that the 400W fan does not meet our cooling airflow target. As a result, the outlet temperature of radiator reaches the maximum desired temperature limit of 110 °C. From above studies, we can conclude that the minimum cooling airflow target can be achieved only by 650W fan.

Published By:
Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP)

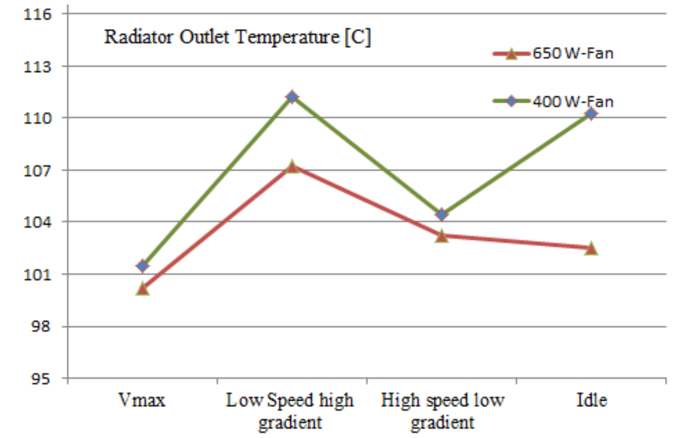


Figure12: Radiator outlet temperature

V. CONCLUSION

Now a day, the numerical analysis of the underhood flow is a vital part of the development process of passenger cars. Additionally, these days, thermal protection is increasingly a subject of numerical investigation. This requires a multi-mode heat transfer analysis considering conduction, convective heat transfer and radiation. This can be done by coupling of sophisticated and specialized simulation codes. Investigation of front end module effects are successfully carried out for our cooling pack, Based on the underhood air flow and Heat Exchanger performance, we optimized the front end module. Due to 1D modeling approach it is very difficult to capture all other flow physics such as up-heat air temperature, air recirculation and underhood air leakage characteristics etc. The condenser was considered as a single zone modeling, this consideration may not be effective while predicting thermal characteristics, so a detailed 3D CFD analysis is required to further confirm the modifications. GT Suite has been successfully used with coupling of GEM3D and underhood module (Cool 3D).

NOMENCLATURE

$\dot{m}$	Boundary mass flux into volume, $\dot{m} = \rho A u$
m	Mass of the volume
V	Volume
p	Pressure
ρ	Density
A	Flow area (Cross sectional)
A_s	Heat transfer surface area
e	Total internal energy per unit mass
H	Total enthalpy
h	Heat transfer coefficient
T_{fluid}	Fluid temperature
T_{wall}	Wall temperature
u	Velocity at boundary
U_f	Effective velocity outside boundary
K_f	Skin friction coefficient
K_p	Pressure loss coefficient
D	Equivalent diameter
dx	Discretization length
dp	Pressure differential acting across dx
ΔT	Temperature different between fluid and wall
ΔT_w	Temp different between surface and Avg. wall temp
k	Thermal conductivity of wall
Pr	Prandtle Number
μ	Dynamic viscosity of the fluid

ACKNOWLEDGMENT

The authors would like to thank Dr. Shome Biswadip and Dr. Satya Vallurupalli, TATA Technologies ltd, Pune for their valuable guidance and support during this project. We would also like to thank Mr. Vinayak Gapchup and Mr. Ajay Virmalwar, TATA Technologies ltd, Pune for encouraging us to publish this paper.

REFERENCES

[1] Biswadip; Kumar, Vinod; Kumar, S. V. Ranganath; and Arora, Gyan, *"CFD Prediction to Optimize Front End Cooling Module of a Passenger Vehicle" (2006). International Refrigeration and Air Conditioning Conference. Paper 845*

[2] Neelakandan K, Goutham Sagar M, Ajay Virmalwar *"Investigation of Radiators Size, Orientation of Sub cooled Section and Fan Position on Twin Fan Cooling Pack by 1D Simulation"* International Journal of Science and Research (IJSR) (2014).

[3] GT User manual V7.4 "Flow modelling manual" *Gamma Technologies. (2013)*

[4] Thomas Binner, Heinrich Reister, Ernst Peter Weidmann, Jochen Wiedemann *"Aspects of Underhood Thermal Analyses"*

[5] S.C. Pang, M.A. Kalam, H.H.Masjuki, I.A. Badruddin, R. Ramli and M.A. Hazrat. *Underhood Geometry Modification And Transient Coolant Temperature Modeling For Robust Cooling Networks*

[6] F. G. Tenkel, "*Computer Simulation of Automotive Cooling Systems*," SAE Paper 740087, 1974 in page no. 19.

[7] J.C. Corbel, "*An Original Simulation Method for Car Engine Cooling Systems: A Modular System*," SAE Paper 870713, 1987 in page no. 27

[8] J. A. Sidders and D. G. Tilley, *"Optimizing Cooling System Performance Using Computer Simulation*," SAE Paper 971802, 1997 in page no. 16

[9] D. Ganga Charyulu a, *, Gajendra Singh b, J.K. Sharma "*Performance evaluation of a radiator in a diesel engine- case study* Applied Thermal Engineering 19 (1999)

First Author:

Neelakandan K
M.Eng (Thermal), having 8+ years' experience in 1D CFD simulations working as Technical specialist in TATA Technologies Ltd, INDIA, and his research interests include: Engine, Vehicle thermal management Underhood optimization, Heat transfer and CFD.

Second Author:

Goutham Sagar M
M.Tech (Chemical), having 5+ years' experience in 1D CFD simulations working as Team Leader in TATA Technologies Ltd, INDIA, and his research interests include: Combustion, Vehicle thermal management, Underhood optimization, Heat transfer and CFD.

Third Author:

Tushar Damodare
B.E (Mech), having 4+ years' experience in 1D CFD simulations working as Sr. Engineer in TATA Technologies Ltd, INDIA, and his research interests include: Vehicle thermal management, Underhood optimization, Heat transfer and CFD.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

Fourth Author:

Pooja Nemade
B.Tech (Mech), having 1+ years' experience in 1D CFD simulations working as GET in TATA Technologies Ltd, INDIA, and her research interests include: Vehicle thermal management, Underhood optimization, Heat transfer and CFD.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)