Underhood Flow Management of Heavy Commercial Vehicle to Improve Thermal Performance

Chetan Kulkarni.[1], Deshpande M. D.[2], Umesh S.[3], Chetan Raval[4]
1-M.Sc. [Engg.] Student, 2-Professor, 3-Asst. Professor.
Department of Automotive and Aeronautical Engineering,
M. S. Ramaiah School of Advanced Studies, Bangalore 560 058
4-CAE Manager, Mahindra Navistar Automotives Ltd, Pune.

Abstract

Thermal management of heavy commercial vehicles is crucial for improving their performance, especially in the engine compartment where heat rejection is more and cooling is a challenge. As heat rejected from the engine increases, this may affect other components like intake manifold. Heat localization in the engine compartment may lead to overheating. Hence to solve this problem and to maximize cooling performance of heavy commercial vehicles a CFD approach has been adopted. Steady state fluid flow analysis was carried out for the baseline model of 25 ton truck and with modifications; the results obtained are used for further improvement.

Numerical modeling has been carried out in HYPERMESH.11 and simulation has been done in the FEM based CFD solver ACUSOLVE.1.8. Spalart Allmaras turbulence model is considered for simulation. Complete analysis has been carried out in two stages: cold flow analysis and hot flow analysis. Cold flow analysis was done to improve flow circulation in the compartment whereas hot flow analysis was carried out for studying heat dissipation. On studying base line model results local modifications were done incorporating additional openings, flaps and shroud length extension Simulations were carried out for maximum power and maximum torque conditions, monitoring the mass flow rate, velocity and temperature at different locations in engine compartment.

Flow recirculation zones present are identified in Underhood compartment and it is observed that front openings, shroud length which covers radiator fan are very small for allowing adequate quantity of air through radiator. Modifications in the shroud geometry lead to 20% improvement in mass flow rate through radiator and intercooler. Modifications incorporated improved the velocity of air at radiator and over the engine surface. This led to about 20% increase in the heat rejection from radiator and exhaust manifold, thus reducing the maximum temperature of the hot spots e.g. close to the exhaust manifold maximum temperature reduced by 114.7°C.

Key Words: Underhood, Heavy Commercial Vehicle, Thermal Management, Exhaust Manifold and CFD.

Nomenclature

h	heat transfer coefficient, *W/m²K*
k	Thermal conductivity, *W/mK*
q	Heat flux, *W/m²*
T	Temperature, *ºC / K*
V	velocity, m/s
ρ	Density, kg/m³
μ	Viscosity of air, kg/m-s

Abbreviations

FEM	Finite Element Method
CFD	Computational Fluid Dynamics
CAE	Computer Aided Engineering
FE	Finite Element
MRF	Moving Reference Frame

1. INTRODUCTION

Advancement in heavy vehicle industries in the field of styling, fuel efficiency and performance is taking place at fast pace. This has led to rapid development in Automotive Industries. However there are some areas that need immediate attention like engine cooling. Heavy vehicles are usually meant for high power and transport applications for long distances. Hence keeping low engine surface temperature is very important from cooling point of view. Another factor like aerodynamic styling of trucks has lead to reducing in the front openings of the trucks for minimizing aerodynamic drag. This reduces the amount of air incoming causing poor air cooling of the underhood and hence a reduction in engine cooling performance.

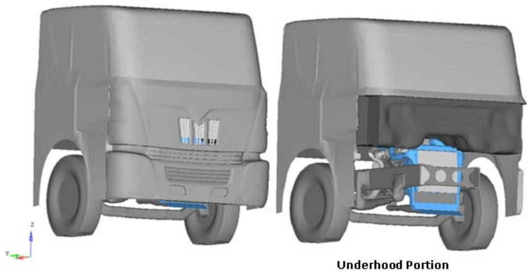

Fig. 1 Front portion of truck and underhood portion

Figure 1 represents the front openings of trick and the underhood portion indicating cooling unit. Front half end of the truck is considered for analyzing the problem and the CFD model is built considering the physical model and its physics. RANS with one equation Spalart Allmaras model is used for analyzing the flow in the compartment. The problem is solved in CFD based solver AcuSolve which employs Galerkins least square method for higher order accuracy.

Model considered for analysis is geometrically cleaned up and complete FE domain of the Fluid enclosed is developed in HYPERMESH .11 which is meant for pre-processing in ACUCONSOLE. Further, boundary

condition application, initialization and solving have been done in ACUCONSOLE / ACUSOLVE, results obtained are post processed in HYPERVIEW. 11.
Results obtained in the form of velocity vectors and temperature contours provide scope for improvement by design modifications in geometry, inlet vents, fan shroud re-design and its location for effective air circulation. This avoids air recirculation in non heat zones and the amount of air deflected towards the heat exchangers to circulate over surface and keep surface temperature low.

1.1 Background Theory

Excess heat from the engine surface is absorbed by coolant which circulates through radiator. If the coolant temperature rises more than 105°C fan coupled to engine gets activated, sucking greater amount of air from the front openings and blows over the engine surface.

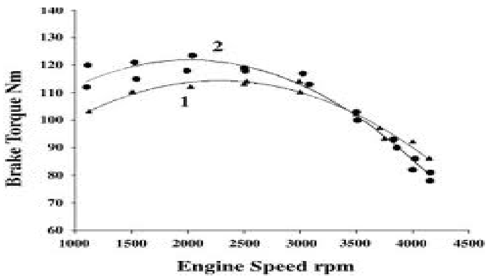

Fig. 2 Brake torque vs. engine speed curve for trucks Error! Reference source not found.

Vehicle moving under maximum cooling load condition i.e. at maximum power and maximum torque emits maximum amount of heat through heat exchanger, causing heat localization in the region where the air doesn't circulate. Especially when the vehicle is in the first gear torque will be high but engine speed will be very low i.e. fan speed will be low as shown in figure 2. In this case the air velocity will be very small as vehicle is moving in very low speed and consequently fan speed is less. This is the condition where heat generation is high and rejection is low. This affects the performance of radiator and intercooler; hence it is necessary for managing proper heat dissipation from the engine compartment. Higher the velocity greater is the heat dissipation, hence it is essential to maintain velocity and mass flow rate higher through the heat exchangers at heat localized zones in engine compartment.
Chacko, Vinod, Shome and Aggarwal [1] at Tata motors Ltd, Pune has carried out analysis for heavy vehicles to understand the flow and thermal characteristics in the engine compartment. This work represents methodology for improving the vehicle thermal characteristics. Results indicated low velocity in the engine vicinity with the recirculation in the compartment blocking the fresh air from the underbody. This resulted in thermal risk; hence iterations were carried out to improve flow fields by changing the geometery. This result was used to relocate the intercooler and to provide enough air circulation in the engine compartment.
Stevens, Bancroft and Sapsford et al [2] has done the work for predicting and improvement in the Underhood airflow in the Ricardo Consulting Engineers Ltd for the Renault's passenger vehicle. The VTHERM program mainly concentrated to improve the underhood cooling. Kurs [3] at Warmemanagement, Essen stated his work on the evaluation of the airflow simulation through the engine compartment with respect to the cooling system of the truck. This paper highlighted the characteristics of airflow in the underhood of heavy vehicles and basics of flow phenomena.
Amodeo, Alajbegovic and Janson [4] in their work made schematic study on the flow simulation through the engine compartment for the passenger car which involved method for numeric effective and flow resolution. Lattice Boltzmann method was used which had effectiveness for handling complex flow geometry especially for the underhood geometry. Simulation of interaction between the airflow and the heat generation has been done using 1-D modeling. Here both the cooling effect and aerodynamic performance were evaluated effectively and correlated for the total vehicle analysis which signified the effect of underhood cooling over overall aerodynamic performance.
Simulation of cooling airflow was done by Alajbegovic, Bing Xu, Konstantinov and Amodeo et al [5] under different conditions. In this work they concentrated on the approach for identifying the effect of air flow cooling for different driving conditions of the Ford Mondero. Several driving conditions were considered for verifying the flow underhood and underbody. For Underhood temperature distribution on the radiator and intercooler face and total wake pressure distribution was evaluated to verify the cooling flow effects and was identified those high recirculation occurring at the complex geometry section in the engine compartment.
Costa [6] considered the aspects of front opening based on the isolated cooling package and flow requirements. Results presented the air flux and temperature distribution management. This approach considered the front fascia design for the air-flow requirements, evaluation of power train cooling packages, performance of other components in the engine compartment affected due to their positioning and localized temperature. Fortunato and Damiano [7] carried out research on Underhood cooling simulation for new vehicle development. Under this work methodology for 3D thermo dynamic simulation has been stated considering external styling and underhood components.
Wambsgnass et al [8] carried out research for identifying the issues related to Thermal Management in the Heavy vehicles. Work illustrates the importance of control over heat rejection from the components coming under engine compartment and its effects over vehicle performance. Effects on emission due to poor thermal management has also been discussed. The research also reviewed innovative concepts for improving cooling systems like including new improvised cooling system which is effective and incorporates less space.
Hallqvist [10] at Scania Commercial Vehicles conducted a parametric study on cooling air flow through the underhood. Study was done by monitoring mass flow rate through the radiator for different configurations of fan and fan shroud. Study identified the effect of fan size, shroud depth its layout and location for improving air velocity. With the help of data generated best configuration of fan along with

shroud was suggested. Modification in encapsulation was suggested for rear portion which improved reduction in air entrapment on top of engine. Complete underhood modification was suggested right from the fan size to shroud improvement for optimisation.

1.2 Problem Definition

Aerodynamic styling and encapsulation has led to reduction in front openings of truck, due to this lesser quantity of air enters in the engine compartment. In case of maximum torque condition vehicle velocity is lower and heat carried away from coolant is higher. Hence heat rejection from radiator and intercooler is poor thereby reducing the performance of radiator. This also causes heat localization affecting components like intake manifold in the engine compartment.

To solve this problem underhood flow and thermal analysis of the heavy duty truck is carried out in order to improve heat dissipation and to make suggestions for improving engine compartment cooling through modifications. Baseline model of truck underhood portion was analysed by creating computational domain in HYPERMESH. 11 and simulated using CFD solver ACUSOLVE 1.8. Flow recirculation and heat rejection from radiator are the two necessary factors to monitor for improving heat dissipation through engine compartment.

1.3 Methodology

- Literature review was been done for geometry simplification and underhood thermal management
- Computational domain of underhood portion for vehicle was done using HYPERMESH 11.
- Based on the assumptions numerical model was developed and analysis is carried out in FEM based commercial CFD solver ACUSOLVE 1.8a
- Analysis has been carried out for minimum speeds at maximum power and torque condition
- Cold flow simulation has been carried out for baseline model analyzing flow and velocity distribution
- Forced convection has been carried out for studying heat dissipation through compartment
- Based on baseline simulation results modification done for improving flow characteristics and heat dissipation

2. DEVELOPMENT OF NUMERICAL MODEL

Analysis is done by developing the numerical model, simulation and post processing. Main aspect of this analysis is air flow and heat transfer, considering this different components coming in the engine compartment have been identified as different heat sources and frontal openings for air flow. Components considered for the analysis are engine, turbocharger, radiator, intercooler which are the major heat sources. Other components like transmission, chassis, suspension, wheels, cabin floor, front grills and openings are also considered which affect the flow path under the cabin floor i.e. the engine compartment. Wheel is considered as the obstruction for the flow. Complex shape of heat exchangers are not considered, instead a rectangle block of similar dimensions are modelled with fluid medium inside. Consideration of engine along with intricate shapes would increase the complexity of the problem, hence side face of engine cylinder block was modelled as a simplified surface.

CFD approach to solve this problem is carried out in two different stages i.e. cold flow analysis and hot flow analysis. Cold flow analysis is carried out for studying velocity distribution and flow path through the engine compartment and high velocity source i.e. fan and over engine surface. Mass flow rate of air entering through heat exchangers was also monitored. Hot flow analysis has been done to identify heat dissipation and temperature distribution in the compartment.

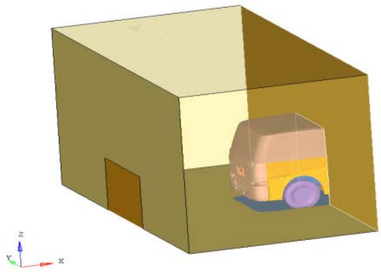

Fig. 3 Representation of computational domain

Computational domain has been created by creating water tight model of truck underhood portion for the components considered as shown in figure 3. The domain is been extended up to 6m in all three direction for analyzing the underhood portion of the truck.

2.1 Discretisation:

Computational domain created is discritized in HYPERMESH. 11 for carrying CFD simulations. Since watertight model was created by surface meshing using triangular elements, fluid medium has been created using these triangular elements as input. Following grid generation scheme was followed

- Tetrahedral elements were used for creating fluid domain
- Inside the underhood domain minimum element size used 2mm and maximum is maximum is 45mm
- Total number of elements were restricted up to 9.2million

Shape of the underhood portion was too complex and front openings were small for allowing fresh air, thus causing convergence and accuracy difficulties. Hence grid independence study was carried out for different combination of grids, pressure drop of radiator was monitored for accuracy.

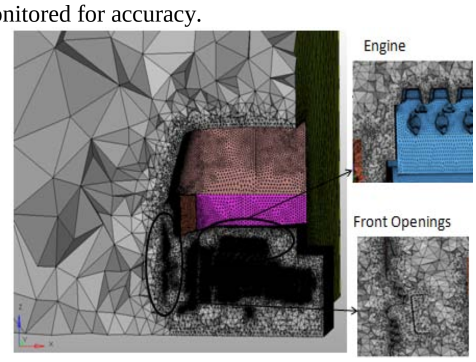

Fig. 4 Grid generated represented at symmetry plane

Figure 4, represents the tetrahedral grid at centre of XZ plane showing refinement at critical locations. Finalized grid was refined near the walls and critical regions like front openings. Low y^+ value i.e. below 75 has been maintained over the complex surface like engine.

2.2 Boundary conditions:

Wall boundary conditions were applied for the walls of truck in underhood portion. Heat source i.e. engine surface has been applied as constant temperature wall. Intercooler and radiator were considered as volumetric heat source where heat rejected was applied to respective fluid medium. Velocity inlet has been specified for the front face of the domain and pressure outlet for domain outlet. Velocity at maximum power is Porosity conditions were applied for the radiator and intercooler specifying Darcy coefficient and Franchimer coefficient, with specifying direction of 3° to X direction. Rotating motion of fan is applied as MRF approach where the mesh doesn't rotates but it numerically calculates the mesh motion at each node and applies while mesh is stationary. This approach reduces computation time and complexity of the problem. MRF is applied to fan surface and fluid domain surrounding. Fan rotation was specified with respect to engine speed at maximum power and maximum torque conditions.

3. RESULTS AND DISCUSSIONS

Results were analysed for cold flow and hot flow simulations separately. Velocity, mass flow rate and recirculation zones are monitored in cold flow. Temperature in the engine compartment and heat flux are monitored in hot flow simulations. Simulations have been done for maximum power and maximum torque condition for respective engine speeds.

3.1 Results of baseline simulation:

At maximum power condition velocity of vehicle is higher, heat rejection is also higher. Chances of recirculation is more, hence results of baseline model are initial analysed for maximum power condition.

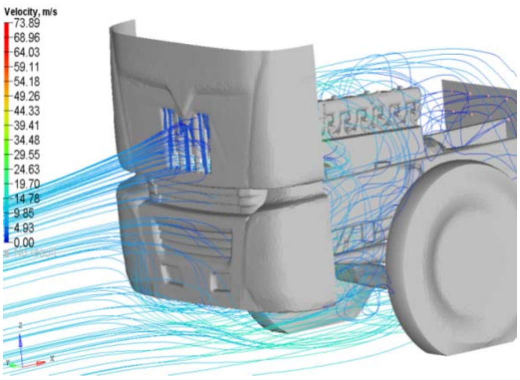

Fig. 5 Streamlines colored by velocity magnituted for baseline simulation

Air entering into the engine compartment through the front openings is shown in the figure 5 where velocity of air gets reduced at the angled openings. This leads for reduced air circulation into the engine compartment especially into radiator and intercooler.

Velocity distribution through the underhood compartment was analysed by plotting on symmetric XZ plane as shown in figure

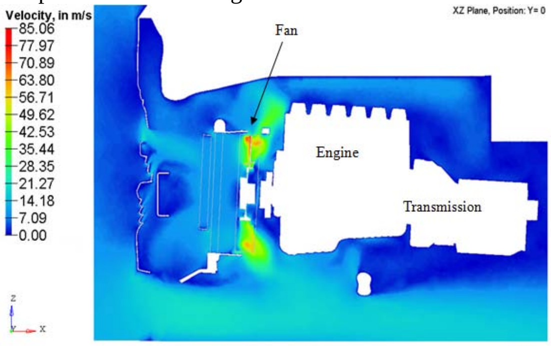

Fig. 6 Contours colored by velocity magnituted at maximum power condition for baseline

Results indicated air entering through the openings is obstructed by the chassis in the front portion. High velocity of air is blown over the engine surface, this causes high recirculation zones near the radiator and other portions of the compartment as shown in figure 6.

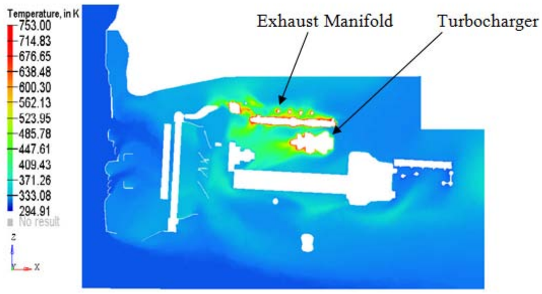

Fig. 7 Temperature contours plotted at exhaust manifold section for baseline

Figure 7 represents temperature contours of fluid under the engine compartment indicating high temperature source at exhaust manifold and the half portion of the turbocharger with higher temperature. Temperature plot indicates the heat rejection from these heat sources where fluid at ambient temperature carries away heat from the heat source. Hot spots are identified at the turbocharger section where less heat is carried away from the fluid

At maximum torque condition velocity of the vehicle is very low and heat rejection is higher. Hence fan is blowing air with greater velocity over the engine. This condition is much similar to idling condtion. Velocity distribution in the engine compartment is represented as contours colored by velocity magnituted at symmetric plane as shown in figure 8.

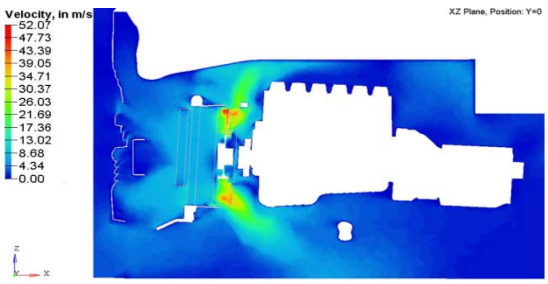

Fig. 8 Contours colored by velocity magnituted at Maximum torque condition for baseline

Under maximum torque condition velocity of air blown by fan in the engine compartment will be high, this leads for higher recirculation. Chances of air circulating back to radiator are higher. This affects heat exchangers performance.

3.2 Modifications

Results of baseline simulation indicated high recirculation zones over the radiator and at the rear portion of engine below cabin floor. Hence to reduce recirculation from fan local modifications are incorporated

- Fan shroud covering fan 35% is extended to 65%. This creates higher suction pressure at fan, sucking air with much higher velocity.
- Flaps are provided at front portion of radiator and over top to avoid recirculation.
- Additional openings are incorporated over the bumper to increase mass flow rate of air entering the radiator and intercooler.

These modifications are incorporated and simulations are carried out for different conditions.

3.3 Results of Simulation Done for Modified Model

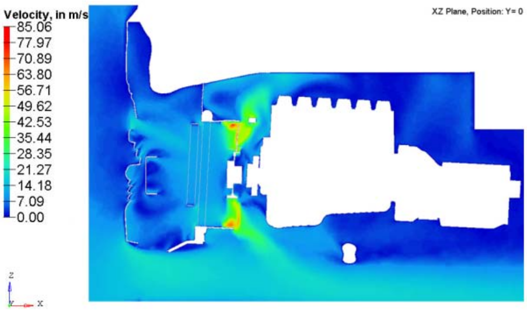

Fig. 9 Velocity contours plotted at symmetry plane for modified model

Modifications done to existing model by incorporating flaps reduced the flow recirculation above the radiator as shown in figure 9. This has led in improvement in the mass flow rate of air entering the radiator. Shroud extension is another cause for improving velocity of air flowing through the engine. Greater amount air sucked from the radiator is forced downwards, hot fluid is mixed with free stream air flowing below the bumper section.

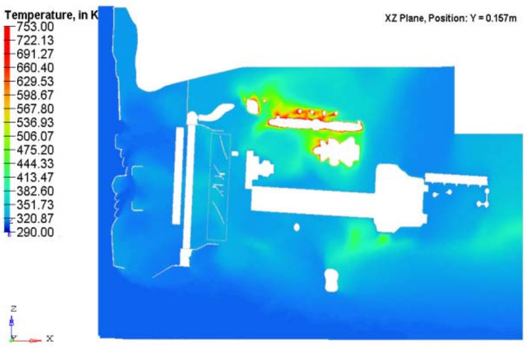

Fig. 10 Temperature contours plotted at exhaust manifold section for modified model

Figure 10, represents the temperature distribution in the fluid plotted at the exhaust manifold section plane, indicating fluid carring the heat rejected from radiator, exhaust manifold and turbocharger. Here the heat carried away from the engine top flows away towards backwards of the truck. Rejected heat also flows away from the turbocharger, this reduces the hot spots near the turbocharger there by reducing the fluid temperature surrounding exhaust manifold. Temperature reduction of 85°C was obtained at the hot spot near turbocharger when compared with the baseline model.

3.4 Comparison of Baseline Results with Modified Results

Simulation for baseline model and modified model has been analysed, velocity, recirculation, temperature and mass flow rate has been compared for incorporating the changes done for improvement.

Flow recirculation zones near the fan shroud has been compared, vector plot colored by velocity magnituted is shown in fig for the baseline simulation (left) and for modified model on the (right).

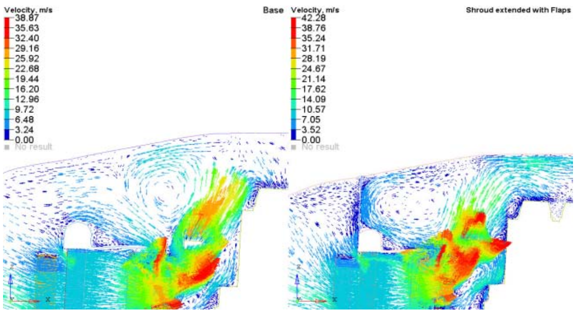

Fig. 11 Comparison of velocity vectors plotted at radiator section

Fan shroud length improvement has reduced major of recirculation's occurring at the fan shroud, this is represented in figure 11. Vector plot for base line model indicates higher recirculation above the radiator. As the shroud extended fluid particles tends to flow towards engine with much higher velocity, this reduces the recirculation occurring.

Reduction in the recirculation from fan reduced the maximum temperature localized on the radiator up to 18%, hence improving radiator performance.

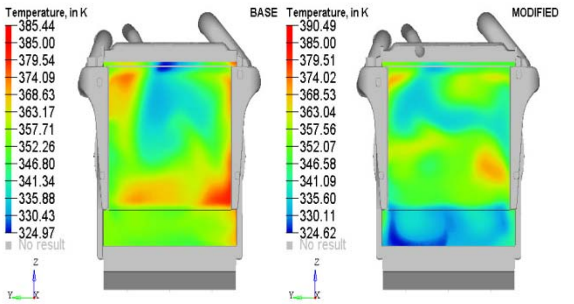

Fig. 12 Temperature contour plotted on Radiator for baseline (left) with modified model (right)

Temperature distribution on the radiator inlet surface has been compared between baseline and modified for studying the hot spots on the radiator. On the baseline model maximum temperature indicated up to 385K whereas on the modified one maximum temperature distribution was up to 360K and also reduced high temperature localized zones. Shroud length improvement increased the velocity of air entering radiator, hence increasing the heat rejection and reducing hot spots on the radiator.

Velocity distribution over the intercooler is also compared and the results of modified model indicated improvement in the velocity of air flowing through the intercooler.

Baseline model consists of a lot of air circulation from the fan back to intercooler; this blocks fresh incoming air leading to reduced velocity. For the modified model the recirculation has been reduced at the fan shroud. Flaps provided at the radiator side try to deflect the air towards intercooler and the top flap tends to deflect the air circulating towards the radiator. Comparison indicated increase in the velocity of air flowing through the intercooler for the modified model up to 3.5m/s. This signified improvement in the intercooler performance.

Different modifications incorporated indicated improvement in the quantity of air entering the heat exchangers. This was studied for each modification and compared with the baseline model. Results are tabulated shown in table 1.

Property		Increase in Mass flow rate (%)		Increase in the velocity (%)	
Component	Conditions	Iteration 1 w.r.t baseline model	Iteration 2 w.r.t baseline model	Iteration 1 w.r.t baseline model	Iteration 2 w.r.t baseline model
Radiator	Max Power	10.52	18.42	10.02	16.29
	Max Torque	10.1	21.36	8.83	17.42
Intercooler	Max Power	12.25	16.19	11.11	16.96
	Max Torque	10.41	24.3	13.78	21.62

Iteration 1: Shroud length improvement.
Iteration 2: All Modifications. (Incorporating flaps, front openings and shroud length improvement)

Table 1. Comparison of mass flow rate and velocity at heat exchangers for different modifications with baseline

Comparison for mass flow rate of air entering through the radiator and intercooler has been done for different conditions. Results were analyzed for different configurations i.e. with baseline model and with modifications. Since the front openings were observed in baseline model air flowing through intercooler and radiator indicated low value. For radiator with shroud extended mass flow rate drastically increased up to 10.52% and for complete modified model it indicated increase in mass flow rate up to 18.2% which has been significant improvement. Shroud extension helped to increase the velocity at intercooler up to 24.3%. This helps when the vehicle is in the maximum torque condition and vehicle velocity is lower. Further simulations were done with complete modifications i.e. along with flaps provided at the radiator and intercooler. This showed that the mass flow rate increased because of additional front openings as well as velocity.

A Comparison of results showed a significant improvement in the mass flow rate at radiator and intercooler which indicated greater amount of cold air entering heat exchangers and thereby improving the heat rejection. Velocity also increased with modifications leading the better heat dissipation rate.

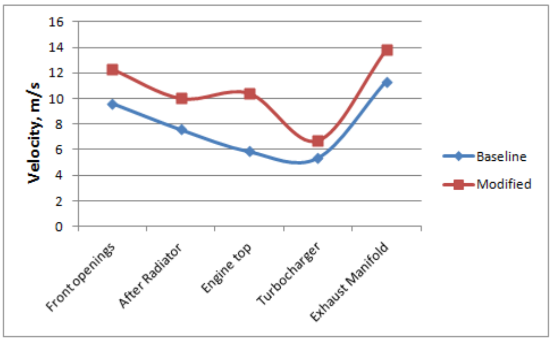

Fig. 13 Velocity monitored at different locations for baseline and modified model

Figure indicates velocity distribution monitored at five different locations. At front openings modified model results showed improvement of 3m/s. Engine top surface showed major difference indicating 4.5m/s improvement from the baseline as shown in figure 13. Front additional opening was an added advantage, for rest other components up to 20% of velocity increased in modified than compared.

Heat dissipation was compared in terms of heat flux rejection from the components with higher surface temperature as shown in figure 14.

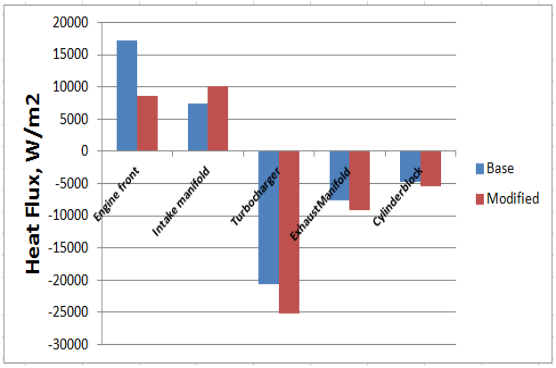

Fig. 14 Heat flux monitored at different locations for baseline and modified model.

Heat source i.e. turbocharger, exhaust manifold and cylinder block side walls are monitored for increase in the heat rejection. For turbocharger, results of modified model showed 15% increase in the heat rejection. This indicated increase in the velocity of air flowing on the engine top portion. Exhaust manifold and cylinder block walls accounted for 10% and 5% for improvement in the heat rejection for modified one.

4. CONCLUSIONS

Underhood flow management for 25T truck has been carried out by cold flow analysis and hot flow analysis by CFD method using commercial software AcuSolve. From this study we found that recirculation zones near the fan and radiator have been reduced. Low velocity of air entering through the intercooler has been identified; modified results indicated increase in of velocity up to 24% at intercooler. Shroud length increase has improved the flow distribution and velocity at the heat source. Mass flow rate of air entering has been monitored in the cold flow analysis, modified results showed increase in the mass flow rate up to 18.5% for radiator and 16.9% for intercooler. Hot flow analysis identified hot spots in the engine compartment and quantified heat dissipation. Modified results indicated an increase in the heat rejection up to 18% from the radiator and 15% from the compartment. Heat flux monitored indicates an increase in heat rejection increased up to 20% from exhaust manifold.

6. REFERENCES

[1] Salvio Chacko, Vinod Kumar, Biswadip Shome, A. K. Aggarwal (2006) '*Underbody simulation for Vehicle Thermal management*', Altair CAE conference, Bangalore.

[2] S.P. Stevens, T.G. Bancroft, S.M. Sapsford,(1999) '*Improving effectiveness of Underhood Airflow prediction'*, Ricardo consulting engineers ltd, Shoreham-by-sea, UK, Paper no: C543/058/99.

[3] H. W. Krus, (1998) '*An Evaluation of the simulated flow through the engine cooling system of the truck*' CYCLONE fluid dynamics, Warmemanagement Tagung, Essen.

[4] Joe Amodeo and Ales Alajbegovic and Wilko Jansen (2006), '*Thermal management simulation for passenger cars – Towards total vehicle analysis*'' Exa corporation at Exa_MIRA, retrived from website www.uib.ac.be .

[5] Ales Alajbegovic, Bing Xu, Alex Konstantinov, Joe Amodeo, '*Simulation of Airflow cooling for different driving conditions*', Exa corporation, Jaguar cars/Land rover, SAE Paper No:2007-01-0766.

[6] Edinilson Alves Costa, '*CFD approach on underhood thermal management of Passenger cars and trucks*', Ford Motor company, Brazil, SAE Paper No: *2003-01-3577.*

[7] Francesco Fortunato and Fulvio Damiano, (2005) ' *Underhood cooling Simulation for New vehicle Development*', Elasis S.C.p.A., PWT, Engine Systems, SAE Paper No: 2005-01-2046.

[8] Martin Wambergnass, (2005) '*Thermal management of Heavy vehicles- A review on identifying issues and research requirements*'', Argonne National Laboratory, London presented at Vehicle Thermal management system 4-conference.

[9] Ganesan, V., (1993) '*Internal combustion engines*', 3rd edition, New Delhi, Tata McGraw Hill Publications.

[10] Thomas Hallqvist, (2008), '*The cooling Air flow of Heavy trucks – a parametric study'* Scania CV, SAE International Journal Volume 1, Issue 1, SAE paper no: 2008-01-1171.

[11] '*Thermal management of the reduction of emission and fuel consumptions in truck'* (2010), technical manual, Behr ltd, Germany, presented at Technical press day.