CFD ANALYSIS OF PLACING COMPONENTS IN THE UNDER HOOD OF A CAR

Praveenkumar. K[1] | Dr.P.Tamil selvan[2]

[1](Thermal Engineering, SNS college of technology, Coimbatore, pk76510@gmail.com)

[2](Thermal Engineering, SNS college of technology, Coimbatore, hodmech@snsct.org)

***Abstract*—*This project reports as the computational study of the aero thermal phenomena in the vehicle under hood compartment as investigated by measuring temperature and convective heat flux. The consequence of the architectural arrangements of electrical and mechanical components on the aero thermal behavior in the under hood compartment. The problem arises from the under hood architecture, specially the positioning of the some components downstream of warmer components in the same air flow. This study makes an assessment of the need for including the thermal objective into the optimization process and also presents an efficient way of performing CFD simulation over the under hood geometry. The objective is to study thermal behavior of the under hood of car by modifying the cooling module distance from the engine components. The distance between the radiator and the engine components are modified horizontally. By increasing distance of placement of the engine from radiator, it shows increase in the mean velocity of the air and creates less resistance to flow when it is compared with base model. Finally, by increasing the distance of engine from radiator increases the temperature of the air downstream from the under hood. The surface temperature components present in the under hood decreases. Generally, additional 10-12 cm is available in the under hood for the placement of the engine block module with respect to the cooling module. However, this space varies from one vehicle to another. Therefore, geometrically there are no potential restrictions if one respects the other allowable distances for an existing car design. However, for a new car design, a choice has to be made if there is a strong downsizing demand of the under hood.*

Keywords*—*Under hood;aerothermal;engine placement ; computational fluid dynamics

1. INTRODUCTION

In recent years, there has been an increase in demand towards the improvement in design of automobiles in particular cars with an effort of achieving higher performance as well as providing better passenger comfort. Large amount of effort in all aspects was invested to improve the aerodynamic behavior of the car in order to attain reduced drag force with improved acceleration and fuel economy. Additionally, advancements were made in the suspension design to improve maneuverability. Also, the engine design was optimized to get better fuel efficiencies and higher power. Thus, a large effort was made to improve the design of each individual component to meet the demands.

Now, placing components under the hood of a car reduces the space. In such compact environments, performance of one individual component may affect the performance of several others in its vicinity. Thus, enhancing the design of individual components alone is not enough to improve the vehicle performance but also their placements under the hood of a car and their interactions with the components surrounding them have to be investigated. An optimization process should be applied to find an optimal placement of these components. This under hood vehicle configuration design problem is a complex multi objective optimization problem. In the past, such multi objective optimization problem was investigated with three major objectives namely; minimizing the height of centre of gravity, maximizing vehicle maintainability and maximizing survivability.

However, the research did not include the thermal aspects of the under hood components of a car. With the evolution of better car designs to cater to the needs of the customer, the under hood space of an automobile car is confined to a much smaller and compact space. But, most of the hot components such as engine, exhaust manifold etc., are placed in this confined under hood space which can adversely affect the overall temperature of the under hood region of a car. Thus, minimizing the overall temperature of the car under hood should be added as an objective to the multi objective optimization problem discussed above.

The under hood thermal management might look simple, but with the engine size being increased for higher power outputs, the under hood space being confined to smaller volumes for compactness and also with more and more components being clustered in to this area, there is not enough space for the air to flow through it. With this extremely complicated and tightly packed environment, it is not easy to carry out experimental studies to predict the temperatures obtained in this region. Also to assist early design changes, it is not always possible to carry out these experimental demonstrations by building prototypes for different designs, as it would increase the design cycle time. Thus, we need to carry out computational methods to understand and improve the thermal behavior of the under hood of a car.

Most designers resort to the technology known as CFD (computational fluid dynamics) to carry out these numerical simulations. This is a powerful tool to model fluid flow and understand the thermal behavior numerically using the finite volume methods.

This tool can also be used to describe and simulate complex thermal fluid flow phenomena such as turbulent flow, flow through porous medium, and flow past heat exchangers. But, integrating CFD simulations with the optimization process is computationally expensive. Thus, an approximation to the CFD simulation should be built for integrating the thermal model with the optimization process.

This project investigate the airflow and thermal behavior in the under hood region of a car by carrying out CFD simulations. Thus, the goal of this project is to come up with a CFD simulation of the complex 3-Dimensional under hood space for different cases, whose results can compared with base model.

2. LITERATURE REVIEW

CFD is a numerical simulation tool to predict the air flow characteristics and its associated thermal phenomena in 2-Dimensional or 3-Dimensional space. The method follows to get the result is that it discretizes the domain space into several small volumes over which the fundamental equations are solved for the velocity and temperature profiles. The equations that are solved over the domain are the energy equation, momentum equation (Navier-stokes equation) and the continuity equation. CFD uses finite volume discretization techniques for dividing the entire domain space into several small volumes. Discretization of the space into smaller volumes is called mesh generation. The size of the cells into which the domain is discretized depends on the users need for accuracy of the solution in that domain.

For the problem proposed in work, as mentioned earlier, we need to perform CFD simulation over the car under hood space. For this, we use the commercial CFD simulation tool FLUENT. FLUENT is the fluid flow simulation tool of ANSYS multi physics software. In the past, there has been an extensive work done on simulating the thermal behavior of the car under hood using various simulate Bancroft et al. 2000 [1] present the importance of the initial geometry preparation by computer Aided Engineering (CAE) tools before running CFD simulations over it. It was shown that there were significant reductions in the time taken to generate a CFD mesh after the CAD geometry clean-up operations were performed. It was mentioned that, however good the quality of the CAD geometry may be, it still needs to be processed before performing CFD simulations over it. Later the CFD simulations were performed on the processed CAD data and were presented in this paper. Validations for the CFD simulations with test data were also presented which showed a good correlation between them.

Huang et al. 2004 [15] had investigated the sensitivity of the size of a flow domain to the air flow rates through the radiator. From the results obtained, a general method for choosing the optimal size of a flow domain to perform under hood air flow simulations using CFD codes was proposed. Finally, the air flow rates simulated by using the proposed optimal flow domain size was compared with the experimentally determined values. Both these data showed a good correlation between them.

Jung and Assai set al 2006 [18] presented an experimental study on Radiators for engine cooling systems, evaporators and condensers for HVAC systems, oil coolers, and intercoolers are typical examples of the compact heat exchangers that can be found in ground vehicles. Among the different types of heat exchangers for engine cooling applications, cross flow compact heat exchangers with louvered fins are of special interest because of their higher heat rejection capability with the lower flow resistance. In this study predictive numerical model for the cross flow type heat exchanger with louvered fins has been developed based on the thermal resistance concept and the finite difference method in order to provide a design and development tool for the heat exchanger From this paper pressure drop vs. velocity curve taken for modeling radiator as a porous media and choose radiator as the heat exchanger.

Chacko et al 2005 [16] studied the significance of the effect of air flow through the radiator. Radiator efficiency was improved by optimizing the air flow through it. The improvement in efficiency was visualized through various numerical simulations that were performed. These CFD simulations were conducted using FLUENT, while the surface and the volume mesh were generated using ANSA and Tgrid, respectively. Later the results were validated with the test data.

Mahmoud khaled et al 2011 [3] presented a methodology for development is towards reducing the size of the heat exchanger volume and weight and increasing thermal efficiency. This projects have shown that a non-uniformity of the upstream velocity distribution increase the heat-exchanger water outlet temperature and thus decrease in the thermal performance.

Mahmoud khaled et al 2014 [4] focused on the optimization of the cooling module. In this result constitute the basis for the new approach of controlling the cooling modulepositioning according to the engine energy requirements. Measurements are carried out on a simplified vehicle bodydesigned based on the real vehicle front block. Measurements are carried out for conditions simulating both the slowdown and the thermal soak phases with the fan in operation. Different fan rotational speeds, radiator water flow, and underhood geometries have experimented. The ultimate aim is to apply the new control approach to a real vehicle so as to reduce the energy delivered to the pump and compressor and therefore to reduce the vehicle fuel consumption.

V B Guolua vink et al 2006 [2] presented a methodology for performing a CFD analysis to predict the air flow and temperature distribution through the vehicle under hood. Through the results obtained, suggestions to improve the air flow through the under hood were made. The method presented was illustrated with the work done over the under hood of a light truck. An under hood simulation model which included all the components relevant for the thermal management like radiator, engine, exhaust manifold, condenser, fans, etc. was used. Instead of modeling radiator and grille in detail, which is complicated, they were modeled as a heat exchanger and porous zone respectively. A standard k-ε turbulence model with Upward Differencing convection scheme with radiation effects was used on a properly refined computational grid. ANSYS FLUENT was used as a simulation tool while the mesh was generated using ANSYS T grid. However, performing the same task over the car under hood is challenging because of its compactness.

Shaolin Mao [5] presented a methodology to carry out computational fluid dynamics simulations for understanding the thermal characteristics of the environment surrounding an under body fuel tank in a passenger vehicle. In this paper, a body-fitted unstructured CFD model of the under body region which included the fuel tank was used. The results for both moving and stationary cases of the car presented in this paper indicated that the major source of the heat transfer to the fuel tank surface was due to the heat convected from the under hood region of the vehicle. The results were validated with the test data from a similar vehicle.

The literature presented describes the challenges that the researchers faced in the past in performing CFD simulations over the complex 3-Dimensional vehicle under hood geometries. Also, suggestions to overcome these challenges were made in the literature.

The general procedure to follow to perform these rigorous CFD simulations over the complex under hood geometry as inferred from the literature is summarized as follows. Initially, the CAD geometry of good quality has to be imported into the CFD simulation tools in an acceptable format. The imported geometry should then be processed by performing the geometry clean-up operations to make it usable for CFD simulations. The processed geometry is then meshes with the desired grid type.

From the results presented in Winnard et al. [17] it can be inferred that the amount of heat transferred by radiation from exhaust manifold constitutes only a fraction of the total amount of heat transferred to the under hood environment. This inference holds true if it is assumed that the results from this paper are applicable to all vehicles.

As presented in Salvio Chacko et al. [16], the radiator can be modeled as a porous model in FLUENT to further simplify and reduce the time taken to perform the CFD simulations without any sacrifice being made in the accuracy of the solutions. The size of the flow domain on which the CFD simulations are made can be chosen according to the method presented in Huang et al. [15].

Finally, from the literature study done, it can be inferred that in the past, researchers aimed at enhancing the automobile manufacturing process by performing early design evaluations with the use of computer-aided engineering tools such as CFD and FEM. This reduced the costly prototyping process for evaluating each new design and significantly impacted the design cycle time.

3. CFD SIMULATION

This chapter discusses the procedure followed in performing CFD simulations using the commercial CFD analysis tool, ANSYS FLUENT over the components and in the domain specified in the previous chapter. The purpose of conducting these simulations was to predict air flow and thermal behavior of the vehicle under hood and thus predict the surface temperature of the components in the under hood compartment. A steady state flow and heat transfer analysis is employed in performing these simulations. Transient simulations were not included as they are computationally expensive and could be traded off, as our prime concern was to simulate the under hood thermal behavior at the most critical working conditions for optimization purposes.

The air flow pattern through the under hood compartment is very complex. Every component in the under hood compartment influences the flow pattern. They also influence the thermal behavior of the under hood compartment. Thus the care has to be taken in modeling the components individually. Modeling of these components is explained in detail below.

A major aspect in the vehicle design process is the design of the front end grille. The airflow through the under hood compartment is primarily dependent on the grille design. Adequate care needs to be taken in modeling the grille to ensure that a proper amount of cooling air is let into the under hood compartment through the grille. Thus, an optimum inlet area is to be specified for the grille design, which ensures adequate air flow through the vehicle radiator core and thus meets the engine cooling requirements under all operating conditions. A schematic of the air flow through the under hood compartment is shown in Figure 1.

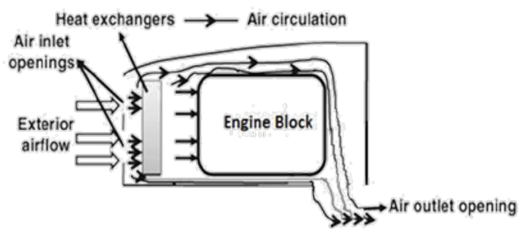

Figure 1: Schematic of the complex air flow through the under hood compartment.

Having considered the complexity of building up a 3-dimensional CAD model of the grille and its associated mesh, it is decided to model the grille as a porous medium. But, with the advancements made in the ANSA and ANSYS FLUENT capabilities, the surface geometry could be included along with the 3-dimensional CAD geometries. Thus, the under hood compartment with the grille is modeled accurately with the right amount of inlet area as a surface geometry. As mentioned above, the under hood compartment is assumed to be adiabatic and so its interactions with the surroundings need not be modelled. Finally, the under hood compartment with the grille was modeled as a thin surface model, which was not included in the meshing step.

As mentioned above, proper regulation of air flow through the radiator core is essential in the vehicle cooling process. The radiator acts as the heat sink for the heat generated from the engine. A coolant circuit travels from the engine compartment to the radiator carrying a part of the excessive heat generated by the engine. This heat is then rejected into the under hood environment as the air flows through the radiator's core.

Over the years, liquid-cooled radiators have become the most preferred radiators and are used in both passenger cars and heavy duty vehicles. In vehicles employed with high-performance engines, the cores of the radiator are modified as flat corrugated tubes to increase the area of heat transfer and thus increase the efficiency of the radiator. This further increases the complexity in modeling the radiator accurately in CFD simulations.

In CFD simulations, the radiator is modeled as a heat exchanger model, to predict the amount of heat rejected by the radiator to the under hood environment. It is also represented as a porous medium, to model the air flow through the radiator core. Viscous and inertial resistance required to model the radiator as a porous medium were calculated and applied.

The following two heat exchanger models are available in ANSYS FLUENT

- Macro model
- Dual cell model

The Macro heat exchanger model either uses the number-of-transfer-unit (NTU) model or the simple effectiveness model for the heat transfer calculations. Whereas, the Dual cell model uses only the number-of-transfer-units model in performing the heat transfer calculations. The Dual cell model is more complicated and computationally expensive as it constructs the solution of the auxiliary fluid flow on a separate mesh, unlike the Macro model, where the flow is modeled as a 1-D flow. These models are used to compute the auxiliary fluid inlet temperature when the amount of heat rejection is fixed and known or to compute the total heat rejection when the fixed auxiliary fluid inlet temperature is known.

In this project, an ungrouped Macro heat exchanger model from ANSYS FLUENT is used to model the radiator. In a standard heat exchanger core, the auxiliary fluid (coolant) temperature is not constant throughout its flow path, along with the direction of the auxiliary fluid flow. As a result, heat rejection from the radiator is also not constant over the entire core. Thus to incorporate this uneven distribution of the heat rejection in the heat transfer calculations, the control volume representing the heat exchanger core is divided into macros along the auxiliary fluid path as shown in Figure. In the Figure shown the heat exchanger core is discretized into 2×4×2 macros. This implies that the auxiliary fluid flows through the heat exchanger core in two passes; each pass is then divided into four rows and two columns of macros. Now, the auxiliary fluid inlet temperature to each macro is computed and then used in the heat transfer calculations. This approach of modeling the heat exchanger core by discretizing it into smaller subdomains called macros provides more realistic solutions for the heat rejection calculations according to ANSYS FLUENT user guide.

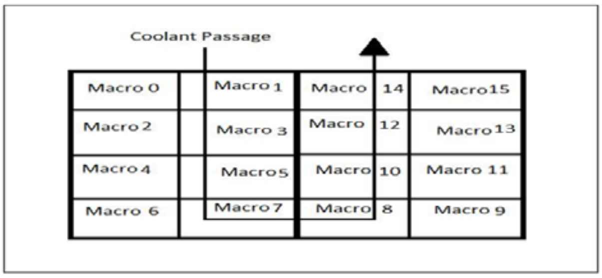

Figure 2: Heat exchanger core divided into 2× 4×2 macros

Having discussed the ease in setting up the problem using the macro heat exchanger model, there are also a few restrictions with using this model which have to be considered. The major limitations in using this model are as follows

The heat exchanger core must approximately be rectangular in shape.

- The direction of flow of the primary fluid must be aligned with one of the three orthogonal axes defined by the heat exchanger core.
- The change in phase of the coolant fluid cannot be modeled and the fluid flow is assumed to be 1-D.

In this project, the radiator modeled as an ungrouped macro heat exchanger model was discretized into the default 2×5×1 number of macros specified in ANSYS FLUENT. These macros are identified in this model with different colors, as shown in Figure. The radiator core model given for this project was approximately rectangular as shown in the Figure below. The direction of flow of the primary fluid is aligned with the x-axis.

As mentioned above, the radiator is modeled using the ungrouped macro heat exchanger model in ANSYS FLUENT. To do so, the following heat transfer data was required.

- The fluid zone representing the heat exchanger core had to be specified. In this project, the radiator modeled as a rectangular core represented the fluid zone.
- Either the fixed heat rejection or the coolant inlet temperature had to be specified and in this project, the fixed inlet temperature of the coolant was specified as 358.15 k.
- The primary fluid temperature also was specified as 298.15 k.
- Then, the “Heat exchanger performance data” and the “Core porosity model” had to be specified.

In this project, the “Heat exchanger performance data” required was obtained from the literature. Dohoy et al. [19] had developed a numerical model using the finite difference method based on the thermal resistance concept to predict the effect of the design parameters on the heat exchanger performance. For the validation of the model, the heat rejection performance of a typical car radiator is simulated for different air flow rates and coolant flow rates. This data is then validated with the experimental data provided by the manufacturer. The experimental data was presented in the paper as a plot of the heat rejections corresponding to different air flow rates and coolant flow rates. The data used in this project is shown below in Figure . In the Figure shown, heat rejection of the radiator was plotted corresponding to three coolant flow rates and three air flow rates.

The experimental data plotted, is represented with symbols, whereas, the simulation data was represented with the dotted lines as shown in Figure . Thus the heat transfer data Table required to model the radiator is built with the data inferred from the above Figure. The screen shot of the Heat transfer data Table used in this project is presented in Figure. The heat rejection of the radiator in watts corresponding to three different flow rates of the primary fluid flow and the auxiliary fluid flow was specified in this Table.

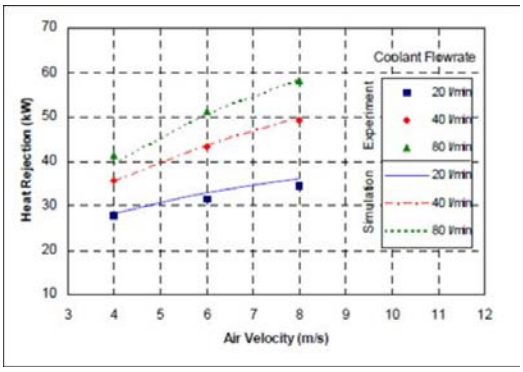

Figure 3 Heat exchanger performance data.

The engine is the prime source of energy for any vehicle. Fuel combustion inside the engine compartment liberates heat energy, which is converted into mechanical energy. The thus generated mechanical energy is delivered to the drive wheels through the vehicle power train and causes the vehicular motion. In typical vehicles, the energy generated by the combustion of the fuel inside the engine compartment is dispersed into three prime areas. Approximately, only about 40% of this energy is converted into mechanical energy and goes to driving the wheels as discussed above. 30% of the combustion energy leaves through the exhaust and heat rejections to the under hood environment. The rest 30% of the energy is carried away by the coolant system and is then rejected to the under hood environment through the heat exchangers, i.e. radiator. The above-discussed breakdown of the thermal energy generated into the three areas is approximate and can vary significantly when the parasitic losses by components such as fans and pumps are taken into consideration.

Thus, one can assume that approximately 20% of the thermal energy generated in the engine compartment is rejected to its surroundings. Also, with the tightly packed under hood environment surrounding the engine compartment, the heat rejected to the surroundings can be critical for any heat sensitive components in its near vicinity. The airflow past the engine acts as a thermal sink for this heat load. Thus proper regulation of the air flow around the engine compartment is crucial.

In CFD simulations, the engine surface is modeled as a non-slip wall. An isothermal constant temperature boundary condition is applied to the engine surface to account for its heat rejection to the under hood environment.

As discussed above, 30% of the combustion energy released leaves the engine compartment through the exhaust system. Thus, the outer surface of the exhaust system is another primary source of heat into the under hood environment. In CFD simulations, the exhaust manifold's surface is again modeled as a non-slip wall and an isothermal constant temperature boundary condition is applied to this surface.

From the results presented in Winnard et al. [17] it can be inferred that the amount of heat transferred from exhaust manifold constitutes only a small fraction of the total amount of heat transferred to the under hood environment. The mode of heat transfer prevalent for this heat transfer is primarily radiation. All other components under the hood primarily transfer heat through convection. Thus, on the basis of the inference made, the radiation effects can be neglected to simplify the thermal simulations over the under hood environment. Thus, in this project, the above-mentioned assumption of neglecting the radiation effects is made.

4. RESULTS

A CFD simulation with the solver parameters and boundary conditions as specified in the previous chapter was performed to provide the user the fluid velocity, temperature and pressure values through the flow domain. In a typical under hood CFD simulation of any vehicle, contour and vector plots for velocity and temperature over different planes in the solution domain are plotted. From the results thus obtained, the need for relocating the components to prevent the under hood thermal environment form being hostile, is assessed.

In this project, CFD simulation over the car under hood region has been setup and run to predict the velocity and temperature distribution for several cases. Also, several CFD simulations were setup by relocating the components in the under hood region, to assess the change in the under hood thermal behaviour. The results thus obtained, were used to assert the importance of under hood` in restraining the under hood thermal behaviour to a safe limit. However, while the flow topology is slightly modified by increasing the distance d, the flow statistics vary significantly. Figure shows the changes with the distance d in the average speed and the standard deviation of the flow velocity distribution in the YZ plane for X = 2 cm.

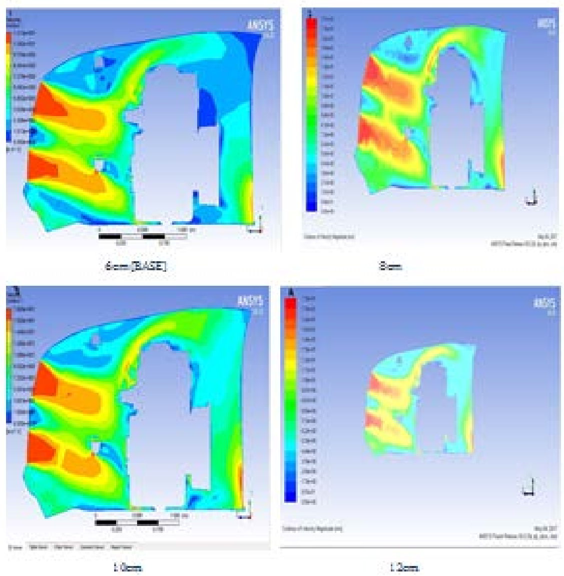

Figure 4: Contours of velocity on a plane through the engine and the exhaust manifold.

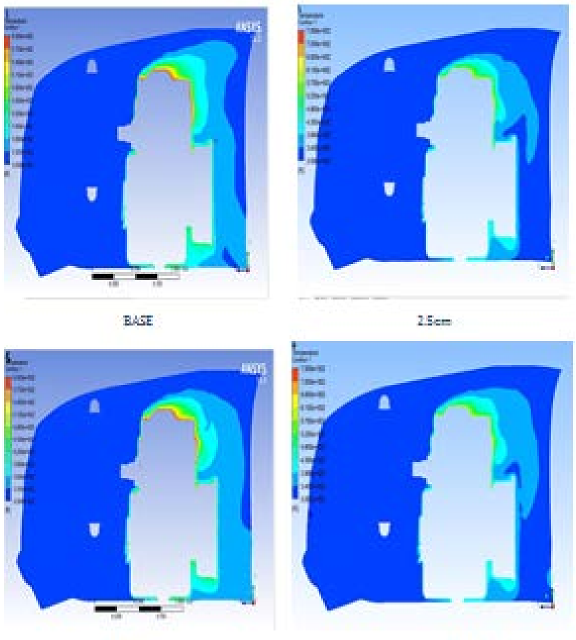

Figure 5: Contours of temperature on a plane through the engine and the exhaust manifold

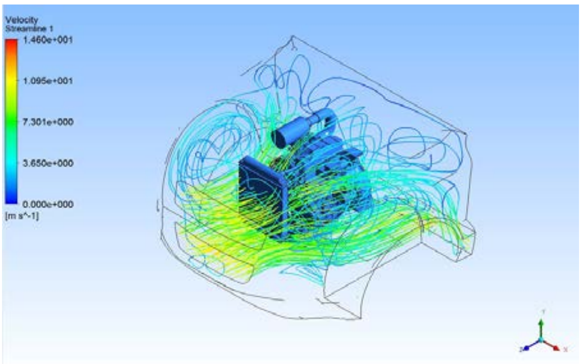

Figure 6 Streamlines launched from the domain inlet, drawn over the plane passing thought the radiator, engine, exhaust manifold and the brake booster.

Applicability of the above result depends, of course, on the availability of the space in the under hood. Generally additional 10-12 cm is available in the under hood for the placement of the engine block module with respect to the cooling module. However, this space varies from one vehicle to another. For example in Audi Q7, approximately 6 cm additional space is available, while for Citroen C6 this space varies between 10 and 12 cm approximately. Therefore, geometrically there are no potential restrictions if one respects the other allowable distances for an existing car design. However, for a new car design, a choice has to be made if there is a strong downsizing demand of the under hood.

5. Conclusion

Implementation of a CFD simulation over the complex 3-Dimensional CAD geometry of a car under hood is presented in this thesis. Appropriate CAD clean up processes are implemented on the initial geometry and the geometry is made suitable for performing CFD simulations upon it. A good quality mesh with 6159538 nodes 13544757 elements is generated. The maximum skewness value of the mesh generated is below 0.94 throughout, indicating that the good quality of the mesh obtained.

Porous medium model and the ungrouped macro heat exchanger model are used to model the radiator. The use of these models significantly simplified the modeling process. Thus, CFD simulations are carried out successfully on the complex under hood geometry by implementing the simplified models for the radiator. The results thus obtained from the simulations performed indicated zones of thermal risk. The prime reason for the formation of these high thermal risk zones in the under hood is due to the improper circulation of air through the tightly packed under hood space. The velocity vector plots obtained for the under hood geometry indicated the presence of recirculation zones which further deteriorated the air flow circulation in the under hood space.

Also, in this project, CFD simulations are performed over four different layouts of the under hood components, to assess the need for relocation of the components in order to keep the surface temperatures of these components in a safe limit. The results thus obtained indicate that the placement of radiator at the different distance from the engine is detrimental for the under hood thermal behavior. Thus, the optimal placement of the components in the under hood space can significantly improve the under hood thermal behavior.

References

[1] T.G. Bancroft, S.M. Sapsford and D.J. Butler, “Under hood Airflow Prediction Using VECTIS Coupled to a 1-D System Model” In Proceedings of the 5th Ricardo Software International Users Conference, Shoreham-by-Sea, UK, 2000.

[2] Guohua Wang, Qing Geo, Tianshi Zhang and Yan Wang “A simulation approach of under-hood thermal Management”, State key laboratory of automotive simulation and control, Jilin University, and college of Automotive Engineering, Jilin University, 130025 Changchum, China, 2016.

[3] Mahmoud Khaled, Hisham El hage, FareedMangi,FabienHarambat, Hassan peer Hossaini,“Fan air flow and heat transfer enhancement of vehicle under-hood cooling system-Toward new control approach for fuel reduction” , PSA Peugeot citroen, Velizy A centre, 2 route de Gisy,78943 Vilacoublay, France, 2011.

[4] Mahmoud Khaled, Mahamad Ramadan, Hisham El hage, Ahmed Elmarakbi, Fabien Harambat, Hassan peer Hossaini, “Review of under-hood arothermal management: towards vehicle simplified models”, Energy and thermo fluid group,school of engineering, Labanese International University LIV,POBOX 146404,Berirut Labanon, PSA Peugeot citroen, Velizy A centre, 2 route de Gisy,78943 Vilacoublay, France, 2014.

[5] Shaolin Mao, ZhigangFeng, EE Michaelides, “Off highway heavy duty truck underhood thermal analysis”, Centre for computation and technology, Louisina state university, Boton Rouge, LA 70803, USA, 2010.

[6] Mahmoud Khaled, Fabien Harambal, Hichamelhge, Hassan peer Hossaini, “Spatial optimization of an under hood cooling module-towards an innovative control approach”, Thermo fluids,complex flows and energy research group, Laboratories de Thermocienetique, CNRS-UMR6607,Ecole polytechnechique, University of nates, rue c.pauc, BP 50609,44306,Nautescedens, France and PSA Peugeot citroen, Velizy A centre, 2 route de Gisy,78943 Vilacoublay, France,2011.

[7] S Seider and F Bet, “Flow and thermal performance prediction for automotive accessory units and their integration into under-hood CFD flow analysis with multi thermal system”, Indes A Gmbbi, Germany.

[8] John R.Pierson, Richard T.Johnson, “Envelopment of automotive battery system capable of surviving modern under-hood environment”, Johnson controls Battery Group, Inc., 5757 North Green Buy Avanue, P.O. BOX 591, Milwaukee, W153201 (USA), 1993.

[9] Mahmoud Khaled, Fabien Harambal and Hassan peer Hossaini, “ Analytical and Embrical determination of thermal performance of louvered heat exchanger- Effects of air flow statistics”, Thermo fluids,complex flows and energy research group, Laboratories de Thermocienetique, CNRS-UMR6607,Ecole polytechnechique, University of nates, rue c.pauc, BP 50609,44306,Nautescedens, France and PSA Peugeot citroen, Velizy A centre, 2 route de Gisy,78943 Vilacoublay, France,2010.

[10] Mahmoud Khaled, Fabien Harambal and Hassan peer Hossaini, “ Under-hood thermal Managements and physical analysis” ,Thermo fluids,complex flows and energy research group, Laboratories de Thermocienetique, CNRS-UMR6607,Ecole polytechnechique, University of nates, rue c.pauc, BP 50609,44306,Nautescedens, France and PSA Peugeot citroen, Velizy A centre, 2 route de Gisy,78943 Vilacoublay, France, 2009.

[11] M. Khaledab, F. Harambatb and H. Peerhossainia, “Temperature and Heat Flux Behavior of Complex Flows in Car Underhood Compartment”, Applied Thermal Engineering Volume 30, pp. 590–598, 2010.

[12] E. Weidmann, J. Wiedemann, T. Binner, and H. Reister, “Underhood Temperature Analysis in Case of Natural Convection”, SAE Technical Paper 2005-01-2045, 2005. I.F. Hsu and W.S. Schwartz, “Simulation of the Thermal Environment Surrounding an Underbody Fuel Tank in a Passenger Vehicle Using Orthogonally Structured and Body-Fitted Unstructured CFD Codes in Series”, SAE Technical Paper, 950616, 1995.

[13] N. Wenyan, “Lift Truck Underhood Cooling Simulation”, Counterbalanced Development Center, NACCO Materials Handling Group, 2006.

[14] K. Srinivasan, G. Woronowycz and M. Zabat, “An Efficient Procedure for Vehicle Thermal Protection Development”, SAE Technical Paper 2005-01-1904, 2005.

[15] K.D. Huang and S.C. Tzeng, “Optimiztion of Size of Vehicle and Flow Domain for Underhood Airflow Simulaton”, Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering vol. 218 no. 9 945-951, 2004.

[16] S. Chacko, B. Shome, and V. Kumar, “Numerical Simulation for Improving Radiator Efficiency by Air Flow Optimization”, Beta-Case International Conference, Greece, 2005.

[17] D. Winnard, G. Venkateswaran and R. Barry, "Underhood Thermal Management by Controlling Air Flow," SAE Technical Paper 951013, 1995.

[18] D. Jung and D.N. Assanis, “Numerical Modeling of Cross Flow Compact Heat Exchanger with Louvered Fins using Thermal Resistance Concept”. SAE Technical Paper, 2006-01-0726, 2006.

[19] D.Allen, M.Lasecki, W.Hnatczuk and R.Chalgren, “Advanced Thermal Management for Military Application”, Proceedings for the Army Science Conference (24th), Florida, 2005.