Research Journal of Applied Sciences, Engineering and Technology 6(24): 4583-4594, 2013
DOI:10.19026/rjaset.6.3472
ISSN: 2040-7459; e-ISSN: 2040-7467

Submitted: January 22, 2013 Accepted: June 21, 2013 Published: December 25, 2013

Research Article

Simulation of Air Flow under the Hood of a Passenger Car Using Computational Fluid Dynamics

[1]Reza Nimtan, [1]Abdollah Khalesi Doost and [2]Navid Madani
[1]Department of Mechanical Engineering, Science and Research Branch, Islamic Azad University, Semnan, Iran
[2]Department of Mechanical Engineering, Bu-Ali Sina, Hamedan, Iran

Abstract: In this study, a method to solve the passing air flow through under-hood by finite volume method is discussed. The flow field existing around a car or passing through it is going to play an important role from different viewpoints. Lateral flow has an important role in fuel consumption, lower emissions, directional sustainability and the wind sound. On the other hand, the internal flow is important from the viewpoint of the good performance of heating systems, air conditioning systems for reducing the temperature of components and thus increasing the life and better performance of components and also engine cooling systems. The study of internal flow is the subject under consideration in the present study. The ultimate goal of this study is to improve the performance of the engine cooling system and decrease the temperature of the components in the space under the hood. In order to achieve the demands, a commercial CFD code for the simulation of air flow under the hood of a passenger car is utilized and finally the method and results of this study are shown.

Keywords: Air flow, car, cooling system, hood, temperature

INTRODUCTION

As we know, the engine in limited space under the hood acts as a heat source which increases the temperature under the hood due to the heat flux. Interestingly, in this case, the low velocity of air in the space under the hood is considered as the main feature of this issue that stipulates further analysis. Through this analysis, we are able to determine the thermal conditions of areas having critical conditions. Determining the critical areas and the amount of heat flux provide a lot of contribution to engineers and designers in line with the motif and design of components, location of many different parts available under the hood, the engine cooling amount, etc.

On the other hand, the airflow through the front end heat exchangers is critical for the cooling performance of a vehicle. Car manufacturers are taking this into consideration when designing a vehicle. Since many contradicting factors are defining the front end design, a clear understanding of the airflow behavior and its influence on the cooling performance is required. Many have resorted to Computational Fluid Dynamics (CFD) to get this understanding (Lawrence, 2001).

The numerical simulation of air flow field in automobile cabin is a heat transfer problem which includes complex heat boundary condition and couple of air and solid (Yang *et al*., 2002).

The study introduces the mathematic model of air flow in cabin dealing with complex boundary condition and expatiates the problem that must be noticed when integer solving method is adopted, the comparison with SIMPLE and SIMPLER arithmetic and the method for dealing with all terms in universal differential function (Song *et al*., 2009).

The simulations include details of the external flow field together with the flow in the under-hood and under body areas (Amodeo *et al*., 2006).

A coupled steady-state CFD and thermal study was undertaken at full-vehicle scale using the Low-Reynolds formulation of the k-epsilon turbulence model, with hybrid wall function modification (Bendell, 2005).

The task is to identify major thermal risks at an early stage in the vehicle design; that is before any prototype vehicle is built and during the evolution of under hood components and external style (Fortunato *et al*., 2005).

It is a reference for improving the security and the comfort of a negative pressure compartment and it is also helpful for the experiment investigation and the optimization design of the air distribution in the negative-pressure compartment (Xiao *et al*., 2007).

Numerical simulation method for turbulent flow in car cabin is a base for evaluation and research of indoor air quality of car cabin. The RNG k-ε turbulent model

Corresponding Author: Reza Nimtan, Department of Mechanical Engineering, Science and Research Branch, Islamic Azad University, Semnan, Iran

was chosen, the method of dividing grid was introduced according to regions and the Monte Carlo method was used to analyze the additional heat flow variation of the indoor solid faces caused by the solar radiation (Gu *et al*., 2008).

A computational fluid dynamics model is established to simulate the defrosting process in vehicles with the air flow velocity field near the windshield quantified. So the defrosting time and effects are investigated and verified by tests (Zhai *et al*., 2007).

The factors that affect the comfortable environment inside the vehicle and proposes parameters for comfortable environment (Peng, 2002).

Therefore, the purpose of the present study was to confirm CFD as a valuable tool for cooling systems' design. In order to serve as a virtual sign-off, the simulation must be able to predict values for critical cooling performance parameters, such as engine coolant temperatures.

MATERIALS AND METHODS

HYPERMESH software: One of the most common mesh software's that is highly spreading among engineers is HYPERMESH software. One of the reasons for the high usage of this software is the capability for matching well with a variety of CAD and CAE engineering and commercial systems. The software is able to create solid geometry simulating, samples partial adjustment, surface geometry simulating, solid mesh generation, crust mesh formation, group mesh formation and automatic generation of intro-surface for engineering applications. Reflecting on all of the above applications has increased general interest of engineers to this software. Another advantage of this HYPERMESH software is that in the software, crust automatic and semi-automatic hexagonal and tetragonal mesh formation is made in a high speed and quality for very high complicated geometry (Lawrence, 2001) (www. hypermesh.com).

CFD thermal analysis of space under the hood by FLUENT software: In this section in order to further clarify the usefulness analysis by CFD, a full sample of space under the hood using FLUENT simulation is displayed. It is assumed that the mentioned sample is inside a 93 km/h wind corridor. For doing computer analysis, STAR CD will be used, which is a computer versatile trading software in Fluid Dynamics simulation. The Software uses finite volume method for solving the transition equations of fluid flow.

The car sample includes CAD data for all surfaces from the front of the car to B pillar which is meshed and it involves the metal plate outside the car, CRFM (Condenser, Radiator, Fan and Module), under-hood components and motor container surfaces and wheels.

Due to the complexity of geometry and amount of available information, all meshes were made of the tetrahedron cells. Star-CD ability to manage unstructured meshes and various types of cells brings possible usage of this cell type that provides the advantages of more speed mesh generation. The tetrahedron cells are generated by an automatic mesh generator. In this method, mesh production will begin from a triangular surface mesh. The size of the mesh is varied from 10 mm in the car front terminal to 30 mm in regions away from the front of the cooling pack. 3D mesh consisted of approximately 1.5 million cells. Figure 1 shows a part of triangular surface mesh.

Fig. 1: Under-hood components triangular surface mesh

Table 1: Outcome of air flow rate

	Air flow rate (m^3/min)			Difference
	CFD	CFD	Test	CFD/test (%)
Fans	Off	On	On	
Radiator	6.6	36.0	38.2	6
Condenser	5.3	30.6	31.2	2
Intercooler	0.9	2.5	2.6	5

The physics of flow arises from equations solution of steady flow, incompressible, isothermal and turbulent. The standard k-ε model with wall function is used for modelling the turbulence. Geometric details of radiators such as pipes and fins are not sampled. Instead, the bulk of the cells related to the nucleus follow. The following equation is used for the flow resistance diffusion:

$$\frac{\partial p}{\partial x_i} = -\left(\alpha_i \left| u_i \right| + \beta_i\right) u_i$$

The α_i and β_i coefficients are determined by the user in any direction. The coefficients are derived from the external curvein terms of air pressure flow for each radiator.

Boundary conditions are consisted of corridor entrance (input velocity, turbulence intensity and length scale), corridor output (zero pressure and zero gradient for turbulence), corridor walls and ceilings (symmetrical plate) and corridor floor (a movable wall). Although this type of boundary condition for corridor walls, ceiling and floor are not in consistent with reality, but due to a lack of specific solution for boundary conditions in this area, these boundary conditions will be used.

Results of accurate test:
Simulation conditions:

- Vehicle speed is 34 Km/h and the fan is off
- Vehicle speed is 34 Km/h and the fan is working with maximum speed

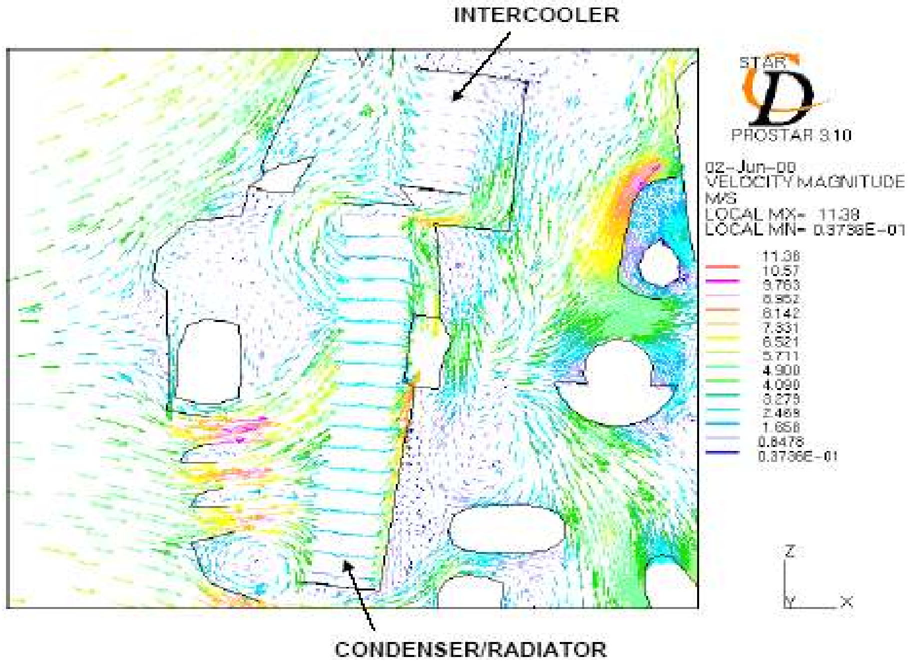

Fig. 2: Car central line plane, velocity map (34 Km/h, fan on, basis sample)

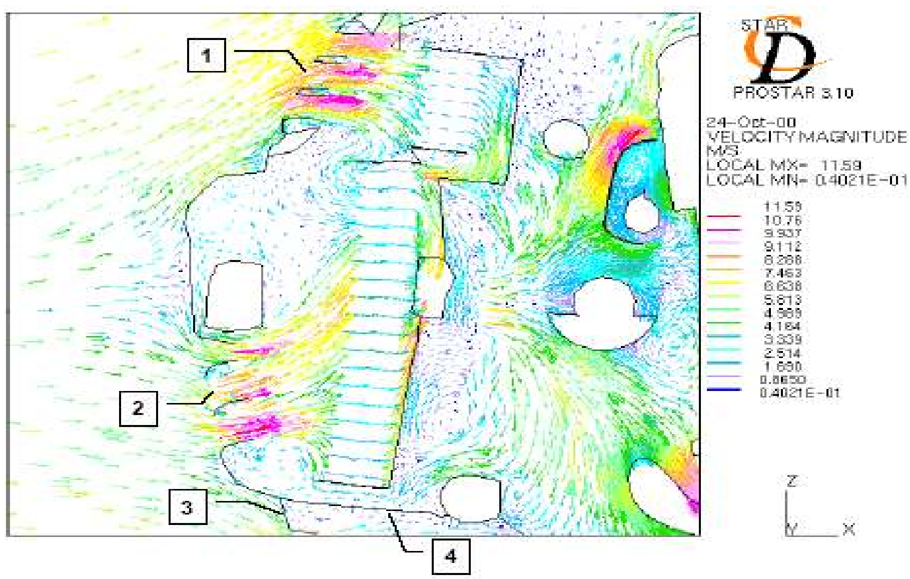

Fig. 3: Car central line plane, velocity map (34 Km/h, fan on, a modified sample)

A similar physical test in the wind corridor with a maximum speed of the car fan was done. The amount of volumetric air flow rate passing through the heat converters was measured:

As it is clear from the Table 1, there is a strong correlation between the data leading to an increase in the confidence level to precede a virtual simulation of car.

In a typical application: When the basis sample was made, any improvements in the model will be implemented and quick responses can be acquired. More than 10 simulations is done with the improvements in CRFM components, front terminal design, graphic deflected air and some other solutions to increase the air flow rate. The sample car has been tested in various speeds; a wide range of car operating conditions is covered. Figure 2 and 3 show a sample of improvement in flow rate after geometry changes in the simulated sample (Power COOL, 2006).

After implementing modifications, there was an increase in air flow rate for radiator from 36 m^3/min

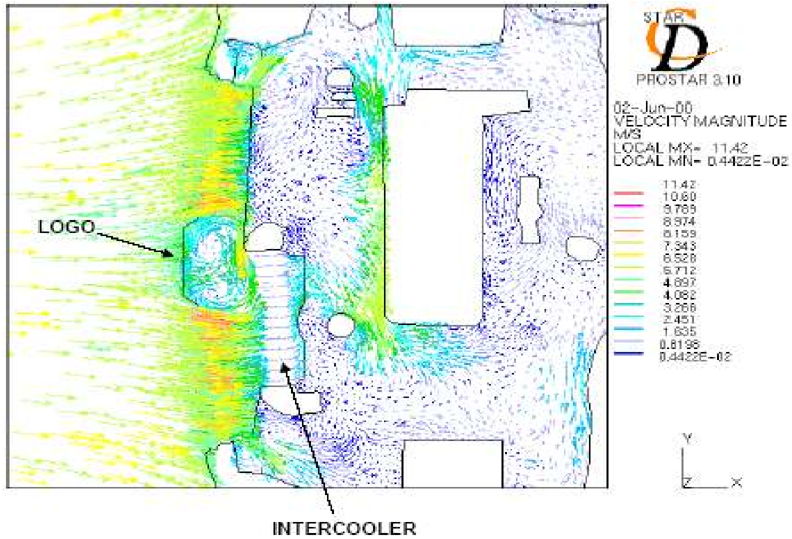

Fig. 4: Intercooler horizontal plane velocity map (34 km/h, fan on, basis sample)

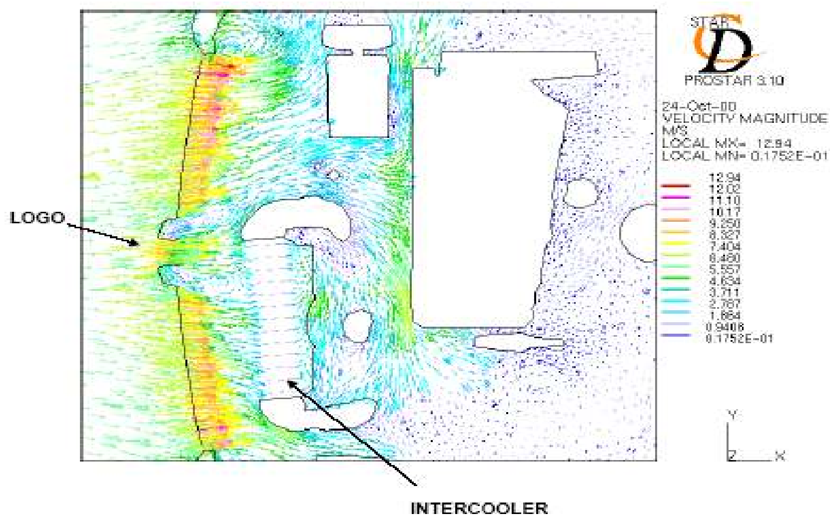

Fig. 5: Intercooler horizontal plane velocity map (34 km/h, fan on, basis sample)

amounts to 42.1 m^3/min, in condenser from 30.6 to 36.9 m^3/min and in intercooler from 2.5 to 2.8 m^3/min. In the front of upper window (Fig. 3 -number 1) additional opening will provide more air intake for intercooler. In order to reduce vortex form at ion in the back lower front-window (Fig. 3 -number 2), the profile was changed. Moreover, side and lower of air deflectors (Fig. 3 -number 3, number 4) will help increase air flow passing through the radiator and condenser.

Figure 4 and 5 indicate improved cooling air flow passing through the intercooler due to a decrease of vortex formation in the back of car logo.

Study on air deflectors was performed at a speed of 150 km/h. A deflector effect of air at this speed is much higher. Figure 6 shows the three air deflectors in basis sample. In this case, there was an increase in air flow rate in the radiator from 54.3 to 65.1 m^3/min, in the condenser from 51.3 to 64.9 m^3/min and in the intercooler from 5.1 to 5.5 m^3/min.

Based on this simulation, we are able to observe and portray the details of flow field which are very difficult by real test to get. This subject is noteworthy in helping further understanding of air flow circumstances in a car (Alajbegovic *et al*., 2007).

CFRM concept and its advantages over the CRFM for automotive thermal systems: Delphi Automotive Systems has analyzed a (CFRM) sample including Condenser, Fan, Radiator and Module in the form of automotive thermal system. Results of this study were measured with a conventional (CRFM) including Condenser, Radiator, Fan and Module. The comparison showed that for a passenger car, under-hood temperature in new CFRM arrangement is more than

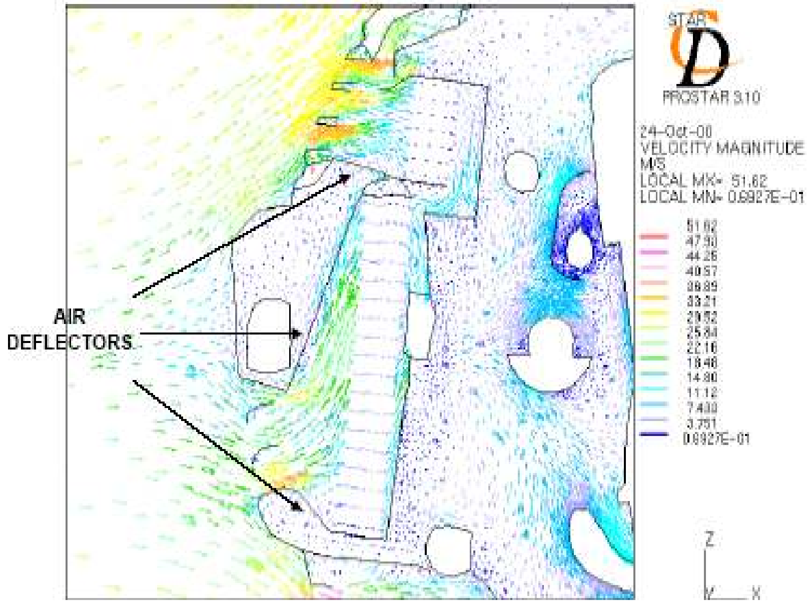

Fig. 6: Car central line plane, velocity map (150 km/h, fan on, basis sample with air deflectors)

10° lower than the traditional CRFM While in both cases the fan speed was 2500 rpm. The main reason for this is higher mass flow rate sucked by the fan in the new CFRM arrangement. In Equal mass flow rate conditions for both CFRM and CRFM under-hood temperature will be the same while in the mentioned conditions, power consumption in CFRM is 19% lower than the CRFM because the CFRM fan speed for the same flow rate is lower than CRFM. The comparison of the performance of CRFM and CFRM with regard to a fall in pressure of the heat converters used in the truck has also resulted in the same findings.

In the old samples, fan was located on the back of condenser and radiator, which is shown by CRFM abbreviation. Pusher Fan arrangement (in the case, fan is in front of the package), initially used as an auxiliary cooling fan. One criticism into this Pusher Fan concept is that a fan under normal condition in a low pressure environment is more efficient. Moreover, the quality of passing air flow through converter in Pusher Fan arrangement is not better than old arrangement CRFM, whose fan is located in the back of the package. Delphi Automotive Systems has introduced a new arrangement in which fan is located in the centre of the package. In other words, the cooling package arrangement for motor power includes Condenser, Fan, Radiator and Module which is abbreviated by CFRM. The test done in this area indicated that CFRM has better performance compared with the traditional arrangement CRFM.

The purpose of this study is to investigate whether the preference of CFRM package in comparison to CRFM that has been seen by Delphi Automotive Systems shows the same behavior or not when it is installed on the vehicle. To achieve this purpose, analysis of automotive thermal systems were done to study and compare the thermal performance of the car with CFRM and CRFM arrangements.

To do this, the first step is a description of system thermal analysis framework. Finally, the result and analysis are displayed, discussed and then a conclusion will be drawn.

Vehicle thermal system analysis: In the past, due to limited computational resources, vehicle thermal analysis has been done by focus on particular sectors, for example, under the body, under the hood and the front end of the car. With increase in computer resources and computational and technological progress, all cars today will be calculated as an integrated system of simulation and analysis of automotive thermal system for thermal environment.

In Thermal analysis of automotive systems, heat converters (condenser and radiator) are still being sampled in spite of having a lot of problems such as geometrical solution of the heat converter with a very small size and the flow passing through it. In this method of construction, instead of separately considering and simulating each heat converter, only the effect of the heat converter on flow field is considered and sampled.

For air field flow rate, radiators are displayed as porous media in which the drag force is imposed by the air passing through it. Due to drag force effect, the turbulent air flow passes through the heat converter with a fall in pressure. Pressure drop is a function of air flow that passes through the heat converter. Figure 7 indicates this phenomenon.

As it is seen, for a given radiator, the greater rate of the air flow passing through, the greater the pressure

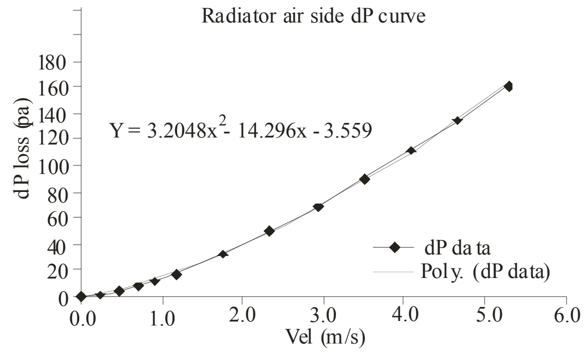

Fig. 7: Heat conventer pressure drop curve

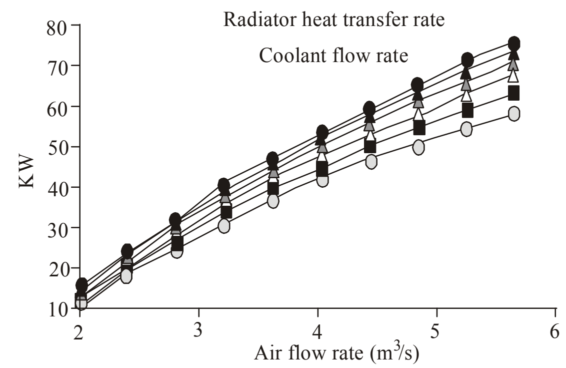

Fig. 8: Thermal characteristics of heat converter

fall. For the thermal field of air flow, heat converter is displayed by the volumetric heat source which injects the heat to the input air that passes through the heat converter (The heat is created by heat converter getting from the engine cooling system). The amount of heat flow which is given to input air flow depends on air mass flow rate and mass flow rate of cooling air passing through the heat converter.

Sample performance heat curve of heat converter shows that the amount of heat repel rate, depends on the air flow rate and coolant flow as it is displayed in Fig. 8. It shows that the rate of heat repel is increased when there is an increase in air flow and coolant flow rate. Both curves of pressure drop and thermal performance of the heat converter can be achieved by a test. These curves are also prepared and presented by the heat converter manufacturers.

In calculation, the flow field is sampled directly due to circulation fan. However, if it is necessary to solve the time average, multiple rotating frame of reference is used, or if necessary, a quick achievement sliding mesh methods are used. Both methods have been well-tested and are available in the commercial CFD codes such as Star-CD code and fluent code (www.cd-adapco.com, www.FLUENT.com). Due to the geometry of fan blades and their velocity, the computational methods mentioned above can produce

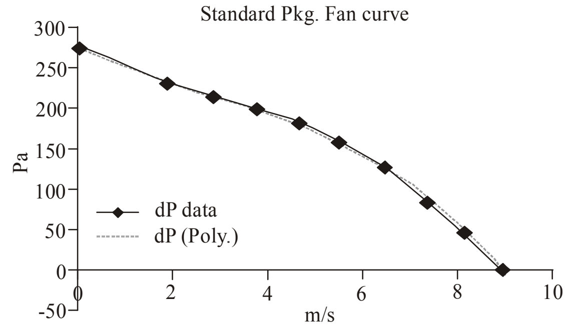

Fig. 9: Fan performance curve

the flow field around the fan blades and the fan performance indicator. In other words, there is an increase in the pressure based on flow curve and fan efficiency. However, the detailed geometry of the fan blades is often unique and it cannot be easily achieved. As a result, fan is sampled by using momentum source. In this simulation method, the same as heat exchanger simulation, only fan effect on air flow was considered and simulated. Simulation is obtained by adding source sentences in momentum equations in such a way that the solution of momentum equations can generate fan pressure increase in terms of flow rate.

As it is shown in Fig. 9, this relation can be obtained by fan performance test or through the ran manufacturer's. Another advantage of this simulation method is using less computational cell analysis and consuming less time because the geometry of the fan blade does not need to be computed.

In the analysis of ahead vehicle thermal system, other under-hood and under body components in the sample is considered and the field flow and heat around these parts are calculated. In one sample, it was assumed that the components are regarded as the 100 mm or more. Under these circumstances, the final sample was composed of 100 pieces. The structure of the above-mentioned sample had a high flexibility through which any other heat sensitive samples can be embedded in to. Final sample was placed over a range of computational length of 23 m, 10 m width and 5 m height.

In improved numerical analysis, the surface of car parts were simulated by discrete triangular mesh. The size of the triangular meshes changes with respect to the desired parts. Mesh sizes of these samples are about 10 to 15 mm. In total, more than 200,000 surface elements were used. Fluid discrete tetrahedron mesh was used in computational interval. Triangular mesh structure or Tetra for making discrete surface causes complex geometry in the most cars to be more easily and quickly displayed. Volumetric mesh number depends on the surface element sizes and the range of computational efficiency. In improved analysis of thermal system, more than two million tetrahedron cells were used for the fluid.

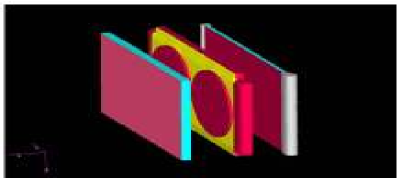

Fig. 10: CFRM arrangement

Fig. 11: CRFM arrangement

Commercial CFD code Star-CD was used for the analysis of this research. The code has a finite volume which takes advantage of the unstructured grid system and most industrial complex analysis were solved using this commercial code.

To solve the velocity field, turbulent flow field was sampled by k-ε. Both the standard k-ε model of Launder and Splading as well as Shih, *et al*.'s k-ε model was used. The results of their study showed a similar result in the under-hood area-the area in our study. For solving the thermal field, all three mechanisms of transferring heat including convection, conduction and radiation were considered in the analysis.

Heat transfer motion due to fluid turbulent convection were sampled by Reynolds Analogue and assuming Prandtl Turbulent number as a fix number. Thermal conductivity of the fluid and the solid was calculated directly. Radiation heat transfer was calculated using the Pearson coefficient method. Heat transfer was also considered at the same time between interface of the fluid and solid thermal components. Also, the effects of the heat exchanger were considered on the thermal field by adding thermal energy sources to energy equations.

The equations of velocity field and thermal field were resolved by coupling method. In this method, thermal field is influenced by the velocity field via convective heat transfer and velocity is influenced by temperature indirectly via the following Eq. (1):

$$\rho = p_a / RT \tag{1}$$

p_a = Atmospheric pressure

The equation is required to describe the physical principles that lead to differences in performance between CRFM and CFRM arrangement.

According to the above analysis and the number of aforementioned cells, the first calculation took approximately 16 h while 8 processors were considered for this function. The reason for this result was that numerical solution had been started from an arbitrary field. It is worth mentioning that any further analysis taking place as a result of changes in operating conditions such as vehicle speed or fan speed are required takes less than 2 h.

DATA ANALYSIS

Both the CFRM and CRFM arrangement, which are given in Fig. 10 and 11, were tested in the analysis of vehicle heating systems.

For both arrangements, it is assumed that functional characteristics of components such as pressure rise curve, pressure drop curve and heat dissipation curves for the heat exchangers are the same. For both arrangements, condenser location will remain the same. The overall dimensions of the package that is taken in vehicle velocity direction remains constant.

Furthermore, the final sample for both CFRM and CRFM arrangement is installed on a sample belonging to a SEDAN from General Motors Company. In this case, it is assumed that vehicle speed is 10 m/s (22 mph) at 40°C in ambient air.

In improved analysis, the engine operating conditions, power transmission system and car load by the rate of heat repel from engine and the temperature of engine surface and car exhaust system are specified. In simulation performed in this study, the same amount (the rate of heat loss and temperature levels of the engine surface and exhaust system) for both arrangements, i.e., CFRM and CRFM are used.

At first, computing is done with fan speed of 2500 rpm. The average temperature of radiator output for CFRM was 105°C, while the value for CRFM was 115°C. Vehicle under-hood space air flow in CFRM arrangement was 10°C cooler than the old CRFM arrangement. To better illustrate the under-hood space temperature, temperature distribution in symmetric plate (y = 0) for both CFRM and CRFM arrangement are given in Fig. 12 and 13, respectively.

Similarly, the temperature distribution from a parallel plate to the ground (z = 400) has been shown in Fig. 14 and 15 respectively for both CFRM and CRFM arrangement.

These figures show that the temperature maps at a specific temperature condition are in 85 to 120°C range. Considering the above figures, it is clear that the space under the hood where the CFRM arrangement is used has more flow of cool air and accordingly we should less worry about the temperature.

The process of the analysis is in such a way that both arrangements, CFRM and CRFM repel the same amount of heat. Therefore, equal heat level is injected

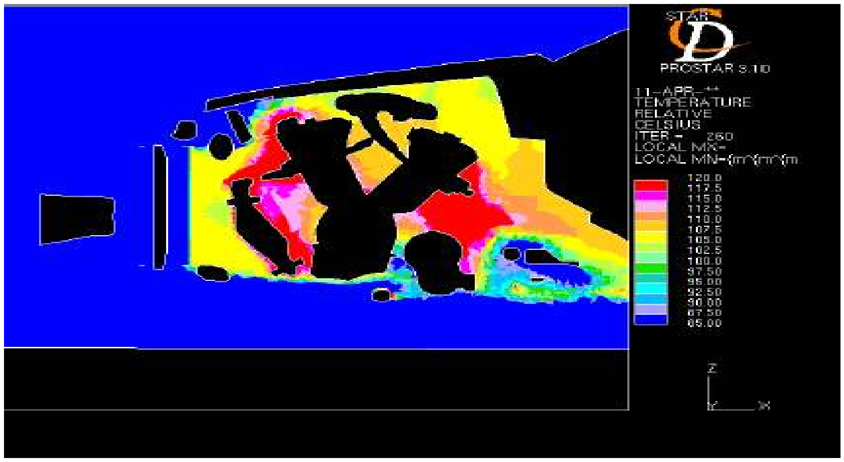

Fig. 12: Under-hood temperature distribution in symmetric plate (CFRM arrangement)

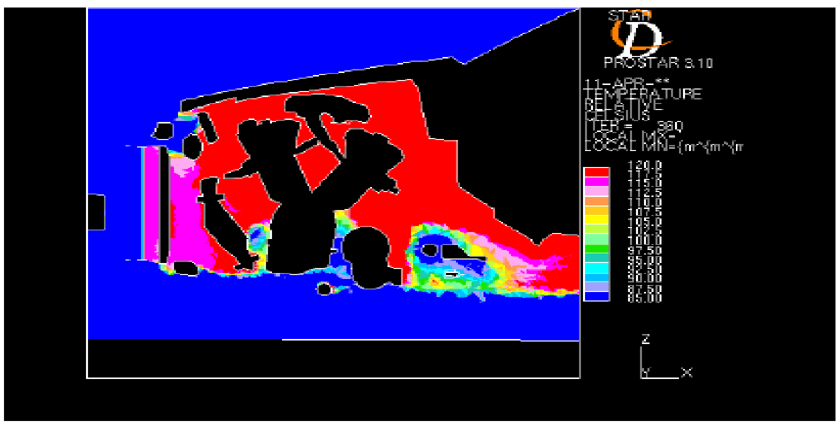

Fig. 13: Under-hood temperature distribution in symmetric plate (CRFM arrangement)

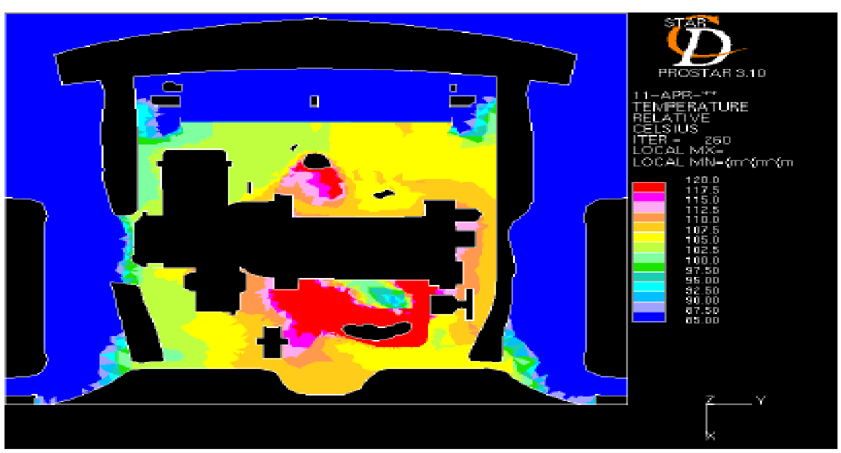

Fig. 14: Under-hood temperature distribution in plate z = 400 (CFRM arrangement)

to the input air flow. When inlet airflow attracts the heat, air flow temperature is increased. Since the temperature is constant at 40°C, the outlet temperature of the converter for CFRM arrangement must be lower because inlet air mass flow is more than that.

In this analysis, air mass flow in CFRM arrangement is equal to 0.808 kg/s, whereas the value for CRFM is 0.696 kg/s. Thus, for 2500 rpm fan speed case, CFRM arrangement transfer 16% more mass flow rate than CRFM arrangement. More mass flow rate of

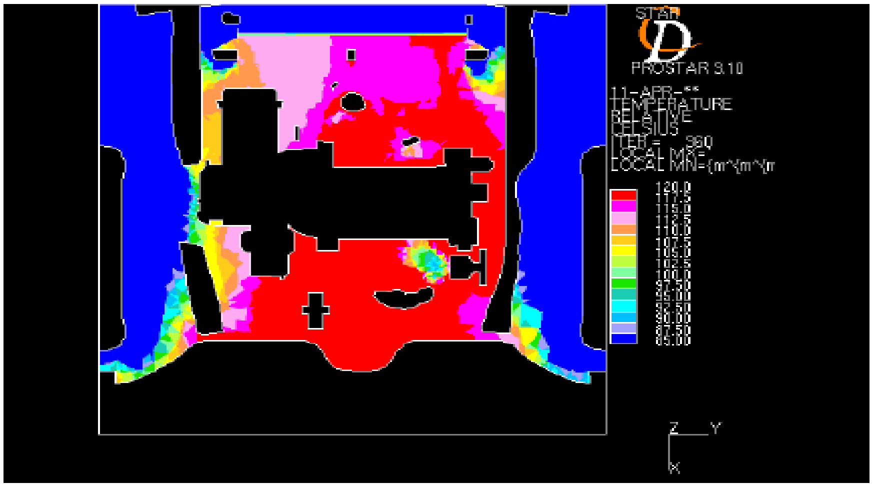

Fig. 15: Distribution of under-hood temperature plate z = 400 (CRFM arrangement)

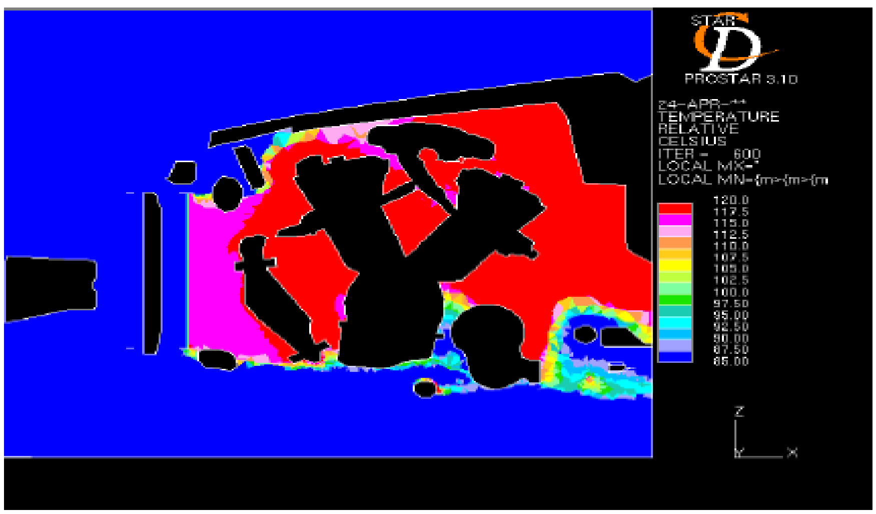

Fig. 16: Under-hood temperature distribution in symmetric plate (CFRM arrangement, 2160 rpm)

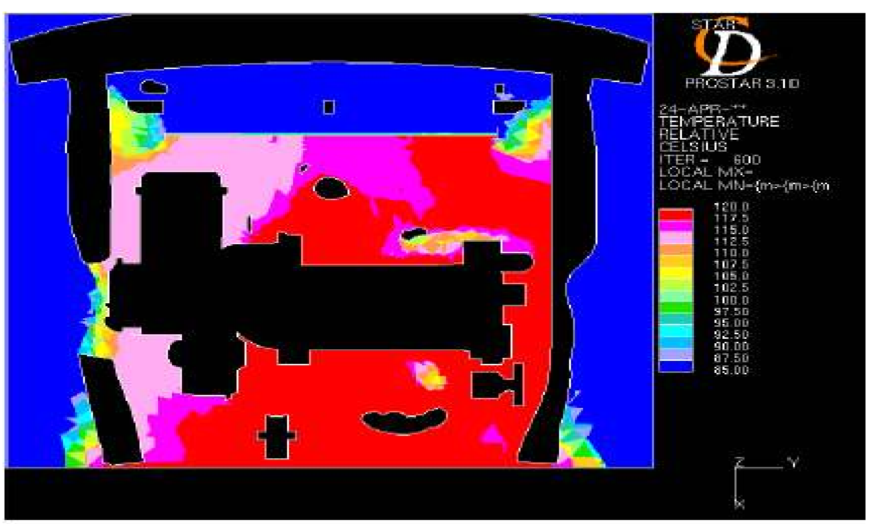

Fig. 17: Under-hood temperature distribution in plate Z = 400 (CFRM arrangement, 2160 rpm)

CFRM is due to fan location before the radiator. Therefore, it functions in an environment with lower temperature in which the air density is higher.

It should be noted that because the air is more dense, fan power consumption in CFRM is more than fan power consumption in CRFM in an equal speed. To better understand this issue that lower under-hood temperature in CFRM is due to higher power consumption of the fan, computer calculations were performed for car heat system in different fan speeds. The fan speed during CFRM analysis was changed in such a way that it provides an equivalent mass flow rate conditions. It was found that the mass flow rate of the fan at speed 2160 rpm in CFRM arrangement is equal to the mass flow rate of a fan at speed 2500 rpm in CFRM. Mass flow rate of both fans is equal 0.696 kg/s. In the case of 2160 rpm fan speed in CFRM arrangement, average temperature of the radiator output is 115/8°C. This amount is half a degree lower than the radiator average output temperature in the CRFM arrangement with 2500 rpm fan speed. Temperature distribution in the under-hood area in CFRM arrangement around 2160 rpm fan speed for y = 0 and z = 400 plates are shown in Fig. 16 and 17, respectively. Obviously, the heat under the hood for current environment (CFRM arrangement and fan speed 2160 rpm), is quite similar to the CRFM arrangement and 2500 rpm fan speed. The overall similarity in the results of the same mass flow rate in each case caused both arrangements to attract the same amount of heat from the engine. A slight deviation is seen due to inappropriate spread speed around the radiator in CFRM arrangement. It should be pointed out that the power consumption is reduced 19% in CFRM with 2160 rpm fan speed in comparison to CRFM with 2500 rpm fan speed.

According to Fig. 17, a heat converter pressure drop curve is a typical for heat converter curve for a car. Therefore, the criteria which have been taken into account in this study for both CFRM and CRFM arrangement is only suitable for cars. A similar analysis was done to measure the performance of CFRM and CRFM arrangement on the truck. A truck radiator is usually thicker than a car radiator. Yet, pressure drop curve for truck radiator is similar to Fig. 7. Thus, sampling the effect of truck radiator on passing air flow is the same as car radiator effect on passing air flow. The analysis done for the higher pressure drop of the radiator resulting from the greater thickness of the radiator shows that the performance of CFRM and CRFM arrangement is similar to car radiator analysis as it was already mentioned.

RESULTS AND DISCUSSION

Two sets of results were reported for CFRM, one for fans speed of 2500 rpm and the other one for 2160 rpm fan speed. In 2500 rpm fan speed, low temperatures were obtained from under-hood space. Of course, fan consumed more power in this case. In the case of fan speed 2160 rpm, the air mass flow is the same as a fan mass flow of a 2500 rpm fan that works in CRFM arrangement. As expected in this case, under-hood air temperature is the same for both arrangement, but CFRM arrangement use 19% less power than CRFM arrangement. In this study, that CFRM arrangement is able to provide the same temperature distribution by consuming less power can prove the superiority of CFRM arrangement in comparison to CRFM arrangement. It should be noted that this is also true in the case of trucks.

In CFRM arrangement, when the fan speed reaches to more than 2160 rpm, the mass flow rate and fan power will increase accordingly. Increased mass flow rate improve temperature distribution in the space under the hood which is due to a decrease of air temperature average in radiator outlet. In this section, fan speed in CFRM arrangement has been estimated equal in case of using same power as 2500 rpm fan speed in CRFM. The speed amount is obtained by trial and error method is equal to 2300 rpm. Since this amount is greater than 2160, then the mass flow rate is more than 2160 rpm in the CFRM arrangement. Therefore, the radiator outlet temperature would be lower in the present case. Therefore, 2160 rpm fan speed could be used in CFRM arrangement, which has the same temperature distribution but lower power consumption compared with the CRFM, or we should use a new fan speed having the same power with the power of a fan speed of 2500 rpm in the CRFM arrangement. However, it has a better temperature distribution in the space under the hood. Of course, the fan can be applied to these two modes under those circumstances resulting in lower power consumption and improving the temperature distribution under-hood space.

It should be noted that the calculation that has been done so far is for the cases in which the vehicle speed is 10 m/sec (36 km/h). In the higher speed of the vehicle like 79 km/h (22 m/sec), the fan causes a decrease in pressure rather than being a source of pressure, or Wind Milling Effect. Since the fan in CFRM experiences more dense and also cool air, in those circumstances in which the same pressure drop curve is considered for both CFRM arrangement and CRFM arrangement, pressure drop in the CFRM fan will be more than CRFM fan. Fan pressure difference between the two arrangements is estimated to be 15%, which was obtained based on the temperature difference between the two arrangements in fan location. In speed 22 m/sec, if we only consider the fan pressure drop caused by Wind Milling Effect, the amount will be 10 to 20 Pa which is negligible. Fan pressure difference in CFRM and CRFM will be even less than this amounting. It means something close to 2 to 3 Pa. In this speed, the dynamic pressure head duo to car velocity is 250 Pa and the pressure drop of the heat converter is about 150 Pa. Thus, although the CFRM fan compared with CRFM has more pressure, the difference is negligible.

A basic assumption in the comparison of fan function in CFRM and CRFM arrangement is using the same performance curve pattern for both proprietary functions. Fan performance curves will depend on the fan working environment. Therefore, it is expected to have different performance curve for both CFRM and CRFM arrangement although identical fan fins is used in both fan patterns. Typically, a fan will have better performance if it is able to control inlet and outlet airflow. The old CRFM arrangement inflow can be controlled by converter which acts as a stator. In the CFRM arrangement, both the inlet and outlet air flow can be controlled by condenser and converter which act as blower stators. Therefore, the analysis which has done in this study is not far from reality.

In this analysis, it is assumed that both CFRM and CRFM arrangement have been well-sealed and air can flow only from the front. Any leakages which may have led to a deviation in performance and may reduce or reverse performance difference between the CFRM and CRFM.

One area of concern in the test is the side effect of incomplete valve grinding during vehicle operation in idol speed. In the tests which have been done on certain vehicles and comparing the new and old arrangements has been taken into account. It is observed that in new CFRM, the warm air in the under-hood area shows a strong desire to return to converter location and the Pusher-Fan arrangement can be regarded as the main reason. Therefore, the CFRM arrangement performance in idol speed and incomplete valve grinding in a car requires more analysis and attention.

CFRM arrangement has other advantages in addition to better performance in terms of under-hood thermal management. For example, since the fan works at low temperature, there is almost no concern about the heat. Fan cool working is also leads to the centre of fan can be considered as a suitable place for installation of components which are sensitive to heat.

CONCLUSION

After the study was done in the field of thermal management of space under the hood, we concluded that the following steps should be followed for analyzing the flow and heat transfer in space under the hood.

The first stage includes the geometric production of space under the hood by one of design simulation software's like Catia, Solid work, Pro-Engineering, etc., which should be done by CAD. Given the enormous potential of Catia software in this part, it was used for geometric simulation of the under-hood space.

The second phase is mesh generation for the produced geometry that was done by different software. Assessing the pros and cons of the available software's discussed earlier indicates that HYPERMESH software might be a good option with regard to a high volume of work in this section.

The third phase is analyzing the air flow and meshed geometric heat. Two CFD popular softwares including Fluent and Star-CD were used for the analysis. Due to the extensive experience of the fluent software, it has been used for flow and heat CFD analysis of under hood space.

REFERENCES

Alajbegovic, A., B. Xu, A. Konstantinov and J. Amodeo, 2007. Simulation of Cooling Airflow under Different Driving Conditions. SAE Technical Paper No: 2007-01-0766, Doi: 10.4271/2007-01-0766.

Amodeo, J., A. Alajbegovic and W. Jansen, 2006. Thermal Management Simulation for Passenger Cars: Towards Total Vehicle Analysis. Exa Corporation at Exa_MIRA. Retrieved from: website www.uib.ac.be.

Bendell, E., 2005. Investigation of a Coupled CFD and Thermal Modelling Methodology for Prediction of Vehicle Underbody Temperatures. SAE Technical Paper 2005-01-2044, Doi: 10.4271/2005-01-2044.

Fortunato, F., F. Damiano, L. Matteo and P. Oliva, 2005. Underhood Cooling Simulation for Development of New Vehicles. SAE Technical Paper 2005-01-2046, Doi: 10.4271/2005-01-2046.

Gu, Z.Q., Q.C. Meng, Y. Yang and Y.B. He, 2008. Research on numerical simulation of air-flow field in car cabin [J]. J. Syst. Simul., Retrieved from: http://en.cnki.com.cn/Article_en/ CJFD TOT AL-XTFZ200807013.htm.

Lawrence, V., 2001. Underhood Airflow Simulation of a Passenger Car Using Computational Fluid Dynamics. SAE Technical Paper 2001-01-3800, Doi: 10.4271/2001-01-3800.

Peng, C.R., 2002. Analysis to the conditioning function of air conditioner to the comfortable environment inside the vehicles [J]. Guangxi Mach., Retrieved from: http:// en.cnki.com.cn/ Article_en/ CJFDTOTAL-GXJX200201006.htm.

Power COOL, 2006. User’s Guide. Release 4.0, Exa Corporation, Massachusetts, Boston.

Song, S.H., C. Yang and X.L. Gou, 2009. Simulation of air-flow and temperature field in air-conditional automobile cabin. Comput. Simul., Retrieved from: http://en.cnki.com.cn/ Article_en/CJFDTOTAL-SJZ200409050.htm.

Xiao, G.Q., J.M. Ni, J.M. Wang and W. Wang, 2007. Simulation of directional uniform air flow in a negative-pressure compartment. Comput. Simul., Retrieved from: en.cnki.com.cn/ Article_en/ CJFDTOTAL-JSJZ200708065.htm.

Yang, Z., Bozeman, J., Shen, F.Z., Turner, D. and S. Vemuri, 2002. CFRM Concept for Vehicle Thermal System. SAE Technical Paper 2002-01-1207, Doi: 10.4271/2002-01-1207.

Zhai, J.Z., Z. Zhang, W. Zhang and Q. Chen, 2007. Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part-1: Summary of prevalent turbulence models. HVAC&R Res., 13(6): 853-870.