Enhancement in Efficiency of Cooling System of Vehicle by optimizing an air Flow and Avoiding Hot Air Recirculation

[1]Saiprasad Mandale, [2]Dr. Pradeep Patil

[1]M.E Student, [2]Professoer,
[1]Mechanical Department SVCE, Indore

***Abstract*—In automobile due to combustion of fuel there is a generation of heat which may lead to rise in temp of automobile system and nearby components of engine, due to rise in temp engine parts viz. inlet and outlet valve may over heat and may damage which may reduces efficiency of engine therefore its necessary to maintain temperature of system. That is obtain by automobile cooling system i.e. radiator and intercooler, fan arrangement which utilize atmospheric cold air and circulate around engine compartment and reduces temperature by convection and become hot. So it's necessary to study cold and hot air circulation to maximize cooling performance of vehicle and to identify easiest method to study different parameter to save time and money.**

***Key Words*: Under hood, Heavy Commercial Vehicle, Thermal Management, Exhaust Manifold and CFD**

I. INTRODUCTION.

The goal in any vehicle development program is to reduce the concept of production cycle time and the number of prototype vehicles and still meet the design targets. For automobile cooling system along with high power requirement generation of heat is also ingresses which will affect efficiency of the system hence to optimize cooling parameter hot and cold air flow through radiator is necessity to study. Given the high cost of building the prototypes, it is becoming increasingly necessary to develop a set of validated numerical simulation tools and techniques that could reduce prototype testing, thereby leading to reduction in vehicle design cost and cycle time. To observe hot and cold air flow circulation CFD approach has adopted to correlate relationship between actual and CFD results at different vehicle running conditions and accordingly identity ways to keep engine compartment cool to enhance its efficiency.

II. LITERATURE REVIEW

To improve efficiency of vehicle lot of research and publication has been done. After studying that a brief discussion and results were discussed in these articles. Under hood Flow Management of Heavy Commercial Vehicle to Improve Thermal Performance Chetan Kulkarni, Deshpande M. D. Umesh S, Chetan Raval Department of Automotive and Aeronautical Engineering, M. S. Ramaiah School of Advanced Studies, Bangalore 560 058 4-CAE Manager, Mahindra Navistar Automotives Ltd, Pune. Found that recirculation zones near the fan and radiator have been reduced [1] .The CFD model developed in cooling system study has been validated against experimental measurements in a planned multistep approach to validation exercises have been conducted for the baseline design configuration to build Confidence in the CFD model prediction quality [2].Davis et al. (1993) describes the impact of conducting under hood airflow simulations on the cooling design and development process [3]. Fellague et al. (1994) provides a correlation for front end cooling airflow studies [4]. Hsu and Schwartz (1995) described an approach wherein local studies of under-body components are coupled by transferring boundary conditions to the local models [5]. Srinivasan et al. (1999) has demonstrated the use of an adaptive Cartesian based mesh for accurate front-cooling airflow predictions, and the feasibility of building local models of the under hood to understand the thermal environments. Two types of inlet boundary were modeled to represent the combination of ram air with a cooling fan, and air flow beneath a vehicle's front end. An outlet boundary was also modeled to represent the flow exiting downstream into the under body of the car. Coupled radiation/convection simulations were performed to obtain the complete airflow and thermal map of the under hood [6].

III. OBJECTIVE

A critical requirement of vehicle design is adequate airflow through the radiator core to ensure adequate engine cooling under all operating conditions so that following objective should be full fill.

IV. To minimize high cost required in the production of number of prototype.
V. To idenentify circulation of hot and cold air around engine to keep it cool.
VI. Possible path of hot air flow, to minimize its circulation around engine.
VII. Possible path of cold air flow, to maximize its circulation around engine.

VIII. METHODOLOGY

Literature review has been done for geometry simplification and under hood thermal management.

- Computational domain of under hood 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.

Boundary Conditions:

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

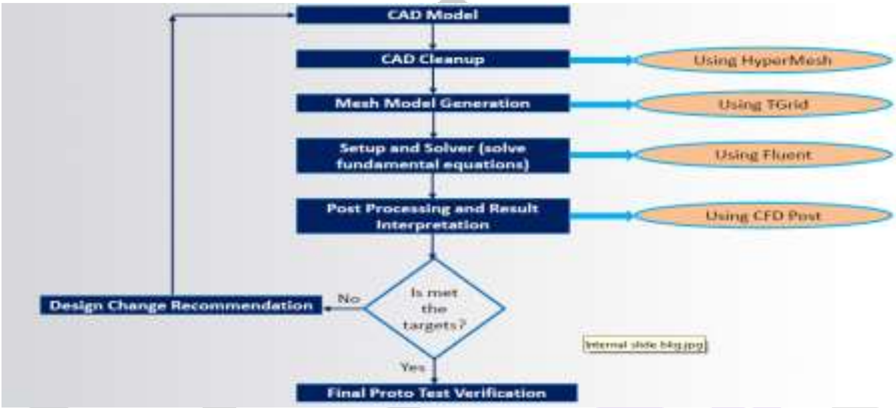

Figure 1: Methodology CFD Simulation

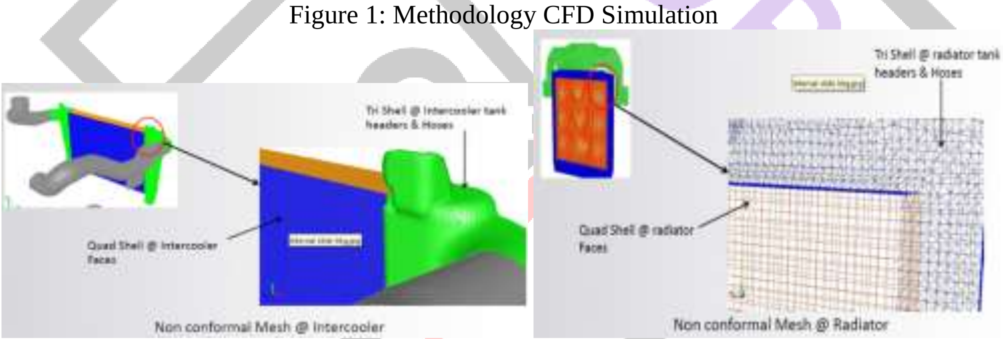

Figure 2: Methodology Mesh Generations Figure

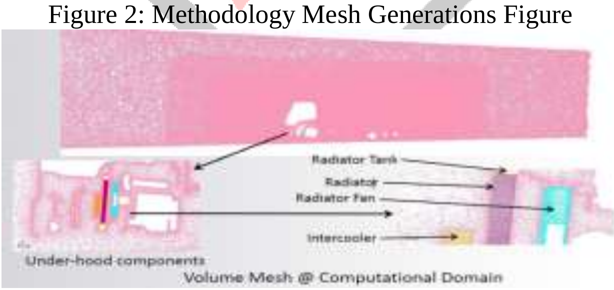

Figure 3: Methodology Mesh Generations

Methodology Input Conditions:

- Vehicle Speed & Fan Speed:

Table 1 Intercooler performance data

	Max Power	Max Torque
Vehicle Speed (KMPH)	V1	V2
Radiator Fan Speed (RPM)	N1	N2

- Input Parameters for thermal analysis:

Table 2 Intercooler performance data

	Max Power	Max Torque
Radiator		
Coolant Flow Rate(kg/s)	mc1	mc2
Coolant Inlet Temp (°C)	Tcin1	Tcin2
Intercooler		
Charged air Flow Rate (kg/s)	ma1	ma2
Charged air inlet temp (°C)	Tain1	Tain2

3 Methodologies: Input Conditions:

Heat exchanger performance data generated through 1-D KULI software for computing heat rejection and outlet temperature of coolant and charged air:

Table 3 Intercooler performance data

Radiator performance data :						
Coolant flow rate	c1	c2	c3	c4	c5	c6
Air Flow rate	Heat Transfer (W)					
a1	h11	h21	h31	h41	h51	h61
a2	h12	h22	h32	h42	h52	h62

Table4 Intercooler performance data

Intercooler performance data :						
Coolant flow rate	c1	c2	c3	c4	c5	c6
Airflows rate	Heat Transfer (W)					
a1	h11	h21	h31	h41	h51	h61
a2	h12	h22	h32	h42	h52	h62

Numerical Analysis:

Although surface temperatures were also measured, this study is aimed to validate the computed air temperatures. The segments of the exhaust system and the applied temperatures for all three configurations are given in table

Fixed temperature	Coolant 115°C	Coolant 105°C	Coolant 105°C
Cylinder block	116	102	103
Cylinder head	116	102	103
Crankcase	116	104	105
Oil pan	106	95	96
Gearbox	113	101	103
Manifold	251	244	244
Turbo turbine	243	240	238
Turbo compressor	120	120	120
Turbo bearing	116	102	102
Exhaust before cat	173	163	171
Catalyst	235	235	237
Exhaust after cat	216	217	218
Mufflers	100	100	100

For the thermal analysis the finite-element code PERMAS is used. Following Equation describes the energy equation solved in PERMAS.

$$\sigma.\mathrm{Cp}\frac{\partial T}{\partial t} - \lambda.\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right) = qi$$

$$\mathrm{C.T} + \mathrm{K.T} \quad = \quad \mathrm{F}$$

The CFD mesh and the PERMAS grid are based upon the same CAD data but are generated with different meshing tools with different accuracy for the representation of the surface. Star CD™ works most accurately with a trimmed mesh consisting of hexahedral volume cells in almost all parts of the computational domain. For PERMAS a tetrahedral grid, which is in most cases automatically created, is sufficient. The common interface of both codes is the surface of both meshes. Therefore a mesh mapping tool is needed to map the physical properties from one mesh to the other . The commercial tool MpCCI™ is used for the mapping. An interpolation tool has been developed at DaimlerChrysler to assure physically reasonable values at each element, although the coupling regions are geometrical not identical.

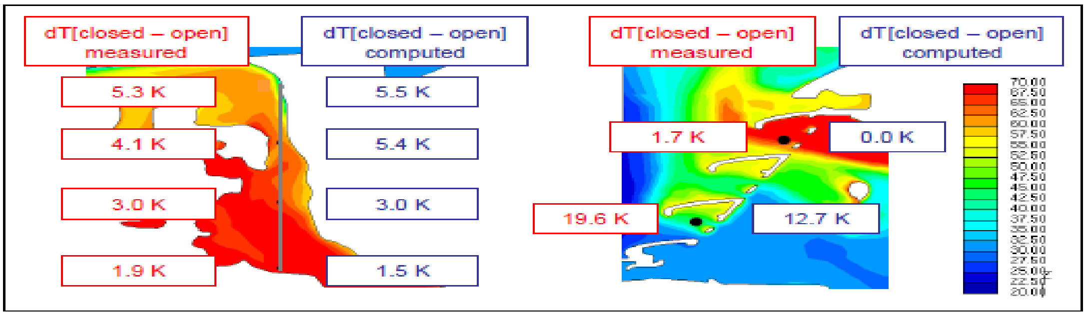

Figure 4: Difference of air temperature closed – open grill for experiment and simulation.

Nevertheless not all measurement points agree well with experimental results. Especially in regions with high temperature gradients deviations are very likely to occur. Fortunately, temperatures of components normally do not show large temperature gradients. So it will be much easier to compare component measurements and numerical results.

IX. RESULT AND DISCUSSION

Results: Existing Vehicle – Max Power Condition

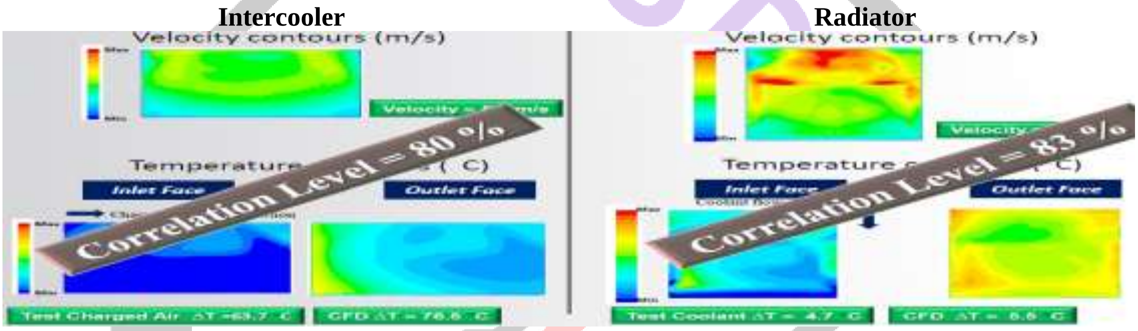

Figure 5: Existing Vehicle – Max Power Condition

Correlation: Existing Vehicle – Max Power Condition
Predicted vehicle level performance of intercooler and radiator at fixed inlet temperature
Table 5 Existing Vehicle – Max Power Condition

Ambient Temp = 28.5°C		Intercooler			Radiator		
		Test	CFD	Correlation	Test	CFD	Correlatio
Coolant/ Charged air side	CFD Inputs						
	Flow Rate	ma1	ma1	--	mc1	mc1	--
	Inlet Temp	Tain1	Tain1	--	Tcin1	Tcin1	--
	CFD Outcomes						
	Outlet	$Taout1_{Test}$	$Taout1_{CFD}$	--	$Tcout1_{test}$	$Tcout1_{CFD}$	--
	Temp	63.7	76.5	80	4.7	5.5	83
	Heat	8.3	10	79.5	42.6	49.9	82.8

Results: Existing Vehicle – Max Torque Condition

Intercooler **Radiator**

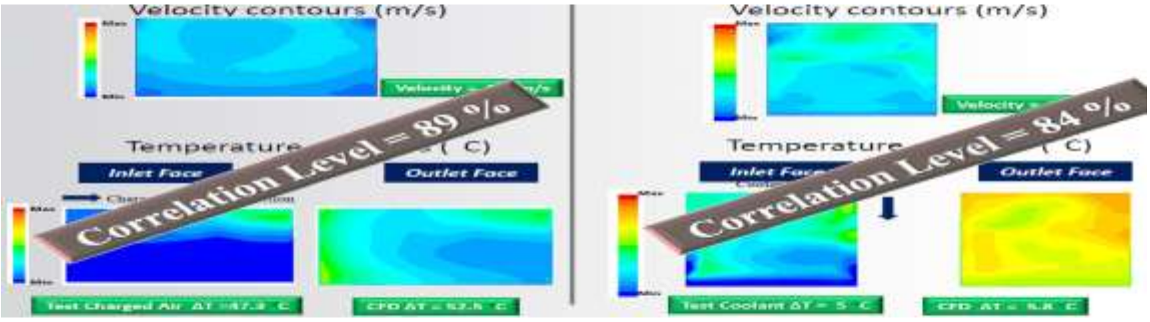

Figure 6: Existing Vehicle – Max Torque Condition

Correlation: Existing Vehicle – Max Torque Condition

Predicted vehicle level performance of intercooler and radiator at fixed inlet temp

Table: 6 Existing Vehicle – Max Torque Condition

	Ambient Temp =	Intercooler			Radiator		
		Test	CFD	Correlation	Test	CFD	Corr
Coolant/ Charged air side		CFD Inputs					
	Flow Rate	ma1	ma1	--	mc1	mc1	--
	Inlet Temp	Tain1	Tain1	--	Tcin1	Tcin1	--
		CFD Outcomes					
	Outlet	$Taout1_{Test}$	$Taout1_{CFD}$	--	$Tcout1_{test}$	$Tcout1_{CFD}$	--
	Temp	47.3	52.5	89	5	5.8	84
	Heat	3.1	3.4	90.3	20.1	23.4	83.6

Hot air recirculation zone identification:

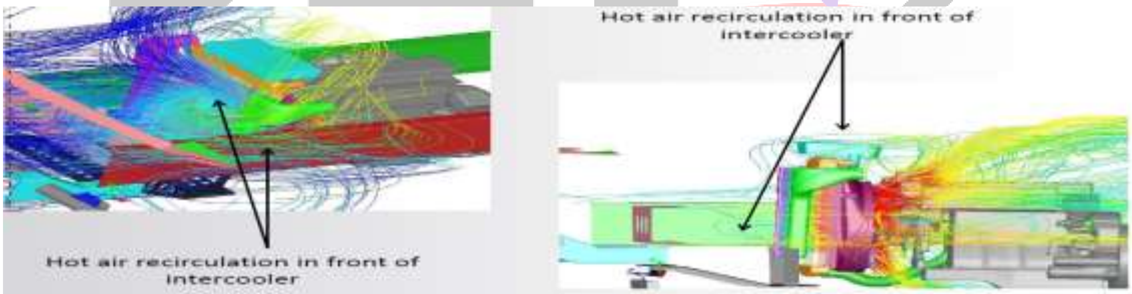

Figure 7: Path Lines colored by Temperature (^{0}C)

Results: New Vehicle – Under-hood Thermal Flow Field

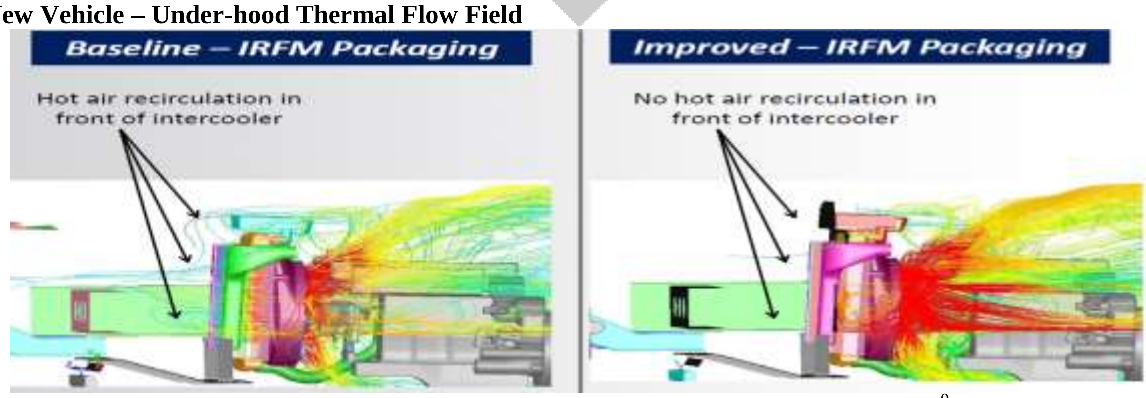

Figure 8: Base line V/s Improved Path Lines colored by Temperature (^{0}C)

Results: IRFM Packing – Intercooler performance at Max Power Condition.
Improvement 12% in Intercooler performance

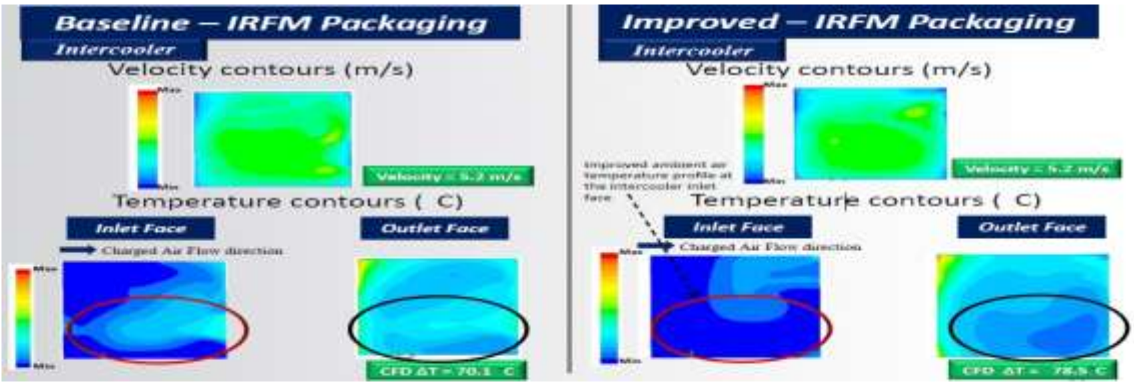

Figure 9: Base line V/s Improved Intercooler performance

Results: New Vehicle – Radiator performance at Max Power condition
Improvement 8.5% in Radiator performance

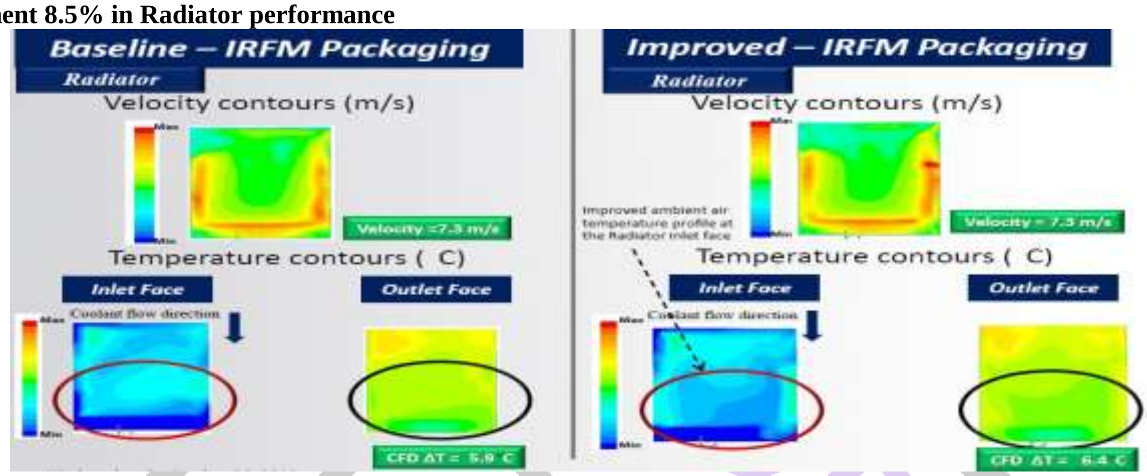

Figure 9: Base line V/s Improved Radiator performance

Conclusion

Following conclusions were derived from the readings and analysis of the project.

1. Using CFD we were directed comparing the heat transfer & pressure drop of heat exchanger with different parameters for optimum performance And CFD analysis has also reduced the cost & time in design and development of radiator as compared to conventional.

2. Methods Correlation level between Field test and CFD simulation is more than 80%.

3. Under hood compartment thermal flow field has been improved by stooping hot air Recirculation by introducing sealing, thus improved.

4. In this study it was found that the proposed heat shield design can act as a good measure in improving the heat distribution and cooling performance on engine components, regardless of its positioning.

5. Intercooler performance improved by **12%.**

6. Radiator performance improved by **8.5%.**

Limitations:

I. Packaging constraint for under hood sealing to vehicle to avoid hot air recirculation can be challenging.

II. The costing has increased by certain amount.

References-

[1] Under hood Flow Management of Heavy Commercial Vehicle to Improve Thermal Performance Chetan Kulkarni, Deshpande M. D,Umesh S., Chetan Raval ,Department of Automotive and Aeronautical Engineering, M. S. Ramaiah School of Advanced Studies, Bangalore 560 058 4-CAE Manager, Mahindra Navistar Automotives Ltd, Pune.

[2] Purdue University Purdue e-Pubs International Refrigeration and Air Conditioning Conference School of Mechanical Engineering 2006 CFD Prediction to Optimize Front End Cooling Module of a Passenger Vehicle Biswadip Shome Tata Technologies Limited Vinod Kumar Tata Technologies Limited S. V. Ranganath Kumar Tata Technologies Limited Gyan Arora Tata Technologies Limited .

[3] Davis, F.V., Veling, T.R., Caltrider, J.L. and Madhavar, R. 1993. Impact of computer aided engineering on Ford motor company light truck cooling design and development processes. SAE Paper 93-2977.

[4] Fellague, K.A., Hu, H. and Willoughby,D.A.1994. Determination of the effects of inlet air velocity and temperature on the performance of an automotive radiator SAE Paper 94-0771.

[5] Hsu, I.F. and Schwartz, W.S. 1995. Simulation of the thermal environment surrounding an under-body fuel tank in a passenger vehicle using orthogonally structured and body fitted unstructured CFD codes in series. SAE Paper 95-0616.
[6] Srinivasan, K., Jan, J.Y., Sun, R.L. and Gleason, M.E. 1999. Rapid simulation methodology for under-hood aero/thermal management. International Journal of Vehicle Design, 23 (1-2): 109-123.