Proceedings of HTFED2004
2004 ASME Heat Transfer/Fluids Engineering Summer Conference
July 11-15, 2004, Charlotte, North Carolina USA.

HT-FED2004-56748

VEHICLE THERMAL MANAGEMENT SIMULATION USING A RAPID OMNI-TREE BASED ADAPTIVE CARTESIAN MESH GENERATION METHODOLOGY

Kumar Srinivasan, Technical Specialist,
Aero/Thermal Development
DaimlerChrysler Corporation, Auburn Hills, MI 48326
E-mail: ks40@dcx.com

Z. J. Wang, Assoc. Professor, Dept. of Mech. Engr.,
Michigan State University, East Lansing, MI 48824
E-mail: zjw@egr.msu.edu

Wei Yuan, Research Engr.
ESI Group, Huntsville, AL 35805
E-mail: Wei.Yuan@esi-group-na.com

Richard Sun, Senior Manager, Aero/Thermal Center of Competence, DaimlerChrysler Corporation,
Auburn Hills, MI 48326
E-mail: rls37@dcx.com

ABSTRACT
CFD simulation of vehicle under-hood and under-body poses several challenges. Specifically, the complexity of the geometry involved makes the use of traditional mesh generation approaches, based on the boundary-to-interior methodology, impractical and time consuming. The current work presents the use of an interior-to-boundary method wherein the need for creating a 'water-tight' surface mesh is not a pre-requisite for volume mesh generation. The application of the new method is demonstrated for an actual passenger vehicle under-hood model with nearly a hundred components. Coupled radiation/convection simulations are performed to obtain the complete airflow and thermal map of the engine compartment. Results are validated with test data. The new method results in significant gains in efficiency over traditional approaches allowing the simulation tool to be used effectively in the vehicle development process.

KEYWORDS
Automotive, Under-hood Thermal, Radiation, Cartesian, Mesh Generation

INTRODUCTION
In the past decade, the automotive industry has witnessed tremendous advances in the integration of CFD into the vehicle development process. Several key aspects of automotive development utilize CFD during early stages of design in order to identify, correct and prevent re-design costs and also to reduce physical testing and prototyping. Simulations also help to reduce overall product development time cycles and allow vehicle manufactures to bring new products to market much faster. Simulation tools have continuously improved over the past several years to make this possible. The availability of unstructured, control volume based commercial flow solvers and grid generators have made it possible to tackle large scale computational problems involving extremely complex geometry.

Customized development of CFD simulation techniques and their validation has been on-going for several years in the automotive industry. The mainstream application areas within automotive development are engine cooling, climate control systems, exterior aerodynamics, power-train component optimization including exhaust system, in-cylinder flows etc. and vehicle thermal management. CFD modeling has made in-roads in all the above areas. The focus of this paper is the vehicle thermal management area.

Thermal management is an important aspect of passenger vehicle development. The need for increased interior room and passenger comfort has resulted in more compact under-hood and under-body packaging environments. Stringent emission requirements have also necessitated that catalytic converters be packaged closer to the engine block resulting in more heat radiation within the engine compartment. In addition, reduced product development cycles have made simulation of vehicle thermal management very important. Early detection of potential thermal issues through simulation allows reduction in the number of prototype based physical tests and associated development costs.

In the past, several approaches have been proposed to perform CFD simulations of the under-hood environment. Davis et.al. [1] describe the impact of doing under-hood airflow simulations on the cooling design and development process. Fellague et.al. [2] provide correlation for front end cooling airflow studies conducted using the UH3D code. Hsu and Schwartz [3] describe an approach wherein local studies of under-body components are coupled with global UH3D simulations by transferring boundary conditions to the local models. Srinivasan, K. et.al. [4] have 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. More recently, Yang, et.al., [5] have simulated vehicle at idle with just the cooling fans operating. Cooling airflow rates at idling conditions and hot air re-circulation are modeled in this work.

Simulations of under-hood and under-body thermal environment involve several challenges. One of the most significant aspects is the complexity of the geometry involved. Typical passenger vehicle under-hood comprises of multitude of components and subcomponents. Although CAD geometry is available, clean-up of the geometry and subsequent processing to allow volume mesh generation typically involves several man weeks of CAD "clean-up" and preparation. Due to this bottleneck, engineering input from the thermal simulations was minimal in the past. The simulation activity lagged behind the design activity and as a result the models were not representative of the current design direction at any given point in time.

In the present work, a new omni-tree based adaptive Cartesian mesh generation technique is used to effectively handle complex geometry and automatically generate three dimensional, body-fitted volume meshes for complex under-hood and under-body components. This new technique has reduced meshing time required for this type of simulations from several weeks to about 1-2 days for initial mesh generations and few hours for subsequent iterations. This has enabled the simulation model to be updated frequently, allowing for valuable input to be provided during the early phases of product development. Most of the vehicle thermal management simulations published in the literature adopt tetrahedral based models. This involves manual clean-up of complex geometry to remove unnecessary details and fix imperfect geometry (gaps). The present Cartesian based approach overcomes this constraint and allows the mesh generation process to be fully automatic with minimal manual intervention.

PROBLEM DESCRIPTION

The engine compartment, also referred to as the under-hood, of a passenger vehicle houses close to a hundred major components including power-train, electrical, suspension, exhaust components etc. The cooling module comprising of several heat exchangers acts as a primary air flow inlet into the engine compartment. When the vehicle is in motion, the airflow stream beneath the front end of the vehicle also becomes part of the under-hood convection mechanism. Figure 1 shows a typical engine compartment. The geometric complexity is self evident.

The outer surface of the exhaust system is the primary source of heat into the engine compartment. The skin temperature of various components of the exhaust system such as manifolds, catalytic converter, connecting pipes, muffler etc. can range from 785K-925K when the engine is outputting close to its maximum capacity with stoichiometric fuel air ratio based combustion. The close proximity of various components in the under-hood environment to the exhaust system results in high temperatures which may be above the permissible material and/or operational limits of the component.

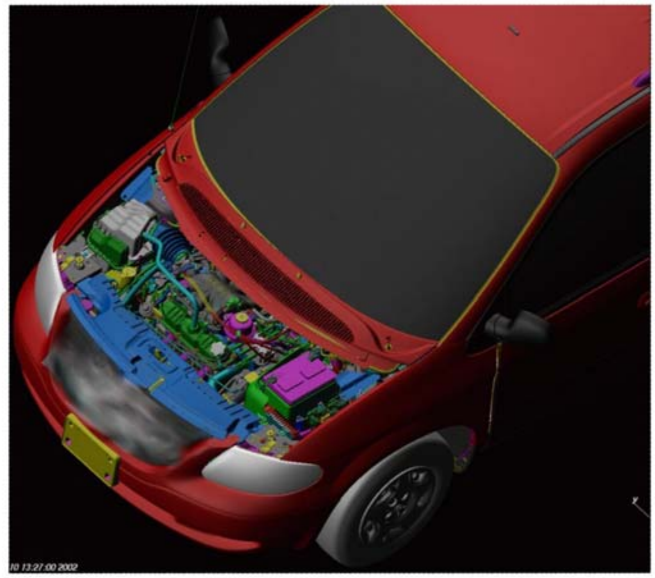

Figure 1. Vehicle under-hood environment.

This makes it essential to monitor the temperatures of all components that may be at risk of failure due to thermal loads and provide appropriate thermal protection. This could be achieved by:

1. Relocating the component
2. Insertion of heat shields between the exhaust and the component
3. Innovative airflow management techniques that increase the convection around the component.

It is obvious that option 1 does not involve additional parts and material costs. As the volume available for packaging components in the under-hood environment is usually very limited, option 1 is typically difficult to accomplish unless this is performed early in the development cycle when the packaging is evolving and some flexibility is available. This is possible if reliable information is available regarding the thermal environment for each component and information about the peak operating temperatures are available early in the design phase. This makes the CFD simulation of the vehicle under-hood environment essential well before any hardware prototypes are available.

COMPUTATIONAL DOMAIN

Figure 2 shows the extent of the physical domain used for the computation. In the X-direction, the domain starts from the cooling module and extends downstream till the under-body of the vehicle starts. This is usually just downstream of the dash panel/A-pillar of the vehicle. In the vertical direction (Z), the ground plane, shown in black, forms the lower boundary and the inner surface of the hood forms the upper boundary. In the lateral direction, two artificial vertical planes are chosen which is outboard of the vehicle's under-hood environment. These are not shown in Figure 2 for clarity. Typically, this plane is chosen to be outboard of the front rails and close to the front tires.

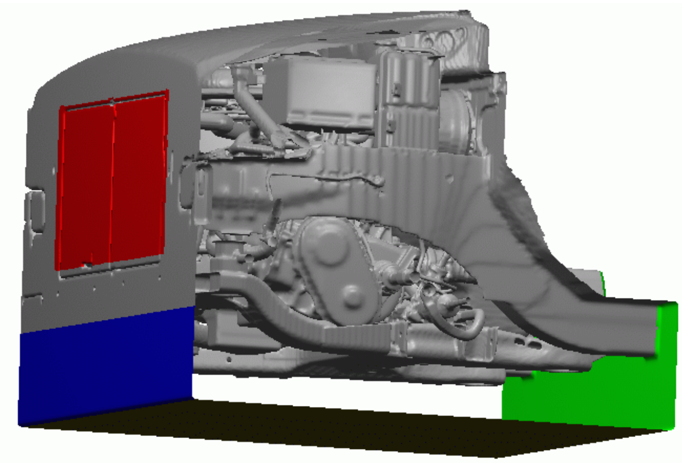

Figure 2. Computational mesh domain

The model is set up with two inlet boundaries and an outlet boundary as shown in Figure 2. The upper inlet, shown in red, represents the flow stream that enters the engine compartment through the fans. This is typically a combination of ram air and the influence of the cooling fan. The lower, rectangular inlet, shown in blue, is chosen to model the airflow beneath the vehicle's front end. The outlet, boundary, shown in green, essentially represents the flow exiting downstream into the under-body of the vehicle. All the components in the under-hood that are enclosed in this volume are included in the model. The boundaries defined above form a closed volume with multiple closed volumes contained inside it.

MESH GENERATION METHODOLOGY

In the past, mesh generation typically forms the primary bottle neck in CFD modeling of vehicle under-hood environments. There are several reasons for this:

1. Most commercial mesh generation packages require a clean surface mesh to be created before volume mesh generation.
2. CAD data is usually "dirty" and needs to be "cleaned" before volume mesh can be created.
3. The CAD data is incomplete – multiple components may be occupying the same physical location since the final location is yet to be determined. This means geometries may intersect with each other and needs to be trimmed etc. This adds to the burden of CAD data clean-up.
4. Large number of CAD models is needed to create assemblies. For example the engine typically comprises of close to a hundred pieces that come together.
5. The detail in the CAD data is typically much more that what is needed for a CFD simulation. Unnecessary details need to be removed from the geometry which again needs to be done manually on a part by part basis.

In the present work, CFD-VisCART, a commercial mesh generation package originally developed by CFD Research Corporation (now marketed by ESI Group) has been used for mesh generation. This tool is based on an Omni-tree based Cartesian mesh generation methodology. This procedure overcomes all of the above difficulties allowing a stream lined mesh generation process that lets the user fine tune the mesh and geometric details to the appropriate level.

This is made possible through a fundamental shift in the mesh generation philosophy. Traditional mesh generation techniques, structured as well as unstructured, falls in the category of Boundary to Interior (B2I) wherein, a "water-tight" surface mesh is needed before the interior volume mesh can be generated (Ref 6). For the present application, 70-100 component assemblies are involved with each component assembly consisting of anywhere from 2-100 sub parts. Generating a water-tight surface mesh for these components is an extremely time consuming process. A detailed under-hood model, could take as much as 4-6 weeks depending on how "dirty" the original CAD geometry is.

CFD-VisCART uses an Interior to Boundary (I2B) approach wherein the grids generation process is reversed. Instead of generating the boundary grid first, the interior volume grid is generated first, and then the interior grid is "connected" with the boundary. In this case, the need for a "water-tight" geometry to start the grid generation process is eliminated. The approach has the potential of completely eliminating geometry repair from grid generation.

In the general I2B grid generation approach for a given set of geometric entities, two meshing parameters, d_{min} and d_{max} are given. They represent the minimum and maximum sizes of grid cells to be generated. The only requirement that the set of geometric entities must satisfy is that the computational domain formed with the entities is "physically" closed if gaps or holes smaller than d_{min} are ignored. This is to say that if a gap or a hole exists in the geometry (which should not have been there); the size of the gap or the hole must be smaller than d_{min}. Note that this enclosure condition is much weaker than the condition of "water-tightness" required by B2I approaches. The implementation details of the B2I approach are available in [7-8].

MESH GENERATION PROCESS

The under-hood environment usually contains several component assemblies that do not get relocated and others that will be re-packaged several times during the development process. For example, the engine, transmission, suspension etc. have designated locations and all other components are packaged around them. The exhaust system routing and locations of components such as battery, ABS module, on board electronic modules, washer bottle, power steering fluid reservoir etc. will change several times during the design evolution. The mesh generation process needs to accommodate changes to the models and their location quickly and efficiently.

Using CFD-VisCART, the fixed component assemblies such as engine, transmission etc. are "shrink-wrapped" into one entity. Figures 3, 4 and 5 show the overall grid generation process. Figure 3 shows the typical components that are enclosed in the computational domain. These are usually read into CFD-VisCART in stereo-lithography (STL) format. Use of the STL format, instead of native CAD data offers two advantages. Firstly, STL files are relatively "light" since it comprises of triangulated version of the geometry as opposed to native CAD data which contains the mathematical details of the geometry. Secondly, STL data is readily available and easy to assemble for large number of components allowing data collection and management easier.

Figure 4 shows the surface mesh on all the components. Although only the surface mesh is shown, at this stage of the mesh generation, the volume mesh is also completed. This is one of the advantages of the I2B meshing philosophy. The surface and volume meshes are generated simultaneously while B2I approaches require that a water-tight surface mesh to be created first.

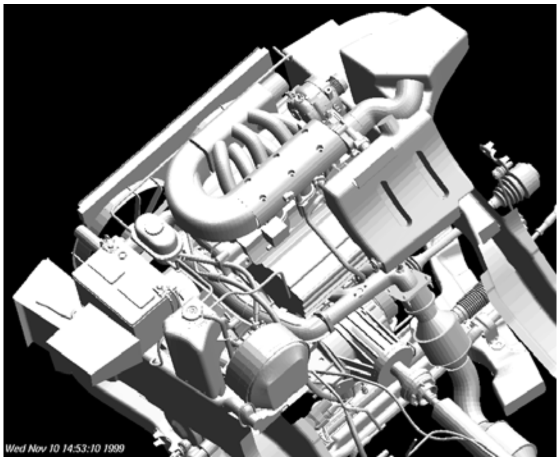

Figure 3. Original Geometry in STL format

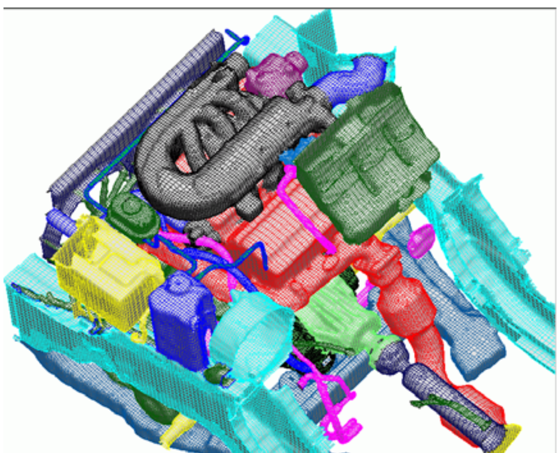

Figure 4. Computational mesh (only surface mesh is shown).

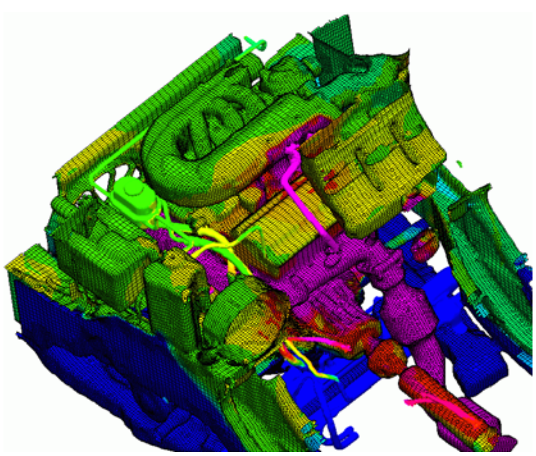

Figure 5. Thermal map on surface of components

Since the present mesh generation methodology is based on a Cartesian background grid, the volume mesh is pre-dominantly hexahedral. This results in much better grid quality and faster convergence of the calculation. Moreover, the number of elements is typically lower than a tetrahedral mesh by a factor of about 4-5 for a given surface mesh resolution. This leads to lower computational time.

Figure 5 shows the overall thermal map of the under-hood components. This is the end result of the analysis. It provides insight into where the thermal 'hot' spots are within the engine

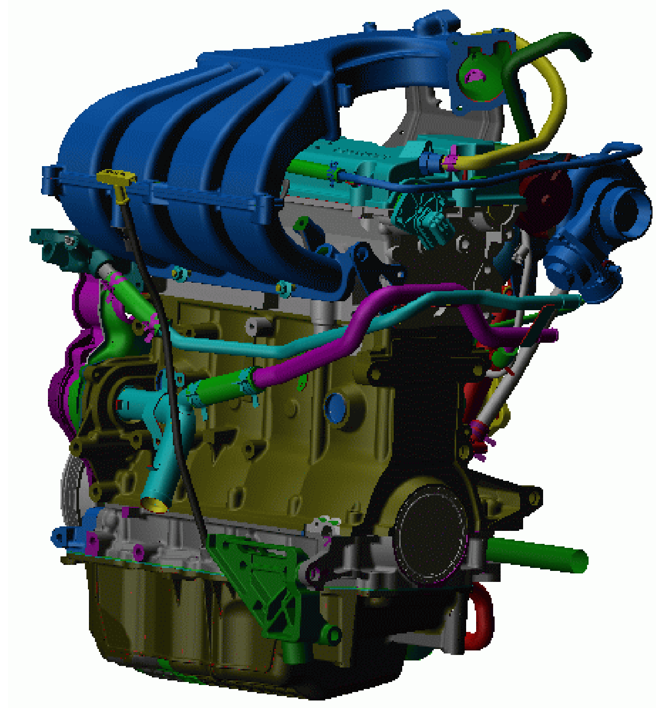

Figure 6a Engine assembly with about 30 sub parts.

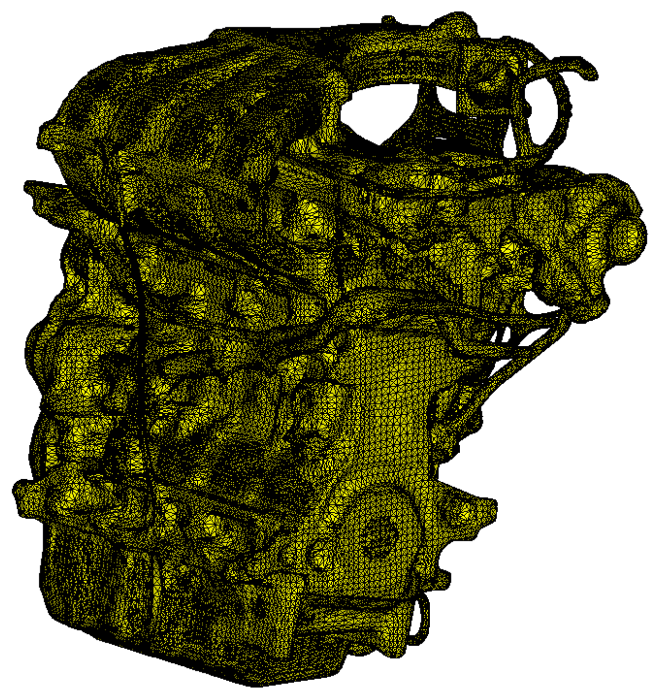

Figure 6b. Shrink-wrapped surface mesh.

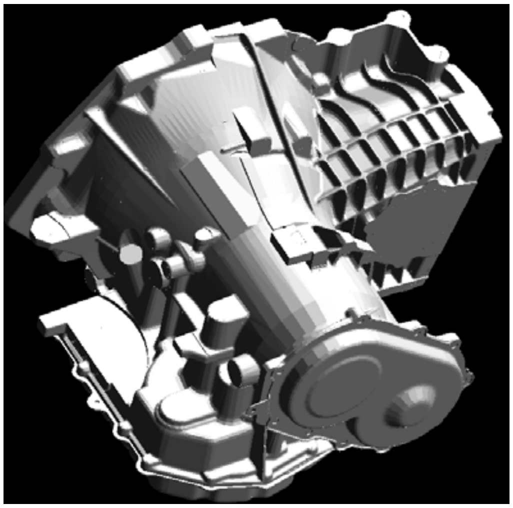

Figure 7a. Original surface – transmission case.

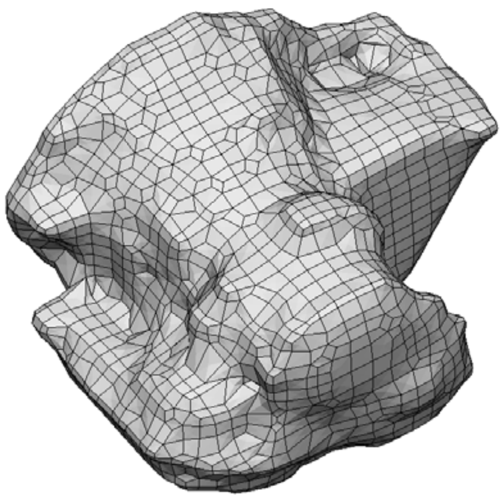

Figure 7b. Shrink-wrapped surface (low resolution).

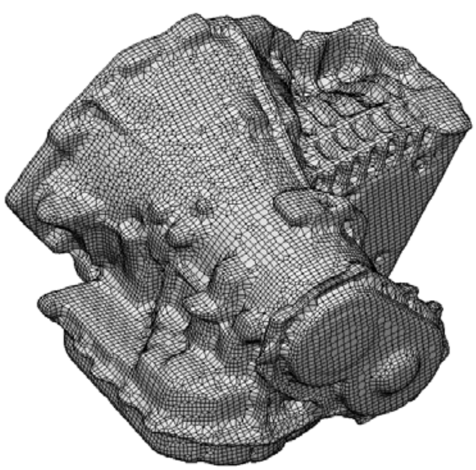

Figure 7c. Shrink-wrapped mesh – fine resolution.

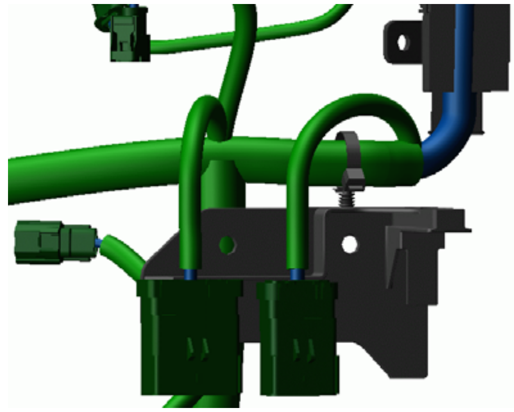

Figure 8a. Geometry showing detailed wiring harness with connectors

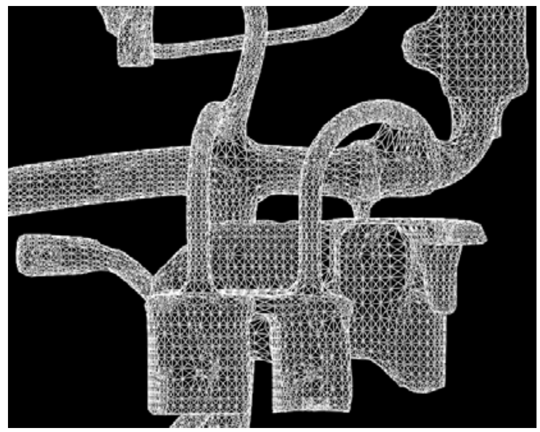

Figure 8b. Shrink-wrapped surface mesh

compartment, and allows for early changes to be put in place. More discussion of how the results are analyzed and utilized is discussed in a later section.

SHRINK-WRAP EXAMPLE

When large number of parts are assembled together to form one entity, such as the engine, the shrink-wrap feature in CFD-VisCART is used to create an envelope of the assembly. This allows for easy part management for subsequent iterations of the under-hood model.

Instead of reading in multiple components, a single file can be created that retains all the details required for the analysis. Figure 6a-6b shows an example of this feature. Figure 6a shows the original STL geometry which is a combination of nearly 30 sub-components. Figure 6b shows the shrink-wrapped surface mesh which is then used for all subsequent mesh generation. The shrink-wrapping operation is fairly quick and only takes a few minutes for the given example.

The shrink-wrapping operation allows for retention of any required level of geometric fidelity as long as the features that are needed are dimensionally larger than the largest “gap” or “hole” in the geometry. The other advantage of the shrink-wrap operation is that it allows for reduction in the number of surface elements the meshing process has to handle during subsequent volume mesh generation sessions. In the example shown, the number of surface elements was reduced by a factor of 3. This factor obviously depends on the surface resolution used and could be higher or lower accordingly.

Figure 7a-7c show another example wherein the surface resolution is varied to demonstrate how the different level of geometric detail can be achieved for the simulation in question. Figure 7a shows the original surface and Figures 7b-7c show different levels of surface refinement which results in geometric details being smoothed over or captured. Each of these shrink-wrap meshes were created in about 2-3 minutes. Keep in mind that the original STL file is not 'clean' and contains physical gaps and disconnected triangles.

Figure 8a-8b demonstrates how the present mesh generation method is able to handle geometry with holes and if the user chooses to close these gaps, it can be achieved automatically. Figure 8a shows the detailed geometry of a wiring harness with connectors. Note that there are several holes in the geometry, along with unnecessary details such as clips, detailed features of the connectors etc.

By appropriately choosing a surface mesh resolution, the holes in the geometry are filled automatically and unwanted details are removed or paved over. This is a tremendous advantage over traditional mesh generation approaches wherein, retaining the un-necessary detail would result in significant increase in mesh sizes. At the same time, if the details have to be eliminated, it would be time consuming because the details would have to be eliminated one at a time for each component. This entails large amounts of manual effort making the overall process impractical.

USE OF SURFACE AND BOX SOURCES

As mentioned in the previous section, one of the primary strengths of the present mesh generation is the ability to resolve components in the under-hood environment to different levels allowing for appropriate surface mesh size for each component through the usage of 'surface' sources. In addition, box sources can be defined if specific regions in the volume mesh need to be resolved with a finer mesh. This is useful to customize the under-hood mesh for specific issues that are being investigated. If a particular component is being studied in detail, then a finer surface mesh size can be defined for this component and also a box source can be defined in the vicinity to better resolve the flow features in this area.

PROBLEM SET-UP AND BOUNDARY CONDITIONS

The simulation of the under-hood environment is carried out for a given set of exhaust temperatures and vehicle speeds. Based on typical vehicle duty cycles, a standard battery of tests have been developed. This includes long term operating conditions such as city traffic, highway driving etc. and short term severe conditions that correspond to pulling a heavy trailer up a 5% or 6% grade highway in an ambient of 317 K

The primary purpose of the under-hood simulation is to identify potential thermal issues and develop counter measures in the form of component relocation, heat shields or airflow management to enhance convection. The short term, severe conditions usually dictate the need for a heat shield. Therefore, the heavy trailer tow type condition is usually modeled in the simulation.

Temperature and velocity boundary conditions are needed at the inlet boundaries described in the Computational Domain section. In addition, the primary heat source in the under-hood is the exhaust system. The skin temperature of the exhaust system is another boundary condition that is required to perform the thermal analysis. The exhaust system comprises of several pieces including the manifolds, pipes and flanges, flexible joints, catalytic converter, muffler and resonators. Each segment of the exhaust system is typically at a different temperature by virtue of its construction and the flow conditions inside. The maximum temperatures are governed by the peak temperature limit dictated by the substrate in the catalytic converter. Based on this value, the other sections of the exhaust system will be at different values. The skin temperature values for the exhaust system are obtained from a variety of sources including previous test data, one-dimensional model of the exhaust system, and bench tests of the exhaust system operating at similar conditions. These temperature values are then specified as isothermal wall boundary conditions on the various sections of the exhaust system.

The velocity boundary condition at the inlet to the engine compartment is known because the cooling system requirements for the vehicle dictate that a certain amount of flow rate passes through the heat exchangers. During the early development phase of the vehicle, CFD simulations are carried out to ensure that the appropriate amount of airflow passes through the heat exchangers (radiator, condenser, oil cooler etc.) – see [4]. As a result the airflow entering the engine compartment through the heat exchangers is known. Moreover, because the heat rejection from the engine is known for any specific condition, the temperature rise of the air across the heat exchangers is also known. Based on the ambient conditions of 317K, the temperature of the air entering the under-hood through the heat exchangers is computed and specified at inlet 1.

Based on the airflow and temperature assumptions, a coupled radiation/convection simulation of the under-hood environment is carried out. The surface of the under-hood components are treated as adiabatic boundaries. This is usually not a good approximation for components made of highly conductive materials such as aluminum, steel etc. But typically, such components are not in thermal risk. Components made of plastic, rubber, polymers etc. have lower temperature targets and need to be monitored closely. These materials are also poor conductors and therefore can be treated as adiabatic boundaries without significant loss of accuracy.

FLUID FLOW AND RADIATION SIMULATION

The coupled flow and heat transfer computation is carried out using the commercial package CFD-ACE+ which is a general purpose unstructured control volume based Navier-Stokes flow solver developed by Jiang et. al. [9]. Standard K-ε turbulence model in conjunction with wall functions is used for turbulence closure. The solver allows for usage of hexahedral, tetrahedral, prismatic, pyramidal and general polyhedral cells. In the current process, the mesh contains pre-dominantly hexahedral cells, but also contains polyhedral cells. The surface mesh contains triangles, quadrilaterals and polygonal faces.

A surface-to-surface based radiation model as described in Tan et. al. [10] is used in the current work. This methodology has the advantage that the view factor calculation scales as NlogN where N is the number of surface elements.

RESULTS AND DISCUSSION

In this section, some of the results and correlation to physical testing obtained from performing under-hood thermal simulations on a vehicle are discussed. The calculations are performed for a vehicle traveling at 24.58 m/s, (55mph) pulling a heavy trailer. Due to the heavy load, the exhaust skin temperatures are at their highest.

Figure 9 shows a section cut through the volume mesh. As seen in the figure, the mesh resolution varies significantly from one region to another. The surface mesh size is chosen so as to ensure a y+ value of 30-75 at all boundaries. K-E turbulence model in conjunction with wall functions is used to model near wall turbulence. The mesh size of 5mm is used for all the elements adjacent to the surface. The computational mesh contains about 2.5 million volume cells. The simulation typically takes about 14 hours of wall clock time. This includes about 3 hours needed for performing the view-factor calculation.

Figure 10 shows the velocity vectors through a section of the under-hood. The vectors have been colored based on the temperature of the fluid. As seen, the hot air entering the under-hood environment mixes with the relatively cooler air entering from below. In local spots, the temperature of the air

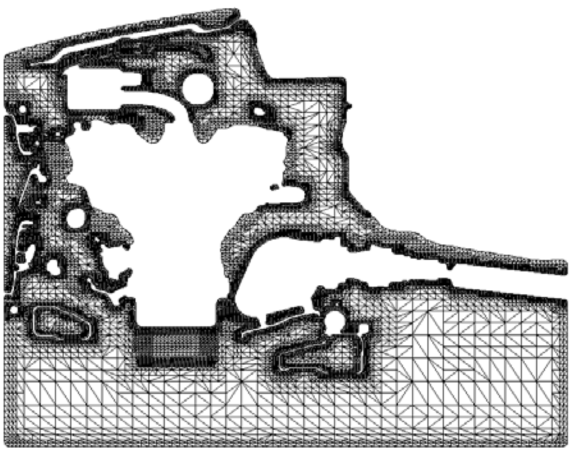

Figure 9. Section cut through under-hood volume mesh

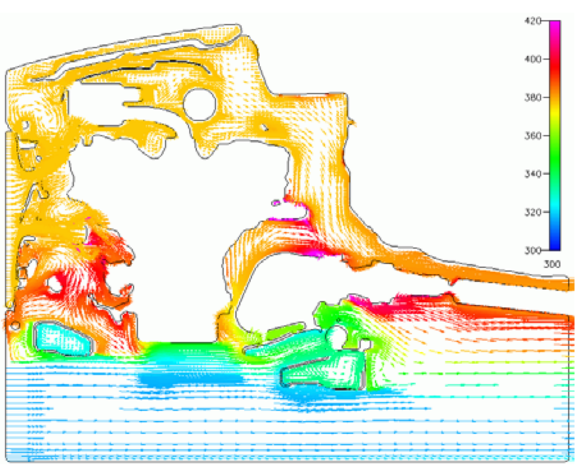

Figure 10. Velocity vectors colored with temperature. Section of the under-hood is shown.

is hotter than the inlet values due to heat pick up from exhaust components. The complete thermal and velocity map of the

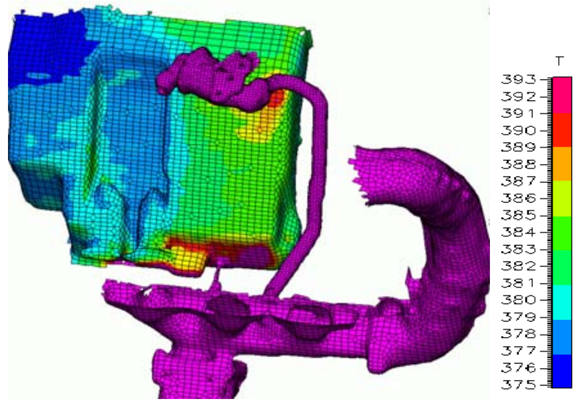

Figure 11. Temperature distribution on a wiper washer bottle.

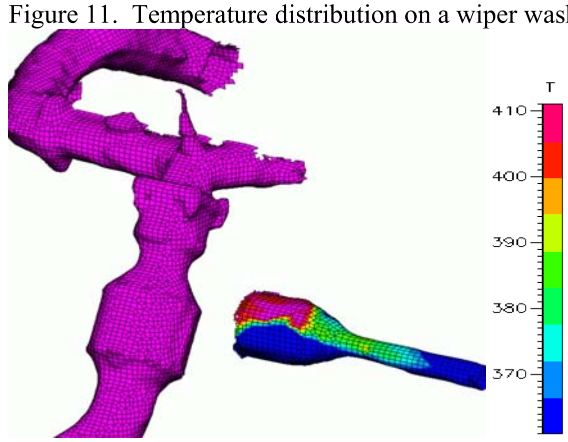

Figure 12. Front CV boot surface temperature distribution.

airflow within the under-hood environment is available. This allows for airflow management and investigations of appropriate component placement options.

Figure 11 shows the temperature distribution on a wiper washer fluid container that was initially packaged in the proximity of the exhaust manifold and EGR tube assembly as seen in the picture. The peak surface temperature on this component was shown through the simulation to be about 393 K. This was above the allowable limit for the material and the component was moved to a different location farther away from the exhaust. This was done well before prototype vehicles were built, thereby saving hardware and testing costs. In the absence of simulation data, potentially the component could not have been re-located later in the development cycle (when actual test data became available) due to packaging constraints, thereby requiring a heat shield. The simulation paved the way for pro-active action that resulted in an effective solution.

Figure 12 shows the temperature distribution on a front CV boot surface. The peak temperature on this surface was found to be about 411K which was above the acceptable limit for the thermoplastic boot material. Since this component cannot be re-located, a shield was added to the lower side of the manifold, which was identified to be the primary heat source for this component. The boot surface temperature was verified to be below its acceptable temperature limit with the heat shield in place.

Several verification tests are done in the prototype and production vehicles to ensure temperature limits are not exceeded for any component. Table 1 shows comparison of peak surface temperatures predicted by the under-hood CFD model and measurements from a physical tests conducted on a vehicle. As shown, the correlation between the simulation and test is excellent.

Another important benefit of performing the simulations is that when the physical tests are being carried out, the locations for placement of thermocouples is based on the thermal map obtained from the simulations. This helps to optimize the number of thermocouples on the test property and also to ensure that the hottest spot is actually instrumented. This may or may not correspond to the closest point, on any given component, to the exhaust surface. Convection patterns may shift the hot spot to a different location. The simulation acts as a guide to the instrumentation process.

Component Number	CFD Model	Test	Difference
1	395	401	-6
2	390	387	3
3	434	438	-4
4	407	404	3
5	380	386	-6
6	397	405	-8
7	405	409	-4
8	378	377	1
9	379	378	1
10	397	404	-7
11	369	374	-5
12	400	402	-2
13	325	328	-3
14	389	385	4

Table 1. Temperature correlation – CFD vs. Test.

Several thermal issues have been identified and corrected through the use of the new methodology on actual vehicle development programs. Exhaust routing changes have been made to ensure sufficient clearance is provided to various components. These were achieved through multiple iterations of the under-hood environment for each exhaust routing option. The thermal results were balanced with vehicle impact and other performance requirements. The simulations allow for informed decisions to be made early in the design process. Incorporation of CFD simulation in the thermal management development process have resulted in nearly $2.5 million cost savings to date for a single vehicle program. Currently, the tool is in the process of being applied for more vehicle programs.

CONCLUSIONS

The application of a novel mesh generation methodology to perform vehicle thermal management simulations efficiently is presented. The new approach overcomes limitations of traditional B2I meshing techniques through the use of I2B approach wherein the need for a water-tight surface mesh is not a pre-requisite for volume mesh generation. This results in significant efficiency improvements in the mesh generation process. The application of this new approach is demonstrated for real life automotive under-hood thermal management simulations. A fully coupled convection radiation simulation is performed to obtain the complete thermal map of the vehicle's underbody environment consisting of several components and assemblies. The model predicts the component temperatures for conditions corresponding to maximum exhaust skin temperatures. The model has been validated against data obtained from actual vehicle tests. The correlation between the physical tests and the digital simulations is excellent. The new methodology reduces the time taken for performing under-hood thermal simulations from several weeks to a few days. This makes it possible to perform vehicle under-hood simulations and provide timely input to the design development process. The new approach has resulted in significant cost savings through reduction in physical testing, prototype and re-designs costs.

ACKNOWLEDGEMENTS

The first author wishes to acknowledge technical support received from Michael Zabat and Mark Gleason, Aero/Thermal Development, DaimlerChrysler Corporation during the course of this work. Thanks are also due to Ryan Fortier, Aero/Thermal Development, DaimlerChrysler Corporation for running the physical tests and providing data for correlation.

REFERENCES

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

[2] 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 94-0771

[3] Hsu, I.F., and Schwartz, W.S "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", 1995, SAE 95-0616.

[4] Srinivasan, K., Jan, J.Y., Sun, R.L., Gleason, M.E., "Rapid Simulation Methodology for Under-hood Aero/Thermal Management", 1999, Int.J.of Veh. Design, **Vol. 23**, Nos ½,

[5] Yang, Z., Bozeman, J., Shen, F.Z., "CFD for flow rate and air recirculation at vehicle idle conditions", 2004, SAE 2004-HX21.

[6] Thompson JF, Warsi ZUA and Mastin CW. "Boundary-fitted coordinate systems for numerical solution of partial differential equations - a review",1982, J. of Comp. Physics; **47**, pp 1-108.

[7] Wang, Z.J., and Srinivasan, K. "Complex 'dirty' Geometry Handling with an Interior-to-Boundary Grid Generation Method", 2001, AIAA 2001-2538, 15th AIAA CFD Conference, Anaheim, CA.

[8] Wang, Z.J., and Srinivasan, K. "An Adaptive Cartesian Grid Generation Method for 'Dirty' Geometry", 2002, Intl J. for Numerical Methods in Fluids, **Vol. 39**, pp. 703-717.

[9] Jiang J., Przekwas, A.J., "Implicit pressure-based incompressible N-S equations solver for unstructured meshes", 1994, AIAA 94-0305.

[10] Tan, Z.T., Wang, D., Srinivasan, K., Przekwas A.J., Sun, R.L.,"Numerical simulation of coupled radiation and convection for complex geometry", 1999, AIAA 98-2677

