

UNDERHOOD AIRFLOW PREDICTION USING VECTIS COUPLED TO A 1-D SYSTEM MODEL

T.G. Bancroft, S.M. Sapsford, D.J. Butler
Ricardo Consulting Engineers Ltd.,
Shoreham-by-Sea, UK

ABSTRACT

Vehicle thermal management is an area of increasing importance in new vehicle product development, consuming significant resources and vehicle thermal development extends over a large proportion of the total programme duration. Ricardo led a European Consortium (VTHERM) to develop new Computer Aided Engineering (CAE) methods in this area of the design process to provide a competitive advantage in the form of substantial reductions in the time-to-market for new products. The resulting process combines the use of a 1-D system thermal model and a 3-D detailed air flow model using Computational Fluid Dynamics (CFD). This paper describes in detail the use of the CFD model. A detailed description of the 1-D system model has been presented in [1]. The method of utilising and preparing CAD data to form the surface geometry file for CFD use is described and is followed by a presentation of the validation of the Ricardo VECTIS CFD calculation. A case study using the process is then described and the paper finishes with an overview of the method to link the 1-D system thermal model and the 3-D CFD model together with validation data.

1 INTRODUCTION

Vehicle thermal management has become an increasingly important aspect of vehicle design as increased engine power, the introduction of underhood encapsulation features for NVH, cabin comfort demands and package space limitations create an increasingly difficult problem to solve. The need for a cost effective solution which satisfies system performance targets means a more detailed understanding of system behaviour is required to engineer an optimised product. To compound the situation there is also increasing demand to reduce product development times to below two years in order to reduce cost, respond to market trends and provide a return on investment more quickly. The work described in this paper has been carried out as part of a part-EU funded consortium project VTHERM. The project was initiated in response to the industry demands outlined above.

To respond to these demands, the VTHERM programme has developed a new approach for vehicle thermal management analysis. The approach involves the use of a 1-D system thermal model which may be constructed at the start of a vehicle programme and may be developed in line with the product. The details of the system model are described in [1]. The system model may be linked to a detailed air flow model of the underhood space, with temperature, heat transfer coefficient and air velocity data automatically exchanged between the two models using the approach described in the last part of this paper.

The detailed air flow model is based on a CFD calculation with geometry derived from CAD data files. The purpose of the VTHERM project has been to develop a data processing route which can produce the CFD model more rapidly than previously possible and to assemble surface geometry from CAD data files taken from source. This has required a flexible process to be created so that engine geometry and vehicle geometry may be taken from different CAD packages, which is commonly the case.

2 OBJECTIVES

- To determine a methodology for constructing a computational mesh using CAD data from any source
- To carry out a baseline calculation of the underhood environment and validate against test data
- To define the process to link the 1-D system model and the 3-D CFD detailed model

3 CONSTRUCTING A COMPUTATIONAL MESH

3.1 Geometry Preparation

The main problem associated with underhood analyses has been the time taken to generate the computational mesh. Reductions in the time taken have usually been realised by simplifying the model geometry (removing components) at the expense of model resolution and accuracy [2][3]. The quality of CAD data is one of the principal factors governing the time taken to generate the mesh. The presence of defects can incur considerable time penalties to prepare the geometry for subsequent CFD use as manual trimming, partial deletion and reconstruction may be necessary. It is important, therefore, that the final use of the CAD model should be considered by the designer responsible. Where possible, problems should be checked for using the original CAD package, where the defects can be most easily repaired for CFD surface geometry use.

Even good quality CAD data still needs to be processed to make it suitable for CFD use. This typically consists of removing unnecessary detail (inside of pipes etc.), adding thickness to thin components (body panels etc.) and removing excess surface detail. These processes are illustrated in Figure 1. Components processed in this way can be combined together using boolean add operations to build up sub-assemblies such as body panels, subframes, undertray and powertrain. For the work presented here, all geometry preparation was carried out using Imageware's Rapid Prototyping Module (RPM) and VECTIS. Preparation time depended on the complexity of the part but the subframe illustrated took about one hour to prepare, simple components take minutes.

The sub-assemblies (as shown in Figure 2a) are combined to form the complete model. The final stage of geometry preparation is to put the vehicle in a suitable virtual test chamber (Figure 2b). Provided the CAD data is good quality requiring little manual manipulation, the total surface geometry may be constructed in approximately 3 to 4 weeks. To further speed up this process would require greater integration of CAD and CFD. At present CAD models are not usually built with the downstream analysis requirements considered. If more of the combining and defeaturing was done to the original CAD model to produce sub-assemblies, or even a completely stitched vehicle model, the time taken to produce the CFD mesh could be reduced dramatically.

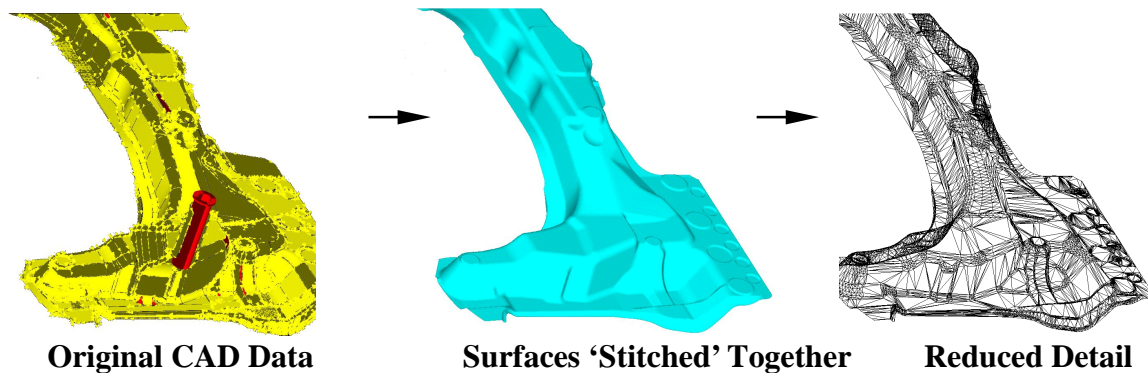


Figure 1 Summary of CAD Data Preparation for CFD Surface Geometry

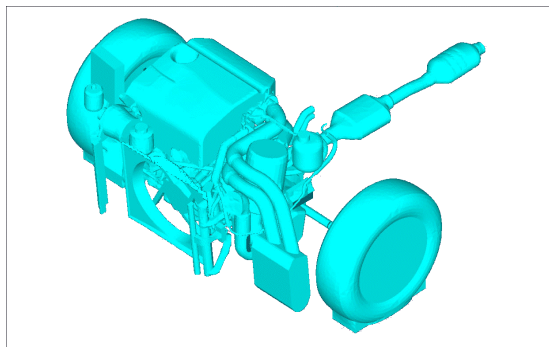


Figure 2a Powertrain Surface Geometry

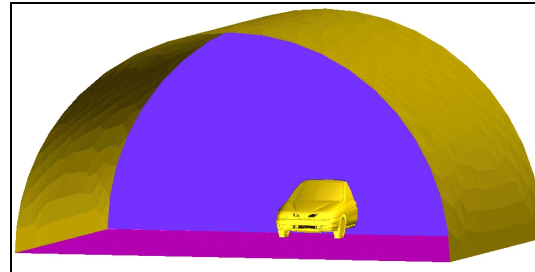


Figure 2b Complete Surface Model

3.2 Mesh Generation

VECTIS has an automatic mesh generator which requires only a surface description of the flow geometry and a control mesh. A cross section through the final mesh is shown in figure 3. The control mesh specifies the global cell sizes and the extent to which these global cells can be further subdivided and refined. Cell sizes ranged from 5mm on surfaces in the underhood region to over 500mm far from the vehicle. Rapid mesh generation is thereby achieved without compromising computational accuracy.

Heat exchangers were captured within the calculation mesh so that the cores were contained completely within a block of global cells. These blocks replace the actual cores which were removed during the geometry preparation stage leaving the end caps. The fan was represented during geometry preparation as a short cylinder with the same diameter and thickness as the real fan. In the calculation, a flow boundary was then specified on either side of this cylinder to represent an inlet and outlet conditions of the fan.

With the above cell sizes typical mesh size was 1.65 million cells. VECTIS took approximately 5 hours to generate this mesh on a SUN ULTRA 2.

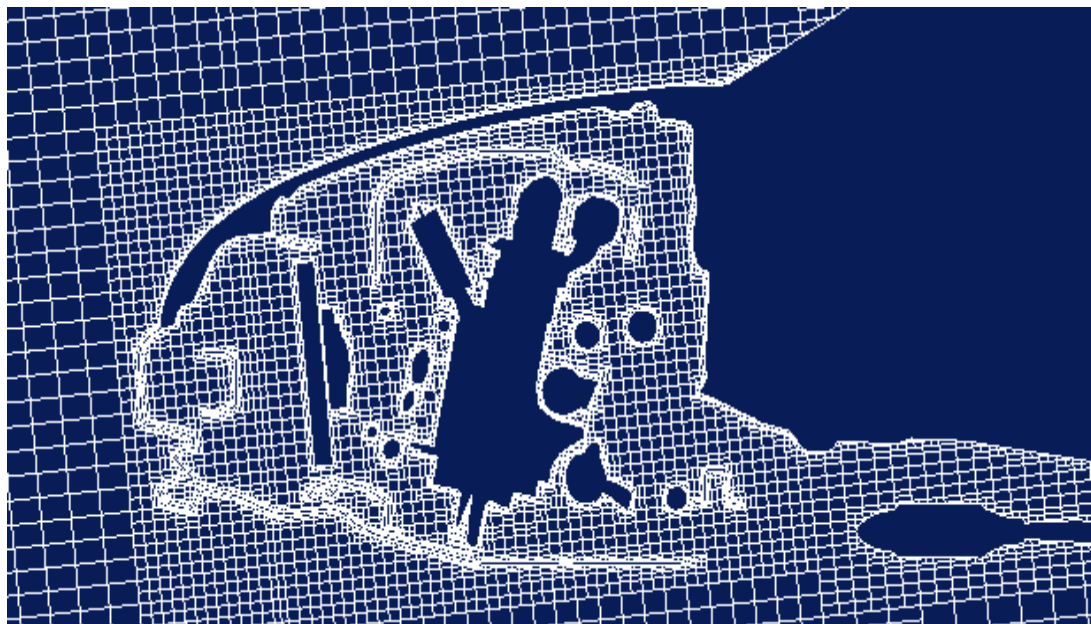


Figure3 Cross section through final mesh

4 COMPUTATION RESULTS AND VALIDATION

4.1 Test Configuration

For the validation exercise, the car analysed was a Renault Megane Turbo Diesel. The vehicle geometry was provided by Renault as part of the VTHERM programme. The vehicle was fitted with air conditioning and a sound deadening cover encased the upper part of the engine. For the baseline calculation, the vehicle was fitted with a full undertray extending past the engine and transmission unit. The test condition analysed was 80km/h wind speed with radiator fan off.

4.2 Analysis Techniques

The VECTIS calculation solves the three-dimensional flow equations governing conservation of mass, momentum and energy and the k- ϵ turbulence model. For this calculation, the flow domain extended in front of the car about 2 car lengths. As shown in Figure 2, either side and above the car the domain was extended out to the wind tunnel walls. Below the car the ground plane represented the edge of the domain and the rear edge of the domain was placed at the position of the B-pillar. This takes the outlet boundary far enough away from the area of interest. As with the corresponding measurements the floor was fixed and the wheels were stationary.

The heat exchangers were modelled as porous media. The flow was restricted to flow in one direction through the heat exchanger cores. Core restrictiveness was represented by specifying a quadratic relationship between pressure drop across the core and velocity through the core derived from experimental tests. A similar relationship between heat transfer coefficient and velocity can also be specified. In future a coupled 1-D system thermal model will be used to calculate heat rejections as described in section 6.

The radiator fan was modelled in a simplified fashion as an actuator disc, coupling together the inlet and outlet (Figure 4). All the outlet conditions are determined by the inlet conditions modified to take account of the fan characteristics.

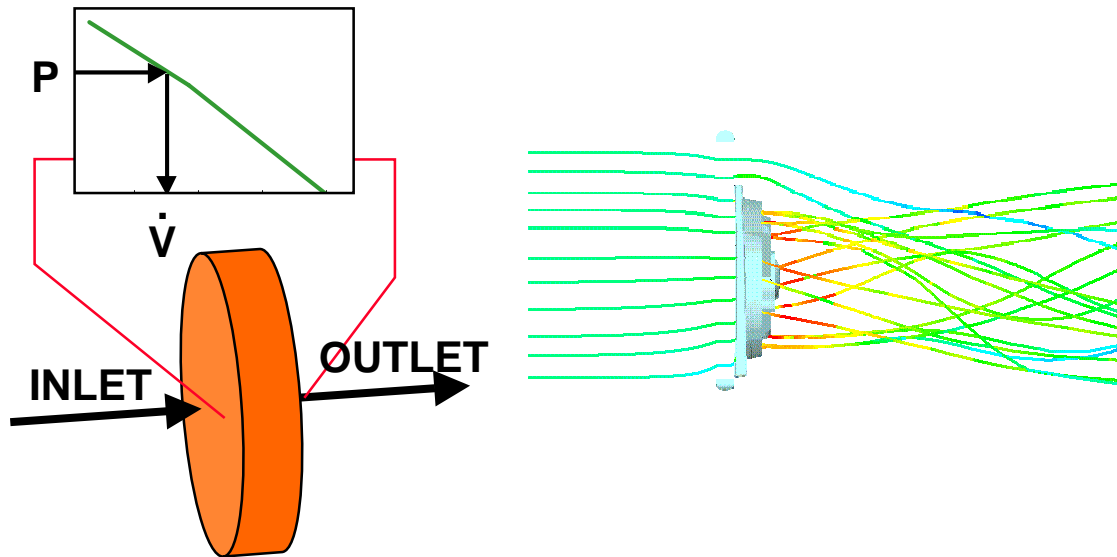


Figure 4 Simplified fan model

The fan characteristics needed are the operating curve relating the change in pressure across the fan to volume flow through it, the tip and hub diameters and the fan blade shape; straight or twisted. Although this model is quite simple, the data needed is easily available from component suppliers; it is also robust and efficient.

4.3 Validation

An essential part of the implementation of analytical tools is the validation of the calculation using experimental data. For this purpose, Laser Doppler Anemometry (LDA) data obtained from the Pininfarina wind tunnel facility were used. Again, these data were provided by Renault as part of the VTHERM programme. To assist in interpreting the data, the LDA measurement locations are shown in Figure 5. Note that the points are arranged in lines across the vehicle (y coordinate direction), the comparisons are presented along these lines.

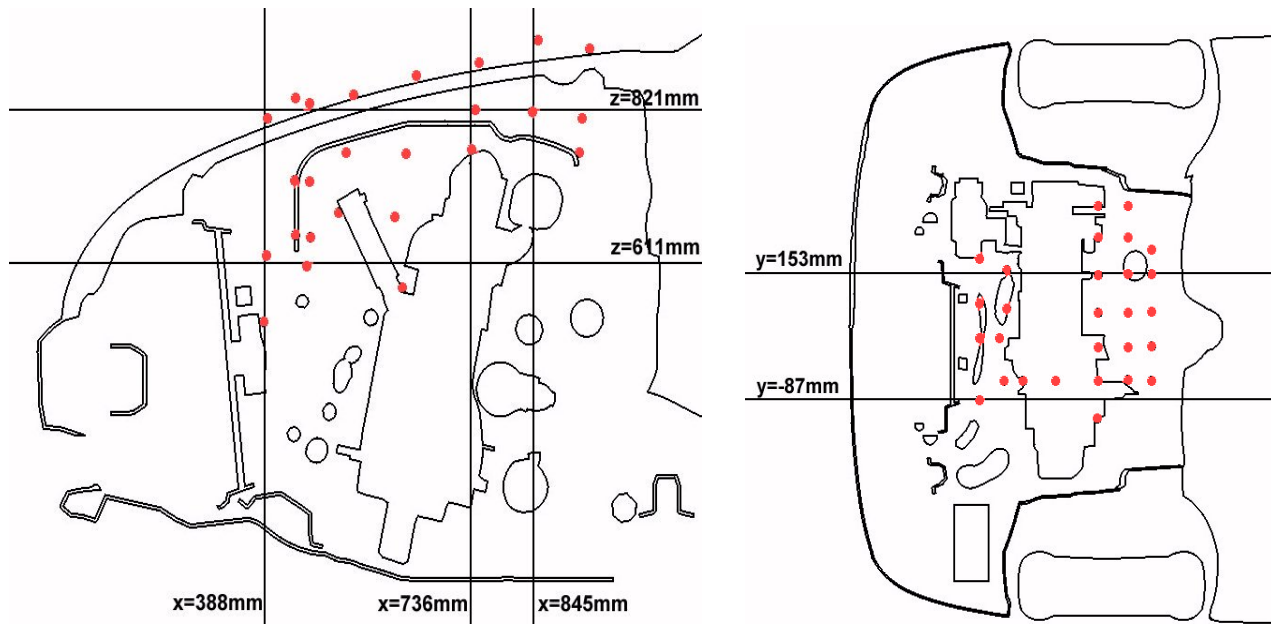


Figure 5 LDA Measurement Locations Superimposed on Vehicle Sections

Comparisons of air velocity were made for points defined from the experimental work which were split into three zones; between the radiator and engine ($x=388\text{mm}$), above the engine ($x=736\text{mm}$) and behind the engine ($x=845\text{mm}$). A comparison between the predicted and measured data is shown in Figure 6. For the comparison, the geometry was defined using a 3-D co-ordinate system as shown in Figure 5 with the $x=0$ plane positioned at the nose of the vehicle, $y=0$ on the centreline of the vehicle and $z=0$ on the floor.

In comparing the CFD and LDA data, two uncertainties associated with the data have been accounted for; location of measurement probe and stability of the LDA measurement. The LDA data is plotted with an error bar of \pm one standard deviation. The CFD data is plotted as a value for the co-ordinate cell and the error bar represents the maximum and minimum range in the adjacent cells. Correlation between the two data sets is considered good if the bands overlap, which is generally the case and the trends are well predicted. The profile of velocity variation along the y -axis appears to be reasonably good with spatial variations detected as shown in Figures 6b and 6c. There are instances, however, where there is relatively poor agreement between the predicted and measured air velocities. Two such cases occur at the co-ordinate points (388,-87,611) in Figure 6a and (736,153,821) in Figure 6b, both of which are highlighted with a red box. The underhood flow velocity fields in the y -plane were plotted for these co-ordinates as shown in Figure 7.

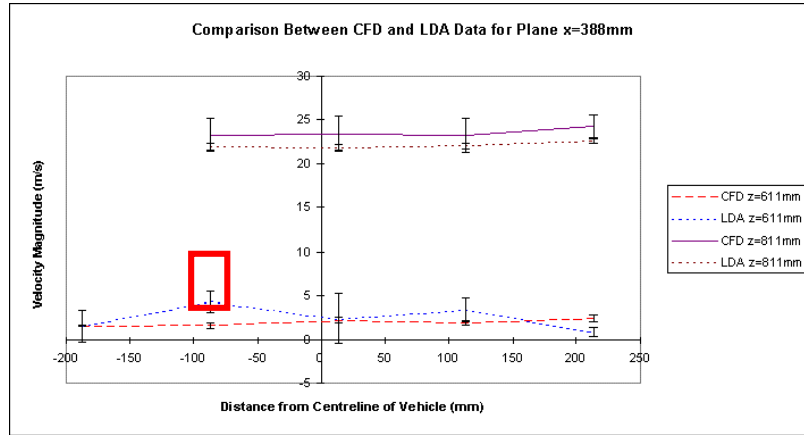


Figure 6a Comparison Between CFD and LDA Data for Plane x=388mm

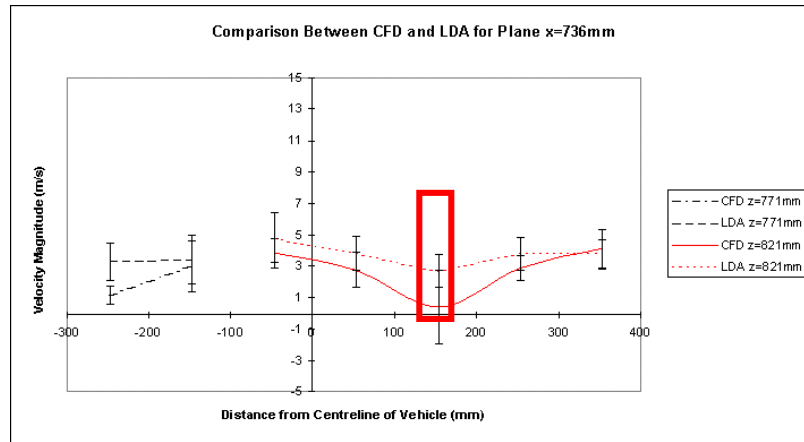


Figure 6b Comparison Between CFD and LDA Data for Plane x=736mm

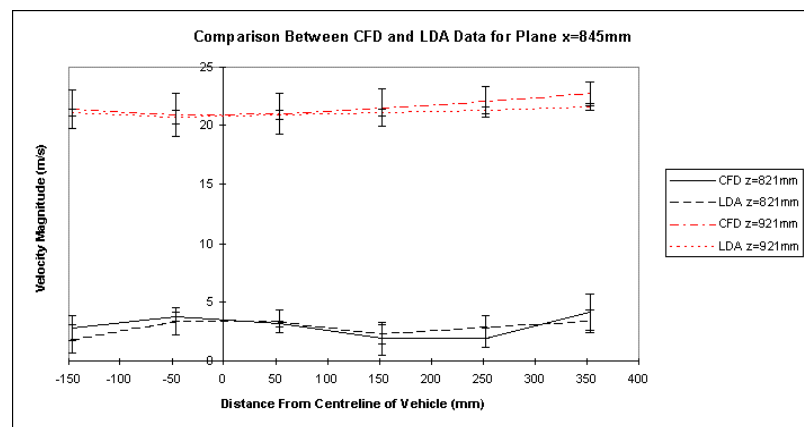
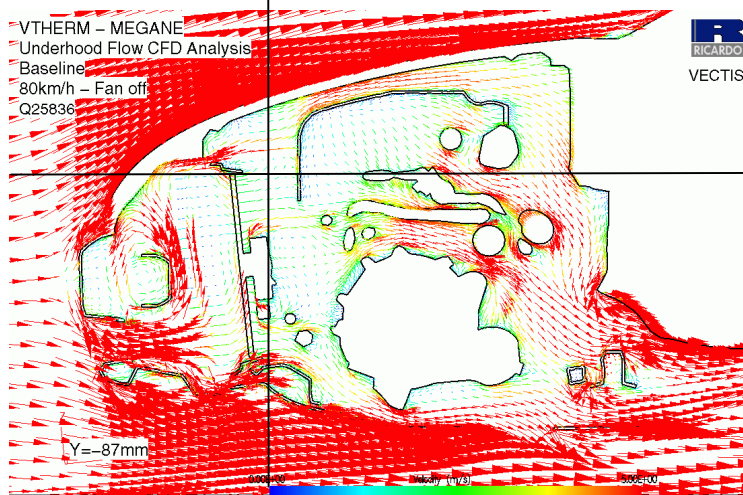
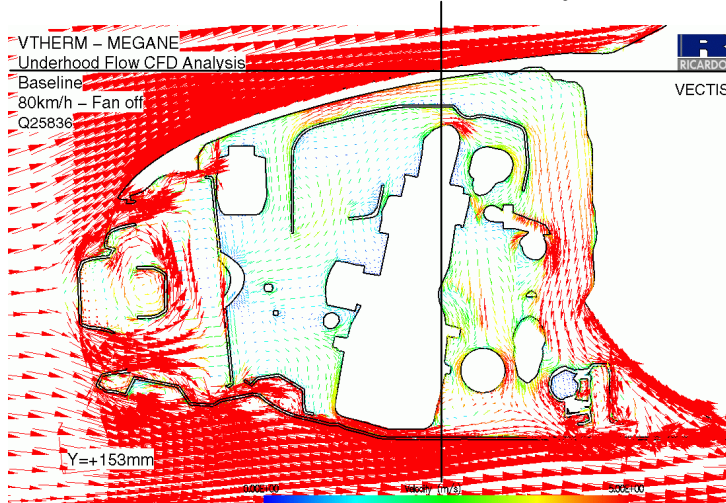


Figure 6c Comparison Between CFD and LDA Data for Plane x=845mm

For the plane $y=-87$ (Figure 7a), the co-ordinate lies behind the radiator in the wake of the outer edge of the fan cowl. When the fan is switched off, the CFD fan model forces the flow to move through the domain in a straight path; no fan-induced component was applied to the flow to simulate the effect of the fan blade as it freewheels. Other studies [6] have indicated that the unpowered fan may influence the flow and so not including this effect may be a source of discrepancy between the CFD calculation and the LDA measurement. An unpowered fan may be simulated using the fan model within VECTIS but the required parameters were not available at the time these calculations were performed.



**Figure 7a Air Velocity Prediction for $y=-87\text{mm}$ (Vehicle Centreline at $y=0$),
Co-ordinate (388,-87,611) Marked by Cross Hairs**



**Figure 7b Air Velocity Prediction for $y=153\text{mm}$ (Vehicle Centreline at $y=0$),
Co-ordinate (736,153,821) Marked by Cross Hairs**

For the plane $y=153$ shown in Figure 7b, the co-ordinate (736,153,821) lies between the engine cover and the underside of the hood. As shown in Figure 6b, there is a minimum in the air flow velocity at this location. This minimum is detected in both the CFD and the LDA data but is more severe in the CFD prediction. This is a region of high velocity gradient as indicated by the high range of predicted values in the cells adjacent to the co-ordinate shown in Figure 6b. Thus any small variation in measurement probe location will contribute to the difference shown in Figure 6b.

As shown in Figure 6, other areas of disagreement also exist which are expected to be due to features present in the test vehicle but not the CFD model (eg. wiring loom). Despite these issues, it is considered that the VECTIS results have generally correlated well with the experimental data and that VECTIS is an appropriate code for predicting underhood flow behaviour. Experience suggests that non-systematic local discrepancies in the predicted flow field do not significantly influence overall predictions of thermal performance.

A case study showing how this verified model has subsequently been used to assess design changes made to the vehicle has been presented previously [7].

5 LINKING WITH 1-D SYSTEM THERMAL MODEL

As already mentioned, a 1-D system thermal model was developed during the VTHERM project and is described in [1]. The system thermal model is capable of predicting cooling performance prior to CAD data being available, but once CFD analysis is available, coupled detailed air flow and thermal calculations may be made. This offers the opportunity to carry out detailed analysis with a low additional overhead in model preparation.

5.1 Thermal model setup

The thermal model is constructed using 1-D fluid flow network analysis software. There are a number of packages which could be used for this purpose. Ricardo have chosen the proprietary software FLOWMASTER2™ because of the availability of standard components which can be used directly in automotive applications together with the ability to customise specific areas and include new techniques. A typical 1-D network built in FLOWMASTER2 is shown below in Figure 9.

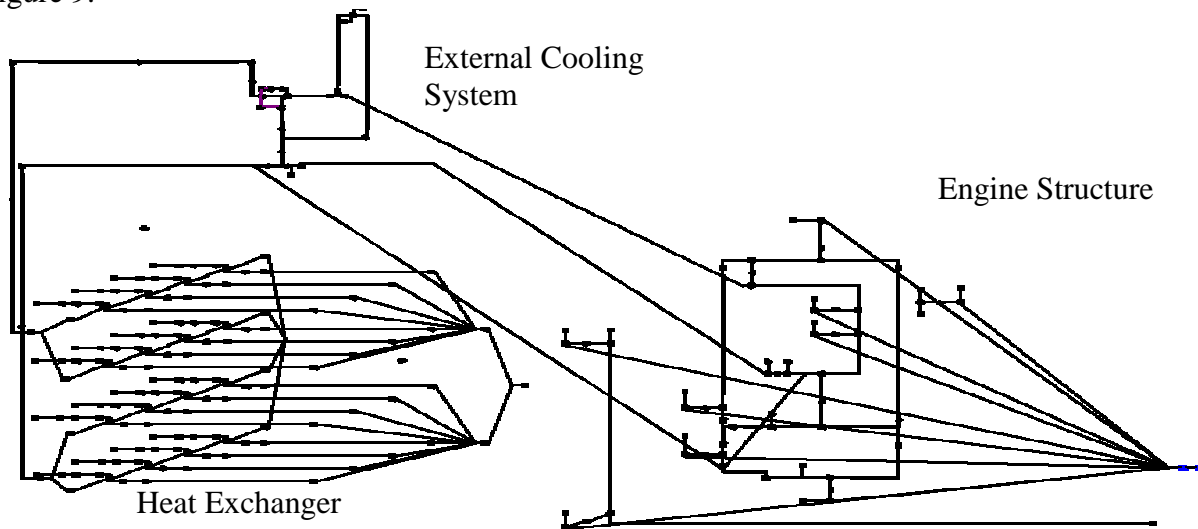


Figure 9 Schematic of 1-D network model

The network includes fluid circuits for the water, oil and external air flow including pumps, valves, pipes, heat exchangers and structural conduction between fluid streams is also included. The data that describes these components is taken either from the component supplier or from classical models. The 1-D model includes a representation of the heat release from combustion into the structure of the engine. This is usually derived empirically but can also be modelled using performance simulation software, such as Ricardo's WAVE package. When the 1-D

model is coupled to the 3-D CFD model the external air flow parameters (flow rates, temperatures and heat transfer coefficients) are taken from the CFD model instead of the 1-D airflow network.

5.2 CFD model setup

The only additional procedure needed to prepare a CFD model is to identify the engine and vehicle surfaces and heat exchangers which are to be coupled. The surfaces are identified by a simple process of painting the boundaries, each colour then uniquely identifies a boundary which has a corresponding component in the flowmaster model. For the heat exchangers it is necessary to define how many zones the core is to be split into to provide spatial resolution of the flow through the radiator. Figure 10 below illustrates both these requirements, note that in this case the radiator has been subdivided into 16 sections.

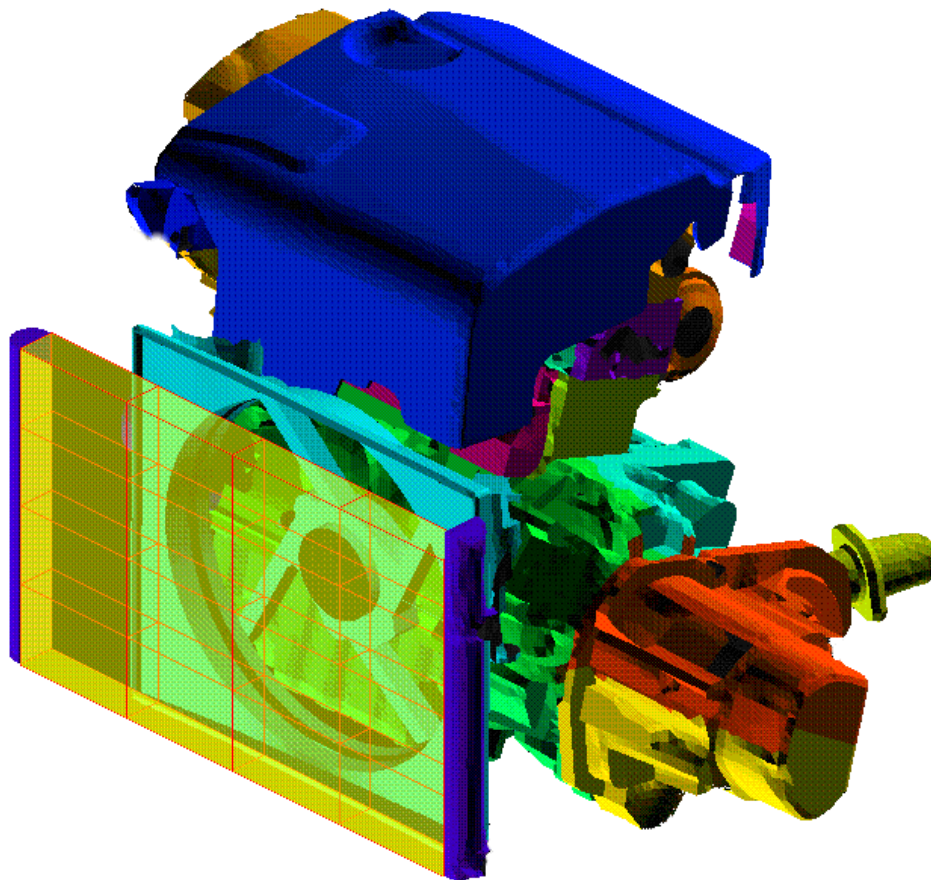


Figure 10 Boundary identification and radiator subdivision

5.3 Coupling

The coupling procedure is shown graphically in Figure 11. The coupling process is controlled by VECTIS, the CFD calculation is run to virtual convergence using estimated surface temperatures and radiator heat rejections. The CFD calculation then initiates an iteration of the FLOWMASTER 1-D network. The CFD calculation passes the area averaged air temperature and heat transfer coefficient for each surface and the upstream air temperature and flow rate through each section of the radiator. The FLOWMASTER2 calculation then uses these air side boundary conditions and recalculates the component and fluid temperatures. Also calculated is

the heat rejected from each section of the radiator. The surface temperatures and radiator heat rejections are passed from FLOWMASTER2 to VECTIS and the 3-D calculation resumes. This procedure continues until the CFD calculation converges and the exchanged values cease to change. At this point the calculation can be stopped and the results post processed.

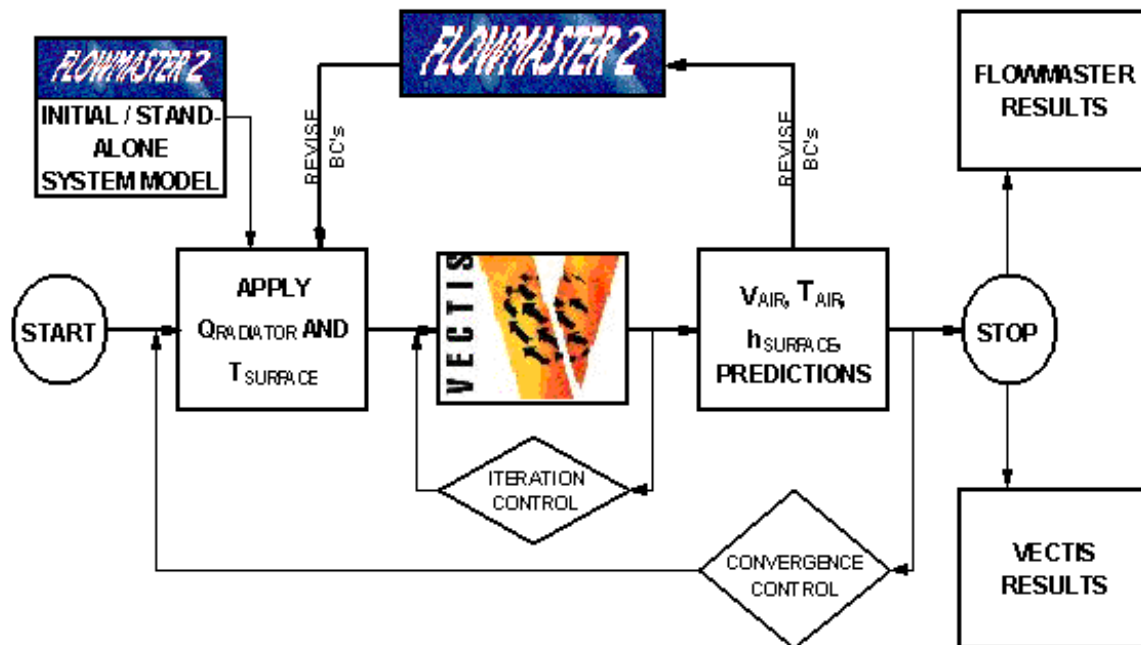


Figure 11 Coupling procedure

5.4 Coupled results

The additional results that can be obtained in comparison to a standalone uncoupled CFD analysis are, for example, full air field temperature prediction (Figure 12). This information can be used to assist packaging studies, the placing of sensitive components and the provision of shielding can be assessed rapidly.

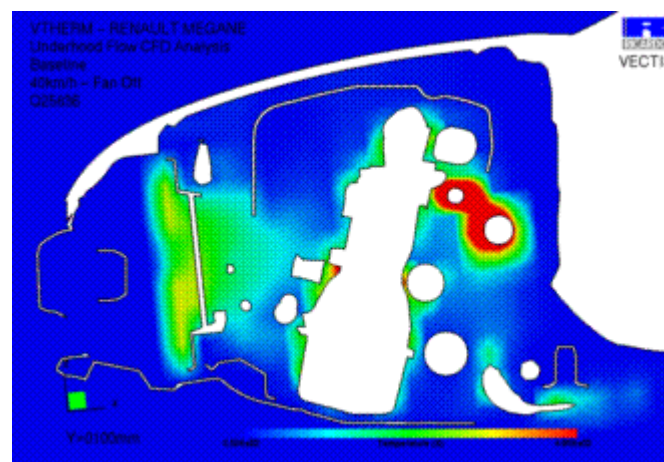


Figure 12 Air field temperature prediction

The temperatures of coupled components (Figure 13) are also calculated. This data allows rapid assessment of the temperatures components will reach and whether or not they will be durable (eg. plastic covers).

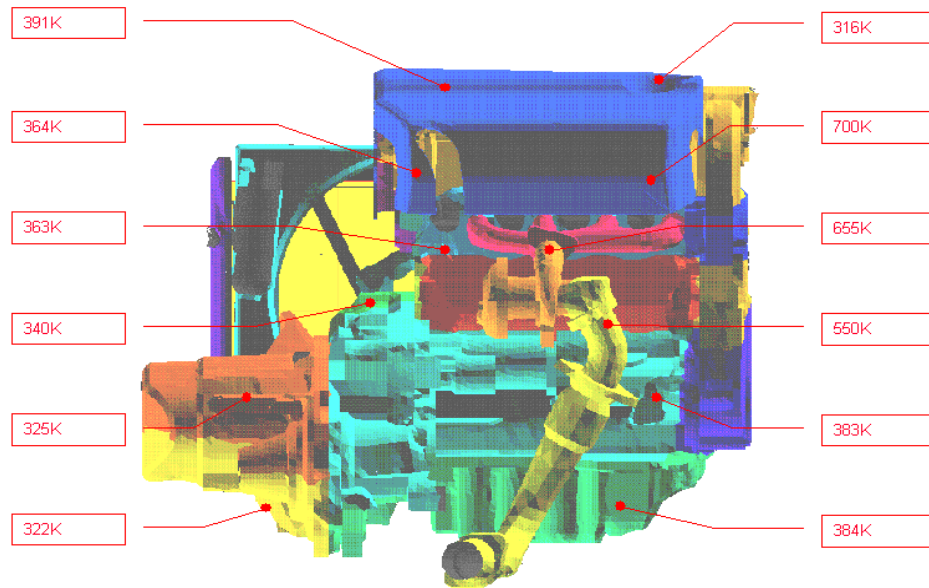


Figure 13 Surface temperature predictions

The oil and water temperature throughout the circuit is also predicted.

5.5 Validation of coupled procedure

As part of the VTHERM programme thermal validation was also performed to assess the suitability of the analytical techniques. Three test vehicles were subject to extensive thermal surveys the results of which were compared to those predicted by the models. Only two of the vehicles (Renault Megane Turbo diesel and Alfa Romeo 155) were analysed using a full coupled 1-D/3-D technique.

The results obtained were encouraging but suggested that some further refinement of the procedures is needed. At low and medium speeds (upto 80km/h) the correlation between the test data and the predicted results was generally very good with the metal and fluid temperatures predicted to within a few degrees. Figure 14 below shows a comparison between measured and predicted component and fluid temperatures results for a 60km/h case using the Renault Megane.

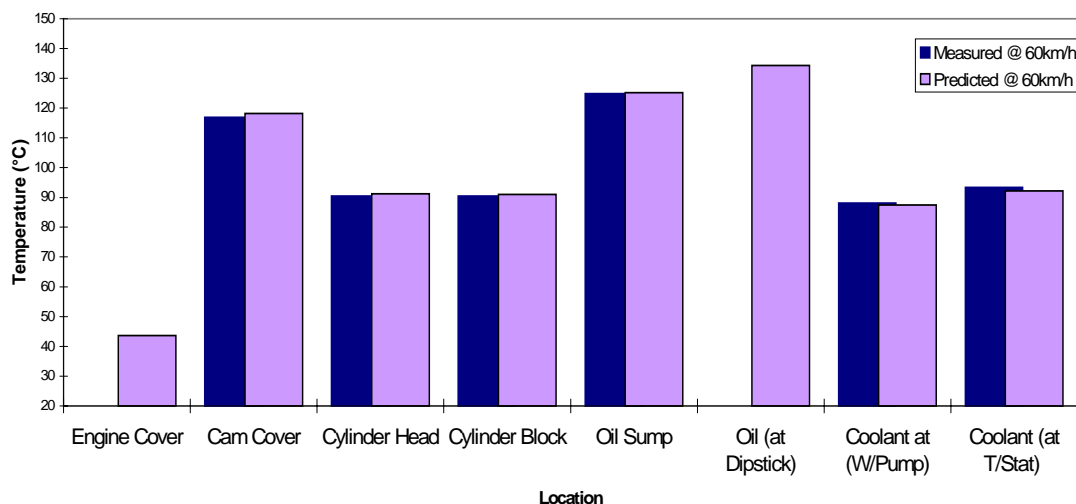


Figure 14 Comparison of predicted measured component/fluid temperatures

At higher speeds (165km/h) the correlation with the measured data was not so good. The calculations tended to underpredict the temperatures. This is thought to be the influence of the boundary conditions. The rear outlet boundary was placed near the B-pillar of the car. This was thought to be far enough away from the area of interest. A fixed mass flow was then applied to this boundary, effectively imposing a mass flow split between the flow over the car and the flow underneath the car. At high speeds this was causing an overprediction of the amount of air that would go underneath the car which in turn caused an excess of air to be drawn through the radiator. To solve this problem a different type of boundary condition was tried which allowed the upstream conditions to control the mass flow split instead of the boundary itself forcing the split. This approach improved the results but the preferred solution would be to include more of the vehicle to allow the flow split to be correctly calculated. Other studies [8] have shown that as long as the boundary is taken far enough away from the underhood region the external model need only be very simple. Indeed in [8] it is suggested that it may be sufficient to extrude the vehicle section at the B-Pillar back to the full vehicle length and impose the boundary condition there, hence no additional CAD is required. To date this has not been investigated at Ricardo.

6 CONCLUSIONS

- A flexible new process has been developed which enables CAD data to be taken from multiple sources and combined to form a single model for underhood air flow calculation. In developing the process, the quality of CAD data has been identified as one of the principal factors governing the time taken to generate the mesh. At present, the most efficient way of improving the quality of the CAD data is at source.
- The Ricardo VECTIS CFD software has been used successfully to generate a CFD calculation mesh of the underhood flow domain in approximately 5 hours and quickly calculate solutions for different geometry cases.
- A CFD calculation has been validated against wind tunnel LDA data provided by Renault resulting in generally good agreement between the two data sets. Accounting for unpowered fan rotation in the fan model and checking/including details such as wiring loom layout is expected to improve the accuracy of the prediction.
- A process for linking the CFD calculation with a 1-D system thermal model has been presented. Promising preliminary results at low and medium vehicle speeds using the method have been presented. Refinement of boundary condition implementation is expected to improve results at all vehicle speeds.

7 ACKNOWLEDGEMENTS

The authors would like to acknowledge the support of the European Commission for their support in funding this work. They would also like to thank Renault SA for their contribution of data presented in this paper, the other partners of the VTHERM consortium for their contributions to the project and the directors of Ricardo Consulting Engineers Ltd for permission to publish this paper.

8 REFERENCES

- [1] Modelling of Vehicle Thermal Systems, D.J. Butler, S.P. Stevens, VTMS4 Paper L07/C543-57.
- [2] Prediction of Aerodynamic Drag and Lift of the Nubira Hatchback Vehicle, N. Haidar, E. Draper, 2nd MIRA International Conference on Vehicle Aerodynamics, October 1998.
- [3] Computational Modelling of the Underbonnet Cooling Flow Characteristics of the New Daewoo Van, N. Haidar, E. Draper, 2nd MIRA International Conference on Vehicle Aerodynamics, October 1998.
- [4] VECTIS CFD Version 3.3 User's Manual, Ricardo Consulting Engineers Ltd..
- [5] Measurements and Prediction of Coolant Velocity in Internal Combustion Engine Cooling Systems, M.R. Heikal, S.G. Cox, R.E. Morgan, N.J. Owen, IChemE 5th UK National Heat Transfer Conference, 17-18 September 1997.
- [6] CFD Quality - A Calibration Study for Front-End Cooling Airflow, J. Williams, G. Vemaganti, SAE 980039.
- [7] Improving the effectiveness of underhood airflow prediction S.P. Stevens, T.G. Bancroft, S.M.Sapsford, VTMS4 Paper C543/058/99
- [8] Numerical sensitivity study on the prediction of radiator performance using CFD underhood airflow simulation, M. Beier, G. Seider, VTMS4 Paper C543/072/99

FLOWMASTER2® is a registered trademark of AMSTRAL HOLDING BV in the United Kingdom, France and Germany.