Melbourne Graduate Fluids Conference, 2001
Monash University, Melbourne, Australia

COMPARISON BETWEEN EXPERIMENTAL AND NUMERICAL METHODS FOR EVALUATING CAR COOLING SYSTEM DESIGN

K. JOHANNESSEN[1], J. SAUNDERS[1], J. SHERIDAN[1] and D. NICLASEN[2]

[1]Department of Mechanical Engineering
Monash University, Clayton, Victoria, AUSTRALIA
[2]Holden Ltd., Port Melbourne, Victoria, AUSTRALIA

ABSTRACT

Numerical studies of cooling airflow using computational fluid dynamics (CFD) software allow engineers to analyse designs earlier in the design cycle and at a reduced cost compared to wind tunnel experiments. However, testing is required to confirm if the CFD codes are correctly modelling the flow near the radiator.

The cooling performance of a car has been studied numerically using the Computational Fluid Dynamics (CFD) code Fluent and experimentally by measuring the Specific Dissipation of a radiator. Modifications were made to the fascia of the car and the cooling performance determined using both the numerical and experimental methods. It was found that the CFD code was accurate in predicting the trends when the fascia was changed. The seven variations tested were ranked correctly with the exception of one pair of results. Flow visualisation conducted in the wind tunnel confirmed the presence of flow structures predicted by CFD models

1. INTRODUCTION

Over the last thirty years there have been increasing demands on the engine cooling system of a car. This has been caused by a steady increase in engine output, combined with a reduction in the size of the cooling inlets and an increase in auxiliary components (Emmelmann and Berneburg, 1990). A water-cooling system is almost universal in new cars using a mixture of water and ethylene glycol as the coolant fluid. This fluid passes through the engine extracting heat before being pumped to a compact heat exchanger (radiator). The radiator transfers heat from the coolant fluid to passing airflow by convection. The cooling air velocity comes from either the forward motion of the car (ram air) or from an fan enclosed in a shroud (fan air) attached to the radiator. In addition, a condenser required by the air conditioning system is often located in the same airflow as the radiator (Newton et.al, 1996).

The fascia covers the front of the car and generally contains radiator air inlets, front bumper and the number plate (see Figure 1). While the fascia of a car has a significant influence on the amount of air reaching the radiator, it is crucial to the appearance of the car and is often finalised early in the design cycle with physical prototypes being expensive. However, computational surface data is usually available on and a computer model can be created at a minimal cost. But, if computer modelling of the effect of fascia changes on cooling performance is going to be of any value, testing is required to determine the accuracy of CFD results relative to current experimental test results. This is made difficult because flow under the hood is complex with regions of flow separations caused by complicated geometry.

The most thorough experimental study of CFD accuracy was conducted by Williams and Vemaganti of the Ford Motor Company (1998). They measured the airflow rate through the radiator using nine vane anemometers for twenty-three geometry configurations on the front-end of the car. The same scenarios were modelled using a Reynolds-Averaged Navier Stokes solver with a standard k-ε turbulence model. The authors found that 14/23 cases were misranked by the CFD analysis however only five were misranked by three or more places. The numerical prediction of airflow rate was an average of 1.8% below experimental values with a 90% confidence of 11.8 % on the individual measurements. The CFD output indicated regions of reverse flow that would have been quite difficult to identify in wind tunnel experiments. The anemometers used in this study were 114mm in diameter and errors could have arisen from averaging across non-uniform flow.

2. EXPERIMENTAL METHOD

The mass flow rate of air through the radiator is difficult to measure accurately. Vane anemometers can be used but these can experience averaging errors and the removal of the fan and shroud is often required. Hotwires are too fragile and other techniques require visual access to the flow. As a result, the cooling performance presented here was found by measuring the Specific Dissipation (SD) of the installed radiator. The Specific Dissipation is a measure of the heat rejection of a radiator and is defined as the heat transfer rate divided by the maximum temperature difference across the heat exchanger given in equation (1):

$$SD = \frac{Q}{T_{coolant,in} - T_{air,in}} = \frac{\dot{m}_{coolant} \times C_{p,coolant} \times (T_{coolant,in} - T_{coolant,out})}{T_{coolant,in} - T_{air,in}} \quad [\mathrm{W/K}] \qquad (1)$$

The subscript coolant stands for the radiator coolant fluid, which is the hot fluid while the air acts as the cold fluid. This value is related to the heat exchanger effectiveness but does not require the measurement of the temperature of air exiting the radiator. This technique has been used previously on a simplified front end (Hird, Johnson & Pitt, 1986) and on complete cars (Lin et al.,1997)

Experimental measurements were conducted in the full-scale wind tunnel located at Monash University. A 30 kW heat bench supplied hot water to the radiator from outside the wind tunnel. The temperature of the coolant at the inlet and outlet were measured by T-type thermocouples, as was the ambient air temperature. The coolant flow rate is measured by a magnetic flowmeter. In this program eight fascias were tested having different sized frontal openings with the rest of the car unchanged.

3. NUMERICAL METHOD

The air flow around the car was modelled using the commercial CFD code Fluent (Version 5). In Fluent, a three-dimensional segregated solver was used to solve the steady Reynolds Averaged Navier-Stokes equations. The turbulence model used was a realizable k-ε type with standard wall functions. The solver used a first order upwind scheme and the pressure-velocity coupling was conducted using the SIMPLE algorithm

The model grid consisted of an unstructured triangular surface mesh created on all external surfaces back to the B-pillar and larger engine bay components (Figure 1). Using the Tgrid software package a tetrahedral volume mesh was created containing ~850,000 elements. Cells were distributed unevenly with the smallest cells located in regions of the largest velocity and pressure gradients.

The boundary conditions were based on test conditions measured in the wind tunnel (Saunders and Mansour, 2000). The inlet was set to a velocity of 16.67 m/s with turbulence levels of 3 % and a length scale of 0.2 m. The outlet was a pressure outlet type with the same turbulence constraints. The sides of the domain were symmetry walls whilst the floor was a no-slip wall, like the wind tunnel floor. The radiator and condenser were modelled as a porous media with flow resistance obtained from manufacturer's performance curves. Each case was solved until the residuals converged to less than 1×10^{-4}. In addition, the mass flow rate through the radiator was monitored to ensure convergence had been achieved.

4. RADIATOR AIRFLOW COMPARISON

4.1. RESULTS

While the CFD model solution provided the air mass flow rate through the radiator in kg/s, the Specific Dissipation test gave the heat rejection in W/K. Thus, the values could not be compared directly and as a consequence the percentage change from the initial baseline case is examined. This approach is acceptable in industry as the performance of a new car is normally based on an evolution from a previous model. The results obtained with the eight fascia configurations tested are shown in Table 1 with the airflow values, relative improvements from baseline and the ranking of the cases from 1 (worst) to 8 (best).

4.2 DISCUSSION

The results obtained suggested that the CFD model can predict performance trends showing whether a change would make the airflow greater or less. An error in this can lead engineers to pursue inferior designs. The fascias tested were correctly ranked in order except for one pair of data where the ranking swapped. The CFD model swaps the ranking of fascias C and D with respect to the experimental results. Further testing is required to identify the reason for this. Resolutionn down to 1% is required for commercial applications and further tests will be done to approach this range.

Condition	Numerical Results			Experimental Results		
	Air Mass Flow Rate kg/s	Change from Baseline and Ranking		Specific Dissipation kW / K	Change from Baseline and Ranking	
Baseline Fascia (BL)	0.67			1.57		
Fascia A	0.51	-23.8 %	1	1.39	-14.6 %	1
Fascia B	0.73	8.4 %	5	1.69	7.6 %	4
Fascia C	0.72	7.5 %	4	1.70	8.6 %	5
Fascia D	0.61	-8.7 %	2	1.45	-8.5 %	2
Fascia E	0.76	13.2 %	6	1.75	10.7%	6
Fascia F	0.84	25.2 %	7	1.45	17.4 %	7
Fascia G	0.66	-2.3 %	3	1.85	-8.0 %	3

Table 1 : Experimental and Numerical results

When looking at the magnitude of the change in performance from baseline the results in Table 1 show that the agreement was quite poor between the numerical and experimental results, especially for the cases with large changes in performance (eg Fascia A had reduction in performance –23.8% vs. –14.6%). This indicates that numerical methods were more sensitive than the experimental tests. It is not known which of these techniques is giving the correct results. Fascia G went against this trend with the experimental tests showing a significantly higher increase in performance. The reason for this was that this particular fascia had the grille moulded into it (unlike any others tested), which was not modelled numerically. The blockage of the real grille was significant enough to reduce the cooling performance by around 6%, which is reasonable according to previous tests. This test highlights the problem with CFD resolving small details like the grille elements.

5. FLOW VISUALISATION RESULTS

Yarn tuft flow visualisation was used to determine flow features between the fascia and the radiator. The features were :

- Air entering the upper nostril with some air passing over the radiator and the rest making a curtain of air about 3 cm thick seen flowing towards the top of the condenser.
- Air travelling forwards against the wind direction between the fascia and the front bumper bar.
- The effect of the air dam was to cause air to enter vertically through the bottom (breathing) opening. Air then flowed up the front face of the condenser.
- Flow circulating behind the numberplate and around the bumper bar.

All these features are present in the CFD output of velocity vectors as shown in Figure 2.

6. CONCLUSIONS

The Fluent CFD code has been used to model the cooling performance of a car with various fascia configurations. The results were compared with those obtained using a standard experimental test measuring the Specific Dissipation in the Monash University Full Scale Wind Tunnel. The tests showed the ability of CFD to predict performance trends and the ranking of the facias was correct for all but two configurations. The experimental and numerical techniques displayed a poor agreement on the magnitude of the change from baseline with numerical results were more sensitive than the experimental results. The CFD results showed clearly that the flow field between the fascia and the radiator is complex, separated flow. In the wind tunnel yarn tufts showed the presence of flow structures observed in the CFD output. This experiment has indicated the ability of using CFD as a valuable tool early in the design process with experimental validation completed before project sign-off.

7. ACKNOWLEDGEMENTS

The authors would like to acknowledge the generous support of Holden Ltd. with this program. The support of the Monash University Mechanical Engineering Department is also greatly appreciated.

8. REFERENCES

EMMELMANN, H.J. and BERNEBUR, H., "Aerodynamic Drag and Engine Cooling - Conflicting Goals?". *SAE Technical Paper 905128*, 1990.

LIN, C., SAUNDERS, J. W., WATKINS, S. and MOLE, L. "Increased Productivity - The Use of Specific Dissipation to Evaluate Engine Cooling". *SAE Technical Paper 970137*, 1997.

LIN, C., "Specific Dissipation as a technique for evaluating motor car radiator cooling performance", *Ph.D Thesis*, RMIT, 1999.

NEWTON, K., STEEDS, W. and GARRETT, T. K., "The Motor Vehicle", 12th Ed, Butterworth Heinemann, Oxfod, UK, 1996.

SAUNDERS, J.W. and MANSOUR, R.B., "On-road and Wind Tunnel Turbulence and its Measurement Using a Four-Hole Dynamic Probe Ahead of Several Cars", *SAE Technical Paper 2000-01-0350*, 2000.

WILLIAMS, J. VEMAGANTI, G., "CFD Quality - A Calibration Study for Front-End Cooling Airflow", *SAE Technical Paper 980039*, 1998.

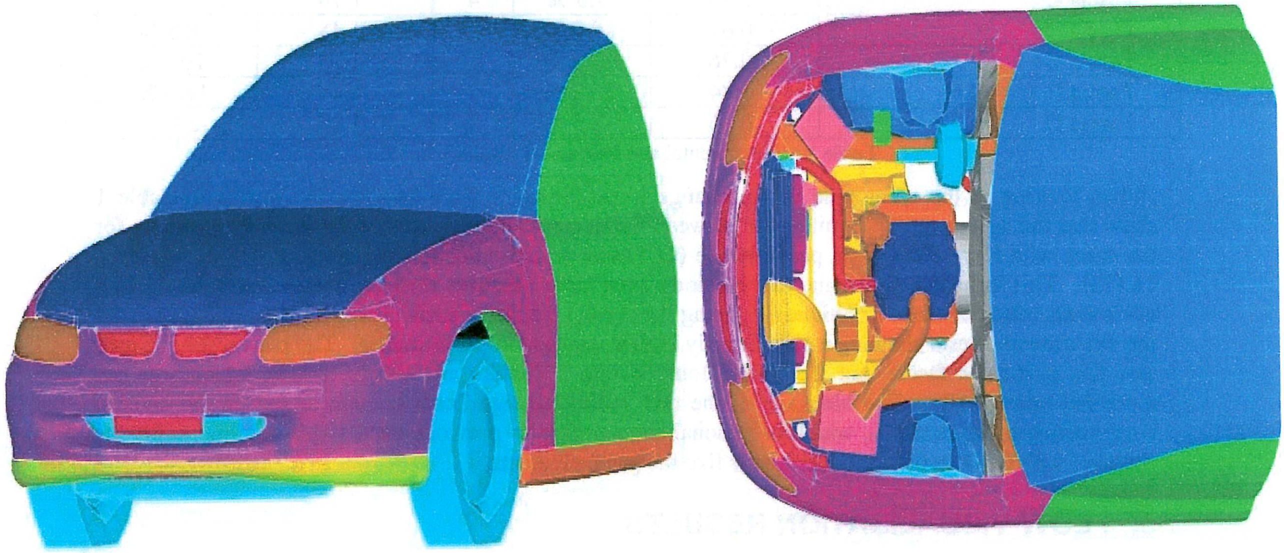

Figure 1 : Surface model of car body with engine components exposed

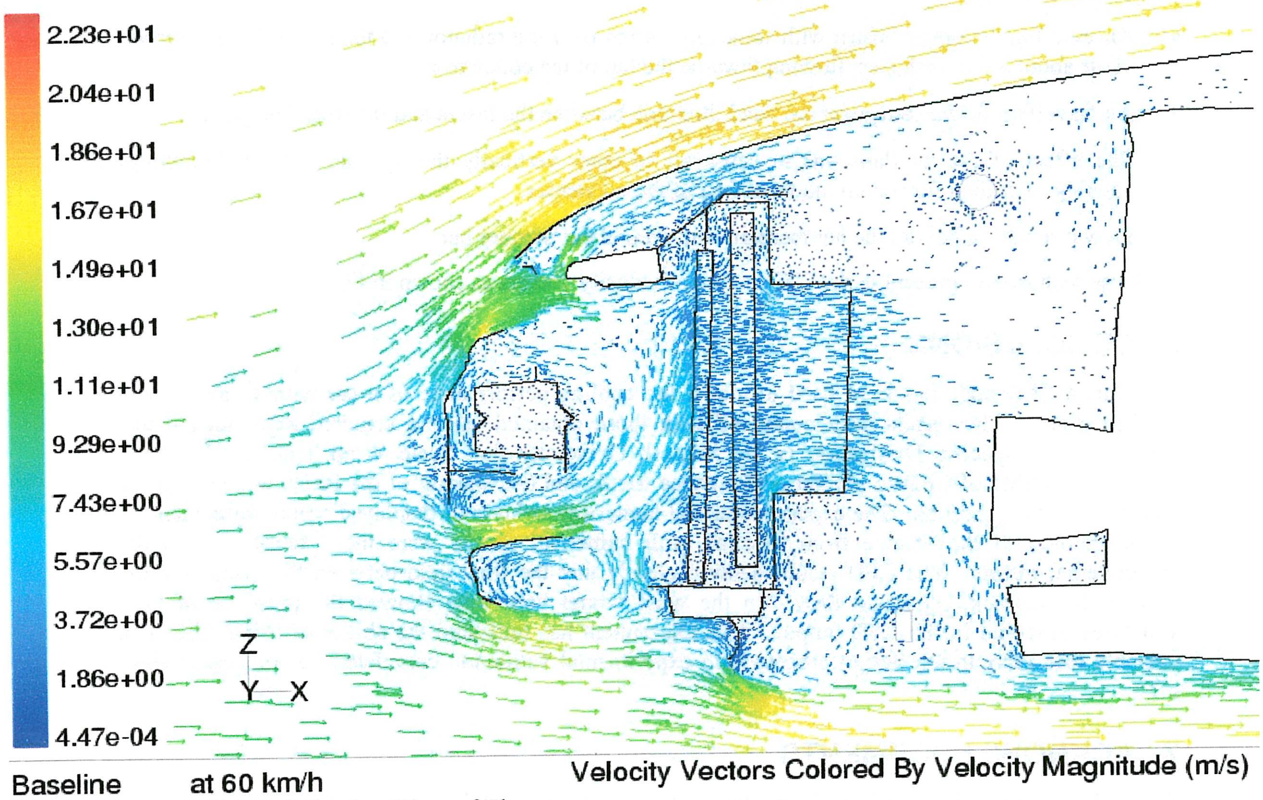

Figure 2 Section View of Flow