14th Australasian Fluid Mechanics Conference
Adelaide University, Adelaide, Australia
10-14 December 2001

Measuring Local Time-Averaged Airflow Velocity through an Automotive Heat Exchanger

Eton Y. Ng[1], Simon Watkins[1], Peter W. Johnson[1] and Lindsay Mole[2]

[1]Department of Mechanical and Manufacturing Engineering
RMIT University, Victoria, 3083 AUSTRALIA

[2]Product Development
Ford Motor Company of Australia, Victoria, 3220 AUSTRALIA

Abstract
In some applications, the airflow in an air-cooled cross-flow compact heat exchanger is complex and non-uniformly distributed. The present study reveals the considerable lack of uniformity across the radiator air face for typical automobiles. This paper presents the development of an experimental technique for measuring the local time-averaged airflow velocity through radiators. Test results using a full-scale vehicle tested in a wind tunnel are presented, including flow visualisation around the radiator and contour plots of the velocity distribution over the radiator core.

Introduction
Heat exchangers have been extensively utilised in many engineering applications for years, particularly in the automotive industry. Automotive radiators are the subject of the present study and are designed to transfer waste heat from the engine to the surrounding air via cooling liquid. Radiators are typically air-cooled cross-flow type heat exchangers, as the air passes perpendicular to the coolant through the heat exchangers. The airflow is usually induced by the moving vehicle (ram air) and/or the cooling fan(s) (fan air). In order to enhance the heat dissipation rate, compact heat exchangers with multi-louvered fins are commonly used. The use of the louvered fins can boost the heat transfer rate by a factor of 2 or 3 compared with equivalent unlouvered surfaces, as the louvers act to interrupt the airflow and provide a series of flat-plate leading edges causing new boundary layers. Consequently, a lower thermal resistance and a higher heat transfer coefficient can be achieved. The flow structure in the louvered fins has been investigated in detail by several researchers, such as Wong and Smith [1], Kajino and Hiramatsu [2], Davenport [3], and Webb and Trauger [4], etc. Figure 1 shows a typical heat exchanger surface with louvered fins used for radiators.

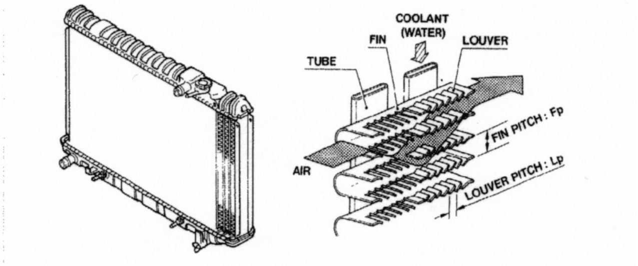

Figure 1 Louvered fin type heat exchanger [2]

As radiators are generally located under the bonnet, in front of the engine and behind the grille and the bumper, the airflow and temperature fields are complex. Nevertheless, knowledge of the airflow through radiators has become more important recently because of:

- more powerful engines producing more heat to be dissipated;
- compactness of the under-bonnet area;
- the trend for small cooling air intakes for styling improvements and drag reduction;
- possible reductions in the internal airflow drag;
- suggestion that airflow velocity distribution should be as uniform as possible to effectively use the total radiator area, or allow possible reduction in the sizing and costing of the radiators; and
- providing a better basis for the development and validation of CFD techniques.

There have been a number of experimental investigations devoted to the subject of quantifying the airflow velocity near radiators. The common industrial practice for flow distribution is to employ banks of propeller type anemometers. The rotation speed of each propeller gives an indication of the averaged velocity over a circular area, which is proportional to the average axial air speed. For instance, Williams [5] employed this type of method for measuring the total radiator flow. More applications can be found in SAE J2082 [6] which reported an overview of the different measuring techniques deployed in the automotive industry. Since the flow field is complex and typical propeller anemometers are relatively large (from 3 inches to 7 inches in diameter), this method has limited resolution and questionable accuracy for average flow rates.

There are some other techniques presented in the literature. Cogotti and Berneburg [7] used two-component fibre-link Laser-Doppler-Velocimetry (LDV) to collect a set of data on the velocity field existing in some critical areas inside an engine compartment. Berneburg and Cogotti [8] evaluated the engine compartment airflow by developing a "test radiator" which consists of a number of air vanes; and using improved LDV enabled measurement of the third component of the flow. The LDV method, although it is a state of the art and precise technique, involves high cost and is time consuming, making it unsuitable for typical engineering applications. Fujikake et al. [9] developed a thermocouple type air velocity sensor for the measurement of airflow rate through a radiator. Thermisters have been investigated for use as single point thermal anemometers for airflow measurement, however due to its sensitivity to turbulence intensity [10], this type of anemometer is not yet suitable for the measurement in unknown turbulence level environments, including in car engine bays. Conventional Pitot-static tubes are normally also not suitable due to the airflow complexity and relatively low airspeeds. On the other hand, research using CFD in this area has recently increased considerably, as it offers a clearer picture for an enclosed area, but it often requires experimental validation.

This study is aimed at developing a measuring technique that can eliminate the weakness of other techniques; can be utilised in variety of testing environments, including laboratories and field applications; and provides the industry with an alternative method. The specific objectives are:

- to develop a robust, cost-effective and non-intrusive technique for measuring local time-averaged airflow velocity through automotive radiators;
- to understand the flow behaviour near a radiator core;
- to identify the internal airflow pattern in a road-going vehicle; and
- to assess the new technique by testing with a real vehicle.

Problem Formulation

This research was initiated in order to develop a pressure-based system for determining the magnitude and the direction of the local airflow through the radiator core. The local airflow velocity is interpreted via the largest measurable pressure drop across the core.

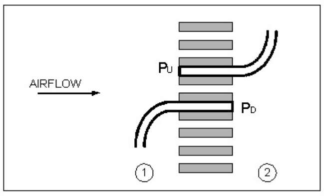

Figure 2 Radiator core model for pressure-drop analysis

It is assumed that the pressures p_U and p_D are measured on the streamlines that have similar characteristics, and p_U is approximately equal to the total pressure at the front face and p_D is close to the static pressure at the back of the core. Therefore, the modified Bernoulli's equation for incompressible flow along a streamline applies;

$$p_1 + \frac{1}{2}\rho V_1^2 + \rho g z_1 = p_2 + \frac{1}{2}\rho V_2^2 + \rho g z_2 + \Delta p_L \qquad (1)$$

Assuming, $z_1 = z_2$

$$p_U \approx p_1 + \frac{1}{2}\rho V_1^2$$

$$p_D \approx p_2$$

$$V_1 \approx V_2 = V_a$$

Therefore,

$$p_U - p_D \approx \frac{1}{2}\rho V_a^2 + \Delta p_L \qquad (2)$$

The pressure drop of the radiator (Δp_L) can be expressed as the summation of the pressure drop from entrance effects, the pressure drop from flow acceleration, the pressure drop from the core friction and the pressure recovery at the exit. Kays and London [11] gives a mathematical model for the flow stream pressure drop calculation for most heat exchanger cores. The expression can be simplified when isothermal pressure drop measurements have been made [3];

$$\Delta p_L = \frac{1}{2}\rho V^2\left(K_c + K_e + f\frac{A_t}{A_c}\right) \qquad (3)$$

where f is the core friction factor; K_c and K_e are entrance (contraction) and exit loss (expansion) coefficients respectively.

There is no simple theory for describing the pressure drop across the core (Δp_L), due to the unstreamlined shape of the louvered fins and the complexity of the airflow, except semi-empirical solutions and numerical simulations (not cited in this paper). Kays and London [11] presented comprehensive experimental data for compact heat exchangers. Davenport [3] performed a practical study of louvered fins and presented correlations of heat transfer and flow friction characteristics for 32 test fin cores. Achaichia and Cowell [12] presented data and correlations for louvered plate fin surfaces. It is noted from the studies that the friction factor (f), hence pressure drop (Δp_L), depends on the core geometry, Reynolds number based on louver pitch, and air properties.

In general, it is possible to express;

$$p_U - p_D = f(\text{Air Velocity}, \text{Core Geometry}, \text{Air Properties}) \qquad (4)$$

Therefore, for a specific radiator core and with knowledge of ambient conditions, the air velocity through a radiator core can be determined from measuring p_U and p_D. Reverse flow can be indicated if the value of (p_U - p_D) is negative.

Experimental Set-up

In order to verify the above formulation and establish a correlation for equation (4), a simple test facility was built enabling a simulation of the non-uniform airflow through sections of radiator cores. A schematic drawing of an open circuit closed test section wind tunnel with the test section of 300(w) x 350(h) x 965(l) mm^3 is shown in Figure 3. A section of radiator core filled the entire cross section. Removable block(s) in the middle of the test section were used for distorting the airflow, producing a nominally two-dimensional flow, as documented with a commercially available four-hole Cobra pressure probe (refer to [13] for details). The probe was used to map the flow around the radiator core, and provide values for the absolute local three-component time-average velocity. The probe used in this study featured a head size of just 1.3mm and each of the pressure taps in the probe head was 0.25mm diameter (see Figure 4). Pressures p_U and p_D were measured via a pair of hypodermic tubes (1.2mm O.D, 0.8mm I.D) connected to a Scanivalve system, which is a 48-channel pressure scanner.

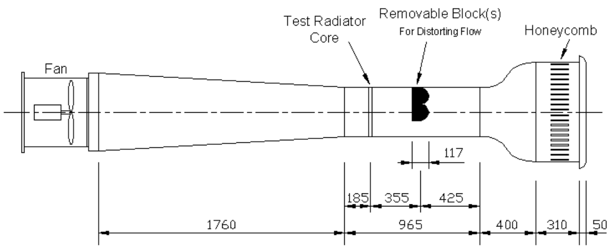

Figure 3 Layout of the test rig set-up

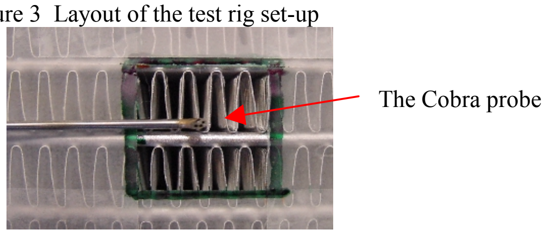

Figure 4 The Cobra Probe in front of a section of the test radiator core; the marked area is 15mm x 15mm

After introducing a correlation for a particular radiator core (the results will be discussed in the next section), it was intended to assess whether the technique was applicable to full-scale automobiles by testing with an Australian production passenger vehicle in the RMIT Aerodynamic Wind Tunnel. The tunnel is closed circuit with a working section of 3m x 2m and a test section length of 9m. Comparison was made with another independent measuring technique using a 16mm diameter vane type handheld anemometer.

Results and Discussion

Figure 5 shows the normalised velocities near the test radiator core for three nominal velocities in undisturbed flow (i.e. no flow distortion blocks upstream). Each data point is the average value of 25 measurements in a 12mm x 12mm square (the distance between radiator core tubes is about 12mm). The contour plots in Figure 6 were aimed at understanding the flow behaviour in a small area adjacent to the core. Measurements were taken by the Cobra probe mounted horizontally on a computerised three-axis traversing gear. Each plot in Figure 6 is generated by 121 measurements from a square of 15mm x 15mm (the marked area in Figure 4). The first three plots correspond to the upstream planes of 60mm, 20mm and 0mm respectively, while the bottom three plots refer to the flow patterns at the exit planes.

Referring to Figure 5, close to the core the averaged velocities appear to rise. It is not clear why the averaged velocities just upstream and downstream of the core face appear higher, but the changes are relatively small. It seems likely that the size of the Cobra probe relative to the size of the jets or the distance from the core face creates a blockage effect.

Referring to Figure 6, at the exit of the core the flow consists of an array of jets and wakes. The core velocities of the jets are approximately three times the mean flow, whereas no flow is present behind the coolant tubes, see Figure 6(d). (It is noticed that there is a "step", as the traverse was probably not set perfectly parallel to the core.) The flow is reasonably well mixed at a distance of 20mm downstream of the core, as can be seen in Figure 6(e). The local flow velocities would be expected to asymptote to approximately free stream velocity.

In order to distort the flow to more closely simulate the complex flow in a real vehicle, blocks were placed upstream of the test core (see Figure 3). In Figure 7, the (p_U - p_D) parameter on the y-axis is plotted against the normal component (U) of the local airflow measured at the front face, with the angle of flow indicated in the legend. Large flow angles are evident. Flow perpendicular to the radiator face is defined as zero degrees. The Cobra probe enables the three-components of the velocity associated with flow angles to be resolved within the range of ±45 degrees. The probe was rotated if the approach angle was larger than the calibrated range. It can be seen that the (p_U - p_D) parameter has a good relationship with the normal component of the airflow even at various approaches of flow direction (accuracy is typically within ± 5%), except for the measurements at flow angles larger than 80 degrees. Hence, measurements of the (p_U - p_D) can enable engineers to determine the airflow magnitude and its distribution over radiator cores accurately even if the flow approach angles are relatively large, as may occur in car engine bay.

The radiator sample was calibrated enabling the establishment of a correlation for (p_U - p_D) against U at several flow approach angles. It was also noted that the influence of the air temperature was minor and was not included for this study. The best way of correlating results was found to be power regression, that is,

$$p_U - p_D = a \times U^b \qquad (5)$$

where a and b are constants depend on the radiator geometry.

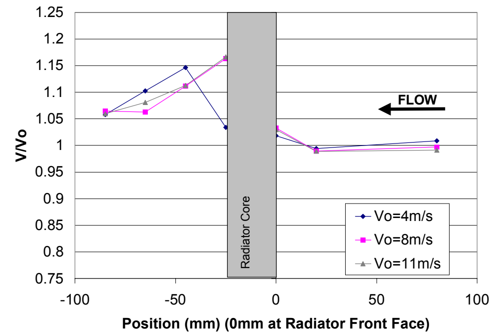

Figure 5 Normalised air velocities plotted along the wind tunnel (Flow - from right to left)

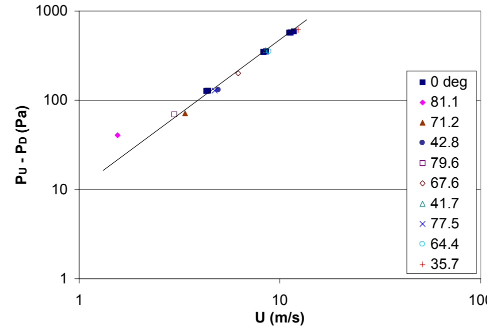

Figure 7 Plot of (p_U - p_D) verses U

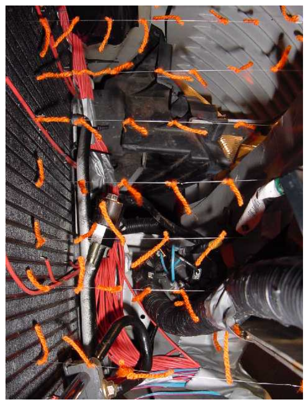

Figure 8 Flow visualisation between the grille and the heat exchangers

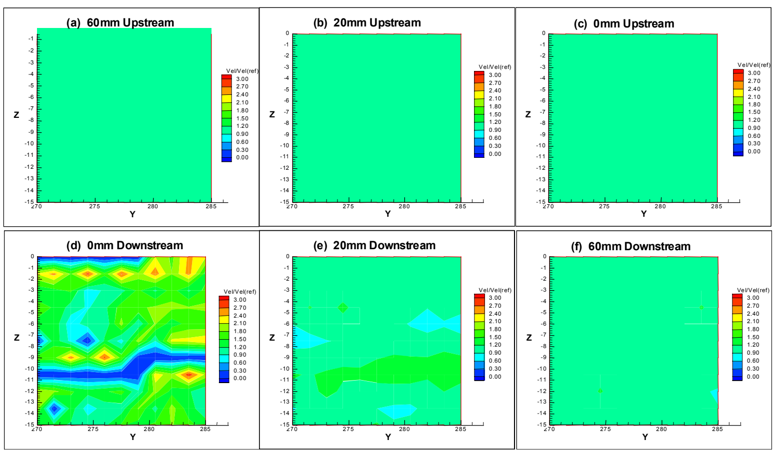

Figure 6 Flow mapping in a square of 15mm x 15mm at six vertical planes in an undisturbed flow
(a) 60mm upstream
(a) 20mm upstream
(a) at the front face
(a) at the back face
(a) 20mm downstream
(a) 60mm downstream

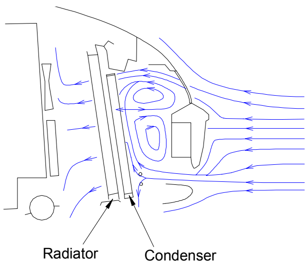

Figure 9 Sketch of the observed flow pattern

Testing with a complete vehicle in the RMIT aerodynamic wind tunnel was also carried out for flow visualisation and evaluation of the new technique. As the baseline vehicle for this study, there were two heat exchangers installed, i.e. a radiator and a condenser, and the airdam and the fan module were removed. Figure 8 is a photograph taken under the bonnet using wool tufts for flow visualisation. It was initially suspected from the figure that there was reverse flow existing in the heat exchanger cores in areas that were in the wake of the bumper bar. After using a smoke trace, it was proved that there was no reverse flow present in the cores, but in some locations the flow velocity was close to zero. The suggested flow pattern was sketched in Figure 9. It was shown that from the flow visualisation the flow was three-dimensional and there were two vortices in front of the heat exchangers due to the restriction of the cores and the wake of the bumper.

Typical results of the proposed technique and the small handheld vane anemometer are shown in Figure 10 for comparison. It was noticed that the similar-size condenser physically located ahead of the radiator acted as a flow straightener. Twenty-four pairs of pressure tubes were set up across the radiator in a 4 x 6 array (squares in Figure 10). The anemometer was held behind the radiator to measure the flow leaving the core. The accuracy of the anemometer was ± (0.1m/s + 1.5% of m.v.). The complete contours for the whole radiator were interpolated by the Kriging method using the commercial software Tecplot (see [14] for details). The results from the two methods are comparable and show higher flow in the top part and very low to zero velocity in the middle of the radiator indicating that the bumper and/or the engine block affect the airflow significantly. The dissimilarity in the top part of the contour plots may be due to limited resolution and a finer resolution of measurement points may be needed.

Conclusion and Future Work

A relatively simple and non-intrusive technique has been proposed and evaluated for use in measuring time-averaged velocity distributions through automotive radiator cores. The technique has been evaluated in a wind tunnel, is suitable for use on the road and complements other existing methods of air velocity measurement. Air velocity distribution through an automotive radiator is shown to be highly complex, three-dimensional and non-uniform, with areas of very low flow through the radiator evident in the wake of a bumper bar. Large flow angles exist close to the condenser front face due to a pair of contra-rotating vortices. The method was found to give satisfactory results even in complex turbulent flows. The next phase of the work is to complete an analytical study to investigate loss of cooling efficiency due to the radiator flow non-uniformity.

Acknowledgement

This study was generously supported by Ford Motor Company of Australia which provided the test vehicle. A scholarship from the Department of Mechanical and Manufacturing Engineering, RMIT is also acknowledged.

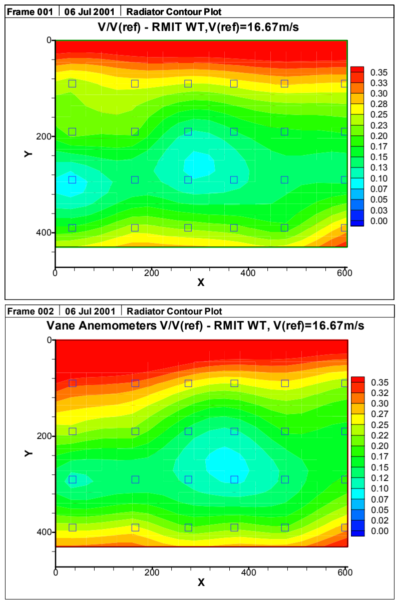

Figure 10 Example plots of the radiator velocity distribution at free stream velocity of 60km/h (a) by correlation for (p_U - p_D); (b) by small handheld anemometer

References

1. Wong, L.T., and Smith, M. C., *Airflow Phenomena in the Louvered-Fin Heat Exchanger*, Society of Automotive Engineers of America (SAE) Paper 730237
2. Kajino, M., and Hiramatsu, M., *Research and Development of Automotive Heat Exchangers*, in *Heat Transfer in High Technology and Power Engineering*, 1987, Hemisphere: Washington, D.C. p. 420-432.
3. Davenport, C.J., *Heat Transfer and Fluid Flow in Louvred Triangular Ducts*, *Department of Mechanical Engineering*. 1980, Coventry (Lanchester) Polytechnic.
4. Webb, R.L., and Trauger, P., *Flow Structure in the Louvered Fin Heat Exchanger Geometry.* Experimental Thermal and Fluid Science, 1991. **4**: p. 205-217.
5. Williams, J., *An Automotive Front-End Design Approach for Improved Aerodynamics and Cooling*, SAE Paper 850281
6. J2083, *Cooling Flow Measurement Techniques*, SAE Report
7. Cogotti, A., and Berneburg, H., *Engine Compartment Airflow Investigations Using a Laser-Doppler-Velocimeter*, SAE Paper 910308
8. Berneburg, H., and Cogotti, A., *Development and Use of LDV and Other Airflow Measurement Techniques as a Basis for the Improvement of Numerical Simulation of Engine Compartment Air Flows*, SAE Paper 930294
9. Fujikake, K., Katagiri, H., and Suzuki, Y., *Measurement of Air Velocity Distribution and Airflow Rate through Radiator in an Automobile*, SAE Paper 780587
10. Burton, D.M., Saunders, J., and Sheridan, J. *Effect of Free Stream Turbulence Level on Embedded Thermister Anemometers. Melbourne Graduate Fluids Conference*. 2001. Monash University, Melbourne, Australia.
11. Kays, W.M., and London, A. L., *Compact Heat Exchangers*. Third Edition (Reprint Edition with corrections) ed. 1998: Krieger Publishing Company.
12. Achaichia, A., and Cowell, T. A., *Heat Transfer and Pressure Drop Characteristics of Flat Tube and Louvered Plate Fin Surfaces.* Experimental Thermal and Fluid Science, 1988. **1**: p. 147-157.
13. Hooper, J.D., and Musgrove, A. R., *Reynolds Stress, Mean Velocity, and Dynamic Static Pressure Measurement by a Four-Hole Pressure Probe.* Experimental Thermal and Fluid Science, 1997. **15**: p. 375-383.
14. *Tecplot User's Manual (Version 7)*. 1996, Bellevue, Washington: Amtec Engineering Inc.