

A New Approach for the Simulation of Multi-Louver Fin Surfaces

Desmond Aubery¹, Pongtorn Charunyakorn² & Sompong Putivisutisak³

¹ *adTherm Technology,
Bangkok, Thailand
E-mail : des@adtherm.com*

² *Department of Mechanical Engineering, Faculty of Engineering,
Chulalongkorn University, Bangkok, Thailand
E-mail : pongton.c@chula.ac.th*

³ *Department of Mechanical Engineering, Faculty of Engineering,
Chulalongkorn University, Bangkok, Thailand
E-mail : fmespt@eng.chula.ac.th*

ABSTRACT

Heat-transfer and pressure drop prediction of corrugated multi-louver fins is of major interest to the automotive, refrigeration and computer cooling industries. Usually air-side performance characteristics are determined from experimental data and empirical equations derived from that data. In this paper, the CFD simulation of an automotive radiator is compared to experimental data. A 2D mesh is used in order to control simulation errors. New equations are introduced for calculation of the fin louver boundary temperature. The simulated heat-transfer and pressure drop performance is found to be in very close agreement with the experimental data.

Keywords

Conjugate heat-transfer, multi-louver fin

1. INTRODUCTION

The optimum design of automotive cooling devices e.g. radiators, condensers, evaporators, in the modern automobile, has become extremely critical.

Current automotive cooling devices typically use the Corrugated Multi-Louver Fin design. This fin design has been in production for at least thirty years, with very few new major technical advances in the optimization of the design geometry. The principle reasons for this are due to severe space constraints within the multi-louver geometry, making it virtually impossible to

insert measurement elements, without significant disruption of the gas flow paths. Thus, researchers and designers have had to resort to conducting geometry-specific macro-performance tests in wind-tunnels; and curve-fitting of the resulting data to empirical equations, for both fin heat-transfer coefficient and pressure drop. The resulting equations usually have large associated inaccuracy^{1, 2, 3, 4}.

There is an urgent need for modern computer simulation techniques to be developed which can provide both accurate and fast solutions to the automotive heat-transfer industry, to enable the complex flow paths within the multi-louver corrugated fin geometry to be understood, and to perform heat-transfer and pressure-drop performance prediction. The interplay between experiment, theory and simulation is considered to be vital to the development of these new, accurate, design and optimization tools for the automotive heat-transfer industry. This paper discusses exploratory research which has already been undertaken by the author in this area, and for future research areas to be explored.

2. PRESENT RESEARCH

A new, novel approach was developed for simulating the complex flow paths through the gas-side of an automotive corrugated multi-louver fin array. Throughout this research program, the critical aspects of accuracy and solution time were considered.

Fin geometries are complex to model in 3D, typically resulting in a high number of mesh cells. A large amount of inaccuracy can easily be introduced through inadequate mesh refinement in regions of abrupt geometry change e.g. louver-non-louver interface. In addition, 3D geometries are generally not simple to modify, without extensive re-modeling. Typically, the larger the number of cells – the longer the solution time required.

The objective of this research program was to fully understand and more accurately predict:

- Heat-transfer from the tube surface, lower fin surface and fin louver surface;
- The source and influence of various errors on the overall error of the simulation results.

2.1 Modeling techniques

The flow field around the centerline section of the louvered portion of the multi-louver fin was modeled in 2D, in order to minimize the effects of mesh shape, and to reduce the number of solution cells in the model. The flow regime is laminar relative to both the louver pitch and fin pitch.

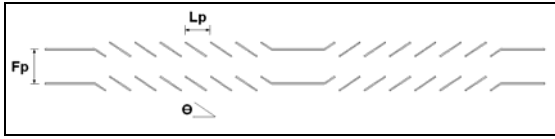


Figure 1 Radiator multi-louver fin geometry

For this conjugate heat-transfer problem, the heat-transfer up the fin was modeled using a set of new extended fin, two-step, heat-transfer equations, which relate the fin surface temperature to the cooling fluid temperature and average gas temperature of specified local fluid cells. These new equations were derived from the equations for an extended rectangular fin of uniform cross-section, published in the literature ^{5,6}.

The general form of the louver boundary temperature equation set is:

$$T_{b,i} = A_i * T_{g,i} + B_i * T_{c,i} \quad (1)$$

$$T_{s,i} = E_{f,i} * T_{b,i} + (1 - E_{f,i}) * T_{g,i} \quad (2)$$

The 2D fin louver boundary temperature, for each respective louver was calculated, and adjusted, directly from these equations, throughout the duration of the solver solution phase of the

simulation. Feedback was obtained from specified cell elements, throughout the solver solution phase, and used in the calculation of local heat-transfer coefficients. The respective heat-transfer contributions of each section of tube, lower fin and louver surfaces were then calculated and summed across the entire depth of the fin geometry. The overall heat transferred to the gas was then calculated directly from the inlet and outlet gas temperatures. This was then compared to the louver-based local heat-transfer calculations, as a cross-check on accuracy. Pressure drop was calculated using the results obtained from the CFD simulation (louver), and corrected by combining these with the drag contributions of both tube surface and lower fin section.

2.2 Benchmark data

The simulation results were benchmarked against data obtained from wind-tunnel tests performed on two different commercial automotive radiators ⁷.

2.3 Error control

In order to obtain extremely accurate results for the simulations, it is imperative that the physical system be well understood. The model was designed in such a way so as to minimize the sources of error at each and every step. Errors are caused by a number of factors: physical data approximations; theoretical approximations; model geometry; mesh shape-induced perturbations; CFD solver accuracy and spatial discretization scheme.

During the initial phases of the research program, it was observed that mesh shape can influence the final simulation result significantly. In some cases, non-physical fluid behavior was observed, especially towards the final stages of the solver solution phase. In many cases, this resulted in complete solver divergence.

2D meshes allow tighter control over the mesh shape in the simulated fluid domain and hence offer the potential for minimization of the mesh shape-induced errors. Thus, to minimize the mesh-induced errors, it was decided to use well-constrained 2D meshes, rather than 3D meshes.

2.4 Modeling flexibility

The approach taken in the research program of using algorithms to determine the average

temperature of the physical fin louver boundary, allowed a large amount of flexibility in introducing, testing and refining the new theory. It was a trivial task to modify a few lines of computer code and re-run the solver. This approach also lends itself to simple geometry modification, under the control of a governing optimization criterion.

3. PRESENT RESEARCH

3.1 Representative data point

A representative data point is listed in Table 1, below. Data is provided from both the physical wind-tunnel test results ⁷, and the output from the simulation.

Table 1 Comparative data for wind-tunnel and simulation model results

Data	Unit	Wind-tunnel	Model	Error (%)
$v_{g,f}$	m/s	7	7	---
$T_{\infty,i}$	°C	25	25	---
$T_{\infty,o}$	°C	---	78.61	---
q'_{louver}	kW	---	58.96	---
q'_{lfin}	kW	---	1.926	---
q'_{tube}	kW	---	1.291	---
q'_{rad}	kW	62.1	62.17	+0.12
ΔP_{rad}	Pa	199	198.7	-0.16

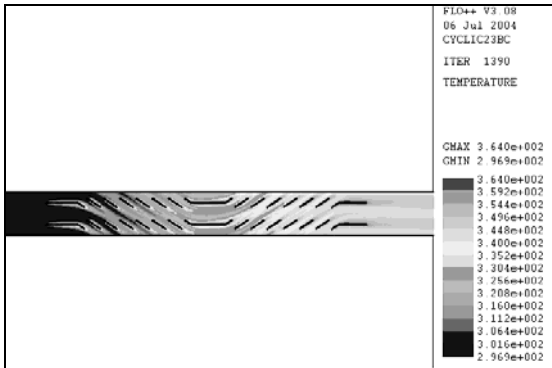


Figure 2 Gas temperature – simulation output

3.2 Temperature

Figure 2 shows the gas temperature simulation output. Figure 3 shows a plot of the gas temperature profile through the core depth, together with a curve-fit of this data.

The gas temperature along the centerline between fin rows is seen to be reasonably oscillatory. This is due to the varying flow regime throughout the core depth. The fin heat-transfer rate was calculated from the temperature at fin outlet, fin inlet, local specific heat and gas mass flow-rate.

The simulation output for temperature shows regions of poor flow, and poor heat-transfer. This allows simple design rules-of-thumb to be developed e.g. the fin louver pitch can be adjusted to align downstream louvers to the centerline between upstream louvers, to maximize fin heat-transfer.

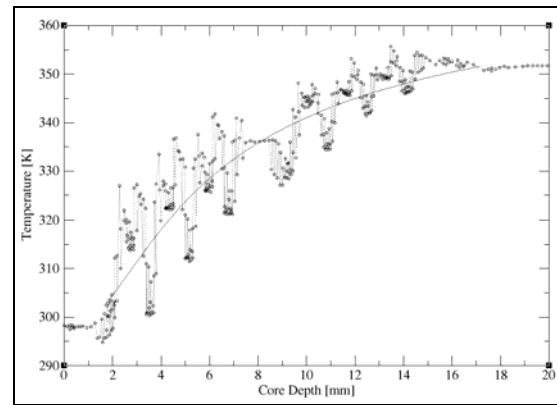


Figure 3 Gas temperature – fin row centerline

3.3 Pressure

Figure 4 shows the gas pressure simulation output. Figure 5 shows a plot of the gas pressure profile through the core depth.

The gas pressure along the centerline between fin rows is slightly oscillatory. This is due to the varying flow regime throughout the core depth. Of interest is the section of pressure recovery at the centre of the core. This relates to the forced deflection of the gas flow in this section. The pressure drop across the fins was calculated from the difference in pressure values between fin louver inlet and outlet, and corrected to account for the influence of the exposed tube and lower fin surfaces.

3.4 General comments

The results of the simulations were found to be in very close agreement with the available benchmark data. Typical errors were in the order of 2 – 3% for

heat-transfer, and 2 - 5% for pressure drop. These results were considered to be well within the measurement error of the physical wind-tunnel test equipment, used for the benchmark data.

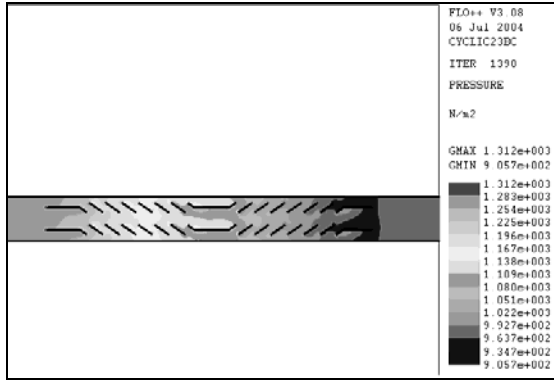


Figure 4 Gas pressure – simulation output

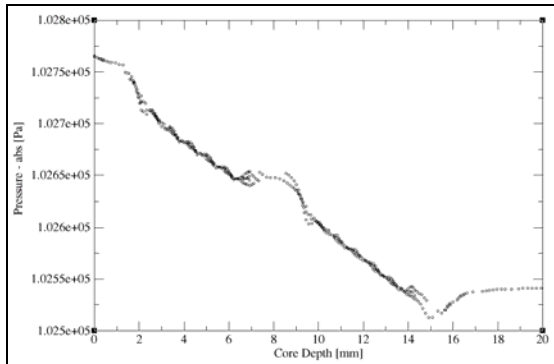


Figure 5 Gas pressure – fin row centerline

4. FUTURE RESEARCH

A three-step equation set which splits the fin into lower & louver sections is in the final stages of testing. Pressure drop calculations are under further development and refinement. A research project is planned with Chulalongkorn University, by the authors, to investigate “Geometric shape modification of an automotive radiator corrugated multi-louver fin, subject to a governing optimization criterion”.

5. CONCLUDING REMARKS

This new approach allows fast, accurate simulation of multi-louver fin surfaces. It is believed that this will eventually be more fully developed into a full range of design and optimization software tools,

which will have significant potential for the automotive heat-transfer, refrigeration and computer-cooling industries.

6. NOMENCLATURE

L_p	Louver pitch.
F_p	Fin centerline pitch.
θ	Louver angle.
$T_{b,i}$	Fin base temperature.
$T_{s,i}$	Louver surface average temperature.
$T_{c,i}$	Cooling fluid temperature.
$T_{g,i}$	Local gas temperature.
A_i	T_g coefficient of louver i .
B_i	T_c coefficient of louver i .
$E_{f,i}$	Fin efficiency of louver i .
v_{gf}	Gas inlet face velocity.
$T_{\infty,i}$	Gas inlet temperature.
$T_{\infty,o}$	Gas outlet temperature.
q'_{rad}	Radiator heat-transfer.
ΔP_{rad}	Radiator pressure drop.
q'_{louver}	Louver heat-transfer.
q'_{lfin}	Lower fin heat-transfer.
q'_{tube}	Exposed tube heat-transfer.

7. REFERENCES

1. Chang, J.Y.; Hsu, K.; Lin, Y. and Wang, C.C. (2000). *A generalized friction factor correlation for louver fin geometry*. Int. J. Heat Mass Transfer. 43 (2000): 2237-2243.
2. Yun, J.Y. and Lee, K.S. (1999). *Investigation of heat transfer characteristics on various kinds of fin-and-tube heat exchangers with interrupted surfaces*. Int. J. Heat Mass Transfer. 42 (1999): 2375-2385.
3. Chang, J.Y. and Wang, C.C (1997). *A generalized heat transfer correlation for louver fin geometry*. Int. J. Heat Mass Transfer. 40 (3): 533-544.
4. Kim, M.H. and Clark, W.B. (2002). *Air-side thermal hydraulic performance of multi-louvered fin aluminium heat exchangers*. Int. J. Refrigeration. 25 (2002): 390-400.
5. Incropera, F.P and Dewitt, D.P. (2002). *Fundamentals of Heat and Mass Transfer*, 5th ed., John Wiley and Sons.
6. Özişik, M.N. (1985). *Heat Transfer – A Basic Approach*, McGraw-Hill Book Company.
7. Aubery, D. (2001 – 2004). *Internal communications and project data*, adTherm Technology.