

Engine Cooling Module Sizing Using Combined 1-Dimensional and CFD Modeling Tools

Timothy C. Scott
University of Virginia

Dhananjay S. Joshi
Chrysler LLC

Copyright © 2009 SAE International

ABSTRACT

Engine cooling module air flows depend on package components and vehicle front end geometry. For years, in the early stages of vehicle development, front end geometry air flows were determined from 3/8 scale models or retrofit of similar existing vehicles. As time to market has become much shorter, finite element modeling of air flows is the only tool available. This paper describes how finite element simulations of front end air flows can be run early in the development program independent of any specific engine cooling module configuration and then coupled with traditional one-dimensional component performance models to predict cooling module air flows. The CFD simulation thus replaces the previous scale model testing process. The CFD simulations are used to determine the two parameters that characterize the front end geometry flow resistance (recovery coefficient and internal loss coefficient). Because these two parameters are independent of any specific set of cooling module components, the number of time consuming CFD simulations is minimized. From this the cooling module heat transfer can be predicted and component selection may proceed.

INTRODUCTION

Discussion of the fundamental procedure for front end air flow and cooling package performance sizing were most recently provided by Kurland [1], Tenkel [2], Emmenthal [3], and Mosier [4] to name a few. But the method has origins going back to the early part of the 20th century.

These procedures have been followed by designers for over fifty years. Over these years they have benefited from computer assistance but mainly in the carrying out of calculations previously done by graphical or manual methods.

The procedure, though 1-dimensional in nature, has proven to be quite robust and accurate. This is largely due to the collection of historical data, and incorporation of empirical interaction parameters and "installed" vs. wind tunnel corrections. For those who have maintained and updated the component database, and trained new engineers in the details of the procedure, 1-dimensional methods are generally sufficient.

The recent trend to dedicated software packages and more CFD modeling indicates to us that some unnecessary complications and costs are being introduced. Division of heat exchangers into discrete

parts and running complete CFD analyses of heat exchanger and fan performance are expensive and time consuming. These are not necessary if one has maintained the database, training, and procedures of the basic 1-dimensional process.

We proceed, therefore, to lay out the basic 1-dimensional process with some techniques and models that make it easy to execute with a simple spreadsheet. Then we show how CFD can be used sparingly to simply replace the front end geometry test process that is no longer possible because of the shorter time to market. Finally, we will discuss the actual vs. perceived value of recent complex software packages designed to carry out this cooling model performance prediction.

THE BASIC 1-DIMENSIONAL METHOD

The fundamental task is to create a graph of pressure drop vs. flow rate for the cooling package components (vehicle configuration, grille, heat exchangers). We then plot this on the pressure rise vs. flow graph for the fan. The intersection of the two curves is the operating point of the system. Generally the analysis is no longer done graphically. Yet the graph is helpful to display the process.

The actual solution process is often carried out by a software package using the component models directly coupled to an equation solver routine of some sort.

We show that this complexity is not required. A simple spreadsheet is quite adequate if the component models are organized as explained below

For this discussion, the fan performance is assumed known. One may have actual fan data or a representative math model. Techniques for modeling electric and mechanical drive fans have been discussed by Scott [5,6]

The basic front end configuration is set down in Figure (1). Air approaches the vehicle at the vehicle speed, V_∞ . There is then a static pressure recovery during deceleration before entering the front end geometry.

There follows a pressure drop through the grille and front end flow passages. The air arrives at the face of the first plane of heat exchangers (generally a condenser or condenser/oil cooler combination) at an average "face" velocity V_1 and static pressure p_1 as shown

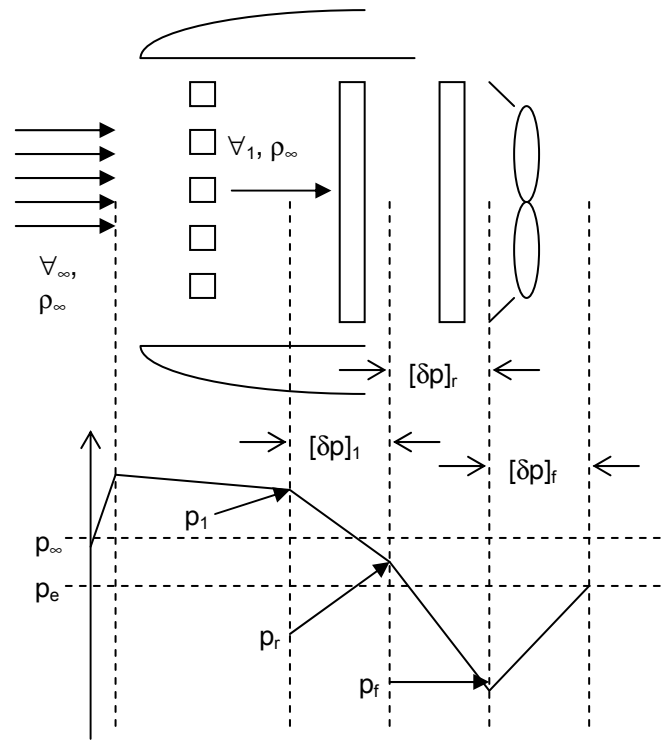


FIG. 1: General Nature of Velocity and Pressure Through the Cooling Package

There is then a pressure drop, $[\delta p]_1$ across the first heat exchanger set and a second $[\delta p]_r$ across the radiator. There may be other expansion/contraction pressure drops as well. We assume these are accounted for in $[\delta p]_1$ and $[\delta p]_r$.

The fan will then increase the pressure to that of the engine compartment, p_e , which is generally very close to ambient. The flow then proceeds out of the engine compartment and eventually back to p_∞ .

This description embodies most of the elements of the problem and is sufficient for explaining the basic analysis procedure.

The static pressure at the inlet to the heat exchangers, p_1 , is related to the vehicle speed, V_∞ , and ambient pressure, p_∞ by:

$$p_1 = p_\infty + C_f \rho_\infty \frac{V_\infty^2}{2} - K_g \rho_\infty \frac{V_1^2}{2} \quad (1)$$

where C_f is the pressure coefficient for the front end and K_g is a grille/front end loss coefficient. We write the loss coefficient in relation to the velocity in the passage approaching the heat exchanger package, V_1 , which is the average face velocity for the first heat exchanger plane.

The pressure in the engine compartment is related to the vehicle speed and ambient pressure by:

$$p_e = p_\infty + C_e \rho_\infty \frac{V_\infty^2}{2} \quad (2)$$

where C_e is the pressure coefficient for the external air flow around the engine compartment (largely the underbody air flow) and is a small negative value.

Thus vehicle geometry effects are characterized by the three parameters C_f , C_e , and K_g .

By subtracting equation (2) from equation (1) we have:

$$p_1 - p_e = [C_f - C_e] \rho_\infty \frac{V_\infty^2}{2} - K_g \rho_\infty \frac{V_1^2}{2} \quad (3)$$

If measured values of $p_1 - p_e$ are available at different values of V_∞ and V_1 we may determine the values of $[C_f - C_e]$ and K_g .

We now arrange equation (3) in a form such as:

$$2 \frac{p_1 - p_e}{\rho_\infty V_\infty^2} = [C_f - C_e] - K_g \left[\frac{V_1}{V_\infty} \right]^2 \quad (4)$$

so that a graph of $2 \frac{p_1 - p_e}{\rho_\infty V_\infty^2}$ vs. $\left[\frac{V_1}{V_\infty} \right]^2$ reveals $[C_f - C_e]$ and K_g

Note that the coefficients, $[C_f - C_e]$ and K_g are treated as being independent of the specific heat exchanger package installed. Indeed, the validity of this model, and the minimum influence of cooling package components on front end effects was confirmed recently by Oler[7]

Further verification of the validity of this approach from actual vehicle data from Schaub[8] is shown by Oler. and this technique is also explained by Moffat [9].

Historically, the coefficients in this model have been determined from wind tunnel or scale model tests. The simplest technique is to draw air through a scale model with an auxiliary fan as illustrated in Figure (2). The auxiliary fan has sufficiently more power than the actual radiator fan and the porous media is a series of perforated or expanded metal plates. With this auxiliary equipment, one can vary V_∞ and V_1 independently over a wide range.

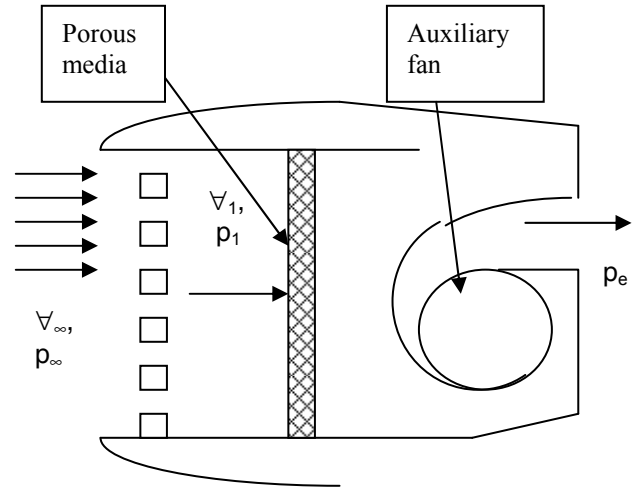


FIG. 2: Auxiliary Fan Method for Front End Coefficients

There were several other ways of dealing with the effects of front end geometry. For example, when vehicle body styles changed annually, a fiberglass mockup of the future geometry would be made early in the development program. This was then fit to the front of a current vehicle for which the air flow vs. vehicle and fan speed were known. The changes in radiator inlet coolant or "top tank" temperature then indicated the air flow advantage or penalty of the new configuration. This could be translated into the new $[C_f - C_e]$, K_g values

To determine the system operating point (air flow rate), the "standard" method for decades was to find the intersection between fan performance and exchanger/vehicle pressure change vs. air flow rates graphically. In the 1970s, this process was computerized by a number of people. The computer program allowed one to quickly investigate a large number of different cooling package components, thus shortening the design time.

The mathematical relations supporting this method are as follows:

Between the face of the heat exchanger package and the engine compartment, the pressure decreases across the two layers of exchangers and increases across the fan. Thus in addition to equation (3) we have:

$$p_e = p_1 - [\delta p]_l - [\delta p]_r + [\delta p]_f \quad (5)$$

For the package/vehicle combination, we put (5) into (3) and solve for:

$$[\delta p]_f = K_g \rho_\infty \frac{V_1^2}{2} + [\delta p]_l + [\delta p]_r - [C_f - C_e] \rho_\infty \frac{V_\infty^2}{2} \quad (6)$$

The air flow rate is then the point where the fan pressure rise from equation (6) crosses the fan performance pressure rise vs. flow rate curve.

To properly solve equation (6) iteratively with the fan curve, we need to properly account for the air temperature change through the exchangers. Thus the fan curve needs to be a plot of $[\delta p]_f/\rho_f$ vs. fan volume flow rate. By dividing (6) by the radiator outlet air density (which is the fan inlet air density) we have:

$$\frac{[\delta p]_f}{\rho_f} = K_g \frac{\rho_\infty}{\rho_f} \frac{V_f^2}{2} + \frac{[\delta p]_l}{\rho_f} + \frac{[\delta p]_r}{\rho_f} + \frac{[\delta p]_{misc}}{\rho_f} - [C_f - C_e] \frac{\rho_\infty}{\rho_f} \frac{V_\infty^2}{2} \quad (7)$$

The solution process can be simplified by noting that equation (7) can be replaced with a simple quadratic in the fan volume flow:

$$\frac{[\delta p]_f}{\rho_f} = C_0 + C_1 \dot{V}_f + C_2 \dot{V}_f^2 \quad (8)$$

without significant error.

CFD EXAMINATION OF GEOMETRY FACTORS

The experimental technique of Figure (2) was commonly used with a 3/8 scale model early in the development of a new vehicle. When such facilities are not available, or when development time is too short to allow for experimentation, one must work directly with the surface geometry in digital form. A computer simulation of the air flows can then be used to accomplish the same task. Indeed, there are a number of examples of this technique and many other studies that show the validity of CFD front end geometry effects modeling

Williams and Vemaganti [10] have performed a detailed study to understand the modeling effort and accuracy needed for front end/engine cooling flow modeling. They examined 23 front end geometry configurations at four different vehicle speeds. The effort put into testing all these configurations experimentally was great and obviously cannot be done for each new vehicle. Yet such studies are necessary to establish a robust modeling procedure.

Statistical methods (Spearman Rank Correlation Coefficient) were used to understand the reliability of the association between experimental and computed values. Aided with results from CFD studies they found that experimental results cannot capture the correct nature of radiator flow in all situations. One such case showed reverse flow through the radiator because of the low pressure region generated by the location of the air dam

with respect to a radiator cross member and lower fascia opening.

Some other aspects such as grid dependency were not addressed in their study.

Moffat [9] combined 1-D and 3-D models to come up with a strategy to model transient behavior of the engine cooling circuit in an efficient manner. Since any testing involves some kind of driving cycle (e.g. FTP or NEFC), it was pointed out that by conducting 3-D CFD calculations for some steady state conditions and combining the results with a 1-D network model for the engine coolant loop, one can get a good idea about engine warm up characteristics. This helps in estimating cabin heater temperature in addition to top tank temperature for the radiator. The differences between experimental and simulation results was mainly attributed to the limitation of modeling the transient behavior of heat rejection with respect to throttle position. The main idea is replacing the bagging methods (i.e. Fig. 2) with a CFD technique to come up with estimation of ram air effect, loss coefficient for heat exchangers and front end opening.

Uhl, et.al. [11] employed a combination of an in-house tool BISS and commercial package (StarCD). They coupled the two simulation tools to reduce the effort in mapping the CFD solution to 1-D code. The tools were linked in a way as to keep the three dimensional nature of flow and account for non-uniformity in the radiator face velocity. Very good agreement was found between testing and simulation results for heat transfer through different heat exchangers.

They have reported a case where a horn sitting right behind the grille showed 7% change in the flow rate, which indicates the need to include as many geometry details as possible in simulations. In general they found better agreement between experimental and simulation results at lower vehicle speeds, while at speeds in the range of 160 to 200 km/h, the deviation was around 13%.

Jerhamre and Jönson [12] have reported a study based on coupling CFD code (Fluent) with additional implementation for heat exchanger modeling via a user defined function (UDF). They have explained the care one must take when using the data for heat exchanger and fan. For heat exchanger performance, data at different air flow and coolant rates was used while taking care that the UDF formulation was based on adiabatic condition. Otherwise one might count the effect of heating the air through the heat exchanger twice. For a stack of heat exchangers (which is a common configuration), addition of individual pressure drops for each core was shown to be different than the pressure drop for the whole package. For fan modeling they argue correctly that the fan curve should be adjusted for shroud losses and then further corrected for temperature change, since the fan testing is typically done at ambient

temperature. We include this in our discussion here by using the inlet fan air density, (Figure (8)).

Kim and Kim [13] report a study of fan location and other effects on air flow. By looking at various items that affect radiator performance, such as fan power, placement, and holes in the fan shroud, they have shown which design changes can provide better results for a given vehicle configuration. This is an excellent use for this computational tool.

All these studies illustrate that the current state of the art of CFD analysis is capable of dealing with the vehicle/front end part of the system analysis. Many of the effects of front end geometry changes can and should be dealt with using this tool. Indeed, as time to market shortens and experimentation becomes impractical, CFD modeling needed.

With a CFD air flow model, one may vary the air flow through the cooling package at constant vehicle velocity by using a different porous media coefficients for the heat exchangers or by varying the fan speed. The

resulting plot of $2 \frac{p_1 - p_e}{\rho \nabla_\infty^2}$ vs. $\left[\frac{\nabla_1}{\nabla_\infty} \right]^2$ is quite close to a straight line as shown in Figure (3)

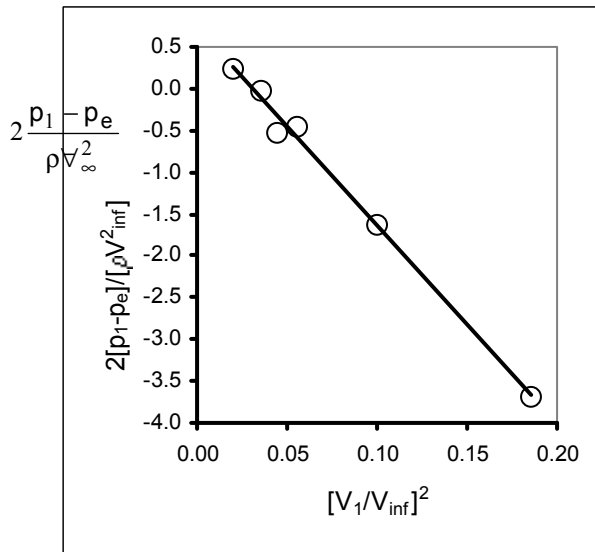


FIG. 3: Typical Results, CFD Air Flow Study

EXCHANGER AIR SIDE PRESSURE DROP MODEL

The air side pressure drop for the heat exchangers is modeled following the presentation by Scott [14]:

$$\frac{[\delta p]_a}{\rho} = \left\{ \left[\frac{T_o}{T_i} \right]^2 - 1 \right\} \frac{\nabla_f^2}{2} + \left\{ C_{1a} \frac{\mu}{\rho_i} \nabla_f + C_{2a} \nabla_f^2 \right\} \left[1 + \frac{T_o}{T_i} \right]^2 \quad (9)$$

The equation is developed in Appendix A. Like the heat transfer model given in the next section, the coefficients for the model, C_{1a} and C_{2a} can be determined directly from pressure drop vs. face velocity data and are exchanger size independent.

In general, the pressure drop with heat transfer can be 5–10 % higher than that without heat transfer. This is due to the additional work required to accelerate the air flow. Since the exiting air is hotter than the entering air, its velocity must be higher in order to maintain the same mass flow rate. Work is required to increase the kinetic energy of the air. This manifests itself in an increased pressure drop. The ratio of the absolute temperatures in equation (9) accounts for this.

EXCHANGER HEAT TRANSFER MODEL

Scott[14] also describes a modeling method for single and 2-phase compact exchangers that does not require detailed geometric information and whose coefficients can be determined directly from heat transfer tests or supplier performance curves. This model is developed in detail in Appendix B.

The coefficients for this model may be determined from exchanger heat transfer vs. face velocity performance curves. One does not need to have details of the exchanger dimensions (coolant passage dimensions, fin thickness, etc.)

FRONT END PACKAGE ANALYSIS PROCEDURE

We now consider the general process for the generation of equation (8) by the analysis of equation (7) from the models of the vehicle and the exchanger package. In this example, we consider the first row to consist of a condenser/oil cooler combination or "combo" cooler as shown in Figure (4).

The condenser face area and air passage configuration are different from those of the oil cooler so that the air side pressure drop will be different for each. This "combo" is in series with the radiator.

The process for determining the pressure drop across the two layers of exchangers as a function of flow rate with heat transfer present is described. There are, of course, several ways one may approach this problem. One is to simply run heat exchanger and fan models in conjunction with a general non-linear equation solver. We present a technique that may easily be implemented by a simple spreadsheet.

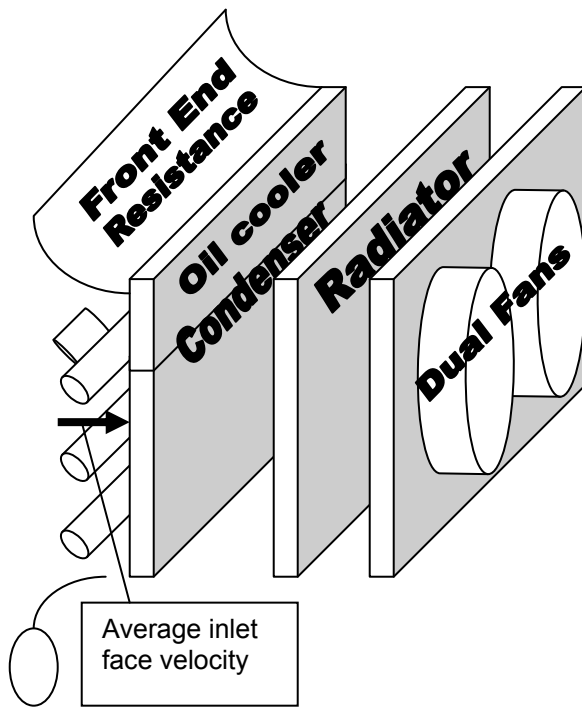


FIG. 4: Example Cooling Package Configuration

We consider a typical situation where the liquid flow rates and inlet temperatures to the oil cooler and radiator are fixed and the condenser operating point is set (condensing temperature, inlet superheat and subcooling). One would, for example choose some specific case for which the air flow is desired to determine if there is sufficient flow for the needed heat transfer. Thus, for example, one knows how much heat needs to be rejected by the radiator at this condition.

With tube side conditions specified, we may generate the air side pressure drop vs. face velocity information for each of the exchangers in advance and fit this with a simple quadratic of the form:

$$[\delta p]_a = C_1 V_f + C_2 V_f^2 \quad (10)$$

For example, Figure (5) shows the air side pressure drop vs. face velocity for a condenser with 30°C inlet air temperature, 70°C condensing temperature with 30°C superheat and 5°C subcooling. The data is generated by using the heat transfer model of Appendix B to determine the heat transfer for each face velocity. Then equation (9) is used to find the pressure drop.

The calculated values are then fit with equation (10) resulting in:

$$[\delta p]_a = 16.36 V_f + 2.635 V_f^2 \quad (11)$$

which matches the calculated values with an error of less than 1%

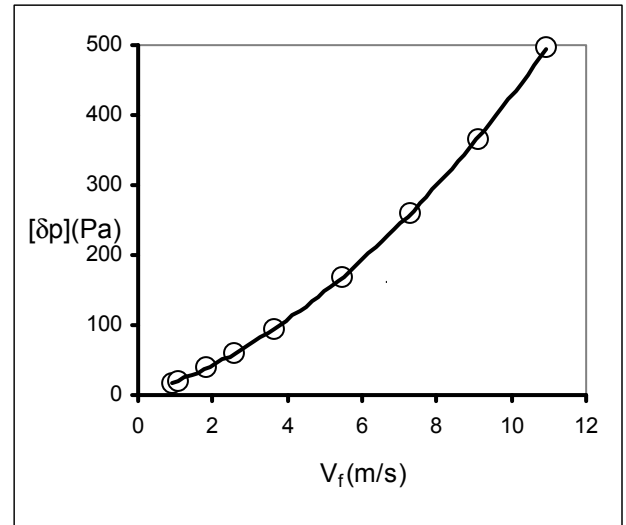


FIG. 5: Condenser Air Side Pressure Drop for Fixed Refrigerant Side Conditions

There are two limiting cases for air flow in the configuration of Figure (4). The first, illustrated in Figure (6), is where the combo cooler and radiator are spaced far enough apart so that the pressure in the space between them is constant. The second, illustrated in Figure (7) is where the combo cooler and radiator are very close so that the air pressure at the interface will be different between the oil cooler/radiator path than between the condenser/radiator path.

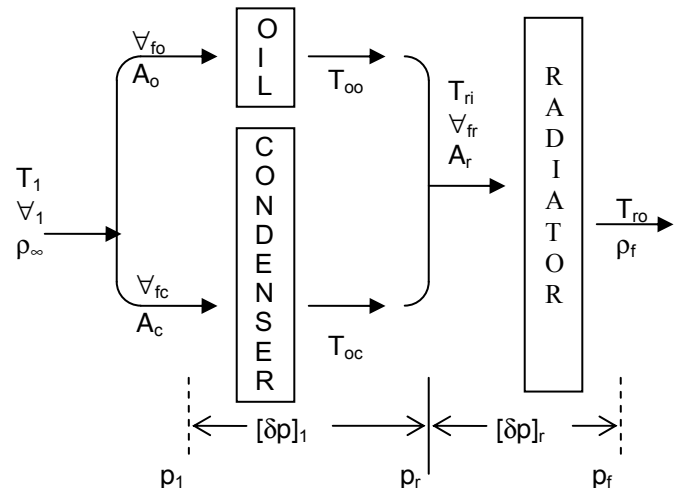


FIG. 6: Mixed Flow Model of the Heat Exchanger Package

In both cases, the face velocity for the oil cooler may be different from that for the condenser because the air side configuration (tube spacing, fin spacing, etc.) and heat transfer are different.