

A New Automotive Air Conditioning System Simulation Tool Developed in MATLAB/Simulink

Tibor Kiss and Lawrence Chaney
National Renewable Energy Laboratory

John Meyer
Visteon Corporation

ABSTRACT

Accurate evaluation of vehicles' transient total power requirement helps achieving further improvements in vehicle fuel efficiency. When operated, the air-conditioning (A/C) system is the largest auxiliary load on a vehicle, therefore accurate evaluation of the load it places on the vehicle's engine and/or energy storage system is especially important. Vehicle simulation models, such as "Autonomie", have been used by OEMs to evaluate vehicles' energy performance. However, the load from the A/C system on the engine or on the energy storage system has not always been modeled in sufficient detail. A transient A/C simulation tool incorporated into vehicle simulation models would also provide a tool for developing more efficient A/C systems through a thorough consideration of the transient A/C system performance. The dynamic system simulation software MATLAB/Simulink® is frequently used by vehicle controls engineers to develop new and more efficient vehicle energy system controls. A MATLAB/Simulink-based transient A/C system simulation model is easier to incorporate into MATLAB/Simulink-based vehicle simulation software; therefore, the availability of a transient A/C system simulation tool developed in the MATLAB/Simulink platform is important.

NREL has recently developed an A/C simulation tool to address these needs. This paper describes in detail the modeling methods used for this new simulation tool. Comparison with measured data is provided to demonstrate the validity of the model. The agreement between simulation and measurement was shown to be good on both the component and system level. The capabilities of the model are also demonstrated by the example of simulating the SC03 cycle.

CITATION: Kiss, T., Chaney, L. and Meyer, J., "A New Automotive Air Conditioning System Simulation Tool Developed in MATLAB/Simulink," *SAE Int. J. Passeng. Cars - Mech. Syst.* 6(2):2013, doi:10.4271/2013-01-0850.

INTRODUCTION

When operated, the air-conditioning (A/C) system is the largest auxiliary load on a vehicle. A/C loads account for more than 5% of the fuel used annually for light-duty vehicles in the United States [1]. A/C loads can significantly impact electric vehicle (EV), plug-in hybrid electric vehicle (PHEV), and hybrid electric vehicle (HEV) performance. Mitsubishi reports that the range of the i-MiEV can be reduced by as much as 50% on the Japan 10-15 cycle when the A/C is operating [2]. The advanced powertrain research facility at Argonne National Laboratory has reported a nearly 20% reduction in range in the Nissan Leaf operating on the UDDS cycle. A hybrid vehicle tested at NREL showed 22% lower fuel economy with the A/C on [3]. Increased cooling demands from the battery thermal management system in an EV may impact the A/C system. Heavy-duty vehicles also need a tool to evaluate the impact of A/C on both "down-the-

road" and idle conditions. Cabin climate conditioning is one of the primary reasons for operating the main engine in a long-haul truck during driver rest periods. In the United States, long-haul trucks (trucks that travel more than 500 miles per day) use 838 million gallons of fuel annually for rest period idling [4]. A flexible open-source analysis tool is needed to assess the A/C system's impact on advanced vehicles. The industry has expressed a need for both a standalone A/C system model as well as an A/C model that can co-simulate with a vehicle simulator such as Autonomie [5]. This model expands the capability of vehicle simulation tools, including Autonomie, and addresses industry needs.

The A/C system contains complex flow, thermodynamics, and heat transfer. On the refrigerant side, the flow is transient and both compressible and two-phase. Calculating refrigerant properties near the phase transitions can also be computationally difficult. Air flow through the condenser can vary widely depending on vehicle speed and condenser fan



Documents

Document Sharing Hub

speed. The heat transfer in the heat exchangers is a very complex process mainly addressed by complex correlations based on measured data. These correlations sometimes reduce the stability of the code. The effects of humidity are difficult to account for, not only in the model but also for the experimental data necessary to calibrate the code. A cabin model is also needed to provide a realistic load on the evaporator. The cabin model must consider all the major pathways of heat transfer into the cabin, including solar and convective loads from the environment, heat from the engine compartment, and sensible and latent heat loads in the air stream. Realistic control methods similar to ones actually used in automotive A/C systems also have to be implemented. The cycling of the compressor can introduce quick transients that are sometimes difficult to handle by the numerical solver. The simulation model also has to be fast enough for the purpose of evaluating vehicle power performance and/or the design of A/C systems and their controls.

There are numerous challenges in developing a suitable automotive A/C system simulation tool. Some examples of previously developed non-commercial full system-level simulation include [6,7,8]. However, to the best of our knowledge, none of them is both developed in the widely used dynamic system simulation platform MATLAB/Simulink (which is helpful for controls engineers) and available to the public at the same time. Commercially available software tends to be very expensive, and typically, their accuracy for predicting the transient processes is difficult to assess based on available documentation. The simulation model subject of this paper addresses these issues. Also, with the model being built on the MATLAB/Simulink platform, most of the inner workings of the model are visible to the user, and the user has considerable freedom to modify the model to better represent their specific A/C system hardware and controls. Only the source codes for some of the basic building blocks of the model remain inaccessible to the user.

SELECTING THE MODELING METHOD

Numerically, the most difficult part is modeling the two-phase refrigerant flow circuit. A number of different methods have been used in the past, and the selection of which method to use was based on the intended purpose of the simulation model.

With the transient nature of automotive A/C systems, the authors thought it was most important that the modeling method capture the transient processes accurately and robustly, even if model execution speed was compromised. The model should be robust for fast transients such as compressor cycling. It should also accurately predict refrigerant redistribution after shut-down and after start-up. It has to conserve mass accurately during simulations of long test cycles.

Another desirable feature of the model was that it should be versatile enough that various complex heat transfer correlations can be programmed into the main algorithm relatively easily. Finally, the model formulation had to be one that was well suited for the simulation modeling platform, MATLAB/Simulink.

The authors decided to use the finite volume formulation for calculating the refrigerant flow. The finite volume formulation is well suited for accurate conservation of mass momentum and energy. Also, in the finite volume formulation, immediately after each step of the integrator, the state variables and the velocity are readily available. Then, heat transfer rates even with the more complex methods can be calculated explicitly. Finally, Simulink is geared toward solving ordinary differential equation systems and, in the finite volume formulation, the problem is established as a set of ordinary differential equations. Therefore, this formulation is very well suited for Simulink.

Challenges were expected with the “stiff” nature of this modeling method, especially in the condenser where pure liquid is present.

DESCRIPTION OF THE MODEL

The transient A/C simulation model consists of two main sub-models that include the cooling circuit model and the cabin model. In the cooling circuit model, the larger volumes containing refrigerant, such as the accumulator, the receiver/dryer, and the headers of the heat exchangers, are modeled with the zero-dimensional volume simulation block (0-D volume block). The refrigerant pipes and the flat tubes / plates of the heat exchangers are modeled with various versions of the one-dimensional pipe simulation block (1-D pipe block). The general structure of the model on the refrigerant side is a network of 1-D pipe blocks connected to each other with 0-D volume blocks.

Using the 0-D volume and the 1-D pipe blocks, models of various multi-pass multi-row compact heat exchangers can be created relatively easily. Models for other A/C system components, such as the compressor and the thermostatic expansion valve, were incorporated with the goal of maintaining sufficient accuracy and acceptable execution speed. The model currently uses R134a exclusively as the refrigerant. However, other refrigerants can be used if the user builds the required property tables in a specified text file format.

In the following sections, the 0-D volume block, the 1-D pipe block and various other modeling blocks are described in detail.

The 0-D Volume Simulation Block

The 0-D volume blocks serve as models for real volumes in the system as well as the connectors between the 1-D pipe blocks. In the 0-D volume block, the sum of the incoming refrigerant mass flow rates minus the sum of the outgoing refrigerant mass flow rates determine the time derivative of

the refrigerant mass in the volume, and the sum of the incoming refrigerant enthalpy flow rates minus the sum of the outgoing refrigerant enthalpy flow rates plus the net heat transfer to the volume determine the time derivative of the internal energy in the volume. Note that when saturated conditions exist in the 0-D volume block, saturated liquid, saturated vapor, or a saturated mix can be selected for the output flow, according to the system component represented by the simulation block. An accumulator, for example, is physically designed to only let out saturated vapor.

Since internal energy and mass are the state variables for the simulation, all other material properties are looked up from two-dimensional tables based on specific internal energy and density. Such lookup tables provide the best numerical performance as no iteration on properties is required.

The 1-D Pipe Simulation Block

In all versions of the 1-D pipe simulation blocks, on the refrigerant side, equations for conservation of mass, momentum and energy are solved through a finite volume method. As with the 0-D volume block, all refrigerant material properties are obtained from two-dimensional tables based on specific internal energy and density. The evaluation of heat transfer between the pipe wall and the refrigerant, and between the pipe wall and air is incorporated through local heat transfer coefficient correlations. The effectiveness-NTU method is applied on the air side, which ensures that the exit air temperature does not overshoot the wall temperature. The time derivative of the wall temperature is calculated from the net heat flow rate to the wall. Therefore, the thermal capacity of the wall has been accounted for. However, the temperature drop across the pipe wall has not been accounted for. It was determined that the introduced error is small for the relatively thin walls of the flat tubes / plates found in typical compact heat exchangers in an A/C system. In the direction of air flow, the temperature of the wall is constant. From the applied finite volume method, it is inherent that in the 1-D pipe simulation block, refrigerant flow can take place in both directions.

The simplest version of the 1-D pipe simulation block is used to model the refrigerant lines connecting the various A/C system components. Here a circular pipe is simulated in a perpendicular cross-flow of air. For these connecting lines, the effects of the relative humidity and condensation of water on the pipe are neglected.

More complex versions of the 1-D pipe simulation block are used for simulating the refrigerant and air flow and heat exchange process for the flat tubes of the condenser and plates of the evaporator. These 1-D pipe model block versions include the options of multiple “parallel channels” on the refrigerant side and more complex fin geometry on the external air flow side. Humidity in the air and condensation of water from air is accounted for. It is assumed that the refrigerant flow properties inside the parallel channels are identical. Also, the wall/fin temperatures in the direction of

the air flow are assumed to be constant. This is certainly an approximation, but one necessary to maintain reasonable simulation execution speed.

The finite volume formulation that is used for the refrigerant flow is a conservative method in the sense that the mass, momentum and energy are conserved very accurately. For the air flow, there are no conservative terms and the flow is described with purely algebraic equations.

Refrigerant-side equations

The general control volume equations for the conservation of mass, momentum and energy as presented in [9] are used as a starting point. The general control volume equations are then written for one dimension, as all flow variables are assumed to change only in the direction of refrigerant flow, and for finite volumes, that is small volumes, over which the flow variables can be considered uniform. Note that the refrigerant is in thermodynamic equilibrium in the finite volumes, and velocity “slip” between the liquid and the vapor phase is not factored in. The refrigerant is also considered homogeneous, that is, all flow properties, including liquid and vapor phase volume ratios, are the same everywhere within the finite volume. Finally, the effects of gravity are neglected.

Conservation of mass

We start out from the conservation of mass ([9], Eq. 3.3.1):

$$0 = \frac{\delta}{\delta t} \int_{cv} \rho \, dV + \int_{cs} \rho \underline{v} \cdot \underline{dA} , \quad (1)$$

where ρ is the refrigerant density, $\underline{v}$ is the velocity vector, $\int_{cv} dV$ is the volume integral and $\int_{cs} \underline{dA}$ is the surface integral. The first integral on the right is the mass in the finite volume, and then for the one dimensional flow for the finite volume:

$$\frac{dm}{dt} = \rho_{in} A v_{in} - \rho_{out} A v_{out} , \quad (2)$$

where A is the pipe cross sectional area, and the *in* / *out* subscripts are for inlet and outlet boundaries of the finite volume, defined also as the left-side and right-side boundaries, respectively, as we use the convention that the positive flow is from left to right.

Conservation of momentum

The conservation of momentum written for a control volume ([9], Eq.3.3.8):

$$\Sigma F = \frac{\partial}{\partial t} \int_{cv} \rho \underline{v} \, dV + \int_{cs} \rho \underline{v} \underline{v} \cdot \underline{dA} , \quad (3)$$

where ΣF is the sum of all forces including shear and pressure forces on the control volume. We define the first integral on the right side as the linear momentum. For finite

volume formulation for one-dimensional flow the following can be written:

$$\frac{dl}{dt} = \rho_{in} A v_{in}^2 - \rho_{out} A v_{out}^2 + (p_{in} - p_{out})A + F_{wf}, \quad (4)$$

where p_{in} and p_{out} are the pressures on the inlet and outlet boundary, respectively, and F_{wf} is the force from wall friction. For the wall friction force, a version of the Darcy-Weisbach formula ([9], Eq. 5.8.7) was used in which the wall friction factor was obtained with the Hagen-Poiseuille equation ([9], Eq. 5.10.12) for laminar flow and from a modified version of the Colebrook equation ([9], Eq. 5.10.13) for turbulent flow. This latter equation does take into account the effect of the relative roughness of the pipe wall. These equations are used for both single and two-phase flow. In the two-phase region, the viscosity was obtained as the quality weighted average of the saturated liquid and saturated vapor viscosity. A user-adjustable calibration coefficient was also implemented.

Conservation of energy

The energy equation written for a control volume ([9], Eq. 3.3.6):

$$\frac{\delta Q_H}{\delta t} - \frac{\delta W_s}{\delta t} = \frac{\partial}{\partial t} \int_{cv} \rho e dV + \int_{cs} \left(\frac{p}{\rho} + e \right) \rho \underline{v} \cdot \underline{dA}, \quad (5)$$

where $\frac{\delta Q_H}{\delta t}$ is the rate of heat addition, $\frac{\delta W_s}{\delta t}$ is the rate of work done by the system via shear forces, and e is the total energy

per unit mass: $e = u + \frac{v^2}{2}$, where u is the specific internal energy and v is the velocity. Note that all components other than the internal and kinetic energy have been neglected. This equation also assumes no other boundary work than that on the inflow and outflow surfaces. We define E as the first integral on the right side, which is the total energy in the finite volume. Then, assuming 1-D flow, no shear force work, and heat addition strictly through the pipe wall (no conductive heat transfer through inflow/outflow boundaries), the following equation can be written for each control volume in the pipe (also will be called as a "segment" of the pipe):

$$\frac{dE}{dt} = A v_{in} \left(p_{in} + u_{in} \rho_{in} + \rho_{in} \frac{v_{in}^2}{2} \right) - A v_{out} \left(p_{out} + u_{out} \rho_{out} + \rho_{out} \frac{v_{out}^2}{2} \right) + Q_{wr}, \quad (6)$$

where Q_{wr} is the heat transfer rate from the pipe wall to the refrigerant flowing inside the pipe segment. Note that Q_{wr} can be written as

$$Q_{wr} = \bar{h} A_i (T_w - T), \quad (7)$$

where $\bar{h}$ is the average heat transfer coefficient over the pipe segment, A_i is the inside area of the pipe segment wall, and T_w is the pipe segment average wall temperature. To calculate $\bar{h}$, the Dittus-Boelter equation ([10], Eq. 8.58) and/or the Chen correlation [11] is used. Note that the transition points between the single-phase and two-phase regions along the length of the pipe need to be determined for the proper application of the heat transfer rate correlations. These transition points are easily obtained from the mass and total energy, which are simulation state variables and therefore available in the finite volumes at the start of each time step.

Spatial discretization

The spatial discretization was implemented as shown in Fig. 1. Note that a "staggered" grid was used. While the control volumes were defined for mass and energy as the volumes shown between the 0-1, 1-2, 2-3, ... location indexes (Type 1 finite volumes), for the linear momentum they were defined as the volumes between the 0-0a, 0a-1a, 1a-2a, 2a-3a, ... location indexes (Type 2 finite volumes). So in fact there are two sets of finite volumes, one set for the mass and energy equations, and one set for the momentum equations. This staggered spatial discretization method had to be used to avoid instability problems. Other authors have also used this method [12], perhaps for the same reason. Note that the first and last Type 2 finite volumes (finite volumes for linear momentum) are "half" size compared to the rest of the ones in the middle of the pipe.

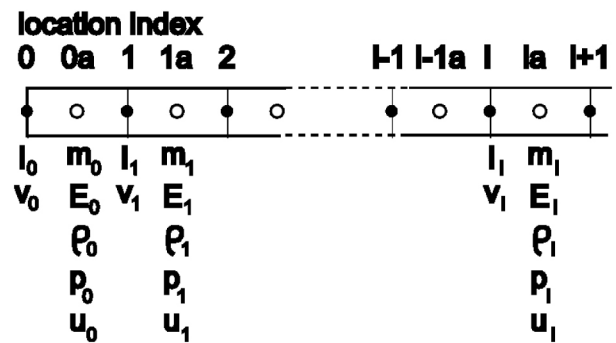


Fig. 1. Spatial resolution for the finite volume formulation

To implement the conservation equations, the variables ρ_{in} , ρ_{out} , p_{in} , p_{out} , v_{in} , v_{out} , u_{in} and u_{out} (the density, pressure, velocity and specific internal energy on the finite volume boundaries, respectively) have to be obtained from the conserved variables m , I and E (the mass, the momentum and energy, in the finite volumes, respectively). The conserved variables are also the simulation state variables that are available immediately after the integrator advances one time step. From the definition of these variables:

$$\rho = m/V$$

$$v = I/m \quad (9)$$

$$u = E/m - v^2/2 \quad (10)$$

$$p = p(\rho, u) \quad (11)$$

Then a simple first-order interpolation of the flow variables on the finite volume boundaries was implemented. For example, for the conservation of mass, the density and velocity on the left-side boundary of the i^{th} Type 1 finite volume is estimated as

$$\rho_{in} = (\rho_{i-1} + \rho_i)/2 \quad (12)$$

$$v_{in} = v_i, \quad (13)$$

and for the conservation of momentum equation, the density and velocity on the left-side boundary of the i^{th} Type 2 finite boundary are estimated as

$$\rho_{in} = \rho_{i-1}, \quad (14)$$

$$v_{in} = (v_{i-1} + v_i)/2. \quad (15)$$

Note how the interpolations are different between conservation of mass and conservation of linear momentum due to the staggered Type 1 and Type 2 finite volumes.

Applying the boundary conditions

In this section we discuss the boundary conditions of the overall computational domain, that is, the left-side boundary of the first finite volume and the right-side boundary of the last finite volume. Flow variables on these surfaces cannot be calculated the same way as on the boundaries between two finite volumes because they are attached to 0-D volume blocks. The boundary conditions have to be treated differently depending on whether inflow or outflow was present on the boundary, determined by the sign of the linear momentum, I . For the inflow, conditions from the connected 0-D volume block could be applied readily to obtain the boundary conditions. Calculating the outflow boundary conditions is more challenging as there are various ways of extrapolating flow variables to the boundary and some of these methods did not work well regarding stability, accuracy or speed. In the end, the following method was found to work well. To calculate the boundary velocity, the linear momentum in the Type 2 boundary finite volume (available as state variable) was divided by one-half the mass in the Type 1 boundary finite volume. The boundary pressure for outflow was simply the pressure in the connected 0-D volume. The specific internal energy on the boundary was extrapolated linearly from the first two regular Type 1 finite

volumes next to the boundary. Finally, once the boundary internal energy and pressure were known, the boundary density could be calculated through an iterative process using the pressure vs. specific internal energy and density tables.

Air-side equations

The heat transfer rate from the air flow to the wall of a pipe segment can be written as

$$Q_{aw} = \bar{h}_a \cdot A_{tot} \cdot (T_a - T_w), \quad (16)$$

where $\bar{h}_a$ is the average air-to-wall heat transfer coefficient in the pipe segment, T_w is the pipe segment average wall temperature, and A_{tot} is the total surface available for heat transfer on the air side in the pipe segment.

The air-side equations are different for the various versions of the 1-D pipe model blocks. For the transport refrigerant pipes, the correlation for heat transfer on a pipe in a perpendicular cross-flow of air was implemented according to Eq. 7.53 of [10]. For air-side heat exchange on the finned heat exchangers, the Chang correlation [13] was implemented. It is noted that although the Chang correlation did not incorporate the effects of humidity and condensation, we still apply this correlation for moist air.

Chang proposed a detailed and a simplified correlation, where the local heat transfer coefficient is calculated by using a correlation for the Colburn j-factor, j . The simplified correlation is

$$j = 0.425 Re_{Lp}^{-0.496}, \quad (17)$$

where Re_{Lp} is the Reynolds number based on the louver pitch, L_p (see Fig. 2). The more complex version of the Chang correlation incorporates a dependency on louver angle, fin length, tube depth, louver length, tube pitch and fin thickness. The model user can switch between the simplified and more detailed correlation versions. The details of how the heat transfer coefficient can be obtained from the Colburn j-factor can be found in [10] p.534.



Fig. 2. Louver pitch used in the Chang correlation

The local heat transfer coefficient, h_a obtained by either the simpler or the more complex Chang correlation has to be adjusted to get an average heat transfer coefficient, $\bar{h}_a$ by incorporating the effects of the fins not being able to maintain the same temperature as the flat tube itself

$$\bar{h}_a = \eta_0 \cdot h_a, \quad (18)$$