2016

A Review of State of the Art in Modeling of Air-to-Refrigerant Heat Exchangers for HVAC&R Applications

Vikrant C. Aute
University of Maryland, United States of America, vikrant@umd.edu

A Review of the State of the Art in Modeling of Air-to-Refrigerant Heat Exchangers for HVAC&R Applications

Vikrant AUTE[1]*

[1]Center for Environmental Energy Engineering, University of Maryland
College Park, MD 20742, United States
Tel: +1 301 405 8726, Email: vikrant@umd.edu

* Corresponding Author

ABSTRACT

Air-to-refrigerant heat exchangers are a key component in all air-conditioning, heat pump and refrigeration systems. The most common of types of air-to-refrigerant heat exchangers are tube-fin and microchannel heat exchangers. There has always been a great emphasis on understating the underlying physics and improving the performance of these heat exchangers. More recently, researchers have been investigating the use of small hydraulic diameter flow channels as well as novel heat transfer surfaces for use in such heat exchangers. The novel designs not only include shape optimized tubes, but also tube bundles with varying tube and fin geometries. In order to design optimum heat exchangers for a given application, it is crucial to use a reliable thermal-hydraulic model to evaluate the performance of air-to-refrigerant heat exchangers. In the last two decades, significant strides have been made in modeling of tube-fin and microchannel heat exchangers. The different modeling techniques include the use of performance maps, LMTD and epsilon-NTU based methods and fully discretized finite volume approaches. In terms of accuracy, the finite volume models are by far the preferred ones. The goal of this paper is to present the state of the art in finite volume modeling of air-to-refrigerant heat exchangers and to highlight research that stretches the boundaries of conventional heat exchanger modeling methods. The review starts out with a comprehensive survey of finite volume models in the literature and their capabilities to account for the various underlying physical phenomenon. High level modeling paradigms are derived and the best practices are highlighted. The various methods of geometry and circuitry representation and solution methodologies are summarized. Majority of these models rely on empirical correlations for local heat transfer and pressure drop evaluations. The use of such correlations has its own challenges and the lessons learned from the literature in this context are highlighted. Air and refrigerant flow maldistribution, especially in microchannel heat exchangers, is a critical phenomenon that needs to be accounted for in such models. Refrigerant flow maldistribution models in the literature range from user-specified quality and mass flow distribution profiles to more sophisticated methods that use CFD-based co-simulation techniques. The different techniques for handling dehumidifying conditions, such as those in an evaporator, are summarized. Lastly, recent developments in the field of optimization of air-to-refrigerant heat exchangers is presented. The review concludes with some thoughts on what the future of air-to-refrigerant heat exchanger design and optimization might be.

1. INTRODUCTION

Air-to-refrigerant heat exchangers have been a subject of active research for more than five decades, since they are a key component in any Heating, Ventilation, Air-Conditioning and Refrigeration (HVAC&R) system. In addition to the improving the performance of these heat exchangers, the modeling, simulation and optimization has also been a topic of research. The goal of modeling and design optimization is to make sure that the first prototype that is built is 90% right. To make sure this happens, it is crucial to have models that are accurate, reliable and easy to use. This does not mean that one should always use the most complex models or the model with the highest fidelity. In fact, each of these models with varying accuracy and speed has its place in the simulation hierarchy. The goal of this paper is to concisely summarize the state of the art in steady state modeling of air-to-refrigerant heat exchangers for HVAC&R applications.

Due to space limitations, some details are omitted here in the interest of staying on topic. As such, the reader is assumed to be familiar with basics of air-to-refrigerant heat exchanger operation and analysis.

2. MODELING OVERVIEW

In this section a brief literature summary is provided separately for tube-fin heat exchangers and micro-channel heat exchangers. We note that most state of the art models (in terms of accounting for most of the phenomenon) are finite volume based, also known as distributed parameter models. The literature reviews are limited to finite volume models.

2.1 Literature Summary for Tube-Fin Heat Exchanger Models

Amongst the earliest simulations of a heat exchanger, Hermann (1962) developed an electronic analog computer flow chart to set up a steam generation process in a heat exchanger. The rapid development of electronic computers led the heat transfer community to realize the vast potential of computers in the study of heat transfer. In another paper titled Katz and Briggs (1965) outlined areas showing increasing use of computers at the time, including development of correlations, solving fundamental equations and optimizing heat exchanger designs. Since then, the field has matured and today, there are several heat exchanger models and tools in the literature used to model both steady state and transient behavior of heat exchangers, which have been validated against experimental results.

Rossi and Braun (1995) presented a heat exchanger model which is part of ACMODEL (Shen, et al.,2004) vapor compression cycle simulation tool. This heat exchanger model is based on the Effectiveness-NTU (ε-NTU) approach, and used correlations to obtain required coefficients for heat transfer, pressure drop, void fraction etc. Domanski (2003) presented a public-domain heat exchanger design and simulation tool, EVAP-COND. It followed a tube-by-tube approach for modeling heat transfer. EVAP-COND offers many features like refrigerant maldistribution through circuits of different lengths and one-dimensional air flow maldistribution. Jiang et al. (2002, 2006) presented a model based on segment-by-segment approach for modeling heat transfer. In this model, each tube is divided into several segments. This allows the user to model two-dimensional air flow maldistribution on coil face. Liu et al. (2004) developed a general steady state model for a fin and tube heat exchanger based on graph theory. Their model accounts for refrigerant distribution through a flexible circuitry arrangement and accounts for heat conduction between tubes as well. Their approach is not based on the ε-NTU method. Rather, they apply conservation of energy to a given control volume, starting with guessed outlet states for air and refrigerant as well as guessed wall temperatures. Liang et al. (2001) studied the effect of refrigerant circuitry on evaporator performance through numerical modeling. They developed a distributed simulation model for predicting the steady state operation of an R-134a evaporator coil. Oliet et al. (2002) carried out a numerical simulation of the dehumidification on tube-and-fin heat exchangers and suggested various modeling strategies. They suggest three different strategies, QUICKchess, BASICchess and ADVANCEDchess, all of them were aimed at accurately solving the dehumidification process. ADVANCEDchess model accounts for tube-to-tube heat transfer in calculating temperatures on a discretized two-dimensional fin surface, which is used to accurately obtain heat transfer to air through convection, details of liquid film formation and actual fin efficiencies. These discretized quantities are lumped into macro volumes formed by fin and tube and applied to relevant equations at that level. Lee and Domanski (1997) suggested a model to account for heat conduction through fins. This model is capable of accounting for tube-to-tube conduction in a tube-by-tube heat exchanger model. In this model, the heat transfer to air and refrigerant is calculated using the ε-NTU approach for every tube in the heat exchanger. On the second iteration, using wall temperatures obtained from the first iteration, and first order neighboring tubes around the current tube being calculated. Recently, Singh et al., proposed ε-NTU and energy balance based models that can account for fin conduction as well as variable geometries, as described in Section 3.

2.2 Literature Summary for Microchannel Heat Exchanger Models

Unlike tube-fin heat exchangers, most microchannel heat exchanger models were developed after year 2000, when they were widely adopted in the automotive radiator and condenser applications, as well as gas coolers in supercritical CO_2 applications. The fundamental equation set for finite volume simulations of MCHX is very similar to that of TFHX. Yin et al. (2001) developed a finite-volume first principles based CO_2 gas cooler model. Empirical correlations were used to predict the heat transfer coefficients, pressure drop and fin efficiency. The model is shown to predict capacity within 2 percent of the experimental values. Kim & Bullard (2001), Yun et al. (2007) and Jin et al. (2011) presented several CO_2 microchannel evaporator models for automotive applications. Energy and mass conservation principles are applied under wet surface condition. In their presented validation, these evaporator models yield to less than 10% deviation on overall heat capacity. However, the condensation prediction of Kim & Bullard's model has a Root Mean Square (RMS) error of 13.1% and the sensible heat capacity prediction's mean deviation by Yun et al. is 17.3%. These indicate that the dehumidification phenomenon is not sufficiently captured in these two CO_2 evaporator models. Although the same model discretization approach and fundamental equations were applied, Jin's model reported lower RMS error of the calculated condensation rate (±8.2%). The use of separate air-side heat transfer

correlations for dry and wet conditions could be the main contribution to this improvement. Asinari et al. (2004) studied the heat conduction effect based on a microchannel gas cooler model. The model adopted a hybrid finite-volume and finite-element approach to model the fin performance.
Effectiveness-NTU method is proven to be an efficient and robust method and has been used in many of the MCHX models (Jiang, 2003; Schwentker et al., 2005; Shao et al., 2009; Brix et al., 2009; Brix et al., 2010; García-Cascales et al., 2010; Tuo et al., 2012). Schwentker et al. (2005) and García-Cascales et al. (2010) discretized the model on a per tube basis. The heat transfer and pressure drop in the ports are assumed uniform within the tube. The R134a and R410A condenser experimental validations demonstrated that such approach is computationally fast and accurate. Brix et al. (2009, 2010) performed parametric studies on refrigerant mass flow rate, quality distribution and air-side flow distribution based on a simplified MCHX model. The control volume (segment-by-segment) has been refined from tube to single port in two of the recent publications (Shao et al., 2009 and Ren et al., 2013). The serpentine type microchannel condenser model by Shao et al. (2009) is the first MCHX model that adopted port-by-port calculation. Three dimensional heat conduction terms are integrated into the fundamental equation set. More recently, Huang et al. (2014b) proposed a model that is capable of accounting fin conduction and also variable geometry designs.

2.2 General Assumptions

Based on the literature, it is observed that the following general assumptions have been made in the models:

1. Steady state
2. Thermally and hydro-dynamically fully developed flow
3. Straight air-flow from HX face into and out of the HX.
4. Refrigerant properties are assumed to be constant for a given control volume.
5. For air-side and fin conduction, a tube is assumed to interact only with its direct neighbors.
6. Negligible heat conduction in the refrigerant flow direction
7. Refrigerant is well mixed in intermediate headers

Majority of the models use the NIST REFPROP (Lemmon et al., 2013) database for refrigerant thermophysical property data. The REFPROP database uses the full equation of state for evaluating properties and can be computationally expensive. Hence, almost all models use some form of approximation such as table look-ups, curve fits, and hybrid methods. On the air-side, the ideal-gas model for air-water mixture as given in the ASHRAE handbook can be used or the real-gas model (computationally expensive) can be used.

3. MODELING DETAILS

In this section, the state of the art in terms of modeling is reviewed according to the various modeling aspects.

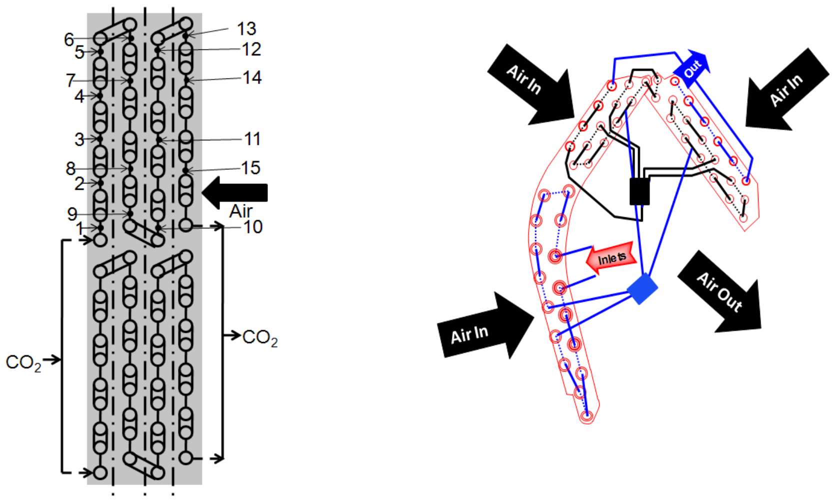
3.1 Geometries

For the purposes of this paper, we are limiting our scope to air-to-refrigerant heat exchangers, mainly tube-fin and microchannel heat exchangers. Fig. 1a shows the conventional tube-fin heat exchangers, whereas Fig 1b shows a more sophisticated design with variable tube diameters, variable fin types and fin densities within the same core. A similar design is shown in Fig 2, but for microchannel heat exchangers. Fundamentally, the control volume can be visualized as a cross-flow heat exchanger with a certain refrigerant flow channel (tube) and fin area (air-side flow channel) associated with it. The models proposed by Singh et al. (2009) has been extensively validated for the variable geometry TFHX, whereas as the one proposed by Huang et al. (2014b), can model and has been validated extensively for the variable geometry MCHX. More recently, Bacellar et al. (2015) have used the models by Jiang et al., (2002) and adapted it for simulation and validation of bare tube bundles as well as non-round tube heat exchangers with and without fins. The hydraulic diameters in their study range from 0.5mm to 5mm.

3.2 Control Volumes

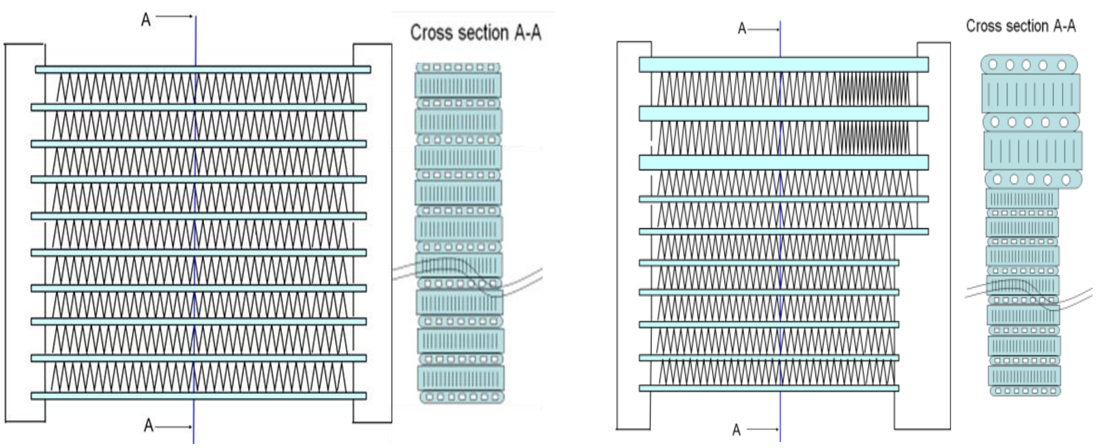
The control volumes, also known as segments are typically formed by dividing a tube into a pre-defined number of sections. The different types of control volumes are shown in Figure 3. For a TFHX, the control volume is simply a tube and an associated fin surface. The same analogy can be used for MCHX as well. As an extension of this, one can arbitrarily define a tube shape and an associated fins surface with it and carry out the simulation using any of the above mentioned methods. For the case of MCHX, the port-level control volumes allows one to account for port-level heat transfer and maldistribution as first adopted by Shao et al. (2009) and later by Huang et al. (2014).

The number of control volumes dictates the accuracy of the simulation as well as the computation time. Based on numerous studies from the literature and in-house validations, it appears that for segmented models, 15 segments per meter of tube length for up to 6 rows of tubes is an acceptable starting point. In the model proposed by Jiang et al. (2002), the individual control volumes can sub-divide themselves into multiple control volumes to track the location of phase change point within a control volume. Appropriate correlations are then used to evaluate the heat transfer and pressure drop in these sub-divided control volumes.



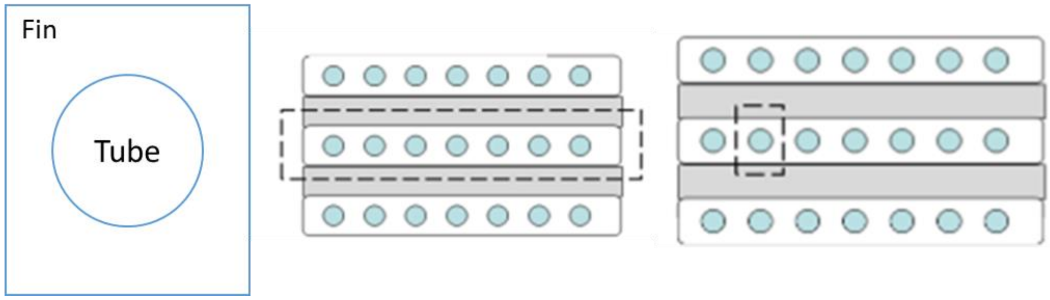
(a) (b)

Figure 1: (a) TFHX based on conventional geometry, (b) TFHX based on variable geometry



(a) (b)

Figure 2: (a) Conventional geometry MCHX, and (b) Variable geometry MCHX



(a) (b) (c)

Figure 3: (a) Control volume for TFHX, (b) tube-level control volume for MCHX, and (c) port-level control volume for MCHX

3.2 Method of Flow Representation

Most air-to-refrigerant heat exchangers have different circuiting (for tube-fin) or pass configurations (for microchannels). The order of the refrigerant and air-flow paths, in relation to the physical location of the tubes typically dictates the calculation methodology. The method of Junction-Tube matrix (Jiang et al., 2002) or the method of graph theory (Li et al., 2004) can be used. Fundamentally, these two methods are based on the concepts of adjacency matrix and adjacency lists, respectively, which are very well described in literature on Graph Theory (first developed by Euler in 1736). Both methods allow for representation of any arbitrary circuitry with any number of flow splits or flow merges for both TFHX, MCHX or even an arbitrary flow network.

3.3 Heat Transfer Calculation Method

Typically, one of the following three methods are used for heat transfer calculations, (a) Epsilon-NTU, (b) UA-LMTD (or AMTD), and (c) Energy Balance. The Epsilon-NTU method has the advantage that it is an explicit method and does not require a solution to a system of equations. The other two methods, both, require one to solve a system of equations in an iterative manner, potentially making them more time consuming. For certain MCHX designs such as the variable geometry MCHX (Huang et al., 2014b), the lesser known three-stream Effectiveness-NTU method was used by Huang et al. in addition to the UA-AMTD method. Other variations of the effectiveness-NTU method have been proposed, such as the one by Zhong et al. (2015) but the specific advantages of their method are not clear, since some of the underlying effects are accounted for through model tuning factors. The calculation of local heat transfer coefficient is discussed in the section on Correlations.

For heat transfer and pressure drop calculations, typically, Pressure and Enthalpy are used as the state variables on the refrigerant side. On the air-side typically, pressure, temperature and humidity ratio (or relative humidity) are used.

The effect of lubrication is generally handled by using an oil-circulation ratio followed by a mass-balance on the oil in each control volume. The corresponding pressure drop and heat transfer coefficient penalties are handled via custom correlations that include the effect of oil. Detailed modeling can be found in the work by Radermacher et al. (2006) for TFHX and more recently by Li and Hrnjak (2015) for MCHX.

3.4 Pressure Drop Calculation Method

Typically, correlations are used to calculate the local heat transfer coefficient and pressure gradient. On the air-side, for most conventional geometries, the pressure drop can be calculated independently of the heat transfer calculation if the outlet air density term is ignored. However, if the full equation (including outlet air density and the mean air density) is used, then the air-side pressure drop calculation needs to happen after the entire core is solved. Depending upon the application, the difference in predicted pressure drop between these two methods can be as high as 5%.

On the refrigerant side, pressure drop is calculated for each finite control volume. The thermophysical properties can be evaluated based on the inlet refrigerant state, or a mean of inlet and outlet. If outlet state is required, then this involves an additional iteration. For finite volume models with sufficient control volumes, the difference between the predicted pressure drops is small and can be ignored, thus giving an explicit pressure drop calculation and higher computational speed.

3.5 Correlations

Almost all the models rely on empirical or theoretical correlations for local heat transfer and pressure drop calculations. In addition, correlations are also used to evaluate void fraction, fin effectiveness, pressure drops in headers and u-bends and fin contact resistance. On the air-side, the correlations for heat transfer coefficient and pressure drop are very specific to the type of surfaces (tube and fin combinations) and cannot be used in a generic manner. Most of these correlations are also plagued by the fact that they are applicable to a limited range of geometrical parameters (e.g., tube diameters, vertical and horizontal spacing etc.) and extrapolation can lead to significantly erroneous results. One could revert to Computational Fluid Dynamics (CFD) to extend the range of applicability of these correlations. More recently, some authors (Bacellar et al., 2014) have used CFD to generate complete performance characteristics for different fin types and bare tube bundles, which can later be tuned as experimental data becomes available.

The refrigerant side correlations are typically categorized into liquid–phase, two-phase and vapor-phase correlations. The use of the relevant correlation is decided based on the refrigerant state at the inlet of each control volume. Generally, the single phase correlations (e.g., Gnielinski) are shown to work well for liquid as well as vapor phase for a variety of refrigerants. However, for certain fluids such as CO2, fluid specific correlations are required, especially for transcritical or supercritical applications.

The state of the art correlations for two phase (evaporation and condensation) include those that are based on flow regime maps. Cheng et al. (2008) provides significant insights into this. It should be noted that many of the flow regime maps based correlations are not as easy to implement and can have discontinuities. Given that the air-side is typically the dominant resistance in such heat exchangers, one can make a judicious trade-off between using a complex flow regime map-based correlation vs. a more generic and simple correlation.

Overall, it is recommended to use correlations that are applicable to a specific class of fluids. For the purposes of this discussion, we can classify typical refrigerants into the following classes: (a) Ammonia, (b) CO2, (c) Water/steam, (d) Hydrocarbons (e.g., R600, R290), (e) pure refrigerants (e.g., R134a, R32) and, (f) Mixture refrigerants (e.g., R407C, R407F).

While accounting for phase transition in a model, one should ensure that the local heat transfer coefficient and pressure drop values obtained from the different correlations are continuous. These issues generally do not affect HX simulations, but can cause convergence issues, when such a model is used in a system level simulation. No matter which correlations are used, one should be aware of the range of applicability (e.g., geometry, fluid states, flow rates, etc.) and any underlying discontinuities.

3.6 Dehumidification

Accounting for dehumidification phenomenon in a model is critical for accurate performance prediction of evaporations operating in wet-mode. In wet-mode, the air-side exchanges heat in sensible as well as latent mode. In the actual model, the tube wall temperature or the average fin surface temperature is compared to the dew point temperature of air, to judge if dehumidification will occur.
The most popular method used in majority of the models to account for dehumidification is the one based on McQuiston (1975), also known as the enthalpy potential method. This method makes some simplifications, but allows one to account for the latent load in each control volume with a simple bounded iteration based on the wall temperature or the fin surface temperature. The other methods include those by Domanski, which also accounts for the heat transfer resistance due to the water film that forms on the fin surface. Yet another method is the one by Braun et al. (1988), in which they develop a new technique using an effectiveness-based formulation to account for latent heat transfer. The difference between the enthalpy potential method and the effectiveness-based method is less than 5% points in terms of sensible heat ratio for typical residential evaporator applications.

3.6 Fin Conduction

The performance degradation due fin conduction can be significant in certain applications. Such applications include CO2 gas coolers, R410A condensers operating in extreme ambient (e.g., 55°C condensing temperature) conditions and sometimes even in evaporators of multi-circuit systems operating under part-load conditions. Zilio et al. (2007) have shown that the heat exchanger performance can degrade significantly (10 to 20%) for some applications.

In order to account for fin conduction, the energy-balance method is the only one that can be used for the core heat transfer calculation. The fin conduction models from the literature include those from Singh et al. (2009), for TFHX and those from Huang et al. and Martínez-Ballester et al., for MCHX. Both studies have shown that the performance degradation can be adequately captured by these models for the above-mentioned applications.

3.7 Air flow maldistribution

In real applications, due to the uneven flow from a fan, there is almost always non-uniform air-flow on the HX face. Majority of the design calculations assume uniform air flow, but this assumption can lead to significant errors in the predicted capacity. For tube-fin heat exchangers, air velocity distribution has been experimentally measured by Kirby et al. (1988) and Aganda et al. (2000). CFD simulations were carried out by Yashar et al. (2014) for tube-fin heat exchangers as well. However, the application of air velocity distribution in MCHX models is sparse due to the limitation of experimental testing. Most tube-by-tube and finite-volume (segmented) models are capable of accounting for air-flow maldistribution. In general, there are two methods of accounting for air flow maldistribution.

In the first method, the flow distribution is an input to the model in the form of air velocity value for each tube or for each control volume on the HX face. The distribution itself can be obtained by lab measurements or from detailed CFD analysis. The data from lab measurements, typically in the form of air-velocity as a function of height for a given fan-HX combination can be converted to a normalized polynomial and used in subsequent analyses. This is typically the easiest approach.

The second approach relies on a co-simulation between HX model and a detailed CFD package. This is the approach used by Singh et al. (2011). The key to success in such simulations is to note that the finite volume models are based on a Cartesian grid, whereas CFD packages use a non-uniform and non-rectangular grid. When the detailed velocity output from the CFD package is mapped from this non-uniform grid to a uniform grid, there is a possibility of mass loss (leading to wrong air-side mass flow rate values) leading to significant errors in prediction. In order to avoid such issues, an appropriate spatial averaging technique must be utilized to ensure mass conservation.

3.2 Refrigerant Flow Maldistribution

Most air-to-refrigerant heat exchangers have different circuiting (for tube-fin) or pass configurations (for microchannel heat exchangers). The refrigerant flow distribution is relatively easy to compute in a tube-fin heat exchangers, when reliable pressure drop correlations are available. This can include heat exchanger with feeder (capillary) tubes at both inlet and outlet. Care should be taken to use flow-regime appropriate correlations for the feeder tubes vs. the core tubes. For the MCHX, however, this is a much more complex problem, especially in evaporators. The flow maldistribution can lead to significant performance degradation.
In terms of modeling, there are three methods of account for this flow distribution. First is direct input of mass flow ratios for each tube in an MCHX header. These ratios can come from other sources such as experimental data or co-simulations. In the second method, specific equations for these mass flow ratios are developed based on measured data for a given heat exchanger and then used in the model. In the third method, a true model could be developed to compute the flow distribution with the header. This model however, still relies on accurate pressure drop correlations and given the uncertainty associated with two-phase pressure drop correlations, it is a matter of trade-off whether one should use such a header model or just apply correction factors to the bulk header calculations.

4. VERIFICATION, VALIDATION & UNCERTAINTY QUANTIFICATION

The task of verification generally refers to making sure that the model “does” what it is supposed to do. Verification can be carried out in several ways such as, (a) code or equation review, (b) trend-analysis, (c) comparison against other models or first principles. During this process, basic heat transfer and thermodynamic principles are used to make sure that the model is working. For example, one can check for trends of air-side heat transfer coefficient and pressure drop against velocity or look for second law violations. An example of comparison against other models is given in [Brix] where they use an established simulation tool to verify their model.

Validation is the explicit task of comparing the model results with measurements from a real physical system. It is important that the system be carefully instrumented for the explicit purpose of validation. It should be pointed out that commercial test standards do not necessarily account for all the state points required for a good validation of air-to-refrigerant heat exchangers. Most models in the literature have been validated with against one or more data sets and all the reported validations are based on the nominal (mean) reported data.

Model tuning or calibration in an integral aspect of model validation. Two types of model tuning methods have been identified based on the parameters to which the tuning corrections are applied. The first set of methods include those in which multipliers are used for local heat transfer, pressure drop, fin effectiveness and void fraction calculations. The use of such multipliers is the preferred way of model tuning, since they account for all the fundamental effects. In the second method, the correction factors are applied on top of the model predictions, i.e., multipliers are used to adjust the predicted capacity or pressure drop. However, such methods can sometimes lead of inconsistent fluid states if one were to actually conduct an energy balance verification on the final results. Depending upon the model end use, one or other or both methods are utilized.

Uncertainty Quantification (UQ) aims at reporting the uncertainty in the model predictions as a function of uncertainties in the underlying calculations. The sources of these uncertainties can include model discretization, numerical relaxation and uncertainties in empirical data and correlations used in the models. In the literature, the work by Domanski and Didion (1987) and Huang et al. (2014a) have shown UQ of model results with respect to the accuracy of the underlying thermophysical property calculations for vapor compression system and for heat exchangers respectively.

Lastly, compared to CFD codes, most of the HX models in the literature do not follow a rigorous VVUQ method (Oberkampf and Roy, 2010) as in some other engineering disciplines.

4. LEVERAGING OF MODELS FOR BETTER DESIGN

The ultimate use of a model is to exercise it with different inputs to better understand how the underlying system, in this case, the heat exchanger, would perform. To that effect, one can identify the following use cases for a model, (a) parametric analysis, (b) co-simulation, and (c) coupling with optimization or other solvers. These three case will be discussed here. Parametric studies simply involve executing the model with a set of pre-defined inputs. Such studies are helpful for trend analysis and general understanding of model behavior.

Co-simulation is a generic term that implies the use of two or more analysis packages/codes that account for a particular phenomenon using a numerically explicit or implicit computation approach. One example of an explicit approach is the coupling of an HX simulation with a CFD package, where in the CFD package provides the air-flow distribution which is then used to compute the HX performance. This is an example of explicit co-simulation. Huang et al. (2014c) show an example of implicit simulation for flow distribution in a microchannel header. The header of the MCHX is simulated using a CFD package, whereas the core of the heat exchanger is modeled using a finite volume code. The relevant boundary conditions are passed back and forth between the two packages in an iterative fashion until convergence is reached. Such kind of automated co-simulations open up possibilities for analysis of very complex HX and supporting components.

Design Optimization is the task where one really begins to appreciate the important of a good model. During optimization, the optimization algorithm generates different values for the inputs, for which the model is exercised and the resulting HX performance is reported back to the optimizer. This allows one to explore a practically infinite design space with relatively few (e.g., a few thousand to ten-thousand) simulations. The use of optimization algorithms with heat exchanger model has been carried out for several decades. A summary of the different objectives and constraints used in heat exchanger optimization problems is given in Aute et al. (2015). Some of the key objectives used in optimization include, cost, capacity, entropy generation, entransy, etc. However, the holy grail of optimization in tube-fin heat exchangers is the circuitry optimization problem. This is an np-hard problem and the design space grows exponentially with the additional of even a single tube. There are various studies in the literature that attempt at developing guidelines for circuiting based on specific applications and refrigerants, but to date, there are only two research groups who have tackled this problem from an optimization point of view. The first is the work by Domanski and colleagues (Domanski et al, 2005, Yashar et al., 2012) and the second is the one by Wu et al. (2008). They have shown capacity improvements of more than 5% for the optimal designs.

5. OUTLOOK

Compared to the early models from 1960s, the modeling of air-to-refrigerant heat exchangers has come a long way, especially with the availability of computational and experimental resources and the advent of new manufacturing technologies. This paper summarized the state of the art in this field and while these developments sound every exciting, there is plenty of room for innovation. Below are some thoughts on where the future research in this area might be headed:

1. Use of CFD to displace the correlations. This is already happening in some areas.
2. Large-scale co-simulation to address complex problems like flow distribution.
3. Multi-scale analysis and optimization, using simulation of novel tube and fin surfaces
4. Development of an "expert system", that would guide the engineer through incremental design improvements
5. Performance-based design optimization, where in an engineer would specify the requirements and the underlying computer program would figure out if the HX should be a tube-fin or a microchannel or some other novel shape, the flow arrangements, the envelope etc.
6. Rapid prototyping using advanced manufacturing such as 3D printing to assist in validation.
7. Rigorous use of VVUQ methods, which in turn would assist system integrators in designing better systems.

REFERENCES

Aute V., Huang L., and Radermacher, R., (2015), A survey of optimization formulations and techniques for the design of heat exchangers using lower GWP refrigerants, ASHRAE Winter Conference, Paper # CH-15-C039, Chicago.

Bacellar, D., Aute, V., Radermacher, R., (2015), A Method for Air-To-Refrigerant Heat Exchanger Multi-Scale Analysis and Optimization with Tube Shape Parameterization, 24th IIR International Congress of Refrigeration, August 16 – 22, 2015 – Yokohama, Japan.

Bacellar, D., Aute, V., Radermacher, R., 2014, CFD-Based Correlation Development for Air Side Performance of Finned and Finless Tube Heat Exchangers with Small Diameter Tubes, *15th Intl. Refrigeration and Air Conditioning Conference at Purdue*, July 14-17, 2014, Purdue, Indiana.

Braun, J. E., Klein, S. A., and Mitchell, J. W., (1988), Effectiveness model for cooling towers and cooling coils, *ASHRAE Transactions* 95(2).

Brix, W., M.R. Kærn, and B. Elmegaard. "Modelling refrigerant distribution in microchannel., *Intl. J. of Refrigeration* 32 (2009): 1736-1743.

Brix, W., M.R. Kærn, and B. Elmegaard, (2010), Modelling distribution of evaporating CO2 in parallel minichannels, *Intl. J. of Refrigeration*, 33: 1086-1094.

Cheng, L., Ribatski, G., and Thome, J., (2008), Two-Phase Flow Pattern and Flow-Pattern Maps: Fundamentals and Applications, ASME Applied Mechanics Reviews, Vol. 61, pp. 050802-1 to 050802-28.

Domanski, P., Yashar, D., and Kim M., (2005), Performance of a finned-tube evaporator optimized for different refrigerants and its effect on system efficiency, Intl. J. Refrigeration, Vol. 28, No. 6, pp. 820-827.

Domanski, P, and Didion, D., (1987), Impact of Refrigerant Property Uncertainties on Prediction of Vapor Compression Cycle Performance, NBSIR 86-3373, United States Dept. of Commerce, National Bureau of Standards.

Huang, L., Aute, V., and Radermacher, R., (2014a), Uncertainty Analysis on the Prediction of Heat Transfer Coefficient and Pressure Drop in Heat Exchangers Due to Refrigerant Property Prediction Error, 15th Intl. Refrigeration and Air-Conditioning Conference at Purdue, July 14-17, 2014, Indiana.

Huang, L., Aute, V., Radermacher, R., (2014b), A model for air-to-refrigerant microchannel condensers with variable tube and fin geometries, *Intl. J. of Refrigeration, Vol. 40*, pp. 269-281.

Huang, L., Lee, M. S., Saleh, K., Aute V., Radermacher, R., (2014c), A computational fluid dynamics and effectiveness-NTU based co-simulation approach for flow mal-distribution analysis in microchannel heat exchanger headers, Applied Thermal Engineering, Vol. 65, pp. 447-457.

Jiang H., Aute V., Radermacher R.,2006, 'CoilDesigner: A General Purpose Simulation and Design Tool for Air to Refrigerant Heat Exchangers', Intl. J. of Refrigeration, Volume 29, Issue 4, June 2006, Pages 601-610 June 2006

Jiang H., Aute V., Radermacher, R., 2002, "A User-Friendly Simulation and Optimization Tool For Design Of Coils", 9th Intl. Refrigeration and Air Conditioning Conference at Purdue, West Lafayette, Indiana.

Lemmon, E.W., Huber, M.L., McLinden, M.O. 2013, NIST Standard Reference Database 23: Reference Fluid Standard Reference Data Program, Gaithersburg. Thermodynamic and Transport Properties-REFPROP, Version 9.1, National Institute of Standards and Technology.

Li, H, and Hrnjak, P., (2015), An experimentally validated model for microchannel heat exchanger incorporating lubricant effect, *Intl. J. of Refrigeration*, Vol. 59, pp. 259-268.

Liu, J., Wei, W., Ding, G., Zhang, C., Fukaya, M., Wang, K. and Inagaki, T. (2004), A general steady state mathematical model for fin-and-tube heat exchanger based on graph theory, *Intl., J of Refrigeration*, 2004, Vol., 27, No. 8, pp. 965-973.

Martínez-Ballester, S., J. Corberán, and J. Gonzálvez-Maciá. 2013a. "Numerical model for microchannel condensers and gas coolers: Part I - Model description and validation." International Journal of Refrigeration 36: 173-190.

McQuiston, F.C. (1975), Fin efficiency with combined heat and mass transfer, *ASHRAE Transactions* 81, no. 1 (1975): 350-355.

Oberkampf, W.L, and Roy, C. J., *Verification and validation in scientific computing*. New York: Cambridge University Press, 2010.

Radermacher, R., Cremaschi, L., and Schwentker, R., (2006), Modeling of Oil Retention in Suction Line and Evaporator of Air-Conditioning Systems, *HVAC&R Research*, Vol. 12, No. 1, pp. 35-56.

Ren, T., G. Ding, T. Want, and H. Hu., (2013), A general three-dimensional simulation approach for micro-channel heat exchanger based on graph theory." *Applied Thermal Engineering* 59 (2013): 660-674.

Rossi, T.M. and Braun, J.E. Computer Models for the Development, Evaluation, and Implementation of Fault detection and Diagnostic Methods for HVAC Equipment, Report No. : HL95-13(A).

Schwentker, R.A., V.C Aute, R. Radermacher, and K.B. Mercer. "Simulation and design tool for microchannel heat exchangers." Hoboken,NJ: *Proceedings of Fifth International Conference on Enhanced, Compact and Ultra-Compact Heat Exchangers*,

Shao, L.L., L. Yang, C.L. Zhang, and B. Gu, (2009), Numerical modeling of serpentine microchannel condensers, *Intl. J. of Refrigeration*, Vol 32, pp. 1162-1172.

Singh V., Aute V., Radermacher R., 2009, A heat exchanger model for air-to-refrigerant fin-and-tube heat exchanger with arbitrary fin sheet, *Intl. J. of Refrigeration*, Vol. 32, Issue 7, pp. 1724-1735

Singh V., Aute V., Radermacher Reinhard,2009, "Numerical approach for modeling air-to-refrigerant fin-and-tube heat exchanger with tube-to-tube heat transfer", *Intl, J. of Refrigeration*, Vol. 31, Issue 8, pp. 1414-1425.

Singh V., Abdelaziz O., Aute V., Radermacher R., 2011, Simulation of air-to-refrigerant fin-and-tube heat exchanger with CFD based air propagation, *Intl. J. of Refrigeration*, Vol 34 (8), pp. 1883-1897

Wu, Z. G., Ding, G. L., Wang, K. J., and Fukaya, M., (2008), Knowledge-based evolution method for optimizing refrigerant circuitry of fin-and-tube heat exchangers, *HVAC&R Research*, Vol. 14, No. 3, pp. 435-452.

Yashar, D. A., Wojtusiak, J., Kaufman, K., and Domanski, Piotr, (2012), A dual-mode evolutionary algorithm for designing optimized refrigerant circuitries for finned-tube heat exchangers, *HVAC&R Research*, Vol. 18, No. 5, 2012.

Yashar, D.A., P.A. Domanski, and H.H. Cho, (2014), An experimental and computational study of approach air distribution for slanted and A-shaped finned-tube heat exchangers, *HVAC&R Research*, no. 20 (2014): 498-507.

Yin, J.M., C.W. Bullard, and P.S. Hrnjak. (2001), R-744 gas cooler model development and validation, *Intl J. of Refrigeration*, Vol. 24, pp. 692-701.

Yin, X.W., Wang, W., Patnaik, V., Zhou, J.S., 2015. Evaluation of microchannel condenser characteristics by numerical simulation, *Int. J. Refrigeration*, 54, 126-141.

Zhong, Y., Glanville, P., Garrabrant, M., Stout, R., (2015), Design Methods for Evaporators with Zeotropic Mixtures on the Refrigerant Side, *ASHRAE Winter Conference*, Paper # CH-15-C038, Chicago.

Zilio, C., Cecchinato, L., Corradi, M. and Schiochet, G., (2007), An Assessment of Heat Transfer through Fins in a Fin-and-Tube Gas Cooler for Transcritical Carbon Dioxide Cycles, *HVAC&R Research Journal*, Vol. 13, No. 3, pp. 457-469.

ACKNOWLEDGEMENT

This work was supported by the Modeling and Optimization Consortium of the Center for Environmental Energy Engineering, at the University of Maryland.