2006

Generic Dynamic Model for Heat Exchangers

Shenglan Xuan
University of Maryland

Vikrant Aute
University of Maryland

Reinhard Radermacher
University of Maryland

Generic Dynamic Model for Heat Exchangers

Shenglan Xuan[1], Vikrant Aute[2] Reinhard Radermacher[3]
[1]Center for Environmental Energy Engineering, University of Maryland-College Park
College Park, MD 20706, USA
Tel: 301-405-8672, Fax: 301-405-2025, Email: *slxuan@umd.edu
[2]Center for Environmental Energy Engineering, University of Maryland-College Park
College Park, MD 20706, USA
Tel: 301-405-8672, Fax: 301-405-2025, Email: Vikrant@eng.umd.edu
[3]Center for Environmental Energy Engineering, University of Maryland-College Park
College Park, MD 20706, USA
Tel: 301-405-5286, Fax: 301-405-2025, raderm@umd.edu

ABSTRACT

A generic dynamic heat exchanger model for transient and steady state simulation has been developed. This model allows the user to easily create and customize a heat exchanger with any level of complexity and choose from a variety of working fluids. A combined moving boundary and finite volume method is implemented to solve the coupled energy and mass conservation equations for a given time step. The point is determined where the flow regime of the refrigerant changes within the heat exchanger and the segment is subdivided to improve the accuracy of the model. Correlations are used for the evaluation of convective heat transfer, pressure drop and void fraction on the refrigerant and air side. A comparison with another, extensively used and verified simulation tool is presented for validation of the model.

1. INTRODUCTION

A number of transient heat exchanger models with a wide range of complexity have been developed since the 1970's, and most of them came with a system model. Most recent attempts divided the heat exchanger into thermal segments to find overall performance.

Two popular distributed parameter methods are used currently in heat exchanger numerical simulation. These are the finite volume method and the moving boundary method. The first method divides the entire heat exchanger into a number of constant control volumes and all conservation equations are discretized into these control volumes to be solved sequentially or simultaneously. The second method divides the heat exchanger into different sections according to the phase state of working fluids. The boundaries between different phase states change with time during the transient simulation.

MacArthur (1984), MacArthur and Grald (1987, 1989) published papers of work to simulate heat exchangers by using a complete distributed parameters method, but it treated the two phase flow as a homogeneous flow and caused an inaccurate prediction of mass flow rate distribution. Williatzen et al (1998) presented a dynamic model which described the transient phenomena of a two phase heat exchanger by solving a set of one dimensional Ordinary Differential Equations (ODE) over three different zones (liquid, two phase and vapor). Rossi & Braun (1999) developed a fast yet large model of a roof-top air conditioning unit. The heat exchanger model is uses the finite volume method to solve mass and energy balance equations.

Jakobsen et al (1999) compared a homogeneous flow model and a slip-flow model in an evaporator with experimental data. The validation showed that the slip-flow model agreed very well with experimental data on dynamic response and is more accurate than the homogeneous flow model on charge calculation. They recommend the use of the slip-flow model when investigating the dynamic behavior of heat exchangers.

Bendapudi,et al. (2004) used both finite volume and moving boundary approaches to develop a shell-tube heat exchanger model. The comparison showed that the moving boundary approach was at least 2 times faster than the

finite volume method based on the execution speed, but the accuracy of both models was not represented. Hence, in order to shorten the computing time and maintain accuracy, it is felt that the moving boundary approach is a better choice when a large segment size is applied on heat exchanger simulations.

The objective of this work is to develop a generic dynamic heat exchanger model which can simulate different types of heat exchangers with any type of working fluids in both steady state and transient state. This model should be accurate, realistic – calculating its pressure drop and local heat transfer coefficient based on the dynamic parameters of heat exchangers.

2. THEROETICAL EVALUATION

2.1 Problem Formulation

A heat exchanger can be described mathematically in terms of equations of fluid properties, heat transfer, momentum, mass and energy conservation. Since transferred heat in radial direction is a very small fraction compared to the total transfer, the following assumptions will be made:

- One dimensional flow, ignoring flow diffusion
- Radial direction heat conduction inside the flow is neglected.

The mass and energy balance equations for the heat exchanger are as follows:

Mass balance on working fluids' side

$$\frac{\partial(\rho)}{\partial t} + \frac{\partial(\rho V)}{\partial z} = 0 \tag{1}$$

Energy balance on working fluids' side:

$$A_{cs}\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\dot{m}h)}{\partial z} = \dot{Q} + W \tag{2}$$

where, ρ is the density of the working fluid, V is the velocity of the working fluid, $\dot{m}$ is the mass flow rate of the working fluid, $\dot{Q}$ is the heat flux of the working fluid, W is the work done on the fluid, t is the time, z is the length, h and u are the working fluid enthalpy and internal energy respectively.

The transferred heat can be calculated depending on the local heat transfer coefficient and temperature difference between the working fluid and the boundary, which is:

$$\dot{Q} = HTC * A * \Delta T \tag{3}$$

where, HTC represents the heat transfer coefficient, A is the heat transfer area between heat exchanger tube and working fluid, and ΔT is the temperature difference between both substances.

For the heat exchanger, when we assume both sides have working fluids, we can write the energy conservation equation as:

$$m\ c_p \frac{dT}{dt} = \dot{Q}_1 - \dot{Q}_2 \tag{4}$$

where, m is the total mass of heat exchanger material, and c_p is the specific heat of the material. Once the heat exchanger is divided into segments in the simulation, m represents the heat exchanger mass of one segment.

In order to obtain the solutions, we need to solve the coupled mass, energy and momentum conservation equations. The momentum equation makes the problem complex and is not easy to solve. Hence, it is substituted using the friction factor to calculate the pressure drop, and the equation is:

$$\Delta P = f\frac{4L}{D_h}\frac{G^2}{2g\rho} \tag{5}$$

2.2 Numerical Algorithm

For a heat exchanger, once we divide it into a number of control volumes – segments, the energy and mass conservation equations in each segment are rewritten as:

$$\dot{m}_i - \dot{m}_{i+1} = \frac{A_{cr}\Delta z}{dt}(\bar{\rho} - \bar{\rho}^0) \tag{6}$$

$$\dot{m}_i h_i - \dot{m}_{i+1} h_{i+1} = \frac{(U - U^0)}{dt} + \dot{Q} \tag{7}$$

where $\bar{\rho}$ is the working fluid average density in the segment, and $\overline{\rho^0}$ is the working fluid average density during the previous time step. U is the current internal energy in the segment, and the U^0 is the internal energy at the previous time step.

The outlet pressure of the segment is calculated by:

$$P_{i+1} = P_i - \Delta P \tag{8}$$

One obvious characteristic of the heat exchanger in a vapor compression system is the change of flow regime, which affects the heat exchanger performance significantly. Hence, a good simulation model has to take care of the phase change when it calculates the heat transfer coefficient and pressure drop.

The Numerical simulation algorithm is described and shown in figure1. In this model, the nonlinear Broyden solver is used to find outlet conditions of each segment. The golden section search method is used to find the transition points between different phase regimes.

The length of different main phase regimes, vapor phase regime, two phase regime and liquid phase regime, of the heat exchanger is determined by calculating vapor quality based on the enthalpy and pressure. These conditions are:

- Liquid regime: when $h(p) < h_l(p)$, then $x=0$
- Two phase regime ; when $h_l(p) \le h(p) \le h_g(p)$, then $0<x<1$
- Vapor regime: when $h(p) > h_g(p)$, then $x=1$

Where, $h_l(p)$ and $h_g(p)$represent the saturation liquid and vapor enthalpy for a given pressure p, x is the quality of working fluid at that pressure.

Temperature, quality of the refrigerant, all of these thermal properties are evaluated using state equations:

$$T = f(p,h);\ x = f(p,h);\ \rho = f(p,h);\ \ldots$$

These state functions are calculated based on a special version of Refprop (2005), NIST Refprop 7.0 (2002).

2.3 Boundary Conditions and Initial Conditions

Boundary conditions: the boundary conditions for solving this numerical model are the inlet mass flow rate, pressure and temperature, or refrigerant quality in case that the inlet is two phase flow. Initial conditions: the initial conditions refer to the initial state of the working fluid in the component and the initial wall temperature of the heat exchanger.

2.4 Time step

Time step is a fixed or variable value depending on the time evolution of the boundary conditions. If the component model is part of a system model, the time step can be given by the system solver. Alternatively, the component can also pass the time step to the system solver and let the system solver determine the time step.

2.5 Convergence

In each segment, the values of flow variables at the outlet are obtained by solving the set of discretized equations from known inlet conditions iteratively. The solution procedure is carried out in a manner that moves forward step by step in the flow direction.

Convergence is verified at each segment using the following condition:

$$(1 - \left| \frac{h_{i+1}^{\ *} - h_i}{h_{i+1} - h_i} \right|) < \varepsilon \tag{9}$$

where h_{i+1}^{*} represents the outlet enthalpy value for a segment, h_i is inlet enthalpy for the current iteration in this segment and ε is a very small number and varies depending on the accuracy requirements of the model.

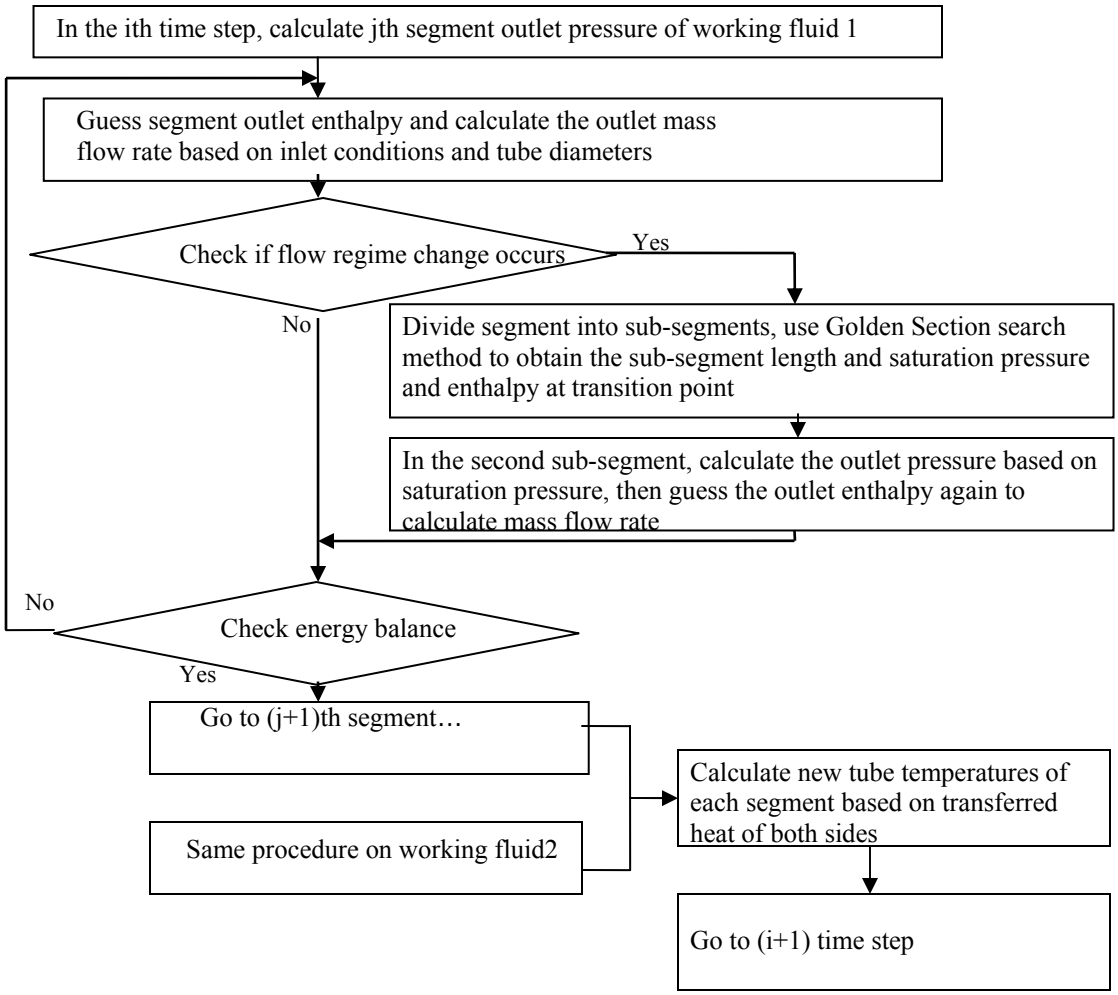

Figure 1: Flow chart of the algorithm

2.6 Choice of Number of Segments

The choice of segment number can be defined by users according to their accuracy and execution time requirements. If users just want a simple but fast model, only one segment representing an entire heat exchanger may be chosen. The segment will be automatically divided into sub-segments at the transition point by using the golden section search method if a flow regime change occurs, as is the case in a moving boundary approach model. If an accurate and detailed model is needed, the heat exchanger is divided into many segments, as is the case for a finite volume approach. However, the transition point is still searched for by golden section search method in the segment where a change in flow regime occurs.

2.7 Pressure Drop, Heat Transfer Coefficient and Void Fraction Calculation

The mathematical model requires local information about the friction factor and heat transfer coefficient. This information is generally obtained from the empirical or semi-empirical correlations. For the refrigerant side, Different correlations are used for single phase and two phase refrigerant pressure drop. Different Heat transfer coefficient correlations are also applied in term of the different situations such as evaporation, condensation or

single phase. For the air side, different correlations will be used to calculate heat transfer coefficient and pressure drop based on the fin type and surface conditions.

3. NUMERICAL RESULTS AND DISCUSSION

The simulation program has been written based on the above mentioned mathematical model and numerical procedure.

3.1 Numerical Results

The numerical results are obtained by inputting representative inlet conditions of both working fluids. The boundary conditions are also inputted before the simulation is run. All heat exchangers are divided into one hundred segments in order to obtain sufficiently accurate results. The inlet condition is two phase flow on the refrigerant side.. A time step of 0.01 second is chosen as the time step for these simulation cases to show the working fluids status change gradually and clearly.

Figure 2 represents the refrigerant temperature distribution along the flow direction in a cross flow air to refrigerant heat exchanger. Different curves represent the temperature distribution at different times. The X axis shows the normalized length of the heat exchanger and the Y axis shows the temperature. In this case, the refrigerant is heated and evaporated. The refrigerant inlet temperature is set up to 298 K and inlet quality equals 0.1. Initially, the refrigerant temperature in the heat exchanger changes little because the refrigerant is in the two phase region and the temperature is saturation temperature. The slight change is due to the pressure drop along the heat exchanger. With increasing time, the refrigerant which is close to the outlet becomes superheated vapor and its temperature increases. The superheated region is enlarged gradually and the refrigerant temperature in this region keeps increasing as a function of time until steady state is reached.

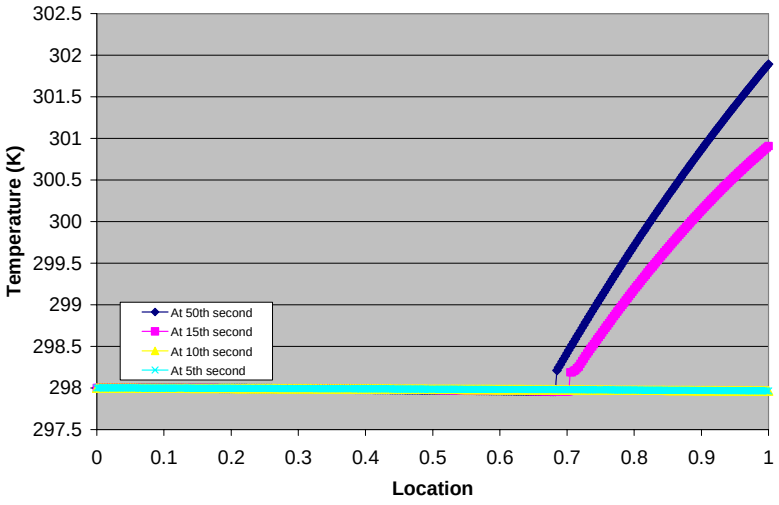

Figure 2: Temperature Profile of an Air to Refrigerant Heat Exchanger

Figure 3 shows the refrigerant pressure distribution along the flow direction in an air to refrigerant heat exchanger. Different curves represent the pressure distribution at different time. The X axis shows the normalized length of the heat exchanger and Y axis is the pressure. From inlet to outlet, it shows that the heat exchanger pressure keeps decreasing. This is because there is pressure drop due to the friction force in the heat exchanger. Since the refrigerant quality keeps increasing along the flow direction and with increasing time in this evaporator case, and more and more vapor is produced at the same location along the flow direction with increasing time, the pressure drop from heat exchanger inlet to outlet also increases accordingly.

Figure 4 shows the mass flow rate distribution along the flow direction in an air to refrigerant heat exchanger. Different curves again represent different mass flow rate distribution at different time. The X axis shows the normalized length of the heat exchanger. Initially, the mass flow rate keeps increasing along the flow direction because the refrigerant is evaporated in the heat exchanger and the refrigerant inventory keeps decreasing. More and more refrigerant is pushed out from the segment. With time advancing, the amount of refrigerant that could be evaporated becomes less and less until steady state is reached and inlet and outlet have the same mass flow rate.

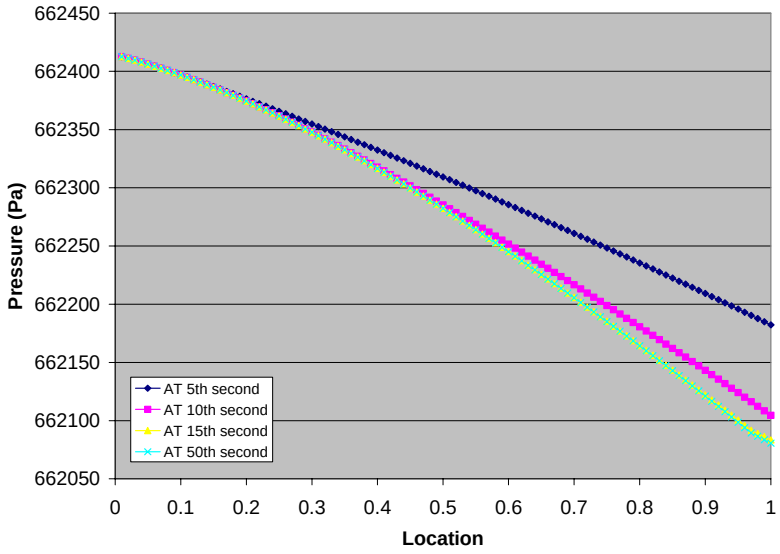

Figure 3: Pressure Distribution of an Air to Refrigerant Heat Exchanger

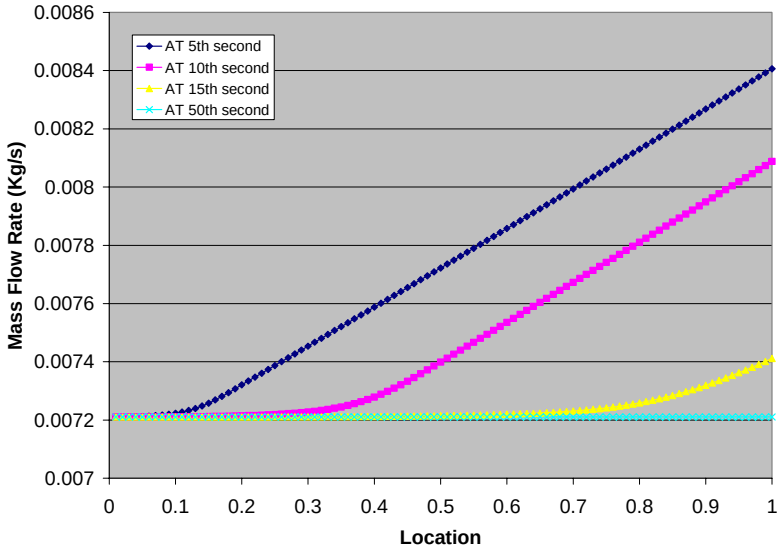

Figure 4: Mass Flow Rate Profile in an Air to Refrigerant Heat Exchanger

3.2 Validation with a Steady State Simulation Tool

Although experimental data have not yet been used to validate these simulation results directly, a comparison between this model and the steady state simulation tool has been conducted for steady state conditions. This steady state simulation tool (Jiang et al., 2002, 2006) has been validated and used extensively and shows a very good agreement with measured data. Hence, an agreement should be expected once the generic dynamic model reaches steady state.

In the first case, the refrigerant inlet temperature is 280.15 Kelvin, the inlet quality is 0.2, and the air inlet temperature is 290 K. The same tube geometry is used in both the steady state software and this generic heat exchanger model. At steady state conditions, the following results were obtained:

	Steady State Simulation Tool	**Generic HX**
Heat Load	121.83 W	122.74 W
Pressure Drop	95.96 Pa	95.01 Pa
Outlet Quality	0.36	0.3583

Table1: Result Comparison for an Evaporator Simulation

In the second case, the refrigerant inlet temperature is 325 Kelvin, the inlet quality is 1.0 and the air inlet temperature is 300 Kelvin. The same tube geometry is also used in both models. At steady state conditions, the obtained results are shown below:

	Steady State Simulation Tool	**Generic HX**
Heat Load	247.96 W	249.398 W
Pressure Drop	1666.74 Pa	1697.7 Pa
Outlet Quality	0.45	0.436

Table 2: Result Comparison for a Condenser Simulation

For both cases, the results from both models are very close at steady state conditions.

4. Summary

A transient generic heat exchanger model was developed. It uses the Golden Section search method to search for the transition point between phase regimes in the segments. The tracking of any flow regime boundary within a segment and sub-dividing the segment as needed is expected to provide still accurate results even when a large control volume is chosen compared with a traditional finite volume method. The combined finite volume and moving boundary approach provides the flexibility for users to trade-off accuracy and execution speed, because in the control volume which exhibits a flow regime change, the moving boundary approach is still implemented. Different correlations are applied to calculate local heat transfer coefficient, pressure drop and void fraction factor for different working fluids and different geometries. Further work still needs be done in order to validate the model with transient experimental data. Also correlations suitability needs be explored in order to confidently use them in transient simulations.

NOMENCLATURE

Symbol

A area, m^2 c_p specific heat, J/kg.K

D	diameter, m	f	fraction factor
g	acceleration due to gravity, m/s^2	G	mass flux, $kg/m^2.s$
h	enthalpy, J/kg	HTC	heat transfer coefficient
L	length, m	m	mass, kg
$\dot{m}$	mass flow rate, kg/s	P	pressure, Pa
$\dot{Q}$	heat flux, W/m^2	T	temperature
t	time		
u	internal energy, J/kg	**Subscripts**	
V	velocity, m/s	cs	cross section
W	work, W	h	hydraulic
Z	spatial distance	l	liquid
Δ	delta	g	gas
ρ	density		
ε	rate of convergence	**Superscripts**	
		°	pervious value

Reference

MacArthur, J.W., 1984, Analytical Representation of the Transient Energy Interactions in Vapor Compression Heat Pumps, ASHRAE Paper, HT-84-19, Vol. 90, Part 1.

MacArthur, J.W., Grald, E.W., 1987, Prediction of Cyclic Heat Pump Performance with a Fully Distributed Model and a Comparison with Experimental Data, ASHRAE Transactions Vol. 93, Part 2.

MacArthur, J.W., Grald, E.W., 1989, Unsteady Compressible Two Phase Flow Model for Predicting Cyclic Heat Pump Performance and a Comparison With Experimental Data, International Journal of Refrigeration, Vol. 12, pp. 29-41.

Williatzen, M., Petit, N.B.O.L., Ploug-Sorensen, L., 1998, A General Dynamic Simulation Model for Evaporators and Condenser in Refrigeration: Part 1 – Moving Boundary Formulation of Two Phase Flows with Heat Exchanger, International Journal of Refrigeration, Vol. 21, No. 5, pp. 398-403.

Jiang, H., Aute,V., Radermacher, R. , 2006 CoilDesigner: a general-purpose simulation and design tool for air-to-refrigerant heat exchangers, International Journal of Refrigeration, Vol.22, pp 601-610

Bendapudi, S., Braun, J.E., Groll, E.A., 2004 Dynamic Modeling of Shell and Tube Heat-exchangers: Moving Boundary vs. Finite Volume, International Refrigeration and Air Conditioning Conference at Purdue

Jakobsen A., Antonius, J., HØgaard Knudsen, H.J., 1999, "Experimental Evaluation of the Use of Homogeneous and Slip-Flow Two-Phase Dynamic Models in Evaporator Modeling, Proc. 20th International Congress of Refrigeration (Sydney), Paper No. 135.

Jiang, H., Aute,V., Radermacher, R., 2002 "A USER-FRIENDLY SIMULATION AND OPTIMIZATION TOOL FOR DESIGN OF COILS", International Refrigeration and Air Conditioning Conference at Purdue

Rossi, T.M and Braun, J.E., 1999, A Real-time Transient Model for Air Conditioners, Proc. 20th International Congress of Refrigeration, Sydney, Paper No. 743.

CEEEprops, CEEE Thermal Properties Calculation Program of Refrigerant and Refrigerant Mixture, 2005.

RefProp v.7.0, NIST Thermodynamic Properties of Refrigerants and Refrigerant Mixtures Database, Standard Reference Data Program, USA; 2002.