International Refrigeration and Air Conditioning Conference

School of Mechanical Engineering

2022

Thermal systems Oriented Two-Phase Heat Exchanger Models. Focus on Numerical Robustness and Calibration

Nicolas Ablanque

Santiago Torras

Carles Oliet

Joaquim Rigola

Thermal Systems Oriented Two-Phase Heat Exchanger Models. Focus on Numerical Robustness.

NICOLAS ABLANQUE[1*], SANTIAGO TORRAS[1], CARLES OLIET[1], JOAQUIM RIGOLA[1]

[1] Universitat Politècnica de Catalunya - Barcelona Tech (UPC), Heat and Mass Transfer Technological Center (CTTC)
Terrassa, Barcelona, Spain, cttc@cttc.upc.edu

* Corresponding Author

ABSTRACT

The simulation of complex refrigeration architectures (that usually include vapor compression cycles) provides useful information for design, study and optimization purposes. Such arrangements may include several interconnected systems and a large variety and quantity of components. All components must meet two crucial requirements, namely, low CPU resolution time and high numerical robustness, in order to achieve relatively fast simulations and to prevent solver resolution issues at the architecture level. Among the usual components present, heat exchangers are the most challenging to address considering both the phenomenological and the numerical point of views.

A generic heat exchanger model oriented for flexible purposes and meeting the aforementioned requirements has been developed under the Modelica programming language. The model can handle both single-phase and two-phase flows based on a simplified approach that considers three different zones for the refrigerant phase. Its numerical robustness has been extensively tested focusing on different boundary characteristics (definition, values, and signal types) and on demanding operating conditions (null mass flow rate, reversed flow, and reversed heat direction). This document presents the main characteristics of the model and a complete assessment of its numerical behaviour in terms of robustness and CPU time consumption.

1. INTRODUCTION

Refrigeration systems based on vapor compression cycle units are widely used to address many domestic, commercial and industrial cooling requirements due to their efficient performance. For instance, applications such as buildings air-conditioning networks, large commercial refrigerated facilities, industrial processing plants, and aircraft environmental systems, usually consist of complex arrangements of different thermal cooling units powered by electrical components and regulated with control actuators.

The simulations of such complex and large architectures are very useful tools for engineering design optimization procedures where a large amount of operational, geometric, and configuration conditions are to be tested. The successful simulation heavily relies on the selection of appropriate models for each constituent component. Due to the large number of components and systems involved, all component models must meet two crucial characteristics: significantly low resolution time (to prevent simulation bottlenecks which could easily derive in unacceptable CPU times) and high numerical robustness (to correctly respond to on-design and also off-design operational circumstances which are often attained during particular transients and/or numerical resolution procedures).

Heat exchangers present in vapor compression cycles are arguably the most challenging components to model (Rasmussen, 2012) due to their phenomenological complexity (phase changes experienced by the refrigerant) and their linking role (a single heat exchanger is usually shared by two different fluid systems). The present work is an attempt to develop a model for two-phase flow heat exchangers that appropriately fulfills the two demanding requirements listed above.

Two different physics-based approaches have been commonly considered for modelling heat exchangers: the fixed volume (FV) (e.g. Morales-Ruiz *et al.*, 2009) and the moving boundary (MB) (e.g. Jensen and Tummescheit, 2002). In the FV method the flow domain discretization consists of an arbitrary number of equally sized and relatively small control volumes while the discretization of the MB method consists of three dynamically resizable control volumes which correspond to each refrigerant phase, namely, vapour, two-phase, and liquid. The MB method can be upgraded into the switching moving boundary method (SMB) which includes the capacity to activate or deactivate any of the three control volumes (e.g. Qiao *et al.*, 2016).

A small number of studies focused on comparing the performance of these two methods have been reported in the open literature such as Bendapudi *et al.* (2008), Rasmussen and Shenoy (2012) and Pangborn *et al.* (2015). Although authors report little differences in terms of accuracy between the two methods, the SMB has proven to require lower CPU time consumption while the FV has proven to be more robust, more detailed, and easier to implement. The present work is an attempt to develop a physics-based method, easy to implement, fast as the SMB method, and numerically robust as the FV method.

The model is inspired in the Switching Moving Boundary approach, where three differentiated zones are considered to account for the three possible refrigerant states but considering several simplifying assumptions and hypotheses. The heat exchanger model combines an steady-state approach used for both fluid flows with a transient approach used for the solid parts. The refrigerant flow is calculated from its inlet state in a sequential manner to determine the distribution of phase zones.

The model has been implemented into the Dymola/Modelica platform and has been used to simulate a real condenser. The model has been subjected to a comprehensive robustness test including: off-design flow conditions (e.g. flow reversals, null mass flow rates), combination of boundary characteristics, and relevant numerical aspects. A full description of the main model aspects in terms of time consumption and numerical robustness is detailed in the present document.

2. MODEL DESCRIPTION

This section is devoted to introduce the most relevant aspects of the numerical model developed in the present work.

2.1 Global structure

Complex cooling systems include several different heat exchanger types in terms of geometries, type of working fluids (liquids, gases, two-phase refrigerants), and phenomenological purposes such as condensation or evaporation. The heat exchanger model structure that has been implemented within this work is focused on addressing all the aforementioned considerations in a flexible way. The model layout is based on two specific sub-components for the calculation of fluid flows thermally linked through an additional sub-component for the calculation of the solid parts. The structure can be easily adapted to tackle different combinations of fluids, geometries and phenomena. The example shown in Figure 1, where an air-to-refrigerant condenser is represented, will be considered throughout the whole present document.

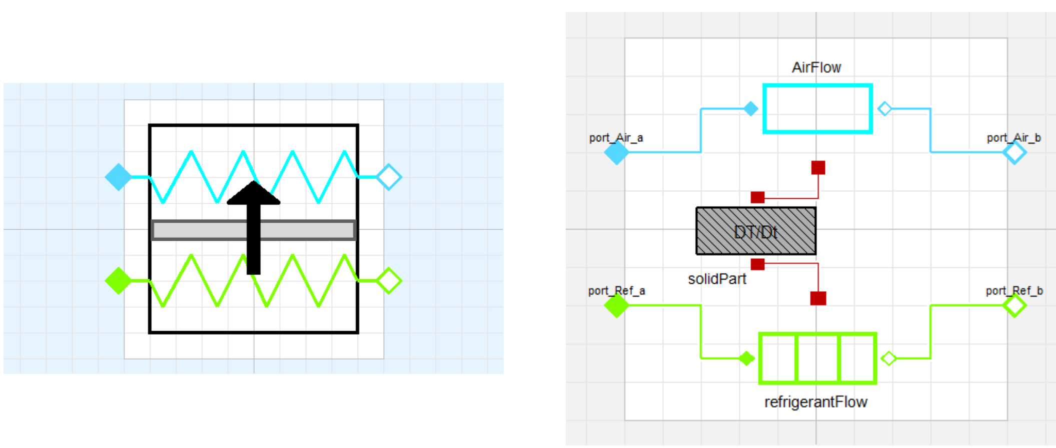

Figure 1: Condenser model (refrigerant vs. air): icon (left) and internal structure (right).

Three different sub-component models, have been developed for the thermal and fluid-dynamic calculation of fluid flows depending on the fluid considered, namely, air, liquid or refrigerant (the latter can include different phases). The sub-component model for the solid parts is characterized by a unique temperature value that changes dynamically but has proven to be effective regarding the model flexibility, CPU time and numerical robustness.

2.2 Single-phase flows

The calculation of single-phase flows is based on a steady-state approach. The pressure loss is calculated based on a traditional approach where the mass flow rate is proportional to the pressure drop where the coefficients K and α are previously determined from reference data:

$$\dot{m} = K \Delta P^{\alpha} \tag{1}$$

The energy conservation equation between the fluid and the solid interface is calculated considering a single zone. The method implemented is based on an $\varepsilon - NTU$ approach in order to optimize the calculation speed and also to prevent the involved temperatures from reaching unreal values. The sensible heat between the fluid and the solid is calculated from their temperatures as follows:

$$\dot{Q}_{fluid} = \dot{Q}_s = \varepsilon C (T_{solid} - T_{fluid,in}) \tag{2}$$

In the case of moist air the heat transfer due to water condensation should be also considered. The total heat transferred by the fluid is then calculated from both its sensible (Equation 2) and latent terms (Equation 3):

$$\dot{Q}_l = \dot{m}_{da} \Delta h_{fg} (W_{out} - W_{in}) \tag{3}$$

$$\dot{Q}_{fluid} = \dot{Q}_s + \dot{Q}_l \tag{4}$$

2.3 Two-phase flows

The calculation of two-phase flows is also based on a steady-state approach. In this case the flow model must deal with complicated phenomena as phase change occurs during evaporation or condensation processes. The method implemented in the model distinguishes three different zones, namely, gas, two-phase and liquid. Each of these zones can be active or not depending on the six different operating modes. For instance, the flow discretization and the corresponding operating modes of the studied condenser are detailed in Figure 2 (a similar approach is considered for evaporation cases).

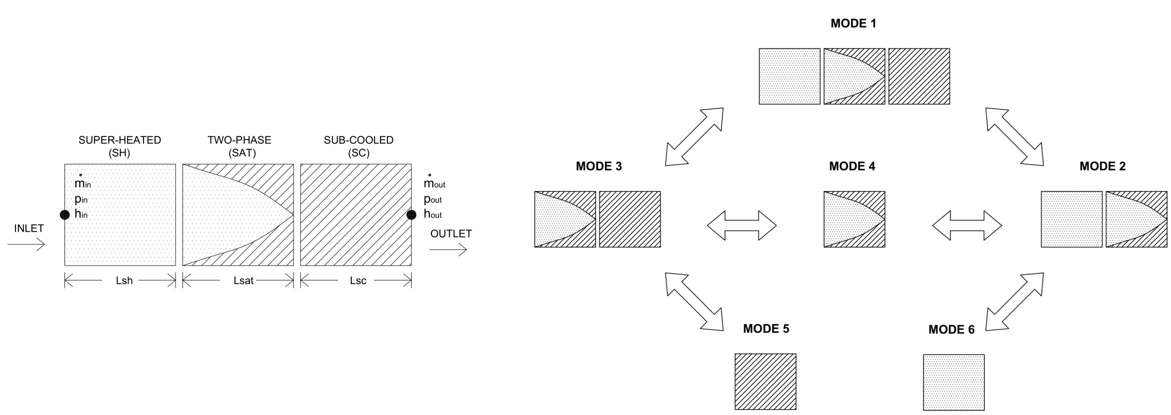

Figure 2: Refrigerant flow condensing: internal structure (left) and operating modes (right).

The pressure drop is considered throughout the whole flow domain and it is calculated from the same approach as in Equation 1 but in this case the coefficients K and α are calculated from a performance map (generated with reference data) where the influence of the inlet pressure, the operating mode and the mean quality value (only for operating modes with two-phase zone) are taken into account.

The energy conservation equation is applied between the fluid and the solid interface. The mean pressure value derived from the pressure drop calculations is used to approximate the specific enthalpy liquid and gas saturation values. The resolution is conducted isobarically from both the inlet specific enthalpy and the mass flow rate values in a sequential manner. For instance, if the inlet condition of a condenser is super-heated, the first zone to be calculated corresponds to the super-heated zone. If the predicted heat surpasses the maximum heat allowed for this particular zone (i.e. heat obtained from the inlet condition and the vapour saturation limit) the corresponding heat for this zone will be that maximum one and the calculation will proceed with the following zone (i.e. two-phase zone). The process keeps going until the heat does not surpass the corresponding maximum heat of the zone being calculated. For single-phase zones the heat is calculated from Equation 2 to prevent unreal predictions while the heat corresponding to two-phase zones is calculated from a standard approach:

$$\dot{Q}_{fluid} = \alpha(T_{solid} - T_{fluid,sat})A \tag{5}$$

2.4 Solid part

The solid part represents the thermal link between the two flows and considers a unique temperature. Its calculation includes dynamic terms:

$$MC_p\frac{dT}{dt} - Q_{fluid,1} - Q_{fluid,2} = 0 \tag{6}$$

2.5 Resolution

The complete resolution is carried out by means of the default differential/algebraic system solver of Dymola. The heat exchanger model combines the steady-state approach used for both flows with the dynamic approach considered for the solid part. Therefore, the model overall thermal response is dynamic as it includes not only the thermal inertia of the solid part but also the possibility to apply artificial relaxations to the energy conservation equation of both flows (i.e. to further overcome the negative impact of the absence of dynamic terms). The pressure drop equation is not only used to calculate the mass flow rate but also to approximate the phase saturation limits needed for the energy conservation equation.

3. MODEL NUMERICAL ROBUSTNESS

This section describes the comprehensive study conducted to assess the model numerical robustness and its CPU time consumption. A full set of tests has been implemented in order to ensure the model robustness regarding all the requirements expected. The results shown correspond to the model of a condenser included in the vapor compression system of an environmental control system designed for the new generation commercial aircrafts.

3.1 Requirements

Heat exchanger models must satisfy many numerical requirements for their successful use when simulating complex thermal architectures where a large number of components and systems are interacting and extremely low simulation time is required for real time control purposes. The following list summarizes the expected model capacities:

- Ability to handle changes of the expected heat flow direction (these conditions could appear at particular transient moments or for specific system operation conditions).
- Both fluid flows must correctly handle null mass flow rate conditions as well as reversed flows. The numerical response must be robust in such possible cases to prevent the system solver from stalling.
- The model numerical robustness must be independent to any particular setup parameter (e.g. time step). No specific numerical tuning must be used for any particular case.
- Robustness to any possible combination of boundary conditions (models could have the pressures defined at both ends or alternatively they can have the pressure defined at one end and the mass flow rate at the other) and signals type (constant, ramps, sines...).
- CPU time must be low enough to prevent any bottleneck from happening at the architecture level simulation.

3.2 Initialization

The initialization process of dynamic solvers could often present numerical issues due to the absence of previous known conditions and the difficulty to determine adequate initial values. The heat exchanger model response to initialization must be comprehensively evaluated. Therefore, the model has been subjected to a large matrix with combinations of boundary condition values covering the complete possible ranges of operation. The test scheme and the considered parameter ranges are presented in Figure 3 and Table 1 (parameters are presented in normalized form for confidentiality aspects). The language used herein, Modelica, follows an acausal equation-based approach (the model inputs and outputs are not defined as in traditional assignation languages), therefore, different boundary configurations must be considered for tests.

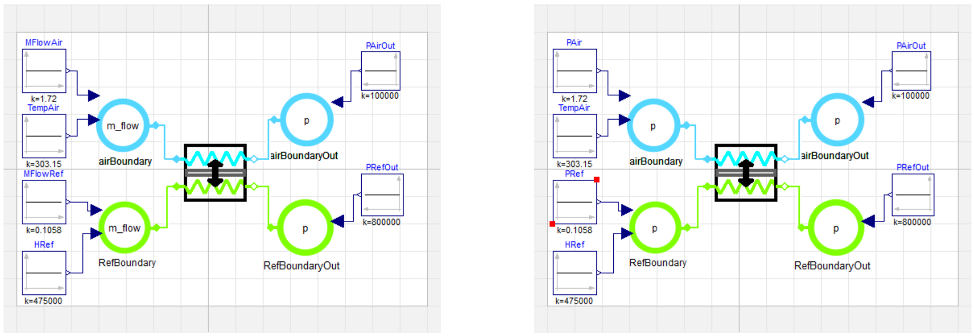

Figure 3: Layout for initialization tests. Boundaries configurations considered: pressure-mass flow rate (left) and pressure-pressure (right).

Table 1: Ranges applied to condenser initialization tests.

Parameter	**values**
Normalized* air mass flow rate	0.4/1.0/1.8
Normalized* refrigerant mass flow rate	0.5/1.0/1.5
Normalized* refrigerant pressure	0.5/1.0
Refrigerant inlet enthalpy	3 values (to ensure inlet conditions for each phase zone)
Air inlet temperatures	4 values (based on saturation temperature)
Boundary conditions	2 configurations (see Figure 3)

*Normalized values are derived from ref. values ($\theta_{norm} = \theta/\theta_{ref}$)

The resulting matrix for the condenser initialization tests consists of 432 different runs in total. The numerical default parameters considered are: simulation stop time of 2000 seconds and number of intervals 2000. The model performance was very good as only one case could be considered as a failure because it needed high CPU time to converge (183 seconds). For the rest of the cases the mean CPU time for convergence was 0.24 seconds. The characteristics of the processor used are: Intel(R) Core(TM) i5-2450M CPU @ 2.60GHz 2.60 GHz.

3.3 Mode switching

The two-phase flow approach implemented into the model considers different operating modes as have been detailed in Figure 2. The model must ensure its robustness when switching from one mode to another during simulations. Therefore the model has been tested for switching conditions considering each particular operating mode as the starting condition. The test scheme and some illustrative results are presented in Figure 4.

The test consists of 6 baseline runs (one for each particular operation mode). The stop time for all simulations is 1000 seconds while three different number of intervals have been considered, namely, 500, 1000 and 2000 which corresponds to time steps of 2, 1 and 0.5 seconds, respectively. These time steps must be respected by the solver although smaller time steps are dynamically used by the solver if needed. In addition two different boundary layout configurations were considered so that the total number of runs was 36. All the 36 simulated cases converge smoothly in approximately less than 1 second.

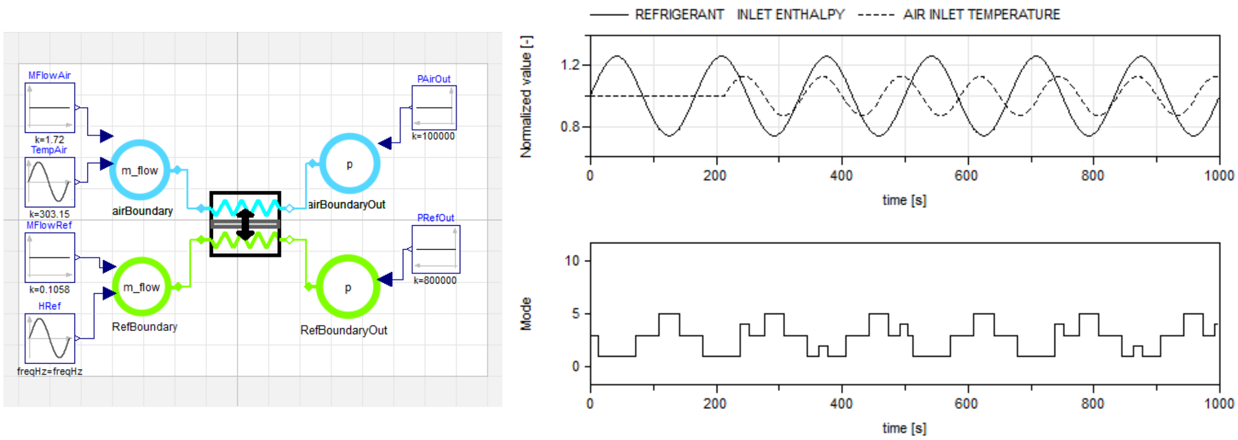

Figure 4: Mode switching: test scheme (left) and illustrative results (right).

3.4 Signals

Another important numerical consideration for the model is to ensure its appropriate transient response when experiencing changes on the boundary values. This aspect is crucial for components used within systems as they are linked to other components, and therefore, exposed to changing values during transient simulations and during solver iterations. In order to ensure the model robustness in this particular sense, a complete set of tests considering two different type of signals (ramps and sines) has been conducted. The test schemes for ramp and sine signals are shown in Figures 5 and 6, respectively.

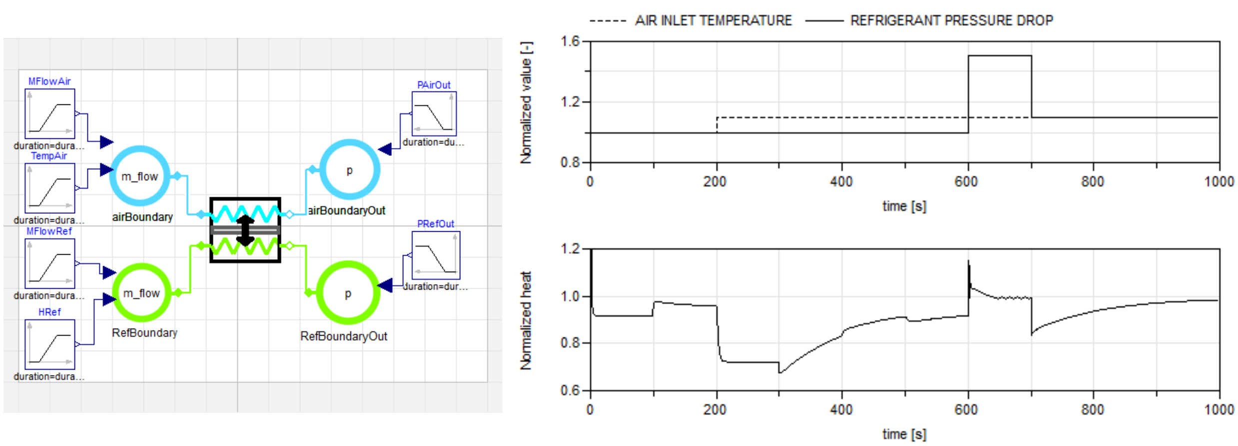

Figure 5: Ramp signals: test scheme (left) and illustrative results (right).

The test consists of 36 runs for each type of signal considered, namely, ramps and sines. Tests are based on the same 6 baseline runs used in the switching mode test in order to include different simulation setups and boundary layout configurations. For the ramp signals tests (Figure 5) each run is subjected to a combination of ramps applied in sequence to all its boundary values throughout the whole simulation (steep slopes are used). For the sine signals tests (Figure 6) frequencies up to 0.1 Hz are used. All 36 runs with ramp signals are calculated in less than 1 seconds, while all 36 runs with sine signals are calculated in less than 2 seconds (except for four cases that are calculated in up to 8 seconds).

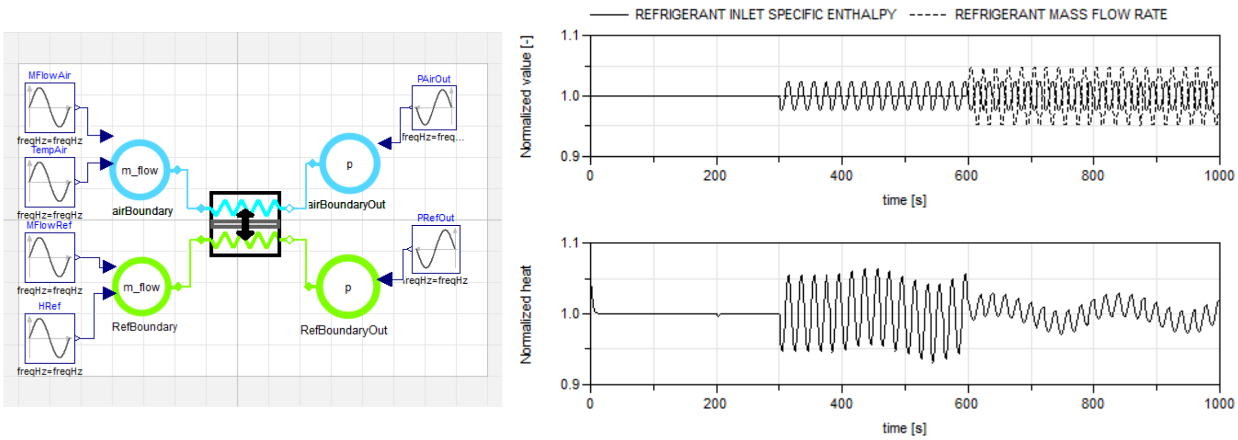

Figure 6: Sines signals: test scheme (left) and illustrative results (right).

3.5 Reversed heat

The model must address changes of the heat direction that could occur at some specific system operation conditions or solver iterating process. For instance, in the studied condenser the heat could eventually flow from the secondary fluid toward the refrigerant. To ensure the model robustness a set of tests where heat direction is forced to change has been implemented. The runs used to evaluate the reversed heat capacity were conducted by modifying the inlet temperature of the air by means of a ramp signal up to a value higher than the refrigerant inlet temperature so that the condenser heat flow direction is changed. The test scheme and a particular illustrative results are presented in Figure 7.

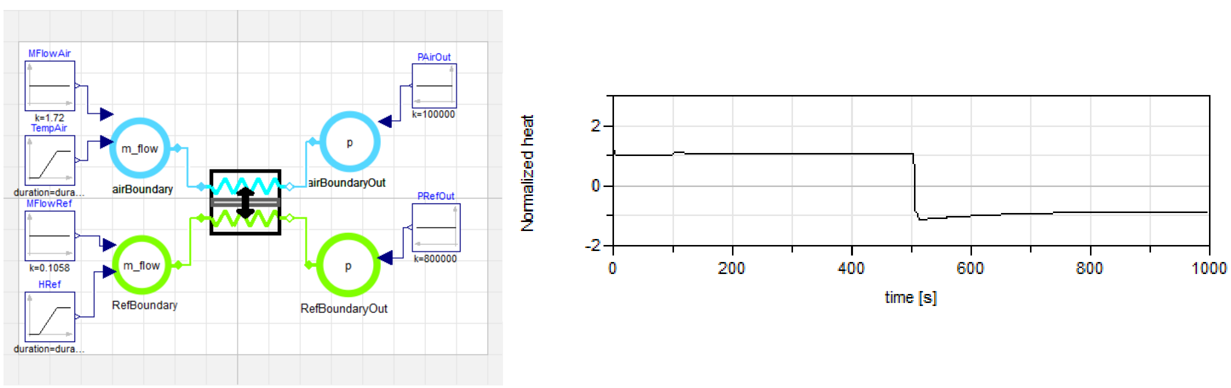

Figure 7: Reversed heat: test scheme (left) and illustrative results (right).

The test consists of 36 runs (they are based on the same 6 baseline runs used in the switching mode test in order to include different simulation setups and boundary layout configurations). All the 36 cases converge smoothly in less than 0.3 seconds.

3.6 Reversed and null mass flow rate

The aim of this final test is to address the model capacity to handle reversed and null mass flow rates on both fluids. This aspect is crucial as these particular conditions could happen during the system start-up, shut-down or any other eventuality. The test scheme and some illustrative results are presented in Figure 8.

This test consists of 72 runs (they are based on the same 6 baseline runs used in the switching mode test in order to include different simulation setups and boundary layout configurations). Each particular run consists of a transient case where the refrigerant is operating at a particular mode and one of the fluids experiences both flow direction changes and null mass flow rate at different moments. The results of a specific run are depicted in Figure 8 to provide a better insight of the runs characteristics. In this particular case flow direction changes are applied to the refrigerant flow.

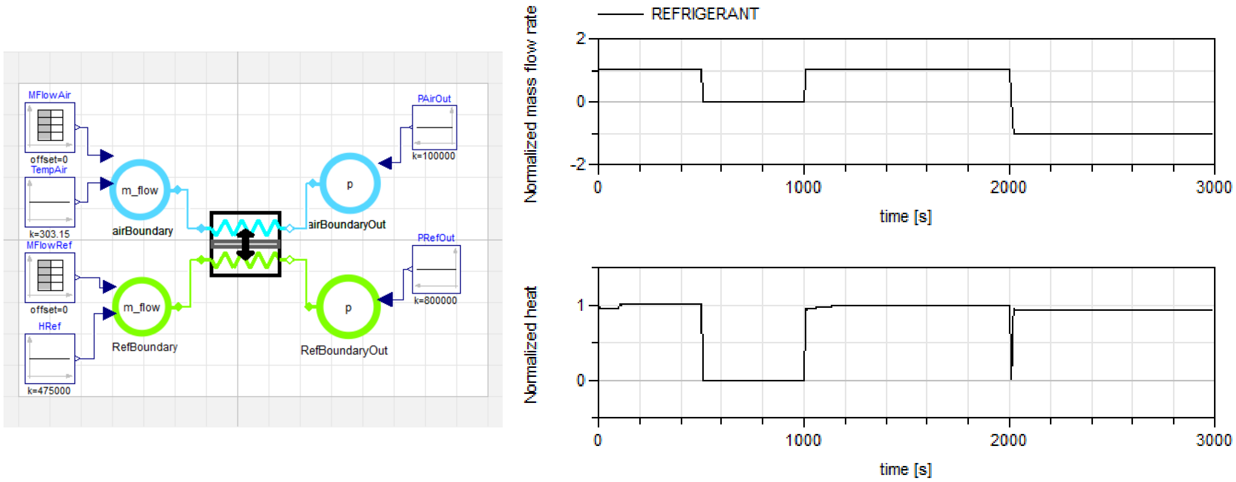

Figure 8: Reversed and null mass flow rate: test scheme (left) and illustrative results (right).

All 72 runs converge without any particular numerical issue. Most of the cases are calculated in less than two seconds except for a very few ones that slightly surpass that threshold.

3.7 Overall assessment

The robustness of the model is evaluated based on the number of run failures. For the particular condenser case that has been studied the results are summarized in Table 2.

Table 2: Robustness tests summarized results.

Test	cases	stop time [s]	intervals	Mean CPU time [s]	failures [-]	failures [%]
initialization	432	2000	2000	0.24	1*	0.2
switching	36	1000	500/1000/2000	0.65	0	0.0
ramps	36	1000	500/1000/2000	0.46	0	0.0
sines	36	1000	500/1000/2000	2.97	0	0.0
heat reversed	36	1000	500/1000/2000	0.26	0	0.0
flow (reversed/null)	72	3000	1500/3000/6000	0.82	0	0.0
Total	**648**				**1**	**0.15**

*High CPU time needed (run not considered for mean CPU time calculation)

It can be seen that all the cases tested converge to a solution without any particular issue (only one case is considered as a failure as it surpasses the expected CPU time). The mean CPU calculation time for all the other tests is significantly low taking into account both the simulation time and the demanding transient characteristics considered for tests. The percentage of run failures is relatively low compared to the number of cases tested, therefore, a good global numerical robustness has been achieved for the condenser model.

4. CONCLUSIONS

- A heat exchanger simulation model specifically aimed to be used within simulations of complex and large refrigeration architectures has been developed.
- The model is based on a combination of steady state and dynamic calculations. Its structure allows to combine different components for the thermal and fluid-dynamic calculations of flows and also to address both condensation and evaporation phenomena.
- The calculation of the refrigerant flow considers three different refrigerant phases which can be activated or deactivated according to the simulation heat level conditions.

- The model has been extensively tested to ensure its numerical robustness at all levels. This aspect is crucial for its confident use within large and complex simulation architectures. The model has proven to be robust regarding many numerical aspects (initialization, numerical setup, boundaries configuration, input signals type) and many off-design operation conditions (reversed heat flow direction, changes on flows directions, and null mass flow rates). All these aspects are crucial to address off-design conditions during specific transients (e.g. start-ups, shut-downs, malfunctions, etc...) but also during solver iterations.
- The model has proven to consume very low CPU time in both standard and off-design conditions. This aspect is critical to prevent bottlenecks from happening during the simulation of large architectures in order to use the model for real time control but also to address the demanding amount of runs needed for design purposes.

NOMENCLATURE

A	heat transfer area	(m^2)
C	heat capacity rate	(W/K)
C_p	specific heat capacity	(J/kgK)
h	specific enthalpy	(kJ/kg)
L	length	(m)
$\dot{m}$	mass flow rate	(kg/s)
M	mass	(kg)
P	pressure	(Pa)
$\dot{Q}$	rate of heat flow	(W)
t	time	(s)
T	temperature	(K)
W	humidity ratio	(kg_v/kg_{da})

Greek symbols

Δh_{fg}	enthalpy of vaporization	(J/kg)
ε	effectiveness	(-)
θ	variable	(-)

Subscript

da	dry air
fluid	fluid
in	inlet
l	latent
norm	normalized
out	outlet
ref	reference
s	sensible
solid	solid

REFERENCES

Bendapudi, S., Braun, J.E., and Groll, E.A., ”A comparison of moving-boundary and finite-volume formulations for transients in centrifugal chillers,” *International Journal of Refrigeration*, Vol.31(8), pp. 1437-1452, 2008.

Jensen, J.M., and Tummescheit, H., ”Moving boundary models for dynamic simulations of two-phase flows,” *In Modelica 2002 Proceedings*, Modelica Association, pp. 234-244, 2002.

Morales-Ruiz, S., Rigola, J., Pérez-Segarra, C.D., and García-Valladares, O., ”Numerical analysis of two-phase flow in condensers and evaporators with special emphasis on single-phase/two-phase transition zones,” *Applied Thermal Engineering*, Vol.29(5-6), pp. 1032-1042, 2009.

Pangborn, H., Alleyne, A.G., and Wu, N., ”A comparison between finite volume and switched moving boundary approaches for dynamic vapor compression system modeling,” *International Journal of Refrigeration*, Vol.53, pp. 101-114, 2015.

Qiao, H., Laughman, C.R., Aute, V., and Radermacher, R., "An advanced switching moving boundary heat exchanger model with pressure drop," *International Journal of Refrigeration*, Vol.65, pp. 154-171, 2016.

Rasmussen, B.P.,"Dynamic modeling for vapor compression systems - Part I: literature review," *HVACR Res.*, Vol.18(5), pp. 934-955, 2012.

Rasmussen, B.P., and Shenoy, B., "Dynamic modeling for vapor compression systems - Part II: simulation tutorial," *HVACR Res.*, Vol.18(5), pp. 956-973, 2012.

ACKNOWLEDGMENT

This project has received funding from the Clean Sky 2 Joint Undertaking (JU) under grant agreement No 886533. The JU receives support from the European Union's Horizon 2020 research and innovation programme and the Clean Sky 2 JU members other than the Union.

Horizon 2020
European Union Funding
for Research & Innovation

Clean Sky 2
JOINT UNDERTAKING

Carles Oliet, as a Serra Húnter lecturer, acknowledges the Catalan Government for the support through this Programme.

Disclaimer: The contents presented in this article reflect only the author's point of view: the authors and Clean Sky JU are not responsible for any use that may be made of the information it contains.