Efficient Numerical Solution of One-Dimensional Governing Equations for Evaporating Flow in a Tube

C.T. Veje, S. Madsen, and M. Willatzen

Mads Clausen Institute, University of Southern Denmark
Alsion 2, DK-6400 Sønderborg, Denmark
veje@mci.sdu.dk, sma@mci.sdu.dk, willatzen@mci.sdu.dk

Keywords: Two-phase flow, evaporation, heat-transfer, numerical methods

Abstract

We present results of the computational stability and efficiency in simulating two-phase flow and heat transfer in a horizontal tube. Two structurally different models of the process are presented and compared for two dynamic computational scenarios; dynamic response for a constant and changing number of flow-zones, respectively. The first model is a simple generic lumped model of the moving boundary type and the second is a fully distributed model in one dimension. The moving boundary model is solved as a set of conventional ordinary differential equations whereas the distributed model is solved using a highly-efficient numerical scheme following Kurganov & Tadmor (2000). The governing equations are formulated in terms of mass flow, enthalpy, pressure, and tube wall temperature based on the continuity-, momentum- and energy equations for the fluid. A wall heat-exchange equation accounts for convective heat exchange between the fluid and the ambient air. The comparison demonstrates a significant difference in the results obtained using the two models, especially in the case where the number of fluid zones is changing. This emphasizes the importance of using either more advanced moving boundary models or fully distributed models in the simulation of systems comprising two-phase flows. In contrast to conventional belief we find that the finite difference model is indeed both stable to discontinuities in boundary conditions; changing number of fluid zones; high frequency perturbations and also at the same time numerically fast enough for real-time simulation and control.

Introduction

Modeling and simulation of two-phase phenomena is important in the development and optimization of a wide range of industrial applications and products. Among others these include essential components of, e.g., the HVAC industry, power plants and the growing efforts within energy storage technologies. As requirements for model accuracy of such systems increase there is a growing if not compelling need for detail in the modeling of the basic phenomena. Such increased detail usually comes at the expense of computation time. The central question is then if a reasonable compromise between accuracy (detail) and computational speed can be met. The present work will address this.

A main component in the aforementioned systems is the evaporative heat exchanger subject to two-phase flow and heat transfer. Traditionally, the lumped component models for evaporators are solved using NTU-\(\epsilon\) - methods. The need to investigate for example evaporator control strategies demands realistic modeling of parameters such as the superheat which has given rise to a widespread use of moving-boundary models, see e.g., He et. al. (1998); Willatzen et. al. (1998); Zhang & Zhang (2006); McKinley & Alleyne (2008). These models show an advantage in demonstrating dynamical behavior of the component in relation to capacities, temperature, and pressure levels although formulated as lumped models. This has enabled evaluation of system performance and analysis with respect to varying conditions and different control strategies. The drawback is loss of detail in the modeling and the handling of fluid-zone switching which may result in numerical instability and increased computation time.

In a general effort to investigate the dynamic behavior of evaporative flow systems several works have been carried out on fully distributed models solving the governing equations, see e.g., Jia et. al. (1995, 1999); Valladares et. al. (2004); Madsen et. al. (2009). In this paper, we present models and comparative simulation results for two implementations of a horizontal tube with an evaporating fluid; a conventional moving boundary
and a distributed model. Two different cases of dynamic response are investigated.

In the following we will give a brief review of the model equations. First, a set of ordinary differential equations is developed based on a conventional moving boundary formalism following the works of He et. al. (1998) and secondly a full set of governing equations is developed following Madsen et. al. (2009). In both cases we make the following assumptions:

- constant tube cross section,
- no gravity effects,
- one-dimensional flow,
- two-phase inlet condition,
- no axial heat conduction in the fluid,
- no viscous heating or viscous pressure loss
- fixed temperature heat source in the air-side,
- constant air-side heat transfer coefficient and area.

The working medium is isobutane (R600a) and thermodynamic quantities such as $T(h, P)$ and $\rho(h, P)$, saturation values and fluid properties are evaluated using Engineering Equation Solver, EES (Klein (2009)) or Refrigerant Equations v. 6.1 (Skovrup (2001)) by generating a lookup table for fast access to these relations.

1 Moving Boundary Model

The system under consideration is shown in figure 1. It constitutes a straight horizontal tube of length $L$ and internal diameter $D_i$ with fluid flowing from left to right while it exchange heat with the tube wall.

A schematic view of the simple moving boundary model is also indicated in figure 1. We treat the tube wall as a single lumped block. On the fluid side the model constitutes a two-phase part $L_{TP}$, and a single phase gas part; the superheat region $L_{SH}$. Clearly $L = L_{TP} + L_{SH}$ and the position of the dryout point $L_{TP}$ is determined by the balance between the inlet mass flow, outlet volume flow, the pressure and heat transfer rates through the tube wall. The air to wall heat flow is:

$$Q_W = f_{fin}\pi D_o \alpha_o (T_A - T_W) \gamma,$$

where $\alpha_o, D_o, f_{fin}, T_A, T_W$ are the heat-transfer coefficient between evaporator wall and ambient, outer tube diameter, fin factor, ambient temperature, and wall temperature, respectively. The outer tube area is reduced by a factor $\gamma = L_{TP}/L$ (as a correction) due to reduced heat transfer over the superheat region given by the liquid volume $V_l = \frac{3}{4} D_i^2 L_{TP} (1 - \alpha)$, total volume $V$ and void fraction $\alpha$,

$$\gamma = \frac{V_l}{V(1 - \alpha)}.$$

The wall energy equation is given by:

$$\gamma M_W C_W \frac{dT_W}{dt} = (Q_W - Q_f),$$

where $M_W C_W$ is the thermal capacity of the wall and $Q_f$ is the heat flow into the tube fluid

$$Q_f = Q_{evap} + Q_{sh},$$

where

$$Q_{evap} = \pi D_i L \alpha_i (T_W - T_e) \gamma,$$

$T_e$ is the evaporation temperature and $\alpha_i$ is the internal heat transfer coefficient which is assumed constant and estimated based on the correlation used in the finite difference model described later. $Q_{sh}$ is evaluated as

$$Q_{sh} = \dot{m}_o (h_o - h_g),$$

where $h_g$ is the dewpoint enthalpy and $h_o, \dot{m}_o$ is the tube outlet enthalpy and mass flow respectively. The outlet
temperature is evaluated using a functional form for the superheat similar to He et al. (1998). Finally the change in liquid mass is given by
\[
\frac{dM_l}{dt} = (1 - x_i)\dot{m}_i - \dot{m}_{evap}, \quad (7)
\]
where \(x_i\) and \(\dot{m}_i\) is the inlet quality and mass flow, respectively, and \(\dot{m}_{evap} = Q_{evap}/\Delta h_{lg}\) is the evaporation flow with \(\Delta h_{lg}\) being the evaporation enthalpy at the relevant pressure. The outlet mass flow is simply assumed to follow the inlet mass flow and the change in liquid content and thus:
\[
\dot{m}_o = \dot{m}_i - \frac{dM_l}{dt}, \quad (8)
\]
An important variable in the model is the determination of the pressure level which results as a combination of the in and outlet flow conditions and the overall heat transfer. The pressure is calculated as
\[
\frac{dP}{dt} = \frac{\partial P}{\partial \rho}^{\rho-O} \frac{\partial \rho}{\partial t}, \quad (9)
\]
where \(\frac{\partial P}{\partial \rho}\) is evaluated through the continuity equation for the total gas volume of the tube \(V(L_{SH} + \alpha L_{TP})/L\). Thus, the pressure dynamics is assumed to be determined only by the balance of inlet and outlet mass flow and not by variations in temperature since temperatures vary slowly compared to mass flows.

The moving boundary model is solved with appropriate boundary conditions using a conventional ODE-solver. For an evaporator tube of length \(L\) we have an inlet mass flow \(\dot{m}_i\) at \(z = 0\) and an outlet volume flow \(V = \dot{m}_o/\rho_o\) at \(z = L\). Pipe wall ends (\(z = 0, L\)) are insulated and the fluid enthalpy is specified at the inlet \(z = 0\).

2 Distributed Model

For the distributed model we assume that variations in physical properties basically take place along one coordinate direction (the length direction of the evaporator). Then, following Madsen et al. (2009) we have
\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho w)}{\partial z} = 0, \quad (10)
\]
\[
\frac{\partial (\rho w)}{\partial t} + \frac{\partial (\rho w^2)}{\partial z} = -\frac{\partial P}{\partial z}, \quad (11)
\]
\[
\rho \left( \frac{\partial h}{\partial t} + w \frac{\partial h}{\partial z} \right) - \left( \frac{\partial P}{\partial t} + w \frac{\partial P}{\partial z} \right) = \frac{4}{D_i} \alpha_i(T_W - T), \quad (12)
\]
where \(w\) is the one-dimensional velocity along the evaporator length coordinate.

These equations are recast in terms of new variables (pressure, mass flow, and enthalpy) \(P, \dot{m} = \rho A w,\) and \(h\)
\[
\frac{\partial \rho}{\partial P}, \frac{\partial P}{\partial t} + \frac{\partial h}{\partial t} + \frac{1}{\rho A} \frac{\partial \dot{m}}{\partial z} = 0, \quad (13)
\]
\[
\frac{1}{\rho A} \frac{\partial \dot{m}}{\partial t} + \frac{\partial (\rho w^2)}{\partial z} = -\frac{\partial P}{\partial z}, \quad (14)
\]
\[
\rho \left( \frac{\partial h}{\partial t} + \frac{\dot{m}}{\rho A} \frac{\partial h}{\partial z} \right) - \left( \frac{\partial P}{\partial t} + \frac{\dot{m}}{\rho A} \frac{\partial P}{\partial z} \right) = \frac{4}{D_i} \alpha_i(T_W - T), \quad (15)
\]
keeping in mind that \(\rho \equiv \rho(P, h), T \equiv T(P, h)\).

The derivative of \(\rho\) with respect to \(P\) equals the inverse square root of the speed of sound and the latter quantity, in the case of two-phase flow, is derived using the model of Nguyen et al. (1981). \(\frac{\partial \rho}{\partial P}\) is evaluated by numerical differentiation. To close the system the energy equation for the wall is given by
\[
(C_W \rho_W A_W) \frac{\partial T_W}{\partial t} = \alpha_i \pi D_i (T - T_W)
\]
\[
+ \alpha_{\nu} \pi D_o f_{fin} (T_A - T_W) + \lambda A_W \frac{\partial^2 T_W}{\partial z^2}, \quad (16)
\]
where \(\rho_W, A_W,\) and \(\lambda\) are the wall density, wall cross-sectional area, and wall heat conductivity, respectively.

As in the case of the moving boundary model we use a constant coefficient of heat transfer for the air-side. For the fluid side we use the correlation due to Kandlikar (1991) for the two-phase flow up to a vapor quality of 0.7 and the Dittus-Boelter correlation for the single phase flow regime. A simple linear interpolation is used between the two correlations. A correlation for the viscous pressure loss could have been included but is assumed to be zero for the sake of comparison with the moving boundary model.

The computational domain as indicated in figure 1 is discretized in \(N\) fixed finite elements as opposed to the two variable elements for the moving boundary case. The above formulation is essentially a homogeneous flow model since we have formulated the equation-set in terms of overall density and enthalpy neglecting the separation in gas and liquid phases as is more conventional in modeling two-phase flows Wallis (1969).

The boundary and initial conditions are similar to those for the moving boundary model and the relations are given by:
\[
\dot{m}(0, t) = \dot{m}_i, \quad (17)
\]
\[
h(0, t) = h_i, \quad (18)
\]
\[
\dot{m}(L, t) = \rho(L, t)\dot{V}, \quad (19)
\]
\[
\frac{\partial T_W}{\partial z} \bigg|_{z=0,L} = 0. \quad (20)
\]
A conventional central finite difference method applied to Eqs. (13)-(16) typically shows spurious oscillations which quickly destroy the numerical solution. In the fortunate case of numerical convergence, which is highly unlikely for a dynamic simulation, the computation time is unreasonably long and dynamic control of a real time simulation is not feasible. As demonstrated in Madsen et. al. (2009), the Kurganov-Tadmor (KT) high-resolution scheme can avoid these difficulties and allows possible discontinuities in the solutions while maintaining computational speed and stability. We have applied the KT scheme in the second order semi-discrete form to Eqs. (13)-(15) and implemented it in c++. The heat equation (16) is discretized using straight-forward second order finite differences. Further details can be found in Kurganov & Tadmor (2000) and Madsen et. al. (2009).

Figure 2: Steady state temperature $T(z)$ at the tube outlet for different discretization levels $N$. Inset shows the result for the entire length of the tube.

Figure 2 shows the steady state solution for the temperature at the tube outlet for different values of discretization level $N$. Already for $N = 50$ not much is gained in increasing to $N = 100$ and the relative difference in temperature $T_0$ between $N = 100$ and $N = 400$ is as low as $5 \cdot 10^{-3}$ K. The inset shows the result for the entire length of the tube. Temperature gradients are converged to $10^{-4}$ K/s after 10 seconds and steady state is obtained by solving the system for 100 s. For the remaining part of this work we use $N = 100$. Computation time is $O(N^2)$, but even for $N = 100$ we only use 1.17 s per simulated second on a 2 GHz desktop. In other words, real-time simulation is definitely achievable using a distributed model!

3 Simulation Results

In the following, we present results for the moving boundary (MB) and the finite difference (FD) models.
using the basic parameters, boundary values, and initial data shown in Table 1 and otherwise stated in the text. In figure 3, we show the fluid temperature $T$ as a function of position $z$ for both methods. The results are steady state values after 100 seconds simulation time. By roughly tuning $\alpha_i$ in the MB model we find approximate agreement between the two models. Notice the acceleration pressure-drop for the FD method. The temperature drops slightly until dryout at $z \sim 7$ m whereafter the temperature rises quickly in the single phase gas zone.

To test the dynamic response and numerical stability of the model we next investigate the model response to steps in inlet mass flow. After simulating the system for 100 s to a steady state we increase the inlet mass flow $\dot{m}_i = 7.5 \cdot 10^{-3}$ kg/s by 10% at $t = 100$ s and decreases it again to the initial value at $t = 150$ s. Figure 4 shows simulation results of $\dot{m}_i, \dot{m}_o$ (top), $T_o$ (middle) and $L_{TP}$ (bottom) for both models as function of time. The models behave as expected and are numerically stable. Differences in initial transients for $t < 10$ s reflect differences in initial conditions between the models. For the response of the outlet mass flows reasonable agreement between the models is demonstrated, however, the temperature dynamics is clearly different for the two models. This is to a large extent due to the lumped character of the wall energy equation in the MB model. Since outlet temperatures are typical signals in the control algorithms for such systems this points to the necessity of increasing the detail of the modeling.

The computation times are of the order of one second for the MB method depending on the choice of integrator. For the FD model it is of the order 100 s. For the FD model, we use $N = 100$ and the timestep is determined by the maximum velocity criterion as given by the KT method. This roughly corresponds to a Courant number based on the maximum flow and sound speed and is therefore orders of magnitude smaller than the typical timestep of the MB method.

When the system is perturbed by an instantaneous change in, e.g., inlet mass flow as in figure 5, a pressure wave is formed shuttling back and forth in the evaporator tube. Figure 5 shows a detailed spatio-temporal view of $\dot{m}(z, t)$ near $t = 100$ s when the inlet mass flow is increased as in the case of figure 4. The wave is formed at the inlet $z = 0$ m and moves with the flow towards the outlet. At $t \sim 100.1$ s, the wave reaches the outlet at $z = 10$ m and reflects back into the system. The speed of the wave is not constant over the evaporator tube due to the changing speed of sound in passing mainly the two-phase region. Obviously, this phenomenon cannot be captured by the MB method. However, although these effects may not be important on timescales relevant in the heat transfer process the resolution of these

**Figure 4:** $\dot{m}$, $T$, and $L_{TP}$ as a function of time. Top: outlet mass flow $\dot{m}_o$ response to a step in inlet mass flow $\dot{m}_i$ at time $t = 100$ s. Middle: Outlet fluid temperature. Bottom: Position of dryout point $L_{TP}$. Results are shown for both MB and FD models. Differences in initial transients for $t < 10$ s reflects differences in initial conditions.
in the FD method points to the stability of the model.

For the integration of ODE’s in the MB model the main part of the computation time is spent at events of discontinuity such as when the inlet mass flow changes abruptly or when the number of flow zones changes. A typical situation in superheat control of evaporative flow systems involves the response of the inlet mass flow to external perturbations. Situations may occur where there is a repeated change of fluid zones if the inlet mass flow is modulated. Figure 6 (top) shows this situation. Starting from a similar situation as in figure 4 we now increase the inlet mass flow and modulate it with a higher frequency to an extent that the number of fluid zones changes with each period.

For this highly perturbed situation the simulation results reveal large differences between the models. Figure 6 (top) shows the mass flow responses and the missing dynamic terms in the MB formulation is now clearly reflected in the dynamics of the model. The corresponding results for the temperature and dryout point in figure 6 (middle and bottom plots) reflect the same short-
Figure 7: Space-time dependence of $\dot{m}(z, t)$ as the inlet mass flow $\dot{m}_i$ is modulated with a high frequency and the number of fluid zones is changing. Pressure waves are seen shuttling back and forth through the system.

Through implementation of two numerical models for the solution of the two-phase flow and heat transfer in a horizontal tube we have found significant differences in the results of the two models. Especially in the case where the number of fluid zones changes there are substantial deviations. This stresses the importance of using either more advanced moving boundary models or fully distributed models in the simulation of systems comprising two-phase flows. The FD model implemented here is an explicit formulation of the complete set of governing equations including dynamic effects such as propagation of sound waves and acceleration pressure losses. Despite this level of detail the FD model is found to be numerically stable and sufficiently efficient that it facilitates real-time simulation and feedback control of such systems.

An interesting test of numerical stability and efficiency of the FD model is shown in figure 7 where the spatio-temporal plot of the mass flow in the tube is shown for the case where the modulation frequency of figure 6 is increased to a period of $0.1\,\text{s}$. The dynamic pressure waves are reflected at the tube ends and crosses while the number of fluid zones changes periodically in a similar way as in figure 6. The computation time of the results in figure 7 is $0.58\,\text{s}$ which clearly facilitates real-time simulation and control of such a system even in the case where detailed distributed models are used.

**Conclusions**

Through implementation of two numerical models for the solution of the two-phase flow and heat transfer in a horizontal tube we have found significant differences in the results of the two models. Especially in the case where the number of fluid zones changes there are substantial deviations. This stresses the importance of using either more advanced moving boundary models or fully distributed models in the simulation of systems comprising two-phase flows. The FD model implemented here is an explicit formulation of the complete set of governing equations including dynamic effects such as propagation of sound waves and acceleration pressure losses. Despite this level of detail the FD model is found to be numerically stable and sufficiently efficient that it facilitates real-time simulation and feedback control of such systems.

**References**


