A homogeneous two-phase flow model of an evaporator with internally rifled tubes

Axel Ohrt Johansen\textsuperscript{a,b}, Brian Elmegaard\textsuperscript{b}

\textsuperscript{a}DONG Energy - Thermal Power, Kraftværksvej 53, Dk-7000, Fredericia, Denmark
\textsuperscript{b}Technical University of Denmark, Nils Koppels Allé, Building 403, DK-2800 Kgs. Lyngby, Denmark

Abstract

In this article a numerical model for solving a transient one dimensional compressible homogeneous two phase model is developed. It is based on a homogeneous model for predominantly one-dimensional flows in a vertical pipe element with internal rifles. The homogeneous model is based on the assumption of both hydraulic- and thermal equilibrium. The consequences and aspects will briefly be discussed in that context. The homogeneous flow model consists of three hyperbolic fluid conservation equations; continuity, momentum and energy and the pipe wall is modelled as a one dimensional heat balance equation. The models can be reformulated in the four independent parameters \( p \) (pressure), \( h \) (enthalpy), \( u \) (velocity) and \( T_w \) (wall temperature). Constitutive relations for the thermodynamic properties are limited to water/steam and is given by the IAPWS 97 standard. Wall friction and heat transfer coefficients are based on the Blasius friction model for rifled boiler tubes and the correlation by Jirous respectively. The numerical method for solving the homogeneous fluid equations is presented and the method is based on a fifth order Central WENO scheme, with simplified weight functions. Good convergence rate is established and the model is able to describe the entire evaporation process from sub-cooled water to super-heated steam at the outlet.

1. Introduction/motivation

Along with the liberalization of electricity markets in Northern Europe and Denmark, there is an increasing need to quickly regulate the large central power plants to cover the current supply of electricity and district heating. Much focus has been put in optimizing the individual power plants, so they can meet the requirements to stabilize the power supply and district heat production, caused by the stochastic nature of wind farms. Electricity generation based on wind has primacy in terms of production and the central power plants have to fill the gap between producing and consuming power. In periods of very high wind generation, the central power plants are thus forced to run down into low load and maintain a contingency in case the wind unexpectedly fails to come. In these situations, there may still be a need for district heating production, why we might consider turbine bypass in the steam power plants and directly produce district heating from the boiler at moderate pressure.

© 2012 Published by Elsevier Ltd.

Keywords: Two phase flow, Vertical evaporator, Dynamic load, Internal Rifled Boiler Tubes, Central WENO, Hyperbolic balance laws.

\textsuperscript{a}Modelled by a fifth order Central WENO scheme for solving hyperbolic balance laws.

\textit{Email address:} axejo@dongenergy.dk (Axel Ohrt Johansen)
Operating flexibility is therefore of great importance for the business economics of the plants and also a prerequisite for a stable electrical system. No matter how strong focus is put on this operational flexibility, power plants, however, will always be subject to technical limitations - e.g. boiler dynamics, coal mill dynamics, flame stability and material constraints. The power plants’ ability to stabilize the electrical system can be increased substantially if we get a better understanding of the thermodynamic and flow changes, which occur in the evaporation process. Siemens has spent years developing a new evaporator concept, in brief; it has developed an evaporator with vertical boiler tubes lined with internal rifles. The system is called the SLMF principle (Siemens Low Mass Flux), see [1], which can be used for very specific evaporator systems. One of the advantages of using SLMF is that the boiler’s primary operating area (Benson minimum) can be moved from the traditional 35-40% load. In new constructions this transition point is to around 20% load. In this way we avoid the very expensive and time consuming Benson transition, when an installation must adapt to the free electricity market and drive down the load. There is very little literature on the subject and there is a modest material relating to the mathematical description of heat transfer and pressure drop in rifled boiler tubes. Back in 1985 Harald Griem, [1] wrote on the subject, and both KEMA and Siemens have performed considerable experimental work that is considered company secrets. Other authors who have dealt with the topic experimentally are [2], [3], [4] and [5]. They have developed consistent algebraic expressions for frictional pressure drop and heat transfer in internally rifled boiler tubes. A Weighted Essentially non-oscillatory (WENO) solver code is implemented in C++ under MicroSoft Visual Studio 2008, and the solver is validated in [6]. The water/steam table is based on a fast bi-linear interpolation scheme, where the lookup table are based on the IAPWS97 standard, which is implemented in FORTRAN 90. The lookup table is described in [7].

2. Evaporation in steam power boilers

A power plant boiler works as a heat exchanger. On one side the fuel is burned and the product of combustion is a hot gas exchanging radiant heat to the water on the other side of the heat exchanger. The boiler is traditionally built as a tower, inside the hot gas is produced and the walls of the boiler are made of pipes welded together, in these pipes the water flows.

The heat flux is approximately 200-400 kW/m² in the lower sections of the boiler and is represented as radiation. At the upper part of the tower, the radiation is still dominating, but it is also necessary to take convective heat transfer into account. At the bottom section, where the radiation from gas to the pipe wall is dominating, the heat transfer on the outside is so massive that it is no longer setting the restriction for the optimal heat transfer. Instead, the limit is set by the heat transfer rate from the pipe-wall to the water inside the pipe.

One parameter that determines the heat transfer rate on the inside of the pipe is the fluid velocity near the inner pipe wall. If the velocity can be increased without increasing the net mass flux through the boiler, the heat transfer rate can be increased. With that assumption it is possible to build a more compact boiler,
by taking into account the specific type of combustion processes (Coal, Gas Wood pellets ect.). Internally rifled boiler tubes (IRBT) are an attempt to speed up the velocity at the wall and keep the vertical tube of a boiler construction. The mass flux through the IRBT are usually in the range of 1000 $kg/m^2s$ and is less than the half as is seen in traditional Benson boiler panel walls, with a moderate pipe inclination.

In addition to the increase in heat transfer, the IRBTs are characterised by an excellent performance concerning two phase flow. The swirl is very good for separation of liquid from gas. The centrifugal force will increase the rate of light fluid to the centre of the pipe and force the heavy fluid components to near the wall, which will improve the cooling of the pipe and thereby increase the heat transfer and decrease the wall temperature of the pipe. Additionally the IRBT have the following advantages: The rifles will enlarge the surface of convective heat transfer, increasing the turbulent intensity in the boundary layer and increase the relative velocity between the wall and core fluid by rotational flow.

The advantages of the IRBT have a price. The pressure loss is higher than in the traditional boiler tubes, but it can be used in a constructive way. When super critical boilers operate at part load, stability problems can occur. The problem is usually solved by building individual pressure loss at each pipe inlet section. Thus the increased pressure loss in the IRBT can be utilized to replace the traditional built in pressure loss and thereby not increase the pumping power.
3. Methods

Although the assumptions of thermodynamic equilibrium are often made in two-phase flow models, the phases rarely find themselves at thermal equilibrium. Some degree of thermal non-equilibrium arises in practically all situations and specially in dynamic situations, thermal non equilibrium must always be present so that heat and mass transfer can take place. Thermodynamic equilibrium does exist between a liquid and its vapour separated by a flat interface e.g., water and steam in a closed vessel. In the classical case of stationary vapour / bubble in large amount of liquid, the vapour and liquid temperatures are equal. However, due to the effect of surface tension, even in this equilibrium situation, the system temperature must be slightly above the saturation temperature corresponding to the pressure of the liquid. It is only in the case of the flat interface, that both phases can be exactly at saturation. Thus, the absence of hydraulic and thermal equilibrium is the rule rather than the exception in multi phase flows. In this chapter we outline a homogeneous dynamic flow model, based on the two layer flow model outlined in [8].

3.1. Thermo-Hydraulic model

The homogeneous model is based on the assumption of both hydraulic and thermal equilibrium and consists of three conservation equations, which can be reformulated in the three in-dependant variables $\rho$ (density), $\dot{m}$ (mass flow) and $E$ (internal energy), where the dependant variables $z$ (axial position in the pipe) $\in [0 ,... , l_z]$ and $t$ (time) $\in [0 ,... , \infty]$. The pipe length is $l_z$. For the massflow given by: $\dot{m} = \bar{\rho} u A$ we find:

Continuity equation:
$$\frac{\partial}{\partial t} (\bar{\rho} A) + \frac{\partial}{\partial z} (\dot{m}) = 0 \quad (1)$$

where $A=\pi r_i^2$ is the cross section area of the pipe and $r_i$ is the inner radius of the pipe. The mixture density is given by $\bar{\rho}$.

Momentum equation:
$$\frac{1}{A} \frac{\partial}{\partial t} (\dot{m}) + \frac{1}{A} \frac{\partial}{\partial z} (\dot{m} u) = - \frac{\partial \bar{p}}{\partial z} - \bar{\rho} g \cos (\theta) - F_w - F_s \quad (2)$$

where the mixture fluid velocity is given by $u$, $g$ is the gravity and $\theta$ is the angle of pipe inclination measured from the vertical direction. The mixture pressure is given as $\bar{p}$ and the shear forces due to wall friction is given by: $F_w = \frac{2}{\pi} \tau_w$ and $\tau_w$ is given by (6) and $S_w$ is the perimeter. The turbulent Reynolds stresses in the mixing fluid is given by $F_s = \frac{2}{\pi} \tau_s$.

Energy equation:
$$\frac{\partial}{\partial t} \left( \bar{\rho} A \bar{h} + \frac{1}{2} \bar{\rho} A u^2 - p A \right) + \frac{\partial}{\partial z} \left( \dot{m} \bar{h} + \dot{m} \frac{1}{2} u^2 \right) = S_w q_e'' - \dot{m} g \cos (\theta) \quad (3)$$

Here the mixture enthalpy is given as $\bar{h}$. Equation (3) can be reformulated by use of the definition of the total specific convected energy: $\bar{\phi} = \bar{h} + 1/2u^2 + gz \cos (\theta)$ and by using the continuity equation to eliminate the gravitational terms on the left side, we find:
$$\frac{\partial}{\partial t} (A(\bar{\phi} - \bar{\rho})) + \frac{\partial}{\partial z} (\dot{m} \bar{\phi}) = q_e'' S_w - \dot{m} g \cos (\theta) \quad (4)$$

where $q_e''$ represents the heat flux per unit surface area through the inner wall and $S_w$ is the perimeter of the heated domain. The internal energy $E$ is given as: $E = (\bar{\phi} - \bar{\rho}) \cdot A$, which is measured in [J/m].
3.2. Hydraulic closure laws

Closure laws in relation to the momentum equation is presented here. The axial shear stress is modelled by for example the Van Driest mixing length theory, see [9]:

\[ \tau_s = -\frac{\partial}{\partial z} \left( \overline{\rho} u' v' \right) \approx l^2 \frac{\partial \overline{u^2}}{\partial z^2} \]

(5)

and is used as an damping in the solution domain where we experienced transients initiated by large gradients in the density (compressibility). This is only applied for a restricted domain where the steam quality \( x \in [-0.02, 0.02] \). The corresponding wall shear stress is given by

\[ \tau_w = f_w \frac{u \cdot |u|}{2} = f_w \frac{G \cdot |G|}{2\overline{\rho}} \]

(6)

The term \( f_w \) is the dimensionless friction coefficient based on the single phase frictional coefficient in heated rifled tubes: \( f_w = \frac{a}{Re_b} + c \). In table (1) we propose coefficients given by [1], for different rifled profiles. In [3] the same formulation of \( f_w \) is used and the author has for specific rifled pipes reported an absolute relative error less than 6.3 \%.

| Table 1. Algebraic relations of \( f_w \) for different profiles. [1] |
|-------------------------|----------------|----------------|----------------|
| type | RR6 | RR5 | RR4 | RR2 |
| a | 1702 | 0.56 | 16.26 | 1.65 |
| b | 1.18 | 0.32 | 0.71 | 0.44 |
| c | 0.032 | 0.01309 | 0.01509 | 0.02344 |

In the two-phase region the friction factor is adjusted according to a two-phase multiplier, formulated by [10]. In that case \( f_w \) is based on fluid properties for saturated liquid. The model that is based on [10] calculates the two phase multiplier as:

\[ \phi^2 = 1 + B \cdot x \cdot \left( \frac{\rho_l}{\rho_f} - 1 \right) \]

(7)

Where the coefficient \( B \) is:

\[ B = 1.58 - 0.47 \frac{P}{P_c} - 0.11 \left( \frac{P}{P_c} \right)^2 \]

(8)

Note that the critical pressure (\( P_c \)) is 221.2 [bar] for water/steam. \( B \) is adjustment as: \( B = B - (B - 1)(10 - x - 9) \). The correlation of (7) is compared to the well known and more computation intensive model of Friedel and is illustrated in figure (2).

3.3. Pipe Wall Model

The heat transfer processes from a combustion process (radiation and convection) to the water and steam circuit in a power plant, is using the pipe wall as the transfer median, to transport the energy from the furnace to the cooling media, in this case water/steam flowing in the panel wall. The solution of problems involving heat conduction in solids can, in principle, be reduced to the solution of a single differential equation, by Fourier’s law. The equation can be derived by making a thermal energy balance on a differential volume element in the solid. A volume element for the case of conduction only in the z-direction is illustrated in figure (3).
The balance equation becomes:

$$\frac{\partial T_w}{\partial t} = \alpha \frac{\partial^2 T_w}{\partial z^2} + \frac{\dot{q}_f}{\rho_w C_{pw}} \frac{S}{A_c} - \frac{\dot{q}_e}{\rho_w C_{pw}} \frac{d_\pi}{A_c}, \quad z \in [0, l_z] \land t \geq 0$$

(9)

where $C_{pw}$ and $\rho_w$ are the heat capacity and the density of the pipe wall and $A_c = \pi (r_o^2 - r_i^2)$ is the cross section area of the pipe wall. $T_w$ is the mean wall temperature forced by the heat fluxes $\dot{q}_f$ and $\dot{q}_e$ expressing the heat flux from the furnace and the heat flux to the cooling fluid respectively.

Hence we can summarize the system of balance laws (SBL), given by (1), (2), (4) and (9), into a compact vector notation, given by:

$$\frac{\partial \Phi(z,t)}{\partial t} + \frac{\partial f(\Phi(z,t))}{\partial z} = g(\Phi(z,t)) + h(\frac{\partial \Phi}{\partial z}, \Phi(z,t)), \quad \Phi \in \mathbb{R}^m, m = 4, \quad t \geq 0 \land z \in \Omega$$

(10)
where the dependent variable Φ and the flux vector f are given as

$$\Phi = \begin{pmatrix} \bar{\rho} A \\ \dot{m} \\ E \\ T_w \end{pmatrix}, \quad f(\Phi) = \begin{pmatrix} -\dot{m} \\ \frac{\dot{m}}{E} \frac{\partial p}{\partial E} + p A \\ 0 \end{pmatrix}$$

and the source and diffusion vectors are given as:

$$g(\Phi) = \begin{pmatrix} 0 \\ p \frac{\partial A}{\partial \dot{m}} - \bar{\rho} g A \cos \theta - \sqrt{\frac{\pi}{2}} \int_{\frac{\pi}{2}}^{\frac{\pi}{2}} \frac{\partial q_{in}}{\partial A} d\theta \\ S_w q_{in} - m g \cos (\theta) \\ \alpha \frac{\partial^2 T_w}{\partial x^2} + \frac{q_r}{c_r} \bar{S} - \frac{q_r}{c_r} \frac{d \pi}{\partial x} \end{pmatrix}$$

and

$$h(\Phi) = \begin{pmatrix} 0 \\ \frac{\dot{E}_{in}}{\pi \rho w} \\ 0 \\ \alpha \frac{\partial^2 T_w}{\partial x^2} \end{pmatrix}$$

Here the dependent variables are $\bar{\rho}$, $\dot{m}$, $E$ and $T_w$ meaning the fluid density, mass flow, total energy of the conserved fluid and wall mean temperature respectively. The pressure can be determined iteratively by water steam tables: $p=p(E, \rho)$. The source term $g$ consists of both source/sink terms and the diffusion term $h$ includes contributions from the mixing length eddy viscosity (5), working as a damping term in the vicinity of $x=0$, and the thermal diffusion in the pipe wall as well.

### 3.4. Constitutive relations for the heat pipe model

For isotropic materials, we introduce the thermal diffusivity given by: $\alpha = \frac{k_f}{\rho w c_w}$ given in $[m^2/s]$, which in a sense is a measure of thermal inertia and expresses how fast heat diffuses through a piece of solid. For a typical panel wall, the thermal diffusivity is approximately $1.98 \cdot 10^{-6}$ $[m^2/s]$ at $200^\circ C$, see [11]. The radiation from the furnace to the pipe surface is given by the heat flux $\dot{q}_r$. The heat flux $\dot{q}_r$ represents the convective heat transfer between the pipe wall inner surface and the flowing fluid in the pipe, and is given as: $\dot{q}_r=\dot{h}_c(T_w-T_f)$, where $\dot{h}_c$ is the convective heat transfer coefficient and $T_w-T_f$ is the driving temperature difference, which is positive for boiling. For isotropic materials (pipe wall), we have expressions for specific heat capacity $C_{pw}$, heat conductivity $k_{pw}$ and density $\rho_{pw}$ as function of temperature in Kelvin from [11] and [12].

A simple, fast and robust model of the heat transfer in film boiling, is given by [13]. The heat transfer coefficient $\dot{h}_{fb}$ is given as

$$\dot{h}_{fb} = c_f \dot{q}_r^{0.673} \quad [W/m^2 K]$$

(11)

where the coefficient $c_f$ is given by the below expression, which is a function of the saturation temperature ($T_s$), measured in $[^\circ C]$

$$c_f = \frac{0.06136}{\left[1 - (\frac{T_s}{373.58})^{0.0025}\right]^{0.73}}$$

(12)

The single phase laminar heat transfer coefficient ($\dot{h}_l$) is calculated from

$$Nu_s = \frac{\dot{h}_l \ell}{k_f} = 4.36$$

(13)

and is valid for $L/d_i > 50$ and $\frac{dG}{\dot{m}} < 2000$. For turbulent single phase flow and $\frac{dG}{\dot{m}} > 10,000$ we use

$$Nu_s = \frac{\dot{h}_l \ell}{k_f} = 0.023 \left(\frac{dG}{\dot{m}}\right)^{0.8} \left(\frac{c_p \mu_f}{k_f}\right)^{1/3}$$

(14)
The total heat transfer coefficient is given by (15), and consists of two contributions; one from the convective heat transfer boundary layer associated to the flowing fluid inside the heat pipe and one that relates to conduction through the pipe wall material.

\[
h = \frac{1}{h_c + \frac{r_i}{k_w} \cdot \ln(r_w/r_i)}
\]  

(15)

where \(h_c\) is expressing the heat transfer coefficient due to the thermal boundary on the inner side of the pipe wall and \(r_w\) is defined by \(T_w = T_i(r_w)\). Since we use the calculated average wall tube temperature as driver in the calculation of the total heat transport to the fluid, we must know \(r_w\).

Due to the knowledge of radial conduction in the pipe, we use a simple analytical wall temperature profile, for estimating the inner wall temperature, expressed by the averaged wall temperature \((T_w)\), based on the heat transfer through the isotropic pipe wall to the flowing fluid. Let \(T(r)\) represent the radial temperature distribution by

\[
T(r) = T_i - T_o \cdot \frac{\ln(r)}{r_o} + T_o
\]

(16)

where \(r\) is the pipe radius with suffix (i=inner) and (o=outer). This temperature profile for radial isotropic pipes, is the steady state solution to the 1D Fourier’s law of heat transfer. Hence, for small values of the thermal diffusivity, the averaged wall temperature can reasonably be estimated by:

\[
T_w = \frac{1}{A_c} \int_{r_i}^{r_o} 2\pi r \cdot T(r) \, dr = \frac{2\pi}{A_c} \left[ a_0 \left[ \frac{x^2}{2} \ln(x/2) - \frac{x^2}{4} \right]_{r_i}^{r_o} - a_0 \ln(r_o) \left[ \frac{x^2}{2} \right]_{r_i}^{r_o} \right] + T_o \cdot \left[ \frac{x^2}{2} \right]_{r_i}^{r_o}.
\]

(17)

where \(a_1\) is given by

\[
a_1 = \frac{r_i^2}{r_o^2} - \frac{1}{2\ln(r_i/r_o)}
\]

(18)

Hence we find \(T_i\) by insertion (17) in (19):

\[
T_i = T_w + \frac{q_r S \ln(z)}{2\pi k_w} (1 - a_1) = \frac{2\pi k_w}{\ln(r_w/r_i)} (T_w - T_i).
\]

(19)

Hence the entire heat transfer can be estimated for the temperature range in between the wall mean temperature \((T_w)\) and the fluid mixture temperature \((T_f)\), which is assumed homogeneous and well mixed with a temperature boundary layer represented by \(h_c\). The one dimensional pipe wall model does only consists of axial heat transfer term, and have no spatial resolution in the radial dimension. The inner wall temperature can be determined by use of the equation for pure conduction through the pipe:

\[
q_r S = \frac{2\pi k_w}{\ln(r_w/r_i)} (T_o - T) = \frac{2\pi k_w}{\ln(r_w/r_i)} (T_w - T).
\]

(19)

Hence we find \(T_i\) by insertion (17) in (19):

\[
T_i = \frac{q_r S}{2\pi k_w} \ln(z) (1 - a_1)
\]

(20)

and hence \(r_w\) in (15) can be determined from (17) and (20) and we find

\[
h = \frac{1}{h_c + \frac{r_i(a_1-1)}{k_w} \cdot \ln(r_i/r_o)}
\]

(21)

where \(h_c\) is smoothed in-between \(h_s\) and \(h_{fb}\) depending of the dryness of the fluid. Additionally \(h_c\) is adjusted on the basis of a smoothing between laminar and turbulent single phase flow as well as for two-phase
flow. The smoothing function is based on a third order function and the associated slopes are determined numerically. Note that the heat flux is positive for $T_i > T_f$. Using the model parameters from table (2) we find $a_1=0.423$ and the temperature fall above the thermal boundary is: $T_o-T_i=27.9 \text{[°C]}$, which gives a temperature gradient in the pipe wall of $\frac{dT}{dr}= 3930 \text{[°C/m]}$ for a heat flux of $q_e=100 \text{[kW/m²]}$. The heat conduction in the material is the most significant barrier for an effectively cooling of the tube wall.

3.5. Auxiliary relations

The Water / Steam library IAPWS 97 by [14] is used as a general equation of state, to derive thermodynamic properties of water and steam. In some relations we need a relationship for the pressure as function of density and enthalpy: $p=p(\rho, h)$. This can be done by a Newton Rapson solver. To improve the computational speed, we recommended to use a look up table within at least 200000 nodes, based on bilinear interpolation, see [7]. Here we create a look up table to ensure water/steam properties within an accuracy below 0.3% as an absolute maximum, due to [7]. Note that the density is smoothed in the vicinity of the saturation line of water to avoid heavy gradients and discontinuities.

3.6. Boundary conditions

It is convenient to use boundary conditions to the model which are physically measurable. Therefore, the following properties are used as boundary conditions; velocity ($u$), pressure ($p$) and enthalpy ($h$). This allows us to rewrite the boundary conditions to those properties, which are described by $\Phi$, see (10). The Dirichlet boundary conditions are given by (22) and the corresponding Neumann boundary conditions are obtained by applying the chain rule for differentiation of complex functions, and are given by (23).

\[
\text{Dirichlet BC : } \begin{cases} 
\rho A \\
\rho A u \\
\rho A (h + \frac{u^2}{2} + gz \cos (\theta)) - p A \\
\rho T_w 
\end{cases}
\]  

where $\theta$ is the angle of the pipe inclination with respect to the horizontal.

\[
\text{Neumann BC : } \begin{cases} 
A \frac{\partial h}{\partial r} + \rho \frac{\partial A}{\partial r} \\
u A \frac{\partial h}{\partial r} + \rho u \frac{\partial A}{\partial r} + \rho A \frac{\partial h}{\partial r} \\
\frac{\partial (\rho \Phi)}{\partial r} [h + \frac{u^2}{2} + gz \cos (\theta)] + \rho A [\frac{\partial h}{\partial r} + \rho u \frac{\partial A}{\partial r} + g \cos (\theta)] - A \frac{\partial \rho}{\partial r} - \rho \frac{\partial A}{\partial r} 
\end{cases}
\]
3.7. Numerical Solution of Hyperbolic Transport Equation

Let us consider a hyperbolic system of balance laws (SBL) formulated on a compact vector notation, given by (10), where $\Phi$ is the unknown m-dimensional vector function, $f(\Phi)$ the flux vector, $g(\Phi)$ a continuous source vector function on the right hand side (RHS), with $z$ as the single spatial coordinate and $t$ the temporal coordinate, $\Omega$ is partitioned in $n_z$ non-overlapping cells: $\Omega= \bigcup_{i=1}^{n_z} I_i \in [0, l_z]$, where $l_z$ is a physically length scale in the spatial direction. This system covers the general transport and diffusion equations used in many physical aspects and gas dynamics as well. The SBL system is subjected to the initial condition:

$$\Phi(z, 0) = \Phi_0(z)$$

and the below boundary conditions given by:

**Dirichlet boundaries:**

$$\Phi(z = 0, t) = \Phi_A(t)$$

and

$$\Phi(z = l_z, t) = \Phi_B(t)$$

**Neumann boundaries:**

$$\frac{\partial \Phi(z = 0, t)}{\partial z} = \frac{\partial \Phi_A(t)}{\partial z}$$

and

$$\frac{\partial \Phi(z = l_z, t)}{\partial z} = \frac{\partial \Phi_B(t)}{\partial z}$$

The above boundary conditions can be given by a combination of each type of boundaries. The Dirichlet condition is only specified, if we have ingoing flow conditions at the boundaries.

The development of a general numerical scheme for solving PDE’s may serve as universal finite-difference method, for solving non-linear convection-diffusion equations in the sense that they are not tied to the specific eigenstructure of a problem, and hence can be implemented in a straightforward manner as black-box solvers for general conservation laws and related equations, governing the spontaneous evolution of large gradient phenomena. The developed non-staggered grid is suitable for the modelling of transport of mass, momentum and energy and is illustrated in figure (4), where the cell $I_j=[z_{j-1/2}, z_{j+1/2}]$ has a cell width $\Delta z$ and $\Delta t$ the time step.

$$\frac{d\Phi_j(t)}{dt} = -\frac{1}{\Delta z} \left[ F_{j+1/2} - F_{j-1/2} \right] + S_j(\Phi)$$

(27)

where the numerical fluxes $F_{j+1/2}$ are given by

$$F_{j+1/2} = \frac{a_{j+1/2}^- f(\Phi_{j+1/2}^-) - a_{j+1/2}^+ f(\Phi_{j+1/2}^+)}{a_{j+1/2}^- - a_{j+1/2}^+} + \frac{a_{j+1/2}^- a_{j+1/2}^+}{a_{j+1/2}^- - a_{j+1/2}^+} \left[ \Phi_{j+1/2}^- - \Phi_{j+1/2}^+ \right].$$

(28)
Notice that the accuracy of this scheme is determined by the accuracy of the reconstruction of \( \Phi \) and the ODE solver. In this chapter the numerical solutions of (27) is advanced in time by mean of third order TVD Runge-Kutta method described by [17]. The local speeds of propagation can be estimated by

\[
a_{+1/2}^+ = \max \left\{ \lambda_N \left( \frac{\partial f(\Phi_{j+1/2}^-)}{\partial \Phi} \right), \lambda_N \left( \frac{\partial f(\Phi_{j+1/2}^+)}{\partial \Phi} \right), 0 \right\},
\]

\[
a_{-1/2}^- = \min \left\{ \lambda_N \left( \frac{\partial f(\Phi_{j-1/2}^-)}{\partial \Phi} \right), \lambda_N \left( \frac{\partial f(\Phi_{j-1/2}^+)}{\partial \Phi} \right), 0 \right\},
\]

with \( \lambda_1 < \ldots \lambda_N \) being the eigenvalues of the Jacobian given by \( J = \frac{\partial f(\Phi(z))}{\partial \Phi} \). Here, \( \Phi_{j+1/2}^- = p_{j+1}(z_{j+1/2}) \), and \( \Phi_{j+1/2}^+ = p_{j+1}(z_{j+1/2}) \) are the corresponding right and left values of the piecewise polynomial interpolant \( [p_j(z)] \) at the cell interface \( z = z_{j+1/2} \).

To derive an essentially non-oscillatory reconstruction (ENO), we need to define three supplementary polynomials \( (\Phi_1, \Phi_2, \Phi_3) \), approximating \( \Phi(z) \) with a lower accuracy on \( I \). Thus, we define the polynomial of second-order accuracy, \( \Phi_1(z) \), on the reduced stencil \( S_1: (I_{j-2}, I_{j-1}, I_j) \), \( \Phi_2(z) \) is defined on the stencil \( S_2: (I_{j-1}, I_j, I_{j+1}) \), whereas \( \Phi_3(z) \) is defined on the stencil \( S_3: (I_j, I_{j+1}, I_{j+2}) \). Now, we have to invert a 3 \( \times \) 3 linear system for the unknown coefficients \( \{a_j\}, j \in \{0, \ldots, 2\} \), defining \( \Phi_1, \Phi_2, \Phi_3 \). Once again, the constants determining the interpolation are pre-computed and stored before solving the PDEs. When the grid is uniform, the values of the coefficients for \( \Phi_1, \Phi_2 \) and \( \Phi_3 \) can be explicitly formulated. It is left to the reader to read [15] or [6] for further details about determining the coefficients in the reconstructed polynomials. To implement a specific solution technique, we extend the principle of the central WENO interpolation defined in [18]. First, we construct an ENO interpolant as a convex combination of polynomials that are based on different discrete stencils. Specifically, we define in the discrete cell \( I_j \):

\[
\Phi(z) = \sum_j w_j \times \Phi_j(z), \quad \sum_j w_j = 1 \quad \text{for} \quad w_j \geq 0 \quad \text{for} \quad j \in \{1, \ldots, 4\},
\]

and \( \Phi_1, \Phi_2 \) and \( \Phi_3 \) are the previously defined polynomials. \( \Phi_4 \) is the second-order polynomial defined on the central stencil \( S: (I_{j-2}, I_{j-1}, I_j, I_{j+1}, I_{j+2}) \) and is calculated such that the convex combination in (30), will be fifth-order accurate in smooth regions. Therefore, it must verify:

\[
\Phi_{sp}(z) = \sum_j C_j \times \Phi_j(z) \quad \forall z \in I_j, \quad \sum_j C_j = 1 \quad \text{for} \quad C_j \geq 0 \quad \text{for} \quad j \in \{1, \ldots, 4\},
\]

The calculation of \( \Phi_{j+1/2}^+, \Phi_{j+1/2}^- \) produces the following simplified result:

\[
\Phi_{j+1/2}^+ = \left( -\frac{7}{120} w_4 - \frac{1}{6} w_1 \right) \Phi_{j-2} + \left( \frac{1}{3} w_2 + \frac{5}{6} w_1 + \frac{21}{40} w_4 \right) \Phi_{j-1} + \left( -\frac{1}{6} w_2 - \frac{7}{12} w_3 - \frac{7}{120} w_4 \right) \Phi_{j+1} + \left( \frac{1}{3} w_3 - \frac{1}{60} w_4 \right) \Phi_{j+2}
\]

\[
\Phi_{j+1/2}^- = \left( -\frac{1}{60} w_4 + \frac{1}{3} w_1 \right) \Phi_{j-2} + \left( -\frac{1}{6} w_2 - \frac{7}{6} w_3 - \frac{7}{120} w_4 \right) \Phi_{j-1} + \left( \frac{5}{6} w_2 + \frac{3}{5} w_3 + \frac{11}{12} w_1 + \frac{73}{120} w_4 \right) \Phi_j + \left( \frac{1}{3} w_3 - \frac{5}{6} w_1 + \frac{21}{40} w_4 \right) \Phi_{j+1} + \left( -\frac{1}{6} w_3 - \frac{7}{120} w_4 \right) \Phi_{j+2}
\]

To calculate the weights \( w_j, j \in \{1, 2, 3, 4\} \), we review another technique to improve the classical smoothness indicators to obtain weights that satisfy the sufficient conditions for optimal order of accuracy. It is well known from [15], that the original WENO is fifth order accurate for smooth parts of the solution domain,
except near sharp fronts and shocks. The idea here is taken from [16] and uses the hole five point stencil \( S_5 \) to define a new smoothness indicator of higher order than the classical smoothness indicator \( IS_i \). The general form of indicators of smoothness are defined in [18]:

\[
IS'_j = a_1^2 \Delta z^2 + \frac{13}{3} a_2^2 \Delta z^4 + O(\Delta z^6), \quad j \in \{1, 2, 3\}.
\]

(34)

and the form of \( IS^*_j \) is given by [15]:

\[
IS^*_j = a_1^2 \Delta z^2 + \left[ \frac{13}{3} a_2^2 + \frac{1}{2} a_1 a_3 \right] \Delta z^4 + O(\Delta z^6).
\]

(35)

where \( a_0 \) and \( a_1 \) can be determined by solving the coefficients to reconstructed polynomial \( \Phi_k \) on \( S_5 \). For estimating the weights \( w_k, k \in \{1, 2, 3, 4\} \), we proceed as follows: Define

\[
IS^*_k = \frac{IS_k + \epsilon}{IS_k + \epsilon + \tau_S}
\]

(36)

where \( IS_k, k \in \{1, 2, 3\} \) are given by (34), \( IS^*_k \) given by (35) and \( \tau_S = |IS_1 - IS_3| \). The constant \( \epsilon \) is a small number. In some articles \( \epsilon \) is from \( 1 \cdot 10^{-2} \) to \( 1 \cdot 10^{-6} \), see [18]. Here we use much smaller values of \( \epsilon \) for the mapped and improved schemes in order to force this parameter to play only its original role of not allowing vanishing denominators at the weight definitions. The weights \( w_k \) are defined as:

\[
w_k = \frac{a_k^*}{\sum_{j=1}^4 a_j^*}, \quad C_k = \frac{C_k}{IS_k^*}, \quad k \in \{1, 2, 3, 4\}
\]

(37)

The constants \( C_j \) represent ideal weights for (30). As already noted in [18], the freedom in selecting these constants has no influence on the properties of the numerical stencil; any symmetric choice in (31), provides the desired accuracy for \( \Phi_{opt} \). In what follows, we make the choice as in [15]:

\[
C_1 = C_3 = 1/8, C_2 = 1/4 \quad \text{and} \quad C_4 = 1/2.
\]

(38)

3.7.1. Convection-Diffusion equations

Let us again consider the general System of Conservation Laws (SCL), given by equation (10), where the source term \( g \) is replaced by a dissipative flux:

\[
\frac{\partial \Phi(z, t)}{\partial t} + \frac{\partial f(\Phi(z, t))}{\partial z} = \frac{\partial}{\partial z} \left( g(\Phi(z, t), \frac{\partial \Phi}{\partial z}) \right), \quad t \geq 0, \quad z \in \Omega
\]

(39)

The gradient of \( g \) is formulated on the compressed form: \( g(\Phi, \phi_t) \), as a nonlinear function \( \neq 0 \). This term can degenerate (39) to a strongly parabolic equation, admitting non smooth solutions. To solve it numerically is a highly challenging problem. Our fifth-order semi-discrete scheme, (27)-(28), can be applied to (10) in a straightforward manner, since we can treat the hyperbolic and the parabolic parts of (39) simultaneously. This results in the following conservative scheme:

\[
\frac{\partial \Phi_j(t)}{\partial t} = -\frac{1}{\Delta z} \left[ F_{j+1/2} - F_{j-1/2} \right] + G_j(\Phi, t).
\]

(40)

Here \( F_{j+1/2} \) is our numerical convection flux, given by equation (28) and \( G_j \) is a high-order approximation to the diffusion flux \( g(\Phi, \phi_t) \). Similar to the case of the second-order semi-discrete scheme of [19], operator splitting is not necessary for the diffusion term. By using a forth order central differencing scheme, outlined by [20], we can apply our fifth-order semi-discrete scheme, given by (27) and (28), to the parabolic equation (10), where \( g(\Phi, \phi_t) \) is a function of \( \phi \) and its derivative in space (diffusion). The diffusion term can be expressed by a high-order approximation:

\[
G_j(t) = \frac{1}{12 \Delta z} \left[ -G(\Phi_{j+2}, (\Phi_z)_{j+2}) + 8 \cdot G(\Phi_{j+1}, (\Phi_z)_{j+1}) - 8 \cdot G(\Phi_{j-1}, (\Phi_z)_{j-1}) + G(\Phi_{j-2}, (\Phi_z)_{j-2}) \right]
\]

(41)
where

\[
\begin{align*}
(\Phi_3)_{j+2} &= \frac{1}{12\Delta z} \left[ 25 \Phi_{j+2} - 48 \Phi_{j+1} + 36 \Phi_j - 16 \Phi_{j-1} + 3 \Phi_{j-2} \right], \\
(\Phi_3)_{j+1} &= \frac{1}{12\Delta z} \left[ 3 \Phi_{j+2} + 10 \Phi_{j+1} - 18 \Phi_j + 6 \Phi_{j-1} - 3 \Phi_{j-2} \right], \\
(\Phi_3)_{j-1} &= \frac{1}{12\Delta z} \left[ \Phi_{j+2} - 6 \Phi_{j+1} + 18 \Phi_j - 10 \Phi_{j-1} - 3 \Phi_{j-2} \right] \\
(\Phi_3)_{j-2} &= \frac{1}{12\Delta z} \left[ -3 \Phi_{j+2} + 16 \Phi_{j+1} - 3 \Phi_j + 48 \Phi_{j-1} - 25 \Phi_{j-2} \right]
\end{align*}
\]

and \( \Phi_j \) are the point-values of the reconstructed polynomials.

### 3.7.2. Source Term

Next, let us consider the general SCL given by (10) and restrict our analysis to the source term of the form: \( g(\Phi, t) \) as a continuous source vector function \( \neq 0 \). By integrating system (10) over a finite space-time control volume \( I_i, \Delta t \) one obtains a finite volume formulation for the system of balance laws, which usually takes the form

\[
\Phi(z, t)_{n+1} = \Phi(z, t)_n - \frac{\Delta t}{\Delta z} \left( f_{j+1/2} - f_{j-1/2} \right) + \Delta t g(z, t), \quad t \geq 0, \ z \in \Omega \tag{43}
\]

The integration of (10) in space and time gives rise to a temporal integral of the flux across the element boundaries \( f_{j+1/2} \) and to a space-time integral \( g_i \) of the source term inside \( I_i \). In practice, one must replace the integrals of the flux and the source in (43) by some suitable approximations, that is to say one must choose a concrete numerical scheme. For SBL a numerical source must be chosen. Here, not only the three classical properties are required, but some additional properties are needed for the global numerical scheme: It should be well-balanced, i.e. able to preserve steady states numerically. It should be robust also on coarse grids if the source term is stiff.

### 3.7.3. Boundary conditions for Non-staggered grid

For a system of \( m \) equations we need a total of \( m \) boundary conditions. Typically some conditions must be prescribed at the inlet boundary \( (z=a) \) and some times at the outlet boundary \( (z=b) \). How many are required at each boundary depends on the number of eigenvalues of the Jacobian \( A \) that are positive and negative, respectively and whether the information is marching in or out for the boundaries.

By extending the computational domain to include a few additional cells on either end of the solution domain, called ghost cells, whose values are set at the beginning of each time step in some manner that depends on the boundary condition. In figure (4) is illustrated a grid with three ghost cells at each boundary. The idea behind the ghost point approach is to express the value of the solution at control points outside the computational domain in terms of the values inside the domain plus the specified boundary condition. This allows the boundary condition to be imposed by a simple modification of the internal coefficients using the coefficients of the fictitious external point. This can result in a weak imposition of the boundary condition, where the boundary flux not exactly agree with the boundary condition. By establishing a Taylor expansion around the boundary \( a \) or \( b \), we can express a relationship between the ghost points outside the solution domain and grid points inside the domain. For further details see [6].

### 3.7.4. Time discretization

The semi-discrete ODE given by (27) is a time dependent system, which can be solved by a TVD Runge-Kutta method presented by [17]. The optimal third order TVD Runge-Kutta method is given by

\[
\begin{align*}
\Phi_j^{(1)} &= \Phi_j^n + \Delta t L(\Phi_j^n), \\
\Phi_j^{(2)} &= \frac{3}{4} \Phi_j^n + \frac{1}{4} \Phi_j^{(1)} + \frac{1}{4} \Delta t L(\Phi_j^{(1)}), \\
\Phi_j^{n+1} &= \frac{1}{3} \Phi_j^n + \frac{2}{3} \Phi_j^{(2)} + \frac{2}{3} \Delta t L(\Phi_j^{(2)}), \quad \text{for } j \in [1, n_c].
\end{align*}
\]
The stability condition for the above schemes is

\[ CFL = \max \left( \frac{u^n_j}{\Delta z} \right) \Delta t \leq 1, \]  

(45)

where CFL stands for the Courant-Friedrichs-Lewy condition and \( u^n_j \) is the maximum propagation speed in cell \( I_j \) at time level \( n \).
4. Results

In this section we setup and solve a homogeneous boiler tube model for two cases; one without IRBT and one with Siemens RR5 pipes. The governing equations are defined by the system of balance laws given by equation (10) including the pipe wall model given by equation (9) for the solution domain given by $\Omega \in [0, l_z]$.

4.1. Numerical setup

Three Dirichlet boundary conditions are applied for the hydraulic case and two Neumann boundaries are applied for the pipe wall model, given as zero gradients in the wall temperature at each pipe end (No heat loss). The intention is to model an evaporator, which can induce oscillations initiated by the compressibility, which arise as a result of a phase shift in the lower part of the evaporator. Therefore, we apply a constant downstream Dirichlet pressure boundary condition, that is corresponding to a stiff system, without any pressure absorption effects in the down stream turbine system due to compressibility. An analogy to this is a geyser, where there is a constant surface pressure and an intense heat absorption in the bottom region, whereby an oscillating pressure wave is initiated due to the compressibility of the fluid, caused by intense heat from the underground. Additionally we force the model with both a constant enthalpy and mass flux located on the upstream boundary, supplied by a constant heat flux along the entire heat pipe. The numerical scheme is the fifth order WENO scheme outlined in chapter (3.7) and consists of 400 computational points with CFL number of 0.8. The numerical scheme is tested for consistency and stability with respect to both a scalar- and a system of hyperbolic equations and has been successfully compared to analytical results from the literature as well as other published results. This work is outlined in [7].

The model is soft started in two steps, at $t=0$ [s] is the pure hydraulic model soft started during 4 seconds, without heat flux. After 10 seconds the heat flux is build-up during four seconds. This is done to avoid heavy shock waves moving forward and back in the entire solution domain. If the soft start period is reduced to only 1 second, heavy pressure oscillations occur. The soft start model is based on a third order theory [21], which gives a $C^2$ continuous sequence, which means zero gradients of the first derivative at both ends of the soft start period. The model data are listed below in table (2). The dynamic start-up process can be seen in figure (6), where the density is given in $[\text{kg}/\text{m}^3]$, pressure in [bar], Temperature in [°C], enthalpy in [kJ/kg] and mixture velocity in [m/s].

4.2. Model consistency

The model consists of 400 differential elements, thus ensuring a smooth continuous solution. By reducing the number of computational cells to only 50 elements, one would observe a more intensive standing wave at the entrance of two-phase region, which is due to intensive heating of the differential cell in the vicinity of the boiling zone, where we have an intensive negative slope in the density as function of the enthalpy, hence the density change becomes so violent that a pressure wave is established to ensure momentum balance. Using a CFL number higher than 1.0 is leading to instabilities due to the semi implicit scheme.

4.3. Simulation results - without IRBT

In figure (5) we illustrate the output results for each 25 sec. of simulation, referring to the solution of the full-scale evaporator at Skærkælværket unit 3 (SKV3) in Fredericia (Denmark), without IRBT. Here we have a tower boiler which consists of 4x56 parallel boiler tubes representing an entire mass flow of 90 [kg/s] flowing in 193.5 meter long heat pipes with an inclination of 12 degree. A steady state solution is obtained after approximately 250 seconds, and is depicted in figure (6) together with the initial conditions. The entire pressure drop and heat uptake fit ($\pm$ 5 %) with steady state experiments performed at (SKV3). The simulation results shows how the state of the fluid gradually moves from the inlet condition, in the form of subcooled water, to the two phase zone, in which the boiling is starting, and finally reaches the super heating zone, where the dry steam is superheated to approximately 360 [°C]. The pressure drop is fixed downstream in the form of a Dirichlet boundary condition, corresponding to measured pressure level from (SKV3). The Pressure distribution along the evaporator reflects different pressure loss models, the pressure
### Table 2. Geometrical and numerical specifications. Data in parentheses are referring to simulation without IRBT.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gravity (g)</td>
<td>9.81</td>
<td>[m/s²]</td>
<td>Spatial start position</td>
<td>0.000</td>
<td>[m]</td>
</tr>
<tr>
<td>Spatial end position (L)</td>
<td>38.25</td>
<td>[m]</td>
<td>Inner diameter of pipe (d_i)</td>
<td>23.8</td>
<td>[mm]</td>
</tr>
<tr>
<td>Outer diameter of pipe (d_o)</td>
<td>38.0</td>
<td>[mm]</td>
<td>Heat conductivity in wall (k_w)</td>
<td>10.139</td>
<td>[W/mK]</td>
</tr>
<tr>
<td>Wall density (ρ_w)</td>
<td>7850.0</td>
<td>[kg/m³]</td>
<td>Specific heat capacity of pipe wall (C_p)</td>
<td>527.21</td>
<td>[J/kg/K]</td>
</tr>
<tr>
<td>Heat flux (q_e)</td>
<td>100.000</td>
<td>[W/m²]</td>
<td>Wall roughness (λ)</td>
<td>1.0E-6</td>
<td>[m]</td>
</tr>
<tr>
<td>Initial Enthalpy - Inlet</td>
<td>1187.6988</td>
<td>[kJ/kg]</td>
<td>Initial Enthalpy - Outlet</td>
<td>1187.6988</td>
<td>[kJ/kg]</td>
</tr>
<tr>
<td>Initial Pressure - Inlet</td>
<td>92.3762</td>
<td>[Bar]</td>
<td>Initial Pressure - Outlet</td>
<td>92.3762</td>
<td>[Bar]</td>
</tr>
<tr>
<td>Initial Velocity - Inlet</td>
<td>0.0</td>
<td>[m/s]</td>
<td>Initial Velocity - Outlet</td>
<td>0.0</td>
<td>[m/s]</td>
</tr>
<tr>
<td>Pressure BC (Dirichlet - Outlet)</td>
<td>92.3762</td>
<td>[Bar]</td>
<td>Enthalpy BC (Dirichlet - Inlet)</td>
<td>1187.6988</td>
<td>[kJ/kg]</td>
</tr>
<tr>
<td>Velocity BC (Dirichlet - Inlet)</td>
<td>0.200(1.1711)</td>
<td>[m/s]</td>
<td>Simulation time</td>
<td>200.0</td>
<td>[s]</td>
</tr>
<tr>
<td>Output frequency</td>
<td>0.1</td>
<td>[s]</td>
<td>CFL number</td>
<td>0.80</td>
<td>[-]</td>
</tr>
<tr>
<td>Number of computational grids (N_p)</td>
<td>400</td>
<td>[-]</td>
<td>Riffle type</td>
<td>RR5 1.5994(No rifels)</td>
<td>[-]</td>
</tr>
</tbody>
</table>

![Fig. 5. Solution of SKV3 evaporator model without IRBT after (a):25, (b):50, (c):75 and (d):100 sec.](image)

The pressure drop in the two-phase region involves the two phase multiplier, outlined in (7), which multiplies the pressure gradient with up to 16 times relative to the pressure gradient for saturated water. The inlet velocity is specified as an upstream Dirichlet boundary condition, and is soft started by use of the before mentioned smooth function, having a soft start period of four seconds. The super heated steam leaves the downstream boundary at steady state flow condition with a speed of app. 24 [m/s]. This ensures a smooth hydraulic flow condition of a cold evaporator. After words the heating is build up smoothly, applied by the same smoothing technique, so that undesirable thermal shock phenomena is reduced to a minimum. A standing pressure wave in the front of the boiling zone of the fluid is created by the very intense negative slope in the fluid density at the entrance to the two phase region. This
pressure-drop oscillations could occur, when there exists large upstream compressibility in the flow boiling system, see ([22], [23]). This phenomenon is increased in a vertical evaporator where the heating phase has
a heavy column of liquid to be transported out of the solutions area, which can initiate stability problems. This phenomenon occurs at low operating pressure in the evaporator or low firing, i.e. heating the bottom of the evaporator. The dryness line in figure (5) expresses the mass based percentage of the steam flowing in the evaporator tube, not surprisingly, this process linearly corresponding to a constant heat flux along the tube.

Pressure-drop oscillations can be characterised as a secondary phenomenon of dynamic instability, which is triggered by a static instability phenomenon. Pressure-drop oscillations occur in systems having a compressible volume upstream of, or within, the heated section. Pressure-drop oscillations have been studied in considerable details by Maulbetsch [24] and Griffith [25], for sub cooled boiling of water, and by Stenning et al. [26], [27], for bulk boiling of freon-11. Maulbetsch and Griffith found that the instability was associated with operation on the negative sloping portion of the pressure-drop - flow curve.

4.4. Simulation results - with IRBT

By converting the SKV3 boiler to a system equipped by RR5 internal rifled boiler tubes (IRBT), this will normally lead to a complete redesign of both the furnace- and the evaporator system, but in this fictive case we use the same heat transfer area, despite the fact, that the IRBT considerably improve the heat transfer in the boiling zone. In this new setup, the length of the boiler tubes are reduced from 193.5 [m] to 38.25 [m] and the number of parallel tubes are increased from the original 4 x 56 to 4 x 270 parallel tubes. We have proved used a very low mass flux (corresponding to approx. 10% load), specifically to analyze the effects of the wall temperature distribution. It should be emphasized that this simulation event is a fictional setup and is rather a calculation example of what can happen in an evaporator tubes, if near zero flow momentarily occurs.

The vertical IRBT leads to a decrease in the mass flux, which is illustrated in (7) for instant pictures of 100, 150, 175 and 200 [s] of simulation. The wall temperature are varying in time and reach a peak while the flow locally is approaching zero, caused by local pressure oscillations initiated by the compressibility at the entrance of the two phase region. The bad cooling caused by near zero flow can have disastrous consequences for the pipe material and may ultimately lead to a meltdown of the evaporator tube. In practice, this is avoided by increasing the circulation through the evaporator. The pressure drop through the evaporator tube is unrealistically low, due to the very low mass flux (105 [kg/m²s]). Normally, the mass flux of IRBT is approx. 1200 [kg/m²s] at 100% load. In figure (8) is listed timeseries of the thermo hydraulic data at two stations located at A(z=1/4L) and B(z=3/4L). The thermo hydraulic conditions in station A is situated in the subcooled region while the station B is situated in the super heated region. Both stations are affected by the compressibility effect, initiated in the entrance to the boiling zone. Pressure waves are approaching up- and down stream due to the eigenvalues of the hyperbolic governing equations (λ₁=±c, λ₂=±u±c and λ₃=±u-
c) where $\lambda_i$, $i=1,3$ is the eigenvalues and $c$ is the local speed of sound for the two phase mixture. In the downstream station B we can also see minor slugs of enthalpy for $t=100$ [s], which also is referring to the compressibility phenomena.

4.5. Model consistency

The model consists of 400 differential elements, thus ensuring a smooth continuous solution. By reducing the number of computational cells to only 50 elements, one would observe a more intensive standing wave at the entrance of two-phase region, which is due to intensive heating of the differential cell in the vicinity of the boiling zone, where we have an intensive negative slope in the density as function of the enthalpy, hence the density change becomes so violent that a pressure wave is established to ensure momentum balance. Using a CFL number higher than 1.0 is leading to instabilities due to the semi implicit scheme.

4.6. Discussion

The two simulation cases shows two very different thermo hydraulic conditions. The simulation of (SKV3) without IRBT is verified against steady state measurements and the pressure drop and heat uptake fits quite well ($\pm 5 \%$). The case of IRBT does not reach a steady state condition after 200 [s] and is illustrating an absolute worst case of boiler layout. It is interesting to see that it is possible to initiate local temperature spikes in an evaporator tubes - even before the boiling region - caused by the compressibility phenomena. The above results show that the numerical model is able to simulate the pressure drop and heat transfer in evaporator tubes (with and without IRBT), in both a time and spatial resolution. However, despite the extremely large in-linearities in the fluid density, and the hyperbolic nature of the governing equations, the model is capable to calculate a dynamic response over the saturation zones in the evaporator. Under normal conditions, the sub-cooled section of the evaporator will be separated from the two-phase section, to ensure numerical stability, but by use of the WENO technique, this can be handled in one setup. It is unfortunately not possible to compare the numerical calculations with measured data, since the IRBT evaporator model is a hypothetical example, but the boundary data are taken from measurements from SKV3.

It is interesting to see how the tube wall temperature may be increased, as a result of poor heat transfer due to the low flow rate in the subcooled section of the evaporator. Further downstream, where the flow speed increases, progressively better heat transfer are observed and a more homogeneous axial temperature distribution all the way down to the superheated section, where the material temperature rises again.

Similarly, we can observe that there are several different models of the wall friction into play, which is revealed by considering the slope of the pressure downstream in figure (7). The pressure gradient is ultimately the greatest in the two-phase region, due two-phase multiplier. we see also that the pressure gradient for superheated steam also, not surprisingly, are larger than sub-cooled liquid.

The Central WENO schemes are designed for problems with piecewise smooth solutions containing discontinuities. The Central WENO scheme has been successful in the above applications, especially for solving the pressure distribution down streams an evaporator. The inlet conditions is sub cooled water and the out flow is superheated steam. Minor pressure waves are initiated in the transition zones to the two phase region ($x=0$), because of the compressibility of the fluid. The pressure oscillations generated in the entrance to the boiling zone is controlled by the shear stresses in the momentum equation ($0.01 \ [m^2/s]$), which smooth the oscillations due to diffusion of momentum. The model is very time consuming in solving the system, because the total energy is determined iteratively as well as the density is a function of pressure and enthalpy. The model is stable as long as the CFL number is less than one and the speed of sound is below the highest calculated speed of sound in the fluid domain, determined at each time step. We can conclude that the solution procedure is non-oscillatory in the sense of satisfying the total-variation diminishing property in the one-dimensional space. No numerical wiggles are observed in the hyperbolic models and smooth solutions are observed in the continuous zones of the flow regimes.
4.7. Conclusion

In this article we have solved the dynamic flow equations and associated wall model for a boiler tube, by use of a fifth order WENO scheme. Simulations with and without a model of the inner rifling of the boiler tube has been carried out. The calculations include the entire evaporation process from sub-cooled water to super-heated steam, which includes a massive change in fluid density downstream. The simulations show that there is a very large pressure drop across the boiler tube without rifling, while the tube with rifling has a significantly lower pressure drop, due to the lower mass flux, although the relative pressure drop in the rifle tube is significantly higher compared to the smooth boiler tube. We also see that the mass flux in IRBT for design reasons are significantly lower. The model handles perfect the pressure oscillations occurring in the two phase region, as a result of the increased compressibility of the fluid. This instability generates minor enthalpy slugs downstream in the calculations. In the IRBT simulations we experience very low mass flux just before the entrance to the two-phase region, which locally gives a very poor cooling of tube wall and rising wall temperature. We can generally conclude that WENO scheme both numerically and in terms of stability is well suited to solve such an complicated hyperbolic system of PDE’s with respect to the transformed independent solution parameters.
References