der Bundeswehr
Universität München

Numerical Computation of Two-Phase Flow in Porous Media

D. Droste, F. Lindner, Ch. Mundt, M. Pfitzner
Institut für Thermodynamik, Universität der Bundeswehr München

❖ Introduction

In burner with small thermal power evaporates the fuel in a porous material. To simulate this complex process on a macroscopic scale, the multiphase mixture model is implemented in COMSOL.
In Figure 1 we illustrate a simplified model for the computation.

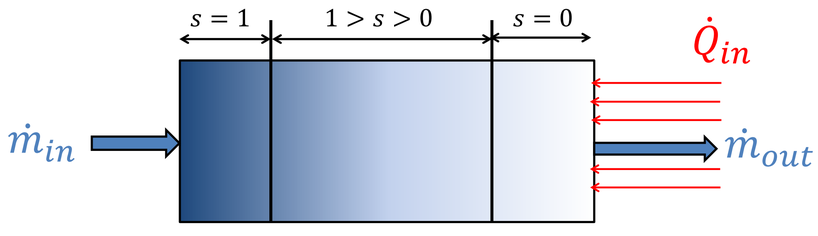

Fig. 1: 1D Model

❖ Two-Phase Mixture Model

The two-phase mixture model (TPMM) introduced by Wang et al [1] is based on the assumption of the local thermal equilibrium (LTE), i.e. T_s=T_f. Because of the assumption generates the model a numerical isolation domain. This variant is a one-equation model.
The second variant of this model is a two-equation model, which is presented by Shi et al. [2]. This variant drops the assumption of the local thermal equilibrium and is called the local thermal non-equilibrium model (LTNE).

$\nabla \cdot (\rho \boldsymbol{u}) = 0$	(Mass conversation)
$\boldsymbol{u} = -\frac{K}{\mu}(\nabla \mathrm{p} - \rho_k \boldsymbol{g})$	(Darcy's Law)
$\nabla \cdot (\gamma_h \boldsymbol{u} H) = \nabla \cdot (\Gamma_h \nabla H) + \nabla \cdot \left(f \frac{K \Delta \rho h_{fg}}{\nu_v} \boldsymbol{g}\right)$	(Energy-equation, LTE)
$\nabla \cdot (\gamma_h \boldsymbol{u} H) = \nabla \cdot (\Gamma_h \nabla H) + \nabla \cdot \left(f \frac{K \Delta \rho h_{fg}}{\nu_v} \boldsymbol{g}\right)$	(Energy-equation fluid, LTNE)
$q_{sf} = \nabla \cdot \left(\nabla k_{s,\mathrm{eff}} \nabla T_s\right)$	(Energy-equation solid, LTNE)

Tab. 1: Conservation equations of TPMM

Löser 1	MATLAB	FV-Code
Löser 2	COMSOL	FE-Code

Tab. 2: Self written solver for the TPMM

❖ Implementation and Results

	$\dot{m}_{in}$	$\dot{Q}_{in}$	Medium
BC 1	$0.3 \frac{kg}{m^2 s}$	$1e6 \frac{W}{m^2}$	Water
BC 2	$0.5 \frac{kg}{m^2 s}$	$2e6 \frac{W}{m^2}$	Water

Tab. 3: Boundary Conditions

We implemented the TPMM in Matlab and COMSOL (Tab.2). In Comsol we used the PDE-Interface and the Darcy's Law module to implement the conservation equations (Tab.1).
A main problem in the implementation process was the diffusion coefficient. Because of the phase change the coefficient has great jumps at the boundaries between the two-phase domain and the liquid/gas domain (cf. Fig 2).

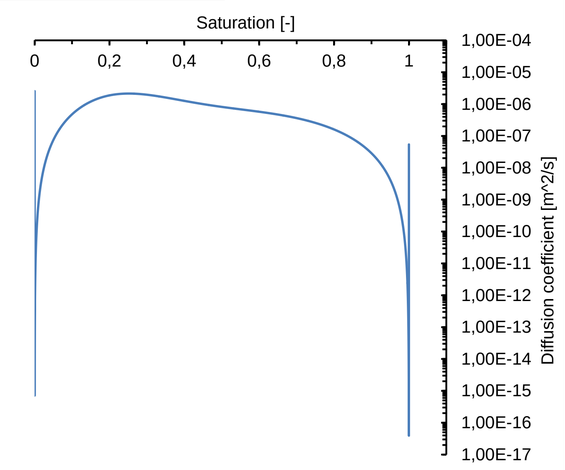

Fig.2: Diffusions Coefficient

Boundary-Conditions	$\frac{T_{f,num.}\|_{y=L}}{T_{f,analyt.}\|_{y=L}}$ Shi et al. [2]	Solver 1	Solver 2
BC 1	0.94	0,99	0,99
BC 2	0,98	1,00	1,00

Tab. 4: Comparison numerical/theoretical temperature

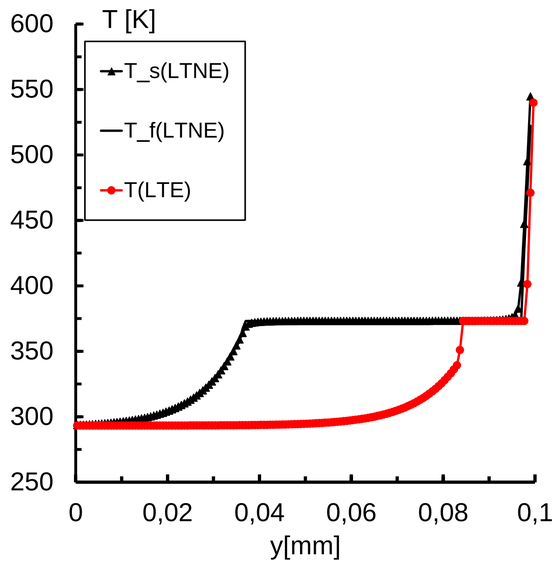

Fig. 3: Comparison of the fluid temperature: LTE vs. LTNE

1D Results:

1. LTE vs LTNE

In figure 3 are shown the results for the LTE und LTNE approach. The LTE model has a very short two-phase domain, because of the isolation, which is caused by the assumption of the approach.
The temperature at $x = l$ is for both methods almost the same. The reason for this outcome is logical, because both models use the same theory to compute the temperature of the fluid:

$$T_{f,analyt.}\Big|_{y=L} = \begin{cases} T_{in} + \frac{\dot{Q}_{in}}{\dot{m}\, c_{p,l}} \\ T_{sat} \\ T_{sat} + \frac{\dot{Q}_{in} - \dot{Q}_{dryout}}{\dot{m}\, c_{p,v}} \end{cases}$$

2. LTNE: Shi vs Solver2

The greatest difference between the two results of figure 4 A und B is the length of the two-phase domains. The reason for this variance is based on the diffusion coefficient. Because of jumps at the boundaries of the two-phase domain, we need to smooth the function of the coefficient. This is done on different ways. The result of the smoothing pushes the two-phase domain upwind .
If we compare the numerical temperature of the fluid with the theoretical temperature, we see (Tab. 4), that the Solvers 1 + 2 deliver better results than the solver written by Shi et al. [2].

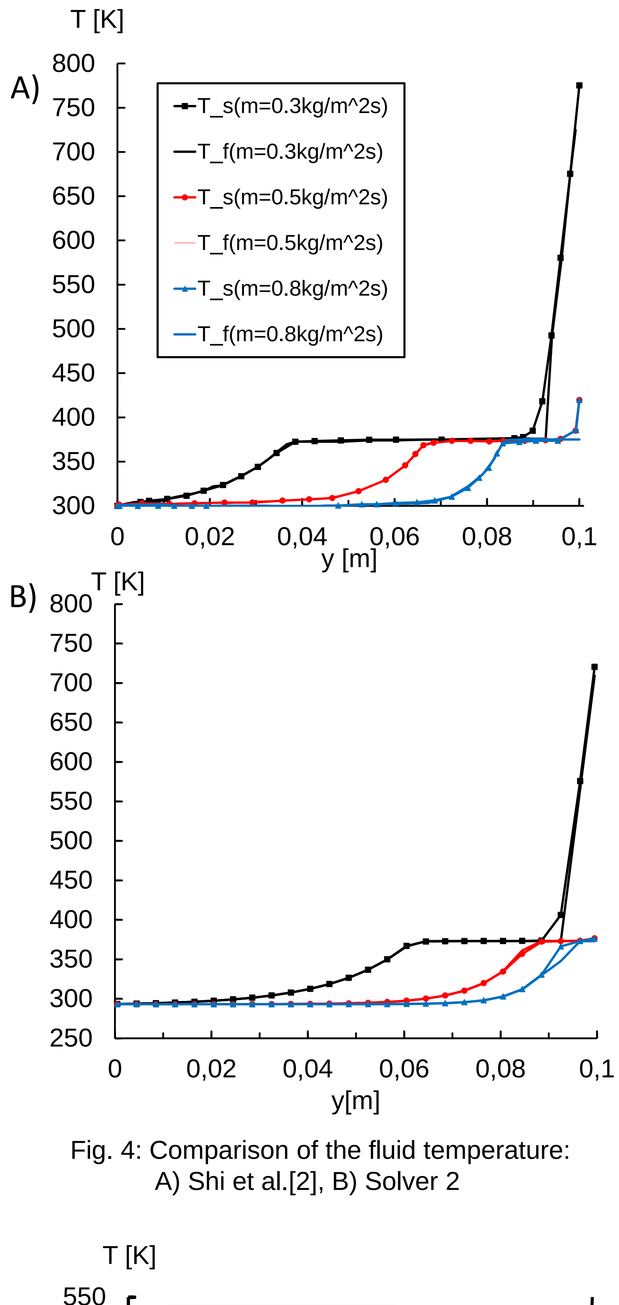

Fig. 4: Comparison of the fluid temperature: A) Shi et al.[2], B) Solver 2

2D Results:

The sketch of the two dimensional problem is displayed in figure 6. Gravitational aspects were neglected.
A nice aspect of this mixture model is, that we can extract the flow characteristics of the individual phases.
In figure 7, we can see the phase velocity of the two phases. At the inlet the phase velocity of the gaseous phase is zero, because the temperature is below the boiling temperature. At the outlet are both velocity unequal zero. The reason for this observation is that there is the two-phase domain. The mean velocity is density based:

$$\rho \boldsymbol{u} = \rho_l \boldsymbol{u_l} + \rho_v \boldsymbol{u_v}.$$

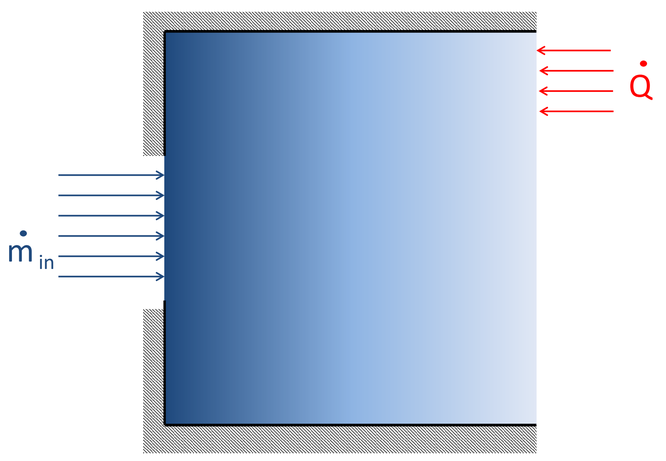

Fig. 6: 2D Model

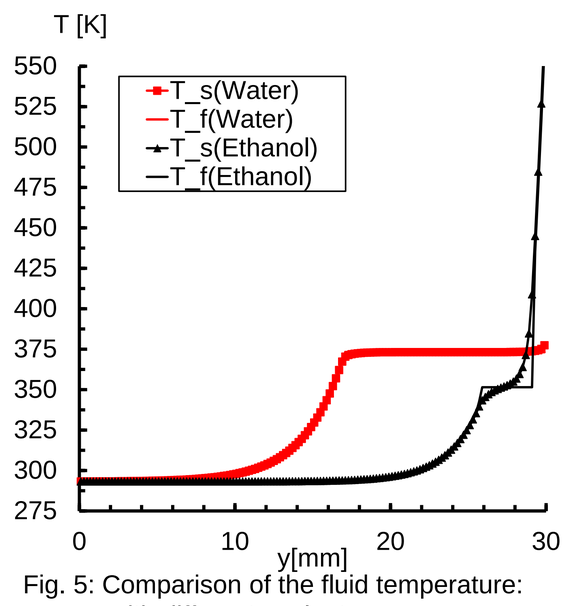

Fig. 5: Comparison of the fluid temperature: with different coolants

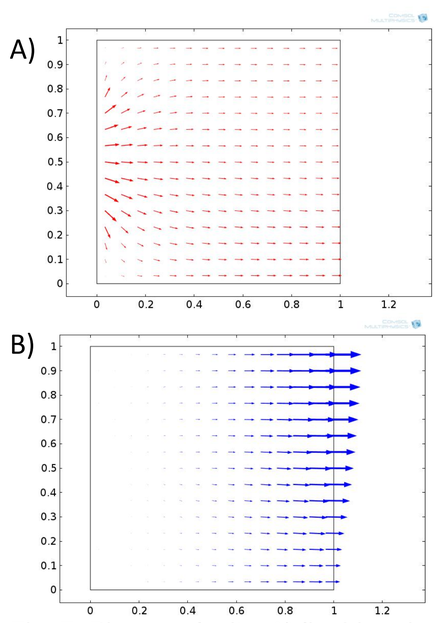

Fig. 7: Phase velocity: A) liquid and B) gaseous

❖ Conclusion and Future Work

The Two-Phase Mixture Model by Wang (LTE) and Shi (LTNE) were implemented in the COMSOL environment. The model was used to compute simple one-dimensional problems. In figure 6 first results for a two-dimensional problem are displayed.
The next steps to taken are a experimental validation. Further more we need to improve the two-dimensional model. To consider non-constant fluid-temperature in the two-phase domain, the model of Wei et al [3] is a promising option.

Literature: [1] C.Y. Wang and C. Beckermann, A two-phase mixture model of liquid-gas flow and heat transfer in capillary porous media, Int. Journal Heat Mass Transfer, 1992.
[2] J.X. Shi and J.H.Wang, A numerical investigation of transpiration cooling with liquid coolant phase change, Transp. Porous Media, 2010.
[3]K. Wei, J. Wang and M.Mao, Model Discussion of Transpiration Cooling with Boiling, Transp. Porous Media, 2012.

Excerpt from the Proceedings of the 2013 COMSOL Conference in Rotterdam