Thermally activated wall system with latent heat thermal energy storage – comparison of 1D and 3D model

Pavel Charvat[1], Lubomir Klimes[1,2], Milan Ostry[2]
[1]Faculty of Mechanical Engineering, Brno University of Technology
[2]Faculty of Civil Engineering, Brno University of Technology

Abstract

The paper deals with a comparison of the 1D and 3D model of a thermally activated wall panel with a phase change material. The 3D model was created in the off-the-shelf simulation tool COMSOL and the 1D model was an in-house developed TRNSYS type. The main advantage of the 1D model is the short computation time. It, however, comes at the expense of lower accuracy and less detailed results. In most building performance simulations, the detailed knowledge of the temperature distribution over the surface of a wall is not important. For this reason, the results of both models were compared in terms of the average surface temperature of the wall panels, the outlet water temperature and the overall heating and cooling capacity of the panels. The two models provided very similar results for the mass flow rates of water from about 0.001 kg/s but the discrepancies increased with the decreasing mass flow rate.

Introduction

Thermally activated building systems (TABS) have attracted a lot of attention in connection with low-temperature heating and high-temperature cooling of buildings. Large surface areas of the TABS make it possible to use the heat transfer fluid (HTF) temperature much closer to the room temperature than in case of other systems, such as fan coil units. That makes TABS suitable for the use with heat pumps. A certain disadvantage of the TABS is that they need to be considered in an early stage of building design and their application in building retrofits is more complicated.

Several building simulation tools provide models of thermally activated building systems or structures. However, some of these models are only usable with sensible heat storage materials. A comparison of the embedded-tube radiant floor models in building performance simulation (BPS) tools was presented by Brideau et al. (2016). The authors used TRNSYS, ESP-r and EnergyPlus to simulate the behavior of a hydronic radiant floor and they compared the results with the finite element analysis performed in the LISA software. The authors reported discrepancies between the results of the BPS tools.

The heat transfer problem becomes more complex when the TABS are combined with latent heat thermal energy storage (LHTES). The simulation model in such case needs to account for the transfer and storage of latent heat. There are three basic approaches to modelling of phase change problems: the enthalpy method, the effective heat capacity method and the temperature recovery method.

Mazo et al. (2012) reported modeling of a radiant floor system with phase change material (PCM). The authors used the one-dimensional finite difference scheme in their model with the effective capacity method applied for PCM modeling. The authors compared the results of their model with the results of ESP-r but only for a case without the PCM and without thermal activation.

A numerical model for walls with phase change materials implemented in TRNSYS was presented by Delcroix et al. (2017). The developed model used explicit finite-difference method for the solution of the conductive heat transfer equation and the enthalpy method was employed to address temperature dependent heat capacity of the PCM. Both the phase change hysteresis and supercooling of the PCM were considered in the model.

Weinläder et al. (2016) reported experimental testing of two types of cooling ceilings containing PCM. The celling designs differed in the position of the PCM relative to the pipes for the HTF. The first type of the ceiling had the PCM positioned above the pipes while the second type of the ceiling had the PCM positioned below the pipes (on the side of the indoor environment). The authors only reported the results for passive cooling operation mode where both types of the cooling ceiling exhibited similar performance in terms of cooling powers at various room temperatures and room cooling loads.

The present paper deals with the 1D and 3D models of the thermally activated wall panels with plaster containing encapsulated PCM.

Modelled TABS with LHTES

The simulated TABS with latent heat thermal energy storage consisted of wall panels with embedded plastic tubes for the liquid heat transfer fluid, the distribution system for the HTF, and a reversible air-to-water heat pump for heating and cooling of the HTF. The TABS was intended for the use in lightweight buildings and thus the panels were designed in a way to be mounted on the indoor surfaces of building structures.

https://doi.org/10.26868/25222708.2017.026

The wall panels consisted of baseboards (made of Oriented Strand Boards - OSBs) with thermal insulation layer on one side of the board and the plaster containing microencapsulated phase change material on the other side. The plastic tubes for the HTF were embedded in the plaster. The plaster layer was 15 mm thick. Each panel contained 17 plastic tubes connected in the U-shaped manner with the supply and return pipe on one side of the panel. The tubes had the inner diameter of 2.25 mm and the wall thickness of 0.5 mm. The distance between the tubes was 15 mm. A schematic view of the wall panel is in Fig. 1.

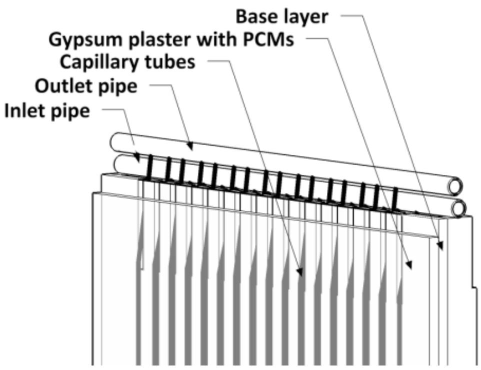

Figure 1: Wall panel

The actual TABS was assembled and tested in one of the university laboratories but the present paper utilizes only the basic concept of the wall panels and no experimental data are presented in the paper.

Simulation models

The main goal of the presented simulation study was the comparison of the results obtained with the 1D and 3D model. The models were compered in terms of the overall heat transfer rates between the panels and the surrounding (indoor) environment and in terms of the discrepancies between surface temperatures and outlet water temperatures. The comparison was done for both steady state and transient boundary conditions in the heating and cooling operation of the panels.

Two configurations of the embedded tubes were considered (Fig. 2). In the first configuration, the supply pipe for the HTF was at the top of the panel and the return pipe at its bottom with the embedded tubes running straight from the supply pipe to the return pipe. Each of the plastic tubes was 2 meters long in this configuration.

In the second configuration, the supply and return pipes for the HTF were at the top of the panels and the embedded tubes were connected in the U-shaped manner (U-tubes). As can be seen Fig. 2, the U-tubes were twice as long as the straight tubes, it means that each tube was 4 meters long.

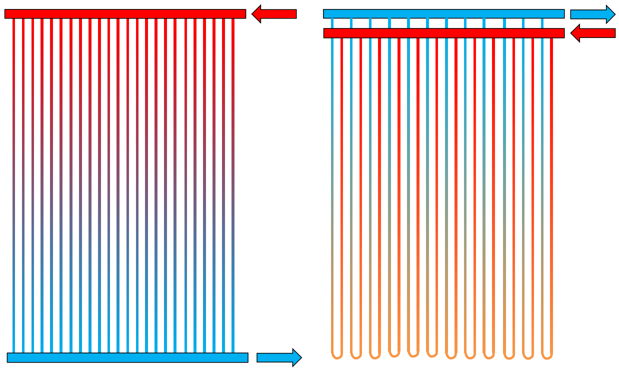

Figure 2: Embedded tube arrangements

The pressure drop of the panels was not analyzed in the study but it is obvious that the pressure drop of the panel with the U-tubes is higher than in case of the panel with the straight tube. The tubes have quite a small diameter and the water flow in the tubes is laminar for the practical range of the water flow rates. The critical Reynolds number (Re_D = 2000, the onset of transition to turbulent flow) is reached at the water velocity of about 0.7 m/s (mass flow rate of about 0.003 kg/s).

Both cooling and heating operation of the panels was simulated. Two arbitrary phase change temperature ranges of the PCM were considered.

The temperature interval of phase change from 20 ˚C to 24 ˚C was considered in the heating scenario. The ambient (room) air temperature was set to 20 ˚C throughout the simulation in this case. The initial plaster temperature was the same as the ambient temperature. The step change of water temperature was introduced as shown in Fig. 3. The mass flow rate of water was constant during the simulations. The initial water temperature was considered the same as the ambient temperature (20 ˚C) and then it was increased to 35 ˚C.

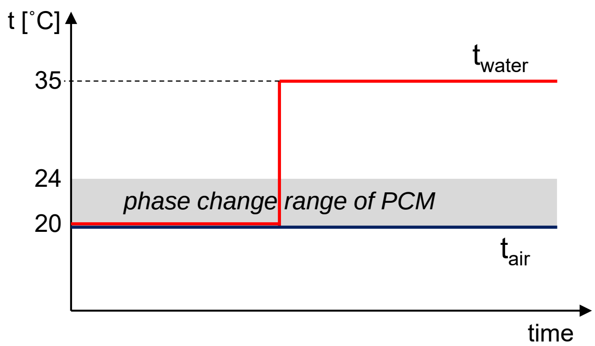

Figure 3: Heating scenario

The cooling operation scenario was defined in a similar manner. The ambient air temperature was set to 26 ˚C and the phase change range of the PCM was between 22 ˚C and 26 ˚C. The water temperature changed from 26 ˚C to 18 ˚C as shown in Fig. 4.

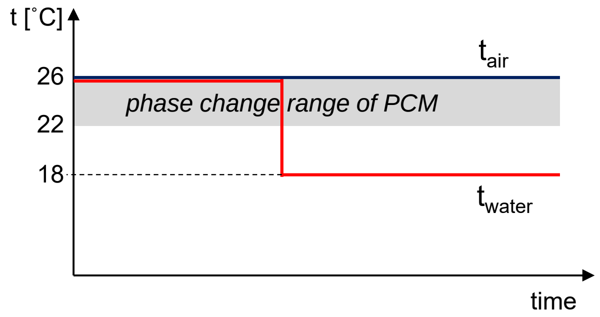

Figure 4: Cooling scenario

Simulations for several mass flow rates of water were performed in each scenario. The effective heat capacity method was used in both 1D and 3D model. It means that the latent heat of the encapsulated phase change material was included in the specific heat capacity of the plaster with PCM. Most PCMs melt and solidify in a certain temperature intervals rather than at constant temperature. As shown in Fig. 3 and Fig. 4 the temperature interval of the phase change was considered 4 K in both heating and cooling scenario. The curve of the effective heat storage capacity was expressed with the following equation (Kuznik et al., 2008)

$$c_{eff} = c_0 + c_m \exp\left\{ -\frac{(T - T_{pch})^2}{sigma} \right\} \qquad (1)$$

where c_0 is specific heat, c_m is the maximum increment of the specific heat due to the latent heat, T_{pch} is the mean temperature of phase change (the peak of effective heat capacity) and *sigma* is a parameter that influences the width of the phase change temperature range.

Eq. (1) can be used in situations where a PCM is incorporated with other material, such as the presented case of encapsulated PCM embedded in the gypsum plaster. The effective heat capacity as a function of temperature for the plaster with PCM that has the mean phase change temperature of 22 ˚C is shown in Fig. 5.

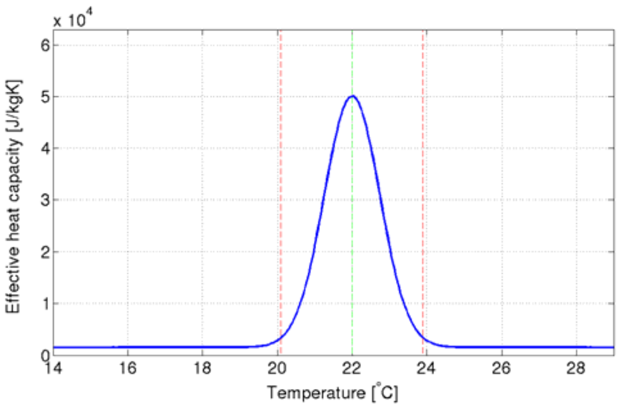

Fig. 5: Effective heat capacity of the PCM as a function of the temperature

Eq.(1) provides a flexible approximation of the distribution of effective heat capacity in a certain temperature range. As the change of the T_{pch} does not influence the melting range or latent heat, Eq. (1) is suitable in situations where the influence of T_{pch} is to be investigated independently of other parameters.

1D model

All heat transfer problems encountered in the real world are 3-dimensional (3D) and their transformation to a 1-dimensional (1D) case requires certain simplification. In case of the wall panels, the heat conduction from the tubes with the HTF to the surrounding plaster is 3-dimensional. A simplification to a 1D problem means that heat conduction is considered only in one direction. In this case, heat conduction with a heat source was considered only in the direction of the thickness of the plaster (Fig. 6). The baseboard and the thermal insulation layer on the backside of the wall panel were modeled as a thermal resistance.

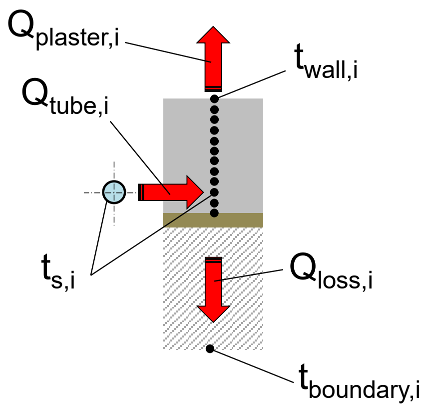

Figure 6: 1D conduction in i-section

The temperature of the HTF in the plastic tubes changes along the length of the tube. The HTF temperature (water temperature in the studied case) was calculated using the energy balance shown in Fig. 7. It was assumed that the mass flow rate of water and the inlet water temperature were the same for all tubes embedded in one wall panel. Therefore, only one tube and the relevant section of the panel was considered in the model. As 1D heat conduction was considered in the direction of the plaster, it was not possible to model the tube itself and only the corresponding heat flux was used as an input in the computational node where the tube was located (a heat source in the node). The heat flux from water to the plaster was calculated with the assumption that the temperature of the outer surface of the tube was the same as the temperature of the plaster at the position of the tube (node temperature). Correlations from the literature (Incropera et al., 2013) were used to calculate the heat transfer coefficient on the internal side of the tubes. The model was written in C++ and implemented as a TRNSYS type.

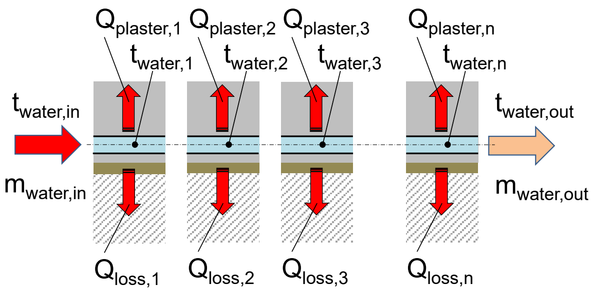

Figure 7: Schematic of the 1D model

The effective heat capacity method, unlike the enthalpy method, does not ensure energy conservation. The accuracy of the heat capacity method, in this respect, depends on the time step. The accuracy generally decreases with the increasing time step. The internal time step of 0.1 second was used in the model of the panel regardless of the TRNSYS simulation time step.

A simple validation test was conducted to assess the accuracy of the effective heat capacity method in this case. The adiabatic boundary condition was applied on the surface of the panel facing the room as well as the surface facing the wall. In such a case both the heat flux to the room and the heat flux to the wall (heat loss) are equal to zero and all heat delivered (rejected) by flowing water is stored in (released) from the PCM plaster. The total amount of heat stored in (released from) the plaster for the step change of water temperature (as shown in Fig. 3 and Fig. 4) can easily be calculated analytically and compared with the numerical results. Very good agreement between the analytical and numerical results was achieved in this test.

3D model

The 3D model was created in the COMSOL simulation software. The COMSOL software is a general FEM-based simulation tool, which allows for modeling of various problems including multi-physical phenomena. The coupled heat transfer with phase change and the fluid flow phenomena were included in the model. The diameter of the tubes is quite small and the temperature of the flowing water is much more important in the investigated case than the detailed knowledge of the fluid flow pattern. The non-isothermal pipe flow module was therefore used instead of the fluid flow module. The use of the pipe flow module means that the simplified non-isothermal 1D fluid flow is computed in the tubes, instead of 3D fluid flow. The computation of the 3D fluid flow would require very fine mesh in the tubes, it would be computationally much more demanding, but it would not provide much additional information about the thermal behavior of the wall panels.

Two geometries were created. One representing a segment of the panel with the straight tubes and the other representing a segment of the panel with the U-tubes. Both segments represented a 30 mm wide section of the panel as shown in Fig. 8 and Fig. 9. The sections are repetitive segment of the wall panel. In case of the panel with the straight tubes, two adjacent tubes were considered and the adiabatic (no heat flux) boundary condition was applied at the sides of the segment (planes of symmetry). As for the panel with the U-tubes, the modeled segment contained only one U-tube (Fig. 9). The periodic boundary conditions imposing the identical temperature distribution and the inverse heat fluxes were applied at the sides of the segment.

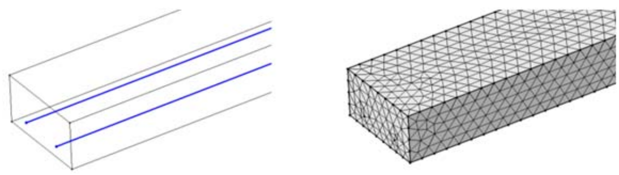

Figure 8: Upper part of the considered section: both cases

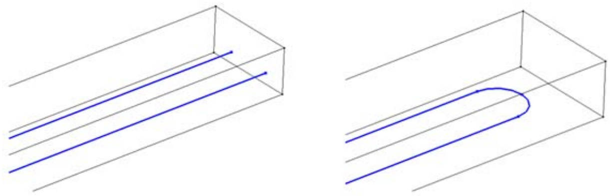

Figure 9: Lower part of the segments: the case with straight tubes (left) and the case with the U-tubes (right)

The effective heat capacity method was used for the modelling of phase change and the laminar flow was assumed in the tubes. The mesh of the PCM domain (shown in Fig. 8) for the case with two straight tubes and with the U-tube consisted of 214 000 and of 224 000 of tetrahedral elements, respectively.

Results and discussion

The main goal of the comparison of 1D and 3D model was to assess the influence that the simplification to 1D problem had on the results. The temperature distribution in the slab is not usually important in building performance simulations of the TABS. The most important results are the surface temperatures and the overall heating and cooling power. The simulations were carried out for the same height of the wall panels. The embedded tubes were 2 m long in case of the panel with the straight tubes and 4 m long in case of the panel with the U-tubes.

Steady state results

Steady state heating scenario was calculated for the inlet water temperature of 35 °C and the ambient air temperature of 20 °C (room temperature). The average surface temperature (surface of the panel facing the room), outlet water temperature and the average heat flux density of the wall panel with the straight tubes is in Fig. 10.

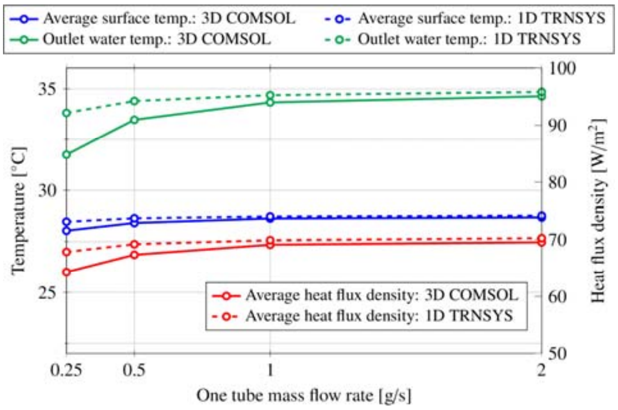

Figure 10: Steady state heating – straight tubes

As can be seen in the chart, from the certain mass flow rates of water both the average surface temperature and the average heat flux density do not increase very much. As the pressure drop of the tubes increases with the square of water flow rate, it would not make much sense to increase the flow rate of water about a certain value.

The results for the cooling scenario are shown in Fig. 11. The water inlet temperature of 18 °C and the ambient air temperature of 26 °C were considered in this case. For practical reasons the heat flux density is shown as a positive value even though the heat flux has the opposite direction to the heating mode operation.

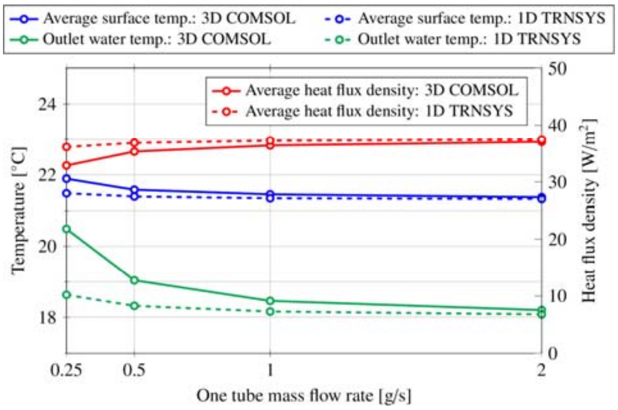

Figure 11: Steady state cooling – straight tubes

The results of the 1D and 3D model are in a relatively good agreement for the mass flow rates of 0.5 g/s and higher. A panel with the height of 2 m containing 34 straight tubes would have the surface area of about 1 m^2. The mass flow rate of 0.5 g/s per tube means the mass flow rate of about 61 kg/hour for the entire panel.

The results for the heating operation of the panel with the U-tubes are in Fig. 12. The discrepancy between the results of the 1D and 3D model is higher than in case of the panel with the straight tubes. The higher discrepancy in this case logical as the 1D model does not address heat transfer between the adjacent tubes. However, the discrepancy decreases with the increasing mass flow rate as the increasing mass flow rate means lower difference between the inlet and outlet water temperature.

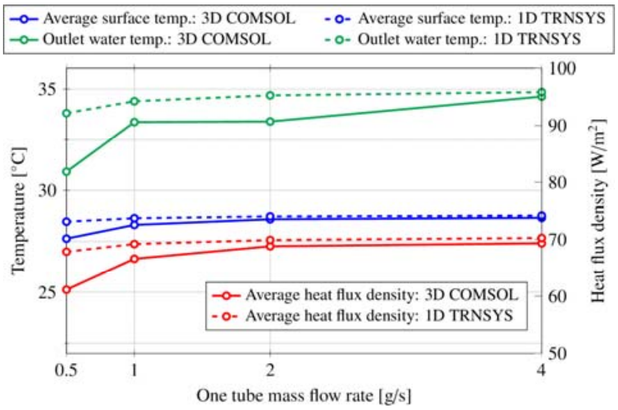

Figure 12: Steady state heating – U-shaped tubes

The cooling mode operation of the panels with the U-tubes exhibited the largest discrepancies between the 1D and 3D model. The simulated cooling mode is characteristic with relatively small difference between the cooling water temperature and ambient temperature (as shown in Fig. 4.). Small absolute differences between the calculated variables thus translate into large relative differences. Similar to other scenarios the discrepancies are decreasing with increasing water flow rate.

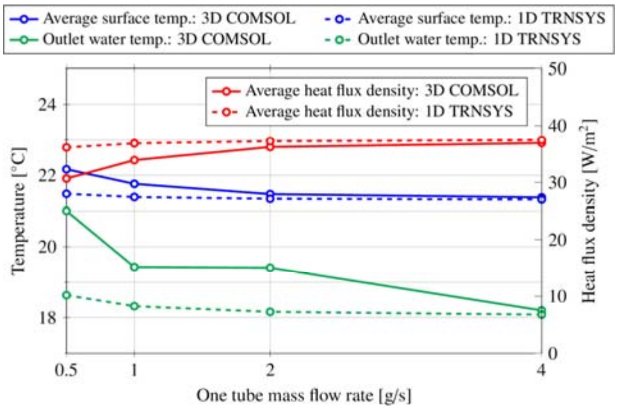

Figure 13: Steady state cooling – U-tubes

Transient results

The results of the 1D model of the panel with the straight tubes for the heating scenario with the step increase of water temperature (Fig. 3) are shown in Fig. 14. The transient simulations reveal the operation dynamics of the panels. The step change of water temperature took place at 0.25 hour. The presence of the PCM significantly influenced the increase of the surface temperature at the early stage of heating operation. As can be seen, the response time of the wall panel significantly depended on the mass flow rate of water. For the mass flow rates of water higher than 0.25 g/s the panel reaches the steady-state operation in less than 1.5 hour. However, in case of the air flow rate of 0.1 g/s it took almost 3 hours to reach steady-state operation.

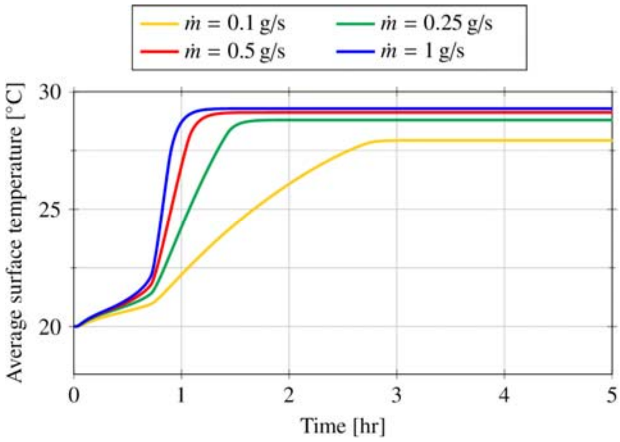

Figure 14: Transient heating – straight tubes

The results for the cooling scenario (Fig. 4) can be seen in Fig. 15. The time needed to reach the steady-state operation is longer then in case of the heating scenario.

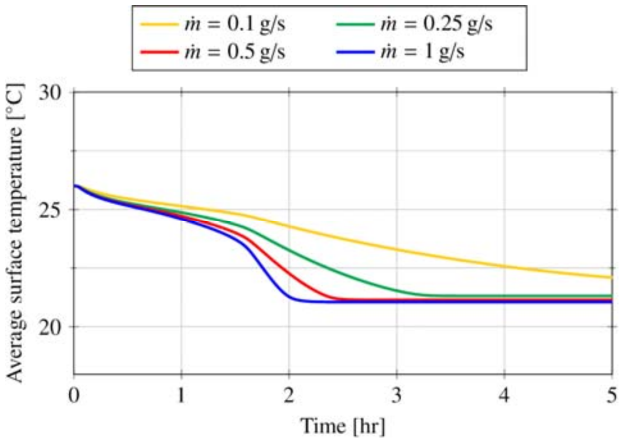

Figure 15: Transient cooling – straight tubes

Conclusion

The results of the 1D and 3D model of the thermally activated wall panel with phase change materials were compared in terms of the average surface temperature, the outlet water temperature and the average heat flux density. For the mass flow rates of water above 0.001 kg/s the two models provided very similar results, however, the discrepancies increased with the decreasing flow rate. The discrepancies were higher for the cooling scenario and for the panel with the U-tubes. From a certain mass flow rate of water further increase of the flow rate does not significantly influence the heating or cooling capacity of the panels. The developed models can be used for optimization of the panels in terms of both design and operation.

Acknowledgement

This work was supported by the Czech Science Foundation under contracts GA15-19162S *Research and development of heat storage media based on the phase change materials for higher energy performance of buildings*. The authors also gratefully acknowledge funding from the Ministry of Education, Youth and Sports of the Czech Republic under the National Sustainability Programme I (Project LO1202 NETME CENTRE PLUS).

References

Brideau, S.A. Beausoleil-Morrison, I., Kummert, M. Wills, A. (2016). Inter-model comparison of embedded-tube radiant floor models in BPS tools, *Journal of Building Performance Simulation*, Vol. 9, Issue 2, pp. 190-209.

Delcroix, B. Kummert, M. Daoud, A. (2017). Development and numerical validation of a new model for walls with phase change materials implemented in TRNSYS. *Journal of Building Performance Simulation.*

Mazo, J., Delgado, M., Marin, J.M., Zalba, B., (2012) Modeling a radiant floor system with Phase Change Material (PCM) integrated into a building simulation tool: Analysis of a case study of a floor heating system coupled to a heat pump. *Energy and Buildings* 47, pp. 458–466.

Incropera, F.P., DeWitt, D.P., Bergman, T.L., Lavine, A.S. (2013) Principles of Heat and Mass Transfer, 7th ed., Wiley & Sons, New York.

Kuznik, F. Virgone, J.Roux, J.J. (2008) Energetic efficiency of room wall containing PCM wallboard: a full-scale experimental investigation, *Energy and Buildings* 40, pp. 148–156.

Weinläder, H., Klinker, F., Yasin, M., (2016) PCM cooling ceilings in the Energy Efficiency Center—passive cooling potential of two different system designs. Energy and Buildings 119, pp. 93–100.