ENHANCED STORAGE TANK SIMULATIONS USING DYNAMIC FLUID MODELING WITH MODELICA

Marco Bonvini[1], Alberto Leva[1], Vladimir Vukovic[2], and Stefan Wischhusen[3]
[1]Politecnico di Milano, Dipartimento di Elettronica e Informazione, Milano, Italy
[2]AIT Austrian Institute of Technology, Energy Department, Vienna, Austria
[3]XRG Simulation, GmbH, Hamburg, Germany

Keywords: storage tank modeling, computational fluid dynamics, Modelica simulations

ABSTRACT

The paper proposes storage tank models that are informative enough to study and assess the correct storage management - a relevant issue for energy efficiency - while being at the same time computationally efficient, and easy to integrate with models of different physical domains. The result is obtained by a convenient use of object-oriented modelling, via the Modelica language. After presenting the model, a simulation study is shown and discussed.

INTRODUCTION

In modern control and management systems targeted at increasing building energy efficiency, storages play a significant role (Ma et al., 2012). To design such systems, accurate modelling and simulation of energy storage components is required. Storage tank models need to account for spatial distribution of the contained fluid properties in order to adequately represent stratification, effects of fluid injection/extraction and thermal exchange with the containment. Therefore, the required models are natively created as systems of *partial* differential equations (PDE). To capture the phenomena of interest, *three-dimensional* (3D) simulations are needed, traditionally falling into the realm of computational fluid dynamics (CFD) tools. However, CFD is limited in terms of coupling the models from different simulation domains such as electrical, mechanical, or thermo-hydraulic (Djunaedy et al., 2003; Gu and Asada, 2004).

As an alternative, usage of object-oriented modelling and simulation (OOMS), specifically by means of Modelica equation-based language, was recently proposed (Ljubijankic et al., 2011). Adopting suitable spatial discretisation schemes for the Navier-Stokes equations results in compact computationally efficient models that natively integrate in multi-domain simulations (Bonvini et al., 2012; Bonvini and Popovac, 2012). In the current study, such novel approach is applied to modelling and simulation of storage tanks.

METHODOLOGY

The Governing Equations

The motion of fluid is described by the equations of mass, energy and momentum balance, and this set of equations is often referred to as the Navier-Stokes equations. In the case of the Newtonian fluid they can be written as:

Mass balance

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \qquad (1a)$$

Energy

$$\frac{\partial (\rho e)}{\partial t} + \nabla \cdot (\rho \vec{v} h) = \nabla \cdot (k \nabla T) \qquad (1b)$$

Momentum equation

$$\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \vec{v}^{\mathrm{T}}) + \nabla p = \nabla \cdot (\mu \nabla \vec{v}) + \vec{f} \qquad (1c)$$

where the scalars p, T, e, h, ρ, k and μ are respectively the fluid pressure, temperature, specific energy, specific enthalpy, density, thermal conductivity and dynamic viscosity; the vectors $\vec{v}$ and $\vec{f}$ are the fluid velocity and the the *external* forces, such as gravity, acting on the fluid.

Considering the scalar projections, and for simplicity analysing only 2D case, the momentum equation (1c) is decomposed into two scalar equations:

$$\begin{aligned} &\frac{\partial \rho v_x}{\partial t} + \frac{\partial \rho v_x v_x}{\partial x} + \frac{\partial \rho v_x v_y}{\partial y} = \\ &f_x - \frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left(\mu \frac{\partial v_x}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu \frac{\partial v_x}{\partial y}\right) \end{aligned} \qquad (2a)$$

$$\begin{aligned} &\frac{\partial \rho v_y}{\partial t} + \frac{\partial \rho v_y v_x}{\partial x} + \frac{\partial \rho v_y v_y}{\partial y} = \\ &f_y - \frac{\partial p}{\partial y} + \frac{\partial}{\partial x}\left(\mu \frac{\partial v_y}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu \frac{\partial v_y}{\partial y}\right) \end{aligned} \qquad (2b)$$

where the subscripts x, y denote the components of the 2D Cartesian coordinate system. In the case of natural convection having y-axis as vertical, holds $f_x = 0$ and $f_y = -\rho g$, where g is the gravitational acceleration. In order to solve numerically the momentum equation (as well the energy and mass balance), each of the terms need to be properly represented.

Modelling

The conservation equation for mass (1a), energy (1b) and momentum (1c) can be represented in the standard Convection-Diffusion (CD) form, which (again for simplicity) in the 2D case reads:

$$\underbrace{\frac{\partial \rho \Phi}{\partial t}}_{local} + \underbrace{\frac{\partial \rho v_x \Phi}{\partial x} + \frac{\partial \rho v_y \Phi}{\partial y}}_{convective} = \underbrace{\frac{\partial}{\partial x}\left(\Gamma_\Phi \frac{\partial \Phi}{\partial x}\right) + \frac{\partial}{\partial y}\left(\Gamma_\Phi \frac{\partial \Phi}{\partial y}\right)}_{diffusive} + \underbrace{S_\Phi}_{source} \quad (3)$$

where the generic quantity Φ is a scalar transported by the fluid moving with velocity $\vec{v} = (v_x, v_y)$, and Γ_Φ is the diffusivity coefficient. The time dependent variable Φ can be either a velocity component, the internal energy or the mass fraction of a chemical species. The source term S_Φ is the generation rate of the scalar quantity Φ per unit volume.

The generic CD equations (3) state that the (unsteady) local change of the scalar quantity Φ is equal to the sum of the convective change, the diffusive change, and the generation from a source. For example, replacing Φ with v_y, Γ_Φ with the viscosity μ and collecting in the source term S_Φ both the gravity $-\rho g$ and the pressure gradient $\frac{dp}{dy}$, the y-momentum equation (2b) is obtained.

Following the procedure for solving CD described in the literature (Patankar, 1980), the general CD equation (3) is spatially discretised over a grid of staggered Control Volumes (CV) as shown in Figure 1. The basic idea behind the staggered grid is to integrate the balance equations over differently selected CVs: the CVs for momentum equations are shifted with respect to those in which energy and mass preservation equations are solved. This is done to avoid numerical problems, resulting in unrealistic pressure/velocity distribution (Patankar, 1980; Versteeg and Malalasekera, 2007). As an additional advantage, the approach eases numerical implementation, as the CVs located in the P cells (see Figure 1) provide boundary conditions for the Vx and Vy cells, and vice versa. Without the staggered approach, interpolation between the adjacent cells would be needed.

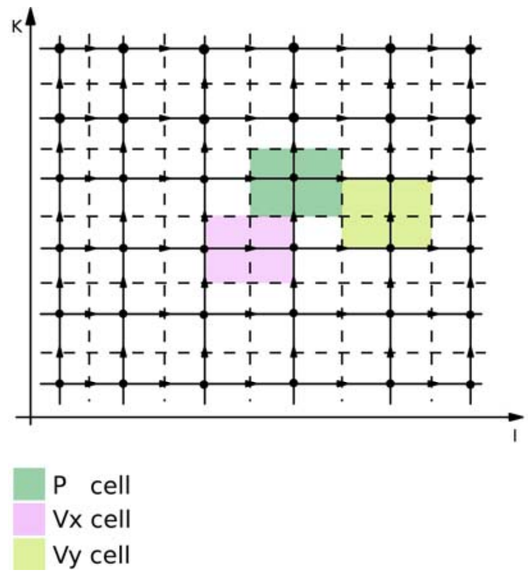

Figure 1 Staggered grid employed for the spatial discretisation of the CD equation

The standard form, employed in all CFD tools (Patankar, 1980) has been adapted for implementation in Modelica as given by equation (4).

$$V\rho \frac{d\Phi_P}{dt} + a_P \Phi_P = a_E \Phi_E + a_W \Phi_W + a_N \Phi_N + a_S \Phi_S + S \quad (4)$$

Here the coefficients $a_{E,W,N,S,P}$ are compact representation of both the diffusive and convective terms, while S are the possible sources (e.g. gravity, heat sources, external forces, depending on the nature of Φ) or, in the case of the momentum equation, the pressure gradients. The coefficients $a_{E,W,N,S,P}$ are defined by equations (5a) through (5e).

$$a_E = D_e A\left(\left|\frac{F_e}{D_e}\right|\right) + \max\{-F_e, 0\} \quad (5a)$$

$$a_W = D_w A\left(\left|\frac{F_w}{D_w}\right|\right) + \max\{F_w, 0\} \quad (5b)$$

$$a_N = D_n A\left(\left|\frac{F_n}{D_n}\right|\right) + \max\{-F_n, 0\} \quad (5c)$$

$$a_S = D_s A\left(\left|\frac{F_s}{D_s}\right|\right) + \max\{F_s, 0\} \quad (5d)$$

$$a_P = a_E + a_W + a_N + a_S \quad (5e)$$

In (5), the capital and small subscripts are associated to quantities located at the cell centers and borders, respectively. $D_{e,w,n,s}$ are the diffusion coefficients related to viscosity in the momentum equations, or thermal conductivity in the energy equation, $F_{e,w,n,s}$ are the mass flow rates through the cell faces, while $A(\cdot)$ is a function representing the convective discretisation scheme employed (e.g. UPWIND or Central Difference) (Patankar, 1980). Such a generalised version of the CD equation can be used for discretising the mass, the energy and the momentum balance equations.

Due to capabilities of the Modelica tools, no explicit implementation of the discretisation method is necessary to solve the equations. Namely, the Modelica tools (e.g. Dymola (Dassult, 2010)) already include several numerical methods/solvers for both ODE and DAE systems (e.g. DEABM, LSODE, LSODAR, DOPRI, DASSL, EULER). To address the fluid motion problem, the current study selected DASSL, an implicit variable step solver for DAE, also suitable for the stiff systems.

To solve the equations, the spatial domain is split by a 3D non uniform grid of volumes, resulting in series of 3D matrices containing temperatures, pressures, densities and velocities. Values in the matrices represent both, the variables inside the domain (i.e. water temperature within the tank) and the boudary values (i.e. temperature of the tank walls or the velocity at the inlet/outlet). One of the advantages of OO modelling, is the possibility of integrating and simulating models belonging to different domains together through the definition of standard interfaces, connectors. The variables representing the boundary conditions are associated to standard connectors, defined in the Modelica Standard Library (Modelica, 2012). For example, the temperatures on the domain boundary can be imposed by an external source as well as defined by a multilayer wall model that exchanges energy with the surrounding environment and is irradiated by the sun. Considering the tank model, velocity, density and temperature of the inlet water stream can be defined by the pump (or the distribution network) that feeds the water into the system and is managed by an automatic control.

Simulation Example

The presented simulation example investigates the effects of inlet and outlet flange positioning and water stream velocity on thermal stratification, i.e. quality of the stored heat. Three different tank layouts have been considered, as shown in Figure 2.

- LAYOUT 1, the inlet and outlet are on the same side of the tank, and located close to the top and to the bottom, respectively;
- LAYOUT 2, the inlet and outlet are located as in the previous layout, but closer together;
- LAYOUT 3, the inlet and outlet are positioned axially and centrally on the top and on the bottom of the tank, respectively.

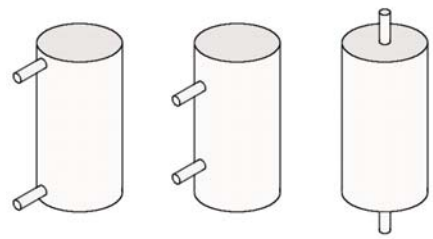

Figure 2 The storage tank layouts considered in the sample simulation case

The considered tank is typically used for storing hot water produced by solar collectors in solar thermal systems. The model is based on a 250 L Rinnai split system 316 stainless steel hot water storage tank (Rinnai, 2008). The tank is 1.25 m high with the inner diameter of 0.5 m. The inlet and outlet, arranged at different heights and with different orientation, have inner diameters of 0.015 m. For the purpose of the current analysis the tank is considered to have adiabatic walls.

The effect of water stream velocity on thermal stratification has been investigated through two different charging cycles: fast and slow. At the initial time, t = 0, the tank is full of water at the temperature of 300 K. In each charging cycle 160 liters of hot water at a temperature of 332 K are fed into the tank (about two thirds of the total tank capacity). In the fast charging cycle the hot water reaches velocity of 0.566 m/s (corresponding to a mass flow rate of 0.1 kg/s), while in the slow charging cycle the maximum velocity is one third of the fast charging, i.e. 0.189 m/s. Figure 3 shows the velocity profiles of the hot water injected into the tank. Each charging is structured such that during the first 1000 s the water reaches maximum velocity, maintained for a period of time, t_{HIGH}, and then rapidly decreasing to zero. After each charging the water temperature distribution reaches a steady state, while the only water flows within the tank are due to convection. At steady state, thermal stratification is thus expected.

The storage tank model is used to investigate six different operating conditions: fast and slow charging for each of the three described tank layouts.

The developed methodology is validated in comparison to a typical stratified water storage model used in building system simulations (Wischhusen, 2006).

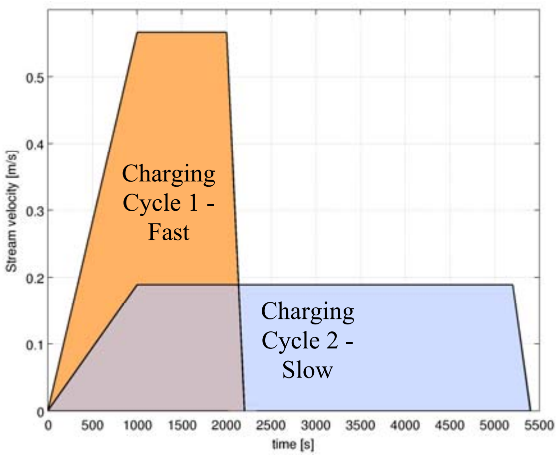

Figure 3 The storage tank charging cycles

RESULTS

The tank reaches a steady temperature distribution at time t = 4000 s and t = 8000 s for the fast and the slow charging cycles, respectively. Using 6x5x10 = 300 grid volumes, the resulting simulation takes approximately 10 minutes on Intel core 2 duo 800MHz PC, with 2GB RAM, running Dymola 7.4 for Linux. Figure 4 shows the time evolution of the mean temperature distribution along the vertical axis of the tank in the six considered cases. The rightmost lines represent the temperature profile at steady state. For comparison, Figure 5 presents vertical mean tank temperature distribution, computed with a one dimensional thermal storage model typically used in building energy simulations.

To illustrate the enhanced model capabilities in computing spatial temperature distribution within the storage tank, Figure 6 shows temperature profiles in the tank LAYOUT 3 vertical cross-section shortly before the end of the charging cycles, while Figure 7 presents steady states for all the considered layouts.

DISCUSSION

The presented temperature profiles vary with respect to the tank layout and inlet water velocity. In each layout a higher inlet velocity reduces the thermal stratification inside the tank. Such effect is a consequence of increased mixing of the hot water (entering the tank) and the colder water (inside the tank). Given a tank layout, the same amount of energy can thus be stored more efficiently if the water is injected with a lower velocity.

Considering the effect of the tank layout on the mean temperature distribution profile, horizontal inlet and outlet help the thermal stratification. The model correctly represents an intuitive phenomenon: vertically oriented inlet tends to destroy the horizontal thermal layers established by the buoyancy effect (as shown in Figure 6). On the contrary, if the inlet is horizontal, water is diffused into a thermal layer, and possibly spreads to the other layers after reaching the tank wall oposite of the inlet. Apart from inlet orientation, inlet location is also important. In LAYOUT 2, the positions of both inlet and outlet tend to limit the maximum and minimum temperatures inside the tank. The bottom of the tank, located underneath the outlet, preserves the cold water, which has a higher density and tends to fall down. On the contrary, the upper side of the tank preserves the hot water (with a lower density). The LAYOUT 2 inlet and outlet positions should then be carefully designed, since the thermal stratification within the upper and lower tank zones is very limited. In practice, such effects impose a lower and higher limit to the tank water temperature gradient.

Compared to the one dimensional thermal storage models, the proposed simulation enhancement using dynamic fluid modeling is able to distinguish between the three considered tank layouts differing with respect to the exact top and bottom inlet/outlet positions. One dimensional modelling does not take into account the inlet/outlet direction/position nor fluid stream velocity.

Compared to the fine-scale CFD tools, accuracy of the obtained results is limited due to the introduced discretisation assumptions. Nevertheless, given the underlying approach, the resulting model well describes dominant phenomena for the considered problem: when energy efficiency is the main issue, one can safely loosen precision requirements on the flow field calculations as long as the temperatures are well represented.

CONCLUSIONS

The presented enhancement using dynamic fluid modeling with Modelica is able to correctly compute spatial temperature and velocity profiles, including stratification. Such capabilities are examplified in storage tank modelling allowing calculaton of differences in the storage fluid temperatures along both, axial and radial tank directions. In the future, such capabilities would allow more realistic consideration of thermal dissipation through the tank walls and enhance traditionally used storage models in building system simulations considering isothermal fluid layers. The satisfactory performance of the developed enhanced storage tank model opens new application possibilities for OOMS and Modelica in cases requiring spatial resolution of fluid simulation results.

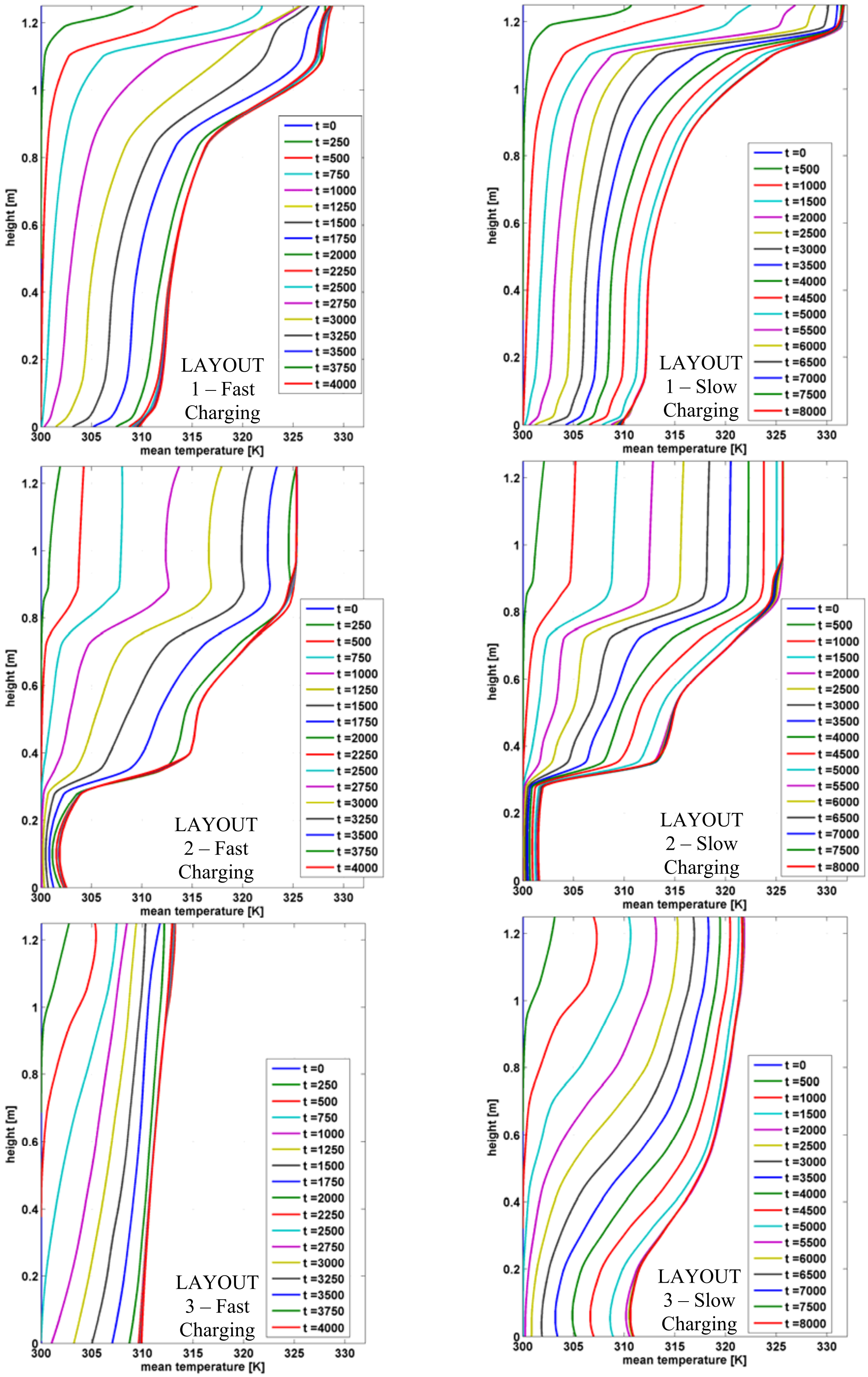

Figure 4 Transient vertical mean temperatures for the considered storage tank simulation cases

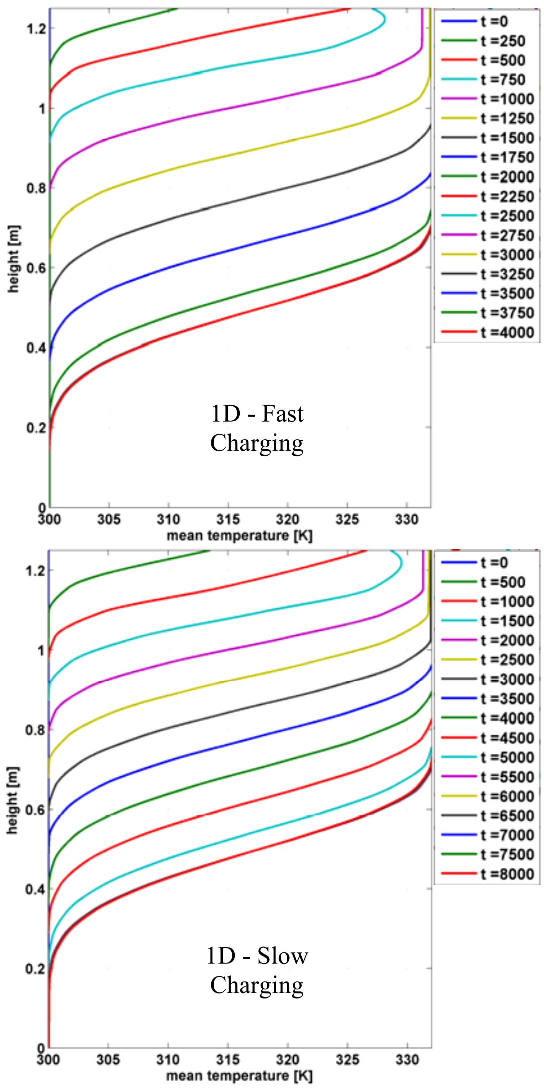

Figure 5 Transient vertical mean temperatures computed with a one dimensional storage tank model

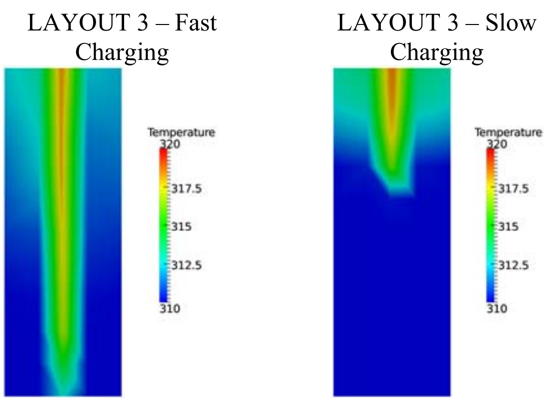

Figure 6 Spatial temperature distribution in the LAYOUT 3 storage tank vertical cross section shortly before the end of charging

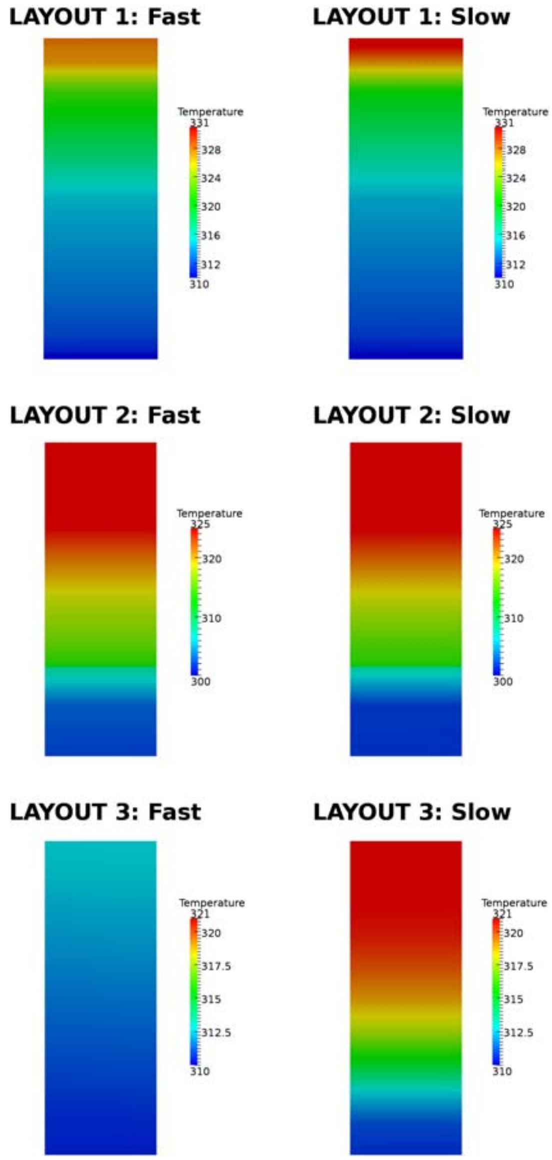

Figure 7 Steady state spatial temperature distribution in the storage tank vertical cross section for the considered simulation cases

The current study exemplifies the concepts above with reference to tanks, and for space reasons remains essentially at the component level. Future works will conversely scale up to the system level, using the created components and applying the presented ideas to design, test and validate detailed airflow models for scalable building indoor environmental simulations.

LITERATURE

Bonvini, M., Leva, A., Zavaglio, E. 2012. Object-oriented Quasi-3D Sub-zonal Airflow Models for Energy-related System-level Building Simulation, Simulation Modelling Practice and Theory 22, 1-12.

Bonvini, M., Popovac, M. 2012. Fluid Flow Modelling with Modelica", 7th Vienna Int. Conf. on Math. Modelling, February 15-17, Vienna, Austria.

Dassult Systemes. 2010. Dymola 7.4.

Djunaedy, E., Hensen, J., Loomans, M. 2003. Towards external coupling of building energy and air flow modeling programs, ASHRAE Transactions 109(2), 771-787.

Gu, B., Asada, H. 2004. Co-simulation of algebraically coupled dynamic subsystems without disclosure of proprietary subsystem models, ASME Journal of Dynamic Systems, Measurement, and Control 126(1), 1-13.

Ljubijankic, M., Nytsch-Geusen, C., Rädler, J., Lőffler, M. 2011. Numerical Coupling of Modelca and CFD for Building Energy Supply Systems, Modelica Conference, Dresden, Germany.

Ma, Y., Kelman, A., Daly, A., Borrelli, F. 2012. Predictive Control for Energy Efficient Buildings with Thermal Storage, IEEE Control Systems Magazine, February, 44-64.

Modelica and the Modelica Association, 2012. https://modelica.org/.

Patankar, S.V. 1980. Numerical Heat Transfer and Fluid Flow. Hemisphere Publishing Corp., Washington, DC, USA.

Rinnai. 2008. Hot Water System Specifications, http://www.rinnai.com.au/.

Versteeg, H.K., Malalasekera, W. 2007. An Introduction to Computational Fluid Dynamics: The Finite Volume Method, Pearson Prentice Hall, Upper Saddle River, NJ, USA.

Wischhusen, S. 2006. An Enhanced Discretisation Method for Storage Tank Models within Energy Systems, Modelica Conference, September 4-5, Vienna, Austria.