ADVANCED ANALYSIS OF COUPLED 1D / 3D SIMULATION MODELS
BY THE USE OF A SOLAR THERMAL SYSTEM

Manuel Ljubijankic, Christoph Nytsch-Geusen, Alessandro Jänicke and Michael Schmidt
Institute of Architecture and Urban Planning, University of Arts Berlin, Berlin, Germany
Hardenbergstraße 33, 10623 Germany, tel.: 030-3185-2097, e-mail:, nytsch@udk-berlin.de,
web: http://www.arch.udk-berlin.de/nytsch

ABSTRACT
This paper presents an advanced analysis of numerical coupled 1D / 3D simulation models in the field of building energy supply systems. Here, the 1D models are described in Modelica, which is a component-oriented modelling description language for DAE (Differential-Algebraic-Equation)-systems. The 3D simulation models are described with the CFD (Computational Fluid Dynamics) method. The goal is to divide the complex model of a building energy supply system in dependency of the complexity of the flow conditions into several 1D or 3D sub-models in such a way, that the coupled sub-models represent the real system as realistic as possible during the simulation experiment. Based on previous studies on this topic, simulation models with different resolutions in space and time are considered to find an optimum between required accuracy and minimal computing time.

INTRODUCTION
The main objective of the described research in this paper is the improvement of the numerical efficiency of 1D / 3D co-simulation in the field of building energy simulation. The general feasibility and obviousness of co-simulation of building and plant simulation in general and in particular for the 1D/3D coupling approach have been studied by various researchers as well as by the authors in detail (Trcka et al., 2010; Djunaedy, 2005; Beausoleil-Morrison, 2000; Ljubijankić et al., 2011).

For the in-depth studies of coupled 1D (DAE-Systems) and 3D (CFD-method) system models, a simple solar thermal system is considered: The main components of this system are an evacuated tube collector (type Viessmann VITOSOL 200 T), a hot water storage and an external plate heat exchanger, transferring the produced thermal energy from the solar loop to the storage loop. With the help of a two-point-controller the solar pump and the storage pump are simultaneously switched on. All components, except for the thermal storage, are modeled with Modelica (Modelica, 2011).

The storage device is computed with CFD. The simulation period is 24 hours on a July day of the region Hamburg (Germany) with weather data from meteonorm (Meteonorm, 2011).

The principal feasibility of coupling approaches between 1D and 3D systems at runtime was already shown and validated in the aforementioned case (see Figure 1) in previous work by the authors (Ljubijankić et al., 2011). The space resolved CFD-simulation of the storage, however, required huge calculation time in this studies. Based on the used middleware TISC for co-simulation, the calculated interface values of both simulation tools Dymola and ANSYS CFD were synchronized after after an equidistant timestep. To reduce the extremely long computing times, the approach here has been extended by a coupling with non-equidistant time steps. In addition, mesh variations with a more coarse space resolution of the CFD modelled thermal storage shall be carried out for a further optimization of the numerical efficiency. In all the new analysis the system model from the previous studies is used as the reference.
Figure 2 shows the integration of the middleware TISC (TISC, 2.3) and the coupled simulators Dymola and ANSYS CFD with their models. On the top left hand side in Figure 2, the component-oriented Modelica system model of the solar thermal plant is shown. In the same figure on top right hand side, the CFD-model of the thermal storage is displayed.

**USED SIMULATION TOOLS**

For the integration of the 1D and 3D worlds to a superior cosimulation approach, the following three tools are used:

- To describe the 1D component models of the solar thermal system (e.g. pipes, elbows, branches, thermal solar collector) Modelica is used as a model description language together with Dymola 7.4 (Dymola, 7.4).
- To describe the 3D space resolved models (here, the thermal storage) ANSYS-CFD 12.1 is used for the CFD approach.
- The framework TISC 2.3. for co-simulation is used to exchange and synchronize the numerical values between both simulation tools.

**Modelica / Dymola**

Modelica is an object- and equation-oriented modelling description language for DAE-Systems, which is suitable for modelling heterogenous and multi-physics problems on the system level (Modelica, 2011).

The Modelica-library FluidFlow for thermo-hydraulic network simulation is developed by the authors (Ljubijankić et al., 2008). The main application domains for this library are the modelling of solar thermal systems, HVAC systems and district heating / cooling systems. The FluidFlow-library comprised a set of “ready-to-use” standard hydraulic models, such as pipes, elbows, distributors and pumps. Furthermore, the library contains more specialized models from several domains (compare with Figure 3), such as solar thermal technologies (collector models), thermal storage technologies (storage models) or energy transformation technologies (e.g. models of heat exchangers, absorption chillers and cogeneration plants).

**ANSYS-CFD**

In order to determine the model state at any point of the volume of the hot water storage (e.g. temperatures, velocities, pressures), the three-dimensionally CFD method is used. In this case, the fluid region of the hot water storage is modelled with ANSYS-CFD Release 12.1 (ANSYS-CFD, 12.1) (see Figure 4), which works with CFX-algorithms, based on the Finite Volume Method (FVM). The FVM method calculates approximated solutions of the partial differential equation system, which describes the transport process of momentum, mass and heat transfer within the flow region (Navier-Stokes equations). For this purpose, the continuous fluid volume of the storage is discretised into a three-dimensional mesh consisting of approximately 20,000 tetrahedron elements. The pipes at the inlet and outlet of the storage (see Figure 4) are divided into finer prisms at the pipes’ boundary to reflect the velocity gradient better. This is necessary, because the Reynold number in the pipes will be larger than in the storage, but less than 2,300. This is a priori checked with the maximum mass flow of 0.026 kg/s at the inlet of the pipe, when the pump switches on. Therefore the CFD is used without turbulence model, i.e. this option is set to “laminar”, because only laminar flow is to be expected. Buoyancy is modelled with the Boussinesq model, which is implemented in ANSYS-CFD (for details see ANSYS-CFD, 12.1).
To map the flow between the pipe-storage connection well, the grid is there very fine resolved and has a smooth transition. (see Figure 4). The synchronization rate between the simulators can be set to equidistant or non-equidistant. First, equidistant time steps were chosen. In order to speed up the simulation, even non-equidistant time steps are chosen (see Figure 6). This requires that all coupled simulators have a list of these non-equidistant timesteps to synchronize correctly.

1D / 3D COUPLED SYSTEM MODEL OF THE SOLAR THERMAL SYSTEM

The overall system of the solar system is modeled with Modelica. (see Figure 7). All Modelica components – except the storage-interface - are part of the FluidFlow-library.

At the upper inlet pipe, mass flow and temperature are impressed as boundary conditions, which are send by Modelica during the transient co-simulation. The temperature at the lower outlet pipe is send to Modelica and the pressure at this location is received from Modelica. The step size of both simulators is the same so that they synchronize their values at the same point in time, regardless if the step size is equidistant or non-equidistant. Interpolation between time steps is not necessary for this purpose.

TISC

For coupling Dymola and ANSYS-CFD, TISC - the TLK Inter Software Connector (Kossel, 2009 and Puntigam, 2006) - is used as a co-simulation environment. The software is platform independent and uses TCP/IP-sockets for communication. TISC supports sequential (“explicit”) synchronization, which is used for exchanging numerical values at runtime.

The inner structure of the Modelica-storage-component in Figure 7 contains interface-components which are able to connect to the TISC-library (see Figure 8), which will be linked to the simulation experiment by Dymola at runtime. On the left hand side in Figure 8 “send-to-TISC” values and on right hand side “receive-from-TISC” values are shown such as mass flow, pressure and temperature. These coupling values from Modelica will be impressed to the CFD-Model of thermal storage (see Figure 4) in ANSYS-CFD as dynamically boundary conditions at runtime. These values will be updated every time step by TISC, which is also coupled to the rest of the solar thermal system – implemented in Modelica (see Figure 8 and 7).
For example, if the massflow in the thermal storage loop is changing, then at the end of the current time step an event will be triggered, which sends this new value (e.g. massflow) to TISC.

**TIMESTEP ADAPTATION**

The numerical coupling of Modelica / Dymola and ANSYS CFD to a transient system simulation, means the slower simulator ("with the more expensive time step") determines the total simulation time, in this case it is ANSYS-CFD. The Solution of the CFD-computation should converge, within each time step. The convergence criteria is 1e-3 for the maximum residuals, which are calculated as the imbalances in the linearized system of discrete equations for solving the mass continuity, the three momentum equations and the energy equation. Therefore some inner loops (iterations) are needed. One loop per time step at best, with a maximum of 15 inner loops, which is set as an upper limit, so that the calculation does not converge within a time step. In a non-changing mesh, the sum of all inner loops is a measure of the total effort of a transient simulation. If over a period the gradients are changing softly, then larger time steps can be set, because less inner loops are necessary (The goal is to reduce the simulation time). This requires that all coupled tools support the control of adaptive time steps.

To realize this, the model of the solar thermal plant (implemented in Modelica), the CFD-thermal-storage and TISC have been extended, so it can also synchronize with non-equidistant time steps numerical values. The coupling interface between ANSYS-CFD and TISC is implemented in Fortran. Until now, for the purpose of this study, brutto type interfaces for both Simulators (also for the Dymola-TISC-interface) were written, in which the current time step size is synchronized dynamically (via the middleware TISC) at simulation time. Therefore ANSYS-CFD must be the “time-step-master” and sets the time step size adaptively and sends this information to Dymola, so that Dymola is able to make at next the same time step size like ANSYS-CFD in its step before. This is still work in progress.

It is possible to give both simulators a static time-step-size-list in which the time steps are a priori defined. After each time step, the simulation stops and numerical values are synchronized via TISC.

However, this process takes an enormous amount of engineering experience to set time step sizes in advance. In order to identify meaningful adaptive time steps, dynamic boundary conditions are impressed to the CFD-model and a stand-alone CFD simulation is started. The considered mesh of the CFD-storage has 19,870 nodes. The boundary conditions are taken from a previously coupled Dymola/ANSYS-CFD simulation with equidistant time step size (1s), which is written before in a binary file format (netcdf). For reading dynamically boundary conditions at simulation time, a Fortran routine was implemented, which is an interface to connect ANSYS-CFD and the ncDataReader2. The ncDataReader2 returns physical values like pressure, mass flow or temperature for any point in simulation time, by interpolation (for details see ncDataReader, 2011). With this impressed "substitute model", the storage gets the same conditions as if it were coupled with the Modelica world.

Now ANSYS-CFD itself can determine the time step during the simulation and choose as few inner iterations (loops) as necessary. This is currently possible only in not coupled mode. In Figure 9, time step size is shown over simulation time. The red graph is the chosen time step size of the stand-alone CFD-simulation with dynamic boundary conditions. With this knowledge, it is possible to divide the simulation period in areas of equal time step sizes.

![Figure 9 Adaptive time steps chosen by ANSYS-CFD in stand-alone mode (red curve). Pre-defined static time steps for coupled mode (green curve)](image)
graph from below (see Figure 9: green graph). Now a list with time steps can be created, which will be fed to the Modelica-model for the solar thermal plant and to the CFD-model for the thermal storage (see Figure 10).

![Figure 10 Scheme for coupled mode with static-time-step-size-list.](image)

Such a coupled simulation is no longer running with equidistant time steps, instead it has a problem-specified-time-step-adaptation, the aforementioned list. It must be mentioned that this list should be used only in conjunction with this coupled model. For another coupled Dymola / ANSYS-CFD simulation experiment with this method, for example if another thermal storage is used or the model of the solar system is changed, then an analysis as shown in Figure 9 should be considered beforehand.

A measure for the CFD-simulation effort is the sum of all required loops during the simulation time. (see Figure 11). The green curve represents the effort of the coupled simulation with an equidistant time step of one second. The total simulation duration is about 48 hours for 24 hours real-time. The red curve represents the CFD-stand-alone-simulation with impressed dynamic boundary conditions and takes about 43 hours.

![Figure 11 No of required coefficient loops in 24h. Green curve: coupled Simulation with 1s time step. Red Curve: CFD-stand-alone simulation with equidistant time steps. Blue curve: coupled simulation with problem-specified-time-step-size-list.](image)

The blue curve represents the coupled simulation with problem-specified-time-step-size-list, therefore about 40 hours are needed. The convergence criteria is the same as explained in the beginning of the section. A Mac-Pro with 8 Xeon (E542)-cores at 2.8GHz and 32GB RAM was used in this case.

In summary, it can not be 'state of the art' to define adaptive time step sizes through lists. Such an approach is too prone to errors and is only possible through conducting pre-studies (see Figure 9), in which the appropriate time step sizes could be determined. Instead the most time consuming simulation tool, in our case ANSYS-CFD should be the master and should communicate the appropriate time step size at the given time to the middleware TISC, in order to allow an adjustment during the simulation experiment in the faster simulation tool, here Dymola.

**MESH VARIATIONS OF THE CFD- THERMAL STORAGE**

Another parameter that significantly determines the duration of the transient CFD-simulation is the number of grid nodes. The more mesh nodes, the higher the computational cost. This is well known in the CFD world. In order to map the 3D temperature distribution in thermal storage well enough, but still keep the computational effort within reasonable limits, three different meshes are considered. The first mesh has 842 nodes, it is very rough. The second mesh has 4,502 nodes and the third one 19,870 nodes.

In Figure 12 the differently fine grids are shown from left to right.

![Figure 12 Mesh variations: from left to right: rough (a), middle (b) and fine (c) mesh resolution.](image)

In this study an equidistant time step size of 1s is used for the calculation. Now only the number of grid nodes determines the overall effort. With this, the following calculation times were obtained:

- Grid a): 17h
- Grid b): 35h
- Grid c): 48h

on the same computer as mentioned in the previous chapter. The different fine grids for the storage show
the thermal layering, which is derived from the loading process, in different levels of detail. In Figure 13 the temperature distribution is shown over the time of the simulation. For this the storage is divided into 10 equal volumes. Shown here is the middle temperature in each volume.

Due to the laminar inflow of the warm fluid in the storage, a well leveled temperature stratification is observed, which begins in the evening hours to distribute easily. It is obvious that the storage with 842 nodes cannot effectively map the thermal layering and distribution. In the more complex grid (4,502 nodes) the properties already approach those of the most complex grid with 19,870 nodes. However, the effort of the simulation is immense. Even this resolution is in need of improvement, but it suffices for the description of the temperature field. For a more exact analysis of the velocity field an even finer grid must be chosen. A transient simulation of 24 hours would then mean a multiple of calculation time (see above). A finer grid than that of 19,870 nodes is therefore disregarded.

Looking at the overall model of the solar thermal system (see Figure 7), it is notable that the Modelica model is not only a dynamic boundary condition model for the CFX-storage, but that the temperature properties in the storage directly affect the turning on/off of the pump. Depending on the temperature in the storage, the pumps are activated or deactivated. For this the sensor temperature is measured in the lower third of the storage.

Figure 14 shows the pump switching procedures for the three above described storages. In the morning, at dawn, the pump engages and disengages several times because the collector is warming up. The incoming cold fluid from the pipes accounts for a deactivation of the pump until a stabilised status is reached. The same behavior can be observed even more intensely in the evening. It becomes clear that exactly this switching behavior cannot be displayed properly with a rough resolution grid storage (852 nodes), because the thermal layering can also not be displayed here. The switching behavior in Figure 14 middle and below is similar, although here it also becomes apparent that the more finely resolved grid storage accounts for a more 'rigorous' switching behavior.
CONCLUSION AND OUTLOOK

First, we are summarizing the essence of the last two paragraphs. While trying to calculate the coupled model of the solar thermal system with a time adaptive procedure, the middleware should be capable to synchronize after any desired time steps. The slowest link in the chain should then be the time master. Here this would be the ANSYS-CFD simulator. But for this, further programming work is required. It was shown that with a list of predetermined time step sizes the fastest calculation speeds could be achieved, but without a precise pre-study of the given model it is very difficult to create such a list.

The grid studies have shown that only the high resolution mesh (19,870 nodes) can sufficiently show the interdependent coupling behavior (for example the shown pump-switch-behavior). The storage behavior of the grid with 4,502 nodes is already close to that behavior of the finest. Rougher grids save calculation time, but have to be handled with care, because one can only determine the qualitative behavior of the physical values in the storage (shown here on the temperature distribution example in Figure 13). In order to create a high quality analysis regarding the current behavior in the storage, a highly resolved grid has to be used. For example a 24 hours simulation would then take much longer calculation times than 48 hours.

Secondly, from the technological perspective the combination of Modelica/Dymola and ANSYS-CFD works very well at the time of the simulation. Therefore it is possible now to simulate a system model with different levels of detail.

In order to simulate the behavior of the thermal storage even better, the more realistic modelling of the storage’s inner constructions (e.g. pipes, branches and connections) would be possible as the next step (see Figure 16). For that we already started some grid studies, where the inlet pipe passes through the storage from bottom almost to the top end, in order to more evenly distribute the hot water. It will be a challenge to find an adequate mesh so that the computing times remain bearable for a whole day simulation.

Another possibility would be to grant the middleware (here TISC) more numerical intelligence, so that the time step size adaptivity can be superordinate controlled by the middleware. An assumption for this is, that the middleware “knows” of the computation accuracies of the single simulators and then it could react accordingly (for example it should repeat the calculation of a specific time step, if necessary).

REFERENCES

ncDataReader 2011. Homepage: http://www.j-raedler.de/projects/ncDataReader2