

COMBINING DIFFERENT LEVELS OF DETAIL IN MODELLING FOR AN IMPROVED PRECISION OF HVAC PLANT SIMULATION

Manuel Ljubijankić and Christoph Nytsch-Geusen

Institute of Architecture and Urban Planning, University of Arts Berlin, Berlin, Germany

Hardenbergstraße 33, 10623 Germany

Email: nytsch@udk-berlin.de

ABSTRACT

In this article, we will present a new multi-level modelling approach to obtain higher precision in HVAC plant simulation models. For this purpose, two different levels of detail in modelling, the coarse component-based approach of Modelica and the detailed physical-based approach of Computational Fluid Dynamics (CFD) were combined.

In a first use case, we analysed the thermal hydraulic network of a simple heating system, both with the CFD-approach (3D-model) and the Modelica-approach (1D-model). Both simulation models consist of an ideal pump, an ideal heating source, two hydraulic loops and an ideal thermal sink. We compared the differences between the results (pressures, temperatures, mass flows) of both modelling approaches. Finally, we tried to improve the accuracy of the fast Modelica model by introducing correction models in the thermal hydraulic network.

INTRODUCTION

Modern modelling approaches like the object-oriented and equation-based modelling language Modelica (<http://www.modelica.org>) enable the configuration and simulation of complex energy systems,

such as the district heating system in Figure 1.

In general, a high number of standard components from a model library (pipes, branches, pumps, heat exchangers) are used, parameterised and connected by the modeller to configure a complex energy system.

One problematic aspect of this component-based modelling method lies in the combination of many single-component models to form a common system model in respect to the physical interactions between the small parts. The example of a small hydraulic system with three components – an elbow, a straight pipe and a second elbow – demonstrates the principal problem: if the length of the pipe is relatively short in comparison to its diameter, then the impact of pressure loss from the elbow on the pressure loss of the pipe is considerable. In this case, the total pressure loss of the three components is not the sum of the single pressure losses of each component, because their equations for the pressure loss calculation assume a distinctive flow profile, which here does not exist. But in the case of a relative long intermediate-pipe, the sum of the single pressure losses approximates the total pressure loss of the small hydraulic system.

Considering this question, we looked for an approach to improve the accuracy of the component-based



Figure 1 Simulation model of the district heating grid of Rheinsberg/Germany. The simulation model includes also 500 simple dynamic building models and the corresponding heat transfer relay stations.

(Klein-Robbenhaar et al., 2000)

simulation method. For this purpose, we combined two different levels of detail in modelling: we use the results of detailed, but slow 3D-CFD-analyses for improving the quality of the simplified, but fast 1D-Modelica models. As described above, the Modelica calculation of the flow-induced pressure loss of several serial connected components can be quite accurate or highly defective. It depends on the parameter values of the combined components. In contrast to this component-based modelling approach, the CFD-approach models the fluid as a continuous volume, whereas all hydraulic interactions between the single hydraulic components are considered automatically.

With this comparative method we analysed a simplified heating system with two hydraulic loops and an ideal thermal source and thermal sink.

ANALYSED SYSTEM

The analysed system (Figure 2) describes the thermal hydraulic network of an abstract heating system, which contains pipes, branches, elbows and a pump. In comparison to a real complex heating system (e.g. in Figure 1), this strong simplified net with only two loops shows the same physical effects as the complex one (e.g. the mass flow distribution in branches), but it is much more clear and thus more suitable for a comparative analysis between the 3D-CFD-approach and the 1D-Modelica approach. It is useful to study pressure losses, mass flow distribution and the heat transfer for the whole system and for the single components.

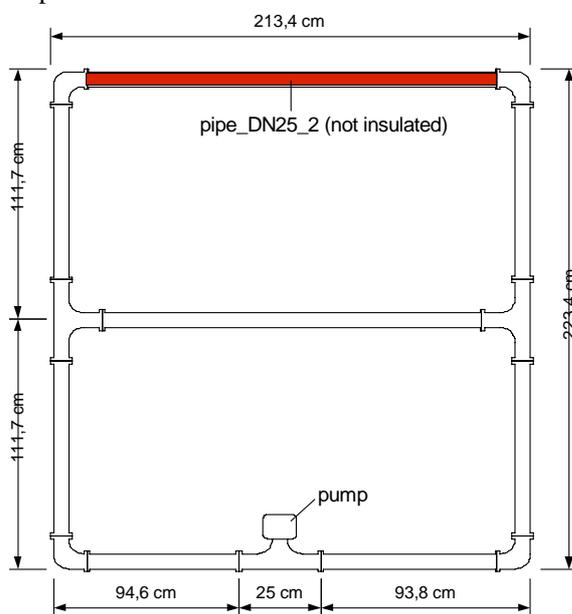


Figure 2 Geometry and topology of the analysed heating system

The pipes have an inside diameter of 26 mm (DN25) and consist of copper walls 2 mm thick. One of the pipes is not insulated. Thus it can be seen as a heat consumer, such as a strong simplified heating

element. All the other hydraulic components are surrounded by a 30 mm thick insulation.

Further, the system contains an adjustable “ideal pump” for controlling the mass flow in a wide range. Mass flows are impressed on the system, which correspond to a Reynolds number from 10 to 100,000 at the pump discharge opening. This wide range of mass flows leads both to laminar and turbulent flow conditions in the single pipes, branches and elbows. The outlet temperature of the pump is constant 70 °C (ideal controlled heater) and the environment air temperature is also constant at 20 °C.

DETAILED CFD-APPROACH

In order to determine the physical dimensions (e.g. pressure and/or temperature) at any point in the model, we modelled the analysed system three-dimensionally with the Computational Fluid Dynamics Method (CFD). CFD as an established method of fluid mechanics has the power to solve complex flow mechanical problems with numerical methods, such as the Finite Differential Method (FDM), the Finite Volume Method (FVM) or the Finite Element Method (FEM). Each of these methods 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 the detailed CFD-approach we used the ANSYS CFX Release 11 (CFX, 2007), which works with CFD-algorithms, based on the FVM-Method. (CFX, 2006)

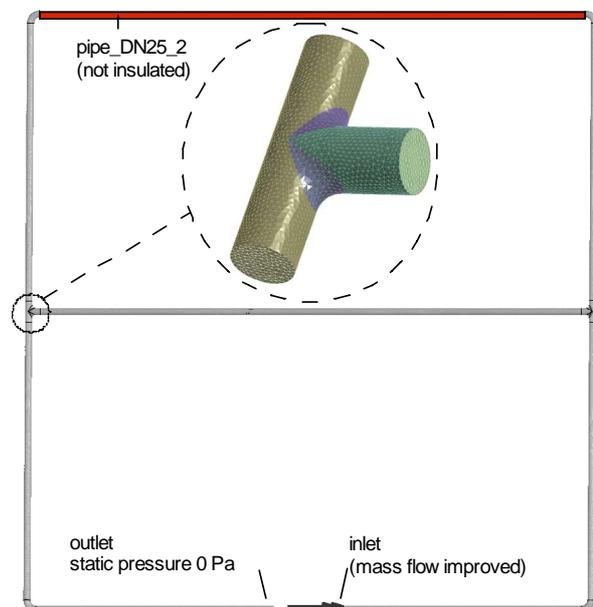


Figure 3 CFD-Model of the ideal heating system, modelled with 450.000 elements. The Detail shows the 3D-modelling and the volume discretisation of the branch.

The analysed system from Figure 2, is geometrically modelled in ANSYS CFX as shown in Figure 3.

The elements of the hydraulic network, such as branches, elbows and pipes are separately modelled in 3-D with the ANSYS-DESIGN-MODELER. Starting with these individual components, the thermal hydraulic net can be assembled step by step to form the complete system.

Following simplifications were made:

- The pump is replaced by two boundary conditions for the network (inlet and outlet). In Figure 3 the inlet is at the right-hand side, and the outlet at the left-hand side.
- The wall roughness is set to zero.
- The fluid is incompressible water.

Following boundary conditions were used:

- Mass flows from 8.49×10^{-4} to 0.849 kg/s enter at the inlet (this corresponds approximately to Reynolds numbers of 10 - 100,000).
- The water flows with 343.15 K into the inlet.
- The static pressure at the outlet is set to 0 Pa.
- The common heat transfer coefficient for convection and long-wave radiation exchange on the exterior surface is set to 4 W/(m·K).
- The environment temperature is set to 293.15 K.

The viscosity depends on the temperature of the water. Because the density of water changes insubstantially within the temperature range of 293.15 – 343.15 K, it can be calculated with the constant value of 997 kg/m^3 .

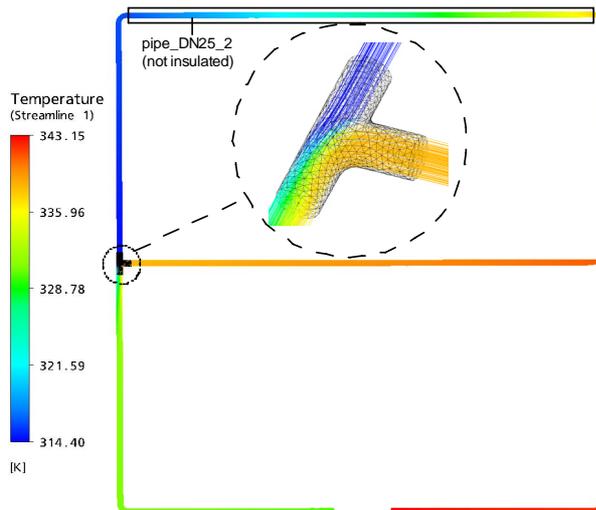


Figure 4 Calculated temperature field of the CFD-model

The system represented in Figure 3 is subdivided with ANSYS-CFX-MESH into 450,000 tetrahedron elements. The generated net contains 100,000 nodes.

A cut-out of the discretised area, is shown for the branch in Figure 4. After the volume is discretised, boundary conditions are set, and the material values are adjusted, the ANSYS-CFX-SOLVER calculates for each of the tetrahedron elements a discretised form of the following 5 partial differential equations:

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0, \quad (1)$$

Three momentum equations (vector notation with the Nabla-operator for the three Cartesian coordinates):

$$\frac{\partial (\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U} \otimes \mathbf{U}) = -\nabla p + \nabla \cdot \boldsymbol{\tau}, \quad (2)$$

Total energy equation:

$$\begin{aligned} \frac{\partial (\rho h_{tot})}{\partial t} - \frac{\partial p}{\partial t} + \nabla \cdot (\rho \mathbf{U} h_{tot}) \\ = \nabla \cdot (\lambda \nabla T) + \nabla \cdot (\mathbf{U} \cdot \boldsymbol{\tau}). \end{aligned} \quad (3)$$

This means we have to solve $450,000^1 \times 5^2 = 2,250,000$ equations.

The turbulence model we used is the $k - \omega$ -based shear stress transport (SST) model. It accounts for the transport of the turbulent shear stress and gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients (CFX, 2006).

Finally, we can see the temperature distribution in the fluid region of the analysed system. Due to the omitted insulation into *pipe_DN25_2* the water cools down here from approx. 343.15 K intake temperature down to 314 K. The detailed view in Figure 4 shows the intermixture of cold and warm mass flow in *branch_DN25_6*.

SIMPLIFIED MODELICA-APPROACH

In comparison to the high-resolution 3D-CFD-approach, the simplified 1D-Modelica approach works with concentrated parameters within the single components of the modelled system. The object-oriented and equation-based modelling approach of the language Modelica (Modelica, 2007) divides a system model into a set of sub-component models.

The physical behaviour of each sub-component - in our case models for pipes, branches, elbows und thermo-hydraulic boundary conditions - is described by a set of ordinary differential and algebraic equations. Further, the models include one or more typed interfaces for connecting the sub-components in the system model.

Figure 5 shows the model of the heating system configured by the use of the predefined models of the Modelica-library *FluidFlow*. This model library for thermo-hydraulic network simulation is being

¹Number of elements

²Equation in each element

developed by the authors and is applied to the modelling of solar thermal systems, HVAC-systems and district heating systems (Ljubijankić et. al. 2008). Each hydraulic component model of this library includes a pressure loss calculation, which depends on the component type, the individual component geometry and the Reynolds number, which is calculated with temperature-dependent fluid properties. Further, the hydraulic component models include a transient energy balance with detailed heat transfer calculation. Each pipe model is able to show a 1D-discretisation of the energy balance in the flow direction; all of the other models have only one concentrated parameter for the energy balance.

Reverse flow is also supported. The flow directions in the modelled thermal hydraulic networks only depend on the external boundary conditions of the network and the power of the net-integrated pumps.

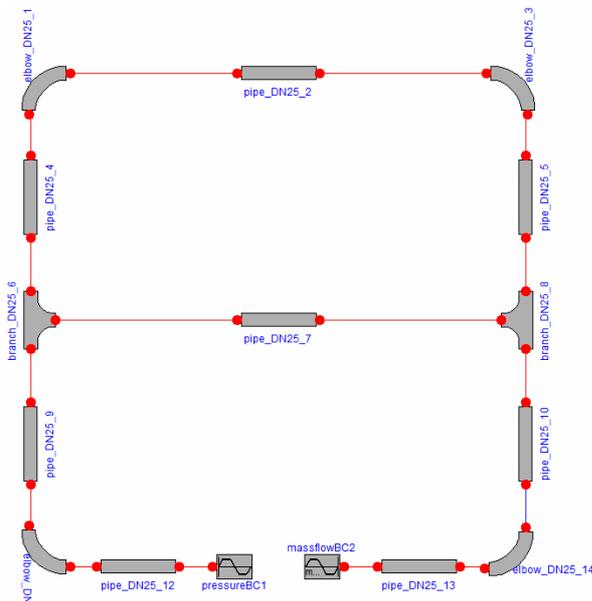


Figure 5 Component based Modelica model of the ideal heating system, modelled with 264 equations.

The following code-snippets illustrate the object-oriented modelling character of the Modelica models. After importing the *FluidFlow*-library, the different model-types are specified and parameterised by the use of class-inheritance (keyword **extends**):

```
...
import FluidFlow.ThermalHydraulics.Models;
model ThermalHydraulicNetworkTwoLoops
  model PipeDN25Insulated
    extends Models.Pipe(rho = 997.0,
      c = 4181.7,
      T_start=273.15+20.0,
      T_env = 273.15+20.0,
      diameter = 0.026,
      k = 0.0,
      U_env=1(1.0/100.0
        +0.002/401.0
        +0.030/0.035
        +1.0/4.0));
  end PipeDN25Insulated;
```

```
model PipeDN25
  extends Models.Pipe(...);
end PipeDN25;

model ElbowPipeDN25
  extends Models.ElbowPipe(...);
end ElbowPipeDN25;

model BranchPipeDN25
  extends Models.BranchPipe(...);
end BranchPipeDN25;
```

Now, the thermal and hydraulic boundary conditions are defined (70°C inlet temperature in the component *pipe_DN25_13* and a mass flow corresponding to a Reynolds number of 100):

```
parameter Real Re[10]={10,100,...,100000};
...
parameter SIunits.Temp_K T_fluid_in=273.15+70.0;
parameter SIunits.MassFlowRate mdot_in
  = -Re[2] * ny(1,T_fluid_in) / diameter
  * pi * diameter^2/4 * rho(1,T_fluid_in);
Models.PressureBoundaryCondition pressureBC1
  (transient=false, p_const=0.0, ...);
Models.MassFlowBoundaryCondition massflowBC2
  (transient=false, mdot_const=mdot_in,
  T_const = T_fluid_in);
```

In the next step, all the component models are instantiated with the predefined model classes. Thereby, the uninsulated element *pipe_DN25_2* is discretised into 10 segments (**n_nodes=10**), because the temperature gradient in this component has to be modelled more precisely than in the other components (modelled only with one segment), due to the increased heating loss:

```
PipeDN25Insulated pipe_DN25_9(length=1.032);
BranchPipeDN25 branch_DN25_8(disjunction = true);
PipeDN25Insulated pipe_DN25_5(length=1.032);
PipeDN25Insulated pipe_DN25_4(length=1.032);
PipeDN25Insulated pipe_DN25_7(length=2.032);
BranchPipeDN25 branch_DN25_6(disjunction=false);
PipeDN25Insulated pipe_DN25_10(length=1.032);
PipeDN25 pipe_DN25_2(length=2.04,n_nodes=10);
ElbowPipeDN25 elbow_DN25_1;
ElbowPipeDN25 elbow_DN25_3;
ElbowPipeDN25 elbow_DN25_11;
ElbowPipeDN25 elbow_DN25_14;
PipeDN25Insulated pipe_DN25_13(length=0.891);
PipeDN25Insulated pipe_DN25_12(length=0.899);
```

In the **equation**-section of the model, the system model topology, expressed by the connections between the single sub-components, is defined by the use of the **connect**-statement and the model-interfaces **p_1**, **p_2** and **p_3**. Here, in the *FluidFlow*-library it is insignificant in which order and with which opening two components are connected, because the hydraulic components have only “openings” but not “inlets” or “outlets”:

```
equation
  connect(massflowBC2.p_1, pipe_DN25_13.p_2);
  connect(pipe_DN25_13.p_1, elbow_DN25_14.p_1);
  connect(pipe_DN25_10.p_1, elbow_DN25_14.p_2);
  connect(branch_DN25_8.p_1, pipe_DN25_10.p_2);
  ...
  connect(elbow_DN25_11.p_2, pipe_DN25_12.p_2);
  connect(pressureBC1.p_1, pipe_DN25_12.p_1);
end ThermalHydraulicNetworkTwoLoops;
```

In this way, the modelled ideal heating system is described with 264 ordinary differential and algebraic equations. Figure 6 shows the results of a dynamic simulation of 20,000 seconds.

At the beginning of the simulation experiment, the fluid in all the hydraulic elements has a temperature of 20 °C (293.17 K). Because the Reynolds number 100 is small and only a part (26 percent) of the fluid leaves the branch in the direction of the uninsulated element *pipe_DN25_2*, the hot fluid (70 °C or 343.15 K from *MassflowBC2*) needs more than 15,000 seconds to warm up the fluid within *pipe_DN25_2*. The negativ temperature gradient in the flow direction within the “heating sink” *pipe_DN25_2* is visible. The fluid enters the pipe at 336 K and leaves it at 314 K.

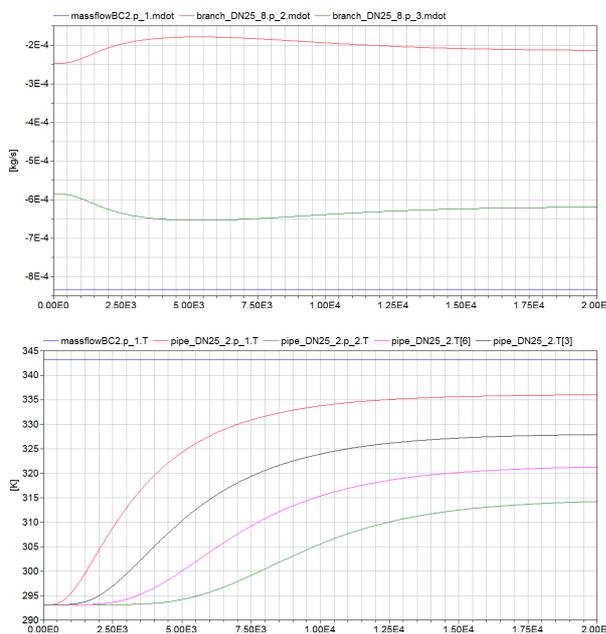


Figure 6 Mass-flow distribution (above) and temperature distribution (below) in the hydraulic network for a Reynolds number of 100

MODEL COMPARISON

For the model comparison we looked for suitable values of both modelling approaches. For this purpose, the high information density of the 3D-CFD-approach had to be reduced to comparable “mean”-values.

In Figure 7 our modality of information reduction is illustrated. Figure 7 above shows the three-dimensional mass flow rate distributions at the openings of the *branch_DN25_8*. We integrated the mass flow distributions over the respective cross-sections and compare these sum values with the mass flow rates of the 1D-Modelica model. (Figure 7, bottom) In the same way we obtained comparable values for the total pressure and the fluid temperature.

We performed simulation experiments for the CFD- and the Modelica-approach for the Reynold (Re) numbers 10, 100, 500, 1000, 2000, 2300, 2500, 5000, 10,000 and 100,000. The comparison between the simulation results are illustrated in Figures 8, 9 and 10. Figure 8 shows the total pressure loss of the heating system that is equal to the pressure difference, which the pump has to produce. The picture shows similar results between Re 100 and Re 1000, but differences for Re 10 and Re > 2000, where the pressure losses of the Modelica-approach are below the values of the CFD-approach.

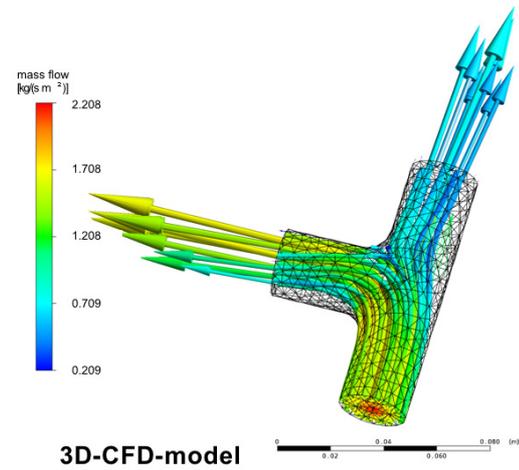


Figure 7 Comparison of the 3D-CFD-approach and the 1D-Modelica-approach by the example of *branch_DN25_8*

Figure 9 shows the mass flow, which diverges at the component *branch_DN25_8* and joins later with the cooled mass flow from *pipe_DN25_2* via *branch_DN25_6*. The picture shows similar results up to Re 5000, but differences for greater Re-numbers, whereas the mass flow of the Modelica-approach falls below the values of the CFD-approach.

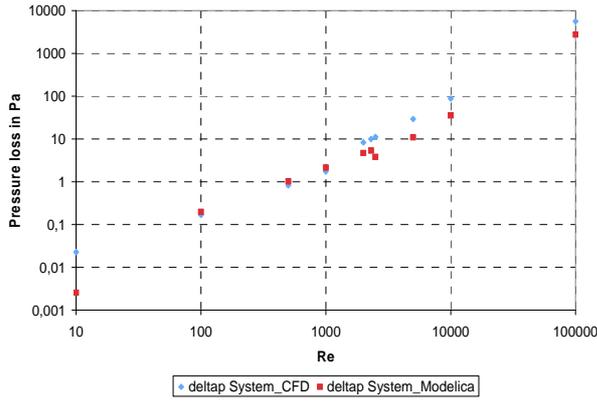


Figure 8 System pressure loss of the ideal heating system, calculated with CFD and Modelica

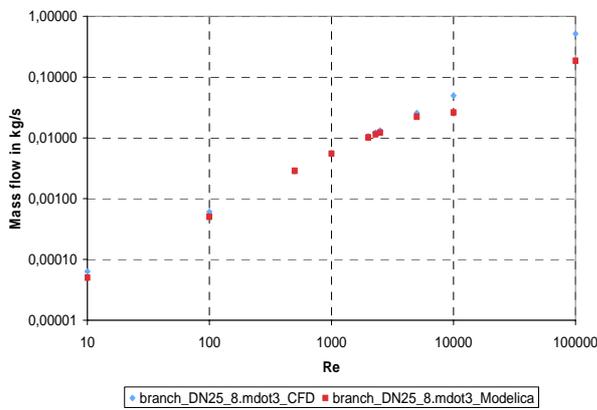


Figure 9 Diverging mass flow \dot{m}_{dot3} in the component $branch_DN25_3$, calculated with CFD and Modelica

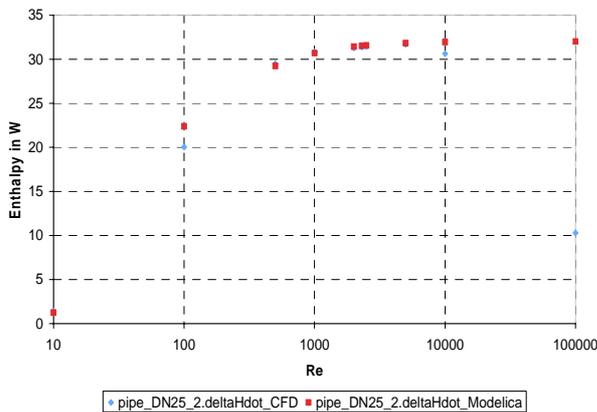


Figure 10 Enthalpy difference between the openings of component $pipe_DN25_2$ (heating sink), calculated with CFD and Modelica

Figure 10 shows the enthalpy difference between the openings of $pipe_DN25_2$, which is caused by the heat loss in the uninsulated pipe. Because the enthalpy difference depends both on the temperature difference between the pipe openings and the mass flow through the pipe, which is again dependent on the calculated mass flow ratio of $branch_DN25_8$

(see Figure 9), this parameter is a good system benchmark for the comparison between both calculation approaches.

The picture shows approximately similar values between Re 500 up to Re 5,000. For Re 100 and Re 10000 the values of the CFD-approach are below the values of the Modelica-approach, but for the high Re number of 100,000 the CFD-approach reaches less than one-third of the value of the Modelica-approach.

IMPROVEMENT OF THE MODELICA MODEL

Based on the results of the previous paragraph, we calibrated the simplified 1D-Modelica model with the help of the 3D-CFD-model. For this, we added one additional pressure loss model into each hydraulic loop. (Figure 11) Each of these models has different positive or negative constant ζ -values for the laminar and for the turbulent flow range. Based on this assumption, the additional “calibration pressure loss” Δp_{cal} is calculated:

$$\Delta p_{cal} = \begin{cases} \zeta_{lam} \cdot \frac{\rho}{2} \cdot \bar{w}^2 & : \text{if } Re < 2300 \\ \zeta_{turb} \cdot \frac{\rho}{2} \cdot \bar{w}^2 & : \text{if } Re \geq 2300 \end{cases} \quad (4)$$

Due to numerical stability, a soft switching between both ζ -values of equation (4) is realised:

$$\zeta = (1.0 - f) \cdot \zeta_{lam} + f \cdot \zeta_{turb} \quad (5)$$

with $f = 0.5 + 0.5 \cdot \tanh((Re - 2300) / 50)$

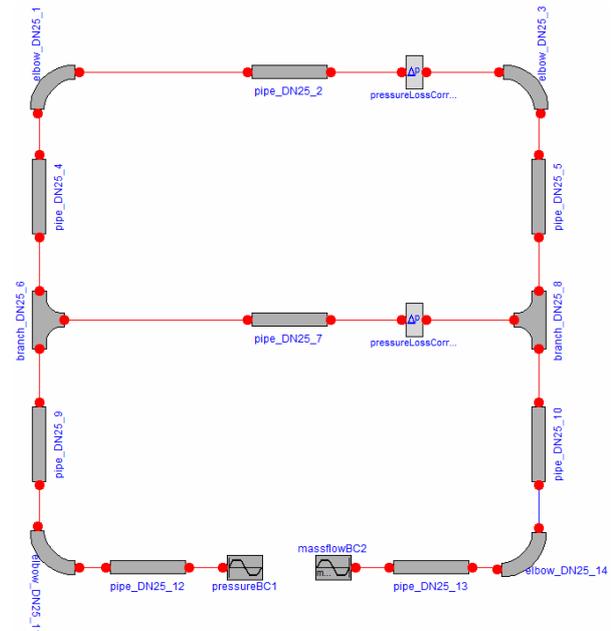


Figure 11 Modelica model of the ideal heating system with additional pressure loss models

Figures 12 and 13 show the results of the simulation experiments with the modified hydraulic systems. Here, for the long loop via *pipe_DN25_2* ζ_{lam} was set on 28 and ζ_{turb} on 14 and for the short loop via *pipe_DN25_7* ζ_{lam} on 10 and ζ_{turb} on 5.

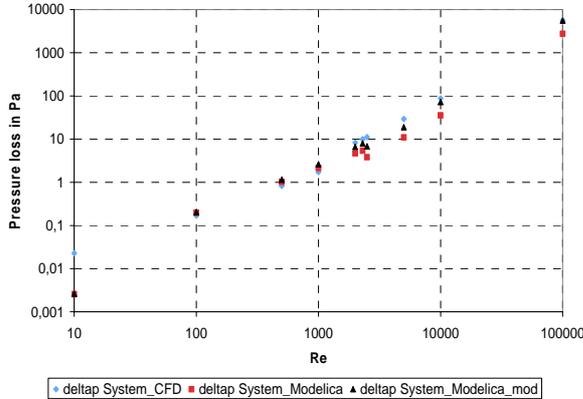


Figure 12 System pressure loss of the ideal heating system, calculated with the CFD and Modelica (with pressure correction)

Figure 12 shows an improvement for the calculated system pressure loss. For $Re = 100,000$ the pressure loss of the modified Modelica model is nearly equal to the CFD-model. For Re between 1,000 and 10,000 the calculated system pressure loss is remarkably closer to the CFD results. For $Re = 10$ the pressure calibration shows no effect.

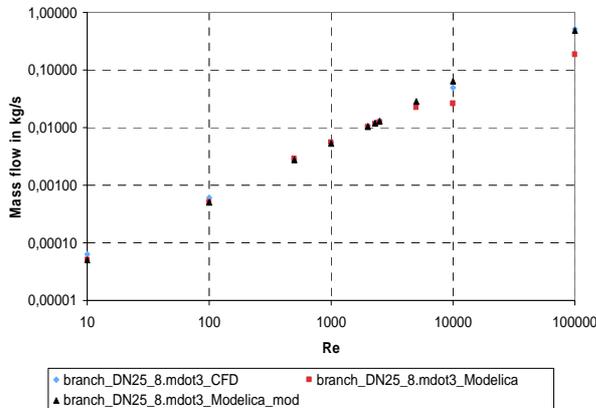


Figure 13 Diverging mass flow \dot{m} in the component *branch_DN25_3*, calculated with the CFD and Modelica (with pressure correction)

Figure 13 shows again the calculated diverging mass flow in *branch_DN25_3*. As a result of the pressure loss calibration, the modified Modelica model shows concurrent values with the CFD-model for all Reynolds numbers.

DISCUSSION AND RESULT ANALYSIS

We have demonstrated the improvement of the 1D-simulation of thermal-hydraulic networks with the help of the detailed 3D-CFD-approach using the example of an idealised simple heating system with

two hydraulic loops. The high-resolution CFD system model, based on 2,250,000 equations enables a continuous flow calculation over the total flow region, including all the pressure losses dependencies between the single network components. Under the assumption, that this detailed model is more accurate than the 1D-approach, we advanced the highly simplified 1D-Modelica system model, based on 264 equations with 2 further calibration equations. As a result we obtained from the Modelica model a good approximation of the accuracy of the detailed model for a wide range of Reynolds numbers with the calculation time of the simplified model.

Conclusion

One of the next development steps of the Modelica *FluidFlow*-Library will be the improvement of the pressure loss calculation within the single component models (pipes, branches, elbows etc.). For this purpose, typical standard configurations of these basic components (e.g. a system with the sequence elbow \rightarrow pipe \rightarrow elbow) shall be comparatively analysed in respect to their interactions with CFD and Modelica.

The next step will be the enhancement of these comparative studies to more complex thermal energy systems, where a part of the flow region shows low and the other part medium or high Reynolds numbers. For example, this could be the 3D-convection flow within a thermal water storage model, combined with the other hydraulic component models of a solar thermal system.

Finally, coarse 1D-Modelica models and fine 3D-CFD-models shall be numerically coupled to common system models using a middleware such as TISC (Kossel et al., 2006). This way will establish new possibilities for detailed system analyses, combined with comparatively moderate computing times. For example, a high-resolution air flow model for a room (CFD) can be coupled with a simplified air duct net model of a HVAC system (Modelica), whereas the Modelica model represents the intelligent “boundary conditions” of the room model.

NOMENCLATURE

h_{tot}	specific total enthalpy [J/kg]
p	pressure [Pa]
Re	Reynolds number [-]
t	time [s]
T	temperature [K]
\mathbf{U}	vector of velocity $U_{x,y,z}$ [m/s]
\bar{w}	mean velocity in flow direction [m/s]
λ	thermal conductivity [W/(m · K)]
ρ	density [kg/m ³]
τ	molecular stress tensor [Pa]
ζ	friction factor [-]

REFERENCES

- CFX, 2007: ANSYS CFX Release 11.0, ANSYS, Inc.
- CFX, 2006: ANSYS CFX-Solver Theory Guide, ANSYS, Inc.
- Klein-Robbenhaar, C., Köcher, R.; Nytsch, C.; Waldhoff, C. 2000: Numerische Simulation von Wärmenetzen. Simulation des Betriebsverhaltens von Wärmenetzen mit Integration der versorgten Gebäude. Brennstoff-Wärme-Kraft 9/2000, VDI-Verlag, Düsseldorf.
- Kossel, R.; Tegethoff, W.; Bodmann, M.; Lemke, N. 2006: Simulation of complex systems using Modelica and tool coupling. In proceeding: 5th International Modelica Conference, September 4th - 5th, Vienna.
- Ljubijankić, M.; Nytsch-Geusen, C. 2008: Thermohydraulic simulation of solar thermal systems with Modelica. In proceedings: 18. Symposium Thermische Solarenergie, OTTI-Technologiekolleg, Regensburg.
- Modelica, 2007. Language specification Modelica 3.0, Modelica Association.