

Comparison of Pipe Flow Vs. CFD Module for Homogeneous Liquid Delivery to a Tank in a Cooling Reactor

L. Leis¹, D. Kray¹

¹Institute for Energy System Technology INES, University of Applied Sciences, Offenburg, Germany

Abstract

We consider the fluid dynamics problem of homogeneously filling a large tank by multiple inlets fed from a piping system. Suitable pipe geometries must be chosen so that the volume flow is constant for all inlets. In this work, we compare both for a piping system as depicted in fig. 1: A main pipe is feeding four secondary pipes that deliver the liquid to the tank. The boundary conditions are the inlet velocity v_{in} at the main pipe entrance and $p=p_{atm}$ at the rising pipe ends 1-4.

The target is to design a pipe systems that leads to homogeneous outflow velocities at the rising pipe ends 1 to 4. For each simulation, we calculate the homogeneity as $H=(v_{max}(1-4)-v_{min}(1-4))/v_{average}(1-4)$.

In order to assess advantages and disadvantages of the Pipe Flow simplifications in COMSOL Multiphysics®, we perform comparative simulations and record mass conservation, convergence behavior, memory allocation and computing time. This will allow us to determine the requirements (laminar / turbulent flow, Re number, pipe geometry, adapted loss factors) for successful Pipe Flow Module application for this problem.

For the Pipe Flow Module calculations, we have to choose loss factors K_{main} and K_{side} for the T junctions as well as K for the 90° bend at the rising pipe 4. Also, we use round geometry of the pipes without surface friction and Churchill friction model with vanishing surface roughness. The CFD model requires much less parameters as the real geometry is entered without using any lumped loss factors e.g.

Stationary studies are conducted with varying pipe geometries and inlet velocities v_{in} . The results are average pipe velocities (Pipe Flow) and local flow velocities (CFD) that are used to test for global mass conservation (total volume inflow and outflow) and to calculate homogeneity H as defined above.

The first simulations are conducted using laminar flow conditions ($Re \ll 2300$) and geometries for maximum homogeneity (large main pipe, small rising pipes) and maximum inhomogeneity (main and rising pipes with identical diameter). Mesh studies are conducted to test that the solutions are mesh independent.

Simulations with the Pipe Flow Module are dramatically faster than using the CFD Module, convergence is achieved more easily compared to the Laminar Flow physics interface (CFD Module). The loss of local information is irrelevant if we just want to get average outflow volume flows from the different rising pipes. However, both modules deliver strongly differing results in case of large inhomogeneity. We will study what are the prerequisites that both model calculations match.

Figures used in the abstract

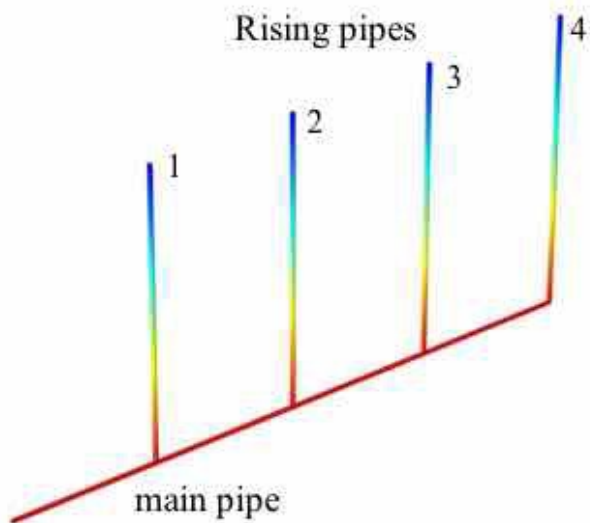


Figure 1: Simulation model as defined in Pipe Flow Module.