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

L. Leis, D. Kray
Institute for Energy System Technology INES
University of Applied Sciences, Offenburg, Germany

Introduction: We consider the fluid dynamics problem of homogenously filling a large tank through multiple outlets of a piping system. Suitable pipe geometries must be chosen so that the volume flow is constant at all the outlets. Figure 1 shows the main pipe which is feeding the four secondary pipes with nozzles that deliver liquid to the cooling tank.

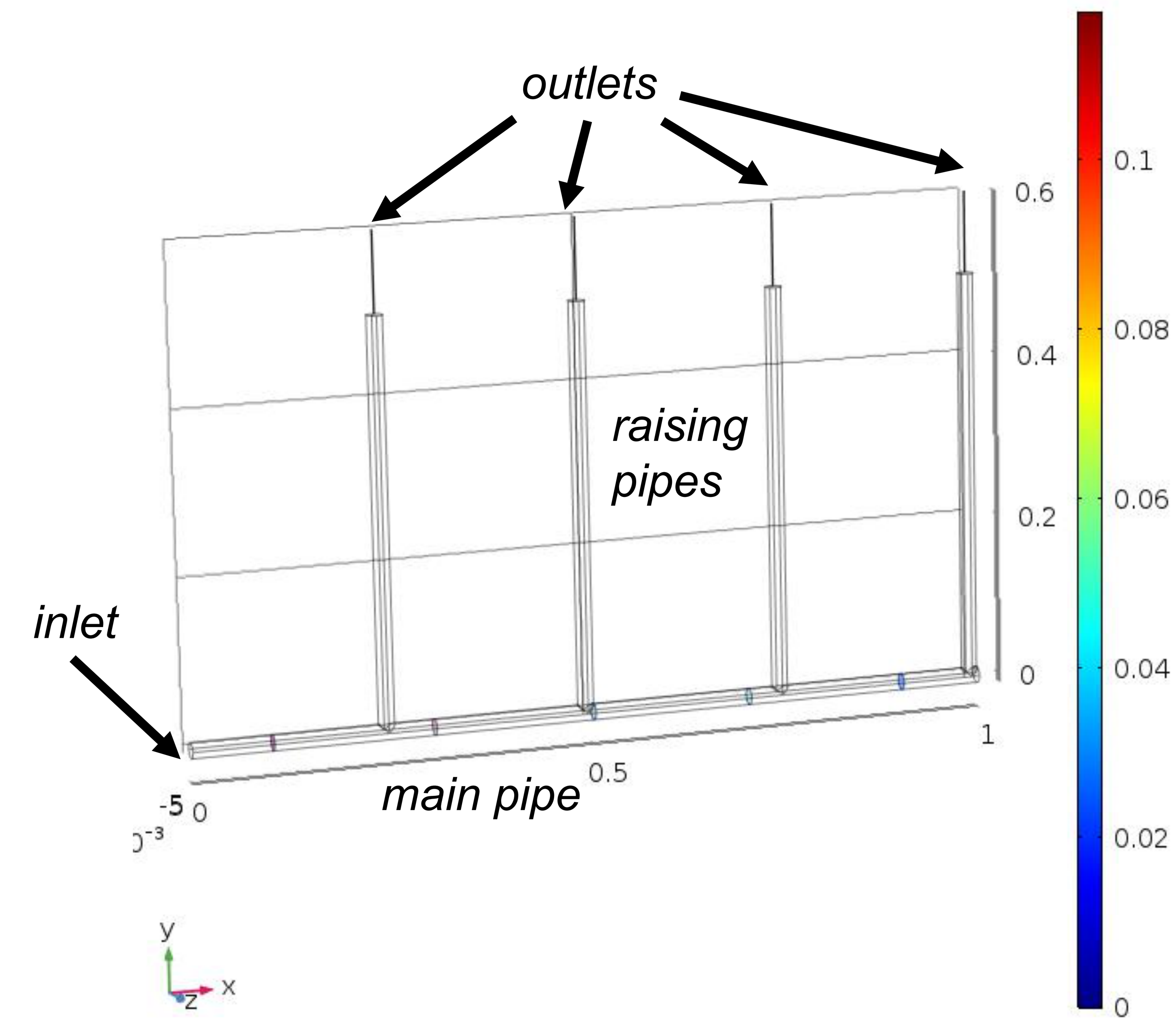

Figure 1. Simulation model as defined in CFD Module.

Computational Methods: The first step is to simulate one T-junction in the CFD module to calculate the values for the pressure loss coefficient ζ for the main and side branches.

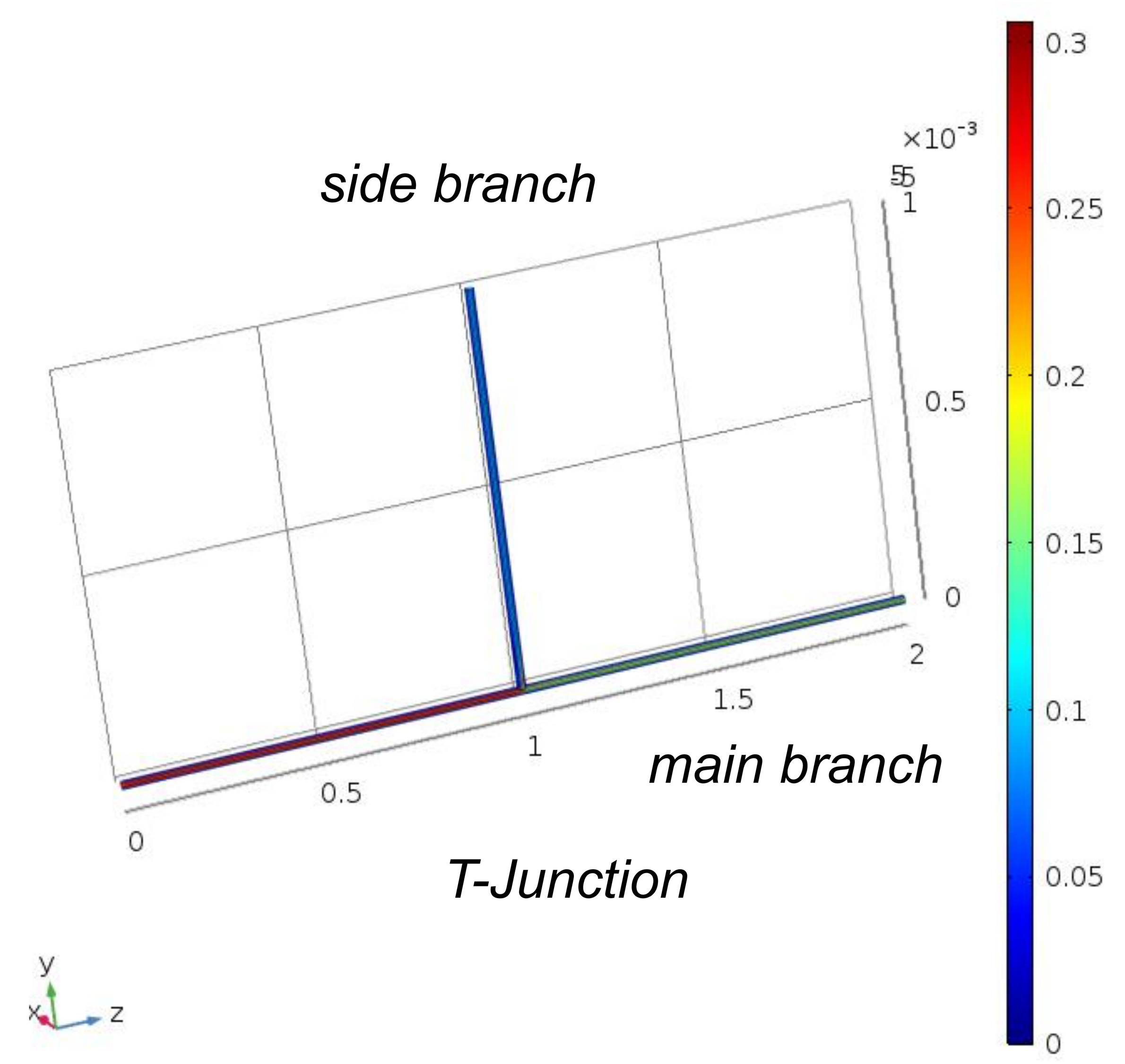

Figure 2. Simulation of a T-junction in the CFD module to calculate ζ.

By using the calculated velocities of the CFD module, the relative volume flow to the side branch can be computed. Then we vary in the Pipe Flow module to match the CFD module results.

$$\frac{\dot{V}_a}{\dot{V}} = \frac{v_1}{(v_1 + v_2)}$$

The calculated pressure loss coefficient are plotted versus the reynolds number.

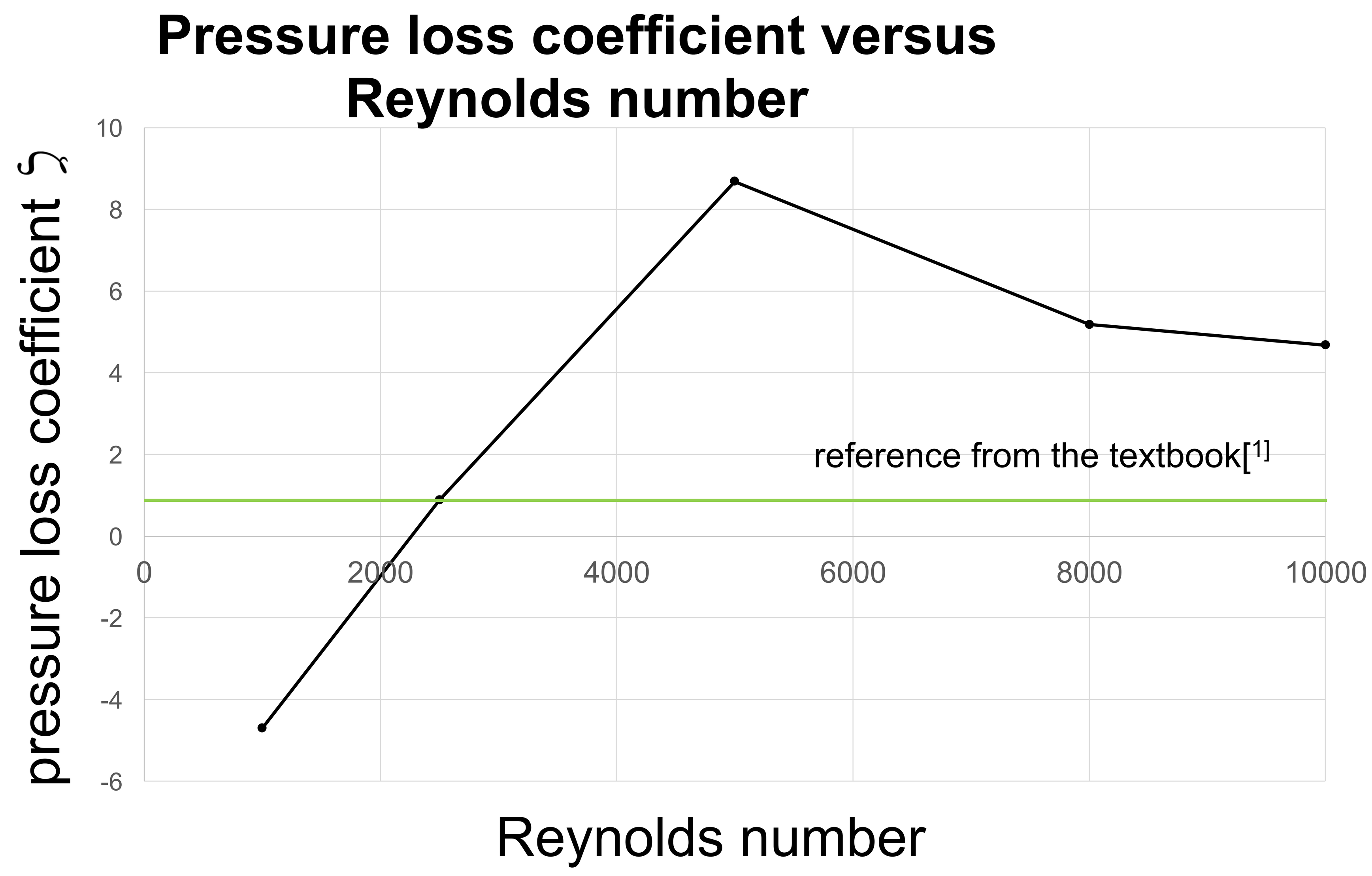

Figure 3. Pressure loss coefficient ζ (Zeta) against Reynolds number .

A parametric sweep was carried out to find the optimum diameters for the 4 outlet nozzles for homogeneous outflow.

$$D = \frac{\dot{V}_{max}(1-4) - \dot{V}_{min}(1-4)}{\dot{V}_{average}(1-4)}$$

Results

Optimal inner diameter nozzles: 3/ 3/ 3/ 5 [mm]		
laminar flow, v= 0.1 [m/s] Re= 1000	**Pipe Flow**	**CFD**
Divergence D	0.93	1.48
Computation time	7 s	4 min 7 s
Memory usage	1.1 / 6.15 GB	3.34 / 10.4 GB
turbulent flow *v= 1 [m/s] Re = 5000*		
Divergence D	0.93	1.44
Computation time	9 s	11 min 5 s
Memory usage	1.21 / 6.07	3.05 / 9.2 GB

Table 2. Comparison of Pipe Flow and CFD Module in terms of divergence, computation time and memory usage for a homogenous outflow.

Conclusion: Simulations with the Pipe Flow module are dramatically faster than using the CFD module, convergence is achieved more easily compared to the Laminar Flow (CFD) module. For the simulation in Pipe Flow it is important to estimate the exact pressure loss coefficients.

References:
1. K.-H. Grote, J.Feldhusen, Taschenbuch für Maschinenbau, 21. Auflage

This work has been supported by the German Federal Ministry for Economic Affairs and Energy under contract no. ZF4098906ZG6