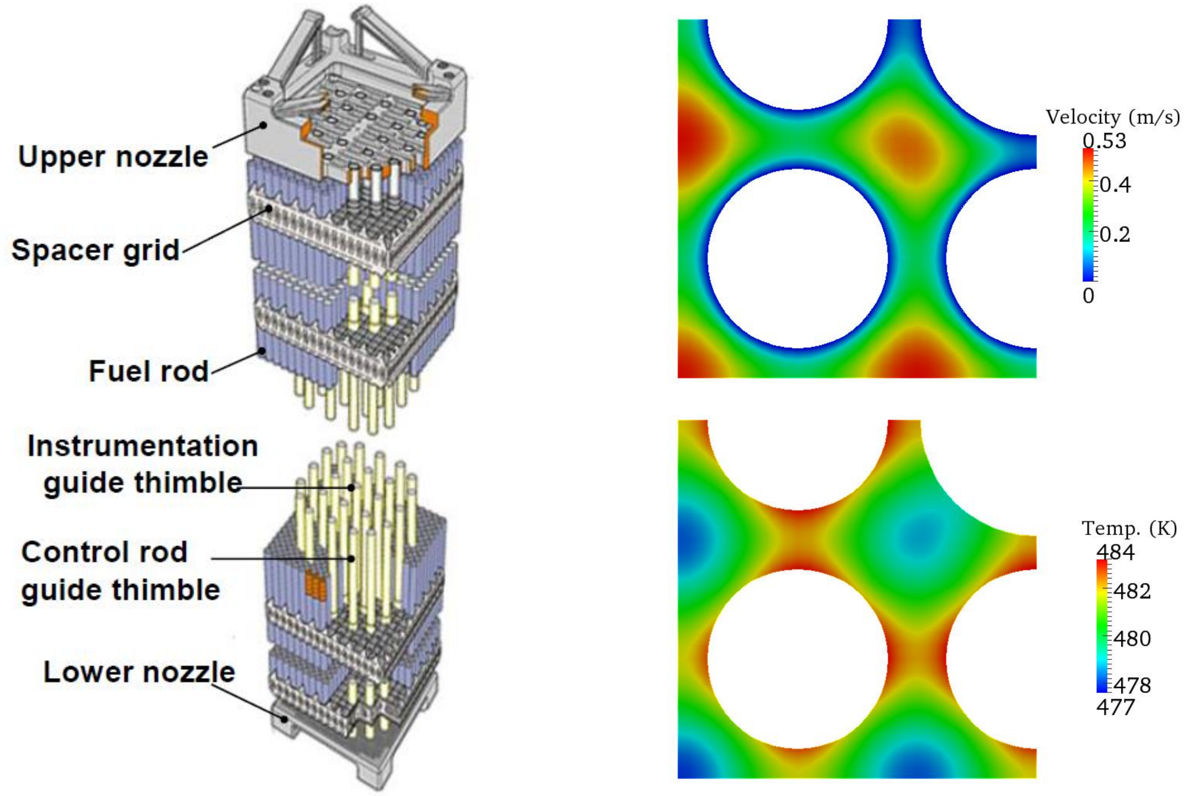
SIMULATION OF THERMAL-HYDRAULIC PHENOMENA IN A SPENT FUEL POOL OF A NUCLEAR POWER PLANT

Daniele Cozzani, Italy
Supervisor: C. Benocci. Advisor: Ph. Planquart.

In this project, the total loss of coolant accident (LOCA) in a spent fuel pool of a nuclear power plant has been investigated by CFD tools. Natural convection heat transfer in a single fuel assembly has been simulated and compared to the results of a lumped parameters code (*MELCOR*), which is currently used to analyse such accidents at system level.
A nuclear fuel assembly is a complex object which is difficult to be simulated in detail. A solver capable to handle density variation and transient flow is needed; in the present investigation an available explicit solver (*OpenFOAM buoyantPimpleFoam*) has been applied but the complete simulation of the accident, whose overall time scale is in the order of several hours, has shown itself to be unfeasible.

However, the wide difference between the time scales (accident duration and flow through the assembly) makes a quasi-steady approximation possible, where the single fluid phase within the assembly is simulated applying a constant heat flux boundary condition at the solid interface. This simulation has been performed for some prototypical channels, instead of the whole assembly to decrease the calculation time. A porous model has been applied to simulate the effects of assembly structure details which could not be resolved by the numerical grid. This approach has provided flow rates which globally match MELCOR output. CFD has shown to be capable to predict hot spots in the bundle having higher temperature than MELCOR results.



Left: Sketch of a 17x17 PWR fuel assembly (Tokyo Institute of Technology).
Right: velocity and temperature fields in a section of channels along a control rod guide tube.