

ONE- AND TWO-PHASE COUPLING OF OPENFOAM WITH THE THERMAL HYDRAULIC CODE ATHLET FOR NUCLEAR SAFETY ANALYSES

JOACHIM HERB¹, FLAVIUS CHIRIAC²

¹*Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Joachim.Herb@grs.de*

²*Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Flavius.Chiriac@grs.de*

Keywords: *Coupling 1D/3D, nuclear safety, thermal-hydraulics, one-/two phase flows*

For nuclear power plants it is necessary to simulate a wide range of operating conditions to prove that regulatory limits and requirements are obeyed during all operating conditions from normal operation to (severe) accident conditions. Thermal-hydraulic codes developed during the last decades have been successfully used which are based on one dimensional models of the thermal-hydraulic behaviour of the plant. At GRS the thermal-hydraulic simulation code ATHLET has been developed to cover the whole spectrum of design basis and beyond design basis accidents for pressurized water reactors and boiling water reactors [1]. Nevertheless, some safety related questions require the simulation of three dimensional effects like the analyses of high-cycle thermal fatigue induced by turbulent mixing, stratification effects of coolants with different temperatures or the mixing of coolants with different boron acid concentrations.

In the last few years OpenFOAM has proven its applicability in the field of nuclear (safety) simulations demonstrated by more than 50 publications in peer-reviewed journals [2]. Therefore, OpenFOAM is now an accepted choice for the use in CFD simulations in the nuclear field.

To reduce the necessary computational costs of three dimensional simulations it is desirable to reduce the areas to be simulated by CFD to only those where three dimensional effects are important and use a thermal-hydraulic code for the rest of the flow domain. Therefore GRS developed an interface for ATHLET which allows the coupling with CFD codes.

Besides a coupling with the commercial code ANSYS CFX [3] also a first version of a coupling with OpenFOAM based on a shared library approach was presented at the 9th OpenFOAM Workshop [4]. This first implementation of the coupling between ATHLET and OpenFOAM was implemented by compiling the FORTRAN code ATHLET as shared library and linking it to the modified OpenFOAM solver executable. This choice proved inflexible, e. g. due to the strong dependency on the used FORTRAN and C++ compilers and limited debugging options. To gain more flexibility the two codes were separated into different processes.

For OpenFOAM a generic coupling library has been developed. This allows minimizing the necessary changes to an OpenFOAM solver which should be coupled. The data exchange with the coupled solver (e. g. ATHLET) is implemented by boundary conditions which can be used like normal OpenFOAM boundary conditions. The actual data exchange between the solvers is implemented by network sockets using the Open Source libraries zeromq [5] and msgpack [6]. They allow an efficient data exchange between different processes, even if these process run on different computer nodes, e. g. on an HPC cluster.

The ATHLET side of the coupling interface has been implemented by wrapping it with a Python interface allowing communicating via zeromq/msgpack.

The coupling is controlled by the OpenFOAM solver which exchanges data with ATHLET several times within one PIMPLE time step and executes the corresponding ATHLET simulation step (see Figure 1). This semi-implicit coupling has shown that it increases the stability of the simulation.

This architecture of the coupling allowed the verification of the implementation by using “dummy” solvers on one side of the solver greatly reducing the debugging effort.

At the moment two OpenFOAM solvers have been extended with coupling capabilities: the one-phase solver buoyantPimpleFoam and the two-phase solver twoPhaseEulerFoam. Both couplings have been validated by comparing simulations with experiments. For the validation of the one-phase coupling an experiment of the TALL facility was selected. In the experiment a lead-bismuth-eutectic was used a working fluid and the transition from forced to natural circulation was simulated. For the validation of the two-phase coupling a simple bubble column experiment was used.

The generic architecture of the coupling allows using it also for the coupling of OpenFOAM with other (one-dimensional) solvers. GRS is planning to release the source code of the OpenFOAM coupling side and a “dummy” coupled solver.

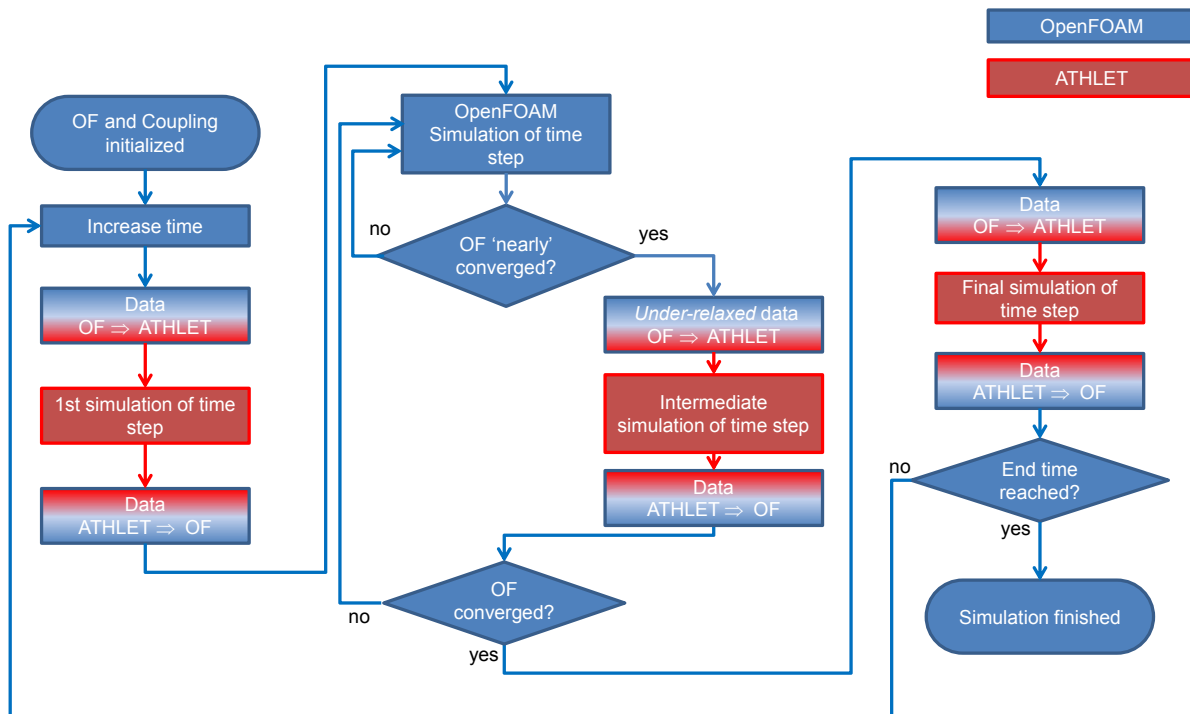


Figure 1: Architecture of the semi-implicit coupling between OpenFOAM and ATHLET

Acknowledgements

The authors would like to thank the German Federal Ministry for Economic Affairs and Energy for the support within the framework of the R&D projects RS1515 and RS1540.

References

- [1] Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH. *The Computer Code ATHLET*. Available: <http://www.grs.de/en/computer-code-athlet>.
- [2] *SIG Nuclear / Publications*. Available: http://openfoamwiki.net/index.php/SIG_Nuclear/_Publications.
- [3] A. Papukchiev and G. Lerchl. “Extension and Application of the Coupled 1D-3D Thermal-Hydraulic Code ATHLET-ANSYS CFX for the Simulation of Liquid Metal Coolant Flows in Advanced Reactor Concepts” in Proc. of the 2012 20th International Conference on Nuclear Engineering collocated with the ASME 2012 Power Conference ICONE20-POWER2012, July 30 - August 3, Anaheim, California, USA, 2012.
- [4] J. Herb. “Coupling of OpenFOAM with Thermo-Hydraulic Simulation Code ATHLET” presented at the 9th OpenFOAM Workshop, Zagreb, Croatia, 2014.
- [5] iMatix Corporation (2014). *Distributed Messaging – zeromq*. Available: <http://zeromq.org/>.
- [6] S. Furuhashi (2013). *MessagePack: It's like JSON. but fast and small*. Available: <http://msgpack.org/>.