OpenFOAM-Interactive (OFI): An Interface to Control Solvers in OpenFOAM

Arpit Singhal*, Raphael Schubert†, Adham Hashibon†

† Fraunhofer-Institut für Werkstoffmechanik IWM
Wöhlerstraße 11, 79108 Freiburg im Breisgau, Germany
E-mail: arpit.singhal@iwm.fraunhofer.de

ABSTRACT

We present a new method for coupling fluid and particle systems that works by directly manipulating the flow field variables, mimicking the presence of solid particles rather than adding additional force terms in the governing equations as, e.g., in the traditional immersed boundary method (IBM). We demonstrate an implementation based on the open source OpenFOAM [2] package.

The OpenFOAM-Interactive (OFI) presented here gives access to all internal field variables of the governing equations. This eases and facilitates complex computational and seamless data exchange and manipulation of the field variable. OFI contributes to reducing the time needed in creating the initial geometry and enables readily re-creating the geometry for the basic computational fluid dynamics (CFD) simulation steps.

The presented methodology is verified for two reference simple problems (i) obstruction to flow with bluff bodies and (ii) heat flow between a sphere and surrounding gas. The verified methodology is then applied to solve a realistic the problem of heat transfer to a gas through particle bed [1]. The particle bed is created “on the fly” with OFI, this facilitates studying fluid behavior over different particle configurations of the particle beds (i.e. porosity variation).

Figure 1. A schematic representation of the OFI architecture. A think controller layer is embedded in the OF solver which intercepts the execution of OF and alters the state seamlessly. The controller communicates through a pipe socket to the Parser which parses commands from the client. The parser in turn communicates through a socket with the client part where currently the user can make changes and issue commands. The Controller converts these commands into actions. The semantic front-end of the simulation package) (under development) and the CFD Wrapper provides a more streamlined plugin option for the interoperable material modelling workflow.

REFERENCES