Connecting OpenFOAM with an External Electromagnetic FDM Solver for Magnetofluiddynamic Simulations

P. Turewicz[1], M. Bibani[1], E. Baake[1]

Numerical simulations have become an essential tool in the fields of engineering. The wide range of various commercial numerical software packages covers almost every scientific field. However, a great disadvantage is that their commercial source codes are usually not freely accessible and the user has no opportunity to adapt the solver for specific or complex problems. This means that it can be complicated or even impossible to take certain influencing effects into account.

Open source numerical software packages available today do not provide a real alternative. Although the source code can be accessed, poor usability and performance make them unattractive. Currently the most promising open source package is OpenFOAM. It offers a great user comfort and a wide range of embedded solvers for different scientific problems. However, the absence of appropriate libraries to solve partial differential equations (PDE) for complex vector fields makes OpenFOAM unsuitable for electromagnetic or magnetofluiddynamic field problems.

Against this background, the objective of this work was to evaluate the possibility of coupling OpenFOAM with a self-coded external electromagnetic solver based on the finite difference method (FDM). For this purpose, the flow of liquid Wood´s metal driven by the electromotive forces inside an induction crucible furnace (ICF) was calculated. The simulation model corresponds to the installation presented in [1].

Connecting Ampere's law with Faraday's law and using the axisymmetry of the problem, the following equation for the vector potential A can be obtained:

$$\frac{\partial}{\partial \mathrm{r}}\left(\frac{1}{r}\frac{\partial r\mathrm{A}}{\partial r}\right)+\frac{\partial^2 \mathrm{A}}{\partial z^2}-\mathrm{j}\omega\mu\kappa\mathrm{A}=-\mu\,\mathrm{J} \qquad (1)$$

where ω is the angular frequency, μ the vacuum permeability and J the current density. This partial differential equation was approximated using the Taylor series and numerically solved applying the flux parallel boundary condition to all boundaries. Taking in to consideration the equation below

$$\mathrm{f_{EM}}=\mathrm{J}\times\mathrm{B}=\nabla\times\left(\frac{1}{\mu}\mathrm{B}\right)\times\mathrm{B} \qquad \text{with} \quad \mathrm{B}=\nabla\times\mathrm{A} \qquad (2)$$

the Lorentz force was calculated for every node and exported to a separate file.

A standard steady state solver for buoyant, turbulent flow of incompressible fluids using the k-ε model, called buoyantBoussinesqSimpleFoam, was selected for the flow simulation. Neglecting the buoyancy and adding the EM force density, the flow can be described by the following Navier-Stokes-equation:

$$v_j\frac{\partial v_i}{\partial x_j}=\frac{1}{\rho}\frac{\partial p}{\partial x_i}+\nu\left(\frac{\partial^2 v_i}{\partial x_j \partial x_j}\right)+\mathrm{f_{EM\,i}} \qquad i,j=1,2,3 \qquad (3)$$

where *vj* are the components of velocity, *p* is the pressure and υ the kinematic viscosity of the melt. The obtained results were compared and verified with good coincidence to the experimental findings in [1].

References

[1] Baake, E. (1994): Grenzleistungs- und Aufkohlungsverhalten von Induktions-Tiegelöfen, Thesis, Leibniz University of Hanover, Hanover.

[1]Institute for Electrothermal Processes, Leibniz University of Hannover, Wilhem-Busch-Str.4, D-30167 Hannover, Germany