

PARALLEL IMPLEMENTATION OF A GEOMETRICAL RECONSTRUCTION INTERFACE ALGORITHM OVER OPENFOAM(R)

Santiago Márquez Damián^{a,b}, Cesar I. Pairetti^c and Norberto M. Nigro^{a,d}

^a*Research Center for Computational Methods (CIMEC), CONICET/UNL, Predio CONICET Santa Fe -
Colectora Ruta Nac Nro 168, Paraje El Pozo, Santa Fe, Argentina santiagomarquezd@gmail.com,
<http://www.cimec.org.ar>*

^b*Universidad Tecnológica Nacional, FRSF, Lavaise 610, Santa Fe, Argentina
<http://www.frsf.utn.edu.ar/>*

^c*Facultad de Ciencias Exactas, Ingeniería y Agrimensura, UNR, Av. Pellegrini 250 - Planta Baja,
Rosario, Argentina <http://web.fceia.unr.edu.ar/> - CONICET*

^d*Facultad de Ingeniería y Ciencias Hídricas, UNL, Ciudad Universitaria. Ruta Nacional Nro 168
Paraje El Pozo, Santa Fe, Argentina <http://www.fich.unl.edu.ar/>*

Keywords: Piecewise-Linear Interface Calculation, Volume of Fluid, Finite Volume Method

Abstract. Even when the Piecewise-Linear Interface Calculation algorithm is a well established technique for interphase reconstruction related to the Volume of Fluid method and is implemented in several privative codes for arbitrary meshes, no freely-distributed version is available to the date for the OpenFOAM(R) libraries suite. This works presents implementation details of this technique over OpenFOAM(R) to be used on parallel platforms. Much of them are related with the underlying numerics and the fundamental requirement of local and global conservativeness. Two main aspects are covered in this work: the computation of face fluxes by intersection of swept volumes and interface cells and the calculation of curvatures in each cell. While the first aspect does not present much difficulties, curvatures computation results in an interesting case in how to manage a problem not well suited for message passing parallelization.