Proceedings of CHT-17
ICHMT International Symposium on Advances in Computational Heat Transfer
May 28-June 1, 2017, Napoli, Italy

CHT-17-209

LESSONS FROM ANUPRAVAHA: TOWARDS A GENERAL PURPOSE COMPUTATIONAL FRAMEWORK ON HYBRID UNSTRUCTURED MESHES FOR MULTI-PHYSICS APPLICATIONS

Jai Manik, Mukul Parmanand, Subrat Kotoky, Preetirekha Borgohain, Amaresh Dalal[§], Ganesh Natarajan
Department of Mechanical Engineering, Indian Institute of Technology Guwahati, 781039, India
[§]Corresponding author. Fax: +91 361 2582699, Email: amaresh@iitg.ernet.in

ABSTRACT

In the present work, the development of a general purpose, indigenous computational fluid dynamics code, ANUPRAVAHA, has been described for simulations of three-dimensional multi-physics problems. The code solves the governing fluid flow and heat transfer equations in a collocated, cell-centered, finite-volume framework, independent of the grid topology. The equations are discretised and solved in an implicit segregated manner using a fractional step approach. The momentum equations cast in a conservative form are linearized using Picard linearization and solved for auxiliary momenta at the cell centers. A new gradient reconstruction approach which is consistent and interpolation-free using a compact stencil is also employed for centroidal gradient computations. Time accurate simulations for unsteady problems are guaranteed using a three-point backward differencing scheme which gives second-order temporal accuracy. Governing equations for scalar quantities such as temperature (energy), concentration (species), volume fraction and turbulence quantities (such as kinetic energy and specific dissipation) are cast in a conservative transport form and discretised akin to the momentum equations. Several two-equation models using eddy-viscosity hypothesis have been implemented for turbulent flows. The pressure is then obtained by solving a Poisson equation obtained by enforcing the divergence condition. A novel generic momentum interpolation approach is applied to ensure proper pressure-velocity coupling, in the presence of large body forces and singular interfacial forces even on hybrid grids. While the solver is primarily devised to handle incompressible flows, the methodology has also been extended to weakly compressible flows, typical of non-Boussinesq convection and radiation, using a low Mach number formulation (Paolucci [1982]). The developed solver has been validated for simulating various multi-physics problems like conjugate heat transfer; droplet rising, falling and splashing; radiative heat transfer problems coupled with fluid flow and turbulent flows along with heat transfer.