Abstract Submitted
for the DFD07 Meeting of
The American Physical Society

An embedded boundary method with adaptive mesh refinement[1]

ELIAS BALARAS, MARCOS VANELLA, PARTICK RABENOLD, University of Maryland, College Park, MD, 20742, USA — An embedded boundary method with adaptive mesh refinements for fluid-structure interaction problems is presented. In the fluid solver the grid does not need to conform to a complex body, which is allowed to move through the fixed grid undergoing large displacements. To adaptively refine/derefine the mesh a single-block, staggered, Cartesian grid solver is employed on a hierarchy of sub-grids with varying spatial resolution. Each of these sub-grid blocks has a structured topology, and is part of a tree data structure that covers the entire computational domain. The Paramesh toolkit is used for the implementation of the adaptive refinement process. The package creates and maintains the hierarchy of sub-grid blocks, with each block containing a fixed number of grid points. A strong coupling scheme is adopted, where the fluid and the structure are treated as elements of a single dynamical system, and all of the governing equations are integrated simultaneously, and interactively in the time domain. A demonstration of the accuracy and range of applicability of the method will be given for a variety of laminar and turbulent flow problems.

[1]Research supported by the National Science Foundation and the Office of Naval Research

Elias Balaras
University of Maryland, College Park, MD, 20742

Date submitted: 06 Aug 2007 Electronic form version 1.4