Abstract Submitted
for the DFD19 Meeting of
The American Physical Society

An immersed boundary method with mesh refinement for turbulent flow simulations SHAO-CHING HUANG, UCLA — The immersed boundary method has been one of the preferred approaches for direct numerical simulation of turbulence that involves irregular or moving boundaries in the computational domain. The method not only bypasses curvilinear or body-fitted mesh generation but also enables the possibility of using fast solvers in the solution procedure, such as FFT-based methods for Poisson equation, making the method highly efficient. However, the convenience of immersed boundary method can be plagued by lack of resolution in the regions where truncation error is large due to the nature of the underlying structured mesh; increasing the local resolution to resolve sharp gradients usually results in rapid growth of the required total computational resources. To address this problem and to enable large-scale computations on parallel computers, an immersed boundary method with mesh refinement capability is developed. A tree-based data structure, as opposed to a block-structured one, is considered to manage the mesh refinement. A number of selected flow cases are used to evaluate the method's accuracy and performance, and to compare to its Cartesian-mesh counterpart.

Shao-Ching Huang
UCLA

Date submitted: 01 Aug 2019 Electronic form version 1.4