Abstract Submitted
for the DFD16 Meeting of
The American Physical Society

An engineering closure for heavily under-resolved coarse-grid CFD in large applications ANDREAS G CLASS, FUJIANG YU, THOMAS JORDAN, Karlsruhe Institue of Technology — Even though high performance computation allows very detailed description of a wide range of scales in scientific computations, engineering simulations used for design studies commonly merely resolve the large scales thus speeding up simulation time. The coarse-grid CFD (CGCFD) methodology is developed for flows with repeated flow patterns as often observed in heat exchangers or porous structures. It is proposed to use inviscid Euler equations on a very coarse numerical mesh. This coarse mesh needs not to conform to the geometry in all details. To reinstall physics on all smaller scales cheap subgrid models are employed. Subgrid models are systematically constructed by analyzing well-resolved generic representative simulations. By varying the flow conditions in these simulations correlations are obtained. These comprehend for each individual coarse mesh cell a volume force vector and volume porosity. Moreover, for all vertices, surface porosities are derived. CGCFD is related to the immersed boundary method as both exploit volume forces and non-body conformal meshes. Yet, CGCFD differs with respect to the coarser mesh and the use of Euler equations. We will describe the methodology based on a simple test case and the application of the method to a 127 pin wire-wrap fuel bundle.

Andreas G. Class
Karlsruhe Institue of Technology

Date submitted: 30 Sep 2016 Electronic form version 1.4