Abstract Submitted
for the DFD05 Meeting of
The American Physical Society

A new solver for Large-Eddy Simulations of turbulent premixed combustion in complex geometries. VINCENT MOUREAU, HEINZ PITSCH, CTR, Stanford University, Stanford, CA 94305-3030 — A new low Mach number numerical scheme for Large Eddy Simulation of premixed turbulent combustion in complex flow geometries has been developed. A Ghost-Fluid like method is introduced that allows the solver to accurately handle large density jumps and small flame brush thickness while using non-dissipative centered schemes for spatial integration. By coupling this solver with the G-equation combustion model, the flame dynamics are well described without resolving the flame front structure. The new numerical method is implemented in the variable-density unstructured flow solver CDP developed at the Center for Turbulence Research. Verification of the numerical methods will be presented in various simple test cases. The formalism has been used to compute a swirling turbulent premixed flame in an industrial premixed combustor. The solver proves to be very robust and to predict velocity statistics with good accuracy. It also captures the main hydrodynamic instabilities caused by the strong swirling motion.

Vincent Moureau
CTR, Stanford University, Stanford, CA 94305-3030

Date submitted: 15 Aug 2005 Electronic form version 1.4