Abstract Submitted
for the DFD15 Meeting of
The American Physical Society

OpenFOAM for beginners IVAYLO NEDYALKOV, MARTIN WOSNIK, University of New Hampshire — OpenFOAM has gained significant popularity in academia and industry, but is still not widely introduced to CFD novices e.g., undergraduate students. This is likely due to the steep learning curve of the software. A relatively short tutorial was developed to introduce students to the basic features of OpenFOAM, allowing them to modify and create simulations, and to better understand other online resources. The tutorial has been successfully introduced to students working on undergraduate capstone projects at the University of New Hampshire and parts of it were presented at a tech-camp for K-12 students.

Ivaylo Nedyalkov
University of New Hampshire

Date submitted: 01 Aug 2015 Electronic form version 1.4