Abstract Submitted
for the DFD16 Meeting of
The American Physical Society

SFO-Project: The New Generation of Sharable, Editable and Open-Access CFD Tutorials TEYMOUR JAVAHERCHI, ARDESHIR JAVAHERCHI, ALBERTO ALISEDA, University of Washington — One of the most common approaches to develop a Computational Fluid Dynamic (CFD) simulation for a new case study of interest is to search for the most similar, previously developed and validated CFD simulation among other works. A simple search would result into a pool of written/visual tutorials. However, users should spend significant amount of time and effort to find the most correct, compatible and valid tutorial in this pool and further modify it toward their simulation of interest. SFO is an open-source project with the core idea of saving the above-mentioned time and effort. This is done via documenting/sharing scientific and methodological approaches to develop CFD simulations for a wide spectrum of fundamental and industrial case studies in three different CFD solvers; STAR-CCM+, FLUENT and Open FOAM (SFO). All of the steps and required files of these tutorials are accessible and editable under the common roof of Github (a web-based Git repository hosting service). In this presentation we will present the current library of 20+ developed CFD tutorials, discuss the idea and benefit of using them, their educational values and explain how the next generation of open-access and live resource of CFD tutorials can be built further hand-in-hand within our community.

Teymour Javaherchi
University of Washington

Date submitted: 02 Aug 2016 Electronic form version 1.4