Title: “I don’t even know where to start with OpenFOAM!”

Instructor: David Boger and anyone else I can find

Description: If you are frustrated and feel as if you’re having trouble getting started with OpenFOAM, then this class is intended to help you. Topics of discussion will be decided by the participants. Possible topics include running tutorials, modifying and compiling solvers and utilities, finding what you’re looking for, using Git, using Doxygen, installing OpenFOAM, starting with paraFoam, understanding the format of codes, files, or directories, terminology or anything else you can think to ask.

Participants are asked to provide some questions to the organizers upon registration so that we are better able to prepare. The hope is that we will be able to answer questions that are of a basic and somewhat general nature. It is highly unlikely that we will be able to dedicate one-on-one time to work with individuals. We anticipate that the class will be hands-on and interactive (participants will need to provide their own laptop and be able to boot from the USB stick that will be provided to participants), but we also expect to answer questions from the front of the room so that everyone can benefit from the discussion.

The organizers will put together some presentation material on running tutorials and modifying and compiling the code that can be used to stimulate discussion if necessary.

Duration: 90 minutes

Date and Location: 13 June 2011, Penn State University, State College, PA, USA