

# CFD WITH OPENSOURCE SOFTWARE

A COURSE AT CHALMERS UNIVERSITY OF TECHNOLOGY  
TAUGHT BY HÅKAN NILSSON

---

Project work:

## A dieselFoam tutorial

---

Developed for OpenFOAM-1.5.x  
Requires: PyFoam

*Author:*  
Per CARLSSON

*Peer reviewed by:*  
NAIXIAN LU  
HÅKAN NILSSON

Disclaimer: This is a student project work, done as part of a course where OpenFOAM and some other OpenSource software are introduced to the students. Any reader should be aware that it might not be free of errors. Still, it might be useful for someone who would like learn some details similar to the ones presented in the report and in the accompanying files. The material has gone through a review process. The role of the reviewer is to go through the tutorial and make sure that it works, that it is possible to follow, and to some extent correct the writing. The reviewer has no responsibility for the contents.

November 5, 2012