

4th Short Course on Computational Fluid Dynamics for Industry

August 11–12, 2016

Department of
Mechanical Engineering



FACULTY OF
ENGINEERING



UNIVERSITY OF
ALBERTA

Course Description

CFD for Industry is a short course that aims at training industry professionals in the proper use of Computational Fluid Dynamics software. Today's CFD packages are increasingly powerful and robust, commercial packages also user-friendly, but their increased ability to deliver a converged result may be misleading. The nonlinear nature of fluid mechanics means that relatively small mistakes in the simulation setup may result in fundamentally wrong flow fields leading to faulty design.

This short course focuses on the methodical process of setting up correctly your CFD simulation and of evaluating the quality of your solution, regardless of the software package used. Qualitative and quantitative criteria for selecting the best grid, domain size, convergence threshold, and other parameters will be taught and demonstrated during practice sessions.

The short course will cover a review of fundamentals, solution procedure, convergence analysis, mesh generation, physical modelling, verification and validation, and best practices. The theory will be complemented by improved hands-on practice sessions using the OpenFOAM® CFD software package, exemplifying concepts learned.

Participants in the short course can apply for Continuing Professional Development credits with the Association of Professional Engineers, Geologists and Geophysicists of Alberta.

The Lecturers

Carlos F. Lange is an Associate Professor of Mechanical Engineering at the University of Alberta. With almost 30 years experience in CFD, he has modelled engineering, biomedical and atmospheric flows. Lange was a Science Team Member of the Phoenix Mars Mission and with his team performed the first CFD characterization of a Mars lander. In 2001, he founded the CFD-Lab at the University of Alberta and 3 years later introduced the undergraduate/graduate course "Applied CFD", which is now training 120 engineers every year in CFD and serves as basis for this short course.

Mohsen Bayati is the R&D Manager at Radix Innovation Corp., specialized in product development by the aid of CFD and CAD modelling. He obtained his MSc from the University of Alberta with research focused on the modelling of multiphase flows at the CFD-Lab. His main areas of expertise are multiphase flows, interface tracking, particle tracking, turbulent flows, high performance computing, and code development.

This offering is not approved or endorsed by OpenCFD Limited, the producer of the OpenFOAM software and owner of the OPENFOAM® and OpenCFD® trade marks.



EDMONTON · ALBERTA · CANADA

Short Course CFD for Industry

Please, register per e-mail sending the information below to carlos.lange@ualberta.ca by

August 5, 2016.

Alternatively, complete this form and return by the deadline with payment to:

Short Course CFD for Industry
10-203 Donadeo Innov. Centre for Engg.
9211-116 Street NW
University of Alberta
Edmonton, Alberta, Canada T6G 1H9

Name: _____

Company: _____

Title: _____

Mailing Address: _____

Phone: _____

E-mail: _____

Price: \$600+\$30 GST = \$630 CAD per person

- Payment form: ☐ Cash
☐ Cheque (payable to the University of Alberta)
☐ Purchase Order (Dept. will issue invoice)
☐ Visa/MasterCard (call 780-492-2950)

Registration and Fee

Register before the deadline by following the instructions in the attached form. Register early as space is limited.

The fee for the two day short course is
\$600 CAD + GST.

This includes course notes, a live and installable Linux appliance with OpenFOAM®, as well as access to workstations with CFD software during the practice sessions. Catering during coffee breaks and lunches is also included.

Dates and Location

August 11–12, 2016

5-013 Engg. Teaching and Learning Complex
9107 116 Street NW, U. Alberta, Edmonton

This offering is not approved or endorsed by OpenCFD Limited, the producer of the OpenFOAM software and owner of the OPENFOAM® and OpenCFD® trade marks.

Registration information is collected under the authority of the Freedom of Information and Protection of Privacy Act. The contact information each participant provides is required by the office to register them in the course, prepare material and courses for their use, plan for future courses and notify them of similar, upcoming courses offered by the department. Financial information is used to process applicable fees and is retained for future reference.

Schedule

Thursday, August 11, 2016

- 08:00 – 08:50 Basic Equations of Fluid Mechanics
09:00 – 09:50 Review of Numerical Methods
10:00 – 10:50 Equation Discretization
11:00 – 11:50 Intro to OpenFOAM
lunch break
13:00 – 13:50 Setup and Solution Procedure
14:00 – 14:50 Physical Modeling I -
Multiphase Flows, Porous Media
15:00 – 16:50 Mesh Generation, Problem Setup
and Post-Processing (practice)

Friday, August 12, 2016

- 08:00 – 08:50 Physical Modeling II - Turbulence
09:00 – 09:50 Convergence Analysis
10:00 – 11:50 Mesh Refinement and Convergence
(practice)
lunch break
13:00 – 13:50 Verification and Validation
14:00 – 14:50 Best Practices
15:00 – 16:50 Multiphase Flow: Setup and Solution
(theory and practice)