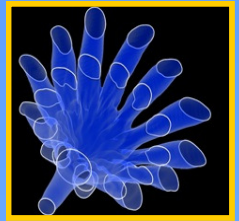




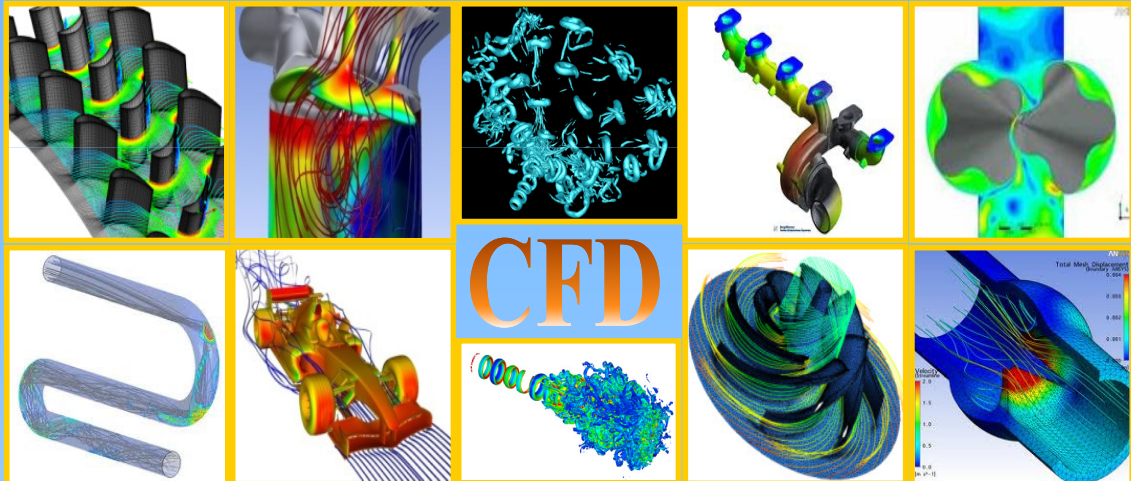
Self Financed Training Program on



Computational Fluid Dynamics (CFD) with OpenFOAM

Batch 1: November 19 – 23
and

Batch 2: December 8 – 12, 2016



Objectives:

- The objective of the present training program is to appreciate the open source CFD code OpenFOAM and simplify OpenFOAM use.
- This training gives a platform to the participant to understand OpenFOAM from basics to advanced to programming level.

Registration Details:

- Last date of registration:
 - Batch 1: November 11, 2016
 - Batch 2: November 28, 2016
- Fee includes training expenses, workshop kit, live USB drive, course material, lunch and morning - evening tea.
- Hostel room (Rs 100/- per day plus Rs 50/- per day bedding charges) or institute guest house room (Rs 1200/- per room) will be provided first come first serve basis.

Course fee:

- Student: Rs 10,000
- Faculty: Rs 15,000
- Others: Rs 25,000

Coordinator:

Dr. Trushar B. Gohil, Assistant Professor

Mobile: 88888 72132, Tel: (0712) 2801169, Email: trushar.gohil@gmail.com

<http://www.cfdtraining.org/cfd-with-openfoam.html>

Organised by

Department of Mechanical Engineering,

Visvesvaraya National Institute Of Technology (VNIT), South Ambazari Road, Nagpur-440010 (INDIA)

Disclaimer & Usage of trade marks

OpenFOAM® and OpenCFD® are registered trademarks of ESI. This offering is not approved or endorsed by OpenCFD Limited, the producer of the OpenFOAM software.