

Tentatives (*please bring your own laptop)

8.30AM : Registration

8.45AM : Setup Machine

9.20AM: Introduction to CFD

10.00AM: Introduction to OpenFOAM

10.30AM: Coffee Break

10.45AM: File Structures (folder1- constant)

- create geometry and mesh
- check mesh
- mesh visualisation

1.00PM: Break (Lunch- prayer)

2.00PM : File Structures (folder1- constant)

- Transport Properties
- Turbulence Properties

File Structures (folder2- system)

- Discretisation schemes
- Solution schemes
- execute OpenFOAM
-

File Structures (folder3- 0)

- Initial conditions

3.15PM : Coffee Break

3.30PM : Post Processing

- Sampling data
- visualisations

5.00PM: Close

**Speaker: Dr. Mohamed Sukri Mat Ali (MJIIT)
Mr. Low Lee Leong (UNITEN)*

Synopsis

It seems a lot of people are looking for free CFD solvers these days. Whether it's a business looking to cut costs or an individual looking to develop new skills without having to take out a second mortgage, quality CFD packages that are freely available are sought after commodities.

There are a few CFD codes, however, which are both free (as in no cost) and also open (as in, anyone can use it, and you can give it to anyone else). Of these, OpenFOAM is, the most advanced. With an active and growing community of users and developers, this free CFD tool seems well on its way to giving the commercial vendors a serious run for their money.

To register, please email to:
researchsukri@gmail.com

Organised by;

Malaysia-Japan International
Institute of Technology (MJIIT)
*Engineering the Nation With Precision For
Sustainable Development*

UTM Kuala Lumpur
Jalan Semarak, 54100 Kuala Lumpur.
+603 – 22031286 (Sukri)

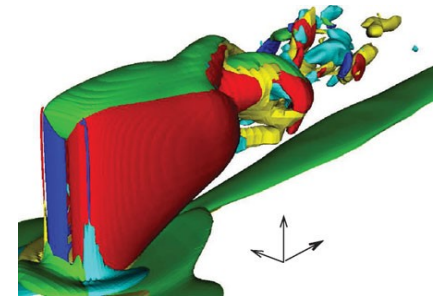
Co-organised by;

Universiti Tenaga Nasional (UNITEN)

Basic Course on OpenFOAM (for beginner)

Open source is the way to go!

17th July 2012 (Tuesday)
MJIIT, UTM Kuala Lumpur



*A one day training workshop designed for
CFD users, Engineers, Researches, and
Students*

*Fee RM30
limited to 20 seats*