Workshop: Starting OpenFOAM on Linux
Tuesday 5th December 2017
14:30-18:00
Computer Lab, Marine Technology Centre, UTM

OSCAE.Initiative and Marine Technology Centre (MTC) will run a workshop "*Starting OpenFOAM on Linux*". This workshop will introduce users to running OpenFOAM on Linux operating system. It caters for the uninitiated to Linux and OpenFOAM, introducing simple flow of commands running off Linux terminal to pre-process, solve and post-process CFD cases using OpenFOAM. Participants will receive a hands-on, interactive introduction to these commands on Ubuntu MATE operating system.

Who Should Attend?: This workshop is beneficial for final year undergraduates who are keen to undertake undergraduate project in research areas of Computational Fluid Dynamics (CFD) simulations using OpenFOAM software.

Registration Fee: Not Applicable

Limited seating: 10

Session 1: 14:30-16:00, Computer Lab, MTC
* OpenFOAM Linux Guide
* Sample Case from Tutorials

Tea break/`Asr: 16:00-16:30, Pantry/Musalla, MTC

Session 2: 16:30-18:00, Computer Lab, MTC
* Sample Case from Scratch
* Exploring Ship Resistance Case

Facilitator:
Assoc. Prof. Dr. Abu Hasan ABDULLAH

Contacts: Nur Syazani Norshaharudin   nursyazani1@gmail.com   (+6013-7227417)
Ummu Saiyidah Najihah   ummusaiyidah95@gmail.com   (+6018-2750663)