CAD-IT Consultants (M) Sdn. Bhd.

Free Event - ANSYS CFD Simulations

Date: 27th March, 2015

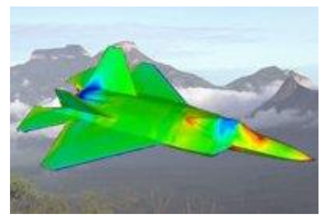
Time: 10:00 am to 12:00 pm (2 Hours)

Location: 605, Block A, Kelana Centre Point,
No. 3, Jalan SS 7/19, Kelana Jaya 47301 Petaling Jaya, Selangor, Malaysia

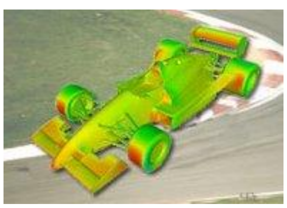
Fee: This is a Free Workshop - Online Registration is Required

Registration: Click Here

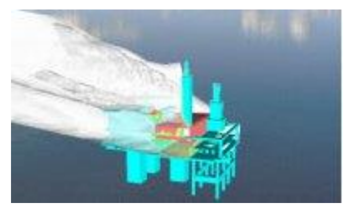
Synopsis



Computational Fluid Dynamics (CFD) has been an important tool in engineering designs and scientific research. The applications of CFD continue to grow in many industries such as oil and gas, automobile, building design, combustions and power generations, heat transfers in electronic cooling and HVAC, and etc.



ANSYS, since the Release 12, has integrated CFD solutions into a streamlined workflow using the Workbench platform. Many companies in Malaysia have benefited from this user-friendly **Workbench** platform and successfully integrated CFD solutions into their business.



Why Join Us

In this 2-hours workshop, we will introduce the integrated CFD workflow in ANSYS Workbench from geometry-creation, grid generation (meshing), CFD simulations, and post-processing.

Demo will be given, using both **Fluent** and **CFX**, to demonstrate the easy-to-use but yet powerful "drag-and-drop" feature in ANSYS Workbench.

Last but not least, you will have a hand-on experience using the latest ANSYS Release 16 during the event.

Who Should Join

Engineers and researches who are new to CFD and ANSYS solutions; existing (or previous CFD) users who are interested to find out more about CFD simulations using ANSYS Workbench.

CAD-IT Consultants (M) Sdn Bhd
www.cadit.com.sg

Bringing you tomorrow's technology… today!