CAD-IT Consultants (M) Sdn. Bhd.

ANSYS R16 Webinar

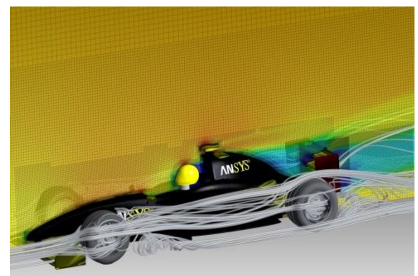
Maximizing Computational Fluid Dynamics (CFD) Productivity

Monday, 15th June, 2015
10:00 am (Kuala Lumpur Time)

Duration: 30 minutes

This is a FREE Online Event; Online Registration is required to join the Webinar.

ANSYS 16 delivers major advancements across the ANSYS's entire portfolio, including structures, fluids, electronics and systems engineering solutions – providing engineers with the ability to validate complete virtual prototypes.

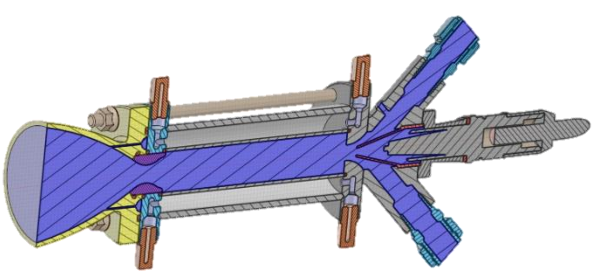


ANSYS CFD Simulations

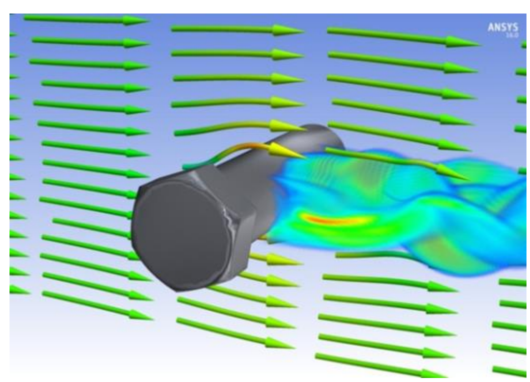
ANSYS Fluent and ANSYS CFX, are the premier software solutions for computational fluid dynamics (CFD) simulations, with the ability to simulate aerodynamics, complex combustion, hydrodynamics, mixtures of liquids/solids/gas, and particle dispersions. In the ANSYS 16.0 Release, ANSYS have incorporated the enhanced capabilities in the areas of pre-processing and user environment and usability – to continually improve the productivity of the ANSYS CFD solutions.

*The 3**0-minutes** Webinar will provide the **UPDATE** and **DEMO** highlighting the innovations for CFD productivity in the Release 16 including:*

- *High Performance Computing in ANSYS Fluent and CFX R16*
- *CFD Geometry Preparation using SpaceClaim Direct Modeler*
- *ANSYS Workbench Meshing R16 Update*
- *Fluent R16 New Workspace and User Interface*
- *Post-Processing Improvements in CFD-Post and CFD Viewer*



SpaceClaim Direct Modeler



CFD-Post Volume Rendering

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

Bringing you tomorrow's technology… today!