

ANSYS CFD Workshop

Introduction

Computational fluid dynamics (CFD) is an engineering method for simulating the behaviour of systems, processes and equipment involving flow of gases and liquids, heat and mass transfer, chemical reactions and related physical phenomena.

ANSYS offers the most complete suite of advanced CFD software tools available to help you achieve unprecedented productivity. With unrivaled modeling capabilities and integrated tools for design optimization, ANSYS truly enables Simulation Driven Product Development™.

Objectives

The aim of this workshop is to inform engineers across all domains about the practical uses of CFD, and to showcase the Power and the Ease of Use of ANSYS CFD solutions.

Who Should Attend

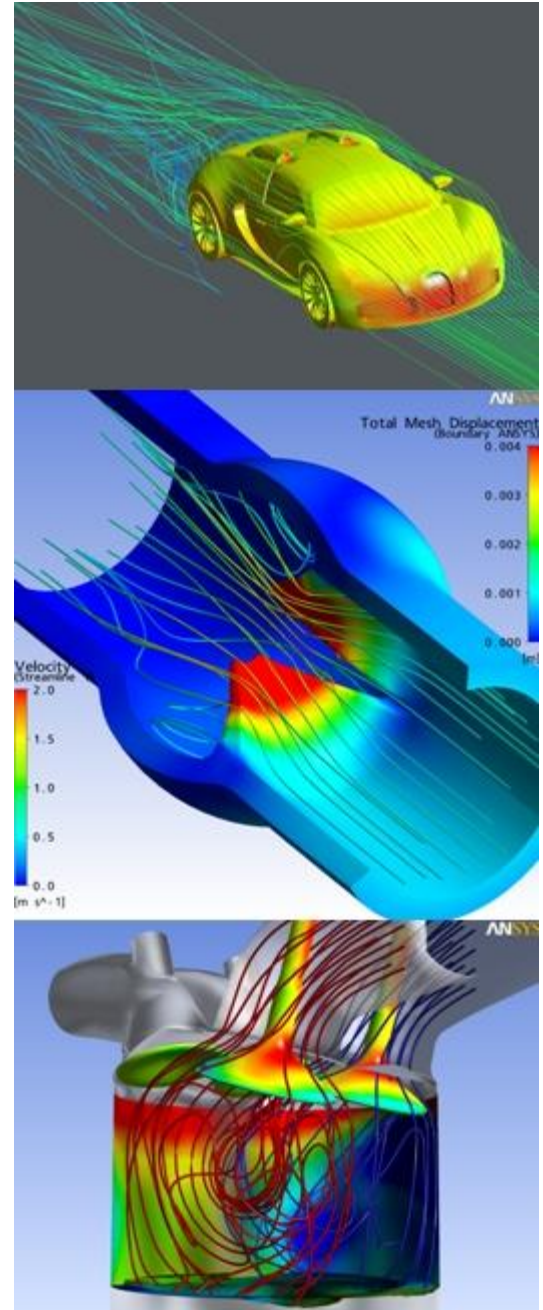
- Engineers who want to find out more about ANSYS CFD solutions
- Engineers who want to integrate CFD into their simulations
- Technical/ Engineering Managers overseeing CFD projects

Agenda of Workshop

- Problem setup in ANSYS Workbench environment
- Performing CFD analyses
- Results checking and post-processing
- Parametric studies

Benefits of using ANSYS CFD

- Directly couples with your CAD software
- Automatically extracts and meshes fluid volumes
- Accurately simulates behaviour of a broad range of flows
- Effortlessly controls the execution, results data and file management for each set of design points
- Seamlessly couples with ANSYS structural mechanics and electromagnetics simulation tools to gain insight into entire system



What Others Had to Say About ANSYS CFD

The accuracy and speed of the CFD simulations provided by ANSYS FLUENT makes it possible to exceed our clients' expectations.

-Shyam Dayanandan, Computational Engineering Solutions LLP

ANSYS CFD software allowed us to input more realistic data such as rotational speed of the wheels as well as pressure of the gas canister, leading to a more realistic simulation of our car performance for our students to understand the pressure and velocity profiles of the car.

-Kenneth Tan, Henderson Secondary School

By using ANSYS FLUENT and performing a detailed analysis of the simulation results, Kranji Solutions have provided the client with more confidence and understanding of their proposed cyclonic vessel design.

-Michel van Vorselen, Kranji Solutions Pte Ltd

For the analysis of flow behaviour of a cyclone-filter device, ANSYS FLUENT is used to calculate the velocity field, pressure field and pressure drop, as well as to visualize the particle trajectory. ANSYS FLUENT enabled us to reduce calculation time, adjust the design with flexibility, and verify our experimentation results.

-Perapong Tekasakul, Prince of Songkla University

Workshop Information

Date	: 8 Mar 2013 (Friday)
Time	: 2 pm - 5 pm
Venue	: 159 Sin Ming Road, #03-05 Amtech Building, Singapore 575625 (view map)

- This workshop is **free of charge**.
- This workshop is meant as a general overview; participants who wish to be proficient in ANSYS FLUENT and/or ANSYS CFX should attend the relevant training course(s). Please email training@cadit.com.sg for enquiries.
- CAD-IT Consultants reserves the right to cancel or postpone any event due to unforeseen circumstances.
- To register, please click [here](#).

For enquiries, please call 6508-7575 or email to caditevents@cadit.com.sg.

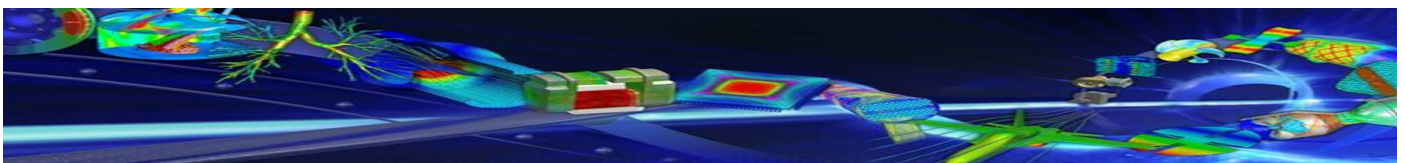
About CAD-IT Consultants

Formed in 1991, CAD-IT Consultants (Asia) Pte Ltd is a leading **ISO 9001:2008** certified global PLM and education provider providing world-class solutions that enable companies to achieve greater product innovation, quality and productivity with reduced lead time and costs. Our priority is in building mutually successful and long term partnerships with our customers and partners.

CAD-IT has received 70 International Awards for marketing/ service excellence. Among its most recent accolades are the Asia Pacific Entrepreneur Award (APEA) - Outstanding Entrepreneur Award 2010 and 2012, the Singapore Prestige Brand Award (SPBA) - Established Brand Award 2009-2012 and Regional Brand Award 2012, and KPMG's Enterprise 50 Awards 2012.

CAD-IT Consultants' suite of PLM solutions include ANSYS (Multiphysics FEA/CFD), Ensign (Extreme Visualization), Goldfire Innovator (Innovation Process Management), Cortona3D (3D Technical Documentation), SpaceClaim (3D Direct Modeler), Mentor Graphics Valor DFM & Valor MSS (Manufacturing Execution Suite), Stampack (Sheet Metal Forming CAE), Rheoware (Blow-moulding and thermo-forming), DEFORM (Bulk Metal Forming CAE) and Autodesk Moldflow (Plastics CAE).

Since February 2010, CAD-IT is honored to partner the Singapore Workforce Development Agency (WDA) to offer Advanced Certificate courses in PLM and Engineering Simulation, under the Workforce Skills Qualifications (WSQ) framework, with the charter to upgrade the technological capabilities of organizations in Singapore so as to enable them to undertake higher value-added design, engineering, manufacturing and R&D activities. From July 2010 to December 2011, CAD-IT has trained over 200 engineering professionals in WSQ courses.



CAD-IT Consultants (Asia) Pte Ltd

159 Sin Ming Road #03-05 Amtech Building Singapore 575625

Tel: (65) 6508 7575

Fax: (65) 6454 3766

Email: training@cadit.com.sg

<http://www.cadit.com.sg>

Bringing you tomorrow's technology... today!