ANSYS Fluid Dynamics Seminar

featuring Hermant Punekar

(Senior Technology Specialist, ANSYS Inc.)

About ANSYS Fluid Dynamics

With the industry's deepest and broadest Fluid Dynamics capabilities, ANSYS has brought together two of the most respected names in CFD simulation – Fluent and CFX – to expertly address the most in-depth CFD simulation demands ever needed.

Backed up by reliable technology that has been verified by academic and independent researchers, technology partners, and a multitude of customers both large and small, ANSYS CFD solutions are the tools of choice for innovative product development organisations around the world, in virtually every industry imaginable.

About the Seminar

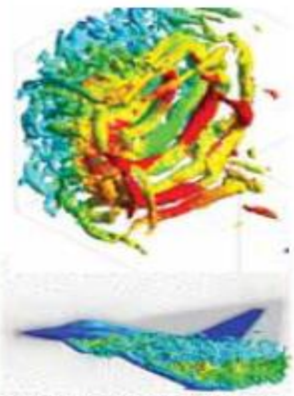
This half-day seminar is highly specialised, focusing on analysis of industrial applications like combustion and multiphase flows amongst other topics.

Who Should Attend

This course is designed for experienced users of ANSYS Fluent or CFX, R&D personnel and industrial practitioners from Marine, Oil and Gas, Defense, Aerospace and HVAC.

Pre-requisites

Participants should ideally have working or application knowledge of ANSYS Fluent and/or CFX.

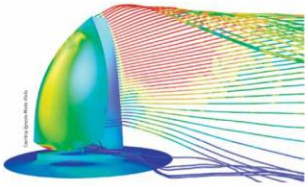


Courtesy EADS Germany GmbH Military Air Systems and the DESIDER Project.

About the Expert

With more than eight years of experience with CFD applications and half a decade of mentoring CFD engineers, Mr. Hermant Punekar is highly respected in ANSYS Inc. circles as the analysis expert in industrial processes involving multiphase flows, phase change processes and gravity separation.

Also competent in heat exchanger modeling and fire simulation, Mr Punekar was part of Fluent 12's core development team in boiling simulation technology, and contributed to the user-defined function (UDF) and development of dual cell heat exchanger models in Fluent.



CAD-IT Consultants (Asia) Pte Ltd

www.cadit.com.sg

Bringing you tomorrow's technology… today!

Topics covered

- Multiphase Flow
- Combustion modeling for furnace/burners/reactors
- Phase change processes
- Gravity Separation
- Heat exchanger modeling
- Tips & Tricks

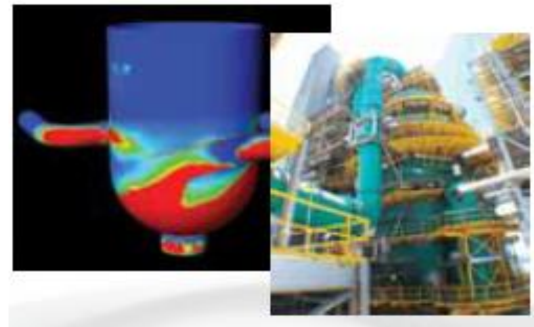
Testimonials

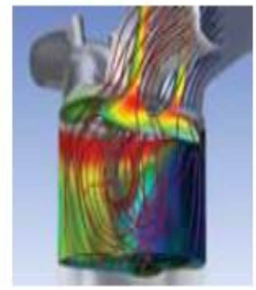
"We use ANSYS CFD because we need to speed up our development process for new products by speeding up all phases of design. With simulation we can investigate inside our products virtually, not physically, and look at detail that would be impossible to evaluate otherwise. We can improve the efficiency of our products by investigating small changes in parameters and spend less time than we would for creating a real prototype and testing."

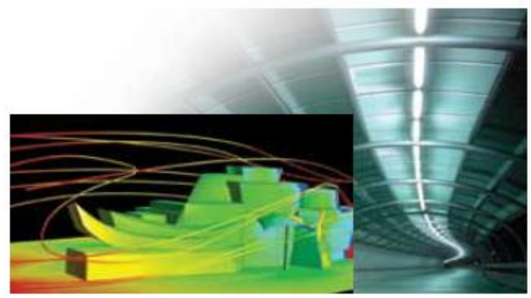
-***Matteo Cipelli, Advanced Engineering COE Manager, Lowara Srl***

"Without ANSYS CFD, we would go into blind designing new reactors or making modifications to existing equipment. The confidence level that we get from using simulation is very high: we generally understand the preferred direction to pursue, rather than going in all possible directions."

-***Matteo Fumagalli, Innovation Engineer, MEMC***





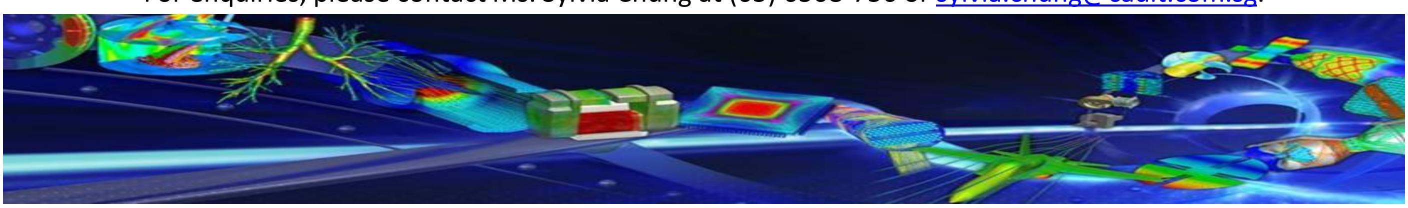


Seminar Information

Seminar Title	**ANSYS Fluid Dynamics seminar featuring Mr Hermant Punekar (ANSYS Inc.)**
Date	**2nd October 2012 (Tuesday)**
Time	**2pm to 6pm**
Venue	**: CAD-IT Consultants (Asia) Pte Ltd Training Center 159 Sin Ming Road, #03-05 Amtech Building, Singapore 575625**

As seats are limited, REGISTER NOW at
http://registration.cadit.com.sg/CADITEvents_Main/Registration.aspx?Event=acc1db58-4a6c-45c7-af62-b6519da2c951

For enquiries, please contact Ms. Sylvia Chung at (65) 6508-756 or sylvia.chung@cadit.com.sg.



CAD-IT Consultants (Asia) Pte Ltd

www.cadit.com.sg

Bringing you tomorrow's technology… today!