Computational Fluid Dynamics(CFD)

Computational Fluid Dynamics(CFD) technique and ANSYS Workbench software are used in almost all of the industry which involves in research and development of Automobiles, Aerospace Applications etc.

The Sessions are handled by **JESHWANTH RAVULA,** *Application Engineer from* ***Innovent Engineering Solutions Pvt Ltd (Authorized Distributor for ANSYS)***.

Date	**5th & 6th of April, 2017**
Time	**9.00 AM to 6.00 PM**
Duration	**Two days with 8 hours per day of Hands on and lecture sessions**
Venue	**Department of Energy and Environment (CEESAT)**

Course Content: Please find the detailed content in Page 3 & 4.

Pre requisites:

The students should be from Mechanical or Chemical background with basics of Fluid Mechanics and Heat Transfer.

The students should bring their laptop with the software installed in it. ANSYS Workbench 16.0 setup file can be downloaded from the link given below.

https://drive.google.com/file/d/0B6tFwKRamkQAVnd2M3d2UWhEcEk/view?usp=sharing

Feel free to contact the coordinators if you have any problem in installing.

Students who do not have laptop please get in touch with the coordinators.

1.Kirubaharan - +91-9443786935

2.Mahesh - +91-9497679458

Registration fee:

Rs.1000/- per head

Only limited registrations are available.

What do the students get?

- The student will get a rich hands-on experience with ANSYS Workbench and get exposed to CFD methodology.

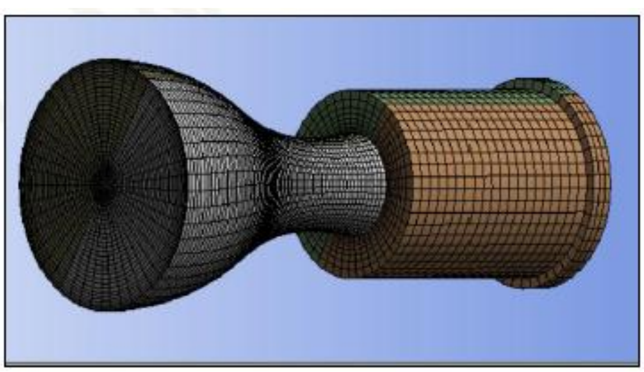
All the students attending the workshop will receive a Certificate from NIT, Tiruchirappalli.

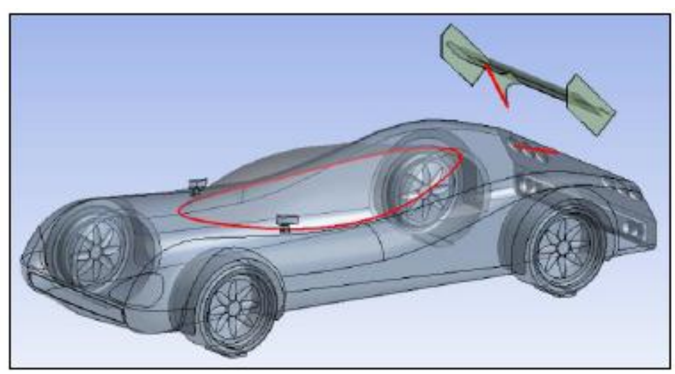
NOTE: Food and Snacks will be provided for the students attending the workshop.

COURSE CONTENT

Day 1:

TOPIC	DURATION	LEARNING OUTCOME
Introduction to ANSYS Workbench and Design Modeler	Pre-Lunch	Overview of ANSYS Workbench, How to launch Workbench? Its Capabilities and file structure Workbench and DM interface details, Type of CAD modeling and GUI navigation
Modelling Sketch and 3-D geometry		Concept of Plane & Sketch, Sketching interface & toolbox, How to create Planes and Sketches How to Draw, Modify, Dimension and Constrain sketches How to create and modify 2D and 3D geometry
Geometry clean up and Advanced 3-D functions		Body Types & States (Frozen & Active) in DM, Boolean operation, Use of Multi-body parts and Share topology, How to Import generic CAD formats (native & Neutral) Clean up process for corrupted and disconnected geometry, How to simplify geometry for CAE analysis How to Decompose geometry into mesh able sections
Lunch Break		
Introduction to ANSYS Meshing	Post-Lunch	What is the ANSYS Meshing? How to launch ANSYS Meshing? Overview on Meshing methods & Mesh controls ANSYS Meshing graphics user interface
Meshing methods And control		Algorithms for Tetrahedral Meshing Difference between Patch dependent and Patch conformal Different methods for Hex Meshing(Sweep and Multizone) How to create Structured mesh Various local & Global mesh settings (i.e. mesh sizing, Refinement, Inflation, etc.)
Workshops on Global Mesh Control		Hand on Exercise to Generate mesh Use of Advanced Size Functions Using Curvature, Proximity, Global Inflation How to check Mesh statistics and quality using various method





Day 2:

TOPIC	DURATION	LEARNING OUTCOME
Introduction to ANSYS CFD and FLUENT		The basics of what CFD is and how it works How to go from the original planning stage to analyzing the end results The different steps involved in a successful CFD project
Materials properties & Boundary conditions	Pre-Lunch	How to define material properties How to define temperature dependent properties The different boundary condition types in FLUENT and how to use them How to define cell zone conditions including solid zones, porous media and rotating frame How to specify well-posed boundary conditions
Heat Transfer		How to treat conduction, convection and radiation How to set wall thermal boundary conditions
		Lunch Break
Workshop on - Mixing tee		Hand on Exercise for Thermal analysis using Fluent, The aim is to learn different steps involved (i.e. reading mesh, defining boundary conditions, selecting material properties, setting up solution monitors, running the simulation and analyzing result) in a successful CFD project
Solver Basics and settings	Post-Lunch	How to choose the solver and the discretization schemes How to initialize the solution How to monitor and judge solution convergence and accuracy
Post processing And Project discussion		Performing flow field visualization and quantitative data analysis How to do this in FLUENT and in CFD-Post How to create Iso-surfaces, Vector plots, Contour plots, Streamlines, XY plotting, Animation creation and more