A SHORT TERM COURSE on "ANSYS Design Modeler and ANSYS FLUENT"

Offered by EntupleTechnologies in collaboration with Department of ME, ABESIT, Ghaziabad

Duration: 5 Days, 15th - 19th January, 2018

About Entuple:

Entuple is a next generation solutions enabler in cutting edge technologies. Entuple delivers world class simulation solutions in Applied Electromagnetics, Semiconductor (VLSI), System Design & Reliability, Mechanical, CFD and RF. Our product solutions include PCB Prototyping, Planer Antenna prototyping systems. Entuple has developed its own range of semiconductor based power drives and process control solutions. We cater to wide range of customers in semiconductor, manufacturing, defense & aerospace and academia.

Entuple has set up center of excellence in the northern region at ABESIT, Ghaziabad to train the faculty & industry professionals in this cutting edge technology.

Scope:

The program provides an immersive learning experience on the concepts of computational fluid dynamics using industry standard simulation tool. It adds the insights of ANSYS Workbench, design modeler, geometric modeling, meshing, ANSYS FLUENT, heat transfer and transient. The program offers lucid examples, case studies and engaging hands-on sessions under the supervision of practicing industry resource.

Target Audience:

The program is designed for

- Under Graduate(UG) / Post Graduate(PG) students interested to acquire skill in Computational Fluid Dynamics and Thermal Analysis
- Research Scholars / Faculty pursuing their research in Computational Fluid Dynamics, Heat Transfer and Transient Analysis
- Practicing Engineers working in the field of Computational Fluid Dynamics

Objectives:

After the completion of this program you will be able to

- Explain the features of ANSYS workbench
- Create models and perform analysis using design modeler
- Illustrate geometry modeling, section formation, mesh creation, mesh control and modelling sections
- Perform heat transfer and transient analysis

About ABESIT:

The ABESIT Institute of Technology, Ghaziabad (UP) is a reputed Engineering Institute in NCR which embarked on its mission in the field of technical education from the Academic Session 2007-08. The institute has lush green and sprawling campus situated next to Crossing Republik, NH-24, Ghaziabad. Institute's philosophy has been to invest all the resources towards students' cause and development. Institute's teaching staff is qualified, painstaking and competently meeting out students' curriculum requirements This is reflected in the institutes result. ABESIT has Human resources with corporate background as a part of faculty which ensures the right blend of theory and practical applicability of the course concept.

About the Department:

Mechanical engineering is a diverse subject that derives its breadth from the need to design and manufacture everything from small parts and devices to large systems. The role of a mechanical engineer is to take a product from an idea to the marketplace. The B.Tech program at – ABESIT, Ghaziabad in Mechanical Engineering commenced in year 2011. Since then three batches have graduated and its alumni are perusing successful careers in the industry and doing higher studies.

To meet ever changing requirements, the Mechanical Engineering department has continuously evolved to incorporate advancements in technology. The department has well-equipped laboratories. Through the Centers of Excellence students are exposed to the new technologies. To bridge the gap between industry and academia, department has been organizing expert lectures from the industry, conducting seminars & workshops and taking students on industrial visits.

Testimonials of Training Program from Entuple:

Satisfied on many varied technical concerns.
– Shaik Nasser, Associate Professor, AMC College of Engineering, Bangalore

I got guidance in true manner. So that I have a confidence to work on myself.
– Shital Parekh, Head Design (R&D), Rotex Automation Ltd, Gujarat

Course Outline and Structure:

Day 1: ANSYS Design Modeler + Meshing

1. Introduction to ANSYS
2. Introduction to ANSYS Workbench
   - Workbench review
   - Basic workflow
   - Data sharing between different solvers
3. Introduction to Design Modeler
   - Preprocessing workflow
   - Planes/Sketches/different toolbox
   - WS 1 - Workshop – Design Modeler Basics
   - WS 2 - Workshop – Sketching
4. Geometric Modelling
   - Design Modeler Concepts
   - How to modify/transform a geometry?
   - Part management
   - Concepts in DM from meshing perspective
5. Geometry clean up and repair
   - Typical geometry issues
   - Analysis tools / Dedicated repair tools
   - Automated cleanup
   - Watertight body preparation & conformal meshing
   - Surface extensions
   - WS 3- Workshop – Geometry cleanup & Fluid body extraction
   - Semi-automatic/manual Repair & defeaturing tools

Day 2:

6. Geometry modelling for CFD
   - Fluid body extraction – internal & external
   - Symmetry modelling
7. Introduction to meshing
   - What is the ANSYS Meshing?
   - Meshing fundamentals
   - ANSYS meshing interface
   - Meshing methods
   - WS 4 - Workshop – ANSYS Meshing Basics
8. Meshing Methods
   - Meshing methods for part / body meshing
   - Assembly meshing
   - Methods & Algorithms for Tetrahedral Meshing / Hex Meshing / 2D Meshing
   - Meshing multiple bodies - selective meshing / recording meshing order

Day 3 : ANSYS Meshing + ANSYS FLUENT

9. Global Mesh Controls
   - Introduction to Global mesh controls
   - General sizing controls & Advanced size functions
   - Global inflation
   - Assembly meshing controls
   - Statistics

10. Local Mesh Controls
    - Local mesh controls (Mesh sizing, refinement, match control, inflation etc.)
    - How to apply local controls?
    - Effects of local controls on mesh
    - WS 5 - Workshop - Mesh Methods
    - WS 6 - Workshop -Using Local mesh controls

ANSYS FLUENT

1. Introduction to CFD Methodology
   - Basics of CFD
   - The different steps involved in a successful project
   - WS 1 -Workshop – Mixing Tee
2. Cell zone and boundary conditions
   - How to define material properties?
   - The different boundary condition types in Fluent and how to use them
   - How to define the mesh interfaces?
   - How to define cell zone conditions in Fluent including solid zones and porous media
   - WS 2 -Workshop - Discrete Phase model Review

Day 4&5:

3. Post-Processing
   - How to perform flow field visualization and quantitative data analysis on your CFD results
   - How to do this in Fluent and in CFD-post
   - WS 3 -Workshop – Post-processing
4. Solver setting
   - How to specify the solver and set the discretization schemes?
   - How to initialize the solution?
   - How to monitor and judge solution convergence?
   - WS 4 -Workshop- Airfoil
5. Heat transfer
   - How to treat conduction, convection (forced and natural) and radiation in Fluent?
   - How to set wall thermal boundary conditions?
   - How to export solution data for use in a thermal stress analysis (one-way FSI)?
   - WS 6 -Workshop -Electronic cooling
6. Transient
   - How to set up and run transient calculations in Fluent?
   - How to choose the appropriate time step size for your calculation?
   - How to post-process transient data and make animations?
   - WS 7- Workshop: Tank flush or vortex shedding Review
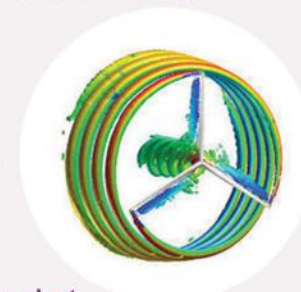
Contact details:

Entuple Technologies
Mr. Aravinth N, Developer,
Mobile No: +91 9513940119

Ms. Sona V, Manager,
Mobile No: +91 7022600846
Entuple Technologies Pvt. Ltd.,
#2730, Trikannika, 80 Feet Road,
HAL 3rd Stage, Indiranagar,
Bangalore - 560038

ABESIT
Prof.S.B.Bajpayee, Head - CEI,
Mobile No: +91 8700173032

Mr.Avinash Trivedi, Head of Dept. - (ME),
Mobile No: +91 9717588974
ABES Institute of Technology,
Campus-2, 19th Km Stone,
NH-24, Vijay Nagar,
Ghaziabad-201009 (U.P.)

Registration Fee:
Practicing Engineers: INR 25,000.00
Teaching Faculty / Research Scholars / UG / PG Students : INR 7,500.00

Registration: Online registration can be done at Entuple website: http://events.entuple.com

Payment Details:
- Online Transaction: Name: ABES Institute of Technology, Account No: 0674009300226366, Bank: Punjab National Bank.
  IFSC Code: PUNB0370300, RTGS Code: PUNB0370300

Venue: ABES Institute of Technology, Campus-2,19th Km. Stone, NH-24, Vijay Nagar, Ghaziabad-201009 (U.P.)