

# **Computational Fluid Dynamics using ANSYS**

## **Description :**

Computational Fluid Dynamics(CFD) is a tool with amazing flexibility ,accuracy and breadth of application. The application of computational fluid dynamics (CFD) in the agricultural industry is becoming ever more important. Over the years, the versatility, accuracy and user-friendliness offered by CFD has led to its increased take-up by the agricultural engineering community. Now CFD is regularly employed to solve environmental problems of greenhouses and animal production facilities. However, due to a combination of increased computer efficacy and advanced numerical techniques, the realism of these simulations has only been enhanced in recent years. ANSYS provides a comprehensive suite of computational fluid dynamics software for modeling fluid flow and other related physical phenomena. It offers unparalleled fluid flow analysis capabilities, providing all the tools needed to design and optimize new fluids equipment and to troubleshoot existing installations.

## **DAY 1**

1. Overview of the CFD
2. Structure of the CFD tools
3. Components of CFD software
4. Geometry
5. Physics

## **DAY 2**

1. Meshing
2. Reports
3. Post processing
4. Solve fluid simulation challenges
5. Hands on experience with Ansys-fluent
6. Case study

**FOR MORE DETAILS, BE FREE TO CONTACT US!!**