

Introduction to ANSYS CFX

The four day course “Introduction to ANSYS CFX” covers the basics of geometry creation, unstructured meshing (tetrahedral and prism elements) and problem set-up, solution and analysis. A mixture of lectures and workshop sessions gives attendees the opportunity for hands-on practice.

Day 1: Creation of Geometry using DesignModeler

We start up and use DesignModeler for our CFD applications. The course includes import of existing CAD, and creation/modification of geometry.

Day 2 - morning: Meshing using ANSYS Meshing Application

An introduction to the Meshing Application is given on the second day. The focus is put on generation of unstructured meshes using the Advancing Front method of CFX-Mesh and swept meshes.

Day 2 - afternoon & Day 3: Solver and Post-processing

We complete the CFD process as we continue with the model set-up in CFX-Pre, running the solver and analysis of results in CFX-Post.

Day 4: Meshing with ANSYS ICEM CFD (CFX) Tetra/Prism

On the last day we will focus on generation of unstructured meshes using the OCTREE and other meshing methods of ICEM CFD (CFX).