

Introduction to CFD pre- & post-processing with ANSA/META

webinar sessions

Session 1:

- Introduction to ANSA CFD layout and basics
- ANSA Topo menu and Draw Modes (ENT, PID etc.)
- How to generate geometry parts (surfaces, points etc.)
- Surface manipulation and modifications
- Preparation of a watertight geometry
- Treatment of problematic areas
- How to check geometry for errors with Checks Manager
- How to properly define PIDs

Session 2:

- Manual meshing techniques (Spacing, CFD mesh, Improve)
- Detect and manage geometry features with the Feature Manager (sharp edges, trailing/leading edges etc.)
- Using the Quality Criteria tab
- Batch Meshing setup
- Size Boxes and Size Field for local mesh refinement
- Checking the surface mesh with Checks Manager and quality improvement
- The Octree/Wrap function

Session 3:

- Layers generation (important options and settings)
- Volume Mesh algorithms (Tetra, HexaInterior etc.)
- Generate Layers and Volume Mesh with Batch Mesh manager
- Checking and fixing volume mesh quality



Session 4:

- The use of DECKS menu for the setup of a case
- Proper PID definition in order to setup correctly the case for a CFD run.
- Demonstration of the output for a test case

- Basic use of META - How to load a model and results.
- Labels and Fringe options.
- The use of User calculations.
- The use of Cut Planes and CFD Post toolbar.
- Generation of streamlines and isofunctions.

Supporting material:

User Guide

- [CFD Brief User Guide](#)

Tutorials

- [The Basics](#)
- [Meshing for External Aerodynamics](#)
- [Batch meshing of a pump](#)

Videos

- [CFD geometry preparation of a Formula-type car](#)
- [Advanced CFD meshing of a Formula-type car and OpenFOAM simulation](#)
- [Webinar on META for CFD](#)