

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

**Length:** 16 hours

**Dates:**

Monday May 5<sup>th</sup> : 14:30 – 18:30 (EEST)  
Tuesday May 6<sup>th</sup> : 14:30 – 18:30 (EEST)  
Wednesday May 7<sup>th</sup> : 14:30 – 18:30 (EEST)  
Thursday May 8<sup>th</sup> : 14:30 – 18:30 (EEST)

**Course Description:**

This course provides a practical introduction to Computational Fluid Dynamics (CFD) pre- and post-processing using ANSA and META

**Learning Objectives:**

Participants will learn essential workflows for geometry cleanup, meshing, and model setup in ANSA, as well as data visualization and result analysis in META. Through this, attendees will develop the skills needed to efficiently prepare CFD models and interpret simulation results. Ideal for participants looking to streamline their CFD processes using industry-standard tools.

**Software used:**

- ANSA
- META

**Audience:** Students

**Prerequisites:** An understanding of the basics of fluid mechanics and FEA



### Modules in this course:

#### Day 1

- Introduction to ANSA CFD layout and basics
- ANSA Topo menu and Draw Modes
- How to generate geometry parts
- 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

#### Day 2

- Manual meshing techniques
- Detect and manage geometry features with the Feature Manager
- 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

#### Day 3

- Layers generation
- Volume Mesh algorithms
- Generate Layers and Volume Mesh with Batch Mesh manager
- Checking and fixing volume mesh quality

#### Day 4

- The use of DECKS menu for the setup of a case
- Proper PID definition 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.



## 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.

You can find useful video tutorials on our [YouTube](#) channel at the following links:



- [ANSA tutorials](#)
- [META tutorials](#)