OpenFOAM training: Detailed course program

Basic training program (3-4 June 2024)

- **OpenFOAM overview (prof. Tommaso Lucchini)**: directory organization, code structure. Solvers and modules. Utilities. Case organization: initial, boundary conditions; physical models; discretization and simulation control parameters.
- **Mesh generation (prof. Augusto Della Torre)**: Usage of blockMesh with basic and advanced options to achieve suitably shaped block structured meshes with complex geometries. Fundamentals of automatic meshing techniques for complex geometries. Application of snappyHexMesh to an external aerodynamics simulation;
- **Incompressible flow simulations (prof. Tommaso Lucchini)**: presentation of 2D and 3D Ercoftac diffuser cases. 2D case: case setup, selection of turbulence models, discretization schemes and mesh resolution, validation with experimental data. 3D case: mesh and validation, challenges of turbulence modeling with RANS.
- **Post-processing (prof. Gianluca Montenegro)**: Basics of mesh and flow field visualization: streamlines, contours, thresholds and basic filtering for better visualization. Time and space sampling of fields, visualization of scalar and vector fields, tips and tricks for advanced functions of post processing.

Advanced Training Program (5-6 June 2024)

- **Advanced Mesh generation with snappyHexMesh (prof. Paolo Schito)**: preparation of geometry, multi-regions, baffles, boundary layers.
- **Conjugated Heat Transfer and External Aerodynamics (prof. Augusto Della Torre)**: definition of the multi-region framework, multi-region mesh-generation, material properties definition and coupled boundary conditions for CHT simulation. Advanced region coupling with source terms at the interface. Setup of an external aerodynamic simulation: choice of turbulence model, best practice for boundary condition and numerical setup.
- **Modeling flows in porous media and moving components (prof Gianluca Montenegro)**: Definition of regions with specific properties (volume forces), case setup and definition of flow resistance and motion properties. Description of the set of equations and validity of the Moving Reference Frame (MRF)and its application to the simulation of an impeller.
- **Advanced post-processing (prof. Paolo Schito)**: simulation validation, automatic and run-time post/processing, data saving.
- **HPC Environments for OpenFOAM (prof. Paolo Schito)**: workflow on HPC environments, use of multiple processes, scalability, large case data management.