Numerical Fluid Mechanics CFD

1. **Course number and name:** 020MFNES5 Numerical Fluid Mechanics CFD

2. **Credits and contact hours:** 4 ECTS credits, 2x1:15 contact hours per week

3. **Name(s) of instructor(s) or course coordinator(s):** TBD (To Be Determined)

4. **Instructional Materials:** PowerPoint slides

   Textbooks/References:
   - Fluid Dynamics; Theory, Computation, and Numerical Simulation, C. Pozrikidis, 3rd edition, Springer. ISBN 978-1-4899-7990-2.
   - CFD Techniques and Thermo-Mechanics Applications, Zeid Driss, Brahim Necib, and Hao-Chun Zhang, Springer. ISBN 978-3-319-70944-4.

5. **Specific course information**
   a. **Catalog description:**
   Computational fluid dynamics (CFD) is a technology based on a fast and reliable calculation methodology for solving complex fluid flow and heat transfer problems. This course introduces the fundamentals and practical technical applications of CFD. Although it provides an overview of some of the fundamental mathematical equations governing fluid flow and heat transfer phenomena, it emphasizes the application of the knowledge gained in the practical use of commercial CFD codes. The course provides a detailed explanation of setting up, running and interpreting CFD model results for different ANSYS Fluent® case studies.

   b. **Prerequisite:** Fluid Mechanics (020MEFES1) or Fluid Mechanics 1 (020MF1ES1).

   c. **Selected Elective** for ME students.

6. **Educational objectives for the course**
   a. **Specific outcomes of instruction:**

   - To introduce and develop the main approaches and techniques which constitute the basis of numerical fluid mechanics.
   - To familiarize students with the numerical implementation of these techniques and numerical schemes, so as to provide them with the means to write their own codes and software.
   - To cover a range of modern approaches for numerical and computational fluid dynamics.
   - Use and run fluid dynamics simulations using ANSYS Fluent® with various case studies.

b. PI addressed by the course:

PI	1.1	1.2	1.3	7.1	7.2
Covered	x	X	x	x	x
Assessed					

7. Brief list of topics to be covered

- **Kinematics Overview**: Fluids and solids – Fluid parcels and flow kinematics – Coordinates, velocity, and acceleration – Fluid velocity – Point particles and their trajectories – Material surfaces and elementary motion – Numerical interpolation – Fluid parcel motion, expansion, deformation, rotation, and vortices – Numerical differentiation – Incompressible fluids and steam function.
- **Flow computation based on kinematics**: Flow classification – Irrotational flow and the velocity potential – Finite difference methods – Linear solvers – Two-dimensional point sources and point-source dipoles – Point vortices and line vortices.
- **Equation of motion and vorticity transport**: Newton's second law of motion for a fluid parcel – Integral momentum balance – Cauchy's equation of motion – Euler and Bernoulli equations – The Navier-Stokes equation – Vorticity transport – Dynamic similitude and the Reynolds number – Structure of a flow as a function of the Reynolds number.
- **Low and high Reynolds-number flows:** Flow in a narrow channel – Film flow on a horizontal or inclined wall – Multi-film flow on a horizontal or inclined wall – Changes in the structure of a flow with increasing Reynolds number – Prandtl boundary-layer analysis – Blasius boundary layer on a semi-infinite plate – Finite difference solutions.
- **Vortex motion:** Vorticity and circulation in two-dimensional flow – Point vortices – Two-dimensional flow with distributed vorticity – Axisymmetric flow induced by vorticity.
- **ANSYS Fluent® case studies/Examples (Focus on 2 dimensional case studies, 3 dimensional problems can be transformed into 2-dimensional subjects upon the discretion of the instructor)**:
    - Air flow CFD Modeling in an industrial convection oven.
    - CFD Application for the study of innovative working fluids in solar central receivers.
    - Computational fluid dynamics for thermal evaluation of earth-to-air exchanger for different climates.
    - CFD modeling of a parabolic trough receiver of different cross section shapes.
    - Numerical simulation and experimental validation of the role of Delta wing privileged apex.
    - Numerical simulation of the overlap effect on the turbulent flow around a savonius wind rotor.