Budapest University of Technology and Economics
Faculty of Transportation Engineering and Vehicle Enginee

1. Subject name	**Advanced CFD in Vehicle Industry**				
2. Subject name in Hungarian	Járműipari áramlásmodellezés				
3. Code	**BMEKORHD005**	**4. Evaluation type**	**exam grade**	**5. Credits**	**4**
6. Weekly contact hours	**2 (0) Lecture**	**0 (0) Practice**	**2 (0) Lab**		
7. Curriculum	**PhD Programme**	**8. Role**	**Basic course**		
9. Working hours for fulfilling the requirements of the subject					**56**
Contact hours	56	**Preparation for seminars**	0	**Homework**	0
Reading written materials	0	**Midterm preparation**	0	**Exam preparation**	0
10. Department	**Department of Aeronautics and Naval Architectures**				
11. Responsible lecturer	Dr. Veress Árpád				
12. Lecturers	Dr. Veress Árpád				
13. Prerequisites					

14. Description of lectures

Specific areas of the application of numerical methods in the vehicle engineering: Fluid dynamics in the vehicle industry, Supersonic internal and external flows, Secondary flows in turbo machinery and coupled flow and thermal processes, Rotors and propellers, Particle tracking, Free surface flows, Combustion in gas turbine combustor, Flow and thermal processes of PCBs, Flow in porous media

-----

The material requires the knowledge of the next topics: Introduction to CFD (Computational Fluid Dynamics) via scientific and industrial applications, Approaches for flow modelling and conditions for applications, Flow modelling by means of continuum mechanics, System of Navier-Stokes equations, The subject of the CFD; actuality, advantages and application areas, Turbulence and simulation techniques for handling turbulence (DNS, LES and RANS), Reynolds and Favre averaged system of Navier-Stokes equations, Reynolds stress and Eddy viscosity models, Turbulence modelling, Modelling approaches close to the wall; logarithmic-based Wall function and Near-wall resolving approach, Placement of the first cell at the wall, Turbulence modelling, Introduction to discretization techniques (Finite Difference, Finite Element and Finite Volume Methods), Finite volume method for solving governing equations, The main steps of a CFD simulation tasks; geometry model preparation and simplification, meshing and mesh metrics, definition of material properties, setting of considered physics, initial and boundary conditions and their definitions, solver settings, convergence characteristics, visualization and presentation of the results in qualitative and in quantitative manner.

15. Description of practices

16. Description of labortory practices

Completing tutor-guided simulation tasks in ANSYS CFX environment: Flow modelling in nozzle of rocket engine, CFD analysis of aircraft wing profile, Numerical simulation of centrifugal compressor, Flow modelling in axial turbine, CFD analysis of X33 re-entry vehicle, Flow modelling of rotors and propellers, Numerical modelling of particle tracking, CFD analysis of free surface flows, Flow modelling in combustion chamber of gas turbine, Coupled CFD and thermal analysis of PCBs for thermal management, Flow in porous media.

17. Learning outcomes

A. Knowledge
- The student knows the advantages, conditions, application ranges and the theoretical and practical aspects of the specific CFD (Computational Fluid Dynamics) methodologies for solving industrial (R&D) problems and for having new scientific results.

B. Skills
- The student can solve CFD simulation tasks independently in the specific areas with especial care for the highest level approximation of the reality and/or at the best “computational cost/accuracy” ratio with verification, plausibility check and validation.

- The student can develop and obtain new industrial and scientific results after understanding and analysing CFD results.

C. Attitudes

- The student aims to complete his/her studies at the highest level, under the shortest time, by providing his/her knowledge and capacity at the best to obtain knowledge for deep and independent professional work.
- The student has strong professional commitment, has developed expectations for finding new, better solutions and has agreement on doing hard work.

D. Autonomy and Responsibility

- The student takes responsibility for guiding mates by the quality of his/her work and by keeping ethical norms.
- The student takes responsibility for applying the knowledge in line with the studied conditions, limitations and constraints.
- The student can friendly accept the well-established constructive criticism and can utilize that in future.
- The student is a creative constructor, proactive, and has leadership skills and argument techniques, capabilities with responsibility during the studies, research work.

18. Requirements, way to determine a grade (obtain a signature)

The criterion of the acceptance of the semester and so getting the signature is the completeness of the solution of a defined problem in a specific area in the agreed time and quality. The exam is oral. The final mark of the exam is the mathematical average of the results for the own task and the exam.

19. Opportunity for repeat/retake and delayed completion

20. Learning materials

1. The presentation about the lectures, simulation guide lines and tutorials provided by the professor,
2. John D. Anderson, JR.: Computational Fluid Dynamics, New York, ISBN-10: 0071132104, ISBN-13: 978-0071132107, McGraw-Hill Higher Education; International edition (1995), 3. ANSYS, Inc., ANSYS CFX-Solver Theory Guide, Release 2019 R1, ANSYS, Inc. Southpointe, 2600 ANSYS Derive Canonsburg, PA15317, ansysinfo@ansys.com, http://www.ansys.com, USA, 2019.

Effective date	27 November 2019	**This Subject Datasheet is valid for**	Inactive courses