

Computational Fluid Dynamics

Unit Title	Computational Fluid Dynamics		
Level of Study			
Credit Value		ECTS Value	
Home Department	Department of Theoretical Physics		
Home Faculty	Physics Faculty		
Unit Co-ordinator	Andrew I. Ivantsov		
Key Words	Computational fluid dynamics, the law of mass conservation, the equation of fluid motion for laminar and turbulent flows, the equation of heat transfer, the mixed problem and spatial discretization methods		
Brief Summary	In the framework of the course the basic principles and equations of hydrodynamics will be studied: the law of mass conservation, the equation of fluid motion for laminar and turbulent flows, the equation of heat transfer, etc. The numerical solution of hydrodynamic problems as the mixed problem and spatial discretization methods will be discussed. The course also contains the introduction to ANSYS Fluent, one of the world leader CFD software.		
Indicative Content	Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flows. Predicting and controlling fluid flow is critical in optimizing the efficiency of so many products and processes. In the framework of the course the basic principles and equations of hydrodynamics will be studied: the law of mass conservation, the equation of fluid motion for laminar and turbulent flows, the equation of heat transfer, etc. The numerical solution of hydrodynamic problems as the mixed problem and spatial discretization methods will be discussed. The course also contains the introduction to ANSYS Fluent, one of the world leader CFD software, that gives tools to successfully simulate the behavior of fluid flows — even with complex interactions between multiple physics — and confidently analyze results.		