Advanced Computational Fluid Dynamics

Module Information

2022.01, Approved

Summary Information

Module Code	7310MECH
Formal Module Title	Advanced Computational Fluid Dynamics
Owning School	Engineering
Career	Undergraduate
Credits	10
Academic level	FHEQ Level 7
Grading Schema	50

Teaching Responsibility

LJMU Schools involved in Delivery
Engineering

Learning Methods

Learning Method Type	Hours
Lecture	11
Tutorial	11

Module Offering(s)

Display Name	Location	Start Month	Duration Number Duration Unit
SEP-CTY	CTY	September	12 Weeks

Aims and Outcomes

Aims	The module aims to explore the underlying theory of commercial computational fluid dynamics (CFD) codes and to investigate their performance and reliability in engineering applications. Whilst the theoretical aspects of the method will be covered in lectures the module is intended to be practical in nature with students having the opportunity to practice via a range of tutorials and assignments using IPython Notebook and an industry standard software.

After completing the module the student should be able to:

Learning Outcomes

Code	Number	Description
MLO1	1	Understand and apply the theory underpinning commercial CFD codes.
MLO2	2	Appreciate the structure of CFD codes for potential amendment and development.
MLO3	3	Set up and validate efficient and accurate CFD models.
MLO4	4	Assess the limitations of CFD simulations as part of the design process.
MLO5	5	Critically evaluate the output from CFD analysis.

Module Content

Outline Syllabus	1. Review of the mathematical formulation of fluid dynamics - Continuity equation (Conservation of mass); Navier-Stokes equation (Conservation of momentum); Equation of energy (Conservation of energy).2. Numerical solution methods - Finite differences; Finite volumes; Spectral elements.3. Efficient approaches for the solution of a system of linear algebraic equations - Why is it important in practical CFD simulations? Issues with direct solution approaches; Iterative solution approaches; Relaxation and convergence; Error and residual; Improve performance of iterative solvers. 4. Equation discretisation schemes - Central differencing; Upwind differencing; Higher order schemes; Diffusion problems (e.g. heat conduction problems); Convection-diffusion problems; Numerical diffusions; 5. Pressure correction algorithms - SIMPLE, SIMPLEC, PISO, other variations.6. Chorin / Chorin-type algorithms (for unsteady problems).7. Introduction to Turbulence – physical descriptions - Why do we study turbulence? Characteristics of turbulence? How to study turbulence? Mesh-based/meshless techniques. Law of the wall. 8. Introduction to Turbulence – modelling - Why turbulence modelling? Closure problem in the modelling? Various turbulence models. How to choose/set up the appropriate turbulence models in the commercial CFD packages? Appropriate treatment of the near-wall region and its importance.9. Considerations on accuracy and validity of CFD numerical results - Assess sources of error in the solution process (from mathematical descriptions to numerical solution) of Fluid/heat transfer problems. How do the sources of error influence the results? How can the errors be quantified? Verification and Validation of the numerical results.10. Further considerations on the use of commercial CFD packages - Capabilities, benefits, limitations of commercial CFD packages. Set-up and run CFD problems and post-processing the results. Some ideas to consider to fix the failed CFD models. Some ideas to consider to improve the performance of the functioning CFD models (in terms of accuracy and simulation time).
Module Overview	
Additional Information	This module aims to appraise and distinguish the features of high performance CFD codes and introduces the student to some of the intricacies associated with the modelling of fluid flow using CFD. The module aims to develop in the student a critical approach towards the appraisal of CFD predictions. This module includes content that relates to the following UN Sustainable Development Goals. Computational Fluid Dynamics (CFD) has become an invaluable tool in the design process of any problem involving fluid flow phenomena. Therefore, the following UN Sustainable Development Goals are, in part, considered:SDG3 – CFD is used in the investigation of blood flow in diseased arteries (i.e. atherosclerosis) and air flow through the lungs. These will be considered in the tutorial sessions.SDG6 – CFD is used in the development of systems that provide clean water and sanitation which is considered in a tutorial.SDG7 – CFD is used in the design of systems that provide affordable and clean energy (e.g. wind turbines, tidal stream turbines) which are considered in a tutorial.SDG9 – CFD is used in industry, innovation (novel products) and infrastructure.SDG11 – CFD is used in the design of sustainable cities and communities whether it be considering the wind profile through large buildings or sewer system. These aspects will be considered in the tutorials.SDG12 – CFD is used in the development of systems that allow for responsible consumption and production.SDG13 – CFD is used in the development of systems that provide for clean, renewable energy supplies playing a role in climate action.

Assessments

Assignment Category	Assessment Name	Weight	Exam/Test Length (hours)	Module Learning Outcome Mapping
Test	VLE Test	40	0	MLO1, MLO2
Report	CFD Project	60	0	MLO3, MLO4, MLO5

Module Contacts

Module Leader

Contact Name	Applies to all offerings	Offerings
Mehdi Seddighi	Yes	N/A

Partner Module Team

Contact Name	Applies to all offerings	Offerings