

Summary Information

Module Code	6314MECH
Formal Module Title	Computational Fluid Dynamics
Owning School	Engineering
Career	Undergraduate
Credits	10
Academic level	FHEQ Level 6
Grading Schema	40

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
JAN-CTY	CTY	January	12 Weeks

Aims and Outcomes

Aims	The module will introduce students to Computational Fluid Dynamics (CFD) and will extend their experience, knowledge and skill with the aid of 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	Apply, analyse, verify and validate efficient and accurate CFD models.
MLO3	3	Critically analyse and evaluate the output from CFD analysis.
MLO4	4	Identify and evaluate the limitations of CFD as part of the design process.

Module Content

Outline Syllabus	<p>Theoretical aspects of CFD:- Qualitative revision of real fluid flow and introduction to CFD with industrial examples of usage. - Review of the governing equations of fluid flow including Navier-Stokes equations, energy equations and continuity equations. - Discretisation of the computational domain and numerical solution of the resulting algebraic equations. - Problems with direct matrix inversion and need for iterative solution methods. Basic iterative schemes, Gauss-Seidel and Jacobi, solution behaviour, convergence, stability, under and over relaxation. - Meshing principles and strategies to improve meshing. - Introduction to the modelling of boundary layer flow, law of the wall (wall function) and its use in CFD modelling. - Turbulence - qualitative understanding. Brief introduction to time averaging and need for a turbulence model. - Validation and verification of CFD simulations. Practical aspects of CFD:- Use of a commercial CFD code to solve engineering problems. - Approach to setting up a CFD model. - Identification of the underlying physics applicable for a given problem. - Prediction of expected results based classical theory and rough hand calculations. - Examination of the typical boundary conditions available within a commercial CFD code and the study of their validity. - Selection of the boundary conditions required to capture the expected physical behaviour at the limits of the modelled region. - Economic use of CFD, run time and computing resources required. - Strategies to reduce the size of model required, use of symmetry in 2D and 3D models, transfer of boundary conditions from other models. - Meshing quality, mesh construction and strategies for mesh refinement, mesh independence, adaptive mesh refinement. - Selection of an appropriate computational domain for external flows, refinement and optimisation of computational domain dimensions. - Monitoring the solution process, convergence control and relationships between convergence criteria and accuracy of solution, strategies for economic solution. - Simultaneous solution of mass and heat transfers (conjugate heat transfer) including flow freezing and its effects on solution time. Use and control of solution adaptive mesh refinement techniques and implications for run time and storage requirements. - Presentation and interpretation of CFD results. Extracting performance indicators, point values, surface integrals for loads and mean values of physical parameters. - Strategies for validation of results including checking of conservation laws, mesh and computational domain independence, comparison with existing data/theory.</p>
Module Overview	
Additional Information	<p>This module includes content which 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 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.</p>

Assessments

Assignment Category	Assessment Name	Weight	Exam/Test Length (hours)	Module Learning Outcome Mapping

Centralised Exam	Examination	100	3	MLO1, MLO2, MLO3, MLO4
------------------	-------------	-----	---	---------------------------

Module Contacts

Module Leader

Contact Name	Applies to all offerings	Offerings
Sean Malkeson	Yes	N/A

Partner Module Team

Contact Name	Applies to all offerings	Offerings
--------------	--------------------------	-----------