MMME3086
Computer Modelling Techniques

INTRODUCTION AND BACKGROUND

1. About the Computer Modelling Techniques Module

The MMME3086 - Computer Modelling Techniques module is concerned with the use of computer modelling techniques and numerical methods to simulate and analyse engineering solid and fluid problems.

The course addresses three topics:
    Numerical Methods (NM)
    Finite Element Analysis (FEA)
    Computational Fluid Dynamics (CFD)

An overview of numerical methods used in engineering analysis is presented. Students will also learn the basic theory of FEA and CFD and how to apply them to model engineering problems.

There are three assessed (compulsory) coursework assignments on Numerical methods (using the Matlab software), FEA (using the ABAQUS software) and CFD (using the ANSYS FLUENT software). The coursework assignments are designed to give students the fundamentals of solving numerical problems and running state-of-the-art computer modelling software (Matlab, ABAQUS and ANSYS) to model relatively simple geometries. These software packages are widely used in industry and cover a wide range of advanced complex engineering applications.

Students who are interested in learning more about FEA and/or CFD can choose the optional 20 credit modules in the 4th year covering these topics.

A brief summary of the Computer Modelling Techniques module is presented below.

Module Name	**Computer Modelling Techniques**
Module Code	**MMME3036**
Credits	20
Year/Level	Level 3
Semester	Autumn Semester
Module Convenor	**Dr Mirco Magnini** (mirco.magnini@nottingham.ac.uk)
Teaching Staff	**Dr Mirco Magnini** (NM) - Coates B100a **Dr Chris Bennett** (FEA) - Coates C43; **Dr Luke Parry** - AMB B20 **Dr Donald Giddings** (CFD) - Coates C33
Prerequisites	Completion of the second year of a Mechanical Engineering undergraduate course, or an equivalent course.
Summary of Content	Introduction/Background Part 1: Numerical Methods (NM) Part 2: Finite Element Analysis (FEA) Part 3: Computational Fluid Dynamics (CFD) Coursework and Seminar Sessions Revision
Method of Assessment	Three marked coursework assignments: NM coursework (30%) FEA coursework (35%) CFD coursework (35%)
Educational Aims	To provide students with a basic knowledge and understanding of the mainstream computer modelling techniques used in modern engineering practice, including Finite Element and Computational Fluid Dynamics methods.

Learning Outcomes	On successful completion of this module students will be able to: LO1 – Understand the theoretical background of numerical methods used in engineering analysis, Finite Element and Computational Fluid Dynamics techniques. LO2 – Identify and apply appropriate computer modelling techniques to solve specific engineering problems. LO3 – Demonstrate an understanding of how computer programmes can be used to solve practical solid and fluid mechanics problems in engineering. LO4 – Evaluate the accuracy and sources of error in solving solid and fluid mechanics problems in engineering using computational models. LO5 – Gain hands-on experience of running state of the art FEA and CFD software codes widely used in industry
Coursework	There are three marked coursework assignments: NM Coursework (30%) FEA coursework (35%) CFD coursework (35%) The assignments do not require supervision. Students are given 2 weeks to complete the assignments. Time required to complete the assignment: 6 hours approx. Submission deadline: 2 weeks after issuing the assignment Educational objective: To gain some practical experience of the fundamentals of solving numerical problems and running FEA and CFD engineering software codes that are widely used in industry.

Teaching week	**Section**	**w/b**	**Lecture topic (2h)**	**Seminar session (2h)**	**Coursework dates**
1	NM	02 Oct	Steady diffusion equation	1D steady finite-volume in Matlab	
2	NM	09 Oct	Solution of linear systems, unsteady diffusion equation	Gauss-Seidel method in Matlab	
3	NM	16 Oct	Solution of nonlinear equations, numerical integration	1D unsteady finite-volume in Matlab	**Mon 16 Oct: NM coursework release**
4	FEA	23 Oct	1D FE, Stiffness Matrices	Stiffness Matrix Assembly, and Solution	
5	FEA	30 Oct	Truss/Pin-jointed FE	Abaqus: Geo, Mesh	**Mon 30 Oct: NM coursework deadline**
6	FEA	06 Nov	Continuum Elements, Plates and Shells	Abaqus: BCs, Loading, Solution, Post-processing	**Thu 09 Nov: FEA coursework release**
7	FEA	13 Nov	Practical Notes on FE	FEA coursework support	
8	CFD	20 Nov	Practical introduction to CFD and a commercial code	Model creation and mesh generation.	**Thu 23 Nov: FEA coursework deadline**
9	CFD	27 Nov	Derive the Navier Stokes equations of fluid motion in 3D for an incompressible, steady flow	Setting up the models in CFD solution and achieving converged solution in Fluent	**Thu 30 Nov: CFD coursework release**
10	CFD	04 Dec	Demonstrate how the finite volume numerical method works in a 1D case	Post solution processing of CFD model and CW support	
11	CFD	11 Dec	CFD overview and revision	-	**Thu 14 Dec: CFD coursework deadline**

2. Some Frequently-Asked-Questions about this module

- **I have used FEA and CFD software before – why do I need this module**?
  Using FEA and CFD software as "black boxes" is very risky.
  As engineers, we need to understand the underlying theory of these techniques and their capabilities and limitations.

- **Does the module cover writing computer software (i.e. computer programming)?**
  Yes and no. We will use Matlab to solve exercises in NM; basic concepts of Matlab will be covered. The module will cover the theoretical background of FEA and CFD, and how the software can be applied to engineering problems.

- **How is this module assessed?**
  - NM Coursework assignment 30%
  - FEA Coursework assignment 35%
  - CFD Coursework assignment 35%

- **What is covered in the coursework?**
  Students will be required to show practical evidence of solving numerical problems using Matlab and run two industry-standard software codes; ABAQUS (for FEA) and FLUENT (for CFD) and answer a number of questions.

- **I have never used FEA or CFD software - Will I struggle with the coursework?**
  No. We do not assume prior knowledge of any FEA or CFD software codes. Students will be given worked step-by-step examples to begin with, and then asked to run a different problem.

- **Are there any tutorial/workshop classes?**
  2h seminar each week with hands-on practice: bring your own laptop.

- **Should I buy any textbooks?**
  There is no specific textbook that closely exactly follows the syllabus covered. There is no need to purchase any textbook for this module. All topics will be fully covered during the formal lectures.

3. An Overview of Computational Mechanics Techniques

Most numerical techniques in continuum mechanics are based on the principle that it is possible to derive some mathematical equations and relationships that accurately describe the behaviour of a small part of the body.

By dividing the entire body into a large number of these smaller "parts" or “elements” and using appropriate compatibility, equilibrium and continuity relationships to link up or assemble these elements together, it is possible to obtain a reasonably accurate prediction of the values of variables such as stresses, displacements, temperatures, etc. in the body. As the sizes of these small elements are made smaller, the numerical solution becomes more accurate, but at the cost of increased computation time.

This introduction covers the background of computational mechanics techniques and presents the classifications of computational mechanics techniques.

3.1 Classification of Computational Mechanics Techniques

The analysis of computational mechanics has evolved into five main approaches, as follows (see Figure 1):

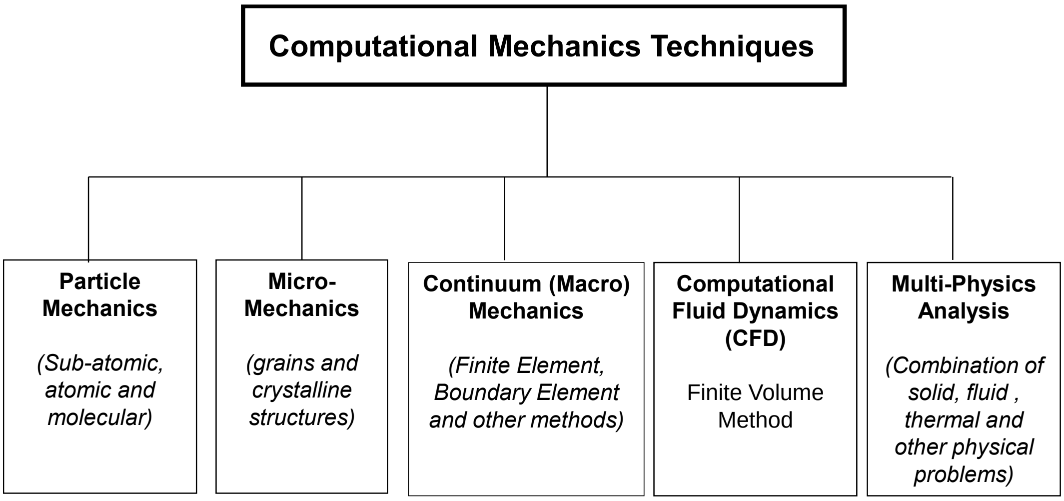


Figure 1: Overview of computational mechanics techniques

- *Particle mechanics* approaches are concerned with modelling atomic, sub-atomic and molecular levels and are mainly used in the fields of physics and chemistry.

- *Micro-Mechanics* methods attempt to model the crystallographic molecular structures, rather than the continuum.

- *Continuum (macro) Mechanics* techniques are widely used in engineering applications using small elements that are assembled together to form a domain or a structure. Finite Element and Boundary Element methods are examples of these techniques.

- *Computational Fluid Dynamics* (CFD) techniques are used to model liquid and gas flow and are widely used in modelling aerodynamics, hydrodynamics and combustion. Finite Volume methods are widely used in CFD applications.

- *Multi-physics* techniques cover the interaction of different physical processes, such as thermal, chemical, mechanical, and electrical processes. Applications suitable for multi-physics analysis include fluid-structure interaction, solidification/melting, aero-acoustics, air bag simulations, etc. Multi-physics solution strategies are usually divided into three types; sequential analysis, loosely-coupled analysis and closely coupled analysis.

3.2 Engineering Simulation Using Computer Modelling Techniques

Computer modelling approaches used to analyse solid and fluid mechanics in engineering problems have traditionally been classified into five main simulation approaches as shown in Figure 2.

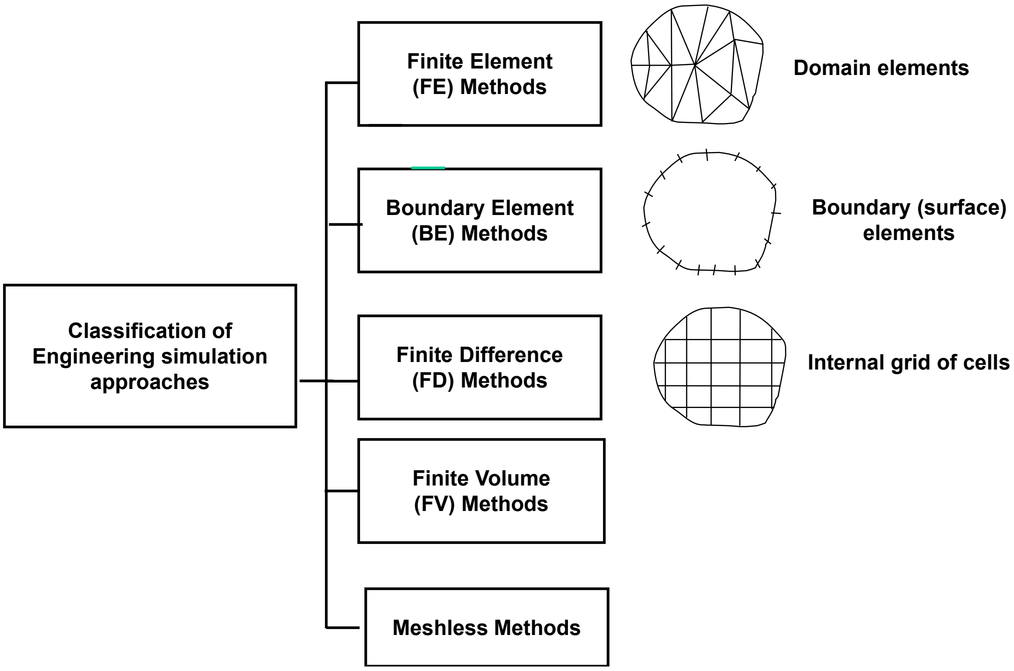


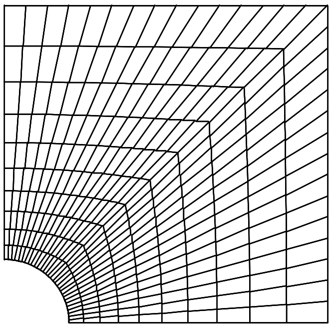
Figure 2: Overview of engineering simulation using computer modelling techniques

The main features of theses engineering simulation approaches are summarised below.

(a) The Finite Element (FE) Method

The main features of FE methods are:

- The entire solution domain (the volume) is divided into small '*finite*' segments (hence the name "*Finite Elements*").
- Each element is defined by its corner points (called "*nodes*"). Typical element shapes are triangular or quadrilateral (in 2D problems) or tetrahedral or hexahedral (in 3D problems).
- Over each element, the behaviour is described by the displacements of the nodes and the material law (stress-strain relationships). This is usually expressed in terms of the "*stiffness*" of the element.
- All elements are assembled together in a "*mesh*" and the requirements of continuity and equilibrium are satisfied between neighbouring elements. The assembly process results in a large system of simultaneous algebraic equations.
- The boundary conditions of the actual problem are applied to the assembly of the elements. This yields a unique solution to the overall system of linear algebraic equations.
- The solution matrix is sparsely populated (i.e. with relatively few non-zero coefficients).
- The equations are solved numerically to compute the displacements at each node. From the displacements, the stresses and strains over each element can be obtained.
- The FE method is very suitable for practical engineering stress analysis problems of complex geometries. To obtain good accuracy in regions of rapidly changing variables, a large number of small (fine) elements must be used.



Finite Element Mesh

Figure 3: Finite element mesh

(b) The Finite Difference (FD) Method

The main features of FD methods are:

- The entire solution domain is divided into a grid of "cells".
- The derivatives in the governing partial differential equations are written in terms of finite difference equations.
- A finite difference approximation is applied to each interior point so that the variables of each node are related to the values at the other nodes in the grid connected to it.
- Provided that the boundary conditions of the actual problem are satisfied, a unique solution can be obtained to the overall system of linear algebraic equations.
- The solution matrix is banded.
- The FD method is relatively easy to program. Its main serious drawback in practical engineering problems is that it is not suitable for problems with awkward irregular geometries. Furthermore, because it is difficult to vary the size of the difference cells in particular regions, it is not suitable for problems with rapidly changing variables such as stress concentration problems.
- FD methods are popular for heat transfer and fluid flow problems, rather than stress analysis problem.

(c) The Finite Volume (FV) Method

The main features of FV methods are:

- FV methods are widely used in computational fluid dynamics software.

- FV methods are similar to the finite difference methods, but without the need for a regular grid inside the domain.

- The solution domain is divided into “control volumes” where the continuity of fluid flow into and out of the control volume is satisfied. The discretized domain is called the “grid” or the “mesh.”

- The control volumes look similar to the elements used in FEA, but the variables of interest are located at the centre of the control volume (whereas in FEA they are at the ‘nodes’ of the element).

- Volume integrals are transformed into surface integrals using integral transformation methods such as the divergence theorem.

- Conservation equations for mass, momentum, energy, etc., are discretized into algebraic equations, and the equations are then solved to obtain the variables in the flow field.

4. Benefits (and Risks) of Computer Modelling

Benefits of Using Simulation and Modelling in Engineering

The use of simulation in engineering has many benefits:

- Ability to model complex designs of engineering components and structures
- Reduction of design and development time of products
- Reduction of the need for experimental testing of prototypes (although some testing is usually inevitable)
- Comprehensive information obtained regarding the distribution of stresses/strains or flow inside or around a structure.
- Better understanding of the effect of geometric features on the stress/strain state
- Identification of weaknesses and failure positions in engineering structures
- Ease of investigating alternative designs
- Demonstration of safety and meeting deign code requirements

Risks and Dangers of Using simulation techniques

Although there are many benefits of using engineering simulation, there are also some risks:

- Incorrect data input by the user ("Rubbish in" = "Rubbish out")
- Errors in translating the real-life boundary conditions into the software
- Incorrect use of the FEA and CFD software
- Using too few elements (the mesh being too coarse)
- Using badly shaped elements
- Attempting to model ill-conditioned problems

- Attempting to solve advanced non-linear problems without understanding the background theory

- Using the wrong type of elements (e.g. in FEA using shell elements when continuum elements would be best)

…and the worst risk is:

- Using FEA and CFD software as “black boxes” without understanding the underlying fundamental theory.

5. References/Further Reading

There is no specific textbook that exactly follows the syllabus covered in this module. Students are not required to purchase any textbook for this module. All topics will be fully covered during the formal lectures and in the handouts.

However, students are advised to study one or two relevant textbooks from the library in order to gain a more in-depth analysis of the topics. This is particularly useful if you experience difficulties in a specific topic.

There are numerous textbooks on FEA, CFD and numerical techniques, ranging from introductory textbooks to advanced mathematical textbooks.

The following textbooks are considered suitable as further reading for this module:

(a) Books on FEA

- Bathe, K.J. *Finite Element Procedures*, Prentice-Hall Int., London, 1996.
- Becker, A.A. *An Introductory Guide to Finite Element Analysis*, Wiley, London, 2003, ISBN: 978-1-86058-410-7
- Becker, A.A. *Understanding Non-linear Finite Element Analysis*, NAFEMS, Glasgow, 2001. ISBN 1-874376-35-2
- Fagan, M.J. *Finite Element Analysis - Theory and Practice*, Longman Scientific & Technical, 1992.
- Hellen, T.K. and Becker, A.A., *Finite Element Analysis for Engineers-A Primer*, NAFEMS publication, Glasgow, 2013, ISBN 978-1-874376-98-9.
- Owen, D.R.J. and Hinton, E. *A Simple Guide to Finite Elements*, Pineridge Press, 1980.
- Zienkiewicz, O.C. and Taylor, R.L. *The Finite Element Method - Volume 1*, McGraw-Hill, 1989.
- Zienkiewicz, O.C. and Taylor, R.L., *The Finite Element Method - Volume 2*, McGraw-Hill, 1991.

(b) Books on CFD

- Anderson, J.D., Jr., *Computational Fluid Dynamics: The Basics with Applications*, McGraw-Hill, Inc., New York, 1995.
- Tannehill, J.C., Anderson, D.A., and Pletcher, R.H., *Computational Fluid Mechanics and Heat Transfer*, 2nd Ed., Taylor & Francis, New York, 1997.

- Versteeg, H.K., and Malalasekera, W. *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, Addison Wesley Longman, Ltd., Harlow, England, 1995.

<u>(c) Other books on Numerical Methods</u>

- Ames, W.F., *Numerical Methods for Partial Differential Equations*, 2nd Ed., Academic Press, New York, 1977.

- Banerjee, P.K. *The boundary element methods in engineering*, McGraw-Hill, New York, 1994.

- Becker, A.A., *The boundary element method in engineering*, McGraw-Hill, London, 1992, ISBN 0-07-707415-7.

- Conte, S.D., and de Boor, C., *Elementary Numerical Analysis*, McGraw-Hill, New York, 1972.

- Mitchell, A. R. and Griffiths, D.F. *The finite difference method in partial differential equations*, Wiley, Chichester, 1980.

- Kreyszig, E. *Advanced engineering mathematics*, John Wiley, New York, 1979.

- Isaacson, E., and Keller, H.B., *Analysis of Numerical Methods*, John Wiley, New York, 1966.

- Press, W.H., Flannery, B.P., Teukolsky, S.A., and Vettering, W.T., *Numerical Recipes: The Art of Scientific Computing (FORTRAN Version)*, Cambridge University Press, Cambridge, 1989.

- Smith, G.D., *Numerical Solution of Partial Differential Equations: Finite Difference Methods*, 3rd Ed., Clarendon Press, Oxford, 1985.