MMME3086 Computer Modelling Techniques

Introduction

Module convenor: Dr Mirco Magnini
Office: Coates B100a
E-mail: mirco.magnini@nottingham.ac.uk

Module specs

Module name	Computer Modelling Techniques MMME3086
Credits	20 (requires ~200 hrs study)
Year/Level	Level 3
Semester	Autumn Semester
Module Convenor	Dr Mirco Magnini
Teaching staff	**Dr Mirco Magnini** (Numerical Methods, NM) **Dr Chris Bennett, Dr Luke Parry** (Finite Element Analysis, FEA) **Dr Donald Giddings** (Computational Fluid Dynamics, CFD)

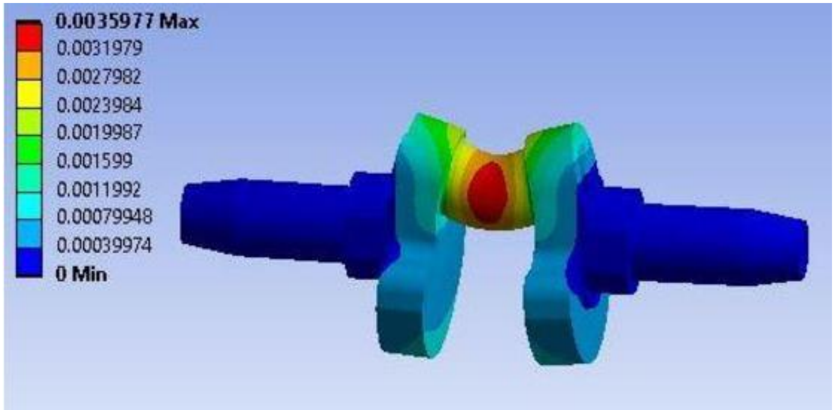
Educational Aims	To provide students with a basic knowledge and understanding of the mainstream computer modelling techniques used in modern engineering practice, including Finite Element and Computational Fluid Dynamics methods.

Introduction

Why do we need computer modelling techniques?

Because many problems in engineering do not have ‘analytical solutions’, and practical tests may be difficult, expensive, dangerous.

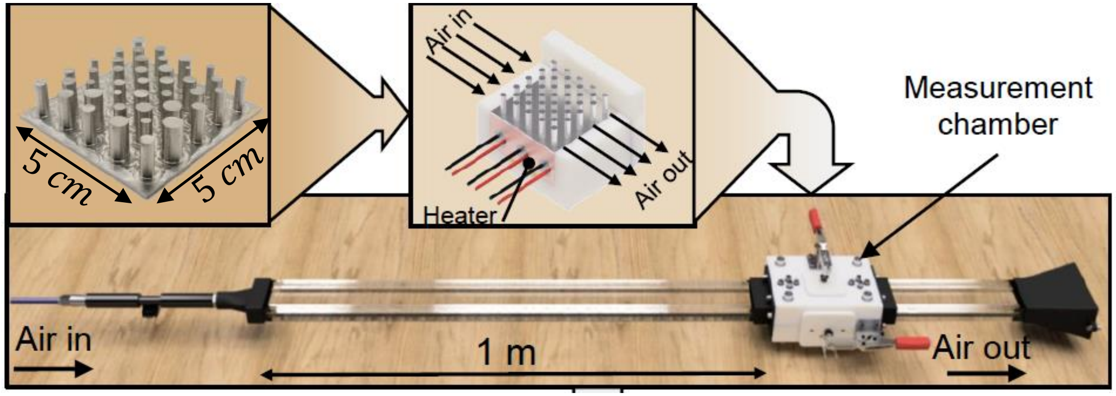
Computational analysis of aerodynamics on F1 car. Source: https://www.simscale.com/blog/front-wing-f1-car-optimize/

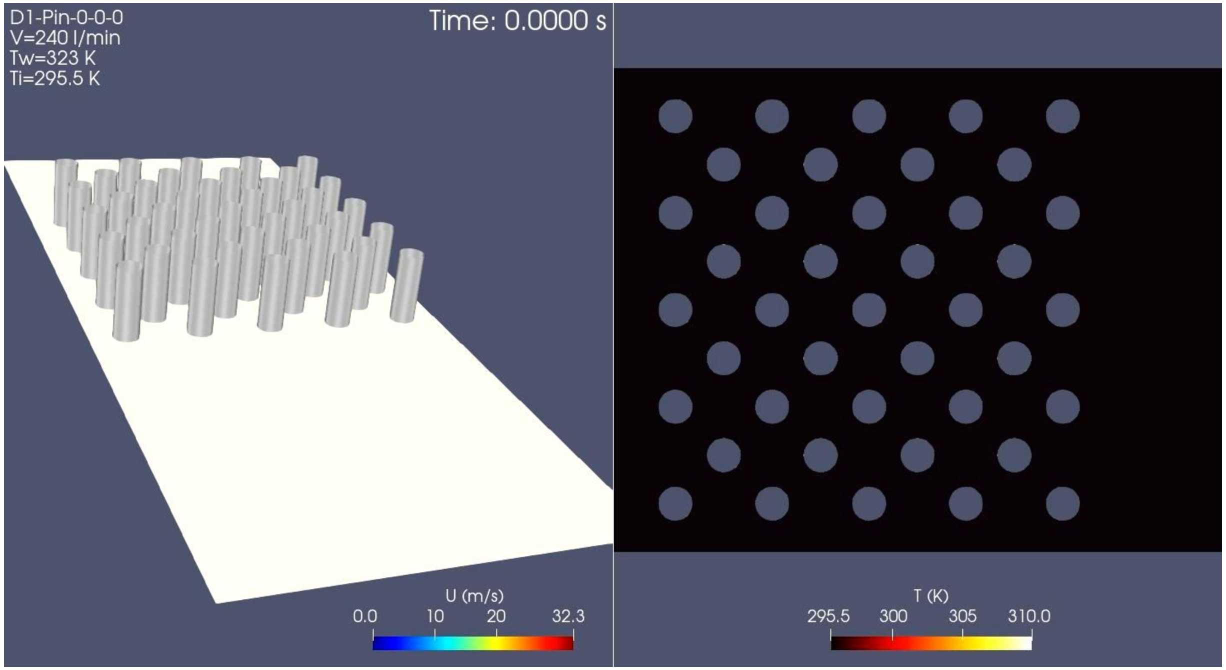


Computational structural static analysis of crankshaft. Source: https://www.semanticscholar.org/paper/Structural-Static-Analysis-of-Crankshaft-Mounika/f67a6d1005952e3d1788588eae677a1857b72199

Examples from my own research

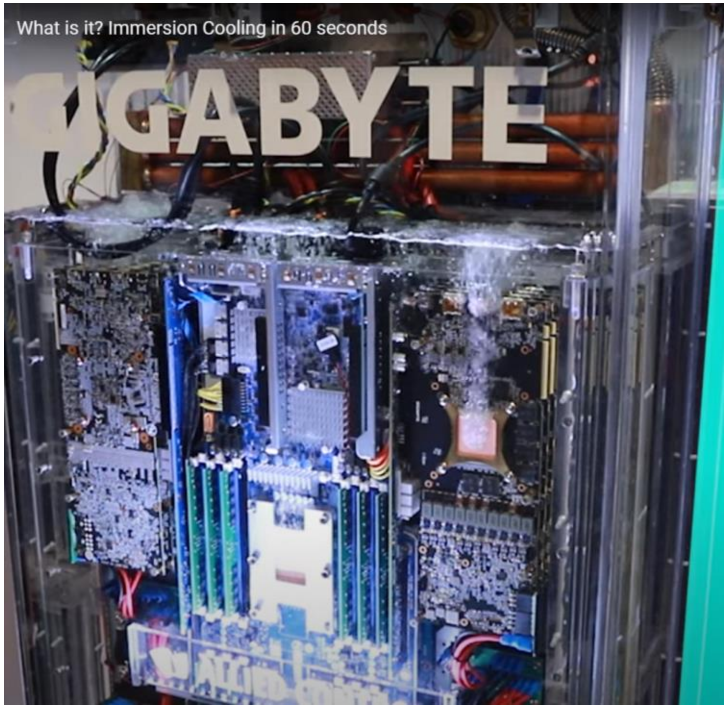
Fluid flow and heat transfer in pin-fin heat exchangers



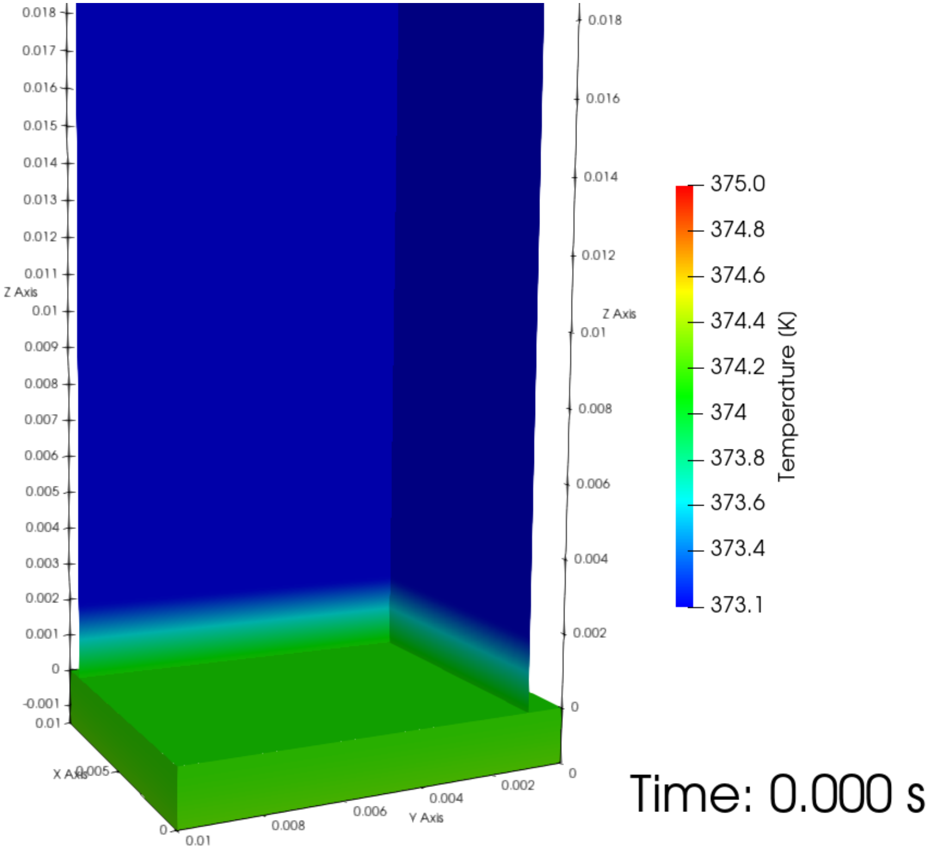


Examples from my own research

Water boiling over a surface at T>100 °C (p=1 bar)



Source: https://www.youtube.com/watch?v=U6LQeFmY-IU

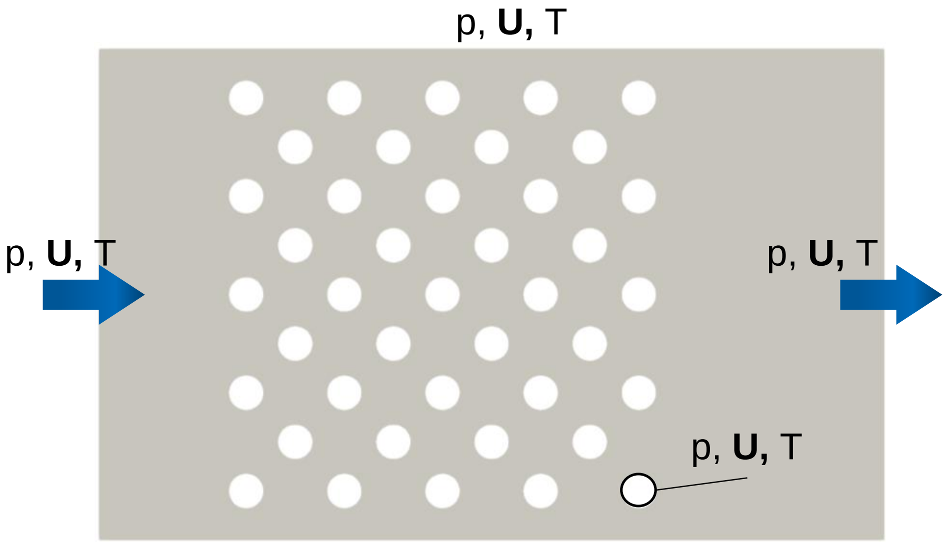


Introduction

In this module, we cover the main computational techniques to ‘simulate’ the dynamics of fluid (CFD) and solid structures (FEA) in engineering.

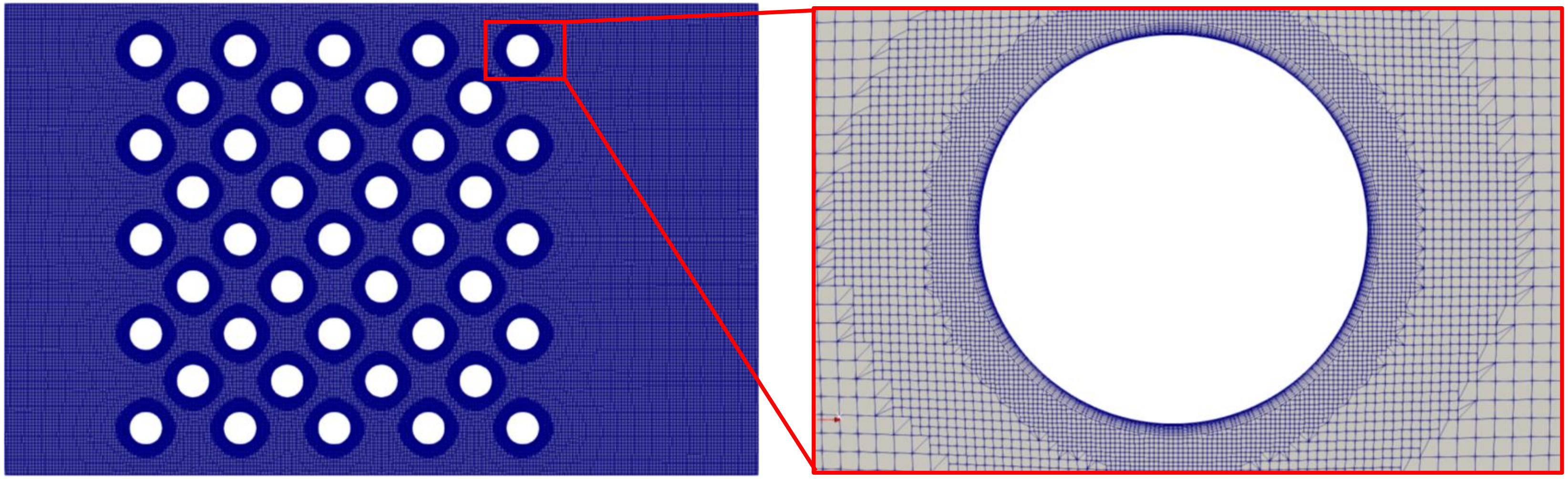
There exists a common procedure: the fluid or structural physical behavior is governed by a set of differential equations; these are discretised on a computational domain made of discrete elements, and then solved to obtain a solution that approximates the fluid flow or structure behaviour.

1. Geometry and boundary conditions are defined (example: fluid flow and heat transfer)



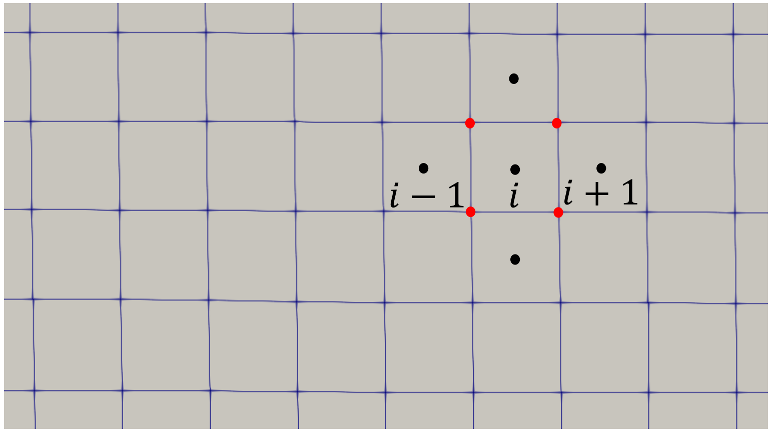
Introduction

2. The geometry is decomposed in n nodes (aka control volumes, finite elements)



Introduction

3. The governing equations are discretised for each node, i=1,…,n



$$a\frac{\partial^2\varphi}{\partial x^2}+b\frac{\partial^2\varphi}{\partial y^2}+c\varphi+\cdots=0$$

Finite Difference (FD)

$$i:\ \frac{\partial^2\varphi}{\partial x^2}\approx\frac{\varphi_{i+1}+\varphi_{i-1}-2\varphi_i}{(\Delta x)^2}$$

Mostly in CFD

Finite Volume (FV)

$$i:\ \frac{\partial^2\varphi}{\partial x^2}\approx\int_V\frac{\partial^2\varphi}{\partial x^2}dV$$

Mostly in CFD

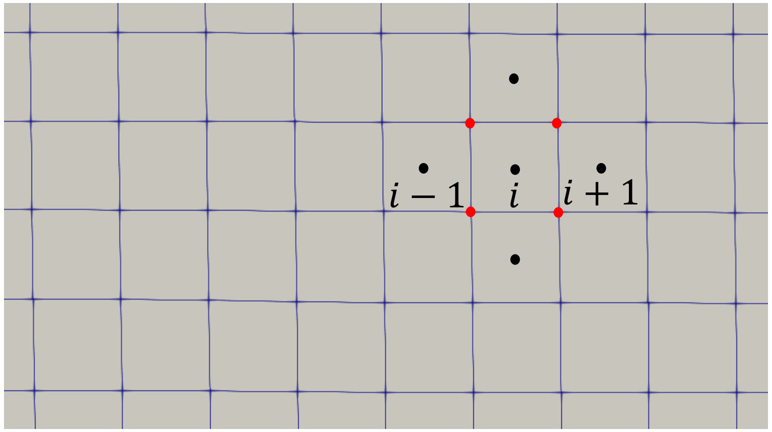
Finite Elements (FE)

$$i:\ \frac{\partial^2\varphi}{\partial x^2}\approx\int_V f(\boldsymbol{x})\frac{\partial^2\varphi}{\partial x^2}dV$$

Mostly used in solid mech, thus FEA

Introduction

4. Regardless of discretisation method, we obtain a linear system of n equations, where φ_i are our unknowns:



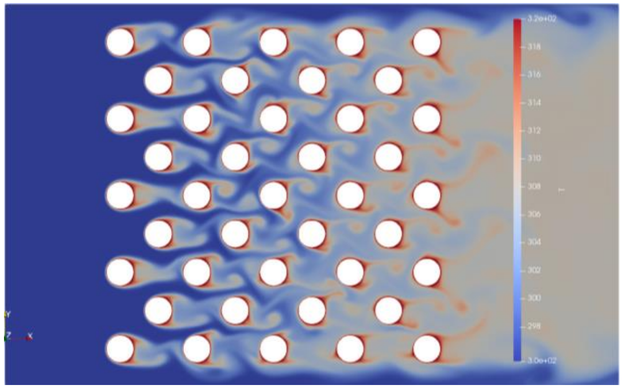
$$\begin{cases} i=1:\ a_{1,1}\varphi_1 + a_{1,2}\varphi_2 + \cdots + a_{1,i}\varphi_i + \cdots + a_{1,n}\varphi_n = b_1 \\ i=2:\ a_{2,1}\varphi_1 + a_{2,2}\varphi_2 + \cdots + a_{2,i}\varphi_i + \cdots + a_{2,n}\varphi_n = b_2 \\ \quad\vdots \\ i:\ a_{i,1}\varphi_1 + a_{i,2}\varphi_2 + \cdots + a_{i,i}\varphi_i + \cdots + a_{i,n}\varphi_n = b_i \\ \quad\vdots \\ i=n:\ a_{n,1}\varphi_1 + a_{n,2}\varphi_2 + \cdots + a_{n,i}\varphi_i + \cdots + a_{n,n}\varphi_n = b_n \end{cases}$$

In compact form: $\boldsymbol{A} \times \boldsymbol{\varphi} = \boldsymbol{B}$

$$\boldsymbol{A} = \begin{bmatrix} a_{1,1} & \cdots & a_{1,n} \\ \vdots & \ddots & \vdots \\ a_{n,1} & \cdots & a_{n,n} \end{bmatrix}, \boldsymbol{\varphi} = \begin{bmatrix} \varphi_1 \\ \vdots \\ \varphi_n \end{bmatrix}, \boldsymbol{B} = \begin{bmatrix} b_1 \\ \vdots \\ b_n \end{bmatrix}$$

5. Solution of the linear system: φ_i is obtained for each node i=1,…,n

Example: φ was the fluid temperature



Module specs

MMME3086 covers the whole simulation procedure and the most popular simulation software for CFD (fluids) and FEA (solid):

Dr. Mirco Magnini
Numerical Methods: FV, fundamental aspects, Matlab
Teaching weeks 1-3; NM coursework

Dr. Chris Bennett
Finite Element Analysis: FE, solid mechanics, Abaqus
Teaching weeks 4-7; FEA coursework

Dr. Don Giddings
Computational Fluid Dynamics: FV, ANSYS
Teaching weeks 8-11; CFD coursework

Timetable

Lectures: Thursday, 11-13, Coates Road Auditorium.

Seminars: Friday, 09-11, Coates Road Auditorium. Bring your own laptop.

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**

Method of assessment

Coursework	Three marked coursework assignments: NM coursework (30%) FEA coursework (35%) CFD coursework (35%) • The assignments do not require supervision. • Time required to complete each assignment: 6 hours approx. • Submission deadline: 2 weeks after releasing 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.
Only for students resitting from 22/23: final exam in January 2023, worth 40% of the module. Format identical to 21/22 and 22/23.	

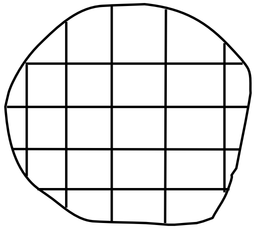
Learning Outcomes

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

Further introduction

The Finite Difference (FD) Method – NOT ON MMME3086

- 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.
- The solution matrix is banded.
- The FD method is relatively easy to program. Its main serious drawback is that it is not suitable for problems with awkward irregular geometries.
- FD methods are popular for heat transfer and fluid flow problems, rather than stress analysis problem.

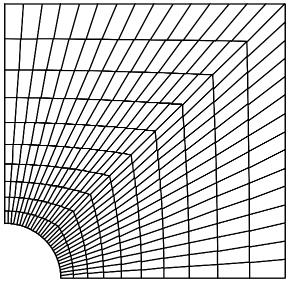


Internal grid of cells

Further introduction

The Finite Element (FE) Method

- 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*").
- All elements are assembled together in a "*mesh*" and the requirements of continuity and equilibrium are satisfied between neighbouring elements. .
- 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.
- The FE method is very suitable for complex geometries.

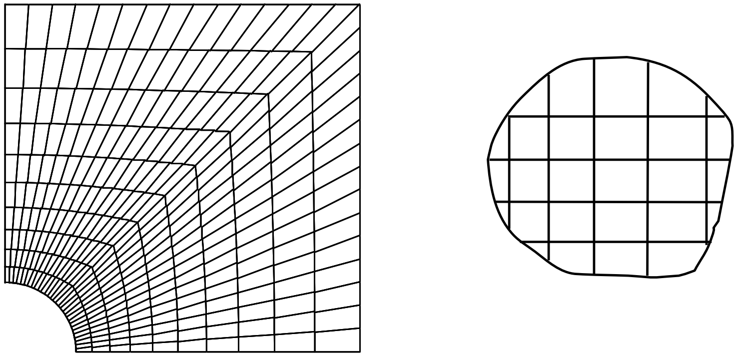


Finite Element Mesh

Further introduction

The Finite Volume (FV) Method

- FV methods are widely used in computational fluid dynamics software.
- The solution domain is divided into "control volumes" where the continuity of fluid flow into and out of the control volume is satisfied.
- 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 corners 'nodes' of the element).
- 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.



Further introduction

Benefits of Using Simulation and Modelling in Engineering

- 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 inevitable)
- Comprehensive information obtained regarding the distribution of stresses/strains or flow inside or around a structure.
- Identification of weaknesses and failure positions in engineering structures

Further introduction

Risks and Dangers of Using simulation techniques

- Incorrect data input by the user (i.e. "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)

- Attempting to solve 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.