University of Nottingham
Department of Mechanical, Materials and Manufacturing Engineering

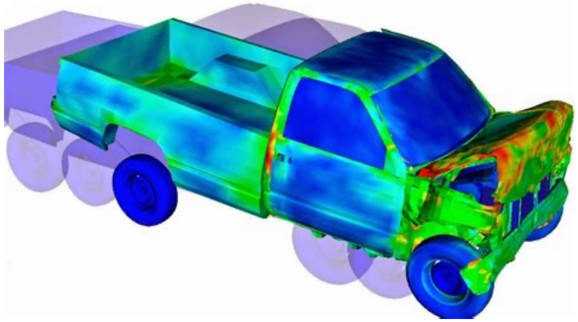
Computer Modelling Techniques

FE-01-01

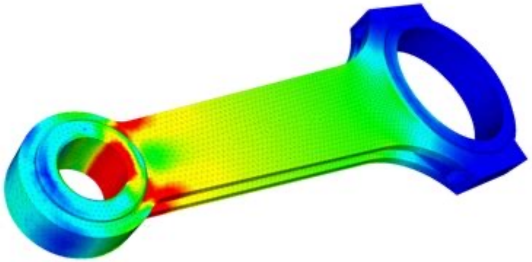
Introduction

1.1 Background

The Finite Element Method (FEM) is a numerical method for solving differential equations.

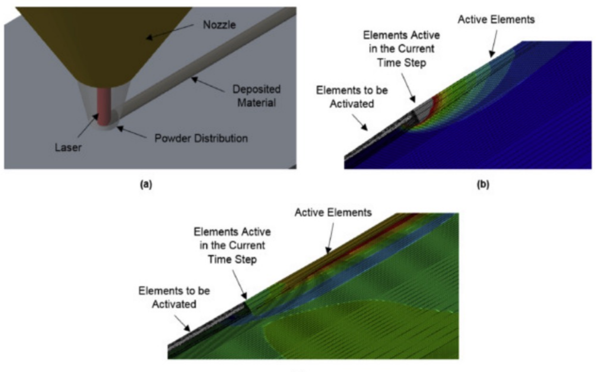


Finite Element Analysis (FEA) is the term generally used when applying FEM to solve a problem.

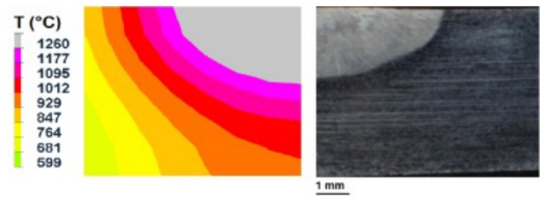


In this module we will be applying FEM to solve structural engineering/solid mechanics problems and linear elastic behavior is assumed throughout.

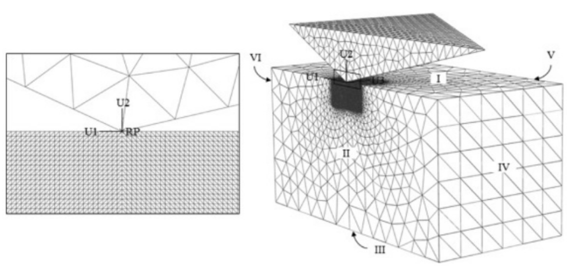
1.1 Background



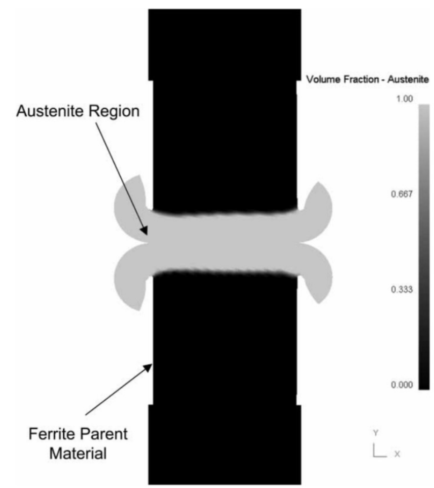
Direct Energy Deposition (Laser Deposition) – Thermal and residual stress prediction



Thermal Modelling of Fusion Welding



Nanoindentation – Property Characterisation using inverse analysis

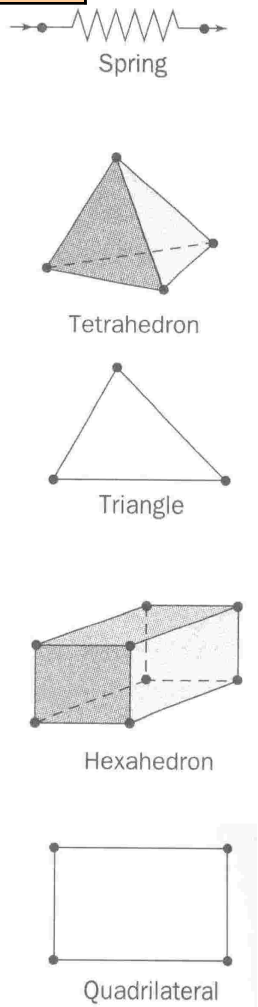


Inertia Friction Welding – Thermal, large plastic deformation & phase volume fraction prediction

1.1 Background

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.



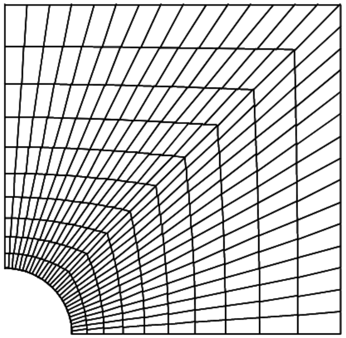
1.1 Background

- 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.

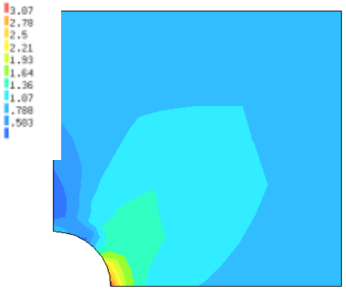
- The solution matrix is sparsely populated (i.e. with relatively few non-zero coefficients).

- 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

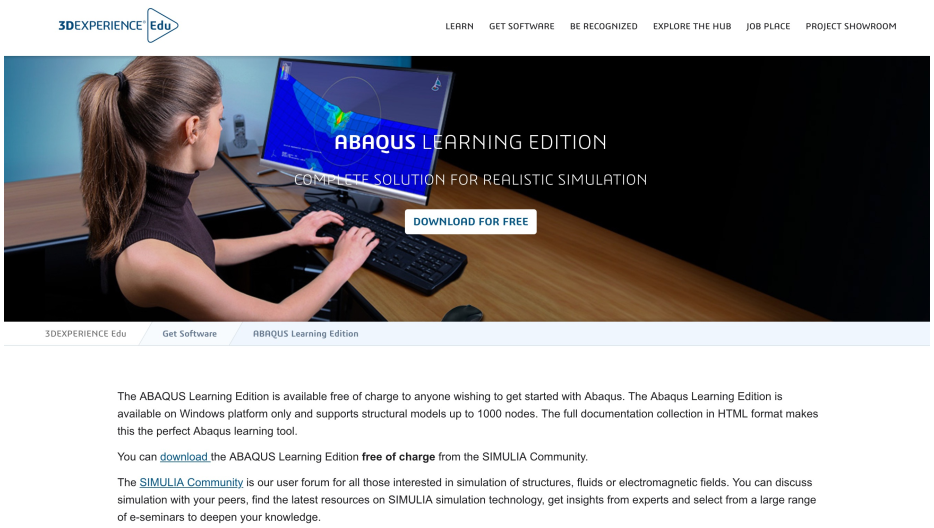


1.2 Abaqus

SIMULIA

Abaqus is a commercial piece of FEA software – it is very capable and flexible.

The Student Edition of Abaqus is available at:
https://edu.3ds.com/en/software/abaqus-student-edition

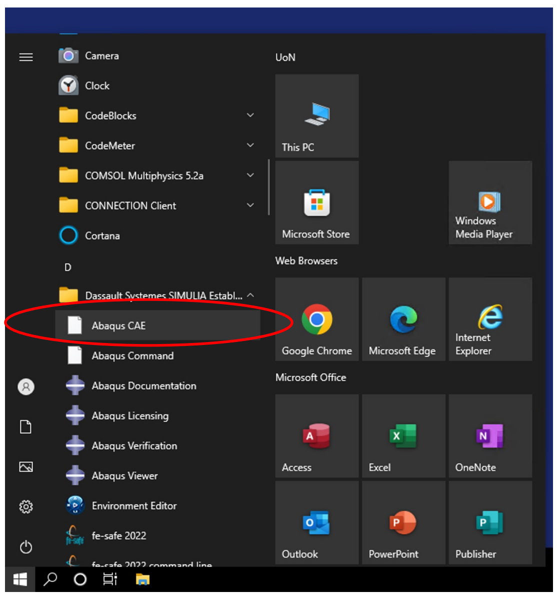


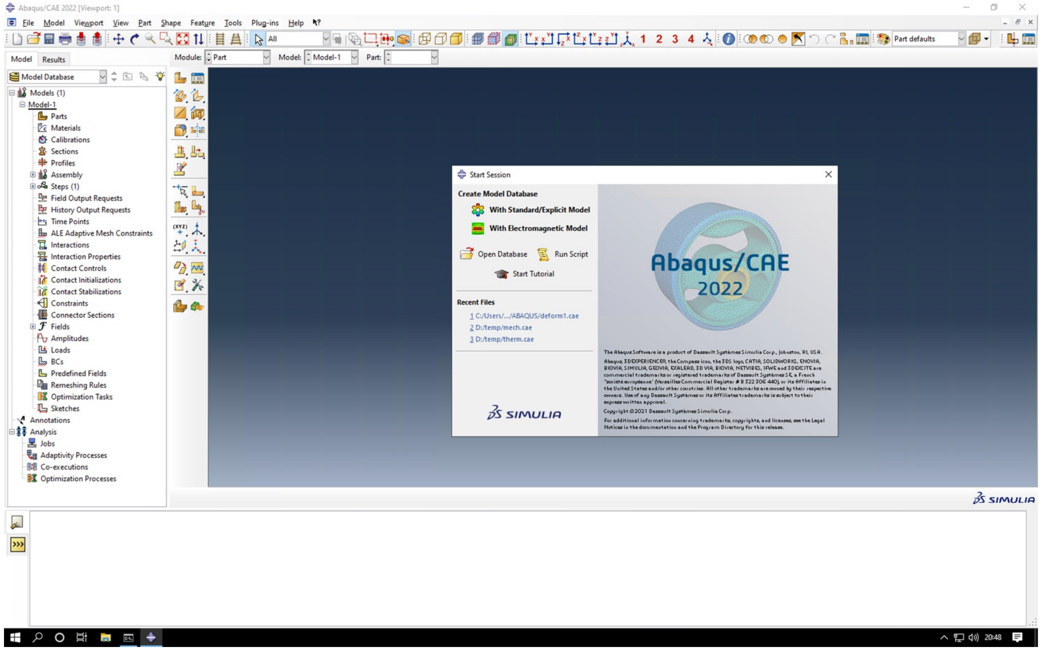
This version is limited to 1000 nodes – sufficient for the exercises and c/w in this module

1.2 Abaqus

Abaqus is a commercial piece of FEA software – it is very capable and flexible.

It is also available on the Engineering Desktop:





This is the full version without the 1000 node limit and can be accessed on and off campus

1.3 Steps for Performing FEA

1. Discretizing the domain – this step involves subdividing the domain into elements and nodes (creating the mesh).

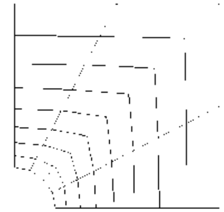
2. Writing the element stiffness matrices – the element stiffness equations need to be written for each element in the domain.

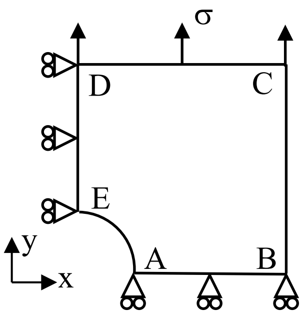
3. Assembling the global stiffness matrix – combining the individual element stiffness matrices to represent the entire domain/structure.

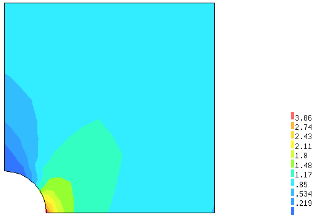
4. Applying the boundary conditions – like supports and applied loads and displacements.

5. Solving the equations.

6. Post-processing – to obtain additional information like the reaction forces and element forces and stresses.

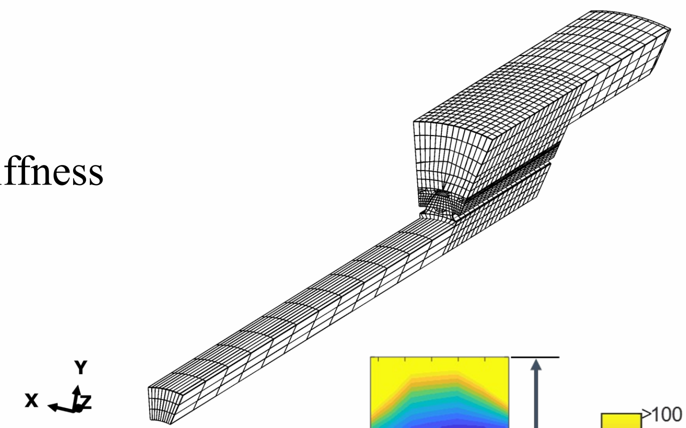


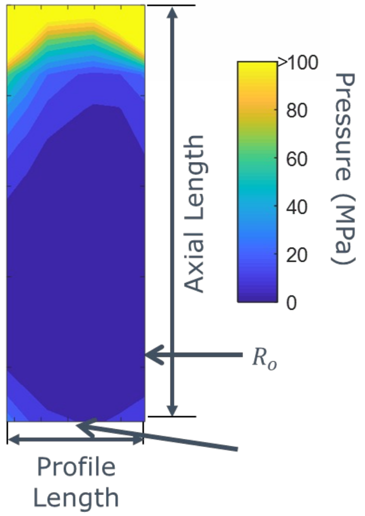




1.4 MMME3086 FEA Content

1. Direct and Energy based formulation of 1D Elements (Stiffness Matrices)
2. Assembly of Stiffness Matrices to form the Global Stiffness Matrix
3. 2D Pin-jointed structures
4. Continuum Elements
5. Structural Elements (e.g. shells/plates)
6. Practical FEA Guidelines





FE Coursework:	Set:	09/11/22
	Due:	23/11/22

35% of module mark