University of Nottingham
Department of Mechanical, Materials and Manufacturing Engineering

Computer Modelling Techniques

FE-04-02

PRACTICAL FE EXAMPLES

Perforated Plate Example

L = 100 mm
H = 50 mm
t = 5 mm
σ_o = 100 MPa

The objective of the analysis is to determine the stress concentration around the hole.

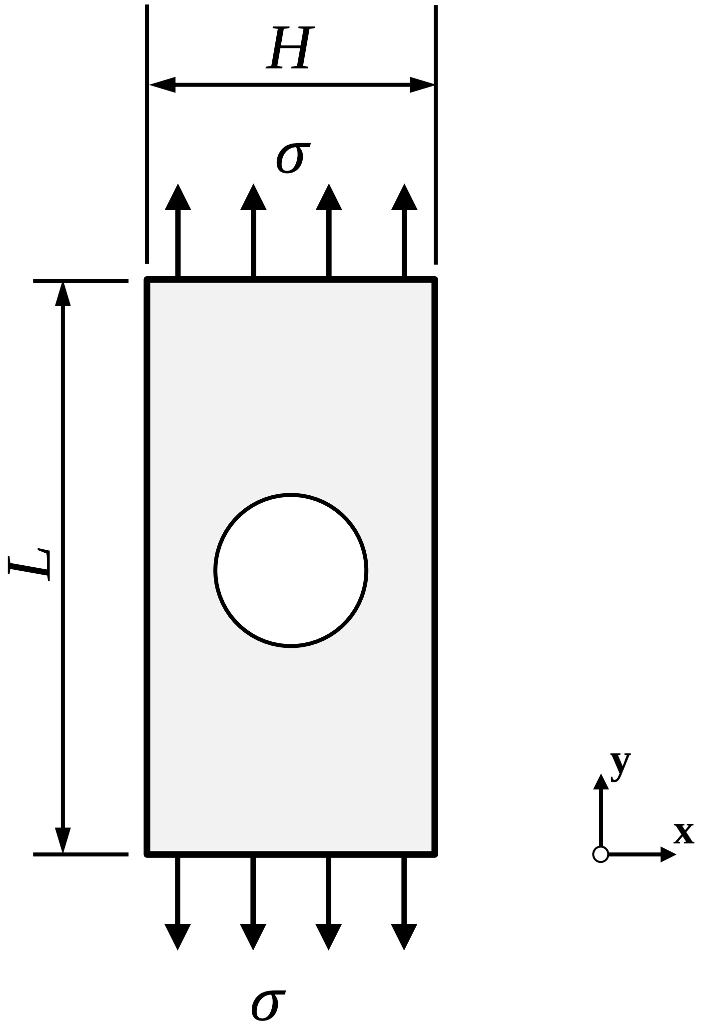


Figure 12: Perforated Plate Subject to Uniaxial Stress

Geometry

Since the plate thickness (in the z-direction) is small, 2D plane stress conditions are applicable.

The plate (both geometry and loads) is symmetrical about the horizontal and vertical axes.

Therefore, only a symmetrical quarter-model needs to be modelled.

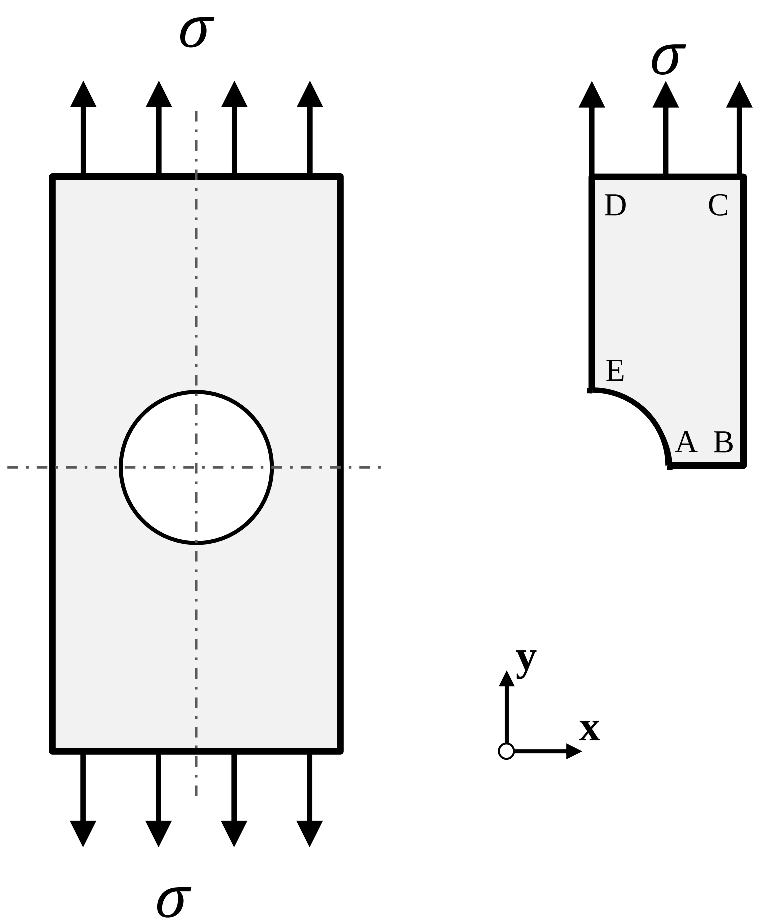


Figure 13: Symmetrical quarter of the perforated plate

Material Properties

Assuming a **linear elastic analysis**, the material properties needed are

E = 70 GPa
υ = 0.3

If the load is high enough to cause local plasticity around the hole, the elasto-plastic stress-strain curve, or at least the yield stress (σ_{yield}) must also be specified.

Boundary Conditions

Displacement Boundary Conditions

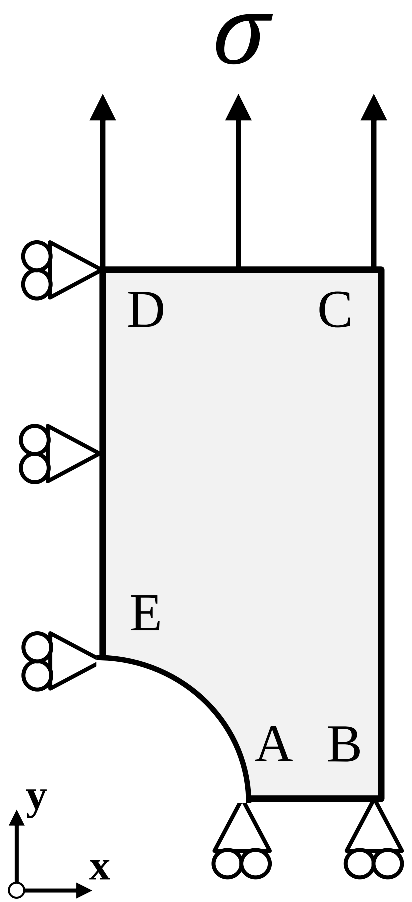
On the axes of horizontal and vertical symmetry;
the nodes can only slide along the symmetry lines.

(a) Zero y-displacements (roller conditions) specified on line AB.

(b) Zero x-displacements (roller conditions) specified on line DE.

Applied Loads

A uniform tensile stress (distributed load), σ_o , is specified at the top surface (line CD).



FE Model

- 2D plane stress linear (4-node) or quadratic (8-node) elements can be used here.
- Either quadrilaterals or triangles, or a combination of the two, can be used.
- Quadratic elements are suitable for this problem, since they can represent the circular hole geometry better than linear elements.
- Since stress concentration is expected around the hole, mesh biasing should be specified around the hole.
- If the yield stress is known, a plasticity check can be performed by checking the maximum value of the effective (von Mises) stress.

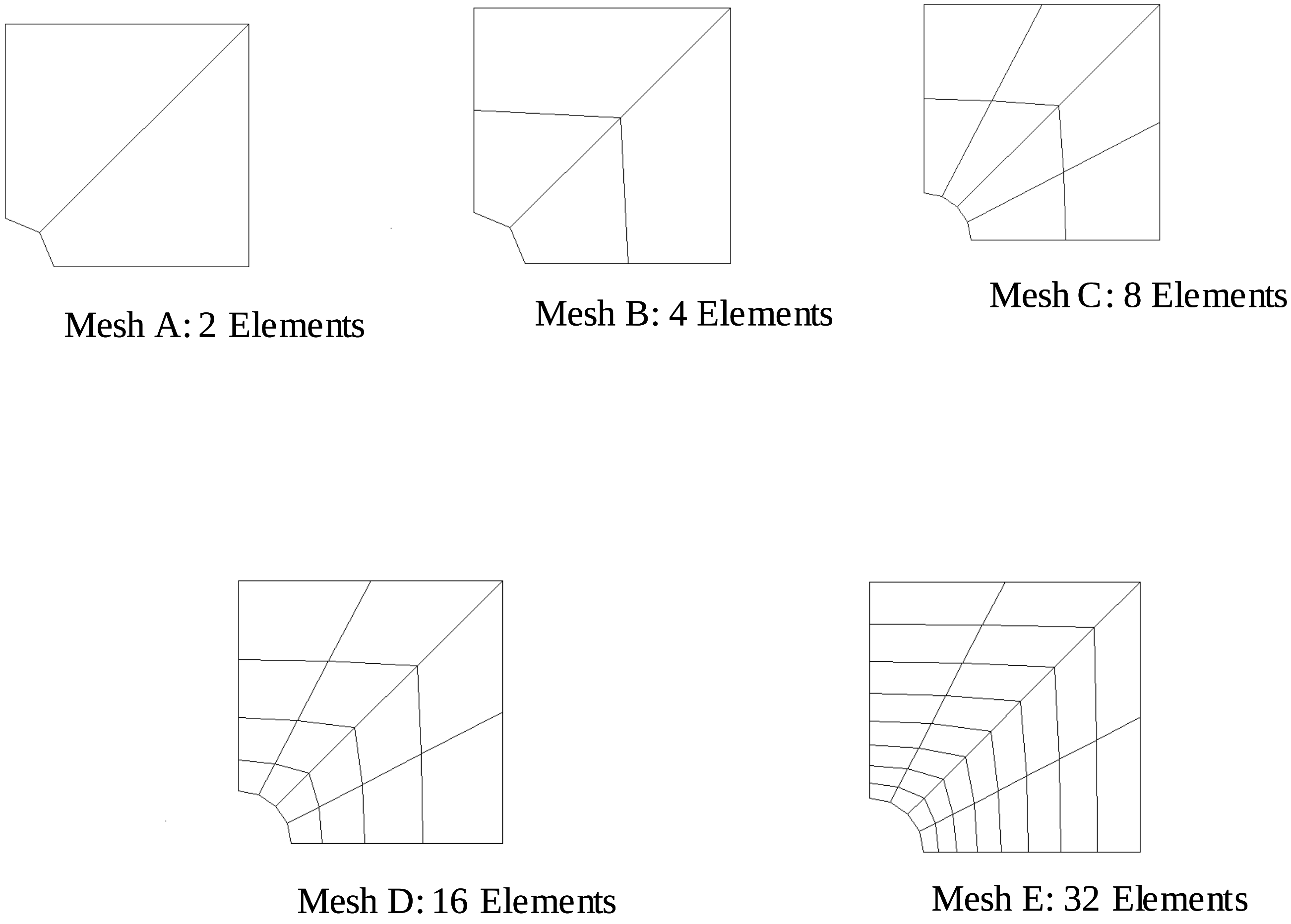


Figure 14: FE meshes used for the perforated plate example

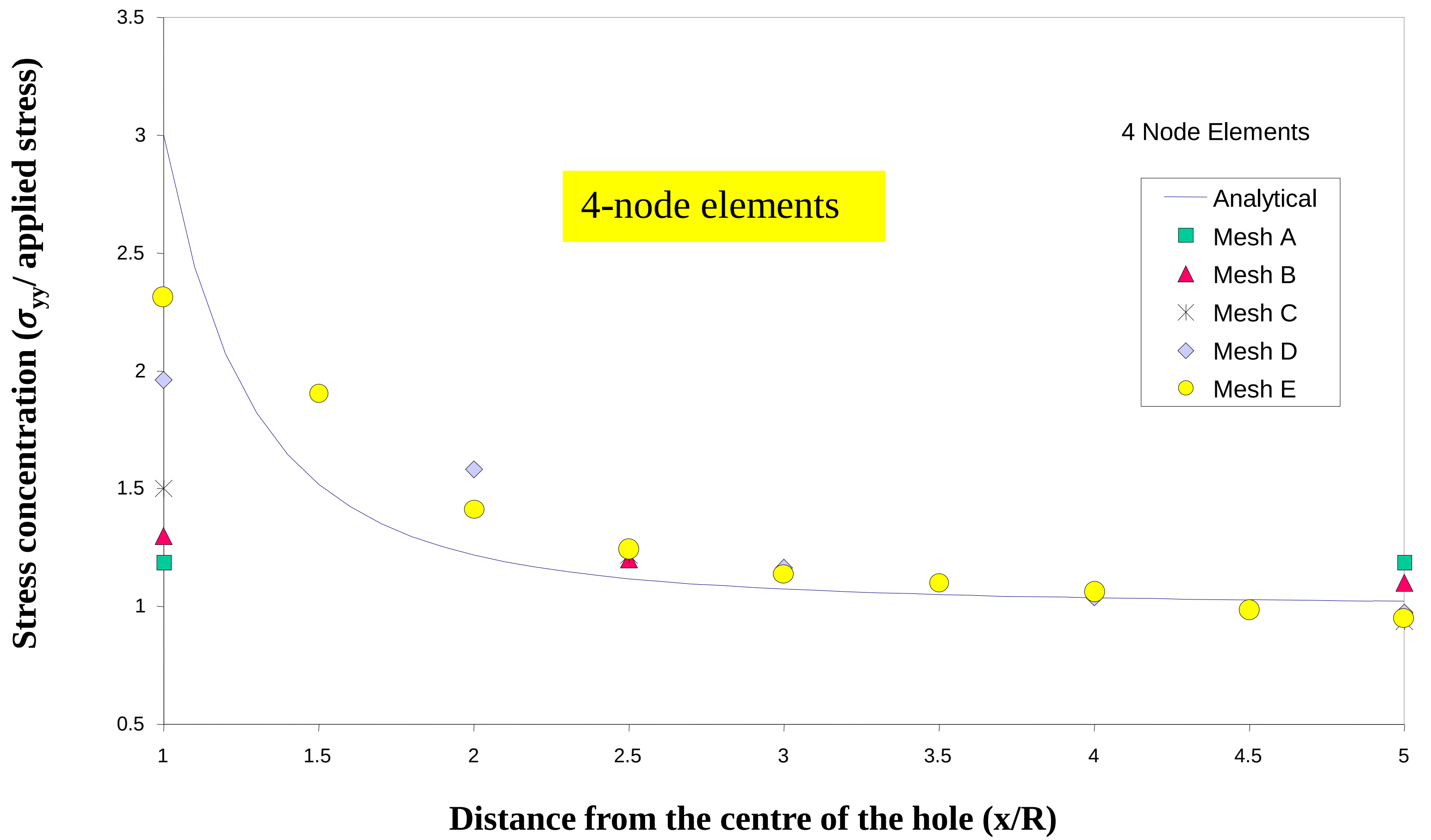


Figure 15: Comparison of FE and analytical solutions for the perforated plate example (4-node 'linear' elements)

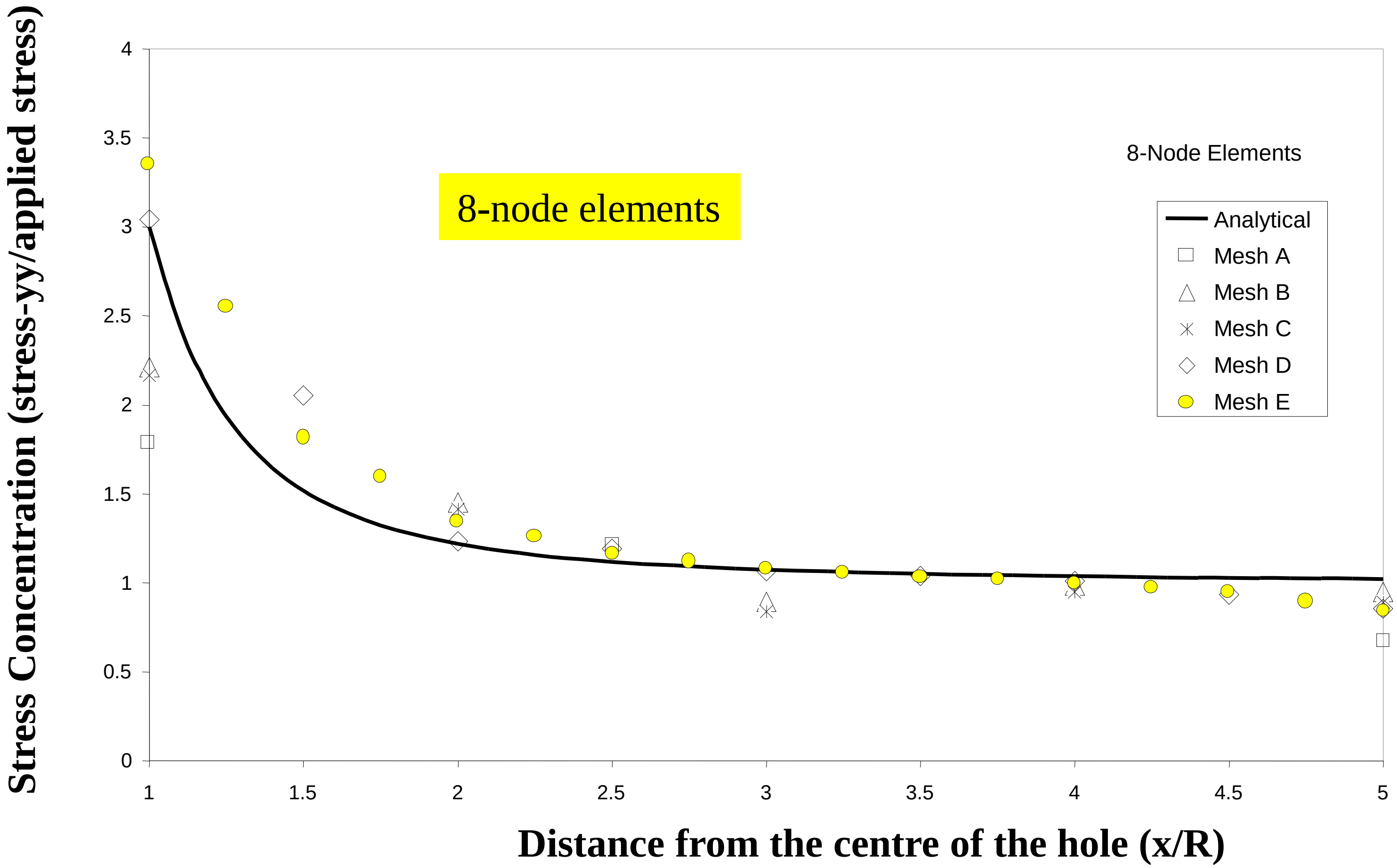


Figure 16: Comparison of FE and analytical solutions for the perforated plate example (8-node 'quadratic' elements)

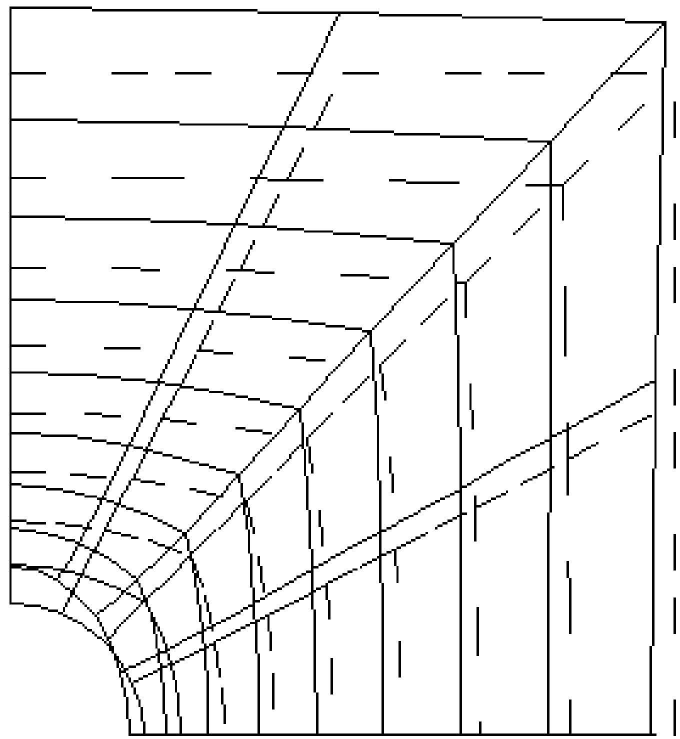


Figure 17: Exaggerated deformed shape (solid lines) for the perforated plate example

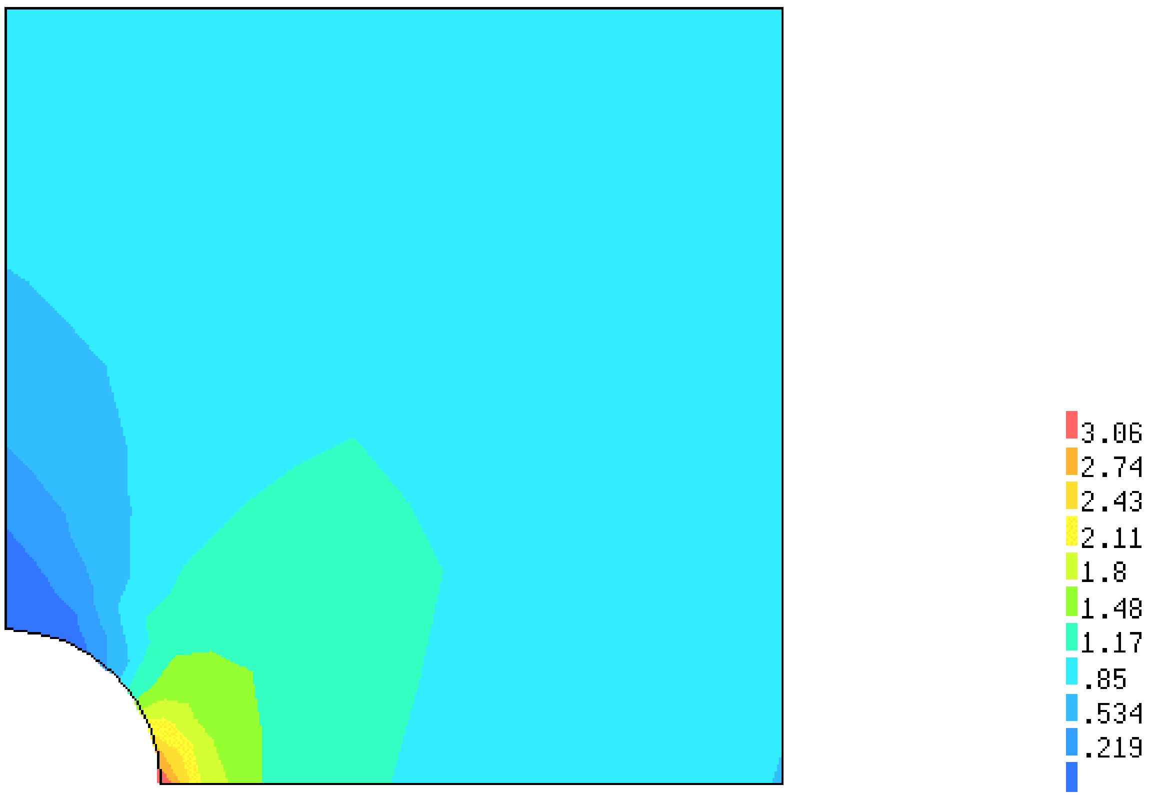


Figure 18: Stress contour plot (σ_{yy}) for the perforated plate example

Cantilever Beam Example

Problem Definition

The beam has a square cross-sectional area of side length t.

L = 2 m
t = 0.1 m
F = 1 kN

The objective of the analysis is to obtain the overall **deflection** of the beam.

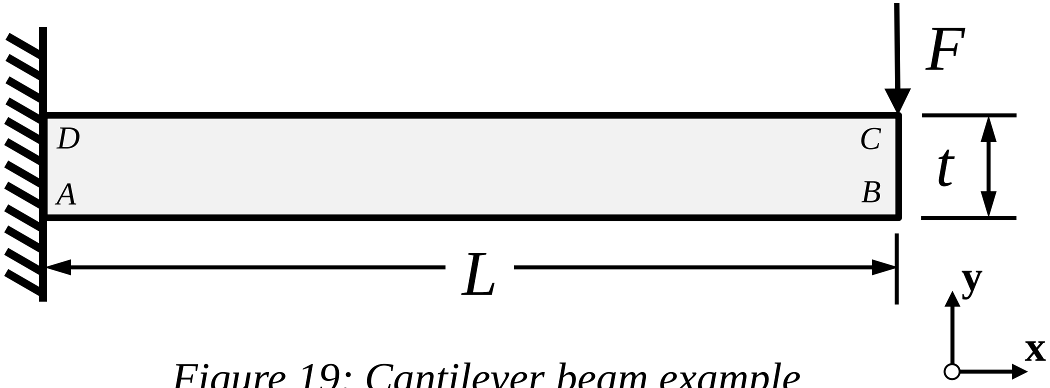


Figure 19: Cantilever beam example

Geometry

- Since there is no symmetry in this problem, the whole geometry has to be modelled.

- The geometry can be modelled with beam elements since the geometry and loads satisfy beam bending conditions, i.e. the geometry is long, slender and subjected to only transverse loads.

- It is also possible to model this problem with 2D plane stress elements since the thickness in the z-direction is sufficiently small.

Material Properties

Assuming a linear elastic analysis

E = 200 GPa

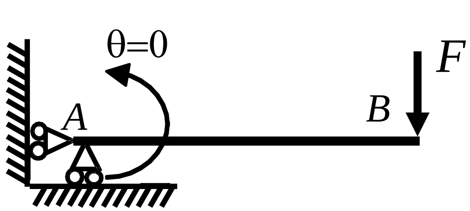
$\nu = 0.3$

Boundary Conditions

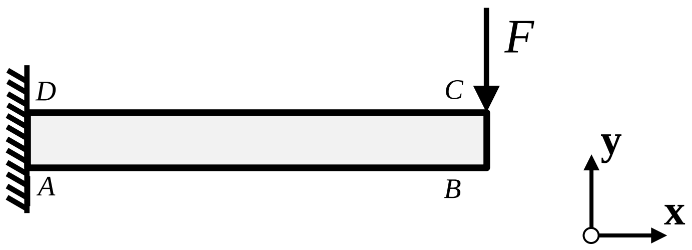
- The cantilever is built-in at the left hand side.

- If beam elements are used, then both the displacement and the slope (gradient of the displacement) at the built-in node must be prescribed as zero.

- If 2D plane stress elements are used, then all nodes on line AD must have zero displacements in the x and y directions, which automatically enforces the built-in condition.



(a) Beam FE model



(b) 2D continuum FE model

Applied Loads

A point load of magnitude *F* is applied to point C.

If a 2D plane stress model is used, this point force can either be applied at point C, or distributed along the line BC.

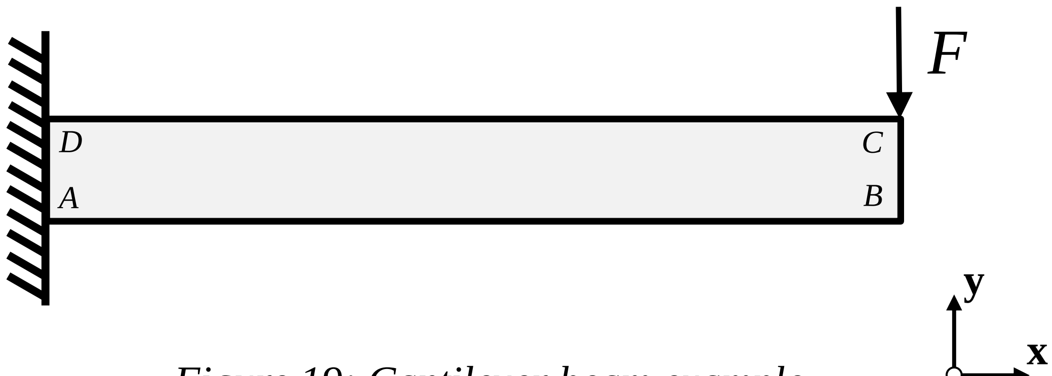
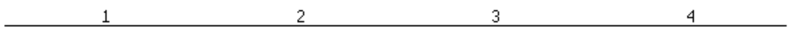


Figure 19: Cantilever beam example

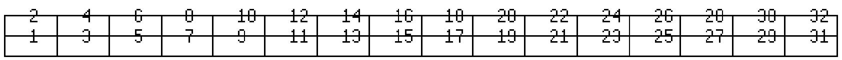
FE Model

Two types of elements can be used to model this problem;
Beam elements or 2D plane stress elements.

Of course, it is always possible to model this problem using 3D elements, but that would be unnecessary.



3-node beam element mesh (4 elements)



8-node 2D plane stress elements (32 elements)

Figure 20: FE meshes used for the cantilever beam problem

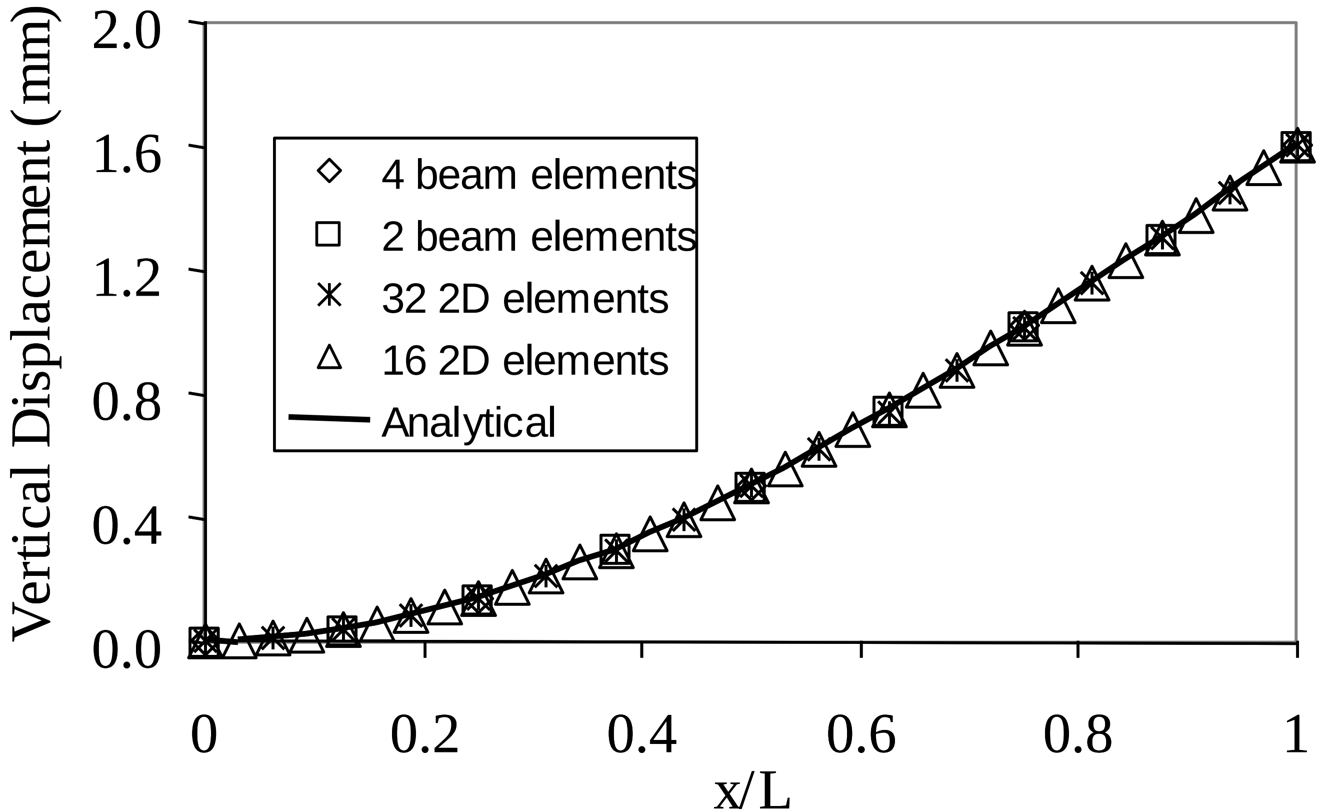
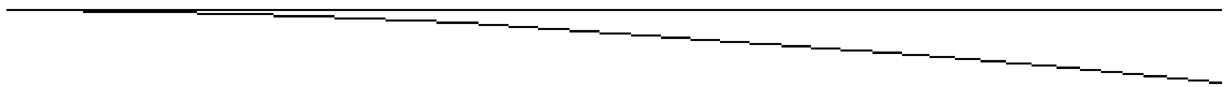
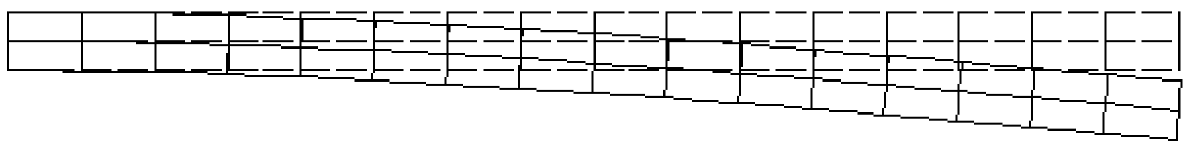


Figure 21: Comparison of FE and analytical solutions for the cantilever beam problem



Beam element mesh



2D plane stress element mesh

Figure 22: Deformed shapes (solid lines) for the cantilever beam problem

Key Points for FE

- FE analysis, specify the geometry, material properties, analysis type, displacement boundary conditions and applied loads.

- **Question** the inputs and assumptions and their sensitivity to your model

- **Simplify** – Reduce the size & complexity of your problem
  - Reduce complexity of your model (geometry, physics, loading conditions)
  - Reduce dimensions of your problem (3D → 2D)
  - Apply symmetry

- **Check** carefully FE solutions – they are and not taken for granted to be accurate.

- **Perform mesh convergence** studies on your analyses to have confidence in FE accuracy

> **!** **Validate** your models either analytically or experimentally

References for Commercial FE Software

On your future engineering journey you will hear about many different FE analysis software.

Na MSC Nastran

Marc® ★

AUTODESK® FUSION 360™ ★

NX [redacted] SIEMENS

LS-DYNA®

COMSOL MULTIPHYSICS®

Altair OptiStruct™ ★

Altair Inspire™ ★

Ansys

PrePoMax

CalculiX

However, what we learnt applies to all the software!

★ Student edition available