University of Nottingham
Department of Mechanical, Materials and Manufacturing Engineering

Computer Modelling Techniques

FE-07

FE Examples

7.1 Introduction

Two examples are presented to demonstrate how the FE method can be used to analyse stress analysis problems. The first example concerns stress concentration in a thin sheet of metal, while the second example highlights the use of beam elements to simulate a cantilever beam deflection. The examples demonstrate the effects of mesh refinement and the choice of elements on the accuracy of the FE solutions.

7.2 Perforated Plate Example (Continuum Elements)

Problem Definition

Consider a square plate of side length *L* and thickness *t* (in the z-direction) with a central circular hole of diameter *D*, subjected to a uniaxial stress σ_o, as shown in Figure 7.1. The numerical values used are *L* = 100 mm, *D* = 20 mm, *t* = 5 mm and σ_o = 100 MPa. The objective of the analysis is to determine the stress concentration around the hole.

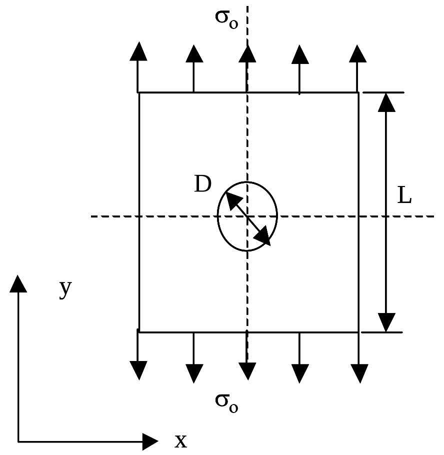


Figure 7.1: Perforated plate example

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 as shown in Figure 7.2.

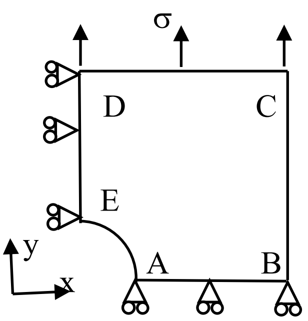


Figure 7.2: Symmetrical quarter of the perforated plate

Material Properties

Assuming an elastic analysis, the material properties needed are Young's modulus (E) and Poisson's ratio (ν). The values used here are E = 200 GPa and ν = 0.3. If the load is high enough to cause local plasticity around the hole, the elastoplastic stress-strain curve, or at least the yield stress (σ_{yield}) must also be specified.

Displacement Boundary Conditions

On the axes of horizontal and vertical symmetry, the nodes can only slide along the symmetry lines, i.e. displacements perpendicular to the axes of symmetry are prevented. Therefore, in this problem, there are two sets of displacement boundary conditions, as follows:

(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.

The analytical solution for the stress in the direction of the applied load in a perforated infinite plate is given by:

$$\sigma_{yy} = \frac{1}{2}\sigma_o \left[2.0 + \left(\frac{R}{x}\right)^2 + 3\left(\frac{R}{x}\right)^4\right] \qquad (7.1)$$

where x is the distance from the centre of the hole and R is the radius of the hole. Note that since the plate used in this problem is not infinite, the computed stresses will be expected to be slightly higher than those predicted by the analytical solution.

To demonstrate the effect of mesh refinement on the accuracy of the FE solutions, a number of meshes are used, ranging from 2 to 32 elements, as shown in Figure 7.3. Also, two types of elements are used; 4-node linear elements and 8-node quadratic elements.

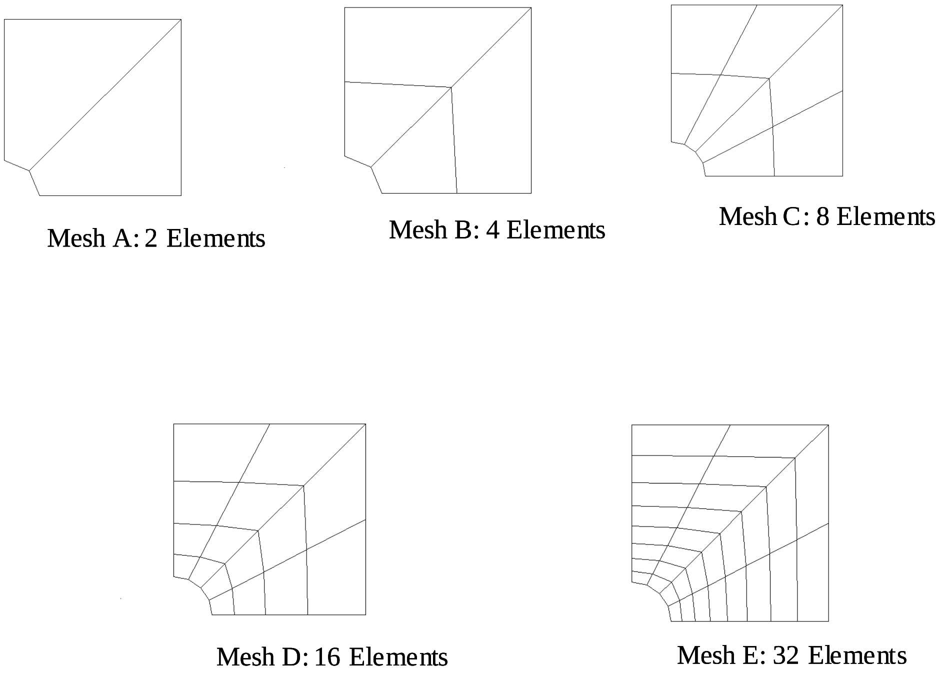


Figure 7.3: FE meshes used for the perforated plate example

Figures 7.4 and 7.5 show comparisons of the FE and analytical solutions for various mesh densities for the 4-node elements and 8-node elements, respectively. It can be seen that the FE solutions converge to the analytical solution as the mesh density is increased and that the quadratic 8-node elements provide better accuracy than the corresponding linear 4-node elements.

The deformed shape for the 32 quadratic element mesh is shown in Figure 7.6 where the deformations are exaggerated by multiplying them by a factor greater than 1. The deformed shape is useful in checking that the overall deformation of the body has followed the prescribed boundary conditions, i.e. the left and bottom sides slide along the axes of symmetry.

The stress contour plot for the vertical stress is shown in Figure 7.7, where, as expected, the highest stresses occur in the vicinity of the hole.

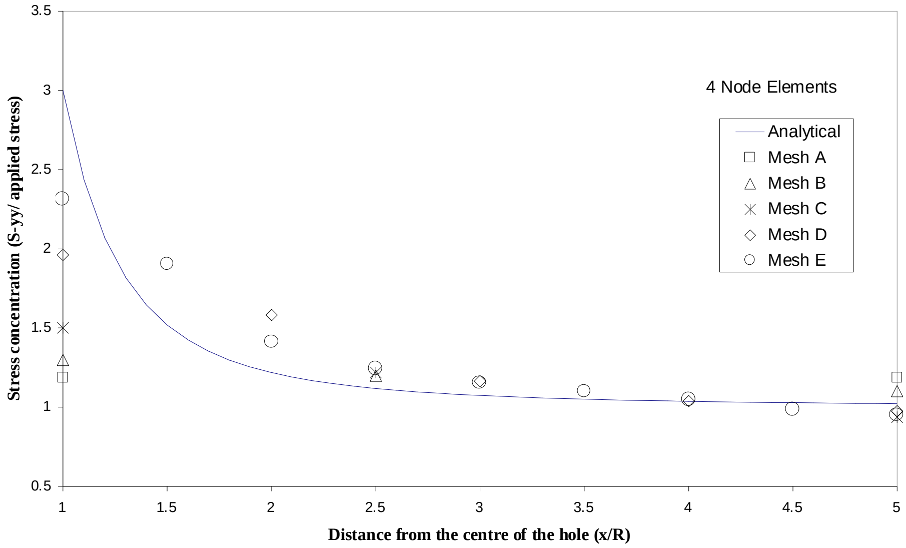


Figure 7.4: Comparison of FE and analytical solutions for the perforated plate example (4-node elements)

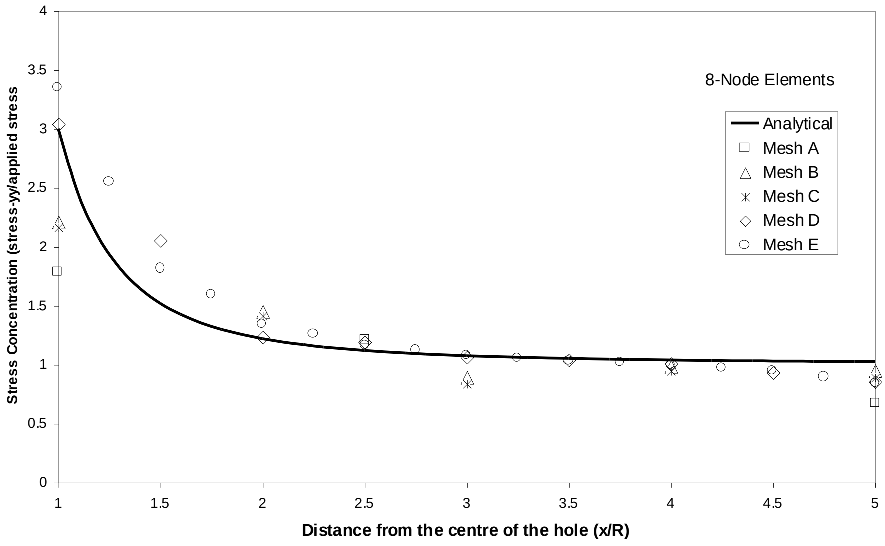


Figure 7.5: Comparison of FE and analytical solutions for the perforated plate example (8-node elements)

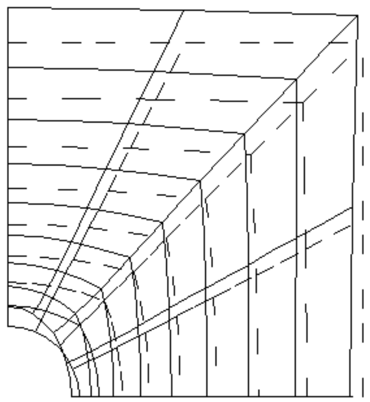


Figure 7.6: Exaggerated deformed shape (solid lines) for the perforated plate example

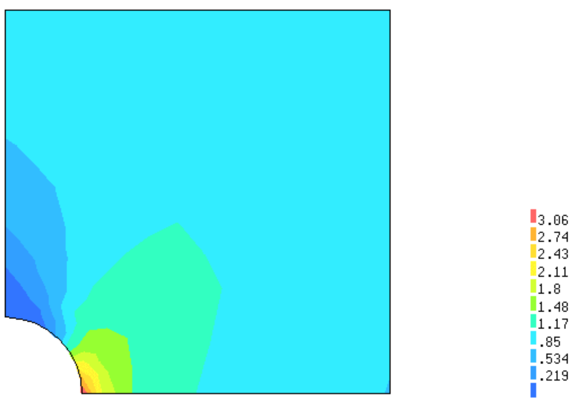


Figure 7.7: Stress contour plot (σ_{yy}) for the perforated plate example

7.3 Cantilever Beam Example (Beam Elements)

Problem Definition

Consider a cantilever beam of length *L* built in at one end and subjected to a concentrated force *F* at the other end, as shown in Figure 7.8. The beam has a square cross-sectional area of side length *t*. The numerical values used are : $L = 2$ m , $t = 0.1$ m and $F = 1$ kN. The objective of the analysis is to obtain the overall deflection of the beam.

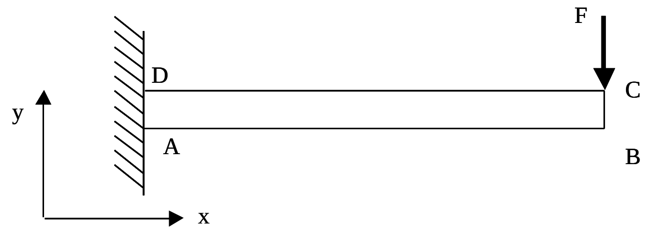


Figure 7.8: Cantilever beam example

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

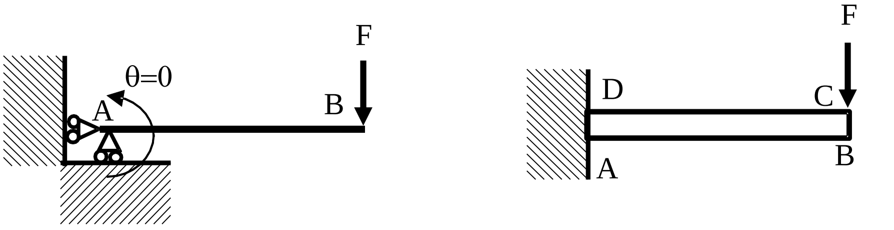
Material Properties

Assuming an elastic analysis, the material properties needed are only Young's modulus (E) and Poisson's ratio (ν). The values used here are E = 200 GPa and $\nu = 0.3$.

Boundary Conditions

The cantilever is built-in at the left hand side. If beam elements are used, then both the displacements (u_x and u_y) and the rotation (θ, gradient of the displacement) at the built-in node must be prescribed as zero, as shown in Figure 7.9. If the rotation (slope) at the built-in end is not specified, then point A becomes pin-jointed which is incorrect when modelling a built-in end.

If 2D plane stress continuum elements are used, then all nodes on line AD must have zero displacements in the x and y directions, which automatically enforce the built-in condition. Note that rotation is not a variable in 2D continuum elements.



(a) Beam FE model (b) 2D continuum FE model

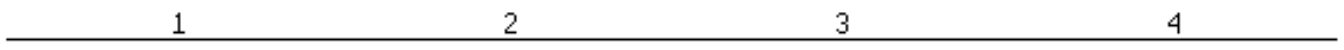
Figure 7.9: Cantilever FE models using beam and 2D continuum elements

Applied Loads

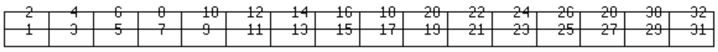
A point load of magnitude *F* is applied to point C. If a 2D plane stress model is used, this point force can either be applied at point C, or distributed along the line BC.

FE Model

As discussed above, 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 and would consume more computation time without improving the accuracy. Figure 7.10 shows a 3-node beam element mesh and an alternative 8-node 2D plane stress element mesh with 2x2 integration points.



3-node beam element mesh (4 elements)



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

Figure 7.10: FE meshes used for the cantilever beam problem

The analytical solution for the vertical displacement, *v*, in a cantilever beam can be derived from beam bending theory as follows:

$$v = \frac{F\,L^3}{E\,I}\left[\frac{1}{6}\left(\frac{x}{L}\right)^3 - \frac{1}{2}\left(\frac{x}{L}\right)^2\right] \tag{7.2}$$

Figure 7.11 shows a comparison of the FE and analytical solutions for a number of beam and 2D plane stress meshes, where it is clear that the FE solutions are in good agreement with the analytical solutions, even when a relatively small number of elements are used. The deformed shapes for the cantilever are shown in Figure 7.12.

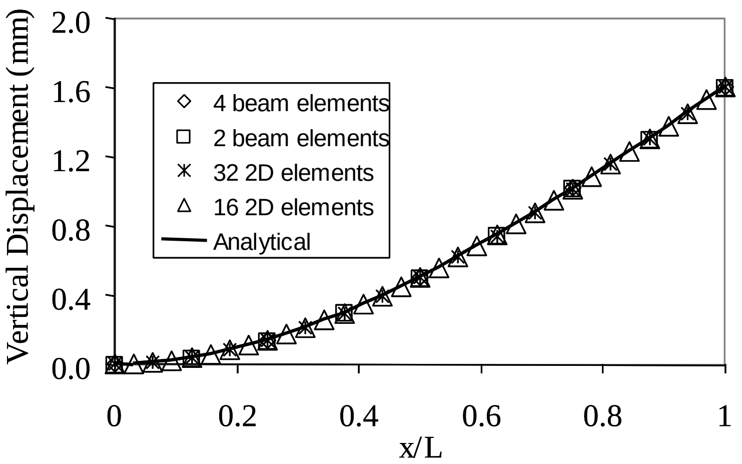


Figure 7.11: Comparison of FE and analytical solutions for the cantilever beam problem

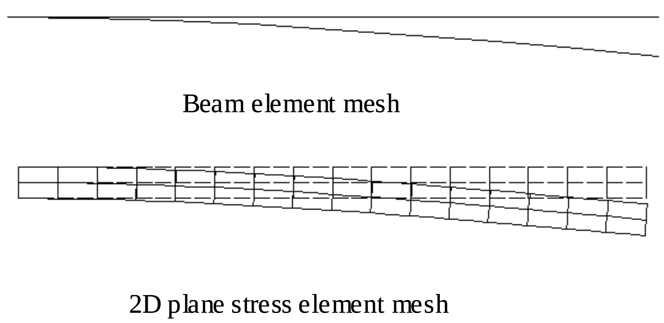


Figure 7.12: Deformed shapes (solid lines) for the cantilever beam problem