ENGINEERING TRIPOS PART IIA

LOCATION: DPO **EXPERIMENT 3D7**

FINITE ELEMENT METHOD

Summary

In this lab exercise the finite element method will be used to analyse the tensioned strip with a circular hole previously considered in the 3C7 lab.

Objectives

- Use a finite element package to solve linear elasticity equations
- Generate finite element meshes using a mesh generator
- Identify and define appropriate boundary conditions
- Compare finite element results with experimental measurements and analytical solutions
- Analyse the influence of the mesh refinement on the final results

---

Important: Before coming to the lab read this handout and load the Jupyter notebook on Google Colab here. Make sure the notebook it is running (use your university google account). Compare the equations given in this handout and their implementation in the notebook.

1 Problem description

This lab exercise involves performing finite element analyses of a tensioned strip with a circular hole using various meshes. The quantity of interest is the stress field in the strip. The meshes are generated with gmsh and the finite element analysis is performed with FEniCS. All instructions for running both are included in the google colab notebook.

The strip has the width 100 mm, length 300 mm, thickness 1.7 mm and hole radius $R = 14.8$ mm. The Young's modulus is $E = 70$ GPa and Poisson's ratio is $\nu = 0.33$. The strip is subjected to uniform tension of σ_0 applied at its two ends through. The total applied force is $F = 10$ kN. Due to symmetry, it is sufficient to model one quarter of the strip as shown in Figure 1.

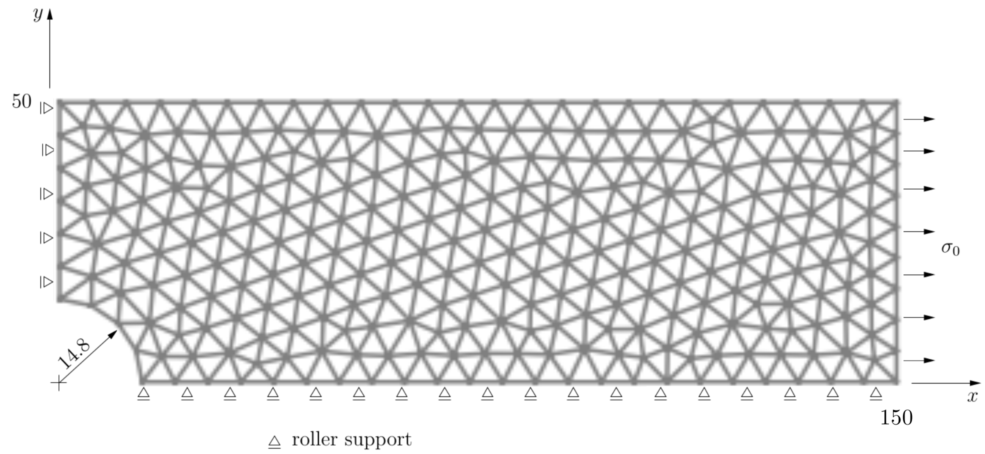


Figure 1: Geometry of one quarter of the strip.

2 Finite element model

2.1 Equilibrium equations in strong and weak form

In planar elasticity the primary unknown is the displacement field with two components

$$\boldsymbol{u} = \begin{bmatrix} u_x \\ u_y \end{bmatrix} \tag{1}$$

The axial strains ϵ_{xx} and ϵ_{yy}, and the shear strain ϵ_{xy} are given by

$$\begin{aligned} \epsilon_{xx} &= \frac{\partial u_x}{\partial x} \\ \epsilon_{yy} &= \frac{\partial u_y}{\partial y} \\ \epsilon_{xy} &= \frac{1}{2}\gamma_{xy} = \frac{1}{2}\left(\frac{\partial u_y}{\partial x} + \frac{\partial u_x}{\partial y}\right) . \end{aligned} \tag{2}$$

Notice that this definition implies $\epsilon_{xy} = \epsilon_{yx}$. In computer implementations of the finite element method, it is convenient to arrange the strains in a column vector

$$\boldsymbol{\epsilon} = \begin{bmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ 2\epsilon_{xy} \end{bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} u_x \\ u_y \end{bmatrix} \tag{3}$$

This equation is written in a more compact form using the *symmetric gradient operator* $\boldsymbol{\nabla}_S$, that is,

$$\boldsymbol{\epsilon} = \boldsymbol{\nabla}_S \boldsymbol{u} \quad \text{with} \quad \boldsymbol{\nabla}_S = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \tag{4}$$

The equilibrium equations in component form are given by

$$\begin{aligned} \frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \sigma_{xy}}{\partial y} &= 0 \\ \frac{\partial \sigma_{yx}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} &= 0 \end{aligned} \tag{5}$$

Notice that there are no distributed body forces in this example. The equilibrium equations can be written as

$$\begin{bmatrix} \frac{\partial}{\partial x} & 0 & \frac{\partial}{\partial y} \\ 0 & \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \end{bmatrix}, \tag{6}$$

or more compactly as

$$\boldsymbol{\nabla}_S^T \boldsymbol{\sigma} = \mathbf{0}\,. \tag{7}$$

The strains $\boldsymbol{\varepsilon}$ and stresses $\boldsymbol{\sigma}$ are related to each other through Hooke's law. The generalisation of Hooke's law to two dimensions can be written as

$$\boldsymbol{\sigma} = \boldsymbol{D}\boldsymbol{\varepsilon}\,. \tag{8}$$

The matrix $\boldsymbol{D}$ depends on whether plane stress or plain strain conditions are assumed. A plane stress model is appropriate because the considered strip is relatively thin so that the through-thickness stress is zero. According to the Data Sheet, the $\boldsymbol{D}$ matrix for plane stress is

$$\boldsymbol{D} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix}. \tag{9}$$

The weak form of the elasticity equations is the principle of virtual work which you already know from Part I of your studies. As it will be derived in the lectures the weak form is given by

$$\int_\Omega (\boldsymbol{\nabla}_S \boldsymbol{w})^T \boldsymbol{\sigma}(\boldsymbol{u})\, d\Omega = \int_{\Gamma_t} \boldsymbol{w}^T \bar{\boldsymbol{t}}\, d\Gamma_t\,, \tag{10}$$

where $\boldsymbol{w}$ is the test function, $\boldsymbol{u}$ is the trial function, and $\bar{\boldsymbol{t}} = \begin{bmatrix} \bar{t}_x & \bar{t}_y \end{bmatrix}^T$ is the applied traction on the Neumann boundary Γ_t.

2.2 Boundary conditions

Essential (Dirichlet) boundary conditions The displacement of all nodes on the symmetry line must be constrained to be zero in the direction normal to the line. At the same time, these nodes should be free to move parallel to the line.

The finite element model must have sufficient constraints to prevent rigid body motion. If rigid body motion is possible the stiffness matrix will be singular, or rank deficient.

Natural (Neumann) boundary conditions The natural boundary condition appears in the right-hand side of the weak form (10). The value of the traction $\bar{t}_x$ is found by dividing the total applied force ($F = 10\,kN$) by the cross-sectional area ($100\,mm \times 1.7\,mm$) $\Rightarrow \sigma_0 \approx 58.8\,N/mm^2$.

2.3 Influence of the mesh and element type

You will define several meshes as described below, using triangular elements. You should always make sure that the element aspect ratio (i.e., length/width) is less than three and never greater than five in order to limit numerical errors due to an ill-conditioned stiffness matrix. When defining graded meshes, use small elements in regions of large stress gradient or where high accuracy is required.

- A uniform mesh of approximately 250 triangular elements.
- A uniform mesh of approximately 15000 triangular elements.
- A mesh with approximately 250 triangular elements. Adjust the number of elements and their size distribution to create a graded mesh in-line with the advice given above. The objective is to obtain the greatest accuracy with the limited number elements. This will be by trial and error.

3 Off-centre hole analysis.

This section is optional for the standard laboratory. In the Laboratory Report, results from this analysis are not required. Students using this laboratory exercise for their Full Technical Report must do this analysis in full and discuss it in their report. A script for the generation of a mesh corresponding to this new geometry can be copied from here and used in the jupyter notebook.

Using a very fine reference mesh with linear triangles and a mesh of quadratic triangles with no more than 600 degrees of freedom, analyse the same plate (length, width and thickness are identical to the previous section) with an off-centre hole. The plate is subjected to uniform tension of σ_0 applied at its two ends through a tensometer. The total applied force is $F = 10$ kN. The distance of the hole centre from the centre-line is $e = 16$ mm and the radius of the hole again is $R = 14.8$ mm. The material properties are the same as those used above. Begin by making a sketch of the boundary value problem to analyse (taking into account any symmetries) and then adjust the number of elements and their size distribution in-line with the advice given in section 1 to obtain the greatest accuracy with these limited number of nodes. Plot the distributions of horizontal and vertical stress along axes passing through the centre of the hole and parallel to the sides of the plate.

4 Write-ups

You should refer to the advice given in the General Instructions document Sections 2.2 and 2.3.

4.1 The Laboratory Report [3 hours]

This report should be submitted online within 15 days of your second lab session, latest at 4pm.

This should be word-processed and no more than three pages, excluding any diagrams and graphs. The report pertains only to the centred hole problem of section 1 and should contain the following in addition to any other issues that you may wish to discuss.

- Along the horizontal and vertical axes, plot the vertical and horizontal stress distributions calculated with the different meshes. Compare these stresses with those obtained for a hole in an infinite plate that is stretched uniformly and also compare the stress at the periphery of the hole with Howland's analytical solution, see Appendix B.
- On the same graph plot the stresses measured experimentally (you can download these on the 3D7 lab Moodle webpage, these are results from the 3C7 Laboratory).
- Discuss reasons for any differences between the measured, analytical and finite element stresses and comment on the relative accuracies of the finite element models with the different meshes.
- Examining the von Mises stress contours indicates where yielding would first initiate in this strip. Estimate the applied stress σ_0 at which yielding initiates in a strip made from an Al-alloy (Yield strength of the Al-alloy is 200 MPa).
- Briefly discuss the finite element modelling approach used to analyse the problem.

4.2 The Full Technical Report [10 additional hours]

This report should be submitted online before the end of the term.

Guidance on the preparation of Full Technical Reports is provided in Appendix I of the General Instructions document and in the CUED booklet A Guide to Report Writing, with which you were issued in the first year. If you are submitting a Full Technical Report on this experiment, you can either use the books indicated in the References section below or do your own literature review. You should include your Laboratory report as an Appendix and refer to it as appropriate.

The Full Technical Report should provide further investigation of aspects addressed in the lab. You are free to pick topics that interest you. Examples of relevant topics are listed below (you should cover at least 3 topics if you pick up from this list).

- Discuss the boundary conditions used to represent the off-centre hole problem. How was rigid body motion prevented in this problem? The loading can also be applied by prescribing displacements (instead of applying traction) at one of the two ends of the plate. Implement this in the notebook and describe how this changes the results.
- For the off-centre hole problem, plot the vertical and horizontal distributions along axes passing through the centre of the hole and parallel to the sides of the plate. Compare these with the strain gauge measurements that you can download from Moodle (these are results from the 3C7 Laboratory). The off-centre hole problem has no known analytical solution. Given more time, how could you test the accuracy of your calculation of the stresses around the hole?
- The Rayleigh-Ritz process of approximation is frequently used in elastic analysis and is similar to the finite element method. Appealing to the Rayleigh-Ritz process or otherwise, explain why element selection and mesh design affects accuracy of finite element calculations.

- Integration over elements in commercial FE codes is usually done using Gauss quadrature, as discussed in the 3D7 lectures. Briefly discuss the choice of quadrature rules and explain how an 'hourglass instability' can occur with an incorrect choice of the order of integration.
- Using python to post-process your results and the 15000-element very fine mesh as a reference, calculate the L^2 norm of the error on coarse meshes and comment on the convergence rate (refer to the guidelines in the google colab notebook).

Fehmi Cirak (fc286)

January 2022

References

Cook, R.D., *Finite element modelling for stress analysis*, 1995.

Cook, R.D., Malkus, D.S., Plesha, M.E., *Concepts and Applications of Finite Element analysis*, 1989.

Zienkiewicz, O.C., Taylor, R.L., *The Finite Element Method. Vol.1:Basic formulation and linear problems*, 1989.

Petter Langtangen, H. and Logg, A., *Solving PDEs in Python: The FEniCS Tutorial I*, 2017. See also: https://fenicsproject.org/tutorial/.

A 3D7 DATA SHEET (extracts)

Element relationships

Elasticity

Displacement	$\mathbf{u} = \mathbf{N}\,\mathbf{a}_e$
Strain	$\epsilon = \mathbf{B}\,\mathbf{a}_e$
Stress (2D/3D)	$\sigma = \mathbf{D}\,\epsilon$
Element stiffness matrix	$\mathbf{k}_e = \int_{V_e} \mathbf{B}^T\,\mathbf{D}\,\mathbf{B}\,\mathrm{d}V$
Element force vector (body force only)	$\mathbf{f}_e = \int_{V_e} \mathbf{N}^T\,\mathbf{f}\,\mathrm{d}V$

Elasticity matrices

2D plane strain

$$\mathbf{D} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \dfrac{1-2\nu}{2} \end{bmatrix}$$

2D plane stress

$$\mathbf{D} = \frac{E}{(1-\nu^2)} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \dfrac{1-\nu}{2} \end{bmatrix}$$

B Analytical Solutions

B.1 Infinite Plate

There are several analytical solutions for the stress distributions in thin plates with circular holes. The best known is the solution for a circular hole of radius R in an infinite plate, see Figure 4, which is obtained in the lectures.[1] The stress components on the centre lines are given by

$$\begin{aligned}
\text{x-axis:} \quad & \frac{\sigma_{xx}}{\sigma_0} &=& \ 1 - \frac{5R^2}{2x^2} + \frac{3R^4}{2x^4} && (11)\\
& \frac{\sigma_{yy}}{\sigma_0} &=& \ \frac{R^2}{2x^2} - \frac{3R^4}{2x^4} && (12)\\
\text{y - axis:} \quad & \frac{\sigma_{xx}}{\sigma_0} &=& \ 1 + \frac{R^2}{2y^2} + \frac{3R^4}{2y^4} && (13)\\
& \frac{\sigma_{yy}}{\sigma_0} &=& \ \frac{3}{2}\left(\frac{R^2}{y^2} - \frac{R^4}{y^4}\right) && (14)\\
\text{both axes:} \quad & \sigma_{xy} &=& \ 0 && (15)
\end{aligned}$$

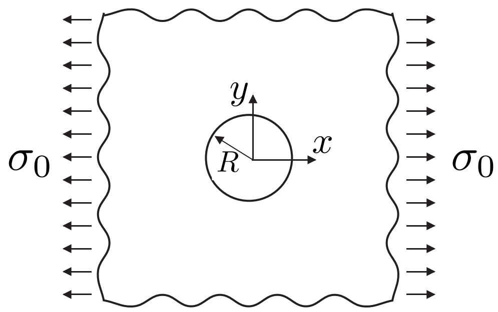


Figure 2: Infinite plate with a hole.

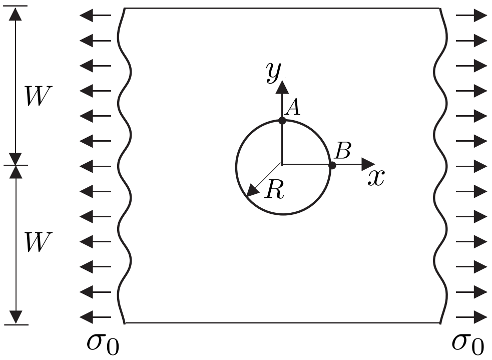
The stress concentration predicted by Equations (11)-(14) decays rather quickly away from the hole. Therefore, this solution can be used for non-infinite plates, provided that the edges of the plate are sufficiently far from the hole that the stresses on the edges are approximately equal to the remote stress field. If this is not the case, alternative solutions may be used.

B.2 Howland

Howland[2] derived semi-analytical expressions for the stress concentration factors at the edge of a symmetric hole of radius R in a plate of finite width $2W$. These factors have been calculated for a range of plate geometries and are shown in the table below.

[1] S.P. Timoshenko and J.N. Goodier, Theory of Elasticity, p. 90-93.

[2] Howland, R.C.J. (1930). On the stresses in the neighbourhood of a circular hole in a strip under tension. Philosophical Transactions of the Royal Society of London A, 229, 49-86.



Stress concentration factors

R/W	Point A	Point B
	σ_{xx}/σ_0	σ_{yy}/σ_0
0	3.00	-1.00
0.1	3.03	-1.03
0.2	3.14	-1.11
0.3	3.36	-1.26
0.4	3.74	-1.44
0.5	4.32	-1.58

Figure 5: Plate of finite width.