
Engineering Analysis using the Finite Element Method

University of California, Berkeley, Spring 2019
 Instructor: Sanjay Govindjee
 GSI: Linus Mettler

Lab 1

The purpose of this lab is to introduce you to the commercial software Ansys, and more specifically the static structural simulation of a thin plate in Ansys Workbench. The procedure consists of the following steps:

- creating a geometry
- setting material properties
- generating a finite element mesh
- defining supports, loads and symmetry conditions
- determining solution parameters
- post-processing and visualizing the results

Please provide a graph (cf. below) to check off this lab. No report required.

1 Problem statement and exact solution

Consider an infinite plate with a circular hole of radius a , subjected to a constant far-field uniaxial tensile stress σ^∞ (see Fig. 2)¹. This is a classical problem in linear elasticity. A cylindrical coordinate system centered at the hole lends itself for the solution of the equilibrium equations. The exact solution for the components of the stress tensor is

$$\sigma_{rr} = \frac{\sigma^\infty}{2} \left(1 - \frac{a^2}{r^2}\right) + \frac{\sigma^\infty}{2} \left(1 - \frac{4a^2}{r^2} + \frac{3a^4}{r^4}\right) \cos(2\theta) \quad (1a)$$

$$\sigma_{\theta\theta} = \frac{\sigma^\infty}{2} \left(1 + \frac{a^2}{r^2}\right) - \frac{\sigma^\infty}{2} \left(1 + \frac{3a^4}{r^4}\right) \cos(2\theta) \quad (1b)$$

$$\sigma_{r\theta} = -\frac{\sigma^\infty}{2} \left(1 + \frac{2a^2}{r^2} - \frac{3a^4}{r^4}\right) \sin(2\theta) \quad (1c)$$

¹Source: L. Anand and S. Govindjee, Introduction to Mechanics of Solid Materials.

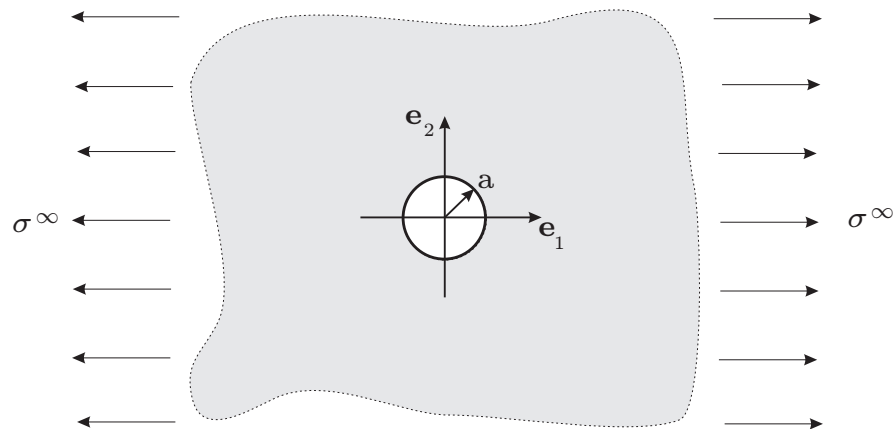


Figure 1: Infinite plate with hole of radius a under far-field uniaxial stress σ^∞ .

Confirm that the analytical solution (1) satisfies the boundary conditions, i.e. that the far-field stress is indeed uniaxial σ^∞ (hint: set $r \rightarrow \infty$) and that the surface of the hole is stress free (hint: check σ_{rr} at $r \rightarrow a$). Identify the location θ along the perimeter of the hole with the largest tensile hoop stress $\sigma_{\theta\theta}^{\max}$. Compute the so-called stress intensity factor

$$K_t = \frac{\sigma_{\theta\theta}^{\max}}{\sigma^\infty} \quad (2)$$

Does it depend on the material properties and why?

2 FE model

2.1 Geometry and material properties

Evidently, our FE model will consist of a *finite* plate whose stress field is expected approach the analytical solution if the hole diameter is small compared to the plate dimensions. Start your analysis by creating a *static structural* analysis in Ansys Workbench. Under the *geometry* block, open DesignModeler (via right-click) and sketch a plate of width $w = 50$ mm and length $l = 100$ mm with a hole of diameter $d = 10$ mm in the x-y-plane. Create a surface using *surface from sketches* and specify a thickness of $t = 1$ mm. For convenience, you may want to set the units of your model to mm. Return to Workbench and make sure to set the analysis type to 2D (select *view* \rightarrow *properties* to access the advanced geometry options).

Double-click on the *engineering data* block. Only one default material is available: structural steel. Inspect the material properties and note the range of different

constitutive models at our disposal. Create a material model for aluminium with a Young's modulus of $E = 70$ GPa and Poisson's ratio of $\nu = 0.35$.

2.2 FE mesh and loads

Double-click the *model* block. This will open Ansys Mechanical, the graphical interface needed for the remainder of our analysis. The tree on the left will guide you through the modeling process.

First, assign the material (aluminium) to your body. Set the 2D behavior to plane stress and verify that the plate thickness is set to $t = 1$ mm. In the *mesh* section, examine the different mesh and sizing options. Generate a mesh and play with the sizing tools - obtaining a good mesh is a key aspect of the modeling process as we'll learn later in the semester. How many elements and nodes does your model comprise (cf. *mesh* \rightarrow *statistics*)? How many degrees of freedom do they represent?

In the *static structural* section define the appropriate loads and supports. Make sure that your body is sufficiently supported².

2.3 Solution

Inspect the various solution options such as the time stepping. Even though the concept of "time" is irrelevant in a static analysis, these are very useful for non-linear problems such as large deformations, plasticity or contact problems. Once satisfied with your settings hit *solve* to compute the solution.

Plot the von-Mises equivalent stress, the total deformation and the circumferential stress component. Hint: for the circumferential stress we first need to create a *cylindrical* coordinate system at the center of the hole. We are then able to plot the directional stress components with respect to that coordinate system.

Compare the resulting stress concentration factor with the analytical solution. HERE Examine how the result is affected by the mesh size.

2.4 Symmetry

The computational cost is directly related to the number of degrees of freedom of your model. You may thus want to reduce your model size by exploiting symmetries.

²Otherwise large rigid-body motions occur. In effect that means the FE equations are singular and thus the solution cannot be determined. Note that Ansys has a built-in "weak springs" feature to prevent this from happening, even when the model is insufficiently supported.

Identify all symmetry planes of your model³. Go back to DesignModeler and modify the geometry accordingly. Return to Ansys Mechanical to introduce symmetry conditions (click on *model* on the tree and look for the symmetry icon) and modify the loads and supports. Are any supports necessary for the model to be sufficiently supported?

Compute the solution and compare the result with your previous analysis.

2.5 Stress concentration vs. plate width

Play with the width w of the plate and examine how the stress concentration factor K_t varies with w/a (keeping the radius a constant). For these types of analyses it is often advantageous to parametrize the input and output parameters in Ansys (this is achieved by ticking the appropriate box where the dimensions are specified - I'll show this during the lab).

Plot K_t vs. w/a in Excel or Matlab for a range of values w/a .

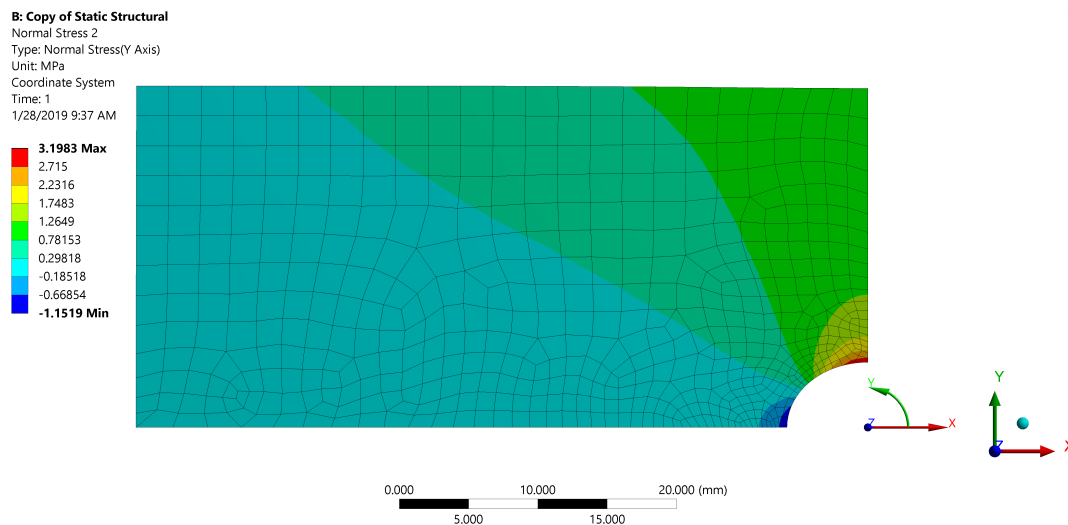


Figure 2: Hoop stress $\sigma_{\theta\theta}$ computed with a FE model exploiting symmetry conditions.

³We require not only that the geometry is symmetric, but also the material properties and external loads. If a model is symmetric, it is sufficient to analyze only half of the body.