

ANSYS - Orthotropic plate with a hole - Problem Specification

Author: Oswaldo Rodriguez, Cornell University

Problem Specification

1. [Create Command Log file](#)
2. [Modify Log file - part1](#)
3. [Modify Log file - part2](#)
4. [Solve](#)
5. [Postprocess the results](#)

Problem Specification

Consider the square plate of uniform thickness with a circular hole with dimensions shown in the figure below. The plate is uniaxially loaded with a uniform pressure $p=1$ MPa. In addition, the plate is made of a Glass/Epoxy composite material with the fibers oriented in same direction as the applied load. The material properties are as follows:

Young's modulus in the fiber direction $E_x = 59.3$ GPa

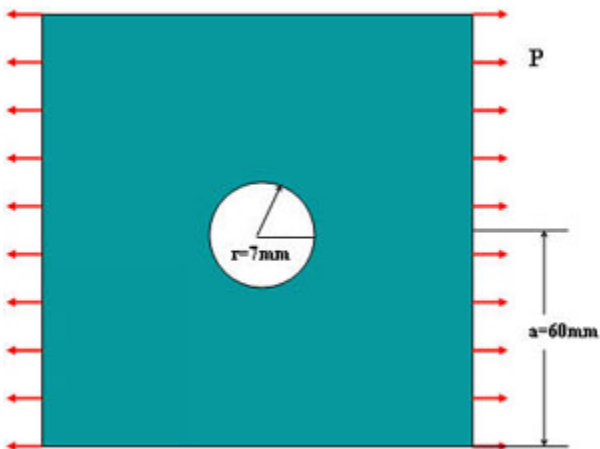
Young's modulus in the transverse direction $E_y = 22$ GPa

In-plane shear modulus $G_{xy} = 8.96$ GPa

Major Poisson's ratio $\nu_{xy} = 0.26$

Minor Poisson's ratio $\nu_{yx} = 0.047$

The circumferential stress concentration on the boundary of the hole is to be determined using ANSYS.



Go to [Step 1: Create Command Log file](#)

Go to [all ANSYS Learning Modules](#)