Universidad de los Andes | FACULTAD DE INGENIERÍA Y CIENCIAS APLICADAS

Finite Elements - IOC5107
Final Report

Profesor:
Jos+e Antonio Abell
Ayudante:
Nicolás
Alumnos:
Felipe Vicencio
Lukas Wolff

20 de abril de 2025

1. Introduction

Finite Element Analysis (FEA) is a numerical method for predicting the response of structures under loading conditions. By discretizing a continuous domain into smaller elements, the method allows to compute distributions of stress, strain, and displacement.

This report presents the structural analysis of a 3D-printed wrench using 2D triangular constant strain triangle (CST) elements under the plane stress assumption. The main objective is to investigate the mechanical behavior of the wrench under different loading scenarios, focusing on its deformation and internal stress/strain states.

Three cases are considered: (a) a total vertical load of 30 kgf applied as a distributed force, (b) the same load applied at a single node, and (c) a distributed load including the effect of self-weight. Each case includes graphs of the deformed shape, stress and strain components, principal values, and a discussion of mesh convergence and precision techniques.

The analysis also includes an optimization study to minimize the maximum principal tensile stress while maintaining constant material volume. The analysis was perfomed using a *Python* program that implementes this methodology. For the mesh generation and optimization, the GMESH *software* was used.