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

24 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.

2. Theoretical background

2.1. Finite Element Method (FEM)

The Finite Element Method (FEM) is a numerical technique used to solve complex engineering problems. It involves dividing a large system into smaller and simpler parts called finite elements. The behavior of each element is described by a set of equations, and the overall response of the system is obtained by assembling these equations. The method used in this report is the CST based model.

2.2. CST elements

The CST (Constant Strain Triangle) is a 2D, three-node finite element used for plane stress/strain problems. It assumes linear displacement fields, producing constant strain and stress within each element. The CST element is defined by three nodes, each with two degrees of freedom (DOF) corresponding to the in-plane displacements. The element stiffness matrix is derived from the shape functions and material properties, allowing for the calculation of nodal displacements and internal forces.

2.3. Plane stress assumption

This condition applies to thin structures where the stress components perpendicular to the plane are negligible. Mathematically, this means $\sigma_z = \tau_{xz} = \tau_{yz} = 0$. The dominant stress components are σ_{xx}, σ_{yy}, and σ_{xy}. This assumption is typically valid in thin plates or sheets subjected to in-plane loading and is commonly used in finite element analysis (FEA) with 2D CST elements under plane stress conditions.

2.4. Plane strain assumption

This assumption is used for long or thick bodies where deformation in one direction (usually the z-axis) is constrained. In this case, the strain components are $\varepsilon_z = \gamma_{xz} = \gamma_{yz} = 0$, although σ_z can be non-zero. This is applicable to structures such as tunnels, dams, and thick-walled pressure vessels. All in-plane stress components remain active, and a different constitutive matrix **D** is used compared to the plane stress condition.

3. Materials for the model

For this asignment, a wrech was design using the GMESH *software*. The wrench was designed to be 3D printed, and the material used was PLA.

This material is a thermoplastic polymer that is widley used in 3D printing due to its mechanical properties. Moreover, depending on the printing orientation, fill density and extrusion temperature, the mechanical properties of PLA can vary significantly. For this analysis, the following properties were used:

- Young's modulus: $3{,}5 \times 10^9 Pa$
- Poisson's ratio: 0.36
- Tensile Yield strength: $50 \times 10^6 Pa$
- Density: $1{,}252 \times 10^3 kg/m^3$

These values were chosen based on the standard levels used in other experiences. CITAR PAPER

4. Convergencia del modelo

Graficos de referencia inicial:

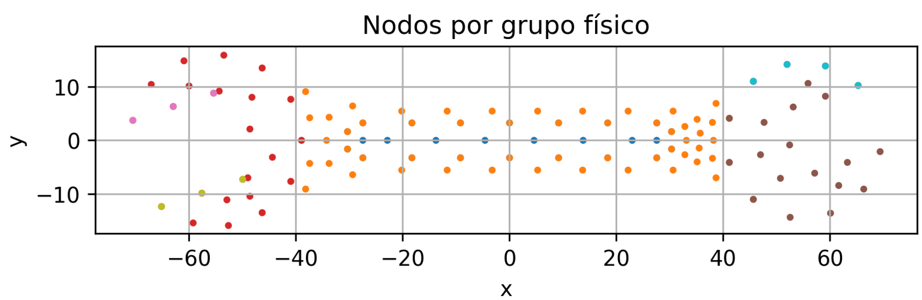


Figura 1: Caption

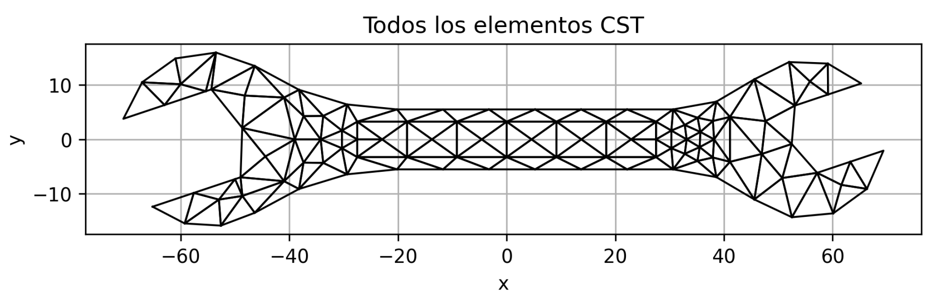


Figura 2: Caption

Luego se genera un codigo para hacer la convergencia

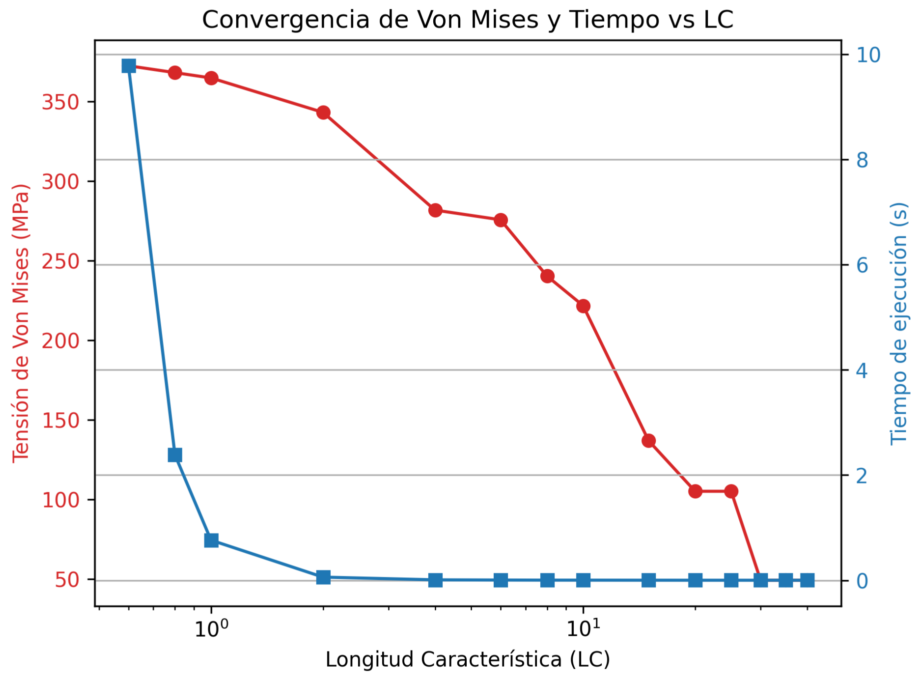


Figura 3: Caption

Se concluye que un lc = 0.8 es optimo balanceando el tiempo de computo y la precision del modelo.

De esta forma, el modelo se ve como:

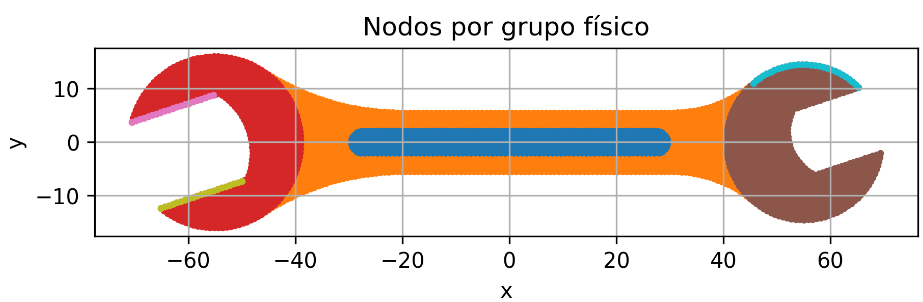


Figura 4: Caption

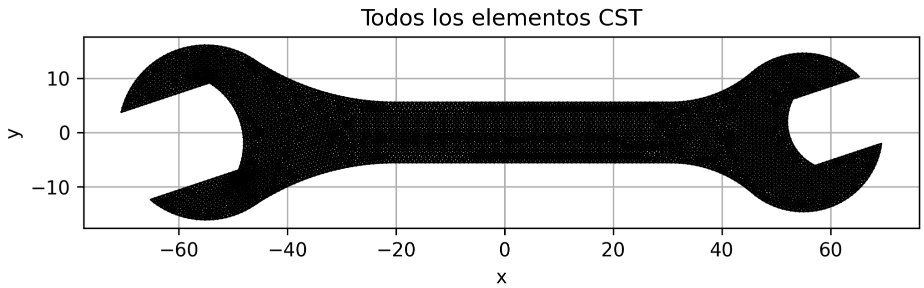


Figura 5: Caption

5. A case

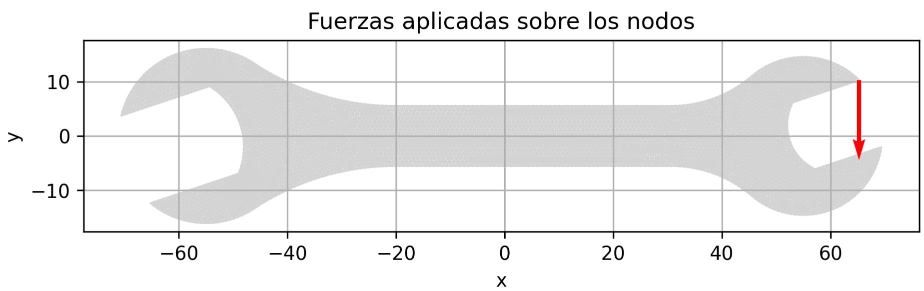


Figura 6: Caption

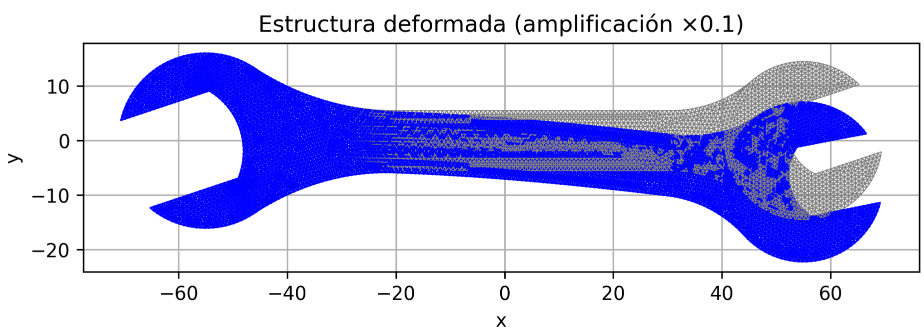


Figura 7: Caption

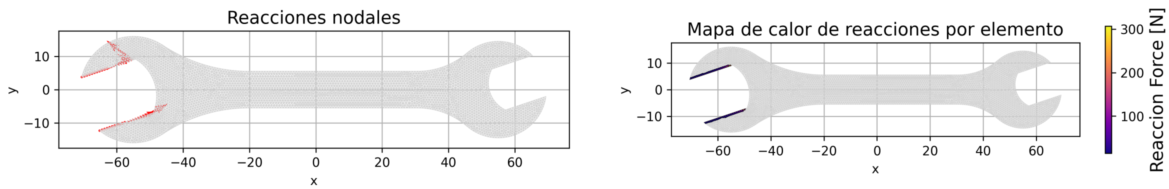


Figura 8: Caption

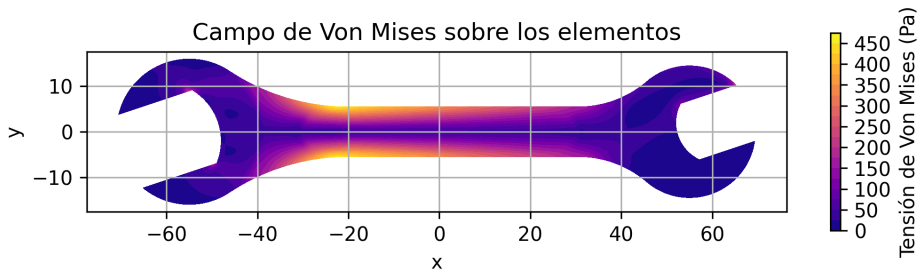
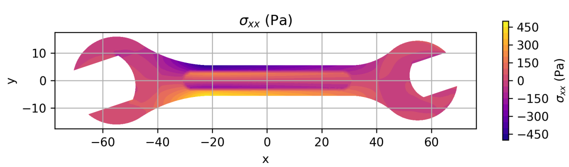
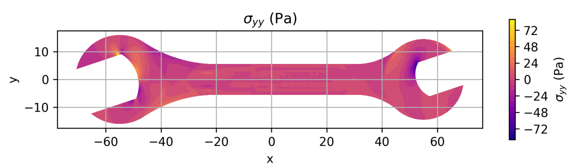


Figura 9: Caption

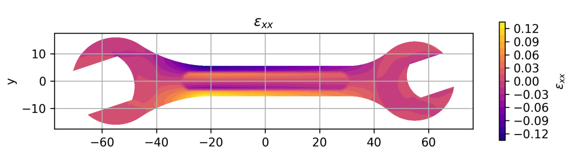


(a) Caption

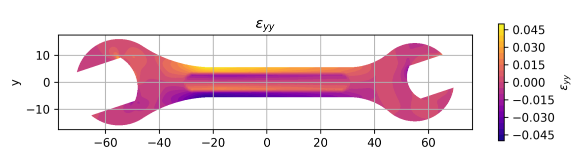


(b) Caption

Figura 10: Caption

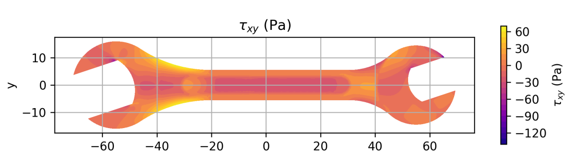


(a) Caption

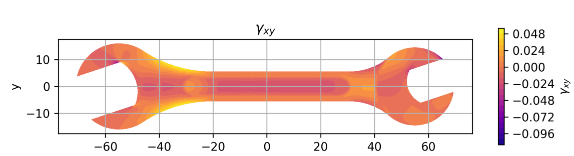


(b) Caption

Figura 11: Caption

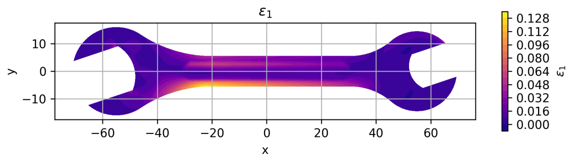


(a) Caption

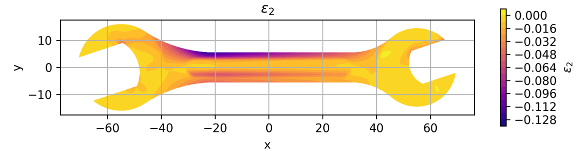


(b) Caption

Figura 12: Caption

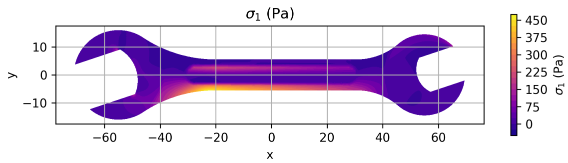


(a) Caption

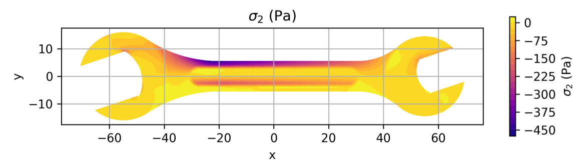


(b) Caption

Figura 13: Caption



(a) Caption



(b) Caption

Figura 14: Caption

6. B case

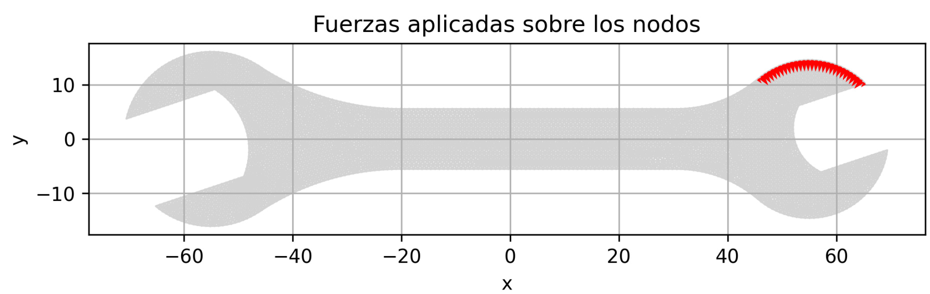


Figura 15: Caption

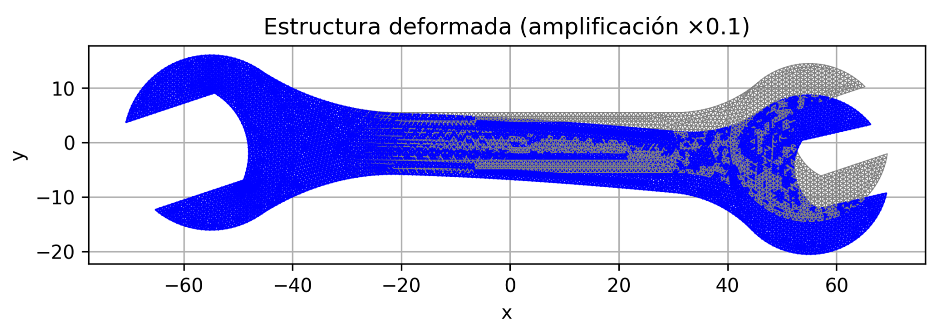


Figura 16: Caption

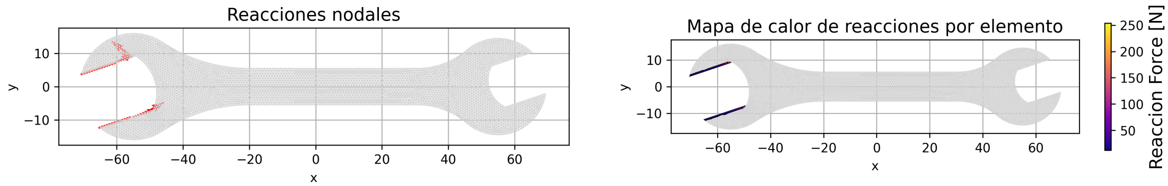


Figura 17: Caption

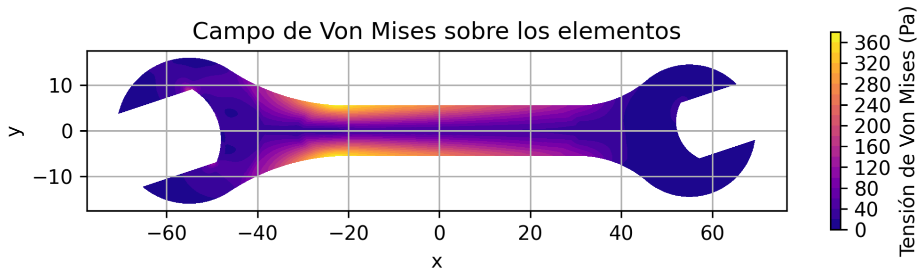
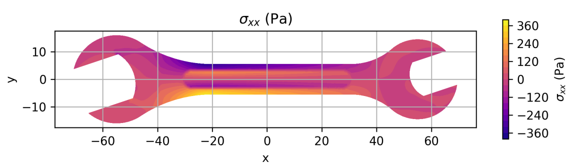
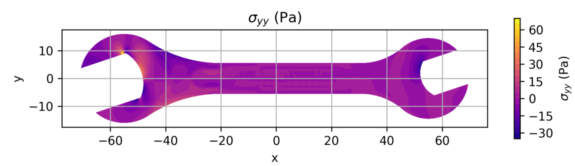


Figura 18: Caption

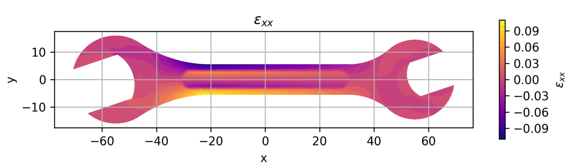


(a) Caption

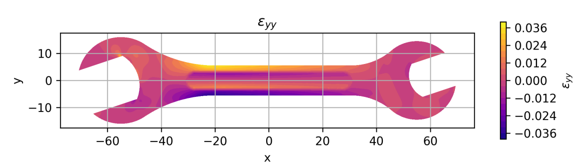


(b) Caption

Figura 19: Caption

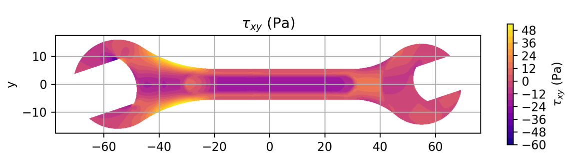


(a) Caption

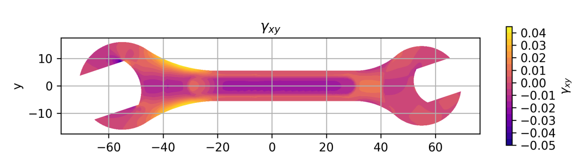


(b) Caption

Figura 20: Caption

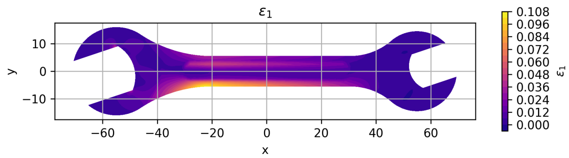


(a) Caption

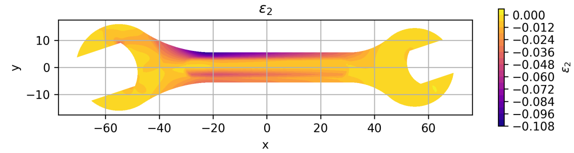


(b) Caption

Figura 21: Caption

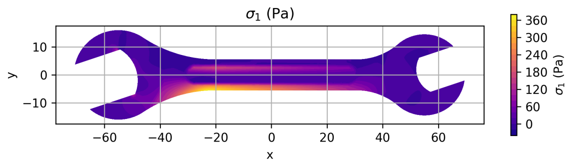


(a) Caption

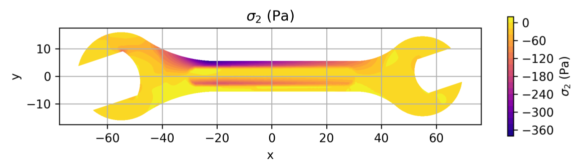


(b) Caption

Figura 22: Caption



(a) Caption



(b) Caption

Figura 23: Caption

7. C case

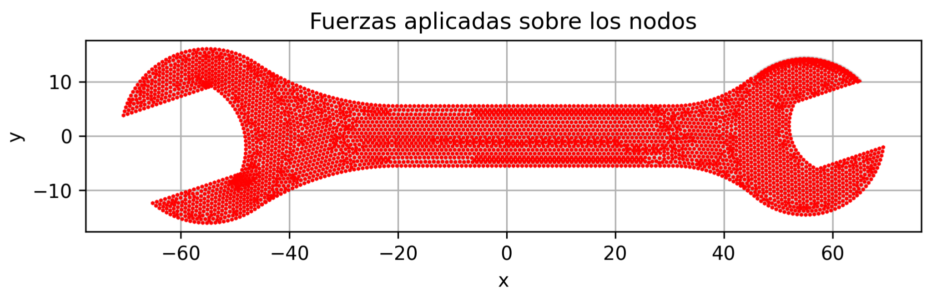


Figura 24: Caption

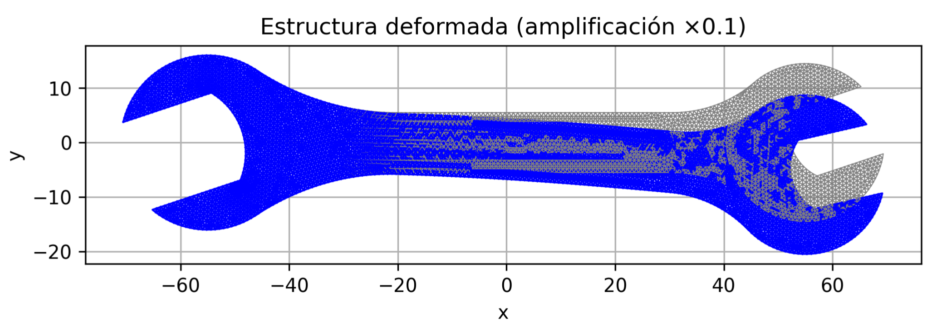


Figura 25: Caption

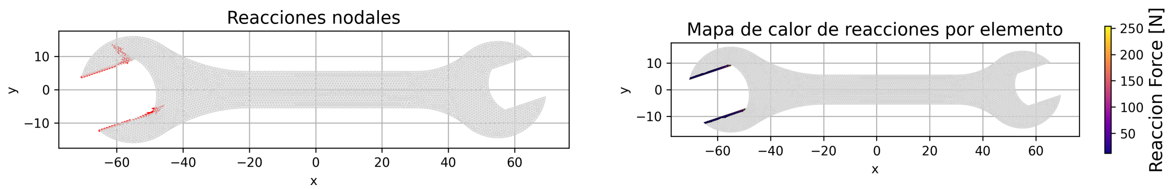


Figura 26: Caption

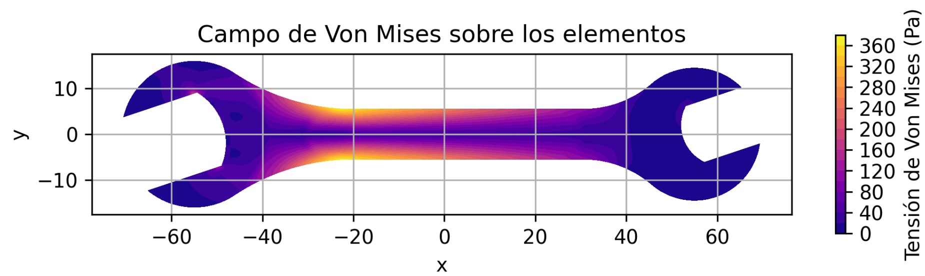
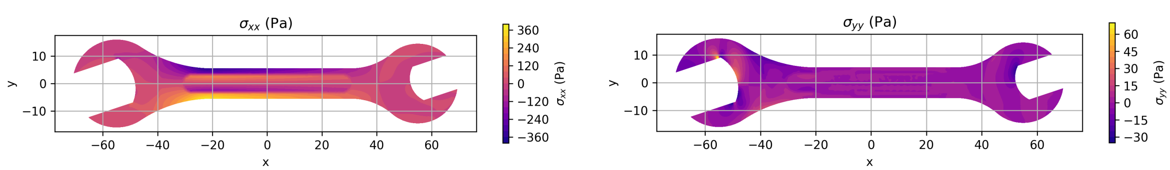


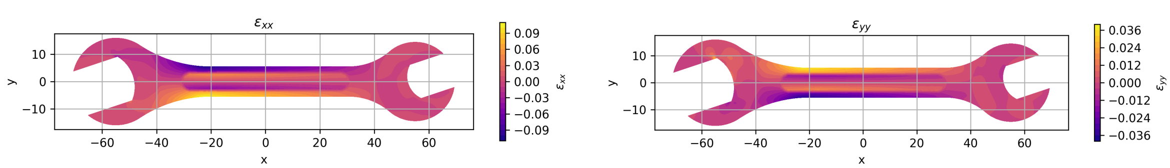
Figura 27: Caption



(a) Caption

(b) Caption

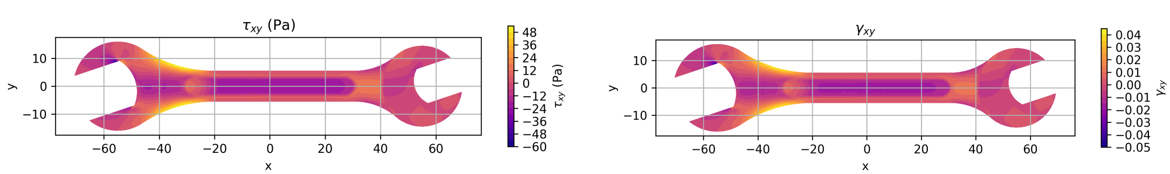
Figura 28: Caption



(a) Caption

(b) Caption

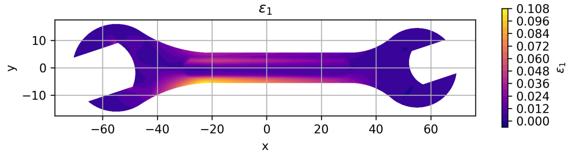
Figura 29: Caption



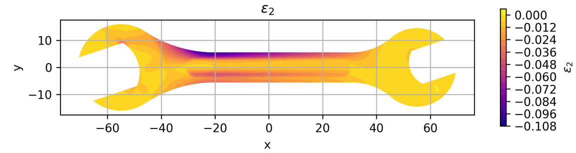
(a) Caption

(b) Caption

Figura 30: Caption

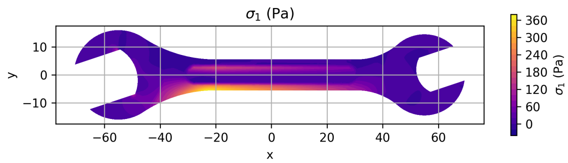


(a) Caption

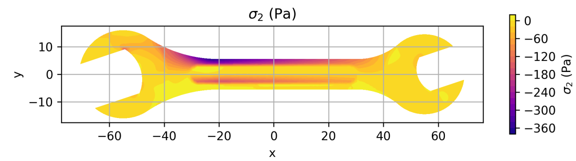


(b) Caption

Figura 31: Caption



(a) Caption



(b) Caption

Figura 32: Caption

8. D case, only self weight

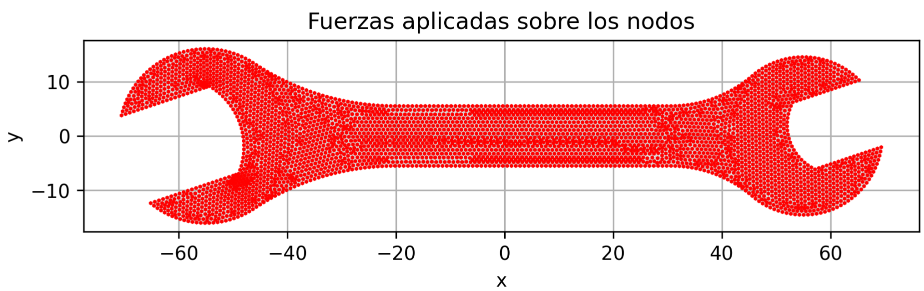


Figura 33: Caption

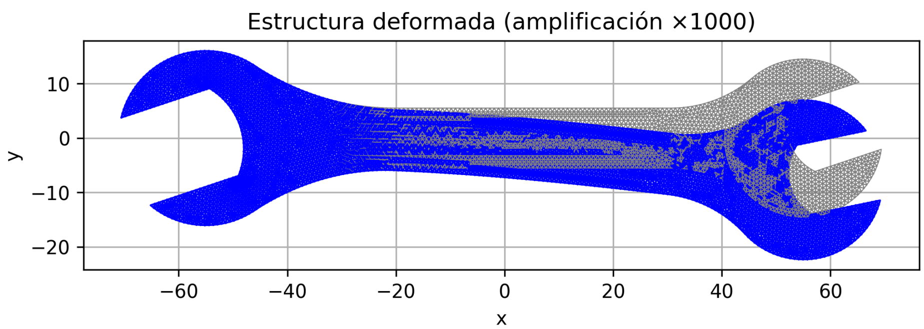


Figura 34: Caption

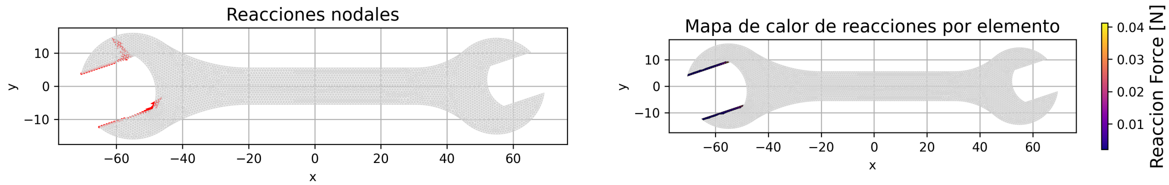


Figura 35: Caption

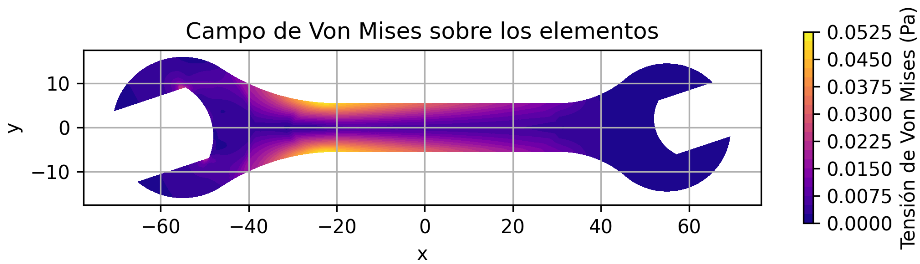
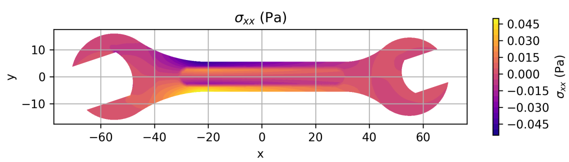
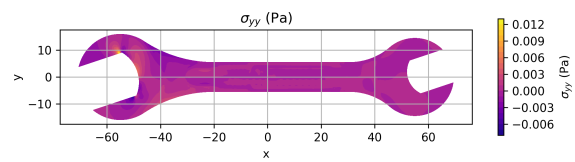


Figura 36: Caption

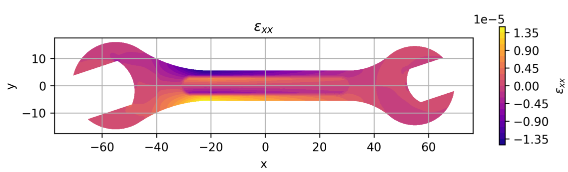


(a) Caption

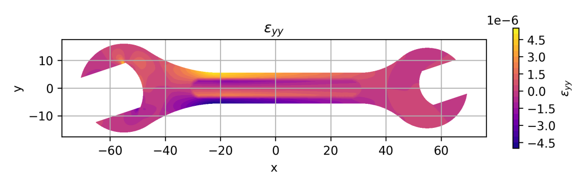


(b) Caption

Figura 37: Caption

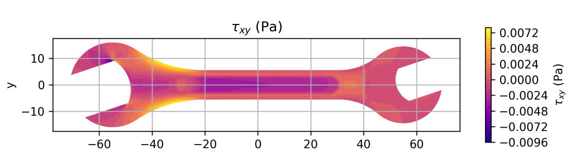


(a) Caption

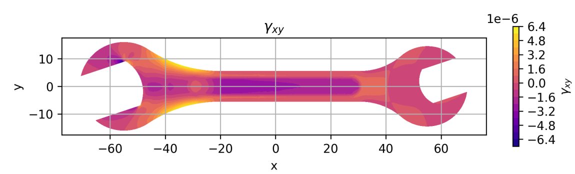


(b) Caption

Figura 38: Caption

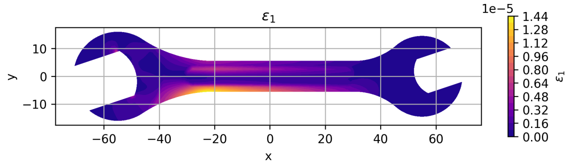


(a) Caption

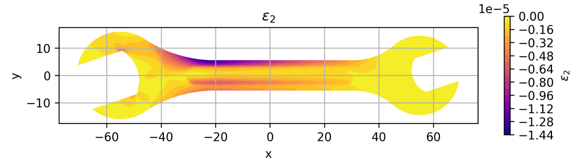


(b) Caption

Figura 39: Caption

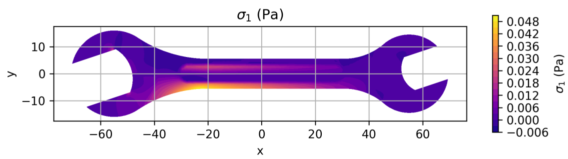


(a) Caption

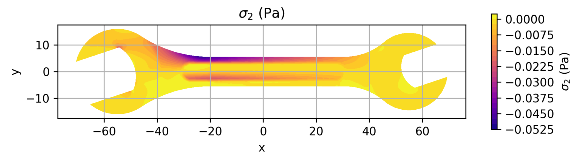


(b) Caption

Figura 40: Caption



(a) Caption



(b) Caption

Figura 41: Caption

9. Topologic Optimization

En base a un elemento con tamaño de malla 5 se pueden analizar los esfuerzos de von mises:

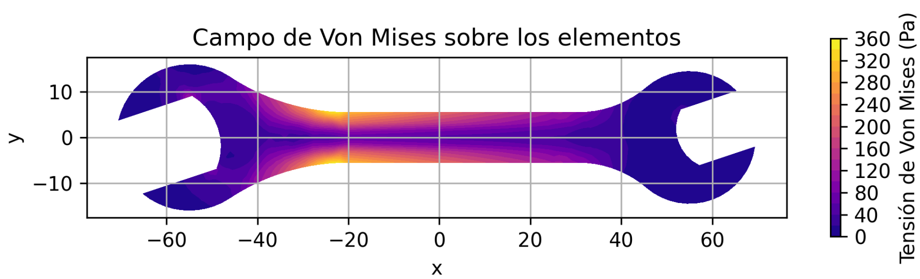


Figura 42: Caption

Luego se pueden modificar los espesores segun los esfuerzos maximo y minimos

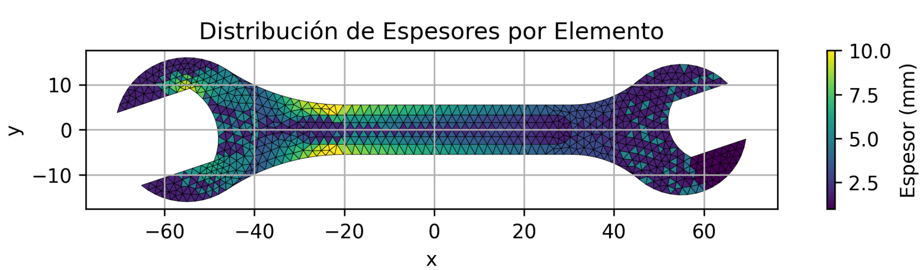


Figura 43: Caption

Y asi se optimizan los esfuerzos a lo largo del elemento

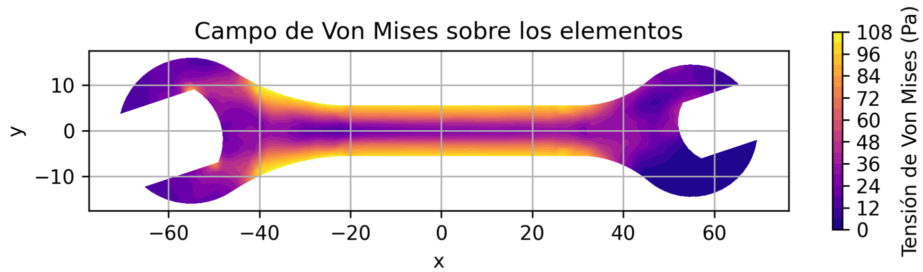


Figura 44: Caption