Universidad de los Andes

FACULTAD DE INGENIERÍA Y CIENCIAS APLICADAS

Finite Elements - IOC5107
Final Report

Profesor:
José Antonio Abell
Ayudante:
Nicolás
Alumnos:
Felipe Vicencio
Lukas Wolff

27 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.

2.5. Shape functions

The shape functions are used to interpolate the displacements within the element based on the nodal displacements. For a CST element, the shape functions are defined as:

$$N_i(\xi) = \begin{bmatrix} \xi_1 & 0 & \xi_2 & 0 & \xi_3 & 0 \\ 0 & \xi_1 & 0 & \xi_2 & 0 & \xi_3 \end{bmatrix} \tag{1}$$

Where ξ_1, ξ_2, and ξ_3 are the coordinates of the nodes in the local coordinate system. The shape functions are used to relate the nodal displacements to the displacements at any point within the element. The shape functions are linear and can be expressed in terms of the local coordinates ξ and η as follows:

$$N_1 = \frac{1}{2}(1 - \xi - \eta), \quad N_2 = \frac{1}{2}(1 + \xi), \quad N_3 = \frac{1}{2}(1 + \eta) \tag{2}$$

2.6. Strain displacement matrix

The strain displacement matrix relates the nodal displacements to the interal deformation of the element. For a CST element, the strain-displacement matrix **B** is defined as:

$$B = \begin{bmatrix} \frac{\partial \xi_1}{\partial x} & 0 & \frac{\partial \xi_2}{\partial x} & 0 & \frac{\partial \xi_3}{\partial x} & 0 \\ 0 & \frac{\partial \xi_1}{\partial y} & 0 & \frac{\partial \xi_2}{\partial y} & 0 & \frac{\partial \xi_3}{\partial y} \\ \frac{\partial \xi_1}{\partial y} & \frac{\partial \xi_1}{\partial x} & \frac{\partial \xi_2}{\partial y} & \frac{\partial \xi_2}{\partial x} & \frac{\partial \xi_3}{\partial y} & \frac{\partial \xi_3}{\partial x} \end{bmatrix} \tag{3}$$

This matrix derives the shape functions in order to obtain the internal strain components as follows:

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} = \mathbf{B} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix} \tag{4}$$

Where ε_x and ε_y are the normal strains in the x and y directions, respectively, and γ_{xy} is the shear strain. The vector of nodal displacements is represented by u_1, u_2, and u_3.

2.7. Stiffness matrix

The stiffness matrix is used to calculate the internal forces and displacements ot the element from the nodal displacements.

$$K^{(e)} = \int_{\Omega^{(e)}} B^T E B t \, d\Omega^{(e)} \tag{5}$$

Where $K^{(e)}$ is the element stiffness matrix, E is the material's elastic modulus, t is the thickness of the element, and $\Omega^{(e)}$ is the area of the element. The stiffness matrix relates the nodal forces to the nodal displacements as follows:

$$\mathbf{F} = \mathbf{Ku} \tag{6}$$

2.8. Nodal forces vector

To obtain the nodal forces vector, a body force is applied (self-weight), then the nodal forces vector is calculated as follows:

$$f^{(e)} = \int_{\Omega^{(e)}} t \left(N^{(e)} \right)^T b \, d\Omega^{(e)} \tag{7}$$

Where $f^{(e)}$ is the nodal forces vector, $N^{(e)}$ is the shape function matrix, and b is the body force vector. The nodal forces vector relates the body forces to the nodal displacements as follows:

$$\mathbf{F} = \mathbf{Ku} + \mathbf{f} \tag{8}$$

2.9.   Von Mises stress Criterion

In structural analysis, the Von Mises stress is widley used as a yield and failure crtierion. The Von Mises stress is a scalar value derived from the stress components and provides a singles equivalent stress that can be directly compared to the material's yield strenght. Under plane stress conditions, the Von Mises stress σ_{VM} is calcultaed using the in-plane stresses as follows:

$$\sigma_{VM} = \sqrt{\sigma_{xx}^2 - \sigma_{xx}\sigma_{yy} + \sigma_{yy}^2 + 3\sigma_{xy}^2} \tag{9}$$

Where σ_{xx} and σ_{yy} are the normal stresses in the x and y directions, respectively, and τ_{xy} is the shear stress in the xy plane.

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. Description of the model

The model for this assignment was a wrench designed using the GMESH *software*. The purpose of this was to analyze the mechanical behavior of the structure under 3 different cases.

The first case was a distributed load applied to one end of the wrench, as shown in Figure 1.

The second case was a point load applied to the same end of the first case. The variation from this case to the previous one is considerable, due to the force being adjusted to only one node instead to a list of nodes.

At last, the third case was similar to case 1, but with the addition of a self-weight load. This was done by adding a body force to the model in that same region.
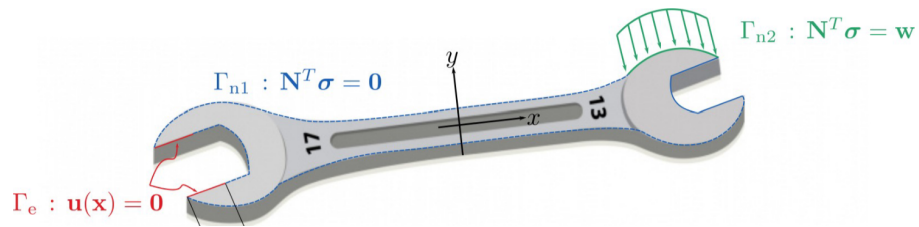
Figure 1: Wrench model with limited boundaries and regions of particular behavior.

5.   Results and Analysis

This section presents the results obtained from the simulation of the proposed algorithm. The objective of this assignment was to evaluate and analyze the wrench behavior under 3 different cases presented below. It is worth noting that for the results to be precise and coherent, a convergence study was performed. This process ensure the accurasy and physical validity of the reuslts, resulting in a mesh size of about $l_c = 0.8$.

5.1.   Case 1: Distributed load apllied to one end of the wrench

For this case, a $30kgf$ distributed load was applied at the wrench deforming it in the y direction. This can be observed in Figure 3.
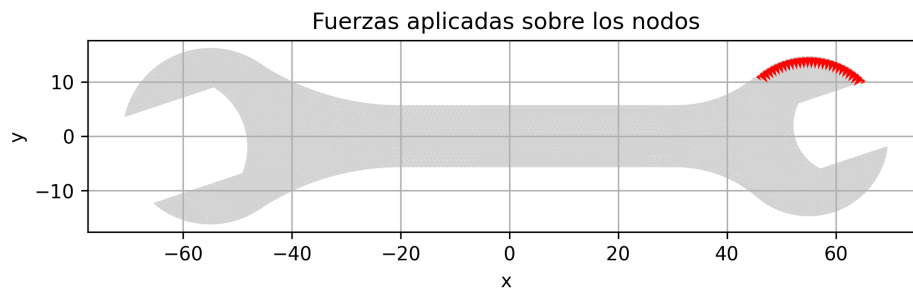
Figure 2: Distributed load applied to the wrench
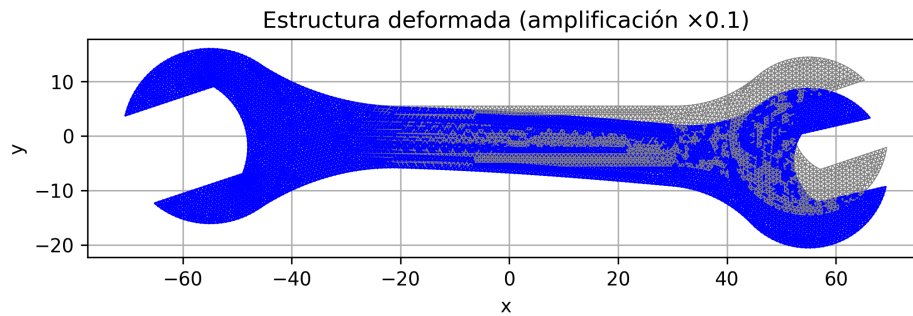
Figure 3: Deformed wrench with distributed load

The results of normal stresses, shear stresses, and displacements are shown below, as well as the principal stresses and strains.

Table 1: Stress (Pa) and strain components

σ_{xx}	σ_{yy}	τ_{xy}	ε_{xx}	ε_{yy}	γ_{xy}
360	60	-60	0.09	0.036	-0.05

Table 2: Principal Stresses (Pa) and strain components

σ_1	σ_2	ε_1	ε_2
360	-360	0.108	-0.108

Finally, the Von Misses stress is plotted below, reaching a maximum value of $\sigma_{VM} = 360Pa$.
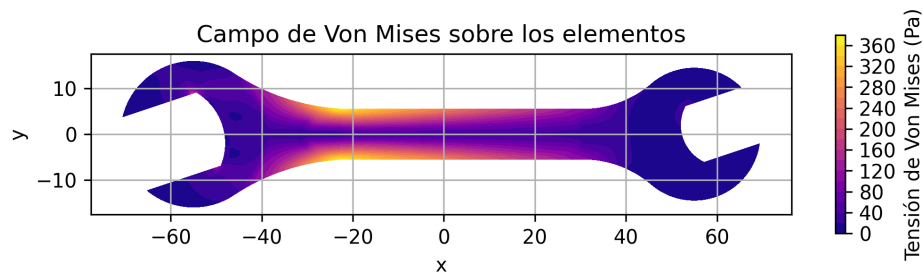

Figure 4: Von Misses stress distribution for the distributed load

5.2. Case 2: Point load applied to one end of the wrench

Different to the previous case, a point load of $30kgf$ was applied to one end of the wrench. This can be observed in Figure 5. Also, the deformed state is shown in Figure 6.
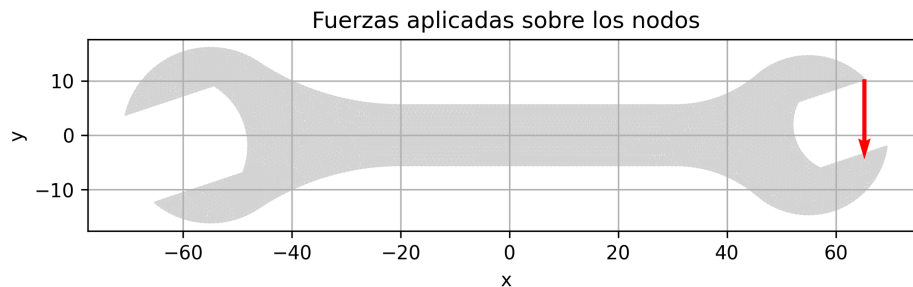
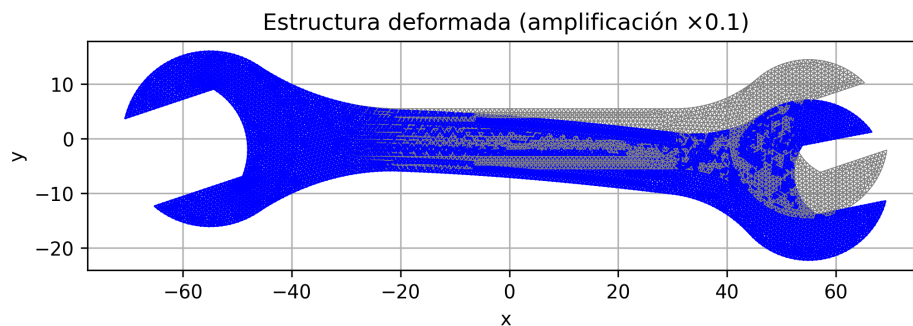Figure 5: Point load applied to the wrench

Figure 6: Deformed wrench with point load

In the following tables, the stress and strain components are shown.

Table 3: Stress (Pa) and strain components

σ_{xx}	σ_{yy}	τ_{xy}	ε_{xx}	ε_{yy}	γ_{xy}
450	72	-120	0.12	0.045	-0.096

Table 4: Principal Stresses (Pa) and strain components

σ_1	σ_2	ε_1	ε_2
450	-450	0.128	-0.128

It is noticeable that, compared to the previous case, the principal strains are significally higher. This is due to the fact that considering the type of material and its mechanical properties, the force magnitude and single point of application increases the stress concentration throughout the wrench. This is also reflected in the Von Misses stress, which reaches a maximum value of $\sigma_{VM} = 450Pa$.
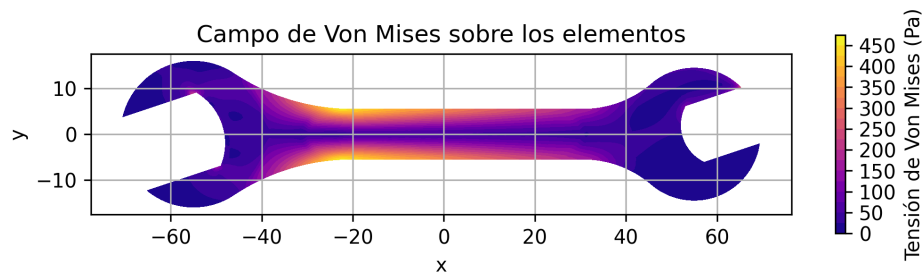

Figure 7: Von Misses stress distribution for the point load

5.3. Case 3: Self-weight added to the distributed load

Now, a similar case to the first one was analyzed, but this time the self-weight was added as a body force ($m_{wrench} = 8gr$). The results are shown below.
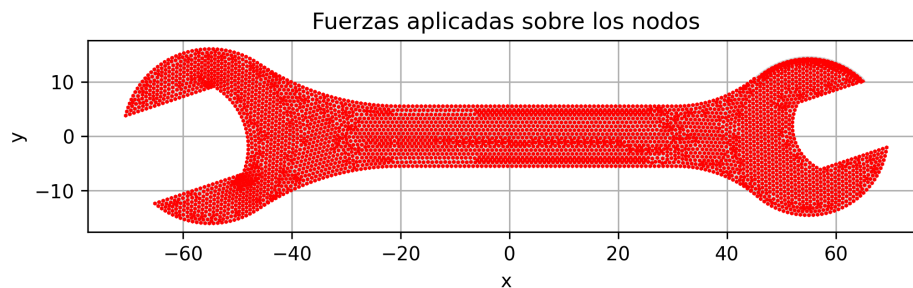

Figure 8: Caption

We can observe in Figure 8 that the vector of the self-weight is acting through the entire body of the wrench, more specifically, in the mass centroid of each CST element. The deformed state is shown in Figure 9.
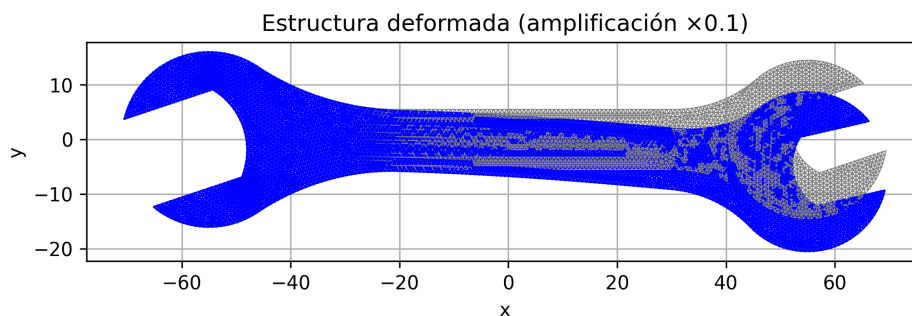
Figure 9: Caption

Table 5: Stress (Pa) and strain components

σ_{xx}	σ_{yy}	τ_{xy}	ε_{xx}	ε_{yy}	γ_{xy}
360	60	-60	0.09	0.036	-0.05

Table 6: Principal Stresses (Pa) and strain components

σ_1	σ_2	ε_1	ε_2
360	-360	0.108	-0.108

Comparing this scenario with the one presented in Case 1, the results are the same. This is because the self-weight of the wrench is negligible compared to the distributed load of $30kgf$. Therefore, the internal stresses and strains are not affectedd by it.

5.4. Case 4: Only self-weight applied as a body force

The decision to analyze this singular case was made in order to observe and compere how this body force condition affected the wrench. This way, we can observe that, due to the conditions of restrcited nodes in one end, the wrench still deforms, but in a much smaller magnitude.
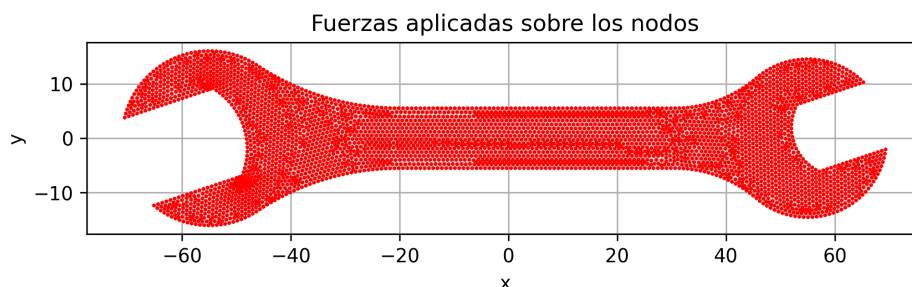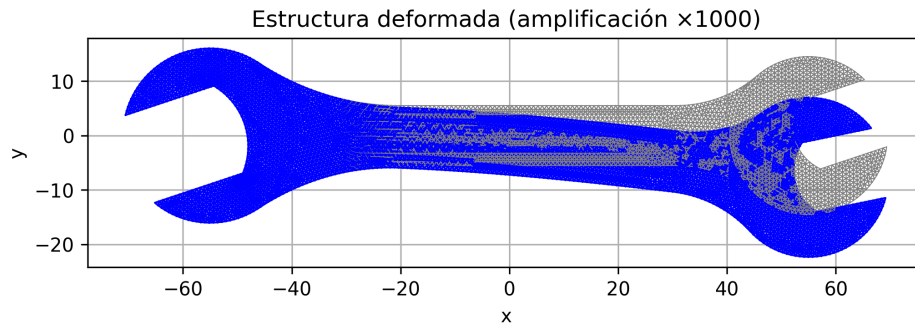

Figure 10: Self-weight applied as a body force

Figure 11: Deformed wrench with self-weight

Notice the amplification factor of the deformed state, which is 10^3. Which is a clear indication of the small magnitude of the displacements compared to the previous 3 cases. The stress and strain components are shown below.

Table 7: Stress (Pa) and strain components

σ_{xx}	σ_{yy}	τ_{xy}	ε_{xx}	ε_{yy}	γ_{xy}
0.045	0.012	-0.0096	$1{,}35 \times 10^{-5}$	$4{,}5 \times 10^{-6}$	$6{,}4 \times 10^{-6}$

Table 8: Principal Stresses (Pa) and strain components

σ_1	σ_2	ε_1	ε_2
0.048	-0.0525	$1{,}44 \times 10^{-5}$	$-1{,}44 \times 10^{-5}$

Observing the plot of the Von Misses stresses, we can see that the maximum value is $\sigma_{VM} = 0{,}0525 Pa$. This is a clear indication of the low impact of the self-weight on the wrench.
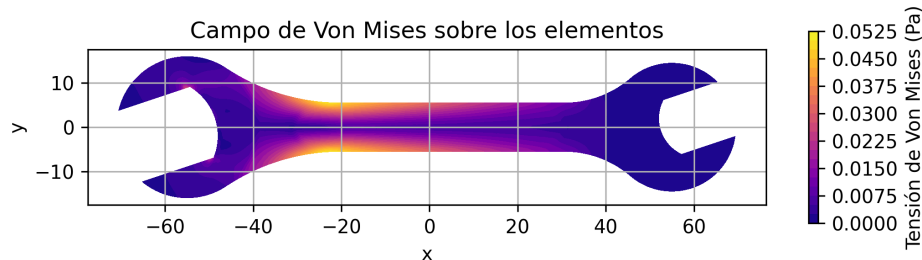

Figure 12: Caption

Finally, the simulation shows the clear mechanical behavior of the PLA wrench under different loading conditions. Where the CST model was able to capture accuratly and with precision the internal stresses and strains of the element. The results were coherent with the physical behavior of it, and the Von Misses stress was able to capture the stress concentration in the areas of interest. The results were also consistent with the expected behavior of a wrench under different loading conditions.

6. Preguntas

6.1. How do you know convergence in mesh size has been achieved?

Mesh size convergence is achieved when the solution of the problem does not change significantly with a further refinement of the mesh. This can be checked by comparing the results of the displacements, Von Mises stresses and principal stresses compared to other mesh sizes. In this case, convergence was assessed by plotting the maximum Von misses Stress with the characteristic lenght, and observing the time of processing in each case. The result shows that at an L_c of 0.8, the mesh was already converging.
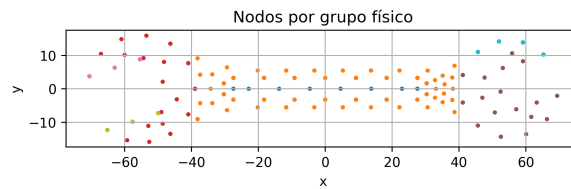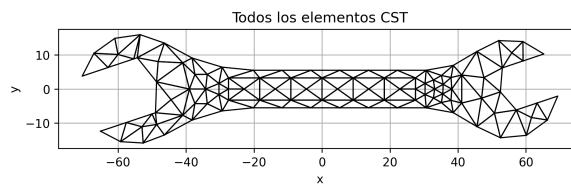

Figure 13: Caption

Figure 14: Caption

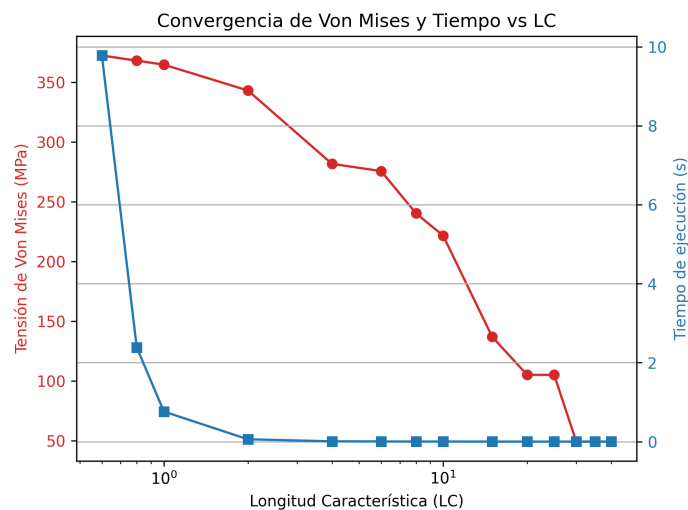Then, a code was programmed to plot the convergence of the mesh depending on the time of ejecution.

Figure 15: Caption

The result was an $L_c = 0,8$, balancing the time of computation and the precision of the model.

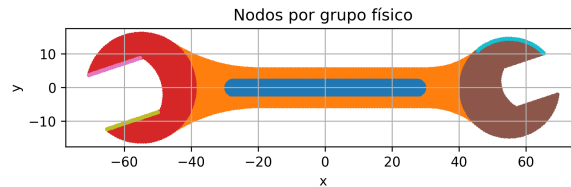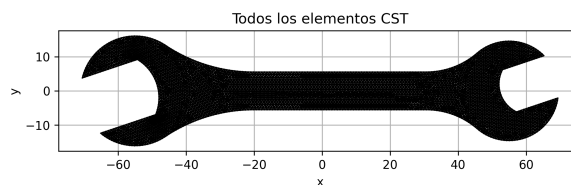Therefore, the final model is plotted as follows:

Figure 16: Caption

Figure 17: Caption

6.2. How do the stress/strain fields look like before and after smoothing?

Before smoothing, the stress and strain fields shows abrubt jumps between adjacent elements, producing a non-physical behaviour and dicontinuous fields. It is expected due to the implementation of CST elements, which have a constant stress and strain state. After applying post-processing smoothing, the fields becmoe continuous, whit better transitions across the domain, making it physically accurate.

6.3. What differences do you observe when applying the load as a distributed load versus a point load?

As a distributed load results in smoother and more realistic stress fiels, as the force is spread over an area, avoiding axcessive stress concentrations. On the other hand, a point load produces localized hight-stress peaks near the application point, leading to unrealistic stress values, unless the mesh is highly refined.

6.4. Is it necessary to consider self-weight for an accurate analysis of the tool?

In this case, it is not necessary to consider it, because the external forces applied are significantly grater than de self-weight of the PLA material. However, numercial comparisons showed that including self-weight, slightly increases the overall stress and displacement values, especially un ling or thin sections.

6.5. Research or provide a reasonable stress-based failure criteria for PLA. For this loading condition, and based on your failure criteria ¿where do you expect the wrench to fail?

According to CITAR PAPER, PLA (Polylactic Acid) exhibits a tensile strength near 50 MPa, depending on molecular weight, crystallinity, and processing method. Additionally, the usual Young's module is about 3.5 GPa,

and the material has a relatively low elongation at break (about 4-7 %), confirming its brittle nature.

Therefore, a reasonable stress-based failure criteria for PLA under mechanical loading is the Von Mises yield criterion, assuming brittle fracture at an equivalent stress level of around 50 MPa. Given PLA's limited ductility and tendency to fail without significant plastic deformation, reaching the ultimate tensile strength would iniciate cracking or brittle rupture.

As a result, based on the FEM simulations and stress distribution results, the highest Von Mises stresses concentrated at the junction between the wrench handle and the wrench head would be an expected failure location, in other words, near sharp geometrical transitions or areas under bending moments.