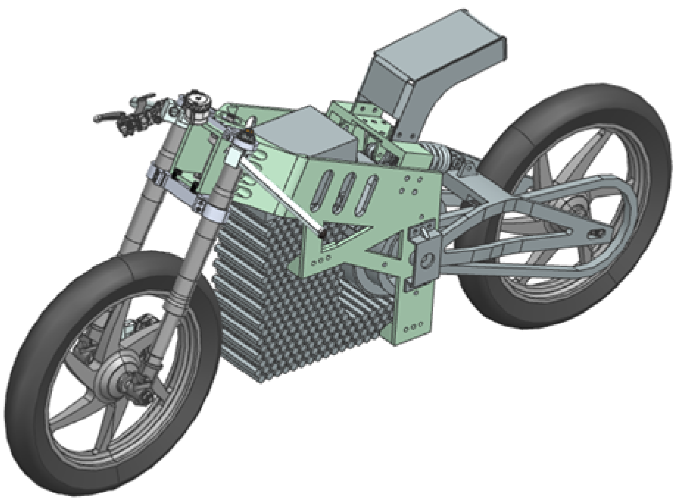
Analysis of the simulation of a motorbike chassis



Computer Assisted Structural Analysis

Students: Unai Álvarez de Eulate Llano, Furkan Aydin, Juan Pablo Maroto Herrera and Maider Ozaeta Fernández de Romarategui

Professor: Ana Isabel Boyano Murillo

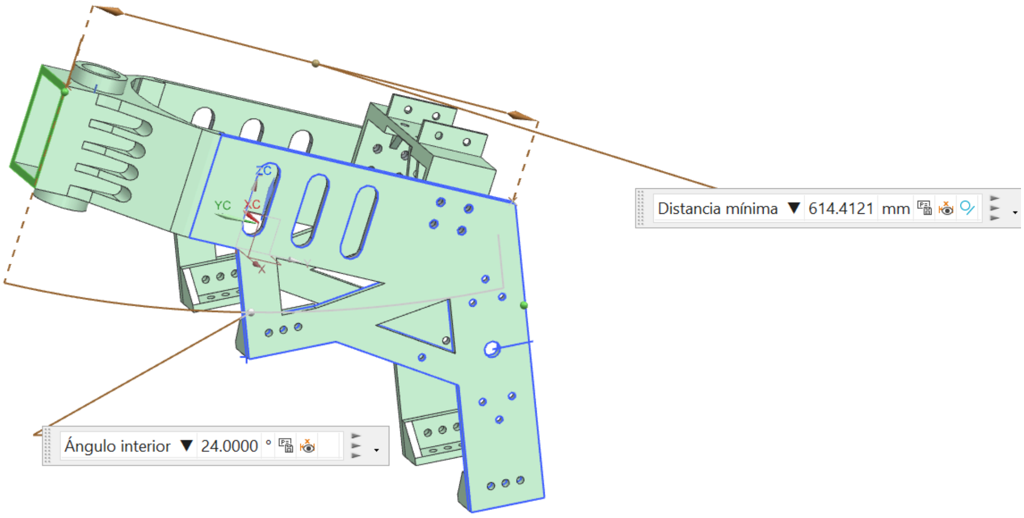
Academic year: 2023/2024

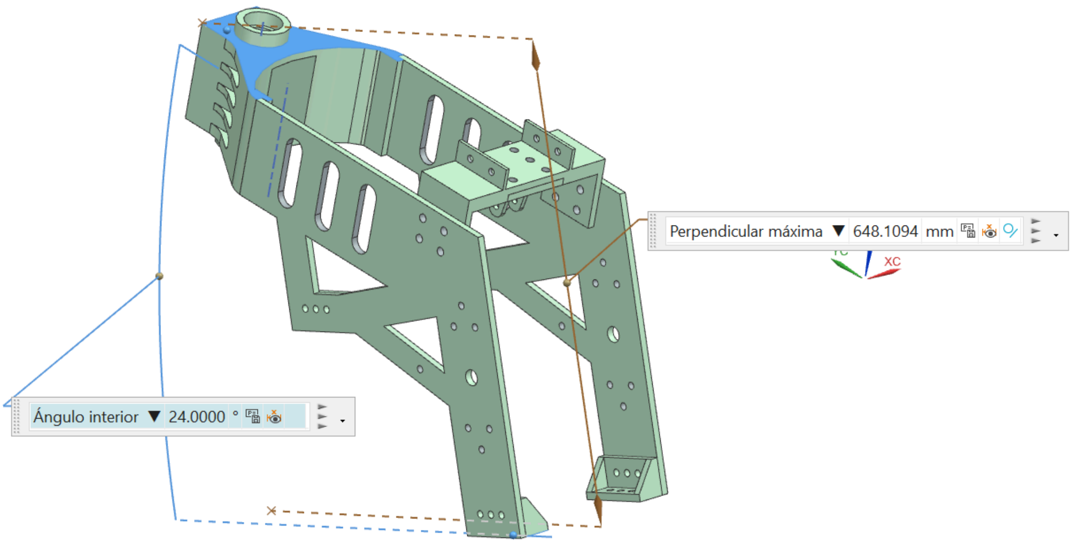
Index

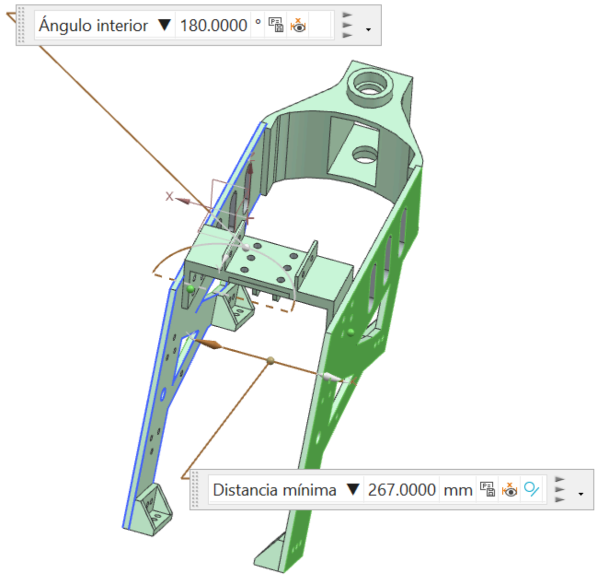
1. Introduction

In this project we decided to design and do the static analysis of the chassis of the motostudent motorbike. In order to make a correct geometry and simulation we take into account the MotoStudent International Competition 2024-2025 Rev.1/2024, where we can find all the competition regulations we need to fulfill our commitment. These regulations are explained below.

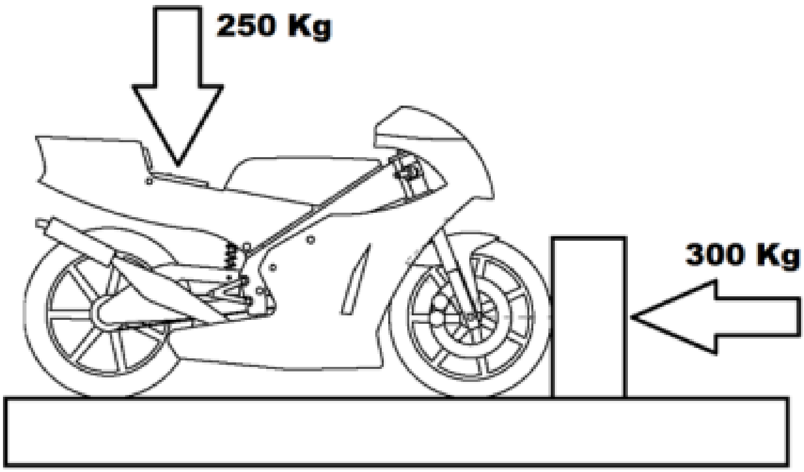
The dimensions of our chassis are restricted by the dimensions of the motorbike. These last ones are regulated in the competition regulations document named above. They are part of *Section B: General technical regulations, Article 2: General design requirements*. The chassis has been made in NX according to those measurements explained in the article. It has to be easy to manufacture and low weighted.







Another requirement for the motorbike is the static scrutineering. At this check, the motorbike prototype will be tested to horizontal and vertical forces in order to verify the rigidity of the frame, the correct assembly and behavior of the suspensions. These specifications can be found in the *Section E: Scrutineering*, inside the *Article 3: Static scrutineering*, where the forces required for the load test are explained and are the ones exposed below.



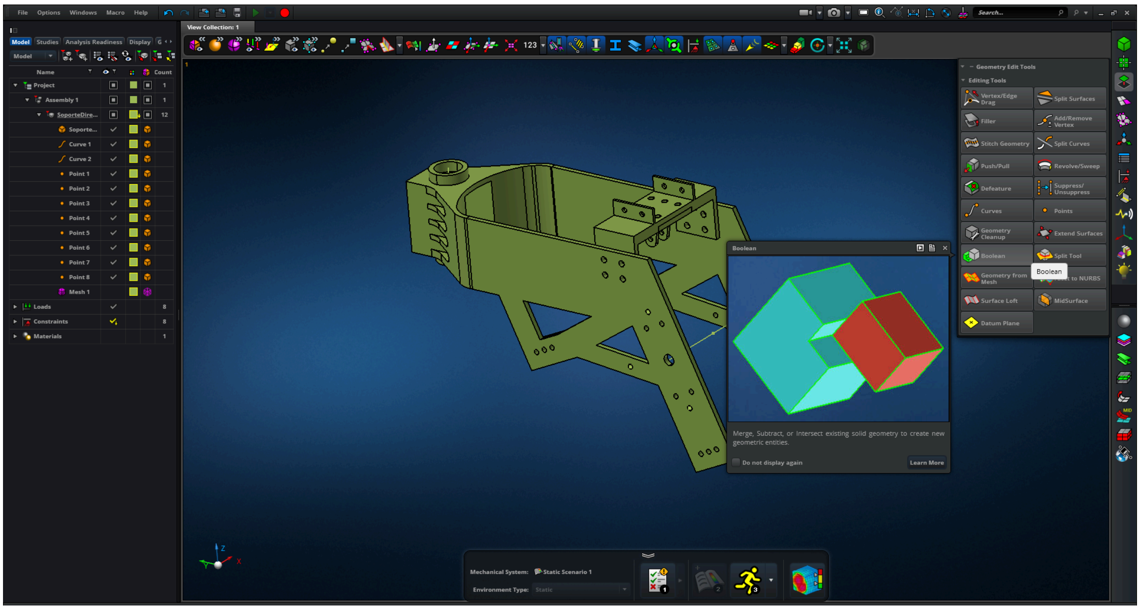
Picture 38 - Load test forces diagram

The material chosen for the chassis of this motorbike is aluminum. It has some very good properties that make it an adequate material for our purpose. The material is resistant to impact, flammable materials and corrosion, it hardly needs any maintenance, it can be easily transported and its weight does not represent a load problem.

2. Simulation process

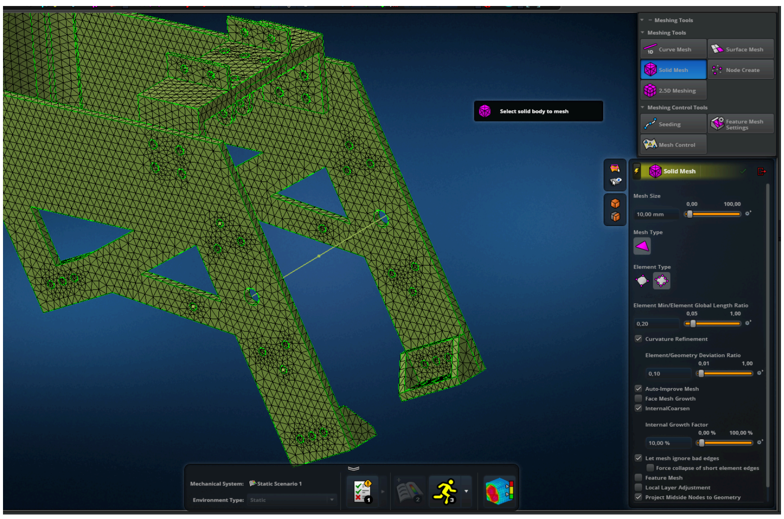
· Boolean operation

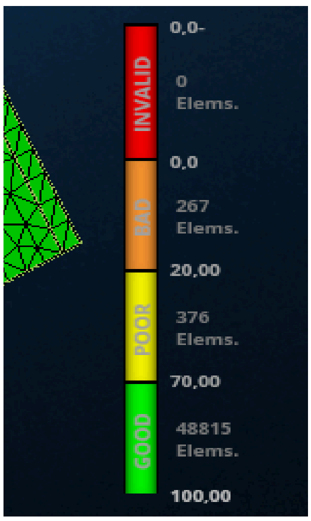
We started importing the model of the chassis to MSC Apex. The format we used to import was .step. This way we got the geometry in different parts. Then we did a merge boolean operation to create a single body and start simulating it.



· Creation of the mesh

We thought it would be interesting as a result of the project to show the stress and the displacement of the body, so the next step was to create the mesh using the solid mesh command, implementing a mesh size of 10 mm. After that, we had to check the mesh quality, which was quite good as it didn't have any invalid elements and the bad elements were very few compared to the good ones.





· Materials

Once we got an optimized mesh with zero invalid elements, we created the material determining its properties. We are using aluminum, with a strength yield of 69500,00 MPa and a Poisson's coefficient of 0.33. Then, we assigned the material to the whole body.

· Constraints and loads

To establish the constraints, we based our design in the real assembly of the motorbike. We put 2 types of them:

- Clamp, which represents the supports on which the motor is placed.
- Movable constraint, where the translation in Y and Z axis and the rotation in X axis are allowed

Once we have the model with the pertinent constraints, we can now put the forces

- Punctual ones
- Distributed

The constraints part

- First constraint is The constraints in the motor. We applied constraints to all holes where the engine is connected. The constraint prevents movement in the x-axis and rotation of the chassis in the y and z axes.
- Second constraint is Clamped constraints from the front side of the chassis. Here we restricted the movement and rotation of the chassis in the x y and z axis.

Loads

- Due to the weight of the battery, pressure is exerted on the points where the battery contacts the chassis. We calculated the pressure from the weight of the battery, taking into account the surface area of the base.
 We used the following equation when calculating the pressure of the battery on 4 different bases connected to the chassis.

Weight : 40kg   F= (40 kg * 9,8m/s^2)/ 4 pieces = 98N
Pressure = F/A   A = 80mm * 30mm - (2π*5^2)*3 holes = 1928,76mm

Pressure load = 98/ 1928,76= 0,0508 N/mm^2 = 0.0508 MPa

- The engine has a force equal to its weight on the chassis. We showed this power from a single point by connecting the middle point of the holes where the engine is connected to the chassis.

Weight : 26kg □ F= (26 kg * 9,8m/s^2) = 254,8 N

- To test the chassis structure, we added it to test the effect of 250 kg of force acting vertically on the motorcycle.

  Force : 250kg □ F= (250 kg * 9,8m/s^2) = 2450 N
- To test the chassis structure, we added 300 kg of force acting horizontally on the motorcycle.

  Force : 300kg □ F= (26 kg * 9,8m/s^2) = 2940 N
- This 300 kg power also creates a moment on the chassis. We found this effect by multiplying the force by the perpendicular distance from the midpoint of the chassis.

  M = F * d

  M = (300 * 9,8) * (650 - (150/2) * cos (24º)) = 1544.348 Nm

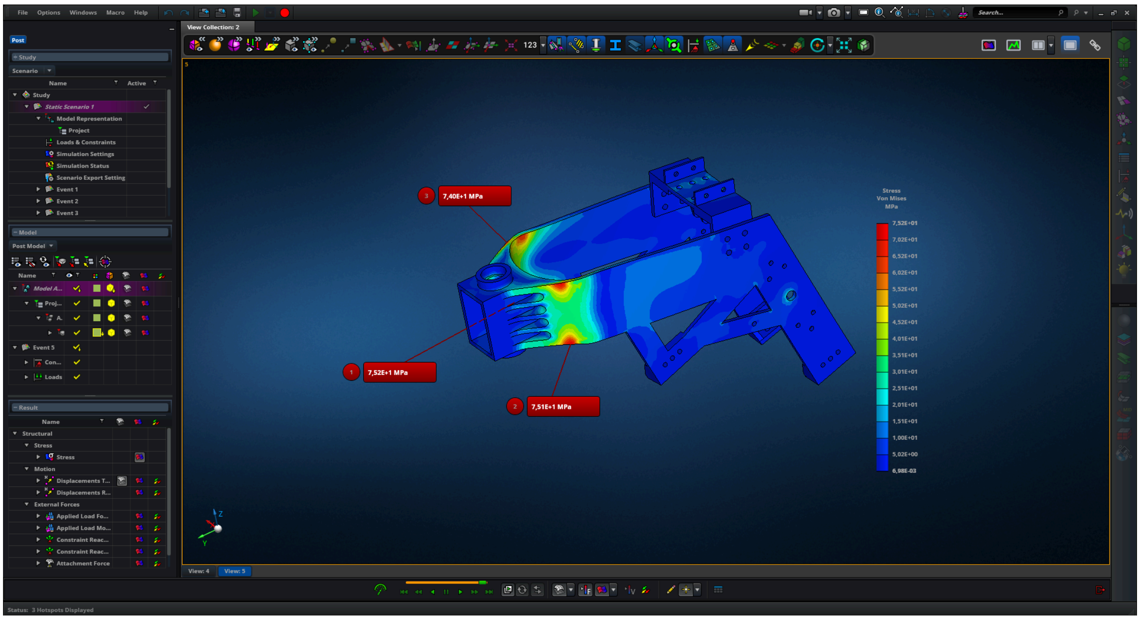
· Events

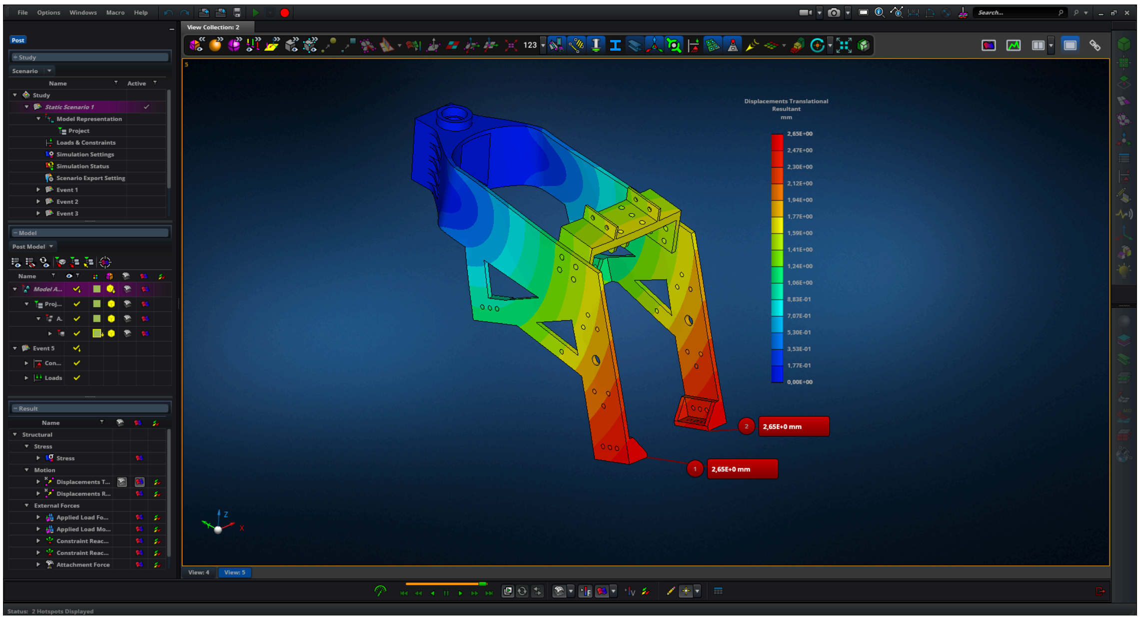
Before simulating the project we decided to create an event for each load, in this way we could see which load creates more stress in the structure and which less. In addition, if we do not have an adequate structure, we could reinforce it more easily because we would know the load that produces that stress.

3. Results

After completing our work on the geometry, we ran the analysis screen and determined how much stress was caused at which points by the forces acting on the geometry, detecting the maximums and minimums. We also identified the points that shifted the most on the geometry.

The following images that we got after running the simulation show, respectively, the stress and displacement of our chassis with all the constraints and loads applied.





As we can see, the hot spots according to the stress are going to be in the upper rounded part. We get a maximum stress of 75.2 MPa in this area, while the maximum displacement is 2.65 mm.

As a result, we determined that our geometry is strong and designed to withstand maximum stress, because the yield strength of the material we are using is way bigger than the maximum stress.

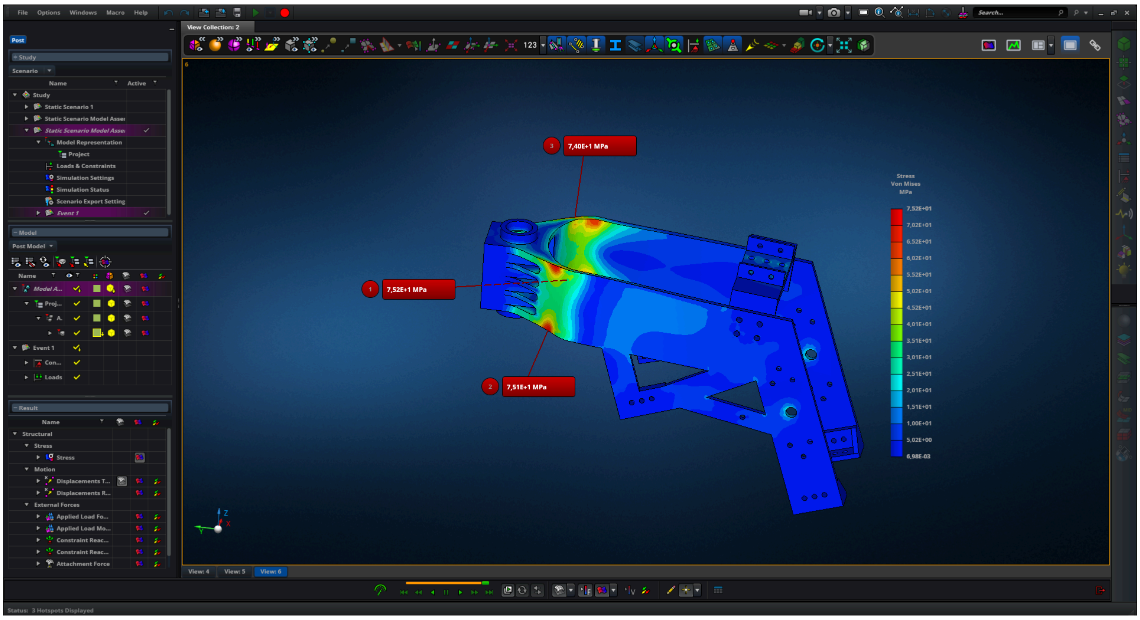
$$Yield\ strength\ =\ Maximum\ stress\ *\ SF\ ; SF\ =\ \frac{230\ MPa}{75.2\ MPa}; SF\ =\ 3.06$$

This way, we get a security factor of 3.06, more than enough for the conditions that we have. At this point, we thought that some changes could be made in order to optimize the body, so in the next section we are going to study how these results differ after applying more details to the geometry and changing the mesh size.

4. Optimization

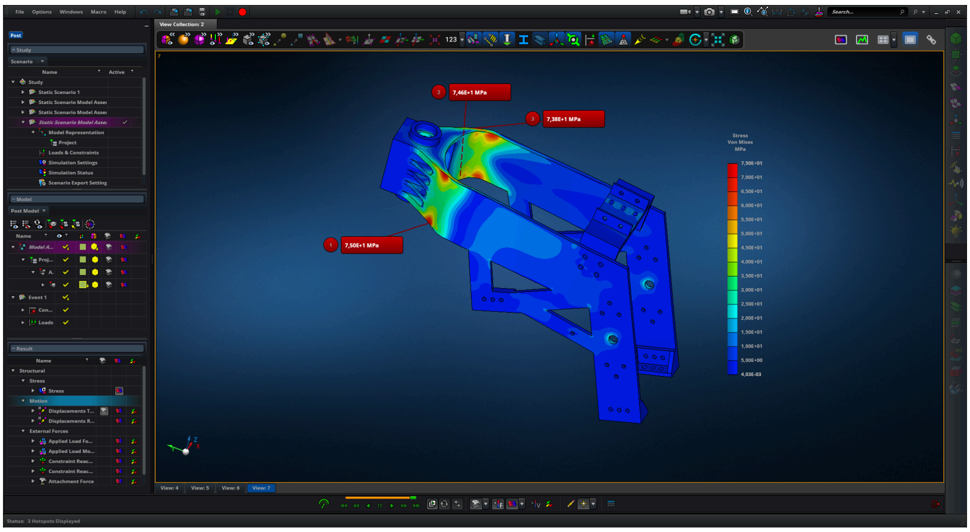
Once we have finished the first model and verified that it is correct by getting an appropriate security factor, we are going to carry out different changes and study all of them.

First of all, we will study how the mesh size alters the stress and displacement results. For this purpose, we change the mesh size into 7 mm and get the following results:

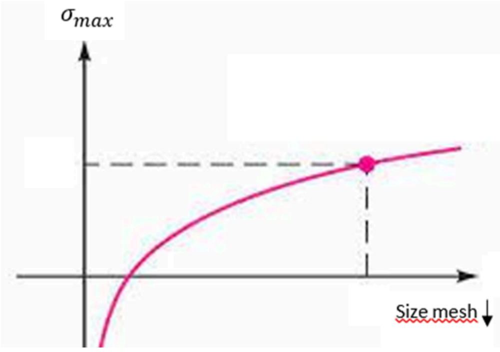


We get a maximum stress of the same value, 75.2 MPa, and in the same area as before. We could say that we reached the value where the stress stabilizes.

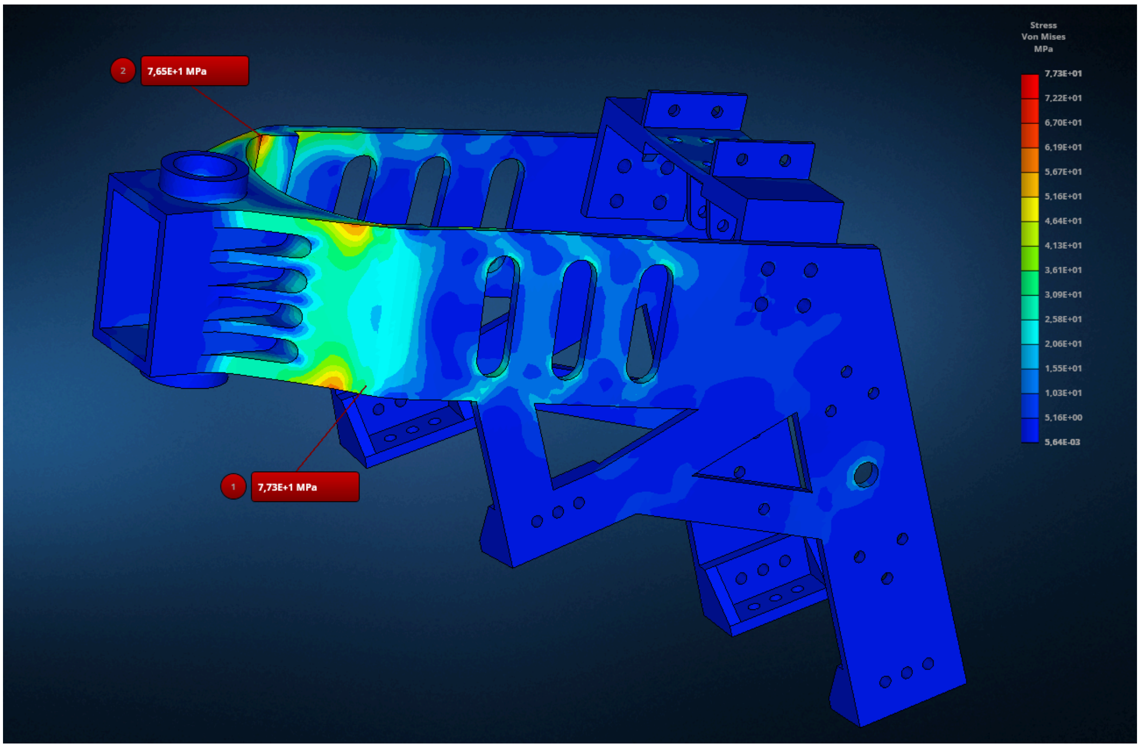
Even so, in order to be completely sure, we decided to try again with a smaller mesh, so we introduced the value of 5 mm for the mesh size, achieving the results below:

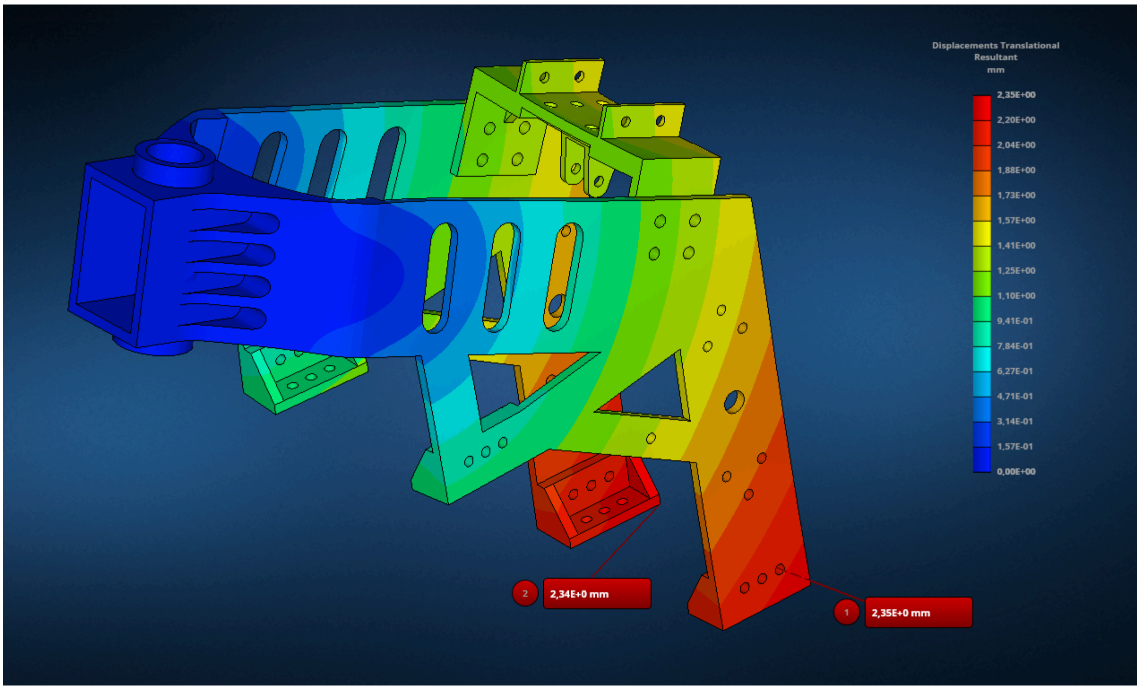


There we can see that this time we get a maximum stress of 75 MPa, the variation is slight, so we confirm that the value is stabilized and we can take it for good. If the results had differed a lot, that would have meant that the mesh size wasn't still good, as we can see in the graph.



After these comprobations on the mesh size, we decided to make some holes in order to use less material and get a lower weight. This way, we get the following final geometry, which we simulated again to be sure that it is adequate.





As we can see, the maximum stress value is 77.3 MPa, so we still have a huge security coefficient.

$$SF = \frac{230\ MPa}{77.6\ MPa}; SF = 2.98$$

We can say that our design is adequate and very reliable thanks to this final security factor.