POLITECNICO DI TORINO

Dipartimento di Ingegneria Meccanica e Aerospaziale

Corso di Laurea Magistrale
in Ingegneria Meccanica

Tesi di Laurea Magistrale

Formula 1 safety buckle modelling in FEM

Relatore

Prof. Elvio Bonisoli
Diego Cagna

Candidato

Edoardo Ricci

Dicembre 2021

Index

List of figures

List of tables

Abstract

The thesis which has been developed in Sabelt spa, has the purpose to analyse a Formula 1 safety harness, focusing on the buckle.
Mainly, the harness is composed by many elements and the most interesting from an engineering point of view is the buckle. This object is used in each car, but his shape depends on the purpose.
Sabelt spa provides a wide range of harnesses, from racing environment, as Formula 1, rally or endurance, to commercial vehicles.
However, the structure of the harness depends on the usage because the homologation required is different. For example, following the normative imposed by the Federation International de l'Automobile (FIA), a Formula 1 harness must be used with 6, 7, 8 or 9 attachment points and the straps involved are six: two passing on the shoulders, two crossing the pelvic zone, two scraping on the crotches. On the other hand, for a car used on the road, according to the Economic Commission for Europe (ECE), the harness can be provided also with 4 attachment points and the straps could be just four, two shoulders and two pelvises.
Anyway, the aim of the project focuses on the buckle of a Formula 1 harness and Sabelt spa uses two kinds it.
Therefore, it has been useful analyse one model, the most performance one, under many points of view, starting from the design to his structural analysis.
The model has been studied in Solidworks which is an industrial CAD-CAE software that represents efficaciously the CAD and FEM environment both, so a perfect trade-off in terms of design and static analysis. Moreover, another industrial software as Matlab played a minor role in this thesis, performing in the graphic representations.

Introduction

The topic of the thesis has been already developed in Sabelt spa, indeed this model of buckle is used, nowadays, from many F1 teams. Therefore, the focus is not on the values got from the simulations, but on the method used to obtain them. The hardest procedure is to get the right set up for the static analysis, as for example the interference between each component. During a FEM simulation, nothing should compenetrate to get the right result. Following a simple procedure tested with the F1 buckle, it is possible to achieve the correct set up avoiding mistakes and loose of time.

Chapter 1 starts proposing an excursus on the structure of a harness. The commercial cars, which could be light and heavy duty both, have a simple harness: two attachment points on the seat, one on the right and one on the left of the driver, and one attachment point on the car door above the left shoulder. A short strap starts from the right points, and it is linked to the buckle, otherwise, a long strap starts from the upper left point and ends on the lower left one. This strap has an adjusting device, to regulate his length, and it passes through a tongue to fill it in the buckle and lock the driver. Pushing on the buckle, the tongue come out and the driver could leave the seat. The aim of the harness is to keep safe the driver from each kind of accident.

Chapter 2 concerns the 3D model in SOLIDWORKS, starting from the design of the parts and proceeding with the assembly. At the same time, each part has been examined under many aspects, as the material proprieties and his mechanical processing. Moreover, it is necessary to analyse his homologation, indeed, the regulations play a fundamental role.

Chapter 3 concerns the preparation for the structural study. It is necessary to remove each kind of interference to allow the right meshing of the assembly.

Chapter 4 concerns the structural study, so it starts with a short description about Finite Element Method, then it proceeds with the mesh theory. It follows with some paragraphs about Solidworks set up for the structural study and it continues with the case study. Indeed, the linear static analysis has been simulated on the buckle and the results has been collected.

Chapter 5 is about the movements made by the buckle, as, for example, the detachment of the tongues from the buckle or the rotation of the lever. These have been simulated through a motion study, still in Solidworks.

1 Physiognomy of a harness

The harness is composed by several elements (Figure 1.1):

- the buckle;
- the straps;
- the tongue;
- the adjusting device;
- the snap hooks.

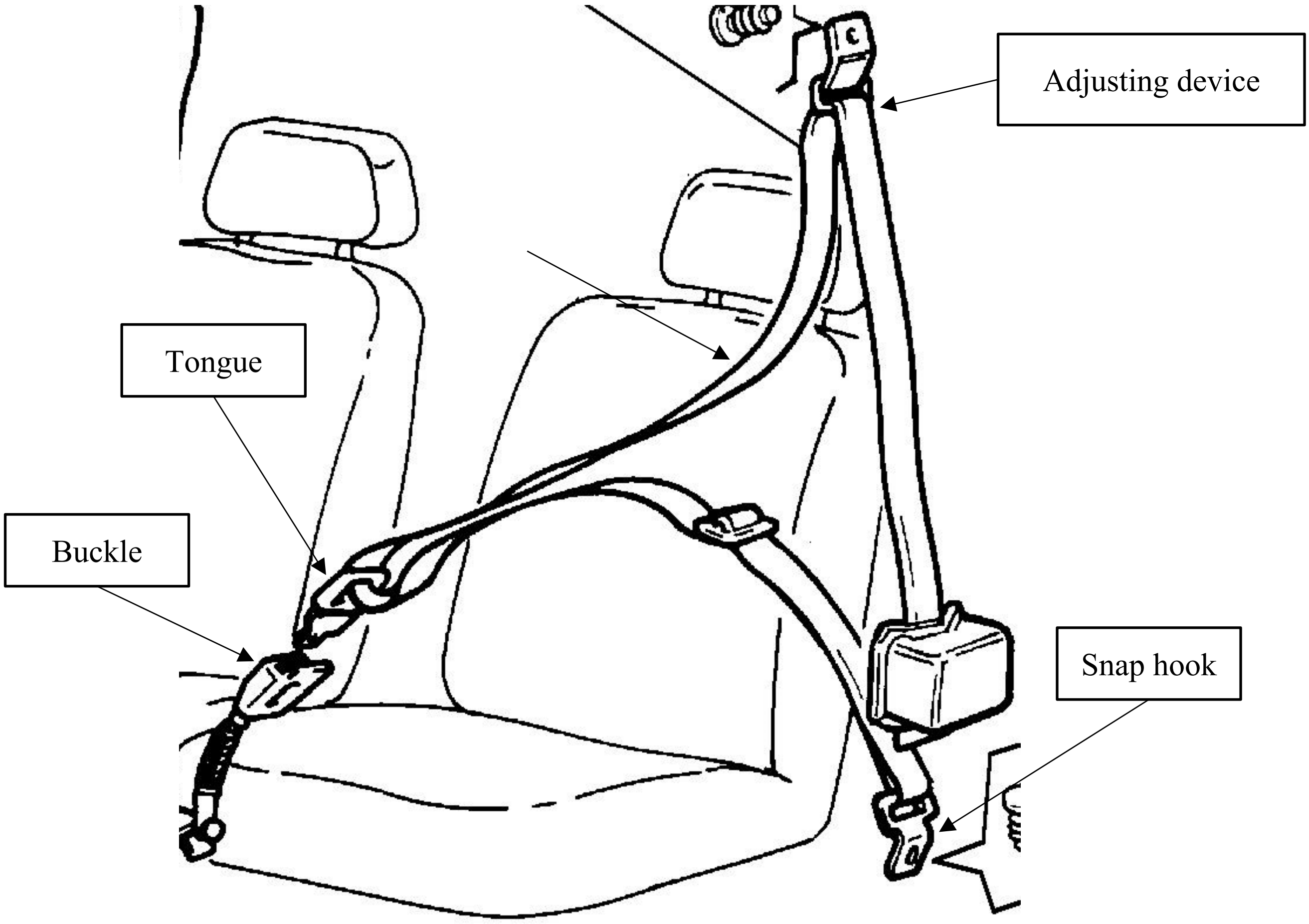

Figure 1.1 – Schematic representation of a commercial car harness.

The buckle is a device which is used for fastening two loose ends, one end attached to it and the other held by a catch in a secure, but adjustable manner. Particularly, as it is shown in Figure 1.1, the snap hooks are fixed on the chassis. The straps are sewn as a loop passing through the snap hooks. One strap is fixed at the buckle through a loop sewing, instead the other strap has an adjusting advice, to regulate his length, and it is sewn around a buttonhole of the tongue. The tongue is placed in the buckle to link the straps and lock the body of the driver.
Indeed, nowadays, its main purpose is to guarantee safety and reliability to an individual which is placed in a seatbelt and the number of the straps linked to it depends on its application.

In the racing environment, according to the Fedaration International de l'Automobile (FIA), the buckle is used with 6,7,8 or 9 number of attachments points to vehicle (Figure 1.2), therefore the number of loose ends increase.

Figure 1.2 – Example of harness with 6 attachment points.

Sabelt supplies harnesses to Formula 1 with 6 or 8 points which coincide with the number of straps, as Figure 1.3 shows.

Figure 1.3 – Example of Sabelt harness used in Formula 1.

Observing the Figure 1.3, it is useful to analyse the three kinds of straps:

- shoulder straps, on the top;
- pelvic straps, on the middle;
- crotch straps, on the bottom.

The tongues are employed to link the straps and the buckle, especially the straps are sewn with the tongues which are fixed by the buckle.
Moreover, the straps could be characterised by an adjusting device to regulate the length of it.
In the end, the straps are linked to the chassis through the snap hooks which are clasped to the roll bar.

Summing up, the harness ensures the safety of the person because the straps are linked to buckle and, at the same time, to the vehicle (Figure 1.4). During a sudden braking or during a collision the straps prevent the hurling of the individual, blocking it to the seatbelt.

Figure 1.4 – Example of Sabelt harness in Formula 1.

Sabelt spa provides two kinds of model for the Formula 1:

- the light one, shown in Figure 1.3;
- the super light one, shown in Figure 1.5.

Figure 1.5 – Super light Formula 1 harness.

Besides, Figure 1.6 shows the visual difference between these two models, focusing on the buckle only.

Figure 1.6 – Light Formula 1 buckle on left, super light Formula 1 buckle on left.

The choice of the buckle depends on the Formula 1 drivers, some prefer the trefoil one, others the circular one. The reason could be the mechanical proprieties and the comfort both. Some aspects are listed:

- the super light weighs 130g, instead of the circular, which is 140g, so the first one guarantees a better performance, talking about aerodynamic;
- the trefoil has a higher centre of gravity to press less the pilot at the pelvis level;
- the super light has a fixed tongue on the left, hence during the unfastening the buckle keeps attached to a strap, instead the circular has a free unfastening;
- the super light has three free tongues, the circular one has four free tongues.

Each Formula 1 driver could choose between these models, according to his racing team. For example, some prefer the super light due to the comfort, others like the circular because of the free release of the tongues.

1.1 Purpose and aims

This project aims to analyse some components of the Formula 1 safety harness: the buckle with his tongues.
They are analysed from an engineering point of view:

- operating principle;
- CAD model of each component, besides the assembly;
- drawing of the parts, in addition to the assembly;
- analysis of FIA specifications;
- FEM analysis of the assembly, through a linear static analysis;
- motion study of the mechanisms of the assembly.

The focus of the thesis is only on the super light buckle because the geometry and the functioning are the same for the buckles both.

1.2 Operating principle

Now it is essential to introduce briefly how the buckle works.
The main goal of the harness is to guarantee the safety of the individual on the seatbelt, therefore it is important to analyse the coupling between the buckle and the straps from a mechanical point of view.

On the upper body of the buckle there is a lever which could rotate in both directions. Through the rotation of the lever is possible to unfasten the tongues which are linked to the straps. Therefore, it is necessary to apply a torque on it.
Moreover, some tongues are unmovable to keep the buckle attached to the straps.
Figure 1.2.1 shows fastening and unfastening mode of the buckle.

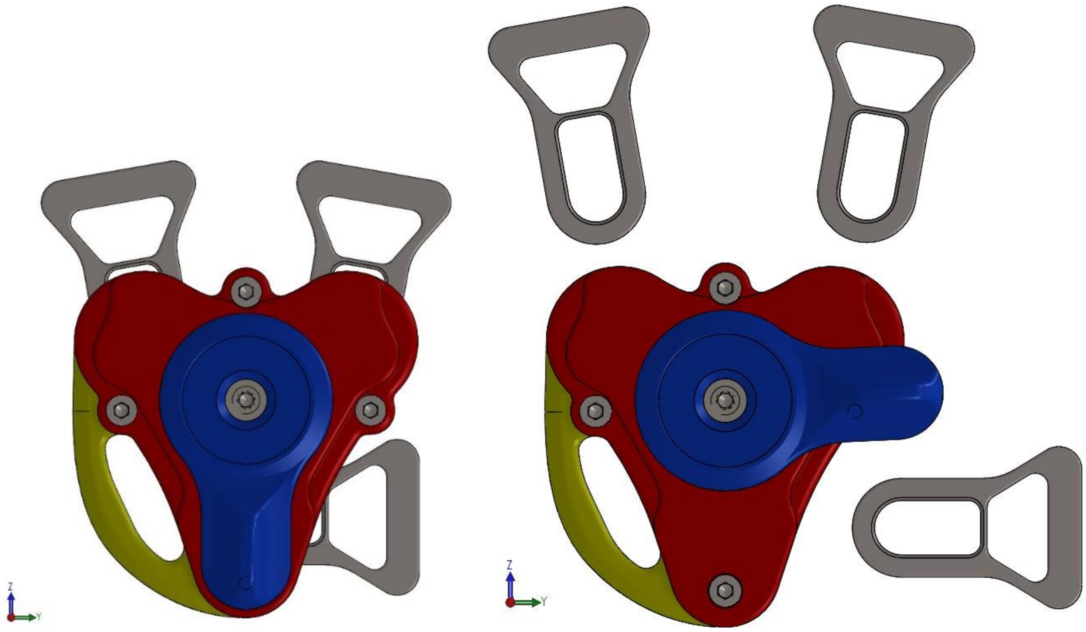

Figure 1.2.1 – Super light buckle in Solidworks: buckled position on left; unbuckled position on right.

In Figure 1.2.1, the left tongue on bottom is fixed, so the left pelvic strap keeps his position, instead of the right pelvic strap and the shoulder straps which are unhooked through their three tongues. Furthermore, removing the shoulder tongues, the crotch straps are pulled out.
After the removal of the tongues, the way to fasten them is to insert them with the lever in the buckled position.

2 CAD model

2.1 Reference system

The first step to create the CAD model in Solidworks, it is the definition of the reference system. The right way to determine it, it is imagining to be a driver while wearing a racing harness, as Figure 2.1.1 shows.

Figure 2.1.1 – Racing pilot while using his harness.

In particular (Figure 2.1.2):

- X axis of the buckle coincides with the advance of the car which represent the X axis of the car reference system;
- Y axis of the buckle coincides with the lateral movements of the car which is the Y axis of the car reference system;
- Z axis of the buckle coincides with the elevation of the car which is the Z axis of the car.

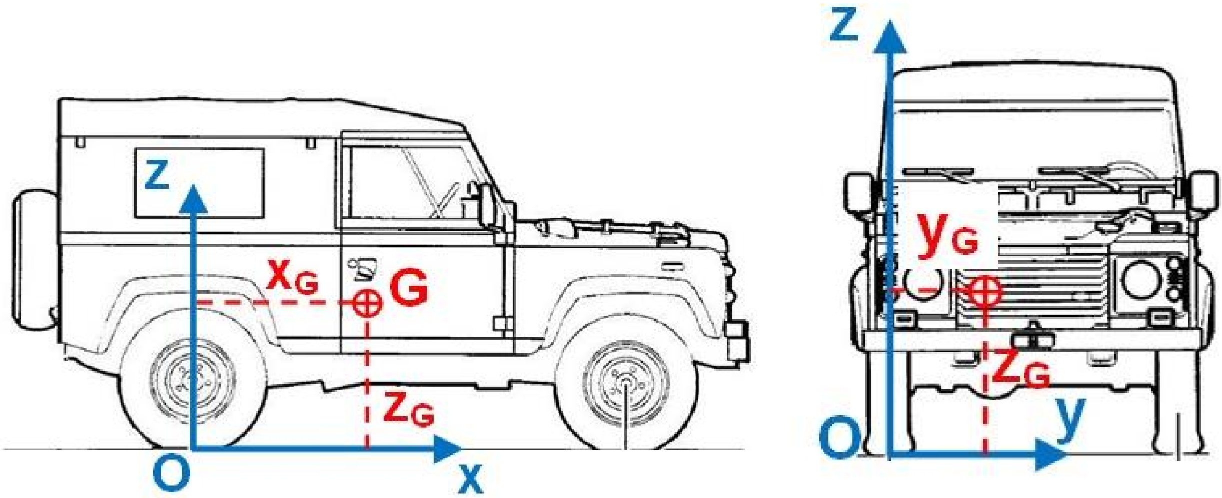

Figure 2.1.2 – Car reference system.

Therefore, in Figure 2.1.3 is shown the buckle reference system in an isometric view.

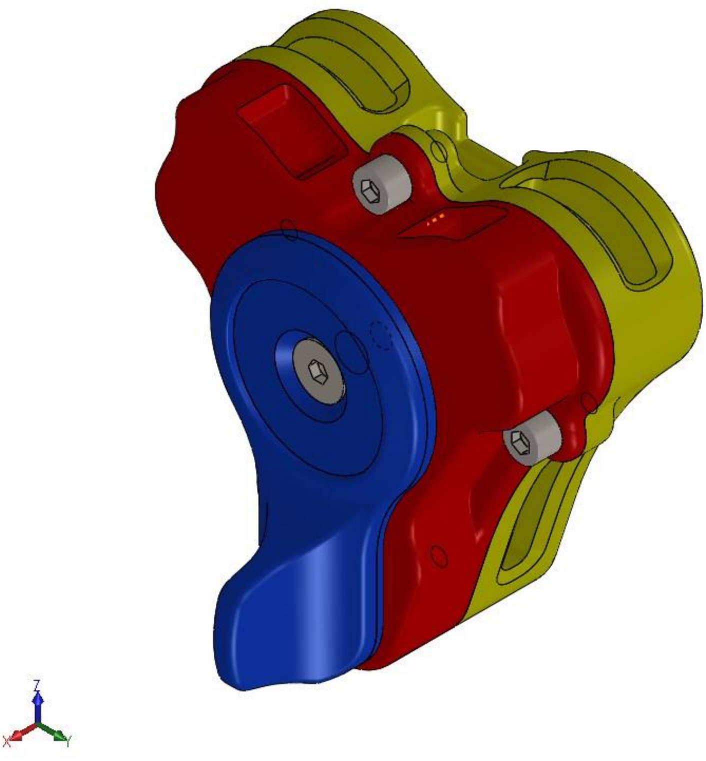

Figure 2.1.3 – Buckle reference system.

2.2 3D model and drawing

The second step after the reference system concerns the Bill of Materials which compose the buckle, hence, it useful to analyse his explosion (Figure 2.2.1).

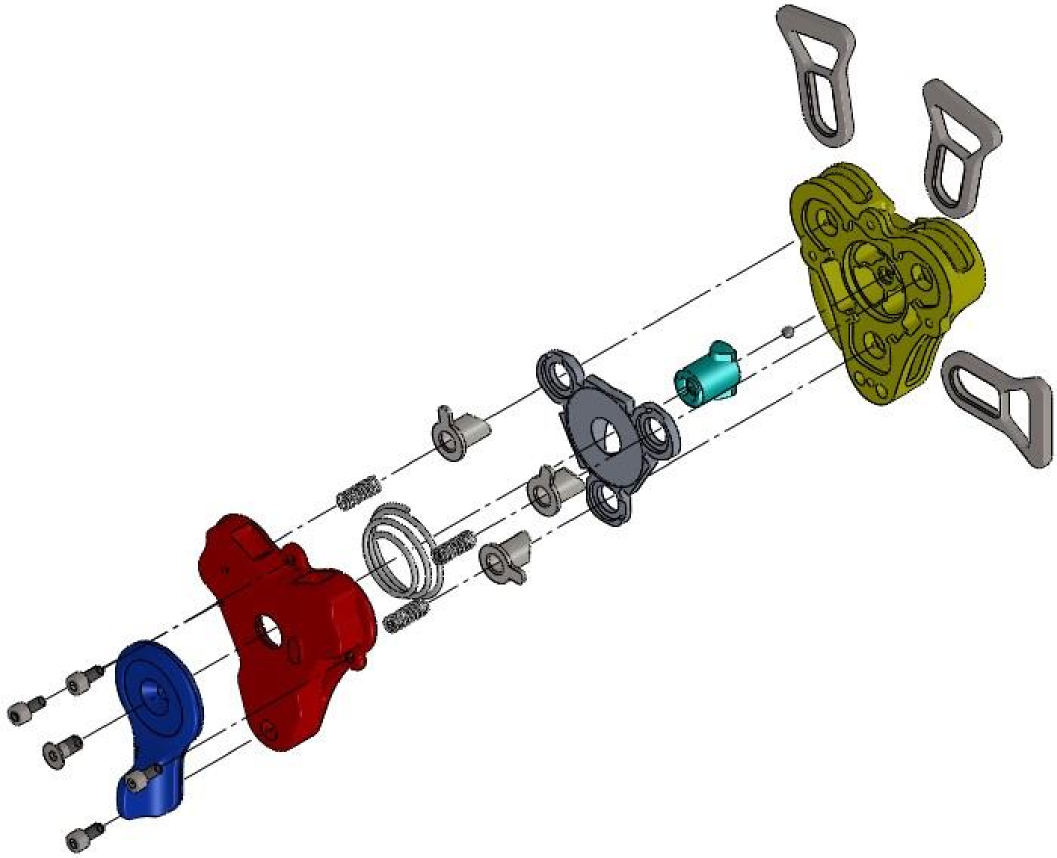

Figure 2.2.1 – Super light buckle explosion.

Figure 2.2.1 shows the explosion of the super light buckle, it is possible to see the assembly of it starting from the lower component which is the lower body till the screws which are the last components, used to lock the lever and the bodies. Now il will be studied each component from a physical point of view, focusing on the material and the mechanical processes involved.

Lower body:

- raw material: Aluminium 7075, known as Ergal, which is an alloy in the Aluminium-Zinc-Magnesium family, so 7xxx series. It has excellent mechanical proprieties and exhibits good ductility, strength and resistance to fatigue;
- raw material preparation: annealing at 400°C for 2 hours, then slow cooling in electric bake with uniform temperature;
- press forging: process made through presses in stamps and put into air bakes at 370°C;
- heat treatment: tempering for 90 minutes at 465°C and rapid water cooling. Then, aging for 12 hours at 135°C;
- mechanical processes: preliminary cutting of the burrs and tumbling to get smooth surfaces. Then manufacturing process and milling;
- galvanic treatment: electrochemical deposition of subtle layer of a metal or an alloy which modifies the support surface proprieties, improving for instance its corrosion resistance and the mechanic characteristics.
  It is composed by three steps: pickling to clean the surface through chemical dissolution, anodizing to get an Aluminium oxide and washing with surfactant liquids.

Sphere:

- raw material: steel AISI 304.

Camshaft:

- raw material: metal powder of graphite;
- pressing and sintering: process of compacting and forming a solid mass of material by heat or pressure without melting it to the point of liquefaction. The atoms in the materials diffuse across the boundaries of the particles, fusing the particles together and creating one solid piece;
- mechanical processes: drilling to get the housing of the screw;
- carbonitriding: metallurgical surface modification technique used to increase the surface harness of a metal, thereby reducing wear. Atoms of carbon and nitrogen diffuse interstitially into the metal, creating barriers to slip, increasing the hardness and modulus near the surface;
- surface treatment: phosphatization. It is a chemical treatment which creates a thin adhering layer of zinc to achieve corrosion resistance and lubrification. Moreover, on the camshaft is applied a lubricant called Molykote and facilitates the contact with the steel sphere.

Lifting plate:

- raw material: Ergal;
- die casting: metal casting process characterized by forcing molten metal under high pressure unto a mould cavity. The lifting plate is pressed by 300 ton with a four holes mould;
- blanking: shearing process where a punch and a die are used to produce parts from sheet stock. In this case, blanking removes the die casting burrs;
- tumbling to smooth the surfaces.

Latch:

- raw material: metal powder of graphite;
- pressing and sintering;
- mechanical processes: drilling to get the housing of the latch spring;
- carburising: it is a heat treatment process in which iron or steel absorbs carbon while the metal is heated in the presence of a carbon-bearing material. The intent is to make the metal harder and increases the wear resistance;
- surface treatment: phosphatization. It is a chemical treatment which creates a thin adhering layer of zinc to achieve corrosion resistance and lubrification. Moreover, on the camshaft is applied a lubricant called Fosfil and facilitates the contact with the steel sphere.

Springs:

- raw material: steel C72.

Upper body:

- raw material: Ergal;
- same mechanical processes of the lower body.

Lever:

- raw material: Aluminium 6082 which is an alloy in the Aluminium-Magnesium-Silicon family, so 6xxx series;
- raw material preparation: annealing at 400°C for 2 hours, then slow cooling in electric bake with uniform temperature;
- press forging: process made through presses in stamps and put into air bakes at 420°C;
- heat treatment: tempering for 90 minutes at 540°C and rapid water cooling. Then, aging for 12 hours at 165°C;

- mechanical processes: preliminary cutting of the burrs and tumbling to get smooth surfaces. Then manufacturing process and milling;
- galvanic treatment: electrochemical deposition of subtle layer of a metal or an alloy which modifies the support surface proprieties, improving for instance its corrosion resistance and the mechanic characteristics.
  It is composed by three steps: pickling to clean the surface through chemical dissolution, anodizing to get an Aluminium oxide and washing with surfactant liquids.

Screws:
- raw material: titanium.

Tongue:
- raw material: titanium;
- milling.

2.3 Mechanical couplings

After this preliminary analysis of the components, it is necessary to study how a buckle works, so what are the mechanical couplings among the parts.
Figure 2.3.1 shows the drawing of the buckle, and it is useful to observe the contacts of the components.

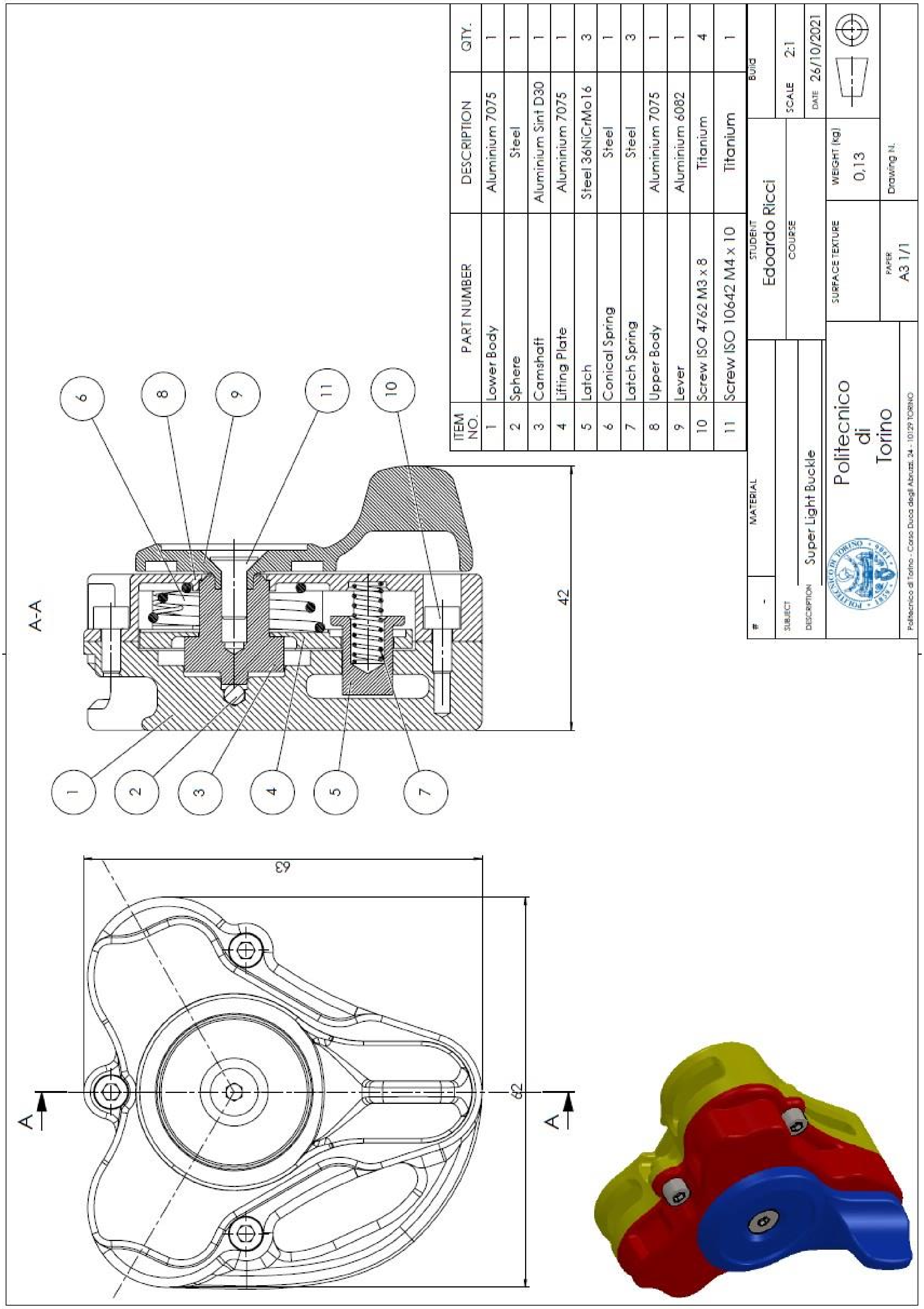

Figure 2.3.1 – Super light buckle assembly.

The rotation of the lever, imposed by the driver with his hand, acts directly on the tongues, which are released, therefore the driver can take off his harness and get out from the car. Otherwise, the way to fasten the harness is to insert the tongues with a constant force into the holes of the lower body; the buckle is hooked when each latch locks his tongue. Particularly, the driver pushes the tongue along the ZY plane, in the hole direction, forcing the latch (Figure 2.3.2).

Figure 2.3.2 – Fastening of one tongue.

Thanks to the latch surface, as it is shown in Figure 2.3.3, the tongue can raise the latch and cross it. When the latch meets the loop of the tongue, the latch falls and locks the tongue.
The double inclination is necessary to facilitate the insertion of the tongue and to resist the breaking load.

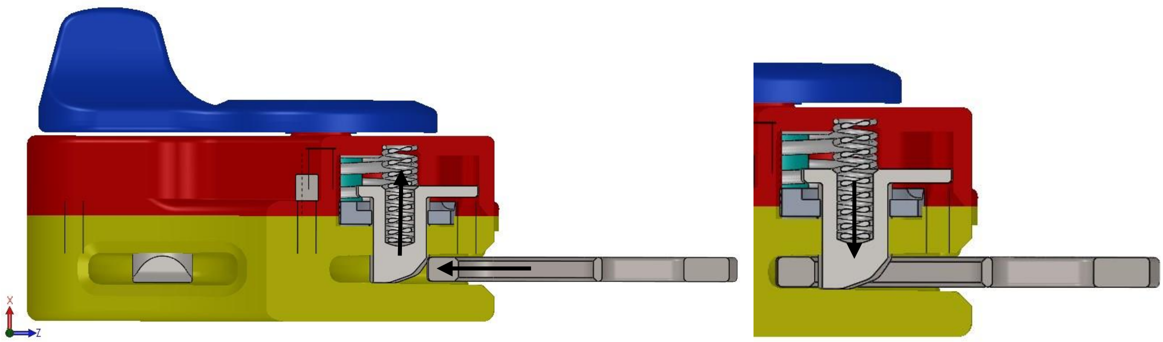

Figure 2.3.3 – Buckle section to show latch-tongue coupling.

How the buckle works:

1. resting position: a conical spring between the upper body and the lifting plate keeps this one in his low position, moreover, three cylindrical springs between the upper body and the latches keep these ones in contact with the lower body;
2. the rotation of the lever acts on the rotation on the camshaft. If the lever turns of 60°, the camshaft follows this movement;

3. the camshaft is in contact with the cams of the lifting plate, indeed, during the camshaft rotation, the cams act on the lifting plate elevating it on X direction. This movement compresses the conical spring;
4. the elevation of the lifting plate guarantees the same movements of the latches and their springs are compressed;
5. the raising of the latches allows the removal of the tongues because it forms a plat between the latches and the lower body.

2.4 Mates

The only way to create the assembly starts from components: after their design in Solidworks, it is necessary to relate them each other's.
In order:
1. to fix a part on the reference system to have a starting point: lower body;
2. to relate each component with the lower body to compose the entire assembly. The purpose is to define the correct geometry in the space using the mates;

In Solidworks, there is a dashboard for the mates (Figure 2.4.1) where you must select:
- kind of mates, as standard, advanced, mechanical;
- related geometry of the two involved components, as surfaces or axis.

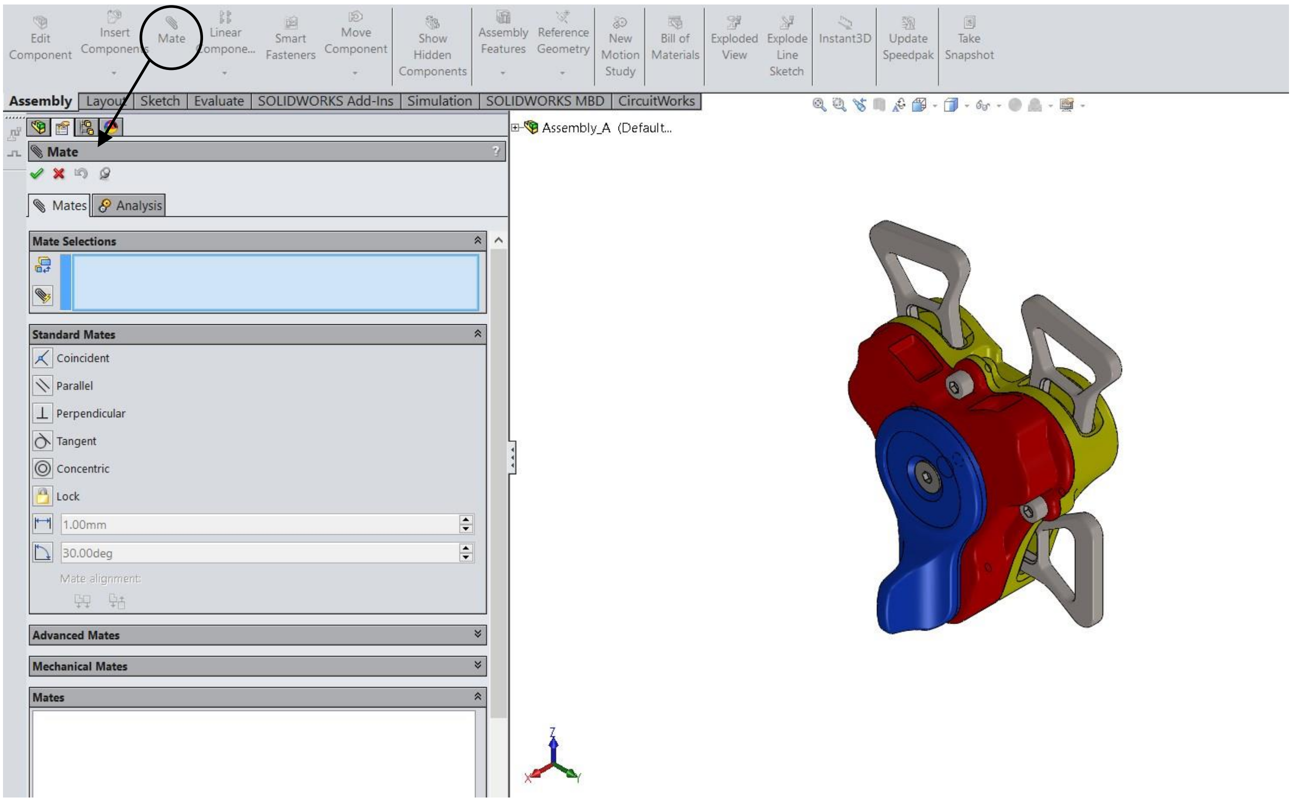

Figure 2.4.1 – Mate dashboard.

Now it is useful to analyse only some relations, just to understand the logic used to complete all the assembly.
To define the starting point, you must mate the planes of the reference system with the planes of the lower body. Figure 2.4.2 shows an example of mate used.

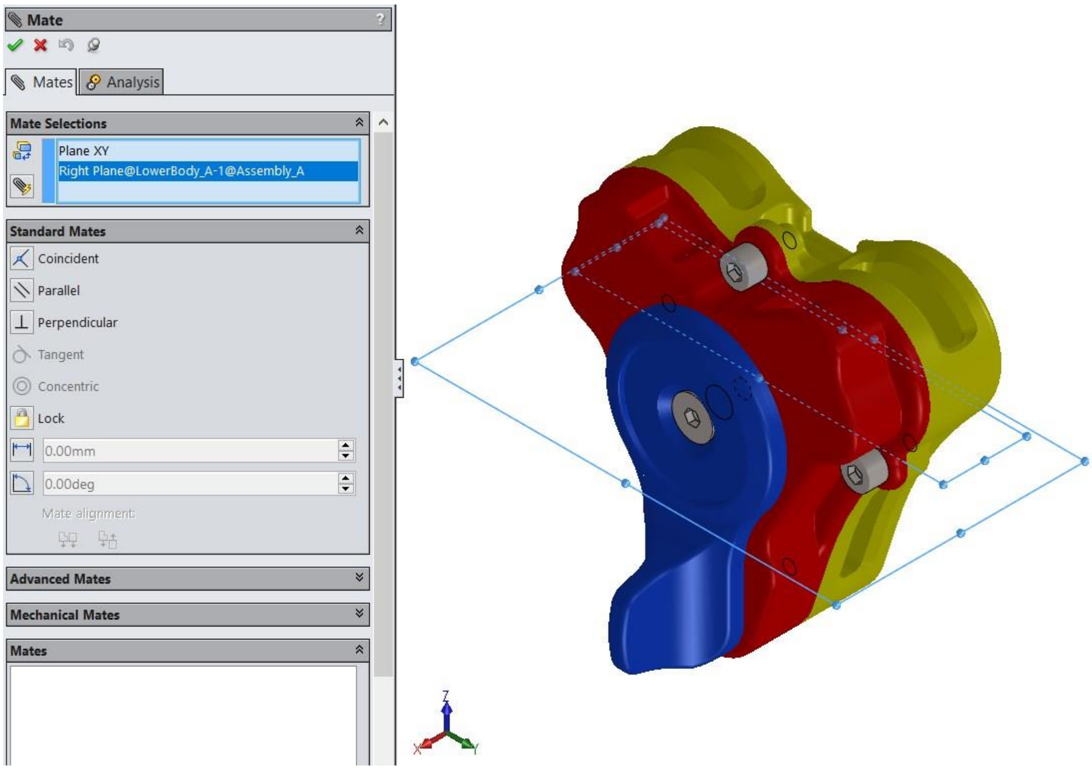

Figure 2.4.2 – Example of mate to set the reference system.

Usually, it needs three mates to lock the component, indeed, the lower body has been fixed through his three planes, instead, for example, the upper body:

- coincident between upper body and lower body faces, as Figure 2.4.3 shows;

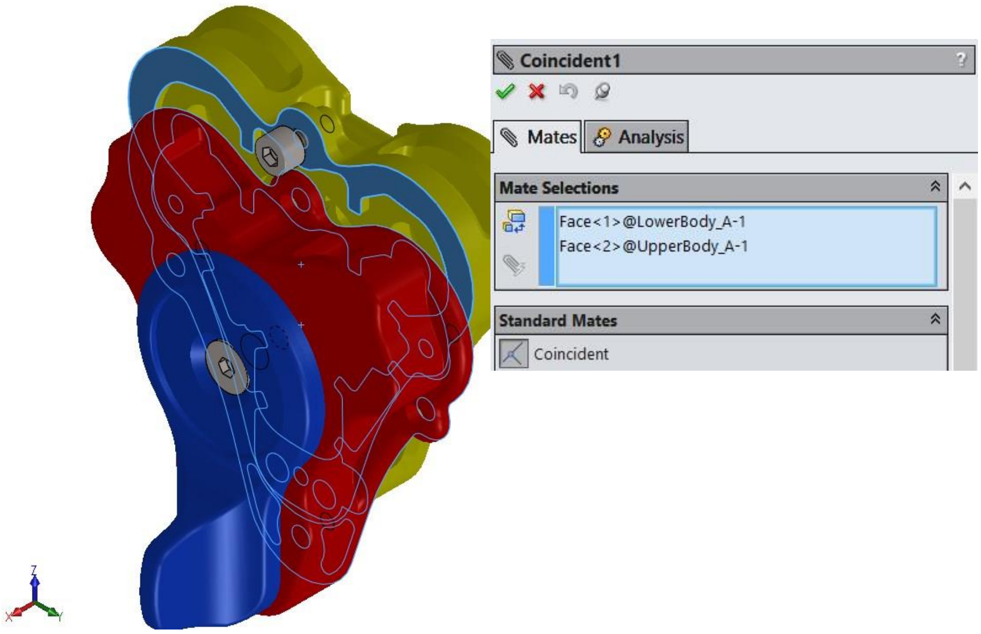

Figure 2.4.3 – First mate to lock upper body in relation to lower body.

- two mates for concentricity (Figure 2.4.4).

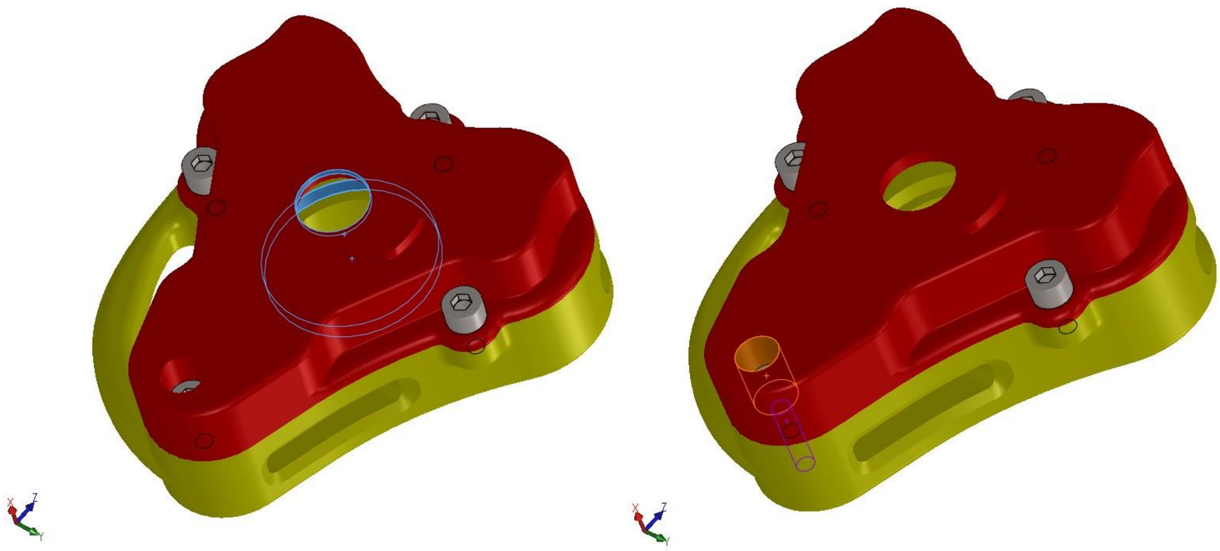

Figure 2.4.4 – Second and third mate to lock upper body in relation to lower body.

These mates allows the locking of the upper body, and the same process must be done to lock each part of the buckle.

2.5 FIA specification

Each component which is used in Formula 1 must be homologated by the Federation International de l'Automobile (FIA). Therefore, each company must respect the standard imposed for the object which provides. In this case, the buckle and his harness are described in "FIA Standard 8853-2016 – Safety Harnesses" which defines the performance and the design requirements.
This specification is only about harnesses made up of two pelvic straps, two shoulder straps and two crotch straps, making a total of six straps in contact with the driver's body. Besides FIA 8853-2016 describes all tests, requirements and conditions for each component of the harness, it will be analysed only the parameters which are necessary for the FEM analysis, so the geometry of the buckle and his breaking load.
Firstly, the buckle could assume two kinds of configurations:

- type "P" which works by pressing a button or a similar device;
- type "T" which works through a turning motion. It shall release the buckle in whichever direction the lever is turned.

The super light buckle studied in this project is a type "T" and, following to FIA, his pre-release free movement shall extend through a minimum angle of 25° measured on both sides of the closed position before the release mechanism begins to operate. This is necessary to avoid useless stress between the cams of the lifting plate and the camshaft. Figure 2.5.1 shows the range of free movement, after this angle which correspond to 25°, the camshaft starts to move the lifting plate, due to the force on the cams.

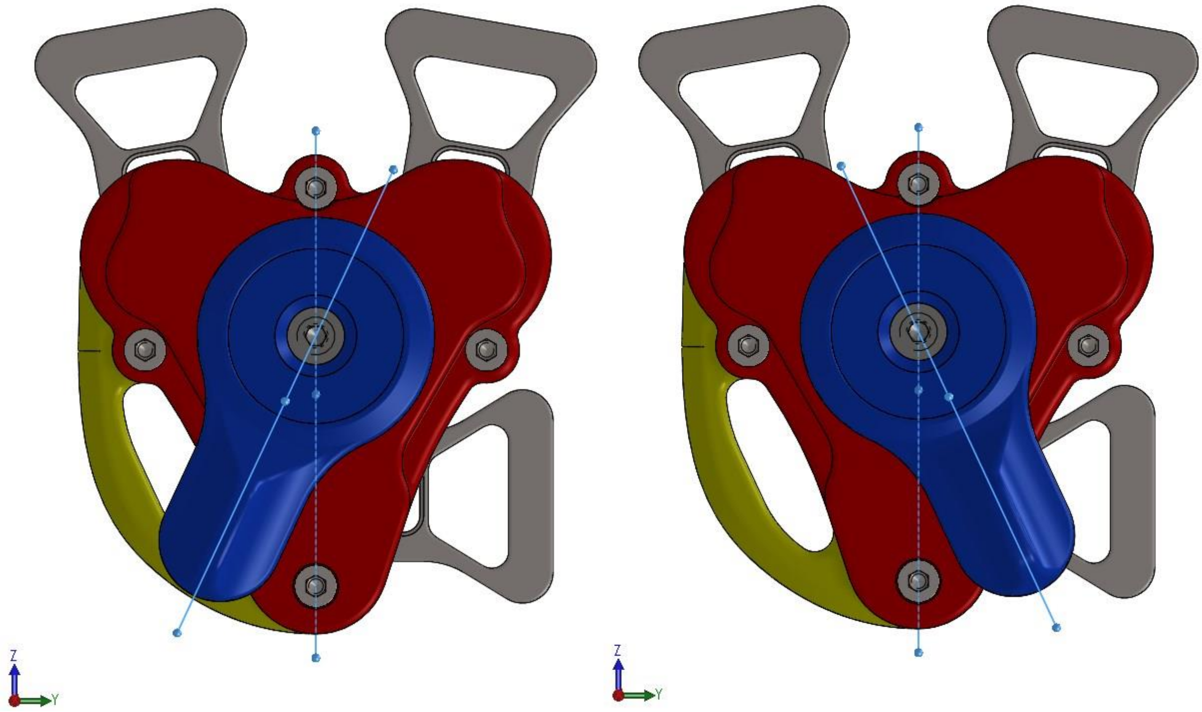

Figure 2.5.1 – Pre-release free movement, 25° each side.

Specifications:

- the rigid part of the safety harness, such as buckles, adjusting devices or attachments shall not have sharp edges liable to cause wear or breakage of the sharps by chafing;
- the contact area of the buckle with the body of the wearer shall be between 20 cm^2 and 40 cm^2. Figure 2.5.2 shows the contact area of the super light buckle which is 20,03 cm^2;

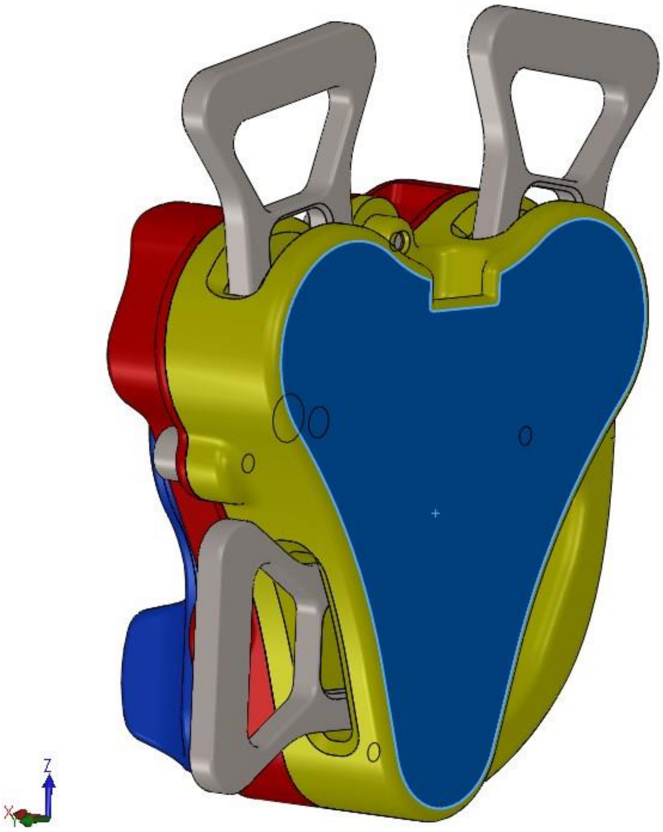

Figure 2.5.2 – Contact area between the buckle and the body of the driver.

- on ZY plane, the buckle lever edges shall not extend more than 10 mm from the edge of the buckle surface in contact with the driver's body. Not applicable if the buckle covers the entire projected surface of the lever (Figure 2.5.3);

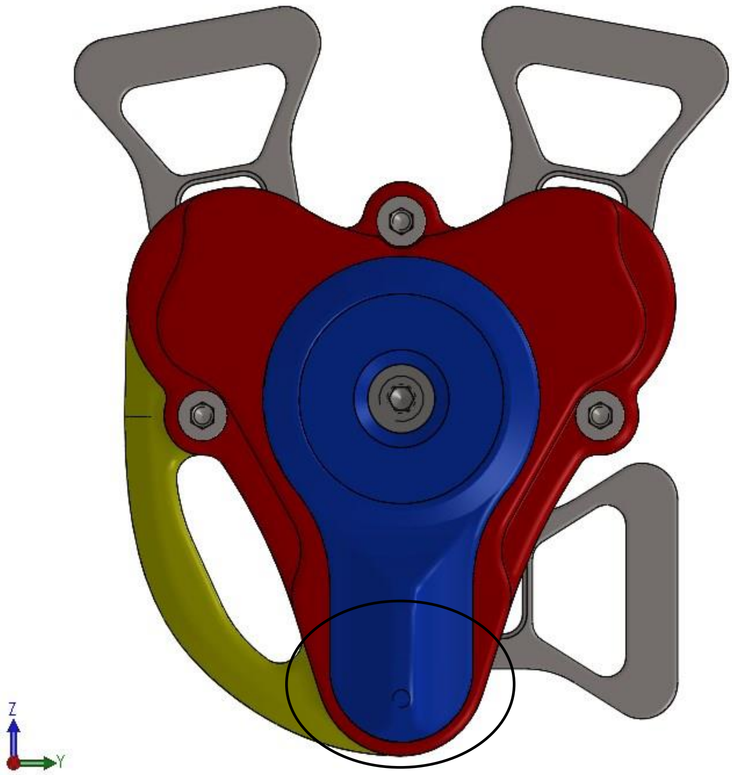

Figure 2.5.3 – Upper body covers the entire projected surface of the lever.

- external edges filleted with a radius ≥ 0,065 mm;
- activation surface of the lever at least 2 cm^2 (Figure 2.5.4);

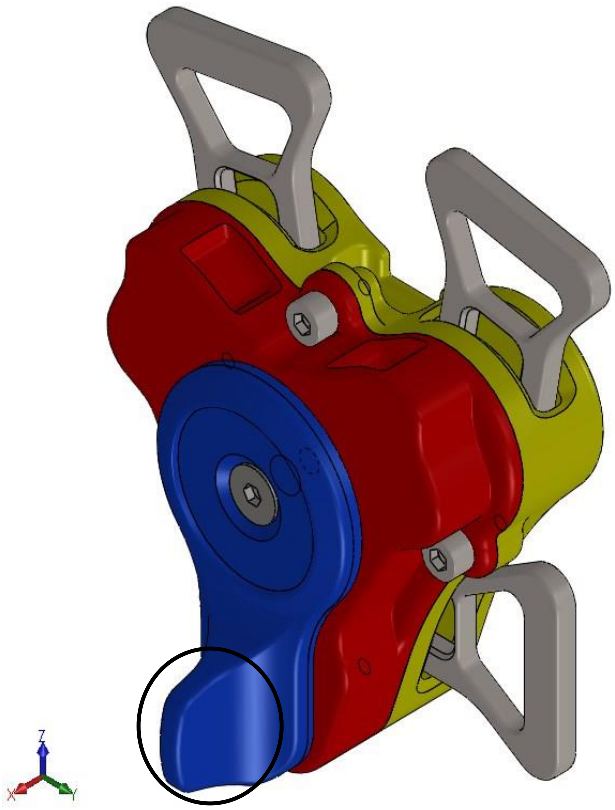

Figure 2.5.4 – Focus on the activation area of the lever.

- loop of the tongues with a rugosity of 1,6 mm, to avoid wear issued between latch and tongue;
- tongues must hook with a click;
- the fastening of the tongues must be guaranteed with the buckle in the rest position;
- once the buckle is opened after the lever rotation, it must keep his position. This is necessary to reduce partial hooking of the tongues;
- the breaking load of each attachment is 15 kN if it is linked to one strap. It increases to 30 kN in case the straps are two;
- the torque required for the opening of the buckle shall be between 2 Nm and 4 Nm; the force required to obtain it shall be between 50 N and 100 N.

The super light buckle follows all FIA specifications which are useful as inputs for the static analysis of the buckle in Solidworks.

2.6 Hertz

Heinrich Hertz proposed his theory to evaluate the contact between two bodies pressed against each other through curved surfaces. He realized that because of the elastic behaviour of the materials, under these conditions the contact does not take place at one point or along a line, but rather on a finite area which is linked to a calculable contact stress.

The theory of Hertz is based on seven fundamental hypotheses:

- surfaces are continuous and non-compliant (the starting contact is a point or a line);
- surfaces can be represented by second order polynomial before deformation;
- isotropic and homogenous solids;
- elastic deformations and respect the Hooke's law;

- small deformation
- the contact area is small compared to the radius of curvature of the bodies involved;
- no friction between surfaces: only the normal force has an effect.

To calculate the contact stress, it is necessary to evaluate for each body in contact the two "principal curvatures", the maximum and the minimum curvatures varying the section plane.
Figures 2.6.1 shows an example of the principal curvatures calculated for a generic body.

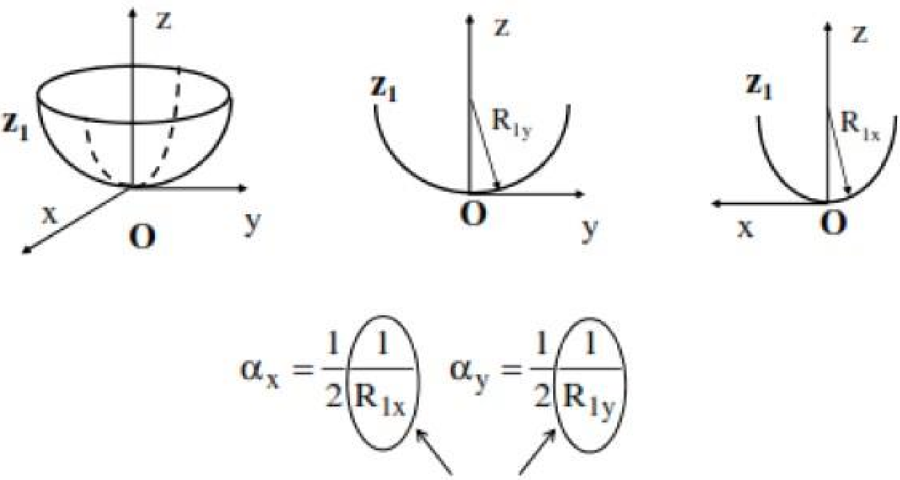

Figure 2.6.1 – Example of the principal curvatures for a generic body.

It is important to consider the following convention for the sign of these curvatures: a principal curvature is considered positive if the centre of curvature stays on the part of the material, otherwise it is considered negative.
Figure 2.6.2 shows this convention of sign.

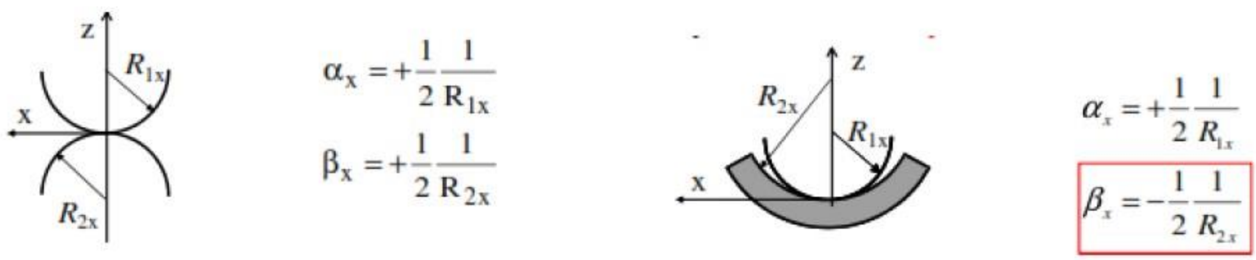

Figure 2.6.2 – Convention of sign for principal curvatures.

In this study the contact occurs between the camshaft and the sphere, hence it can be considered the simplified case of a contact between a sphere and a plane. In a generic case the contact should be a circumference, but under this hypothesis the contact occurs on a square area. Therefore, the curvature of the plane is zero, because the two radiuses are infinite.
In Figure 2.6.3 there is a simplified representation of the contact.

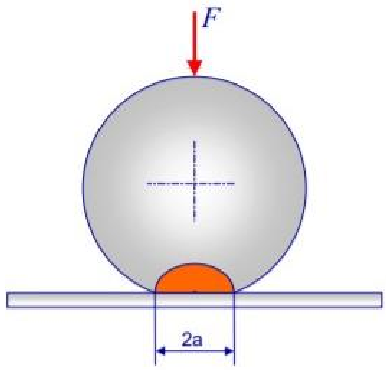

Figure 2.6.3 – Example of contact between a sphere and a plane.

The mathematic equation to calculate the pressure between the two bodies is

$$p_{max} = \frac{3}{2}\frac{F}{\pi a^2}$$

where F represents the force on the camshaft imposed by the rotation of the lever and the length a could be obtain by

$$a = \sqrt[3]{\frac{3}{4}FR\left(\frac{1-\upsilon_1{}^2}{E_1} + \frac{1-\upsilon_2{}^2}{E_2}\right)}$$

R is the radius of the sphere while E and ν are the proprieties of materials.
In Figure 2.6.4 has been represented the contact between the camshaft and the sphere.

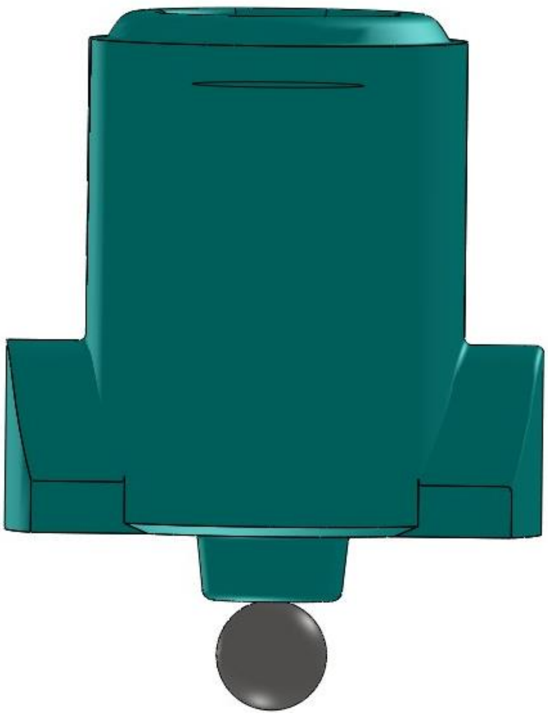

Figure 2.6.4 – Contact between the sphere and camshaft.

Moreover, near the contact area the compressed material tends to expand transversely, but the expansion is prevented by the surrounding material which operates as a constraint. Hertz demonstrated the maximum stress is not on the contact surface, but inside the material at a distance z from the surface.

3 Interference detection

After the definition of the CAD model, it is necessary to prepare it for the static analysis, therefore the assembly should be analysed from a geometrical point of view.
The parts should not guarantee the interpenetration each other because, in that case, Solidworks could not create the mesh, it would not be continuity along the surfaces.
So, through the command "Interference detection", Solidworks shows the interferences of the components for the whole assembly, as it is possible to see in Figure 3.1.

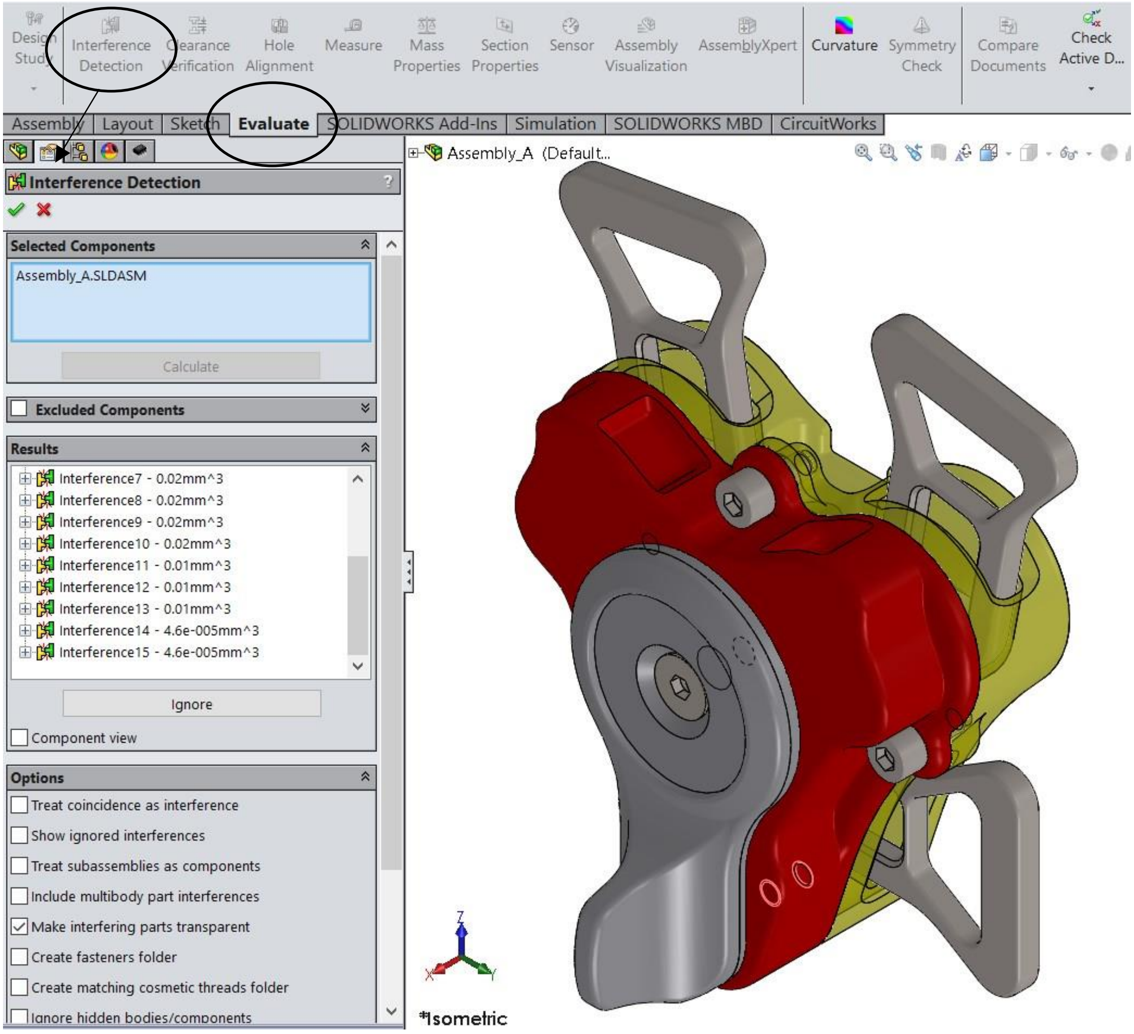

Figure 3.1 – Detection of the interferences among buckle parts from the model A in Solidworks.

The Figure 3.1. shows, in the window "Results", many interferences with their value, so it is necessary to examine each one by one.

3.1 Lower body - Screw

The first hindrance is represented by the interpenetration between the holes of the lower body and the thread of the four screws, as it is shown in Figure 3.1.1.

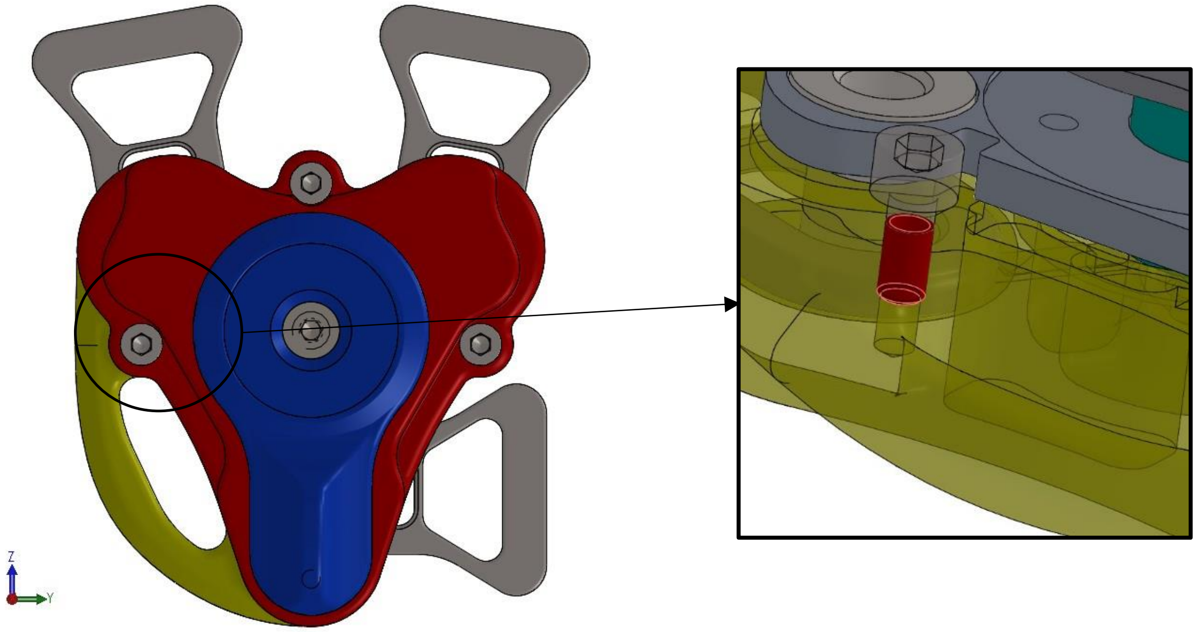

Figure 3.1.1 – Zoom of the interference between the thread of the screws and the lower body.

This issue could be fixed acting on the hole, increasing his diameter, or acting on the screw, reducing the diameter of the thread.
In particular, the hole has been widened from 2,5mm to 3mm and has been done for each matching.
Moreover, the screws report another problem which comprehend the head (Figure 3.1.2).

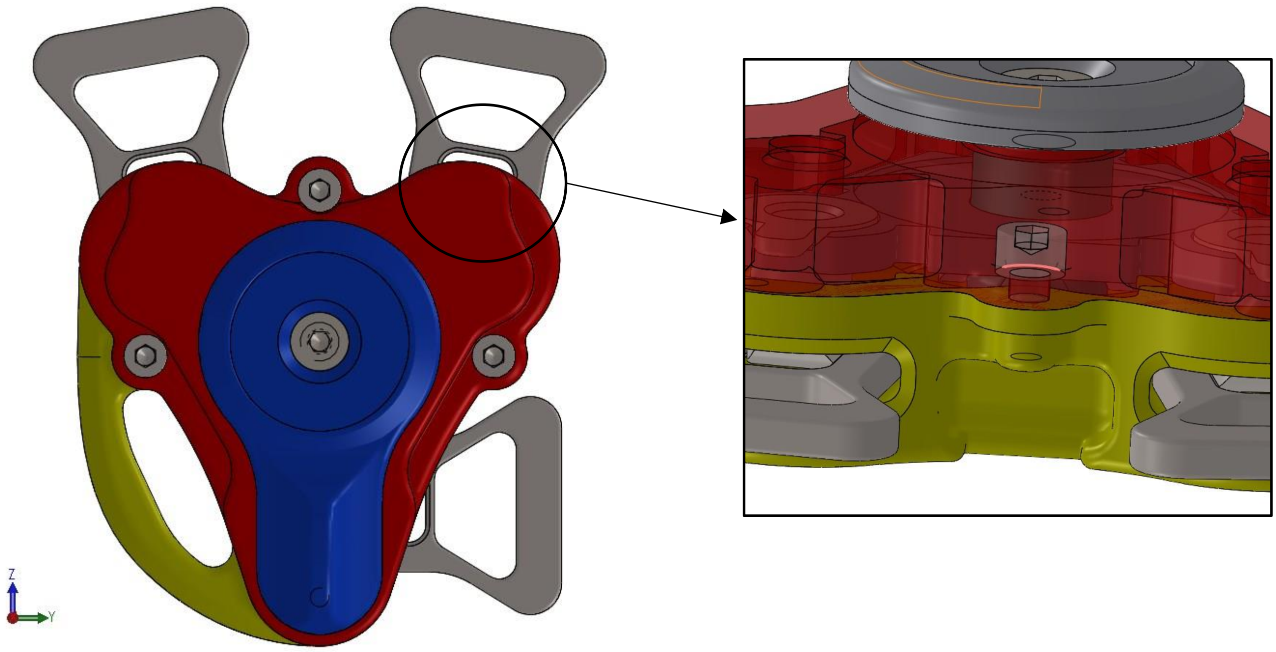

Figure 3.1.2 – Zoom of the interference between the head of the screws and the lower body.

The way to eliminate the interpenetration is to change the fillet on the lower edge of the screws, from 0,1mm to 0,2mm.

3.2 Camshaft - Lever

The contact between the lever and the camshaft is critical because there are two interferences, as the Figure 3.2.1 shows.

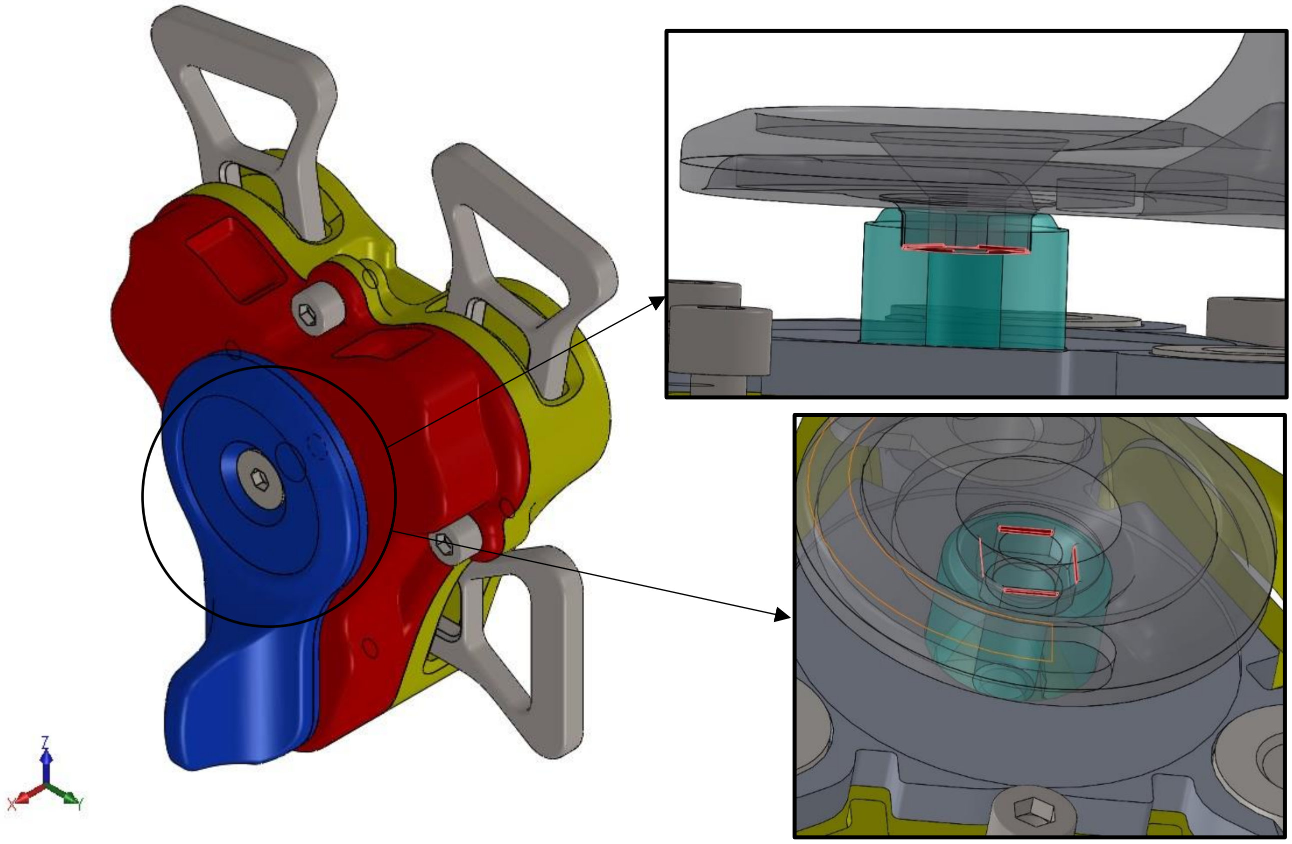

Figure 3.2.1 – Zoom of the interference between the lever and the camshaft.

To avoid these problems:

- camshaft housing has been increased of 0,5mm;
- on the edge of the lever in contact with the camshaft has been done a 0,3mm fillet.

3.3 Lifting plate - Latch

The last problem is characterised by the contact between the latches and the lifting plate (Figure 3.3.1)

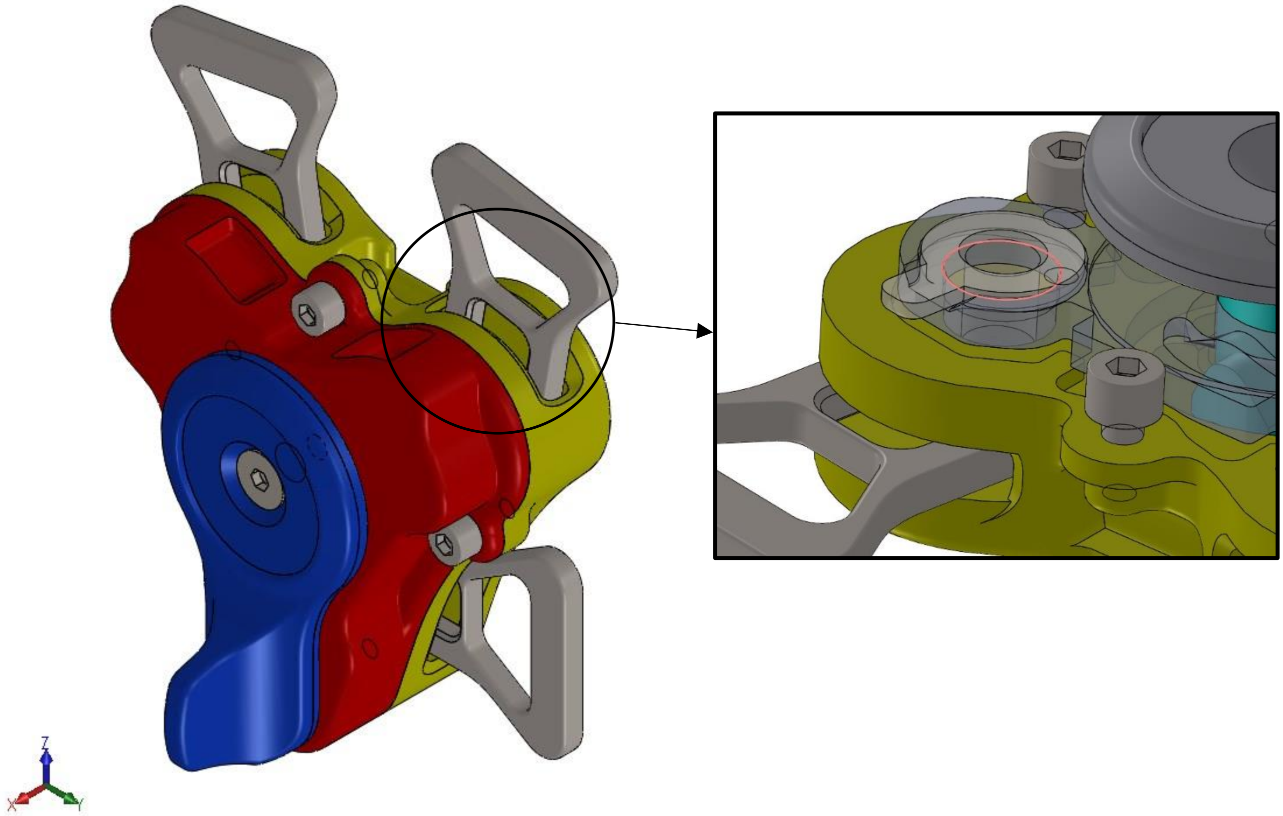

Figure 3.3.1 – Zoom of the interference between a latch and the lifting plate.

As it has been done for the other interference, the way is to use a fillet on the edge of the plate, his value is 0,3mm.

These operations made on the assembly have the target to prepare it for the FEM analysis. These adjustments are necessary to create the mesh because it needs continuity among surfaces.
Therefore, there will be two model, one for the CAD, one for the FEM.

4 FEM model

4.1 Definition

The Finite Element Method is a fully used method for numerically solving partial differential equations arising in engineering and mathematical modelling. His fields of application are structural analysis, heat transfer, fluid flow, mass transfer and electromagnetic potential.
To solve a problem, the FEM divides a large system into smaller equal parts which are called finite elements (triangles and quadrilaterals for 2D domains, hexahedrons and tetrahedrons for 3D domains). This is achieved by a particular space discretization in the space dimensions, which is implemented by the construction of a mesh of the object. The mesh is the numerical domain for the solution which has a finite number of points, called nodes, as you can see in Figure 4.1.1.

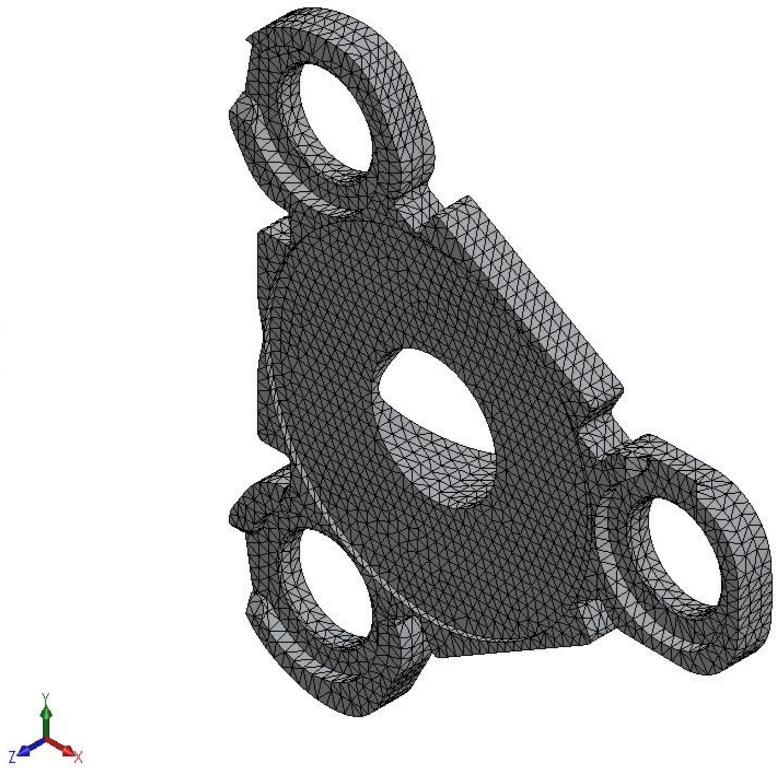

Figure 4.1.1 – Mesh of the lifting plate of the buckle.

The subdivision of a whole domain into simpler parts has many advantages:

- accurate representation of complex geometry;
- easy representation of the total solution;
- capture of local effects.

Moreover, the typical functioning of the method involves:

- dividing the domain of the problem into a collection of subdomains which are represented by a set of element equations to the original problem;
- recombining all sets of element equations into a global system of equations for the final calculations.

4.2 Analytic formulation

To find the solutions of the problem, it is necessary to define a method and a mathematic model. Starting from the formulation of the first one:

- breaking down of the structure/component;
- mathematical characterization of the object;
- formulation based on "imposed displacements" which means:
    - characterization of the elements, so mathematical description of the element depending on equilibrium conditions and deformation;
    - building of the structure, so mathematical formulation of the equations which describe the element belonging to the structure;
    - solution of equation system.

Instead, talking about the mathematic model, it explains the object can be described through three technics:

- wire frame: description through contour lines. The technic is immediate, but it does not provide information about mechanical proprieties of surfaces and it does not distinguish internal and external volumes;
- surfaces: mathematical description of surfaces which define objects. Tough evaluation of volume and mass;
- solids: description of the volumes took up by objects. Geometric, volume and mass proprieties are obtained automatically.

The finite element method leads to write a general equation that the software will solve for each node

$$\mathbf{Kx} = \mathbf{f} + \mathbf{R}$$

Although this equation has been theoretically obtained in the case of linear analysis, it represents the starting point for the solution of all analyses. Indeed, it can be observed even a buckling analysis and a modal analysis with internal stress can be traced to this equation considering an additional term Δk which corresponds to a variation of the stiffness matrix of the elements due to the internal stress state introduced.

4.3 Linear static analysis

The generic model for a static matrix problem is:

$$\boldsymbol{K_g x_g} = \boldsymbol{f_g} + \boldsymbol{R_g} \qquad (1)$$

where the following hypothesis are considered:

- no boundary conditions are applied, hence external forces $\mathbf{f_g}$ and reaction forces $\mathbf{R_g}$ are balanced;
- $\mathbf{x_g}$, $\mathbf{f_g}$, $\mathbf{R_g}$ are real vectors (g x 1);
- $\mathbf{K_g}$ is the global stiffness matrix (g x g) real, symmetric and positive semidefinite, to allow rigid body motions or stable elastic deformations.

The first step is to apply:

- Rigid Joints (RJs)

$$K_{RJs} = T_{RJs}{}^{T} K_g T_{RJs}$$

which is (j x j) with T_{RJs} (g x j);

- Rigid Body Element (RBE):

$$K_{RBE} = T_{RBE}{}^{T} K_{RJs} T_{RBE}$$

which is (b x b) with T_{RBE} (j x b);

- Boundary Conditions (BCs):

$$K = T_{BCs}{}^{T} K_{RBE} T_{BCs}$$

which is (n x n) with T_{BCs} (b x n);

Moreover, now it needs the vector of solutions of the system and to gain it, it is necessary to analyse each transformation:

$$x_g = T_{RJs} x_{RJs} \qquad (2)$$

where

$$x_{RJs} = T_{RBE} x_{RBE} \qquad (3)$$

$$x_{RBE} = T_{BCs} x \qquad (4)$$

and replacing the eq. (3) and (4) in the eq. (2), the result is:

$$x_g = T_{RBE} T_{RJs} T_{BCs} x$$

where

$$T = T_{RBE} T_{RJs} T_{BCs}$$

so, it is possible to obtain the solution:

$$x_g = Tx$$
$$x = (T^T T)^{-1} T^T x_g$$

Therefore, eq. (1) is reduced to active degrees of freedom, applying rigid joints, rigid body elements and boundary conditions. Eventually master/slave and Guyan reduction are secondary applied.
The generic model for a static matrix problem is reduced to:

$$\mathbf{Kx} = \mathbf{f}$$

where:

- **x** is the unknown vector of the system deformed shape (n x 1) with the overall dofs n of the system (active and constrained dofs);

- **K** is the stiffness matrix (n x n) sparse, symmetric, semipositive definite (semipositive property is due to the unconstrained form, when boundary conditions are applied, it must be positive definite) with $n < g$;
- **f** is the vector of the external generalized forces (n x 1) applied to the system with $n < g$;

4.4 Simulation

The next step is to analyse the buckle in Solidworks, through a static point of view. "Simulation" is the command which allows that and it needs many inputs.
Firstly, it is necessary to start a "New Study", as the Figure 4.4.1 shows, and to choose the type of study: static, frequency, thermal, fatigue, ecc.

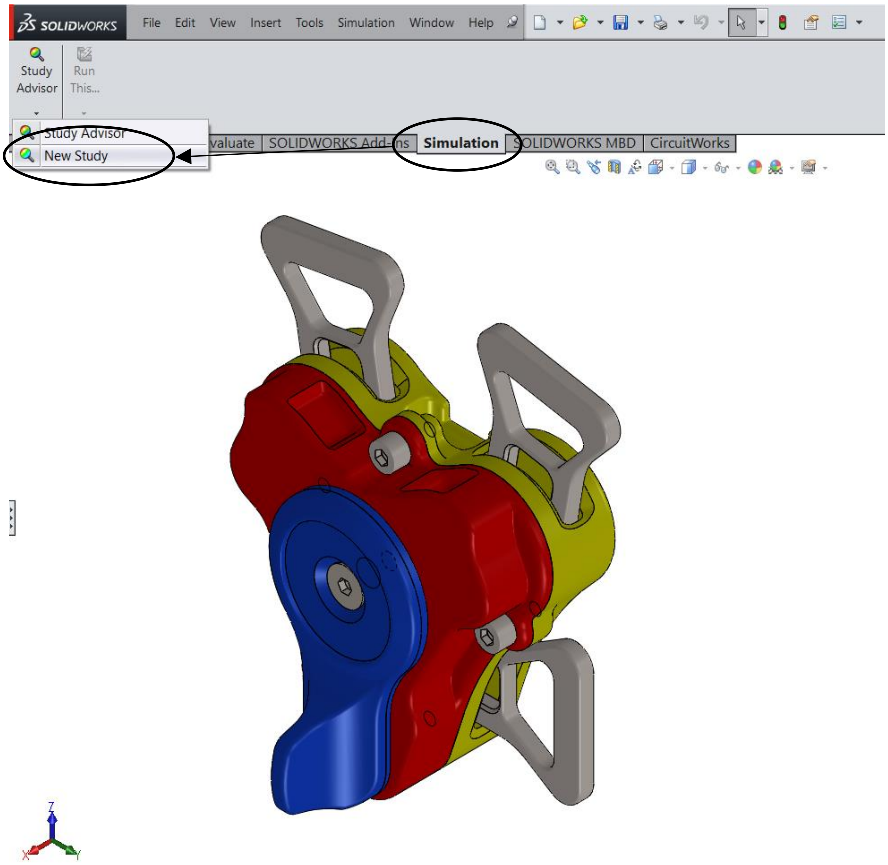

Figure 4.4.1 – Command to start a new study in Simulation.

In this case, the static one, it's useful to follow a plan and define:

- materials;
- fixtures;
- external loads;
- mesh;
- connections.

Afterwards, it's necessary to run Solidworks, hence it can calculate the solution of displacements and stress through the analytic approach shown in Chapter 4.3.

4.4.1 Materials

The first step to run a "Simulation" in Solidworks is the choice of materials. This is fundamental to define the physical properties of each component, particularly, for the static analysis:

- **density** [kg/m^3], which directly affects the mass, so the weight force;
- **stiffness matrix**, which describes:
  - elastic modulus [MPa]. It is a quantity which measures an object's resistance to being deformed elastically when a stress is applied to it;
  - poisson's ratio. It is the deformation of a material in directions perpendicular to the specific direction of loading.

$$K = \begin{bmatrix} E \\ \nu \end{bmatrix}$$

- **yield strength**. It is the point on a stress-strain curve which indicates the limit of elastic behaviour and the beginning of plastic behaviour.

Even though many other proprieties of materials are useful, as thermal expansion coefficient, it is sufficient to focus on the three listed upon because they affect the result in linear static analysis: the displacement and the stress.

4.4.2 Fixtures

Now it is useful to set up the fixtures of the assembly which are necessary to define the service environment of the model, indeed, they directly affect the analysis. They are applied to geometric entities as features which are fully associative to geometry and automatically adjust to geometric changes. Moreover, the fixtures of an assembly are different form the part ones, they own three characteristics:

- to be not ideals;
- to be difficult to apply;
- to be neither pejoratives nor preventives.

From a technical point of view, they could be described as features which allow to prescribe zero or non-zero displacements on vertices, edges or faces.
Basically, there are three standard types:

- fixed geometry which locks all degrees of freedom of the interested surfaces, in this case the translational ones;
- roller/slider which specifies that a planar face can move freely into its plane but cannot move in the direction normal to its plane;
- fixed hinge which specifies that a cylindrical face can only rotate about its own axis. The radius and the length of the cylindrical face remain constant under loading.

It is possible to set them through the option *Fixture* and Figure 4.4.2.1 shows the dashboard where you must choose the kind and the surfaces engaged.

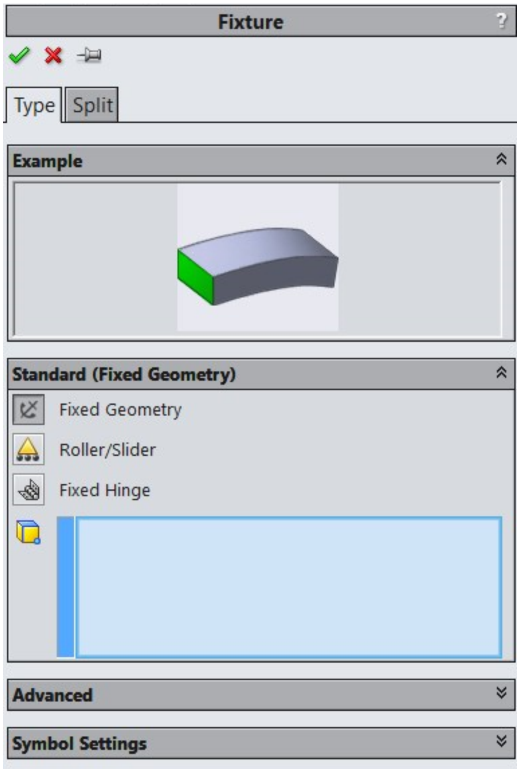

Figure 4.4.2.1 – Fixture dashboard.

In Figure 4.4.2.2 has been represented an example, applied to the buckle, where the lower body has a fixed geometry on the lower base, so the assembly cannot move.

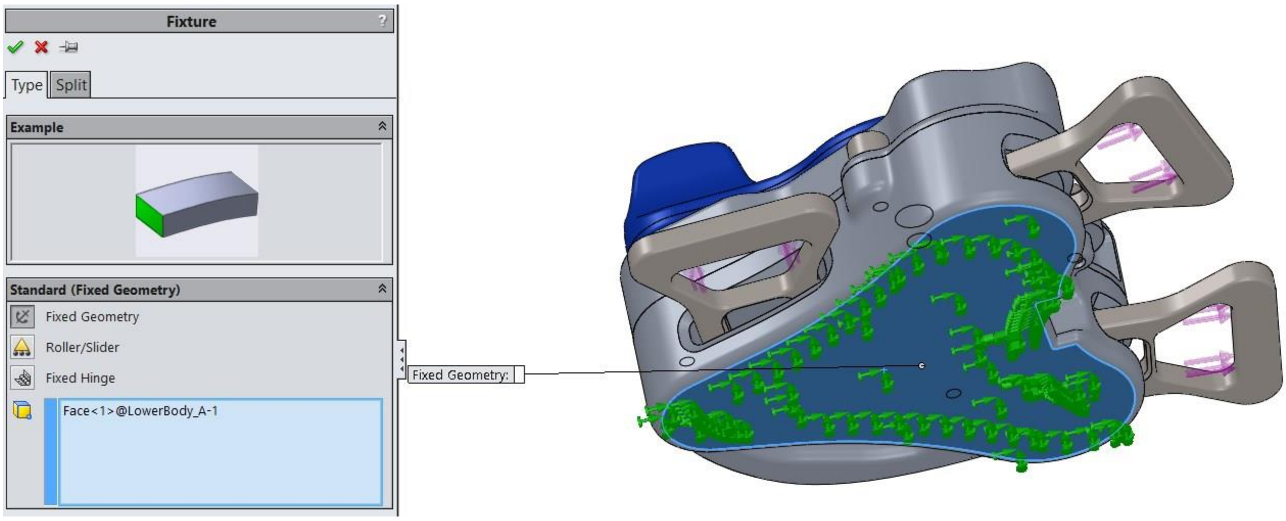

Figure 4.4.2.2 – Example of standard fixture.

Apart from the standard fixtures, there are many others and for the buckle it is useful to focus on the elastic support: it represents the fixture as a spring. This has been necessary to simulate the real environment of the safety buckle, indeed, the contact between the driver's stomach and the lower body cannot be represented through a slider. Moreover, the reaction of the fixed strap can be represented through this fixture, using a different stiffness than the lower body.
Figure 4.4.2.3 shows the dashboard of the elastic support.

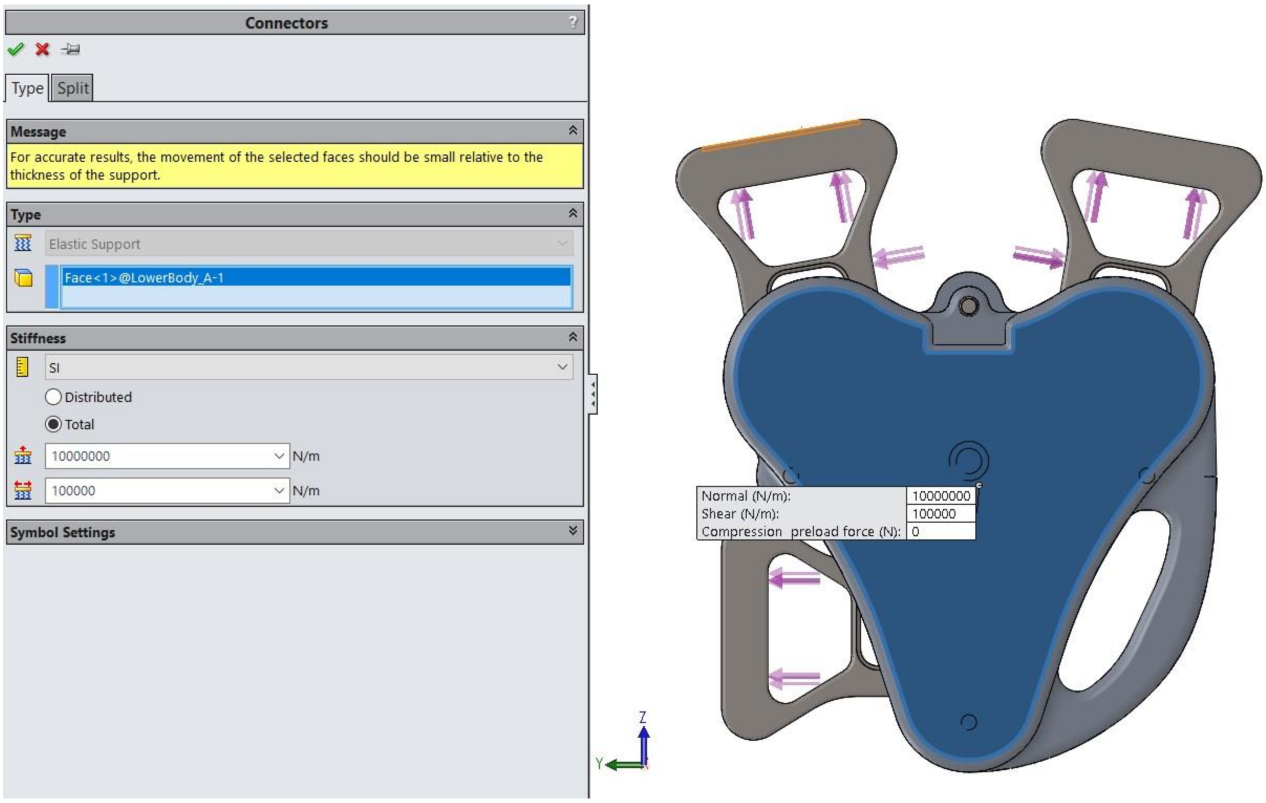

Figure 4.4.2.3 – Example of elastic support.

After the selection of the surface, it is necessary to choose:

- longitudinal stiffness;
- transversal stiffness.

4.4.3 External loads

The next step to get a right setup is the definition of loads which are forces or torques with uniform distribution to faces, edges, reference points, vertices and beams in any direction for use in structural studies. The dashboard of loads is represented in Figure 4.4.3.1, and it is necessary to choose:

- the kind;
- the surface engaged;
- the intensity;
- the direction.

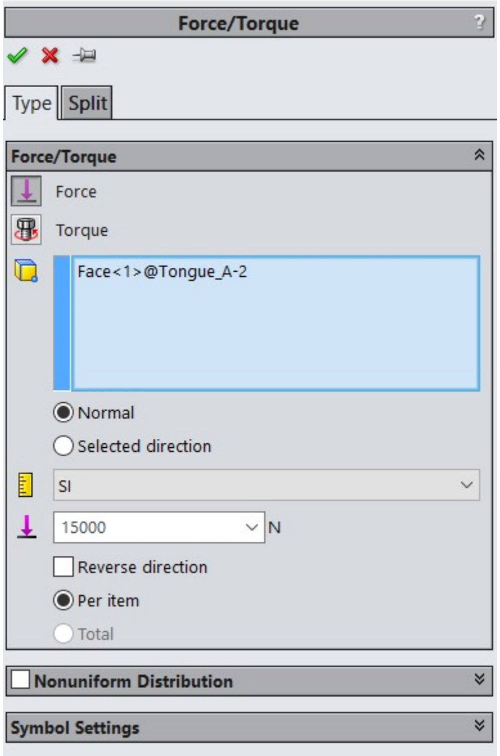

Figure 4.4.3.1 – Load dashboard.

In Figure 4.4.3.2 has been represented an example of loads, applied to the tongues of the buckle.

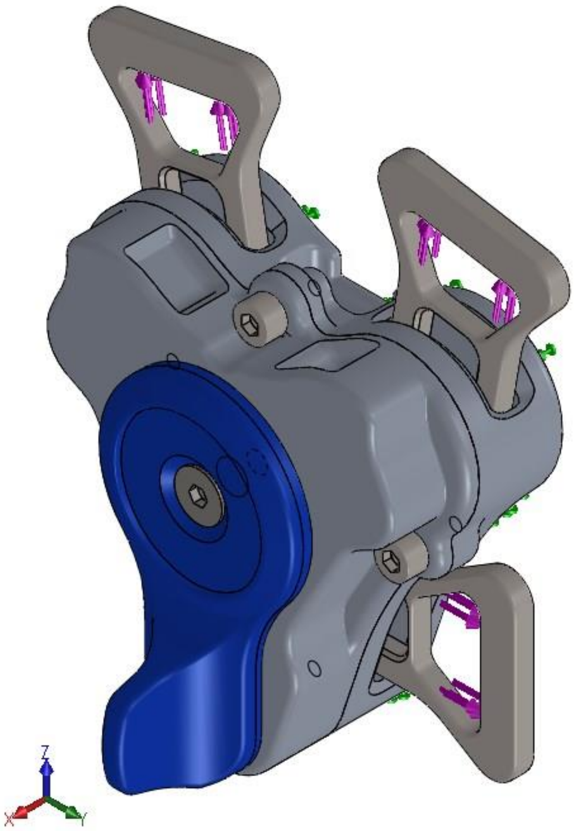

Figure 4.4.3.2 – Example of load.

4.4.4 Mesh

Another step to generate a Finite Element Method simulation is the mesh which is, for definition, a network formed of cells and points. The process starts with the creation of a geometric model. Then, Solidworks subdivides the model into small pieces of simple shapes, elements, connected at common points, nodes. Finite element analysis programs look at the model as a network of discrete interconnected elements.
In Figure 4.4.4.1 there is an example of mesh, got using the lower body of the buckle.

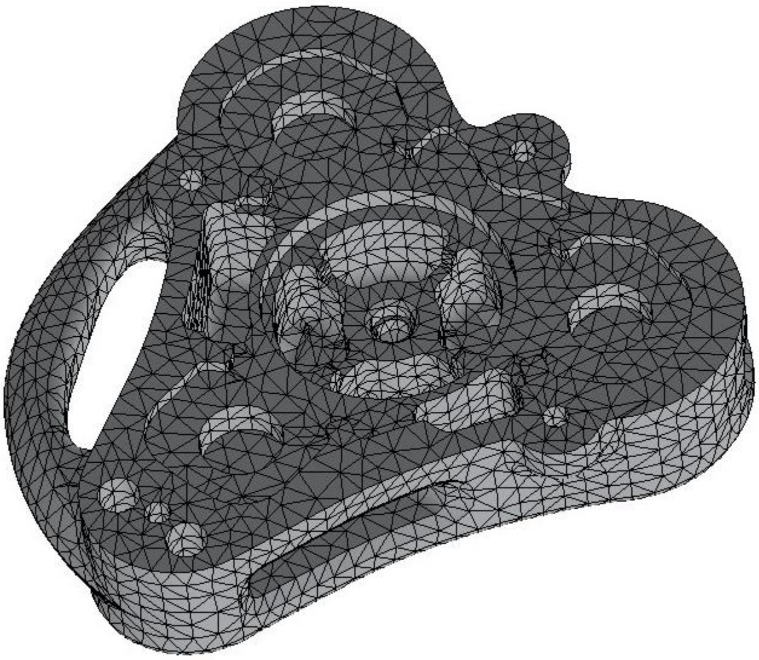

Figure 4.4.4.1 – Example of mesh, using the lower body as sample.

In Figure 4.4.4.1, it easy to see the number of elements which compose the mesh, indeed there is a huge number of tetrahedrals which represent the cells and they are connected by nodes.
The FEM predicts the behaviour of the model by combining the information obtained from all elements making up the model.
In design analysis, meshing is a very crucial step. Solidworks generates a mesh based on a global element size, tolerance and local control mesh specifications. Mesh control lets you to specify different sizes of elements for components, faces, edges and vertices.

4.4.4.1 Solid mesh

When Solidworks meshes a part or an assembly with solid elements, it generates one of the following types based on the active mesh options for the study:

- **first order**, characterised by linear tetrahedral solid elements;
- **second order**, characterised by parabolic tetrahedral solid elements.

The main difference between them is the structure, the first one is defined by four corner nodes connected by six straight edges, instead, the second one is composed by four corner nodes, six mid-side nodes, and six edges. The Figure 4.4.4.1.1 shows schematic drawings of the two types of mesh.

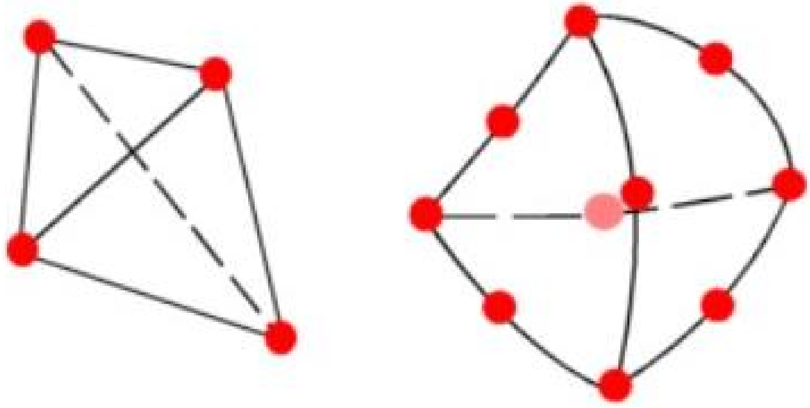

Figure 4.4.4.1.1 – Linear elements on the left, parabolic elements on the right.

Moreover, fixing the number of elements, so the mesh density, parabolic elements yield better results than linear because:

1. they represent curved boundaries more accurately, indeed, as the Figure 4.4.4.1.1 shows, there is a midpoint on each edge;
2. they produce better mathematical approximations. Stress and displacement are clearer and more accurate.

However, parabolic elements require greater computational resources than linear elements.

In Solidworks:

- the option draft quality mesh represents the first order, also called Tetra4, and it is usually used for quick evaluation or where bending effects are small;
- the option high quality mesh is used for the second order, also called Tetra10.

The choice of this option is fundamental because represents the density of the mesh which affects number of nodes and number of the elements. But, before talking about density, it is necessary to explain the convergence which determines how many elements are required in a model to ensure that the results of an analysis are not affected by changing the size of the mesh.
At least three points need to be considered, and as the density of the mesh increases, the quantity of interest starts to converge to a particular value.
There are two types of refinements (Figure 4.4.4.1.2):

- **h-method**: decreasing the characteristic length (h) of elements, dividing each existing element into two or more element without changing the type of elements used. In essence, it consists in decreasing the size of elements, increasing their quantity;
- **p-method**: increasing the degree of polynomials (p) without changing the number of elements used. Hence, to keep the same number of elements using a Tetra10 instead of a Tetra4.

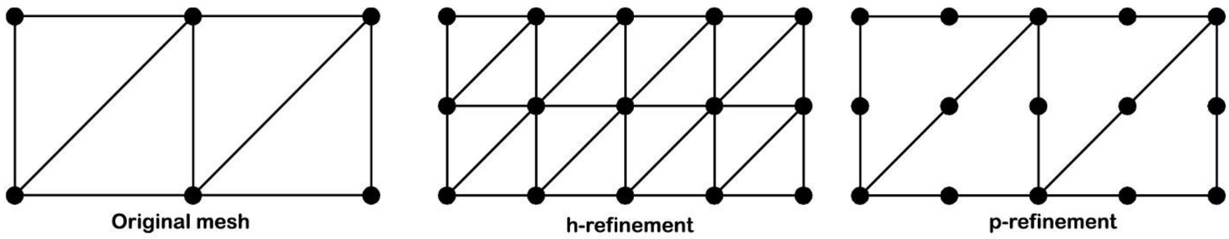

Figure 4.4.4.1.2 – h-method and p-method, shown through a simple scheme of 2D mesh elements.

After the definition of these methods, it is useful to analyse them on Solidworks:

- changing high quality mesh to draft quality mesh, it occurs the p-method;
- changing the density of mesh, from coarse to fine, it occurs the h-method because the characteristic dimension decreases, as element side or his height.

In Figure 4.4.4.1.3, it is shown the mesh density control panel and the characteristics dimensions of each element of the mesh which affect the density.

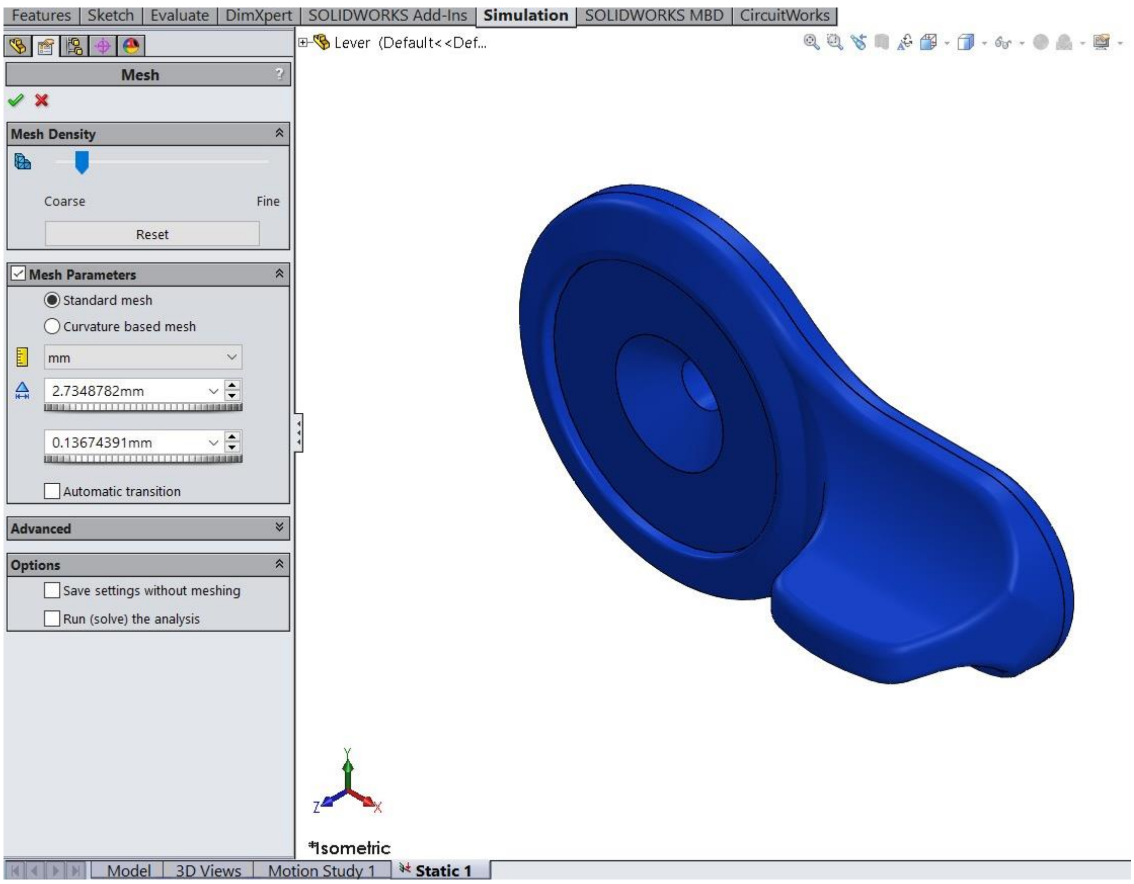

Figure 4.4.4.1.3 – Mesh density dashboard.

There two ways to reach convergence:

- moving the cursor of the density mesh control panel;
- changing the value of height or side of the singular mesh element.

To better understand this concept, it is useful to report many mesh results, obtained with different configurations.

- H-method. Setting high quality mesh, the mesh density has been changed from coarse to fine. The Figure 4.4.4.1.4 shows the aim of the h convergence: the number of elements increases while their size reduces.

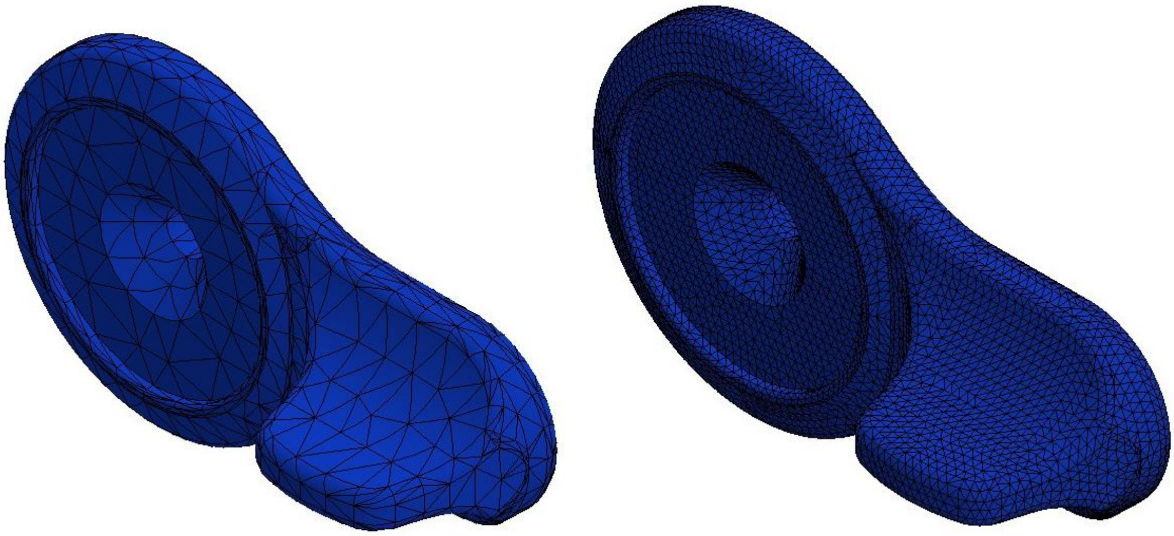

Figure 4.4.4.1.4 – Coarse density at left, fine density at right.

Moreover, the Table 4.4.4.1.5 shows how the number of elements increases and the size of the singular elements is reduced, changing the density from coarse to fine.

Table 4.4.4.1.5 – Table of mesh density. Mesh 1 means coarse until mesh 7 which means fine.

Mesh attempt	Number of nodes [-]	Number of elements [-]	Element size [mm]
1	8360	4564	3,039
2	8825	4777	2,773
3	10911	6022	2,279
4	17879	10305	1,671
5	31219	18822	1,235
6	59721	37518	0,931
7	99266	64075	0,759

The behaviour of the h-method could be represented from a graphical point of view too, indeed, the Figure 4.4.4.1.6 and the Figure 4.4.4.1.7 show it. It is easy to see the huge grown of the number of elements and nodes using a better refinement of the mesh, at the same time, there is a reduction of the size of each element.

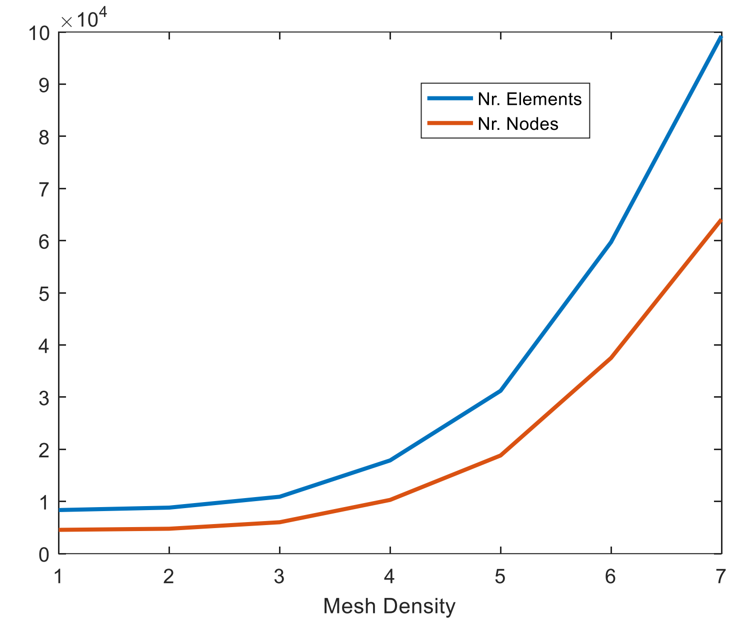

Figure 4.4.4.1.6 – Plot of the number of elements and the number of nodes refining the mesh.

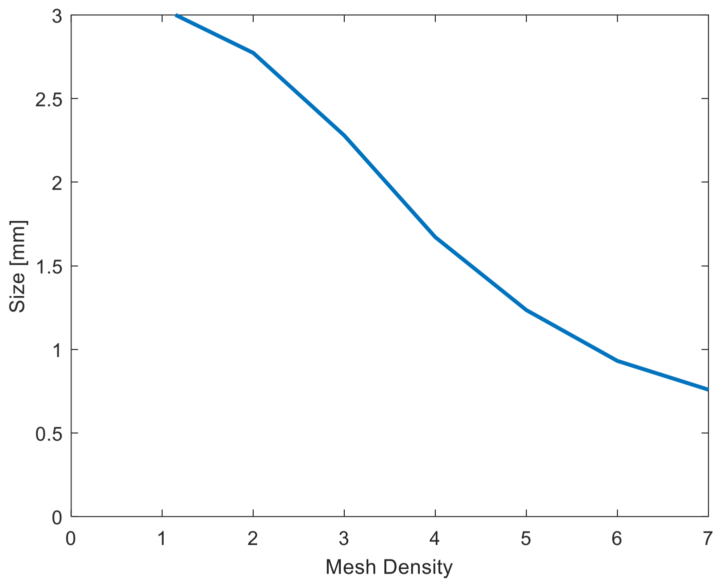

Figure 4.4.4.1.7 – Plot of the size of the singular element refining the mesh.

- P-method. Changing the mesh from high quality mesh to draft quality mesh, hence, passing from Tetra10 to Tetra4, the number of elements is always the same, but the number of nodes decreases. In Figure 4.4.4.1.8 it is represented the same component with Tetra4 and Tetra10 mesh. There is a zoom of the latch and the surfaces obtained with Tetra10 are more defined and smoother than Tetra4, as there is a midpoint between each edge. In addition, they have 7616 elements both, but the mesh obtained with Tetra4 has 1858 elements, the one with Tetra10 12445 elements.

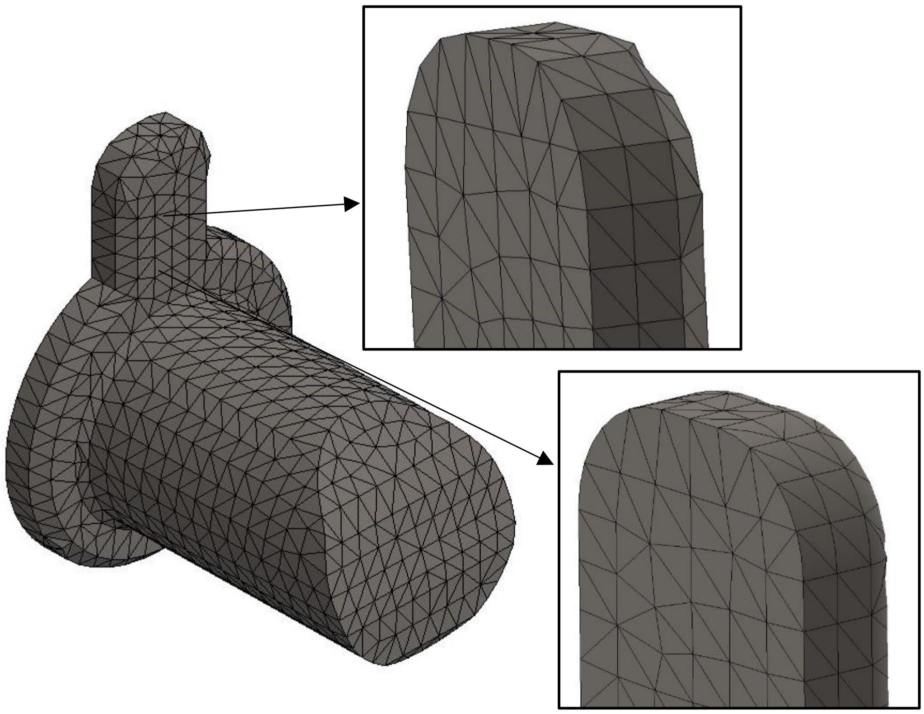

Figure 4.4.4.1.8 – Latch mesh to show the difference between Tetra4, on bottom, and Tetra4, on top.

Solidworks also gives the possibility to build a mesh with zones which are more thickened than others (Figure 4.4.4.1.9). This could be useful to mesh surfaces characterised by curves, indeed, through the command Curvature Based Mesh, the software thickens the zones where the fillet radius is bigger.

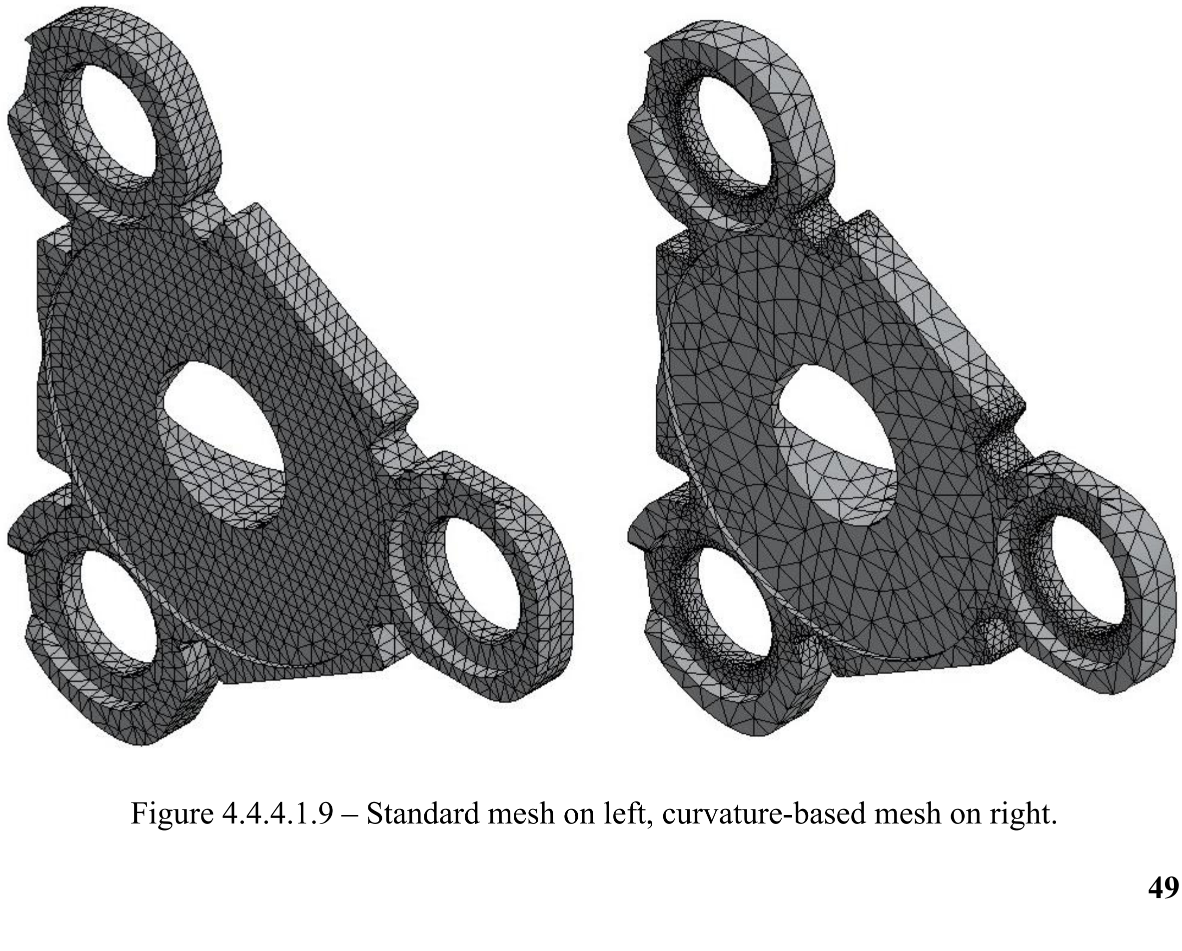

Figure 4.4.4.1.9 – Standard mesh on left, curvature-based mesh on right.

Usually, the standard mesh is used for object with prismatic form because there are not fillet.

The last step to conclude this chapter is the compatibility of the mesh.
The terms *compatible* and *incompatible* mesh refer to the type of mesh continuity on touching geometrical entities that belong to different bodies. Indeed, they apply to the mesh of assemblies, not to single parts.
In a compatible mesh, touching entities are meshed so that there is a node-to-node correspondence between the mesh of each entity.
Nodes in correspondence can be merged or superposed. For a no penetration contact, node to node contact elements is created between the coincident nodes on the source and target faces. In the Figure 4.4.4.1.10 the orange arrow indicates one of the touching nodes between lower and upper body.

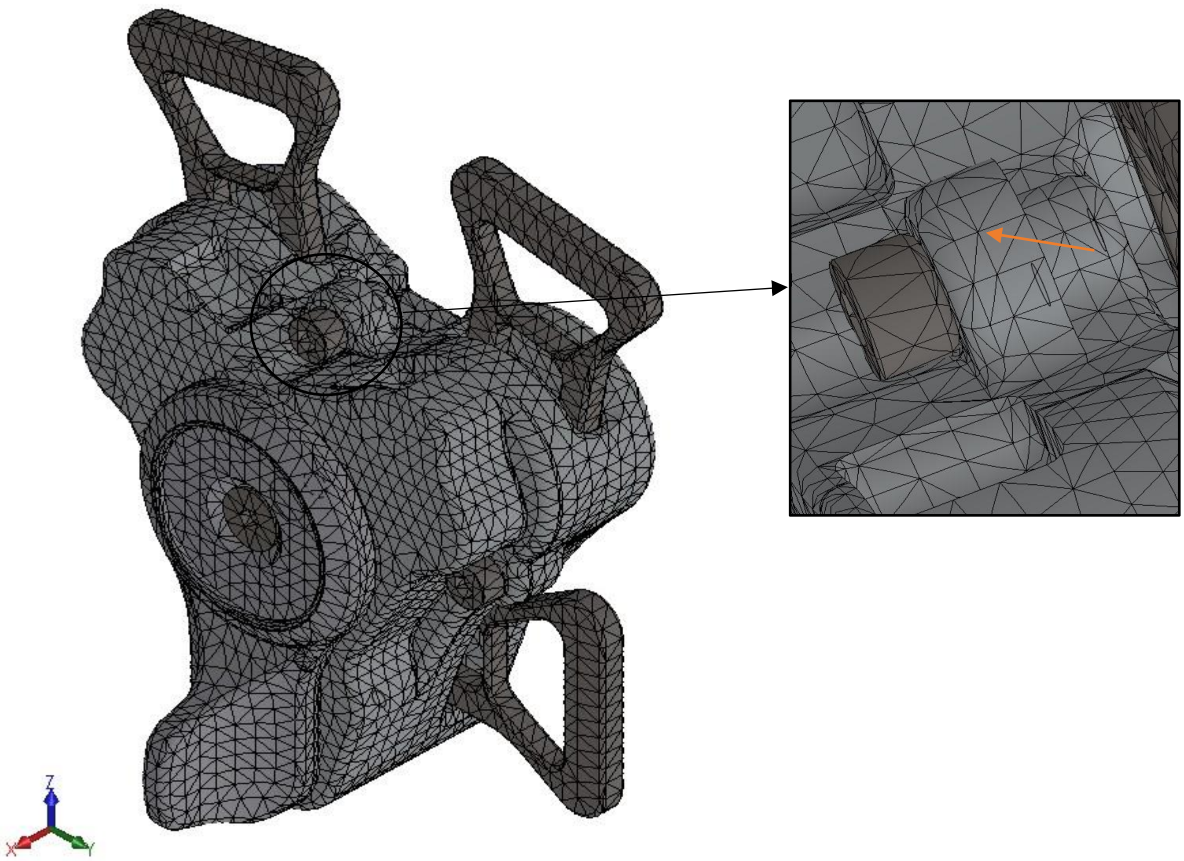

Figure 4.4.4.1.10 – Zoom of lower and upper body contact to show the meaning of compatible mesh.

Moreover, results based on a compatible mesh are more accurate that those based on an incompatible mesh.

4.4.5 Connections

The next step to complete the setup for the simulation concerns the contact analysis. This one describes the interaction among part boundaries which are in contact, or which come into contact during the loading. Any change in connections requires re-meshing of the model. For the buckle model it is necessary to set the component contact, the contact sets and the spring connectors.

4.4.5.1 Component contact

In the static analysis of the assembly, as for the buckle one, the component contact is already set on *bonded* which means all components behave if they were welded during simulation. In Figure 4.4.5.1.1 it is possible to see it and it shows the contact il global, so it means that concerns all parts.

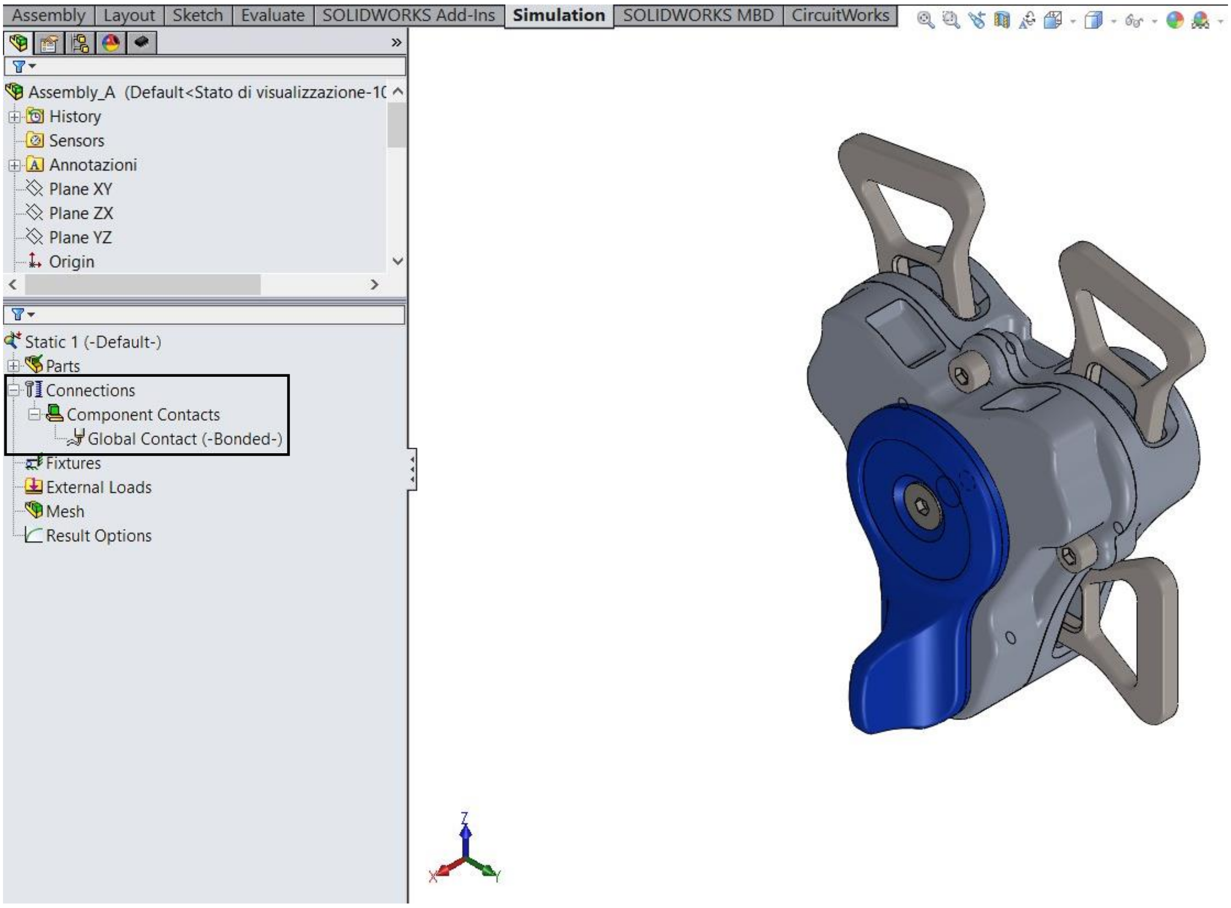

Figure 4.4.5.1.1 –Component contact option in Simulation menu.

Using this approach, the linear static analysis does not consider the non-linearity of the study due to the contact between the components.

4.4.5.2 Contact set

Besides the component contact, the right way to get results for the static analysis is to correctly set the contacts among the buckle parts. Therefore, it is necessary to follow a method in order avoid the wrong setting and to waste time: this step is fundamental to avoid the interpenetration during the loading.
Firstly, it is useful to remove two latches and two tongues to lighten assembly (Figure 4.4.5.2.1), so the simulations will be faster and will not negatively affect the calculation. Hence, just one tongue will be affected by the load.

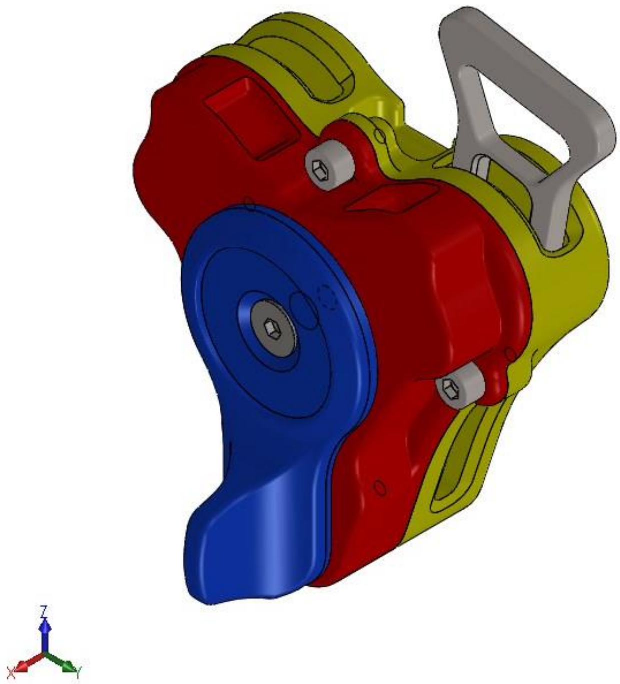

Figure 4.4.5.2.1 – Buckle without two latches and two tongues.

Afterwards, the next step is to fix the assembly in order to lock the three translational degrees of freedom: the lower body is fixed into the space and it cannot move.
Now it is useful to follow this scheme:

- to run the simulation;
- to analyse the results. Look at the displacement and if there is interference between two components, use “contact set” to avoid it (Figure 4.4.5.2.2). The first dashboard is for the object which goes through another, so it is necessary to select the affected surfaces. The second one is for the opposite part where is necessary, anyway, to select his surfaces.
  If there is not interpenetration, all contact sets are correct.

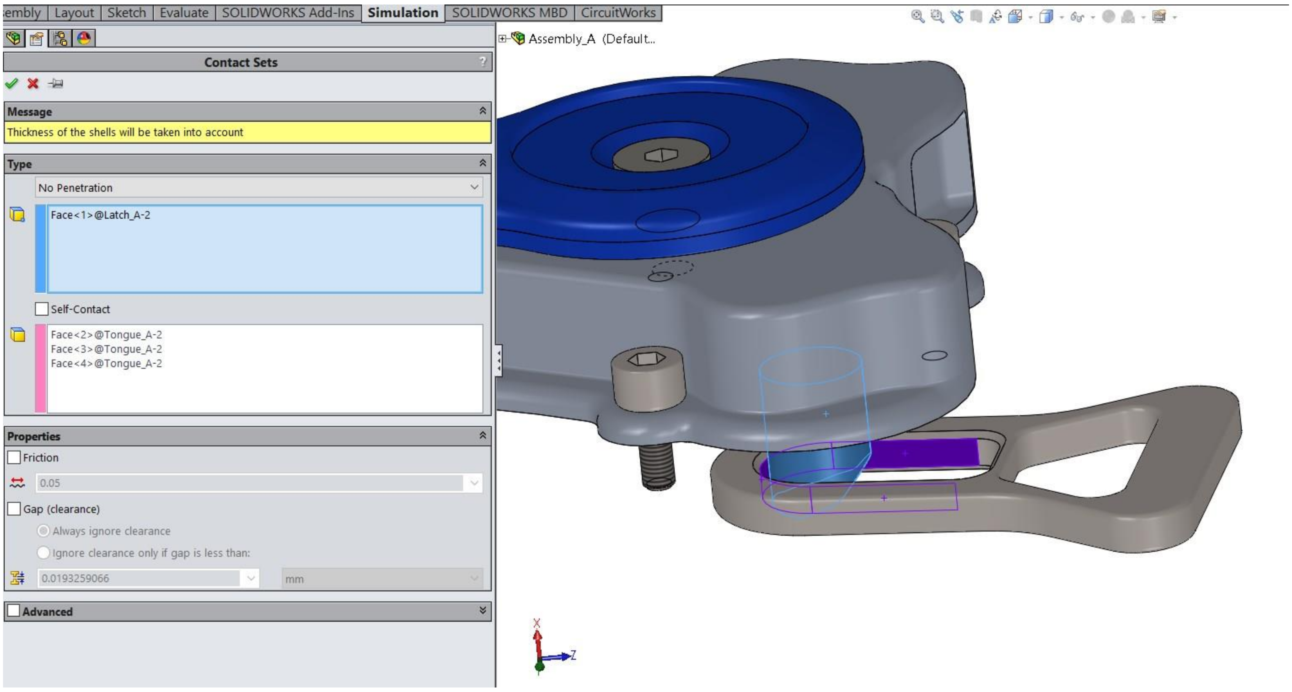

Figure 4.4.5.2.2 – Command to set the contact between two parts.

This scheme is important to get the best result without mistakes. All contact set could be chosen immediately, but:

- the simulation could last much time, as 1 to 3 hours;
- the simulation could not reach the equilibrium; hence it could stop and do not report any result;
- the simulation could show many interferences or illogical result and it could be difficult to understand the causes.

Therefore, the first simulation is without any contact set, just with materials, fixtures, external loads and component contact (Figure 4.4.5.2.3).

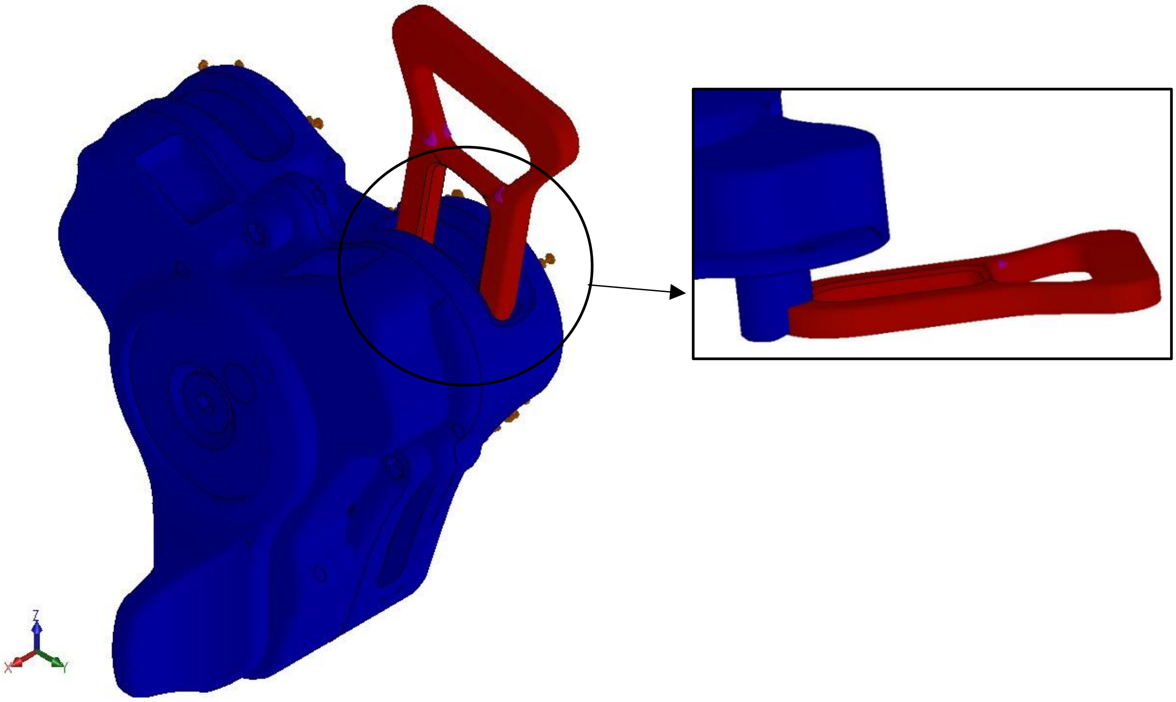

Figure 4.4.5.2.3 – Simulation without any contact set and the zoom of the interference.

In Figure 4.4.5.2.3, there is interpenetration between the remaining latch and his tongue, so it is necessary to use the command "Contact set" and avoid it. In the following list there all the steps which allow to get the right simulation.

- **contact set 1**: the tongue is locked from the latch and the displacement is represented in Figure 4.4.5.2.4. It is evident tongue brings the latch and the lifting plate out of the assembly;

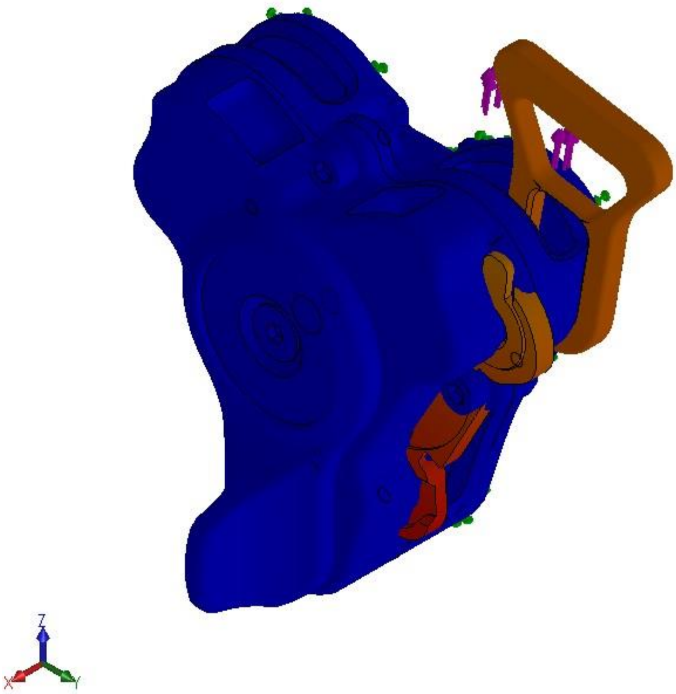

Figure 4.4.5.2.4 – Simulation with contact between tongue and latch.

- **contact set 2,3,4**: tongue surfaces locked by the lower body, displacement shown in Figure 4.4.5.2.5. The lifting plate and the upper body are affected by interpenetration;

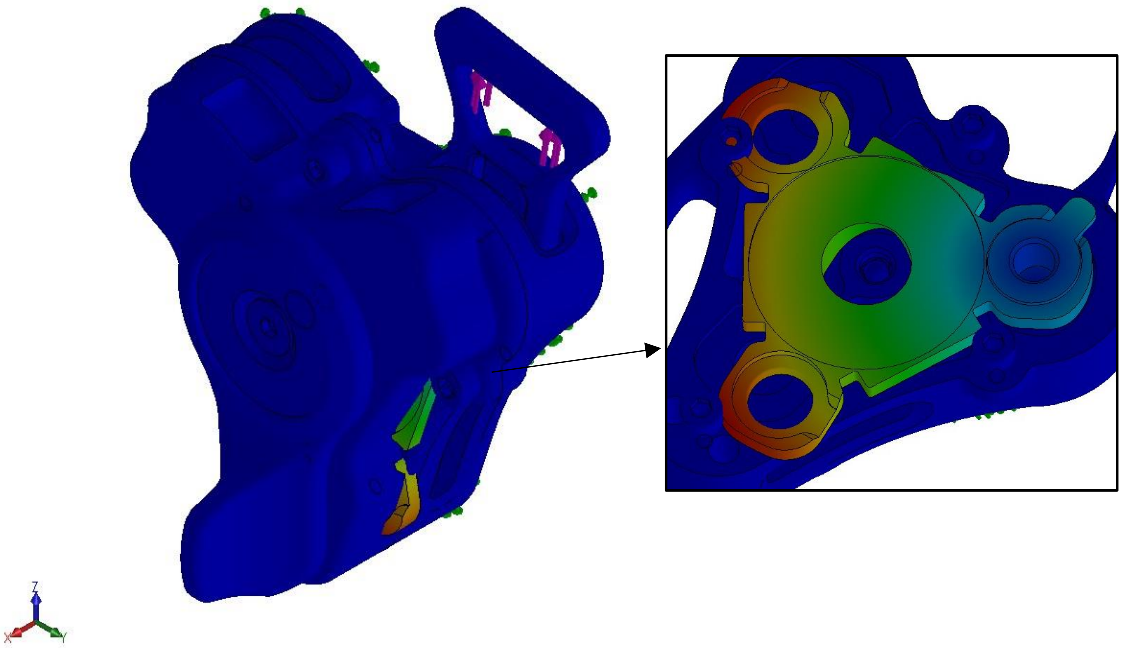

Figure 4.4.5.2.5 – Simulation adding the contact between tongue and lower body.

- **contact set 5**: locking of the lifting plate through camshaft surfaces (Figure 4.4.5.2.6). Now the lifting plate, the camshaft, the latch and the tongue are all linked together and there is an interference with the surface of the lower body where the lifting plate is placed;

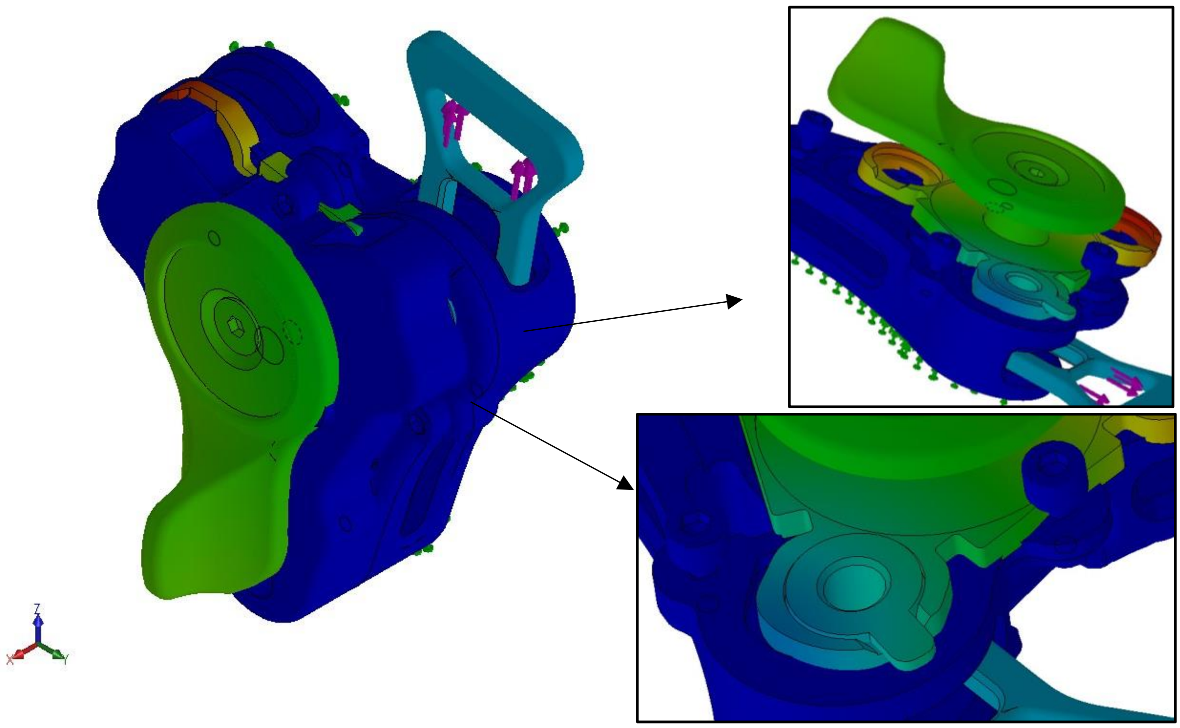

Figure 4.4.5.2.6 – Simulation adding the contact between lifting plate and camshaft.

- **contact set 6**: lifting plate locked by the lower body surface. The Figure 4.4.5.2.7 shows the interference between the lifting plate and the upper body, moreover the interpenetration between the latch and the lower body;

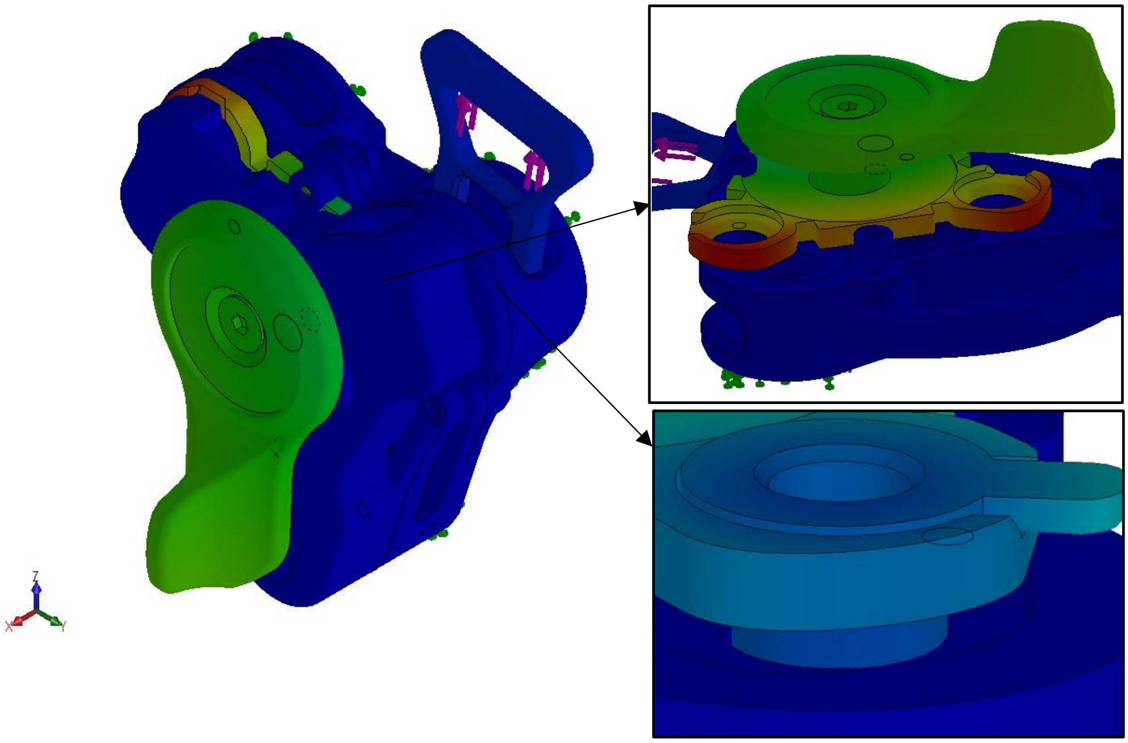

Figure 4.4.5.2.7 – Simulation adding the contact between lifting plate and lower body.

- **contact set 7,8**: the lower body locks the latch, so it fixes many other parts (Figure 4.4.5.2.8), but the camshaft and the lever are not clamped. Moreover, the lifting plate has been locked by the upper body through his internal surface;

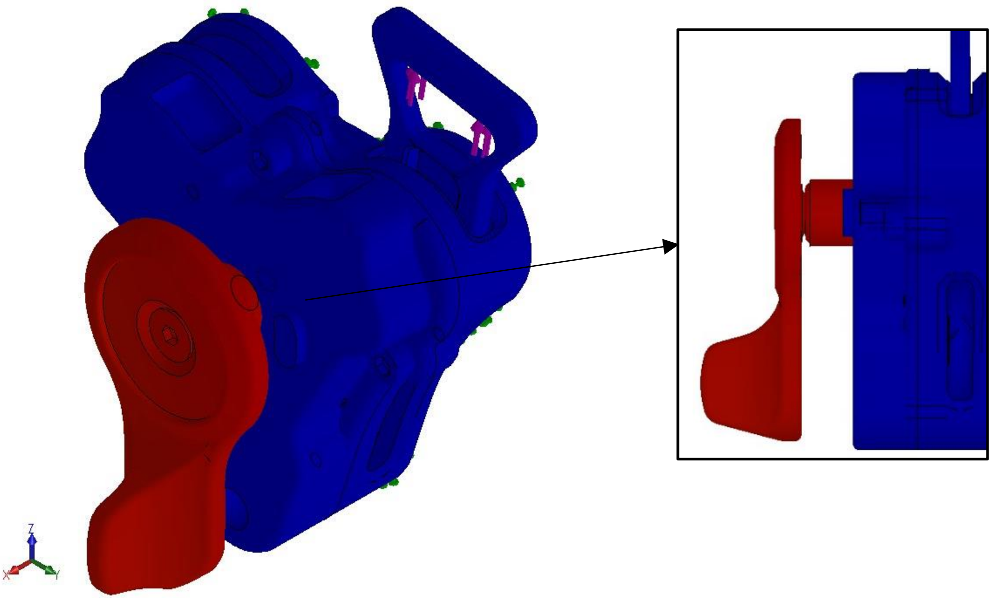

Figure 4.4.5.2.8 – Simulation adding the contact between lifting plate and lower body.

- **contact set 9,10**: the cams of the camshaft have been locked by the surfaces of the lifting plate, so this one drags the camshaft (Figure 4.4.5.2.9);

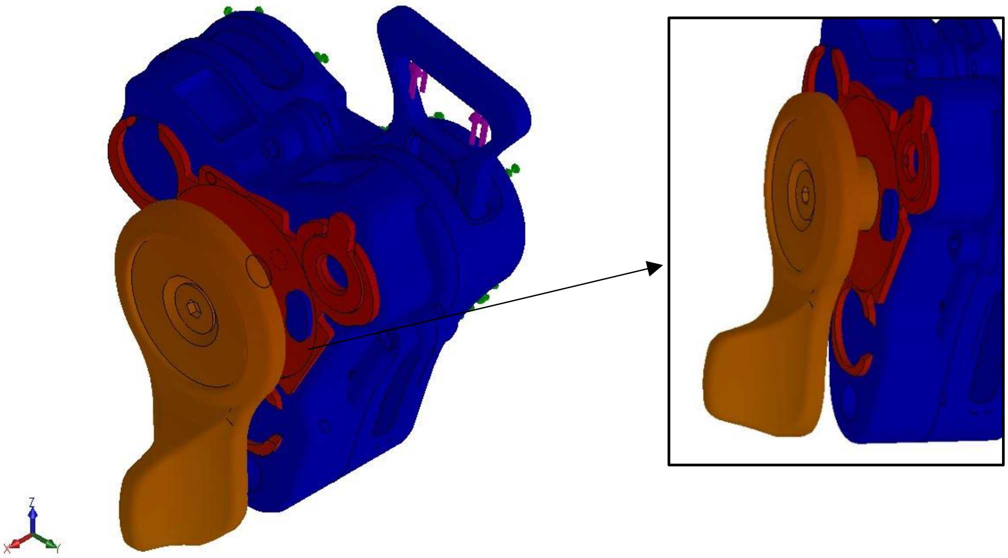

Figure 4.4.5.2.9 – Simulation adding the contact between lifting plate and camshaft.

- **contact set 11**: the internal surface of the upper body locks the lifting plate which is getting out from the buckle in the Figure 4.4.5.2.9. The sub-assembly, lever plus camshaft, pulls down (Figure 4.4.5.2.10) because there is not a relationship with the sphere;

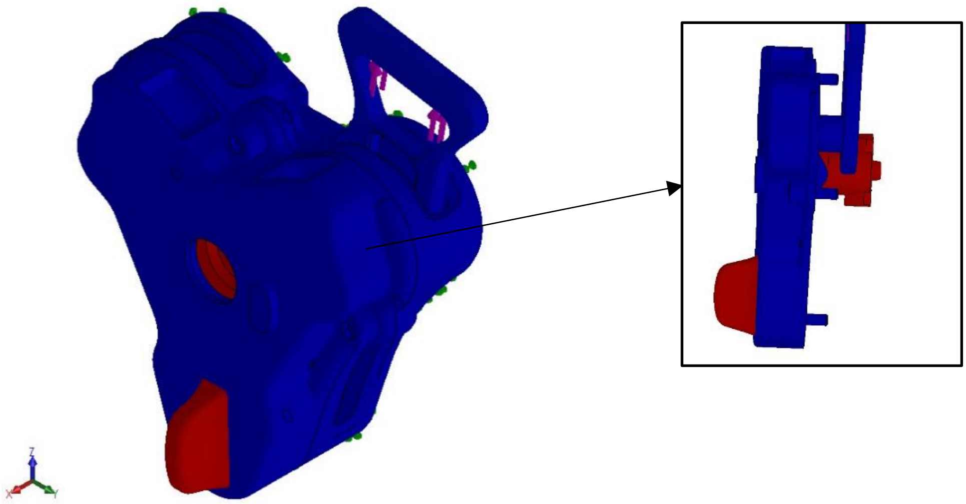

Figure 4.4.5.2.10 – Simulation adding the contact between lifting plate and internal surface of the upper body.

- **contact set 12**: the camshaft has been locked by the sphere, but they pull down together (Figure 4.4.5.2.11), so it will be necessary to add another contact set on the sphere;

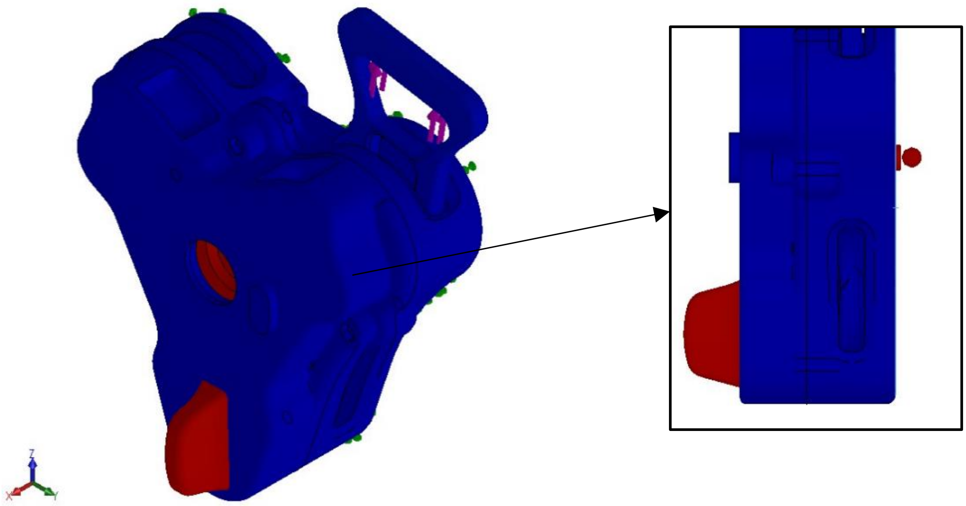

Figure 4.4.5.2.11 – Simulation adding the contact between the camshaft and the sphere.

- **contact set 13**: the sphere has been locked with his housing on the lower body (Figure 4.4.5.2.12), now the problem is the upper surface of the tongue;

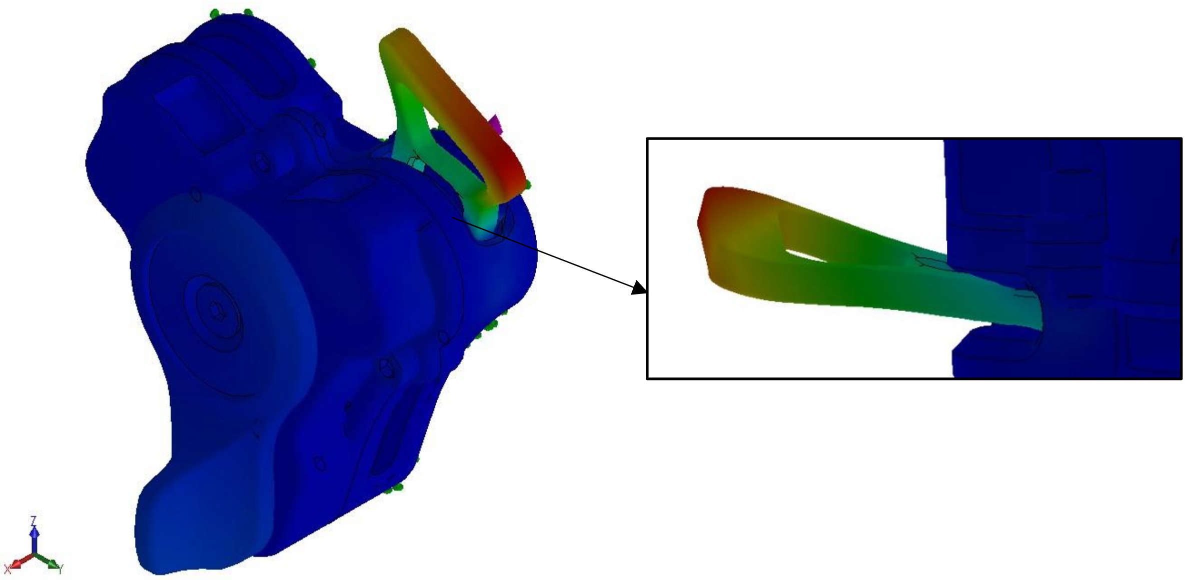

Figure 4.4.5.2.12 – Simulation adding the contact between the sphere and his housing on the lower body.

- **contact set 14,15**: the upper and the lower surface of the tongue could not compenetrate the lower body (Figure 4.4.5.13), but, beside this, the huge force on the tongue guarantees interpenetration with the lower body. Moreover, the latch has no contact set with the lifting plate, so this behaviour is reasonable;

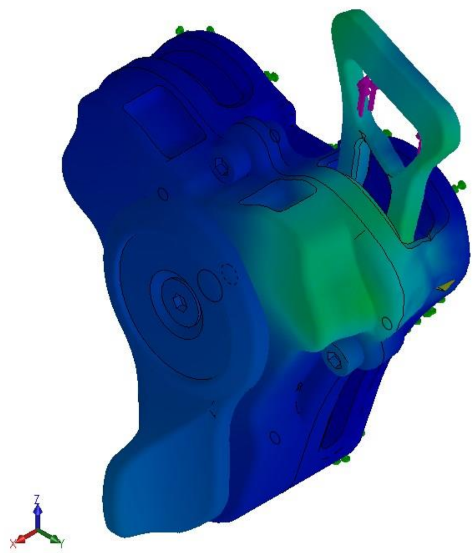

Figure 4.4.5.2.13 – Simulation adding the contact between the upper and the lower surface of the tongue and the lower body.

- **contact set 16**: the lifting plate locks the latch (Figure 4.4.5.2.14), hence the simulation shows the final result: all the components are clamped in the right way and nothing could compenetrate. The displacement affects only the tongue which is stopped by the latch.

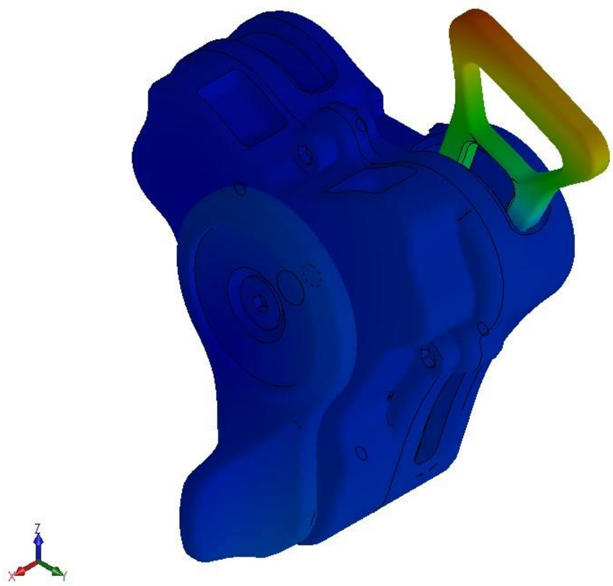

Figure 4.4.5.2.14 – Simulation adding the contact between lifting plate and the latch.

Following this method, the static analysis becomes non-linear because the contact set includes the influence of the load on the solution, indeed:

$$\mathbf{K}(\mathbf{f})\mathbf{x} = \mathbf{f}$$

This sort of analysis should consider the non-linearity due to the contact between each part of the assembly, it's wrong to consider all components welded in a unique one. Particularly, the stiffness of each component is not proportional to the displacement but can be expressed by a general function which depends by the force. The load on the tongues causes the contacts among the components, so their stiffness matrix is influenced by the force.

4.4.5.3 Spring connector

The last step before running the simulation concerns the spring. Since the CAD model has four springs, it is necessary to convert them into FEM entities. The right way is to use *Connectors* which are mechanisms that define how an entity, as vertex, edge or face, is connected to another entity or to the ground. Using them the model is simplified because you can simulate the desired behaviour without having to create the detailed geometry or define contact conditions.
By using spring connectors, the number of elements and analysis time are reduced.
There are three types of springs:

- compression-extension;
- compression only;
- extension only.

In the buckle, the springs work only under compression and to set them it is necessary to:

1. insert a new spring connector (Figure 4.4.5.3.1);

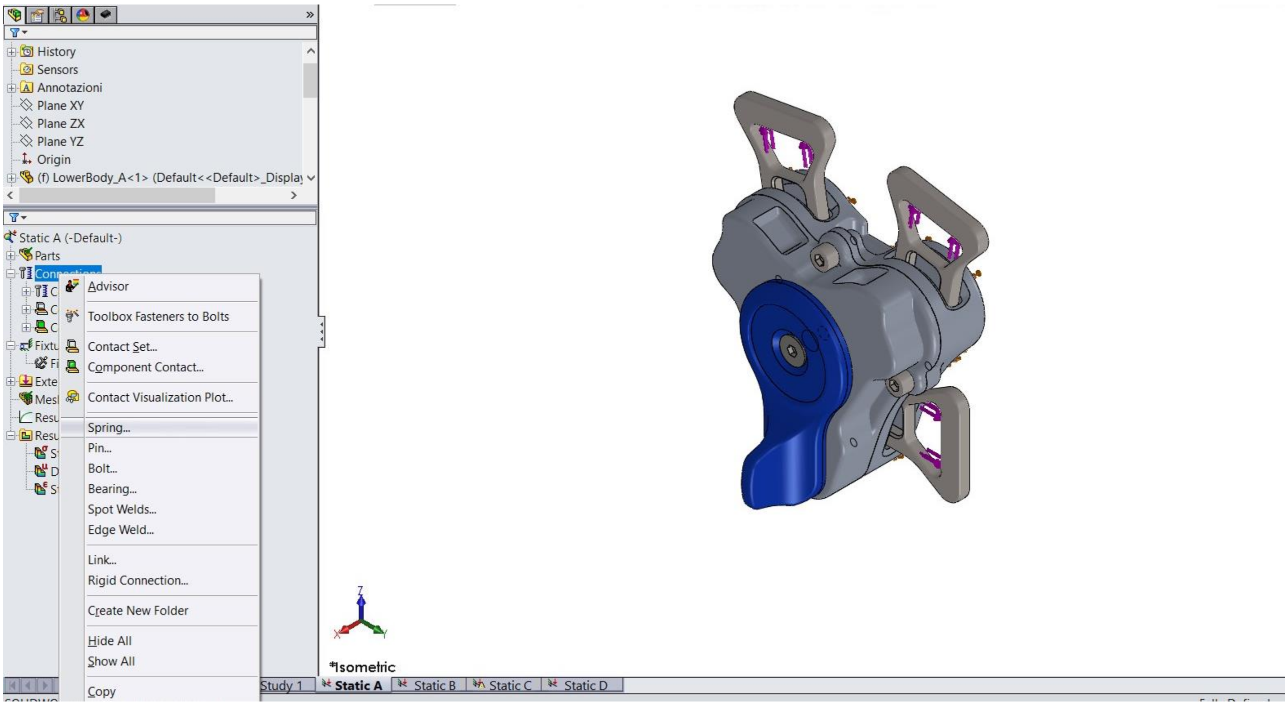

Figure 4.4.5.3.1 – How to open spring connector dashboard.

2. select the type (Figure 4.4.5.3.2) which is flat parallel faces;
3. select the interested surfaces where the spring should work;
4. impose the stiffness value and, in case, the pre-load.

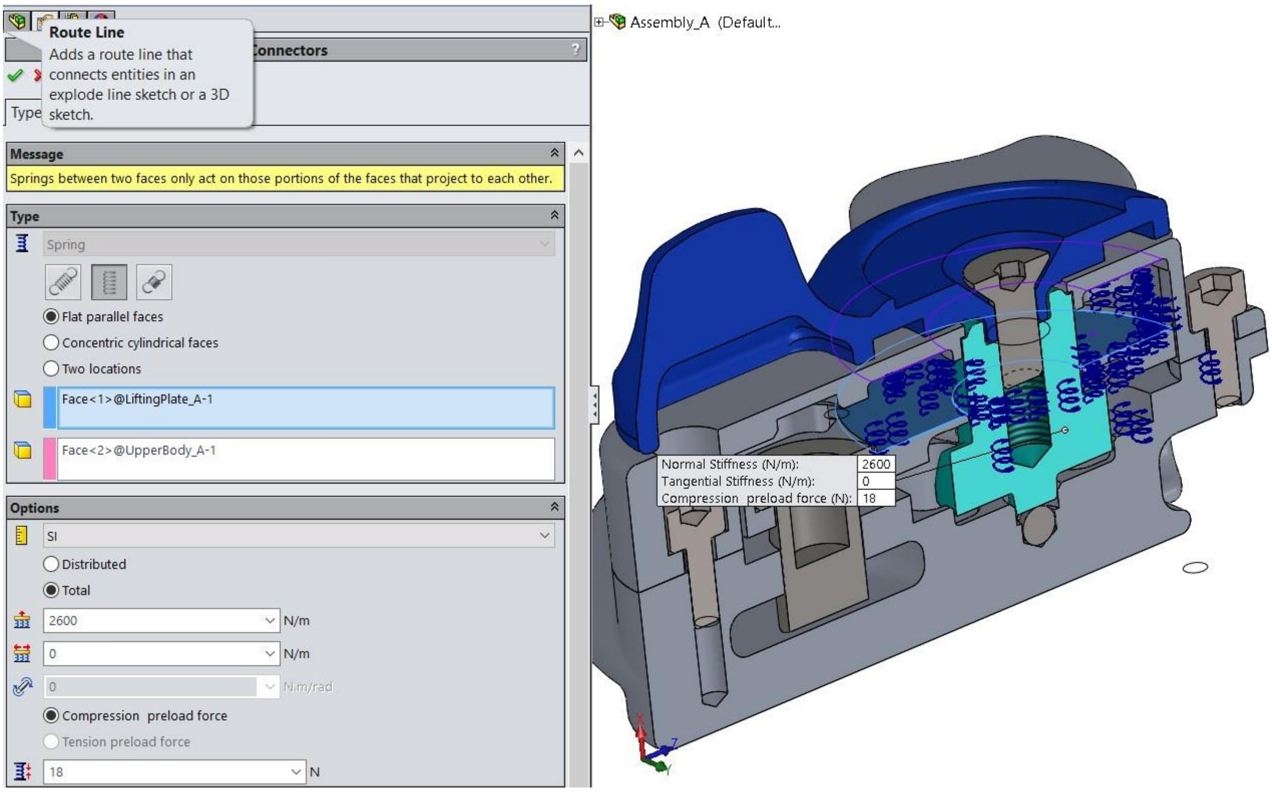

Figure 4.4.5.3.2 – Setup of the spring connector dashboard.

This procedure used to set a spring should be done for each spring:

- the conical one between the lifting plate and the upper body;
- the cylindrical one between each latch and the upper body.

4.5 Case study

Now it is necessary apply all these methods to correctly set the static analysis for the safety buckle.

4.5.1 Material

The materials chosen for the system components are resumed in Table 4.5.1.1.

Table 4.5.1.1 – Table of components material.

Component	Material	Density [kg/m^3]	Young modulus [GPa]	Poisson ratio [-]	Yield strength [MPa]
Lower body	Al 7075	2810	72	0,33	505
Sphere	Steel AISI 304	8000	190	0,29	207
Camshaft	Al Sint D30	7000	130	0,31	330
Lifting plate	Al 7075	2810	72	0,33	505
Latch	Steel 36NiCrMo16	7800	210	0,28	1050

Latch spring	Steel C72	7850	206	0,28	20
Conical spring	Steel C72	7850	206	0,28	20
Upper body	Al 7075	2810	72	0,33	505
Lever	Al 6082	2690	69	0,33	215
Screw ISO 4762	Ti	4500	105	0,37	377
Screw ISO 10642	Ti	4500	105	0,37	377
Tongue	Ti	4500	105	0,37	377

4.5.2 Fixture

Three elastic supports, as shown in Figure 4.5.2.1, which allows to lock the buckle in the space, replicating the real conditions. The fixture applied to lower body simulate the stiffness of the driver's stomach. Instead, the support applied on the fixed attachment represents a reaction constrain. The last one is applied on the lower body in direction Z.

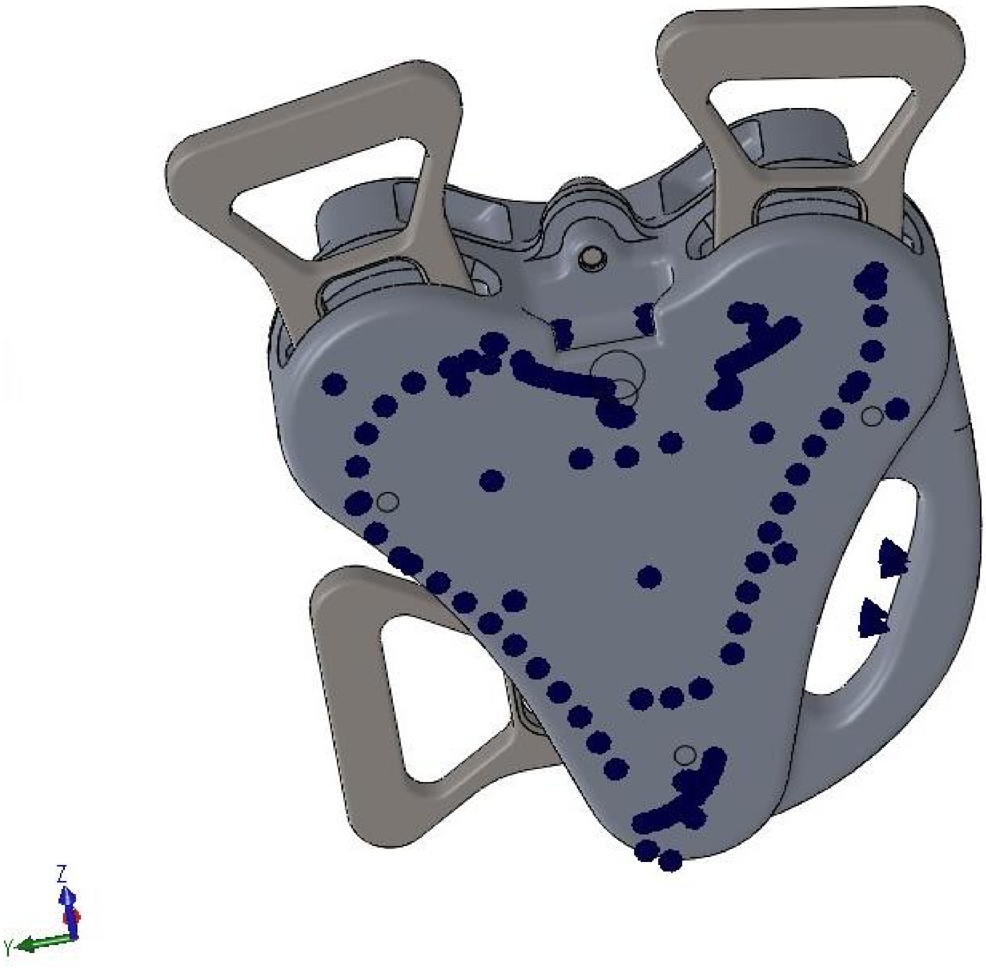

Figure 4.5.2.1 – Setup of the spring connector dashboard.

The values of the stiffnesses have been set using a smart number, hence a number which allows to get a real result. Indeed, the first simulation has a stiffness value of 10^2 N/m, but it generated huge displacements, so it has been increased.
The right value has been set at:

- 10^7 for the normal stiffness;
- 10^6 for the shear stiffness, usually lower of one order of magnitude.

4.5.3 Load

The six straps of the safety harness generate forces which act on the buckle (Figure 4.5.3.1). Particularly:

- shoulder and crotch straps generate a force on the upper tongues;
- pelvic straps generate a force on the lower right tongue and a fixture on the fixed tongue.

Figure 4.5.3.1 – Scheme to represent forces (blue) and fixture (green) on the buckle's tongues.

The value imposed in Soldiworks for every strap corresponds to 15 kN which is required from the FIA 8853-2016, except from the crotch ones where the load should be 10kN.
Hence, these forces have been set in the buckle model following the direction of the straps, as Figure 4.5.3.2 shows.

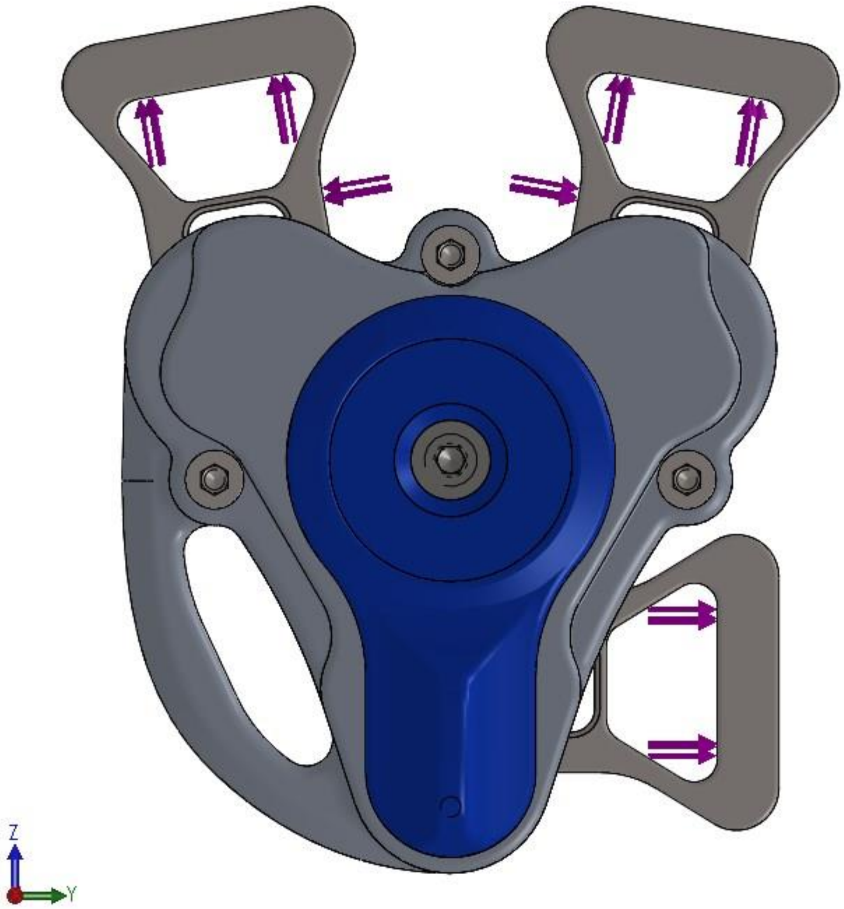

Figure 4.5.3.2 – Representation of the forces which act on the buckle.

4.5.4 Connection

- Global contact: used to weld the component together;
- spring connector: to simulate the stiffness of the conical spring and of the latch springs.

Table 4.5.4.1 – Table of springs proprieties.

Spring	Stiffness [N/m]	Pre-load [N]
Conical	2600	18
Latch	2400	14

- contact set: used to effectively replicate the contact between each part, avoiding the interpenetration. The number of contact set used is 48.

4.5.5 Mesh

The way to get the right mesh for the buckle assembly is through iteration, indeed, it is necessary to try until all parts are meshed. For example, a coarse mesh is faster from a computational point of view, but Solidworks could stop the process because one or more component cannot be meshed. Moreover, it is useful to change the convergence, as high or draft quality mesh, or the curvature.

In Figure 4.5.5.1 is shown the first iteration where the mesh has been set to coarse. As you can see, many components are missing, hence it is necessary to change the mesh density.

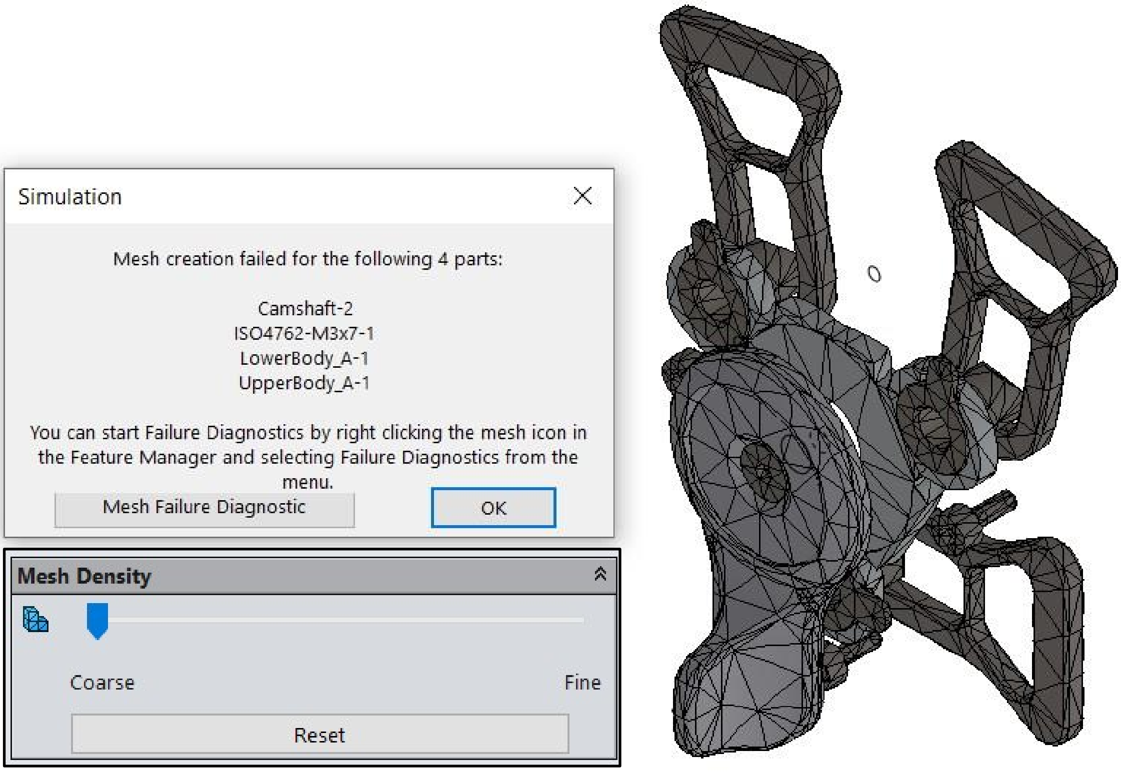

Figure 4.5.5.1 – Mesh got with a coarse mesh density.

Increasing the mesh density, the number of components meshed is higher than before, but still not sufficient, as it is possible to see in Figure 4.5.5.2.

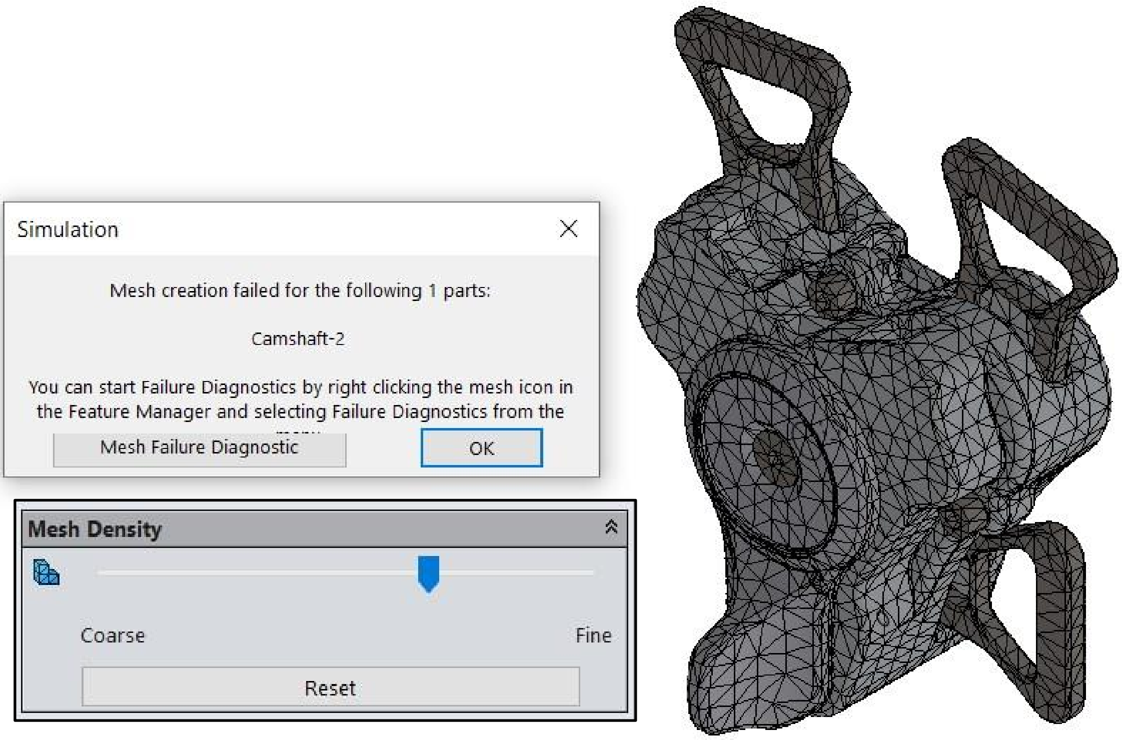

Figure 4.5.5.2 – Mesh got with a medium mesh density.

After the step shown in Figure 4.5.5.2, the right setup has been found and in Figure 4.5.5.3 there is the result. Hence, this configuration represents the final step to get the correct mesh which will be used for the simulation.

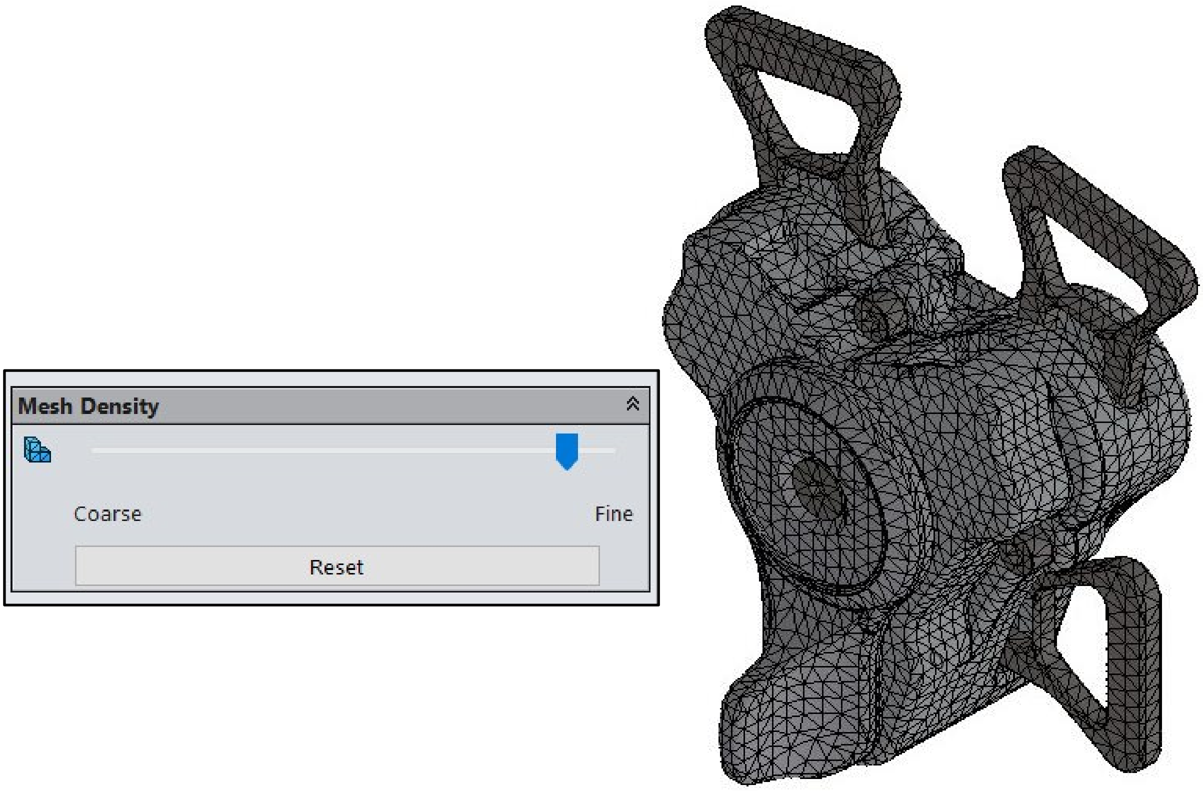

Figure 4.5.5.3 – Mesh got with a fine mesh density.

4.5.6 Result

After the end of the setting, it is necessary to run the simulation to get the result useful for the FEM, firstly it starts with the stress.
The results should show a stressed zone:

- on the tongues, most of all where they touch the latches and the straps;
- on the latches, where they are in contact with the tongues and with the lower body;
- on the lower body, where the latches load;
- on the fixed tongue, but lower than in the other zones.

Figure 4.5.6.1 shows the stress results.

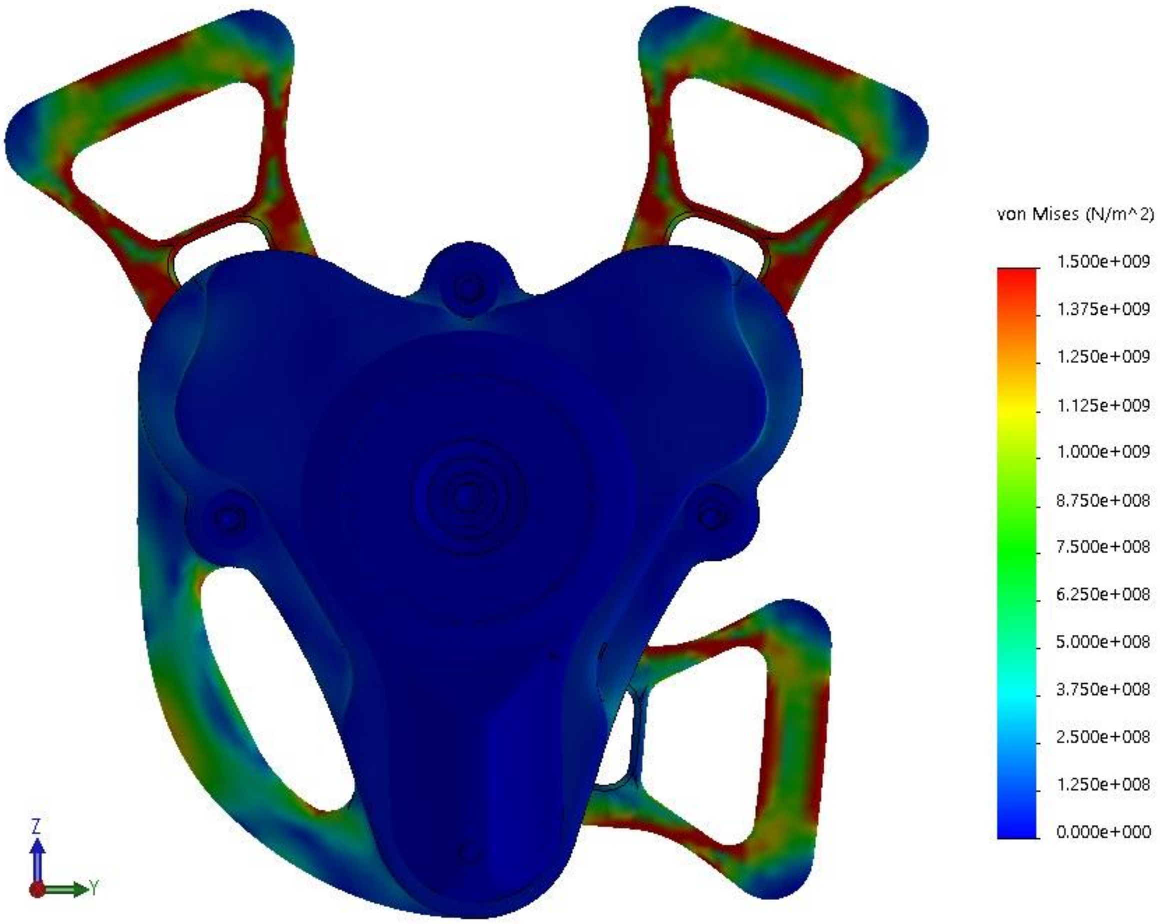

Figure 4.5.6.1 – Stress result on the super light buckle.

Observations on Figure 4.5.6.1:

- Von Mises stress used to analyse the study with a scale which starts from zero to 1500MPa. Useful to comprehend where are the stressed zoned, the focus is not on the values, but on the method;
- correct stress zones (Figure 4.5.6.2).

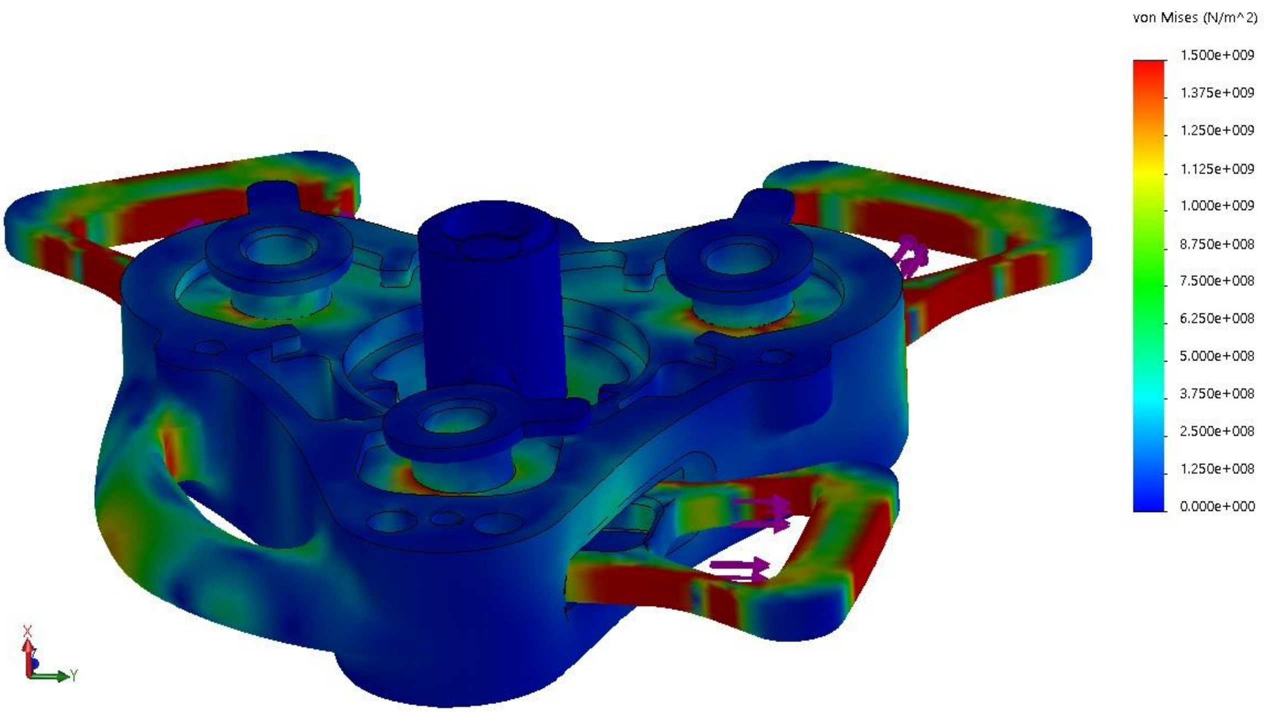

Figure 4.5.6.2 – Stress result on the super light buckle, hiding the upper body.

Figure 4.5.6.2 gives a better representation of the stressed zones, but it is useful analyse each component.

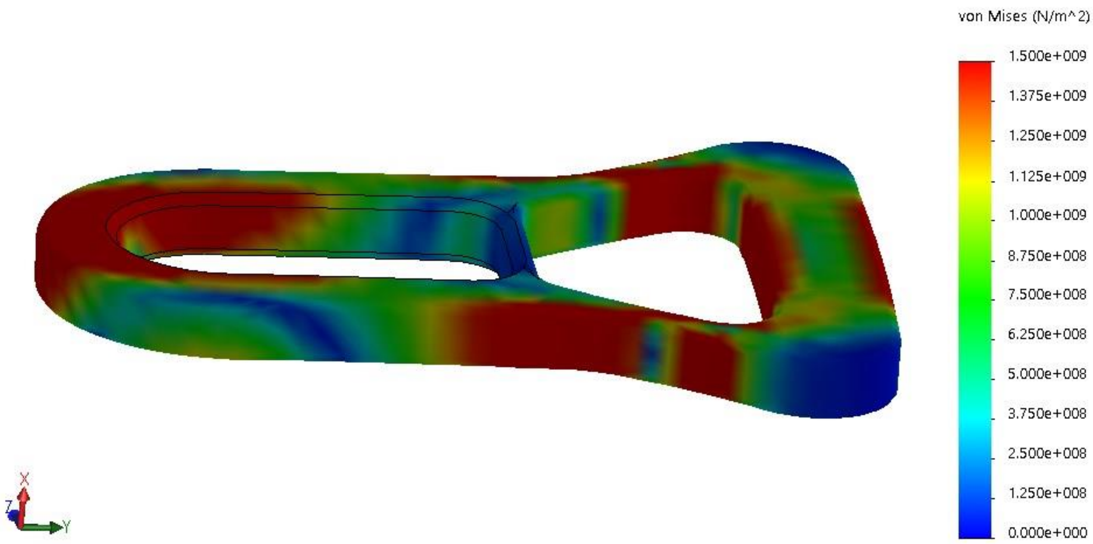

Figure 4.5.6.3 – Stress result on one tongue.

Figure 4.5.6.3 shows the stress caused by the strap load of 15kN and by the contact with the latch, there is a bending and a pressure, most of all where the diameter changes.

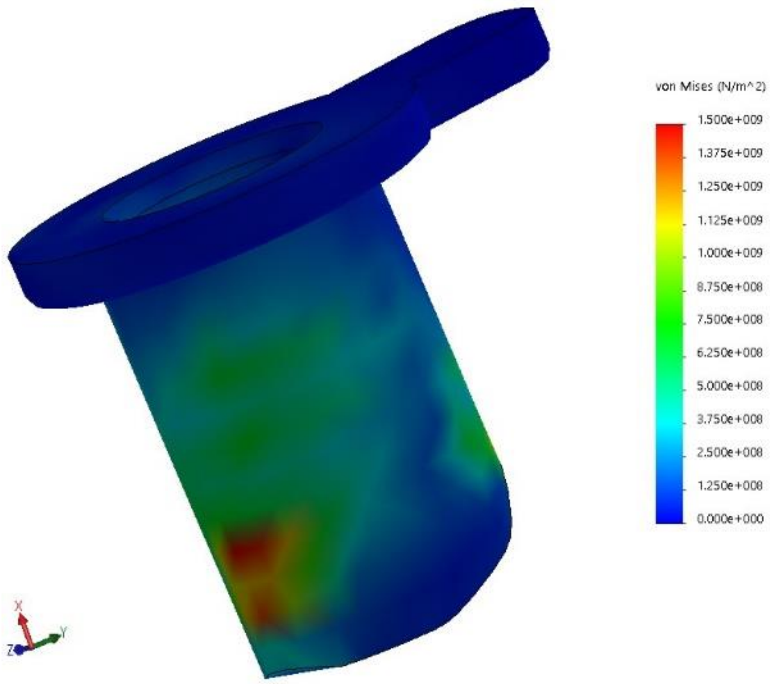

Figure 4.5.6.4 – Stress result on one latch.

Figure 4.5.6.4 shows the stress caused by the contact with tongue, indeed there is a zone of the surfaces which is red.

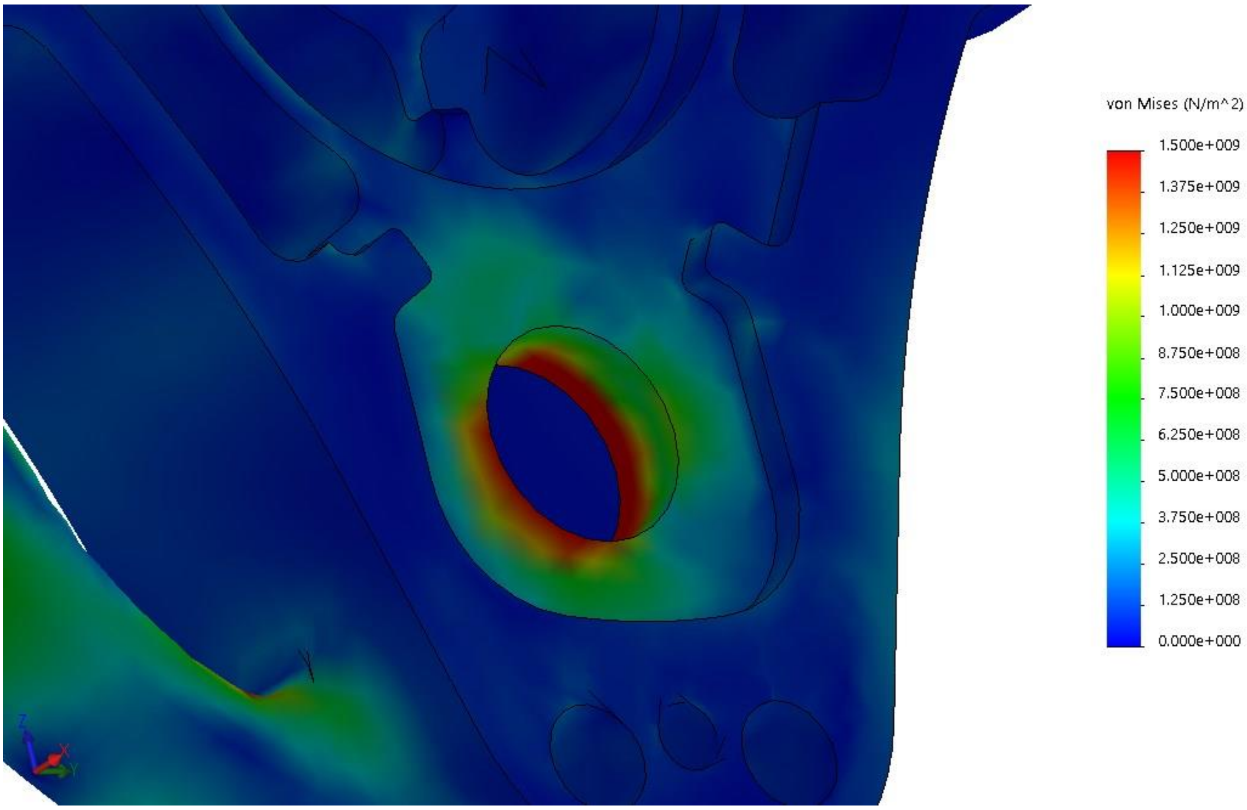

Figure 4.5.6.5 – Stress result on the lower body caused by one tongue.

The red zones in Figure 4.5.6.5 are correct because the latch is loading there, due to the force on the tongue.

The last stressed zone is the fixed tongue which represent an hyperstatic beam. It is clamped from the sides both and the strap generates a reacting load which acts in the middle of the beam. Hence, as you can see in Figure 4.5.6.6, there is traction where the force is applied and compression where the tongue is fixed.

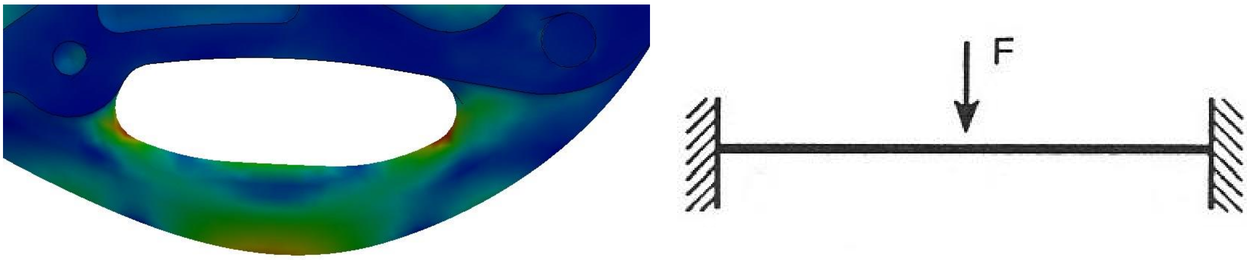

Figure 4.5.6.6 – Stress distribution on the fixed tongue on left and his hyperstatic scheme on right.

Figure 4.5.6.6 clearly shows the neutral axis, in blue, between the compression and traction zones. Moreover, Figure 4.5.6.7 shows the scheme of the bending moment and the elastic deformation of the beam.

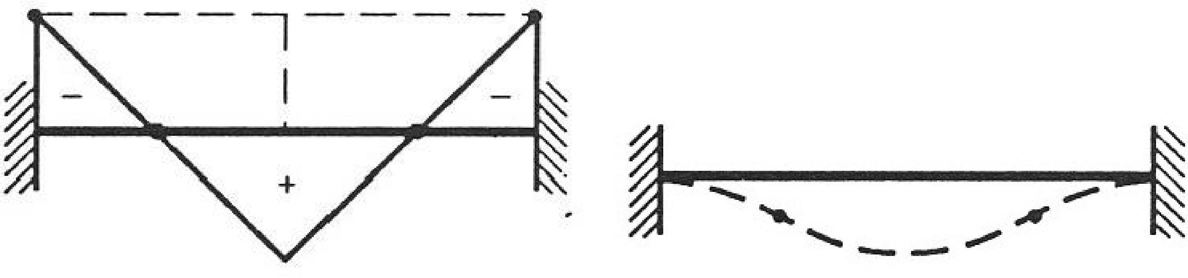

Figure 4.5.6.7 – Bending scheme on left and elastic deformation on right.

On the other hand, analysing the displacements, Figure 4.5.6.8 shows:

- a maximum displacement of the tongue of about 15mm, due to the load of the straps;
- rotation on the tongue on the ZY plane due to combined load of the straps;
- the left tongue is subjected to a higher displacement than the other two because the fixed tongue below does not produce a force; it is a fixture.

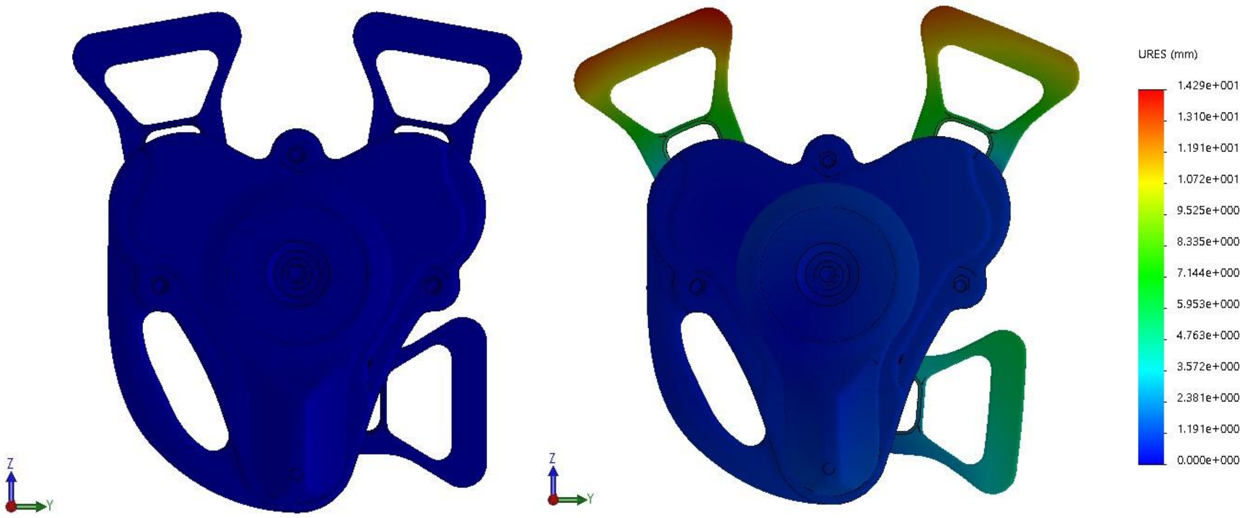

Figure 4.5.6.8 – Displacements of the tongues.

5 Motion

The last step to conclude the analysis is the motion study which is a graphical simulation of motion for the assembly model. It does not change the assembly model or its proprieties, but it simulates and animates the motion you prescribe for a model. Particularly, it is not a linear analysis, but a time study with not deformable bodies. It is useful to show the coupling between each component.
Two studies have been done:

- fastening of the buckle;
- unfastening of the buckle.

First, it is necessary to explain how it works and the method used to get the results.

5.1 Motion Manager

The Motion Manager is the dashboard where you can set and simulate the study, it is shown in Figure 5.1.1.

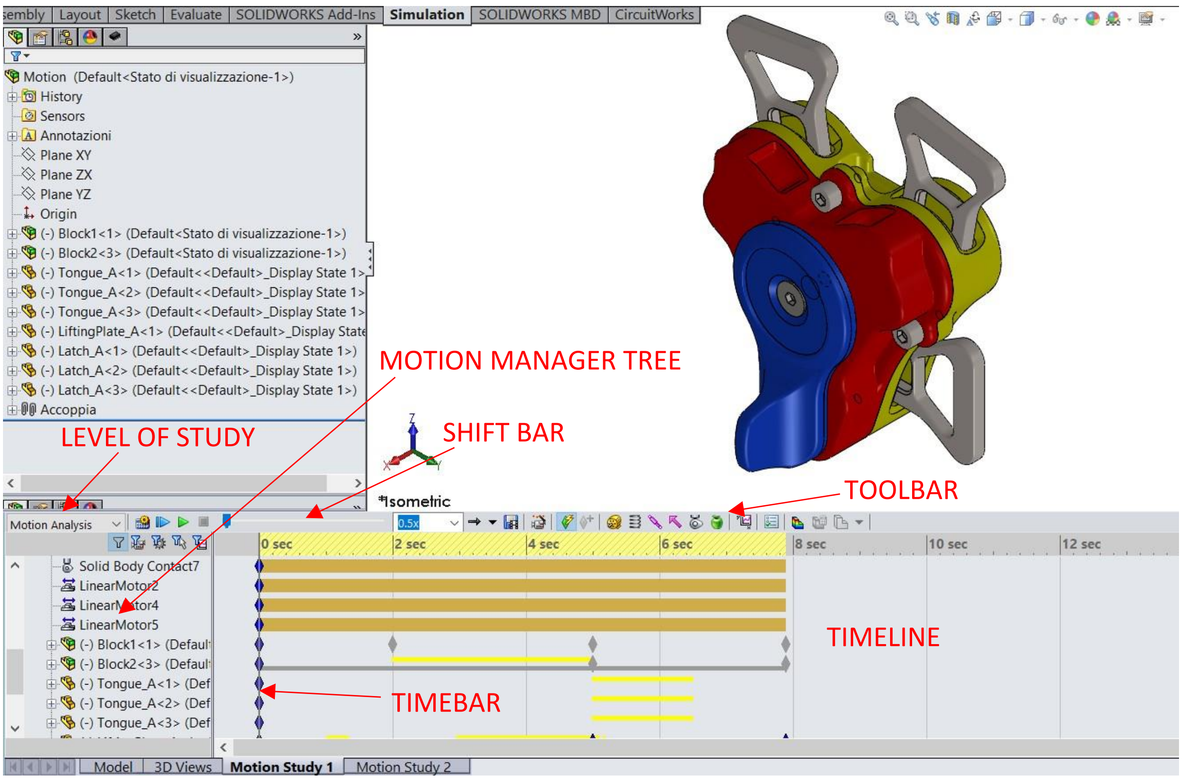

Figure 5.1.1 – Dashboard of the Motion Manager.

The level of study is composed by:

- animation;
- basic motion;
- motion analysis, which is the most complete and the one which is used.

The Motion Manager Tree is where you can find each action and degree of freedom of the part, as, for example, a spring or a motor.

The Shift Bar represents the advance of the motion with his speed, as x1, x2 or x4.
The Toolbar is where is possible to add the gravity, the motors or the springs, for example.
The Timeline is the duration of the motion study which could be set as you prefer, in this lasts 8 seconds.
The Timebar represents the instant where you are.

5.2 Set up

Before starting the motion study, it is smart to agglomerate the component in sub-assembly, to reduce the computational calculation and to handle the assembly in a faster way:

- upper body, lower body, sphere and their screws unified in a block because they behave in the same way;
- lever, camshaft and their screw unified in another block, they rotate together.

Moreover, the movement of each part is determined through:

- degrees of freedom of the components;
- motor applied;
- mass proprieties;
- external forces;
- time.

Therefore, the second step is to reduce the degrees of freedom of the components, to allows the movements you would like to simulate. For example, it has been unlocked the rotation of the lever, removing the coincide between his axis and the lower body's axis.
After the definition of all the right degrees of freedom for each part, it is necessary to set the gravity, choosing a direction and a module. His dashboard is in the Toolbar (Figure 5.2.1).

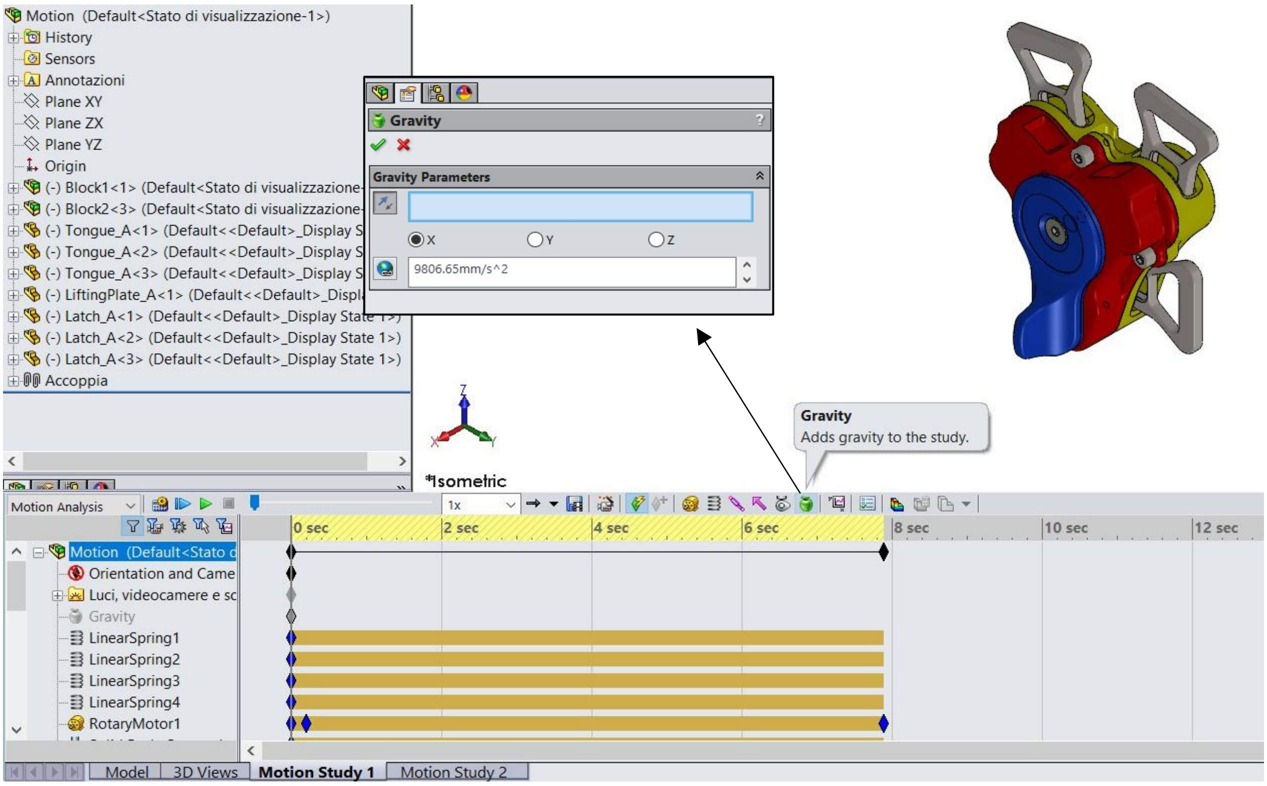

Figure 5.2.1 – Dashboard of the gravity.

Afterwards, it is necessary to set the four spring, Figure 5.2.2 shows their dashboard.

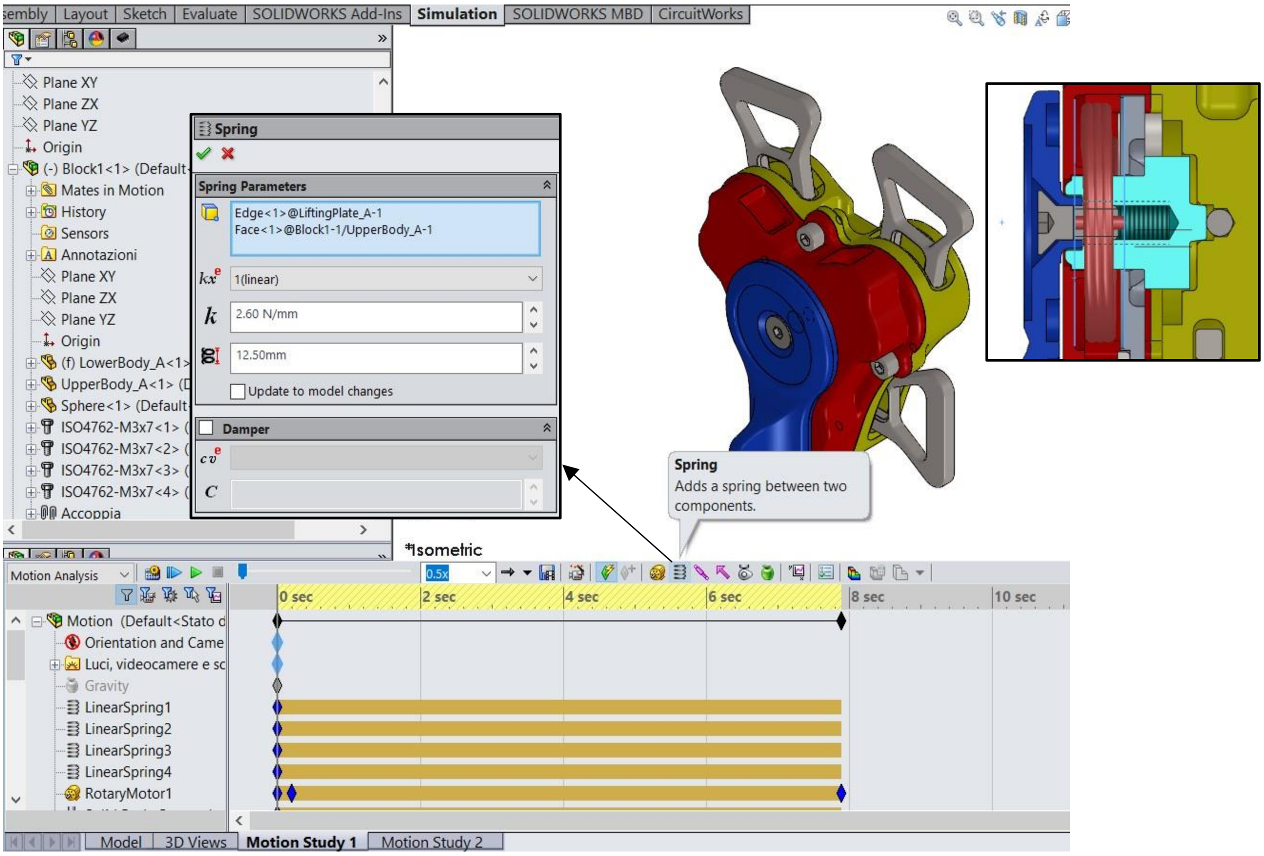

Figure 5.2.2 – Dashboard of the spring.

After selecting the spring in the Toolbar, you must choose the interested surfaces, the kind of spring, the stiffness and his height. This method must be used for all the four springs.

The next tools to set are the motors:

- one rotary for the lever;
- three linear for the tongues.

As Figure 5.2.3 shows, you must select the face and his direction, moreover, you must choose the kind of motion. In this case, the lever one, it has been set a rotation of 90° which starts after 2 seconds and lasts for 3 seconds. The same procedure must be applied for the unfastening and fastening of the tongues.

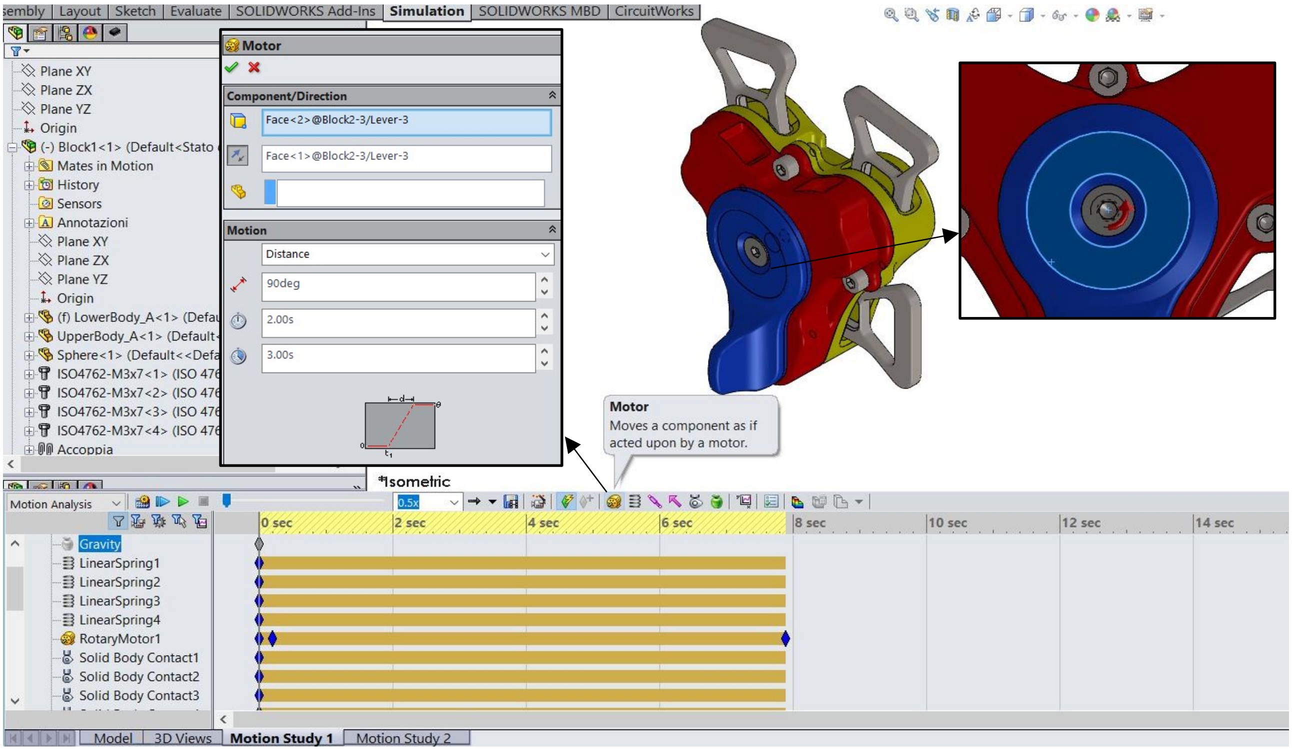

Figure 5.2.3 – Dashboard of the motors.

For the linear motors the setting is the following:

- distance: 35 mm;
- starting time: 5 seconds;
- lasting: 1,5 seconds.

The last step is the setting of the body contacts which means the parts in contact between each other's. There are 6 contacts:

- cams of the lifting plate on the camshaft;
- latches with the lower body;
- latches with the lifting plate;
- latches with the upper body;
- camshaft with the sphere;
- tongues with the lower body.

The way to set them is shown in Figure 5.2.4. You must select the interested components, which is faster than selecting the surface, and his material.

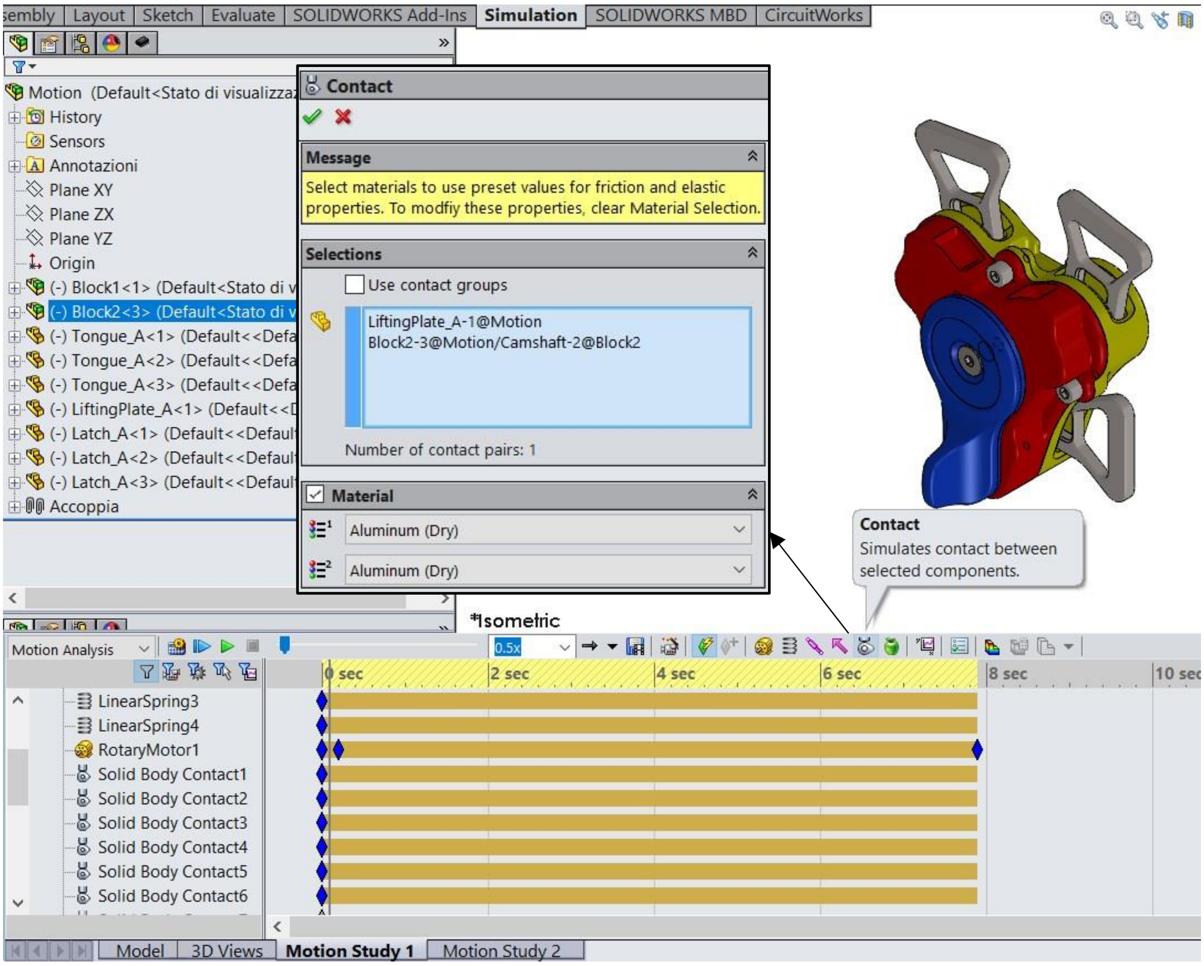

Figure 5.2.4 – Dashboard of the body contact.

5.3 Results

After the setting procedure, it is necessary to run the study, as shown in Figure 5.3.1.
This method has been used for fastening and unfastening both; on the Motion Manager Tree you can find each actions imposed and calculated.
The Motion Analysis has been useful to show the functioning of the mechanical couplings.

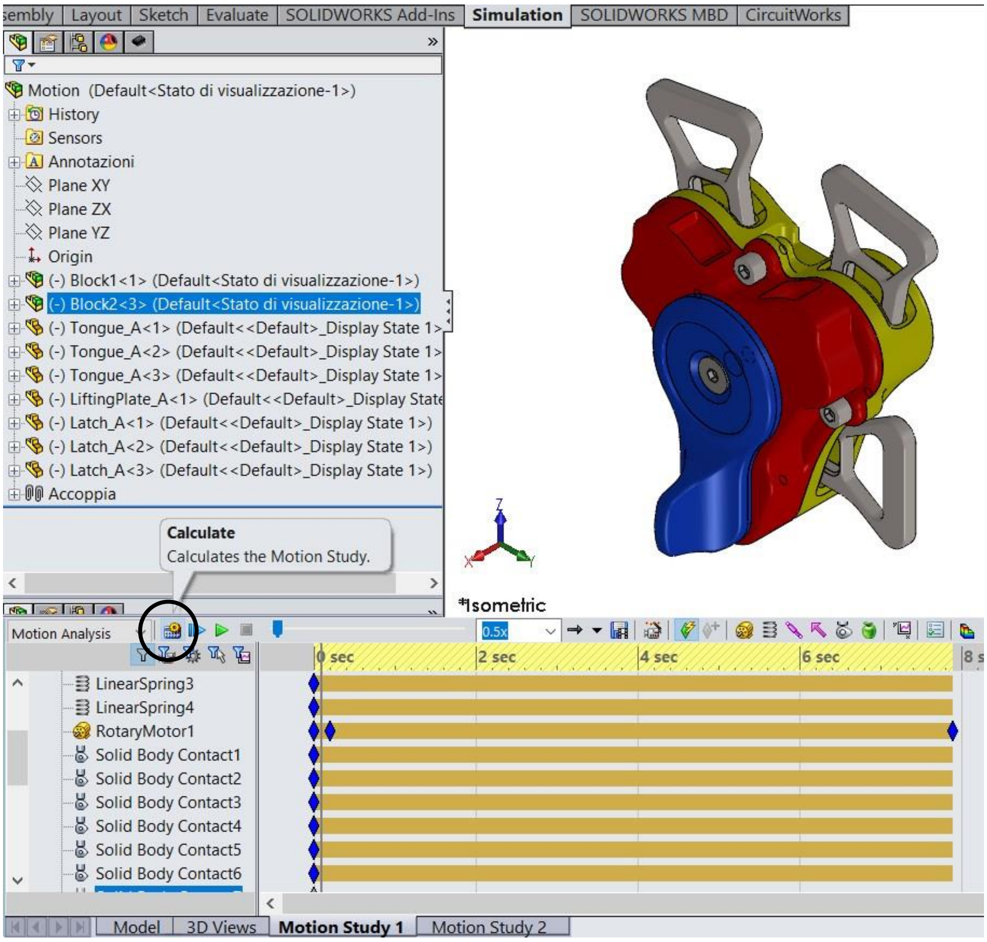

Figure 5.3.1 – Tool to calculate the Motion Analysis.

Conclusion

Working on a project like a safety harness has been interesting and instructive, indeed, you can learn to analyse a mechanical component from many points of view.
The design of the buckle in Solidworks has been fundamental to achieve a huge know-how of the software, as, for example, the techniques used to get it or the drawings of the parts. Moreover, the comprehension and the interpretation of the FIA 8853-2016 make you confident with the industrial world because each component must be tested and homologated.
The study and the analysis of the material and of the mechanical processes of the components allow you to enhance your knowledge, most of all in the automotive world. It is important to know how the buckle has been created and from what materials, besides the engineering difference between each material or process. Using an Aluminium bar is different from a Titanium one, for the proprieties and for the price both.
Afterwards, it has been studied the static analysis from an analytical point of view, but also applied to the Finite Element Method. Hence, the theory and his application played the most important role in this project: a brief excursus on the analytic formulation of the linear and non-linear system gives you the skills to approach them on the software. Without this knowledge, it has been impossible to achieve the FEM results on Solidworks; in addition, it has been explained, in broad terms, the theory of the FEM analysis and how it is used for the mesh of an object. Thanks to this study has been possible to apply it on the buckle on Solidworks, but it is necessary to follow a method, tested with this buckle. The most difficult step to achieve the right setting for the static analysis is the interference between the parts: using the method of the Contact Set it is possible to get the right set up, without high computational calculation or wrong simulation.
Therefore, the focus must be on the procedure used to get the result and not on the value get, because they are relative. The stress and the displacement got on the software are exceptional from a physical point of view, indeed, they have a logical sense, correctly representing the realty. This kind of study can be used for each assembly studied through a FEM analysis.

Acknowledgements

Appendix

It contains the drawings of the components of the super light buckle.

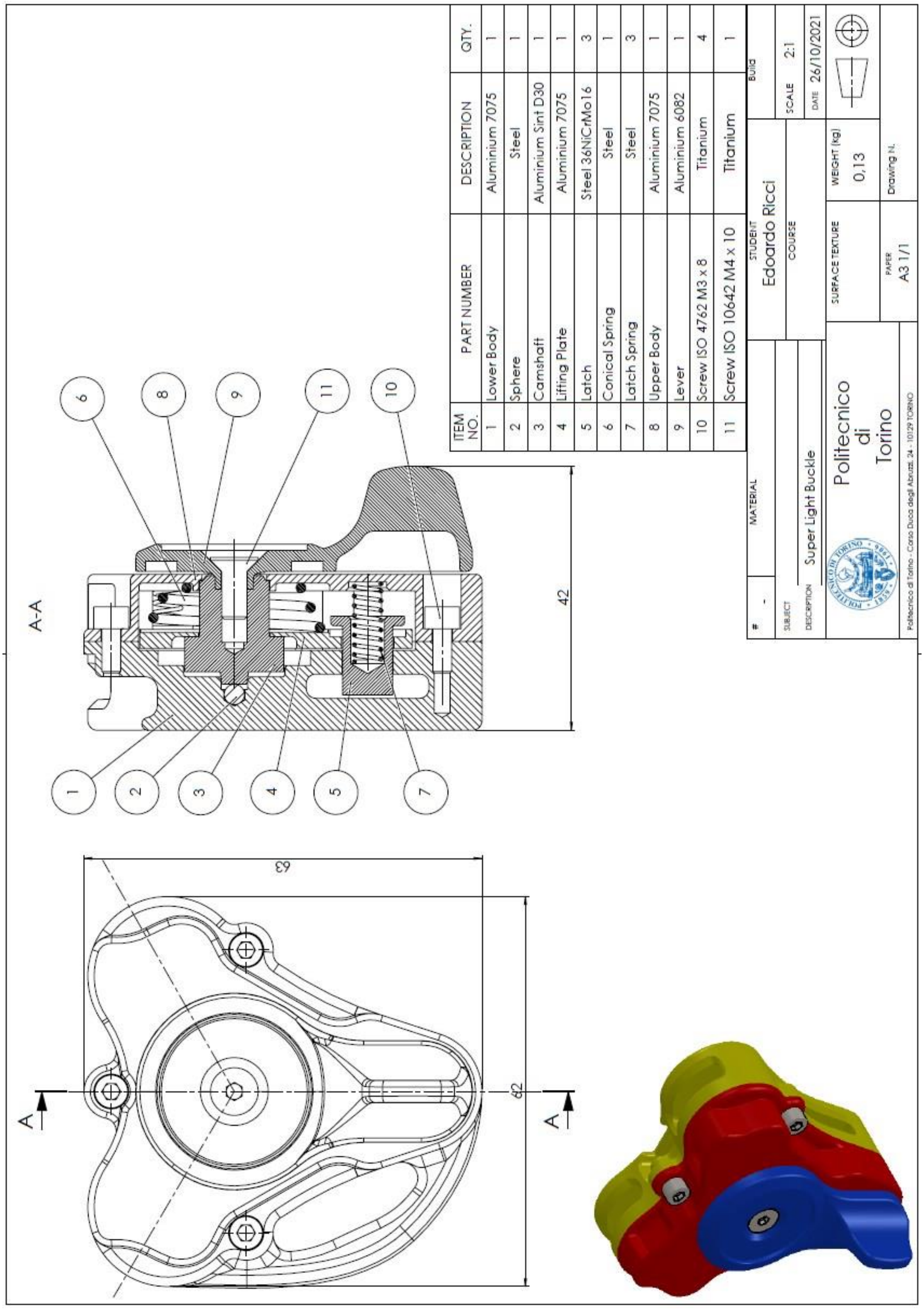

Figure A-1 – Buckle assembly.

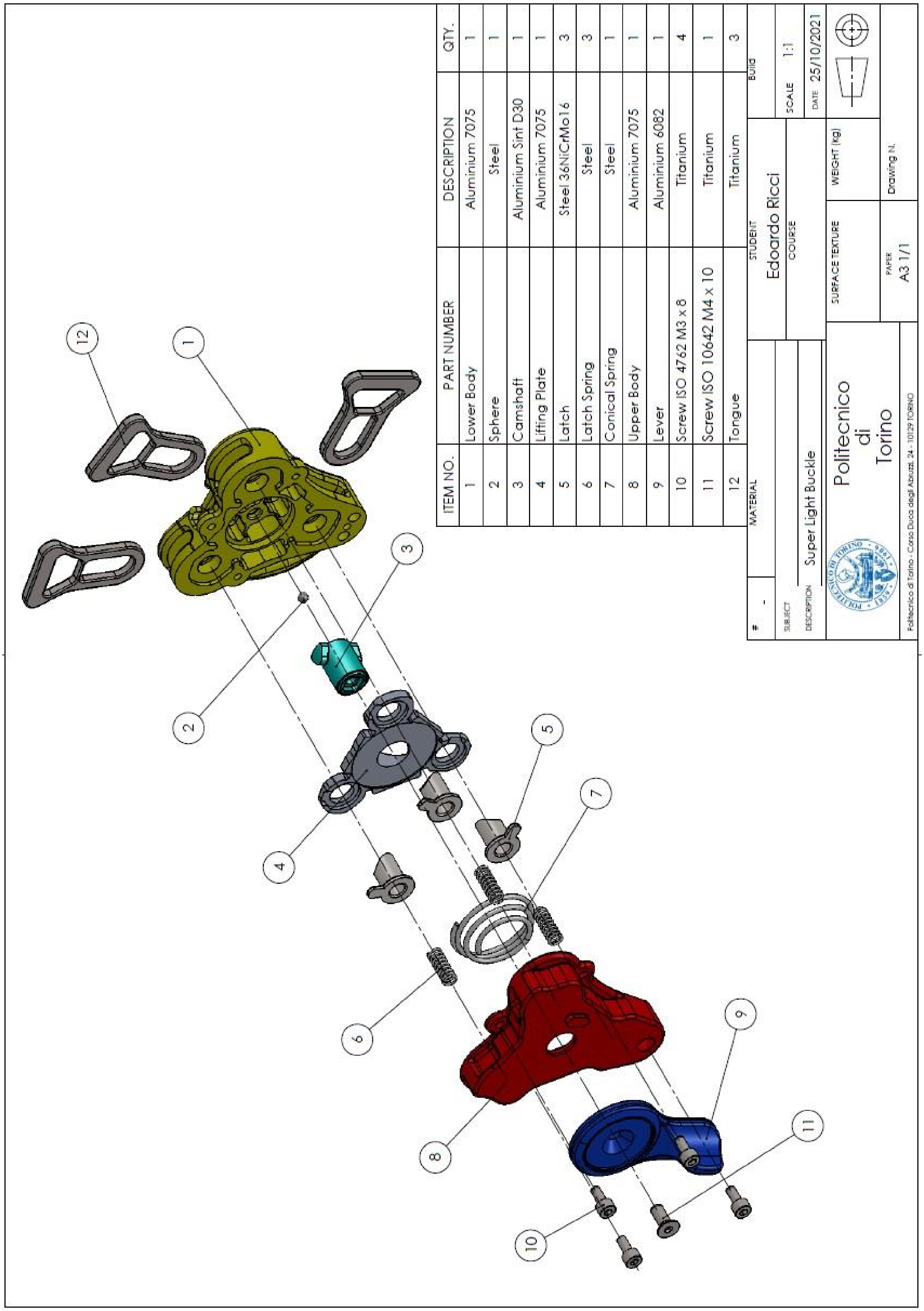

Figure A-2 – Exploded buckle assembly.

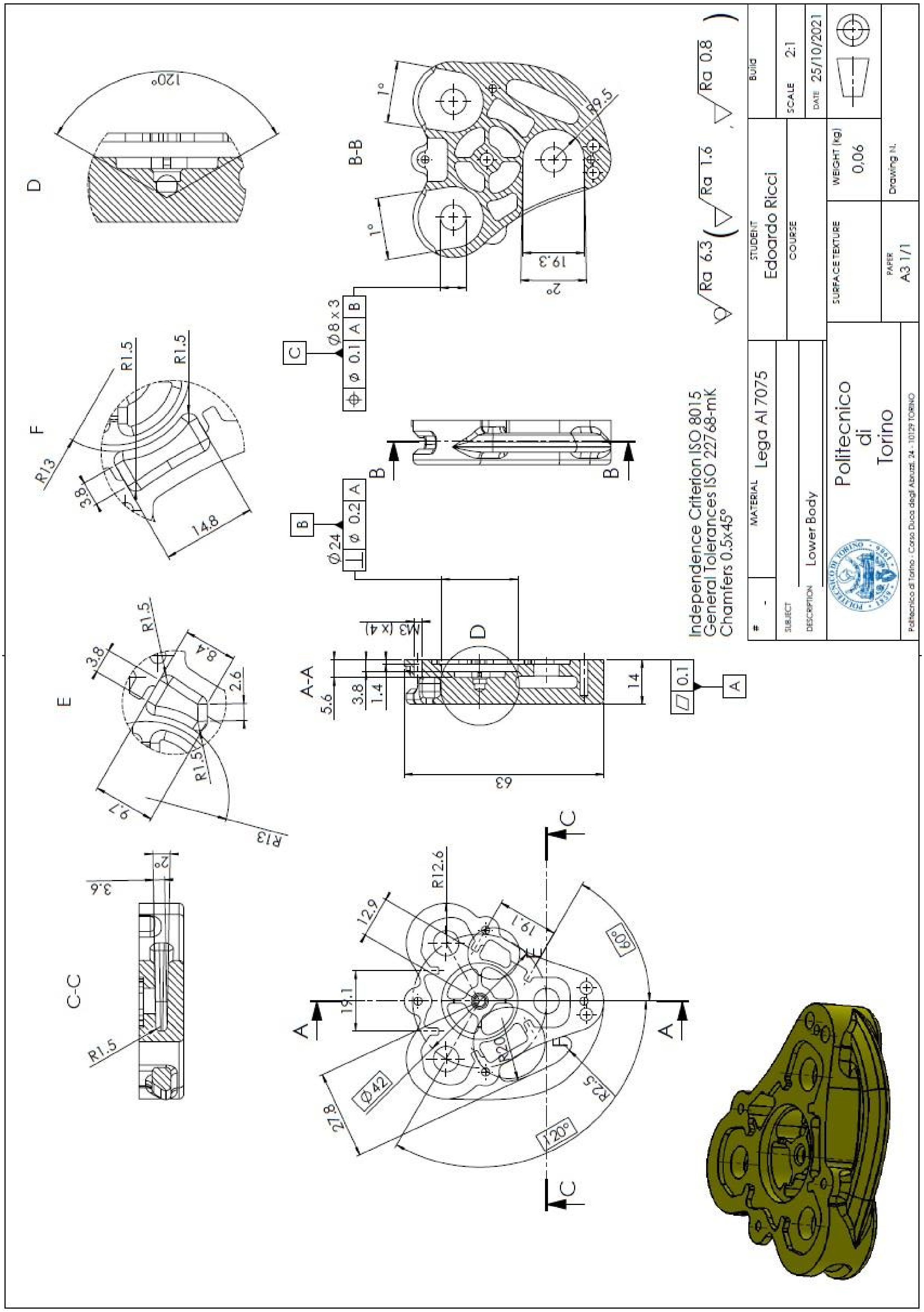

Figure A-3 – Lower body.

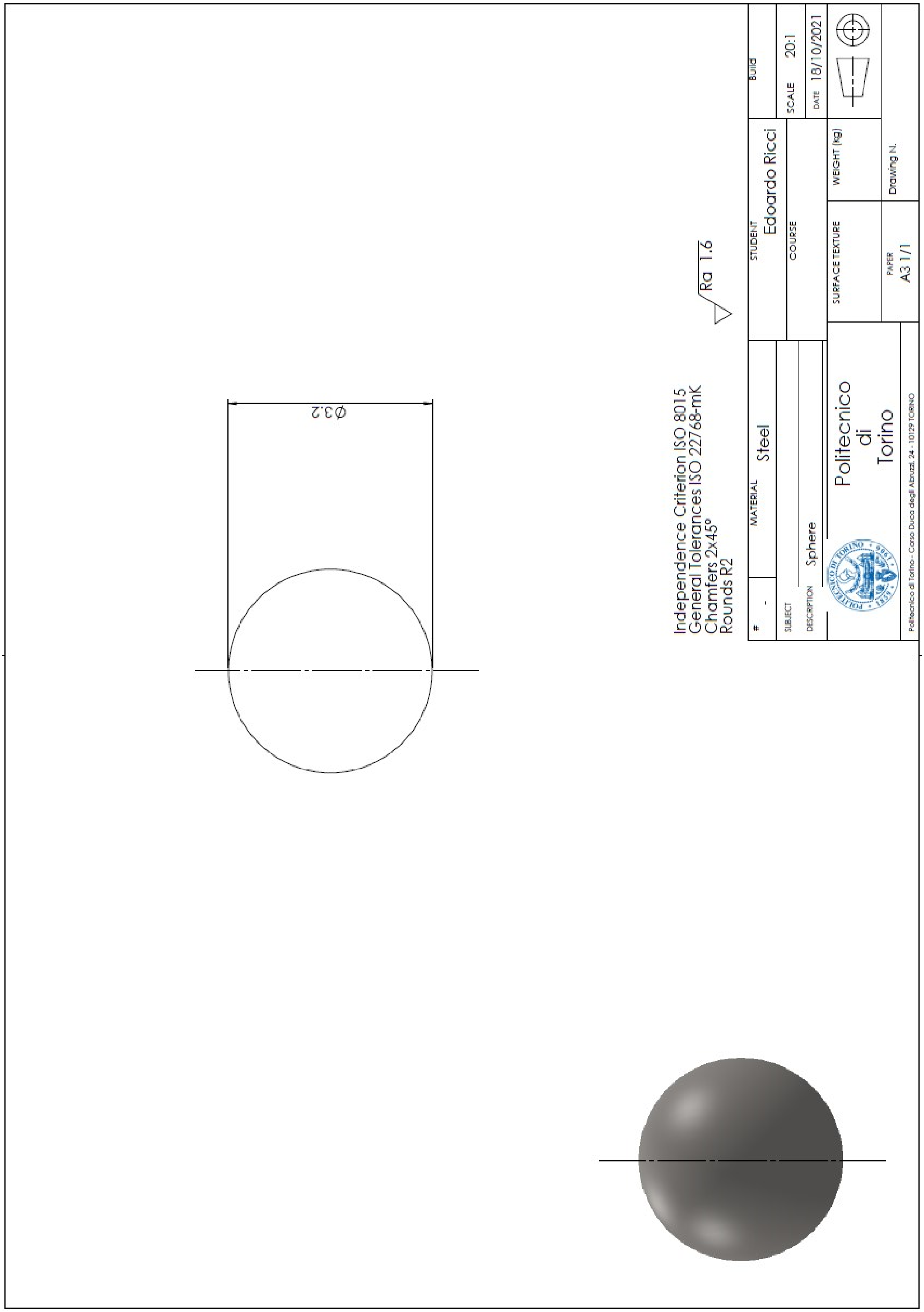

Figure A-4 – Sphere.

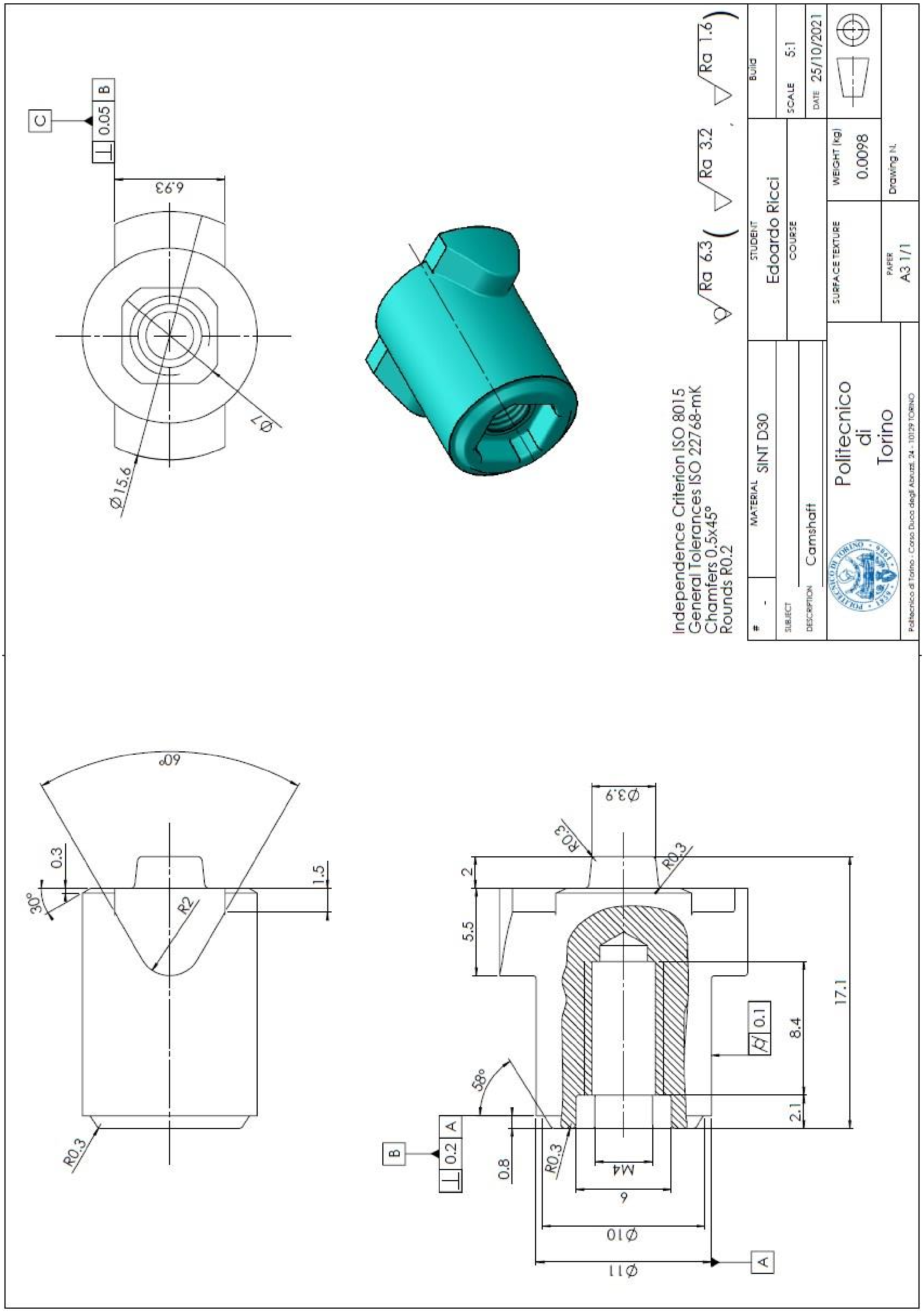

Figure A-5 – Camshaft.

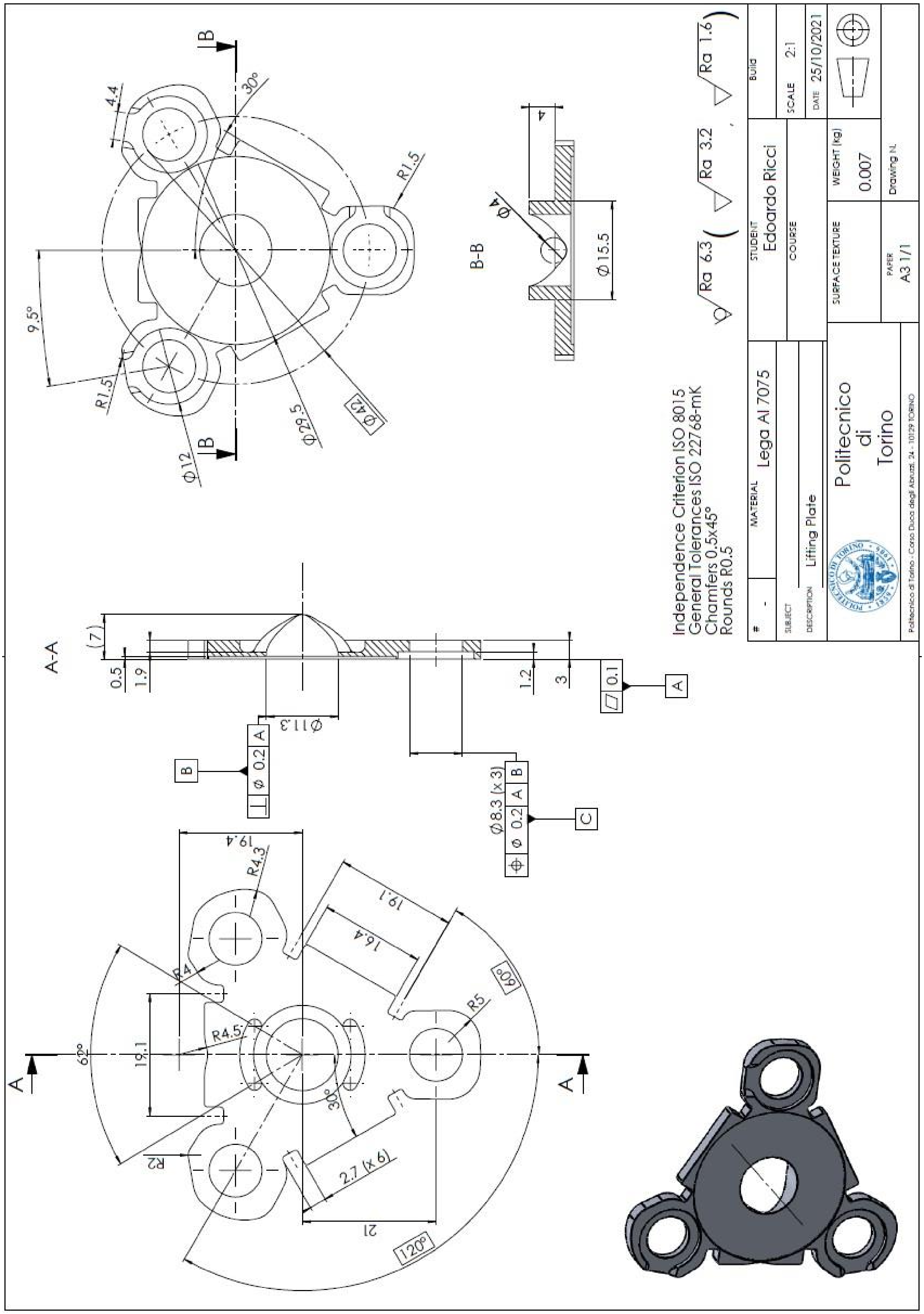

Figure A-6 – Lifting plate.

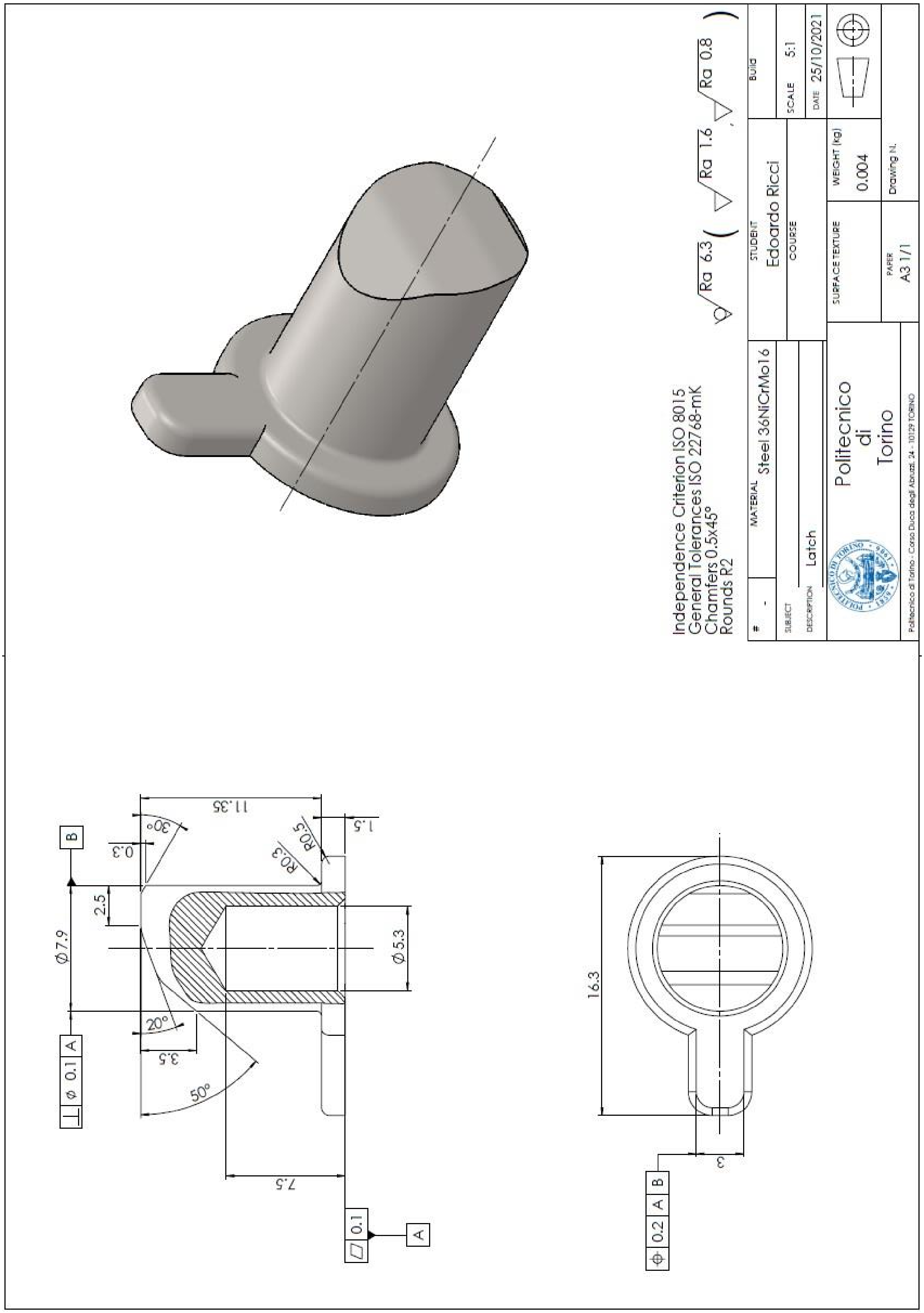

Figure A-7 – Latch.

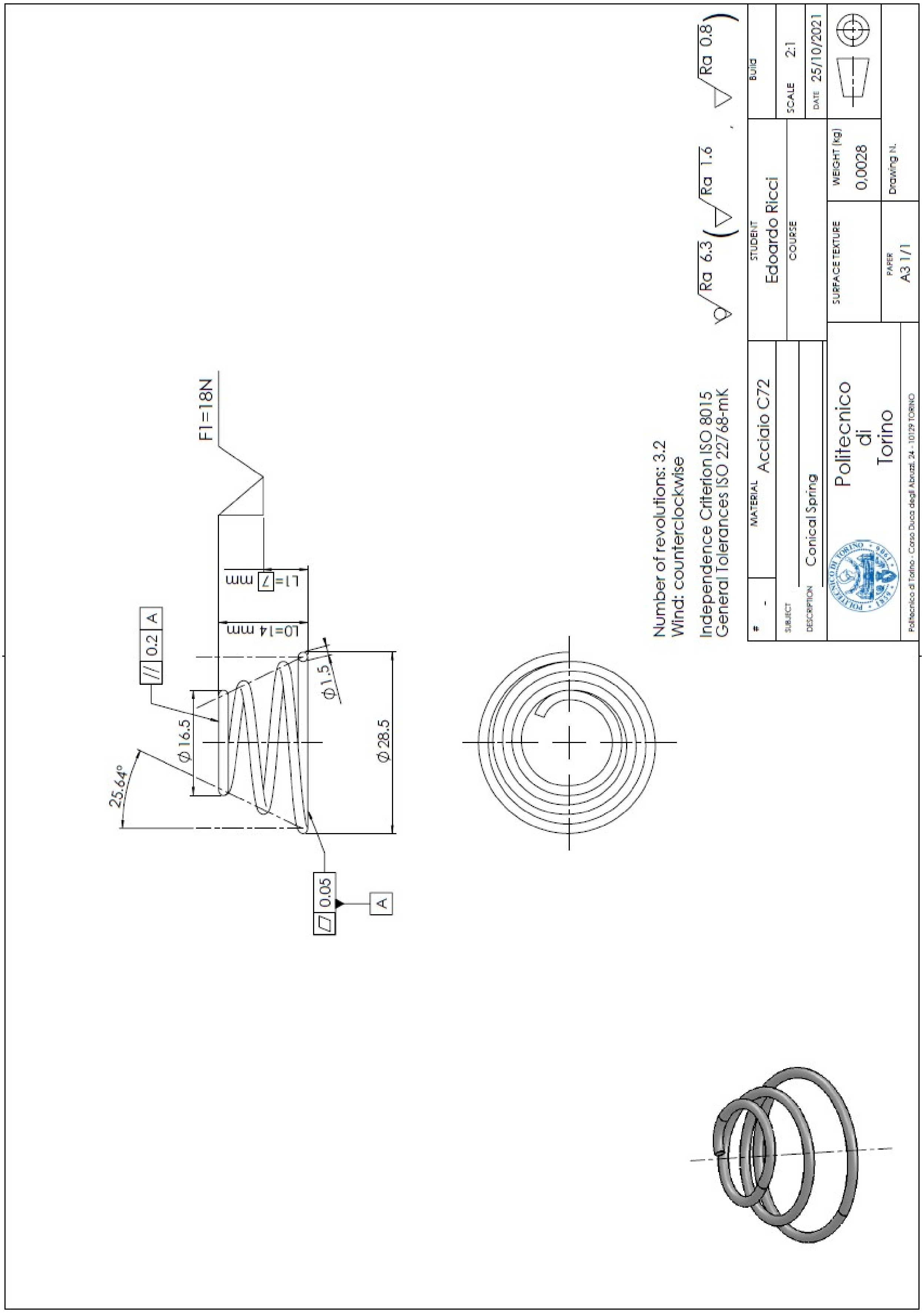

Figure A-8 – Conical spring.

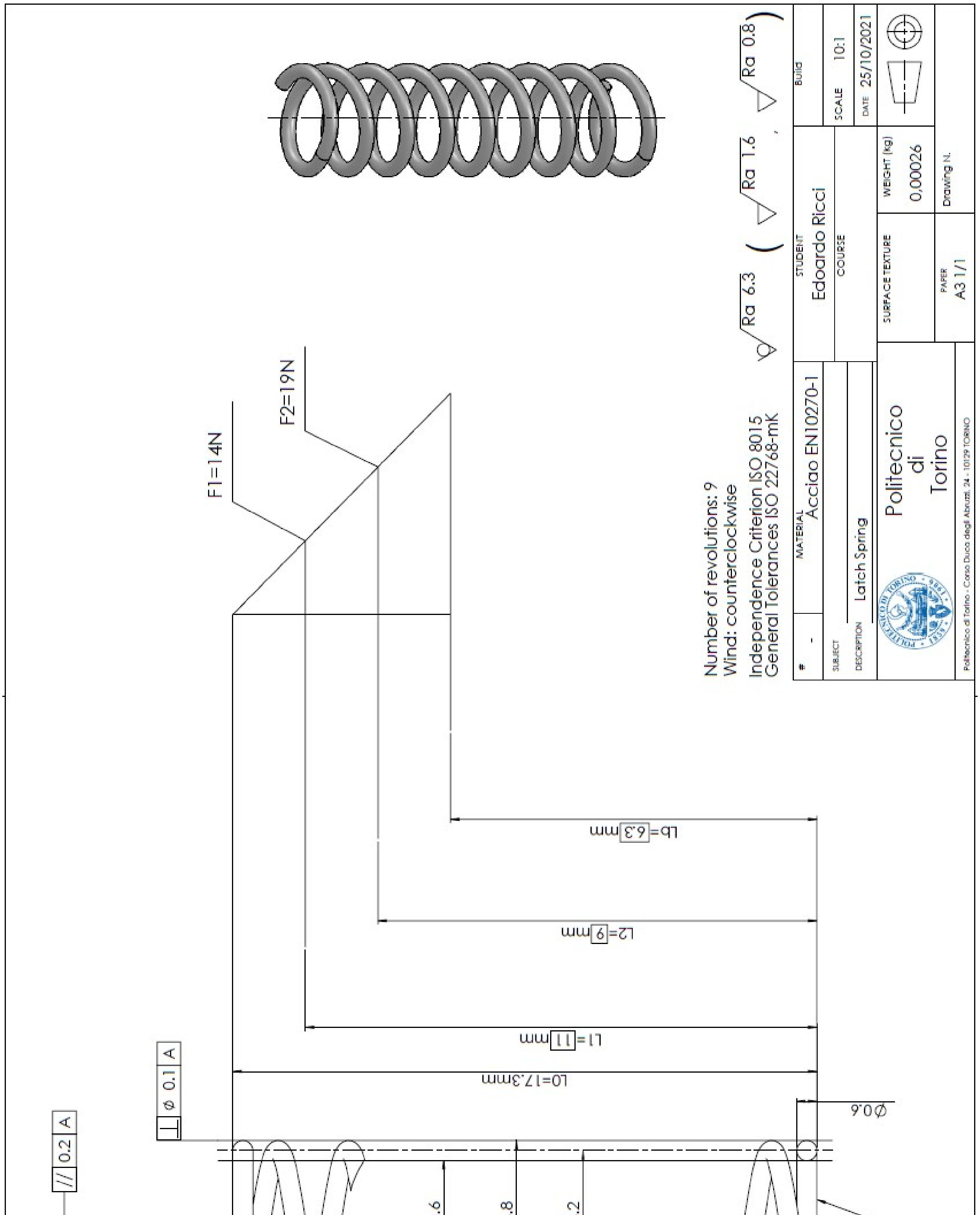

Figure A-9 – Conical spring.

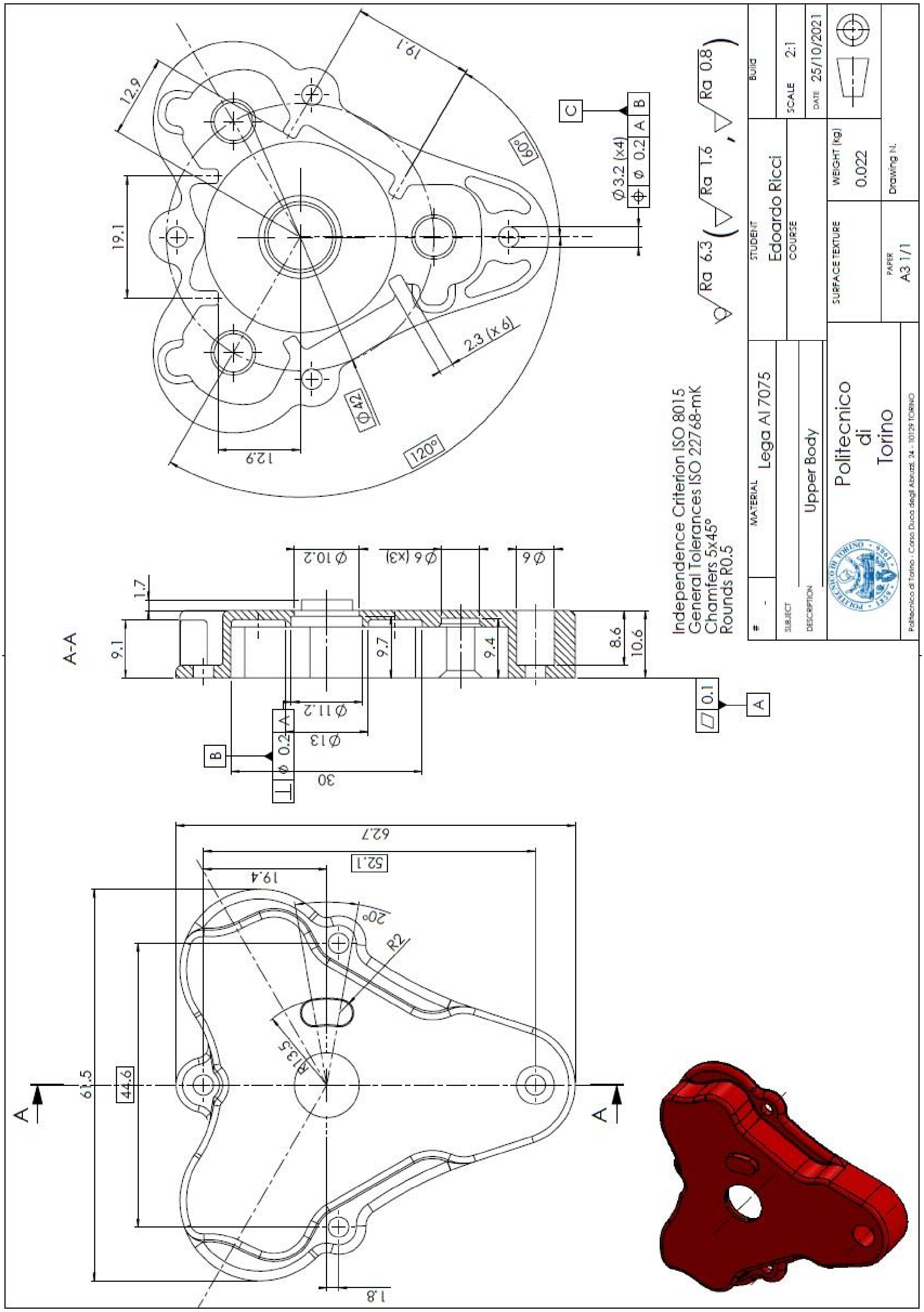

Figure A-10 – Upper body.

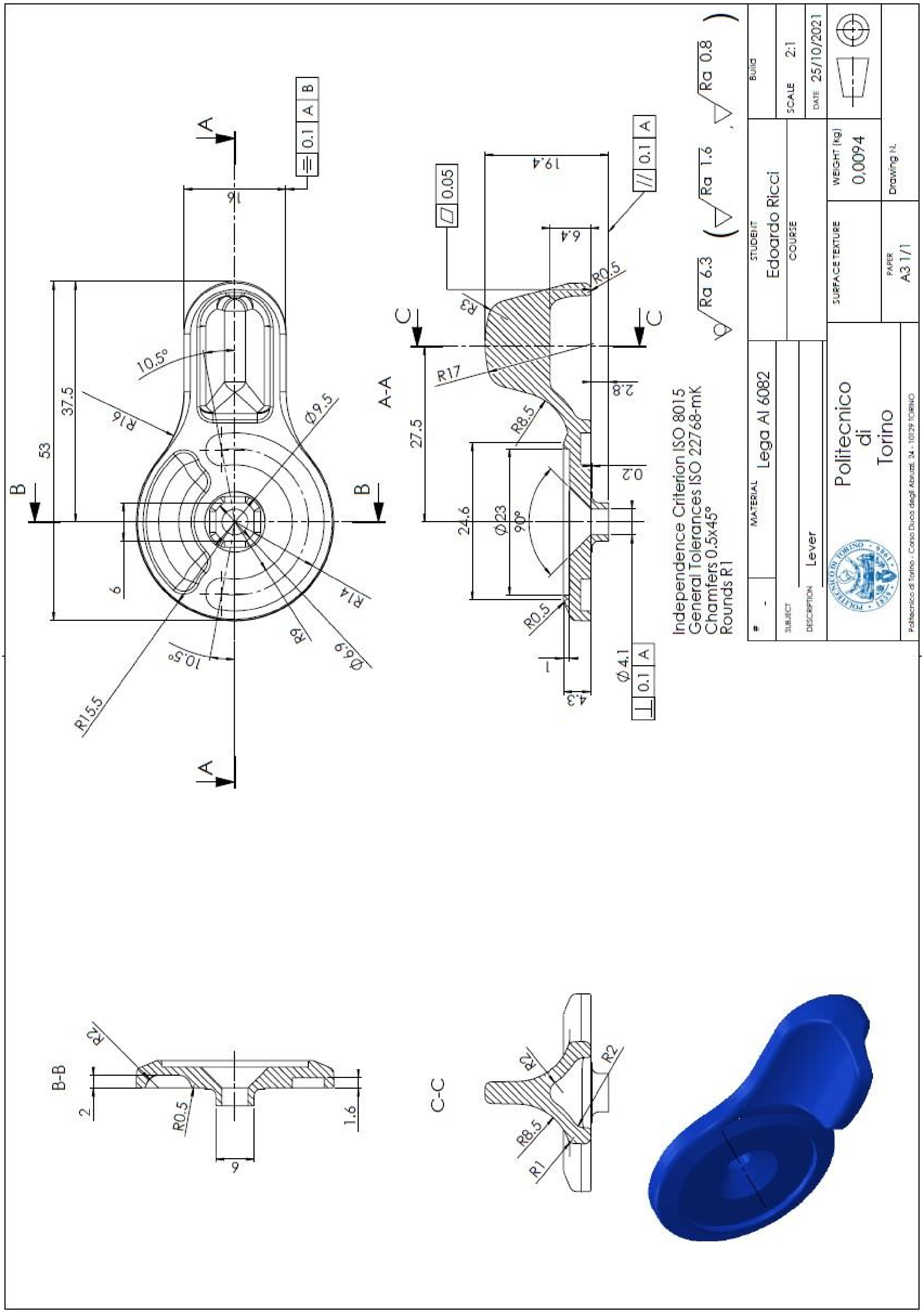

Figure A-11 – Lever.

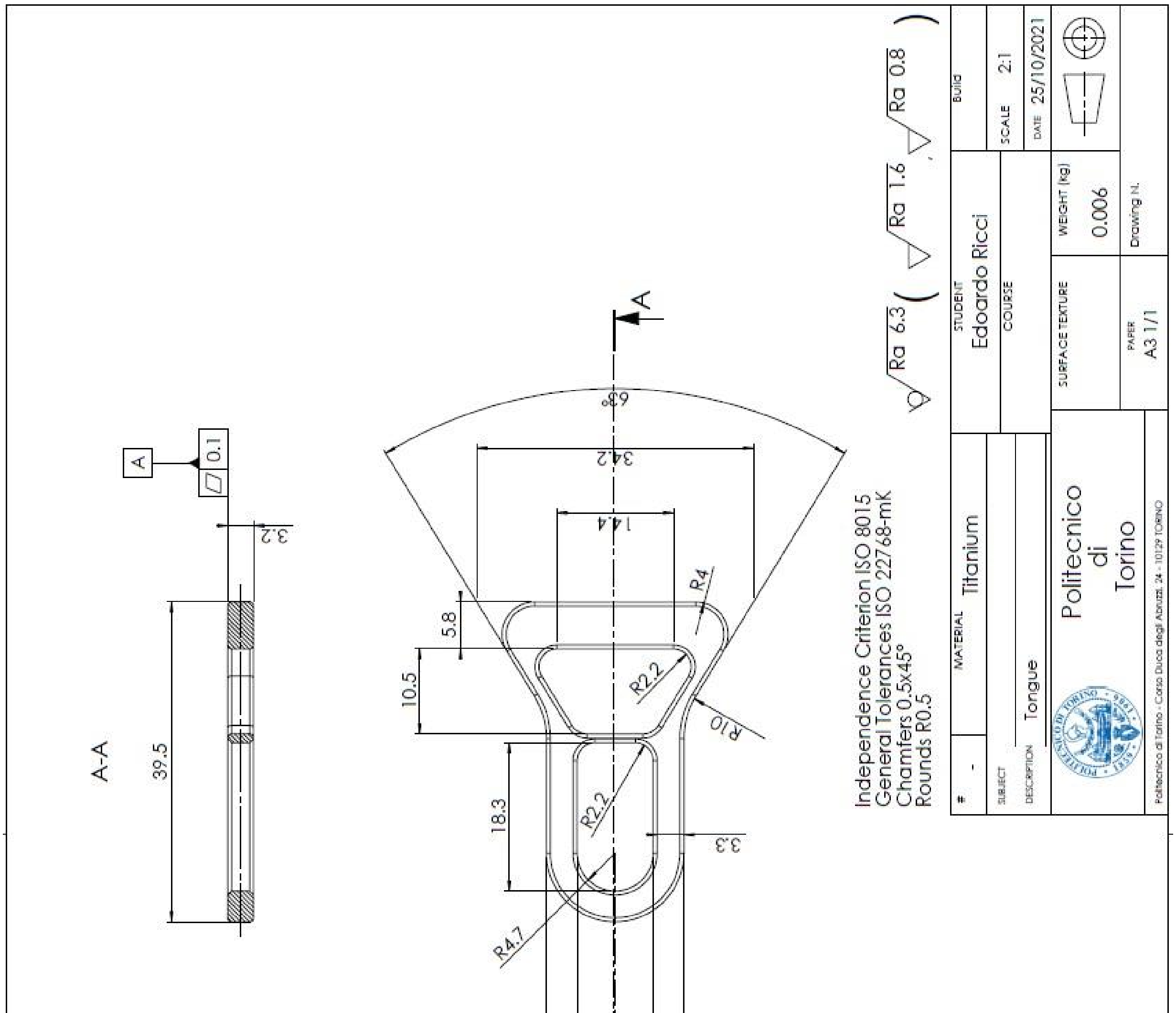

Figure A-12 – Tongue.

Reference

[1] Bonisoli E., LUPOS: Lumped Parameters Open Source FEM code, Tutorial v.2019-08-16, Politecnico di Torino, Departement of Mechanical and Aerospace Engineering, 2019.

[2] Bonisoli E., *Tools and tech for Prod Develop*, PhD Course, I Facoltà di Ingegneria Politecnico di Torino, 2014.

[3] Corrado P., *CAD-CAE Integration: from parametric models to static, buckling and modal with stress analyses*, MSc Thesis, I Facoltà di Ingegneria Politecnico di Torino, 2015.

[4] Thomée V., *From finite differences to finite elements: A short history of numerical analysis of partial differential equations*, 2001.

[5] https://www.simscale.com/docs/simwiki/preprocessing/what-is-a-mesh available in 2020-09-02

[6] https://www.simscale.com/blog/2017/01/convergence-finite-element-analysis/ available in 2020-03-10

[7] http://help.solidworks.com

[8] https://deust.wordpress.com/2014/11/30/h-method-p-method/ available in 2021-10-31

[9] Kurowski P., *Good Solid Modeling, Bad FEA*, 2002

[10] Jhonson K.L., *Contact mechanics*, Cambridge University Press, 1985.

[11] Chirone E., Tornincasa S., *Disegno tecnico industriale vol.2*, Il Capitello, Torino,2014.

[12] https://en.wikipedia.org/wiki/Sintering

[13] Oberg E., Jones F., Ryffel H., *Machinery's Handbook 23rd Edition.* New York: Industrial Press Inc., 1989.

[14] FIA Standard Safety Harnesses 8853-2016.

[15] Rethwisch D., Callister W., *Scienza e ingegneria dei materiali,* EdiSES, 1991.

[16] Carpinteri A., *Scienza delle costruzioni vol.2,* 1971, Pitagora, Bologna, 1995, pp. 70-71.