

ILLINOIS INSTITUTE
OF TECHNOLOGY



FINITE ELEMENT STUDY OF A 3 DEGREE OF FREEDOM ROBOT

RESEARCH PROJECT FOR MMAE597

Jon ander Sagarduy



INDEX

FINITE ELEMENT STUDY OF A ROBOT	4
Introduction:	4
CAD model:	4
Static Analysis	21
Results for the 250000N load:.....	25
Results for the 500000N load:.....	28
Results for the 1000000N load:.....	29
Dynamic Analysis	32
Conclusions:	35

Index of pictures

Picture 1: General model.....	4
Picture 2: Catya design.....	5
Picture 3: Sketch of the lower accomodation.....	5
Picture 4: Front design of the lower accomodation.....	6
Picture 5: back design of the lower accomodation.....	6
Picture 6: Catya design of the cap.....	7
Picture 7: Sketch of the circular cross-section bar.....	7
Picture 8: Catya design of the circular section bar.....	7
Picture 9: Sketch of the rectangular section bar.....	8
Picture 10: catya design of the rectangular section bar.....	8
Picture 11: Sketch of the crank rod crap.....	9
Picture 12: Catya design of the crank rod crap.....	9
Picture 13: Catya design of the guides.....	10
Picture 14: Sketch of the platform.....	10
Picture 15: Catya design of the platform.....	11
Picture 16: Catya design of the assembly of the different componenets of the structure.....	12
Picture 17: Position of the three guides.....	12
Picture 18: Assembly of the spherical DOF.....	13
Picture 19: General 2D mesh in Hypermesh.....	14
Picture 20: List of the elements that have been created in the Hypermesh file.....	16
Picture 21: 3D mesh with the rigid.....	16
Picture 22: Upper view of the rigid.....	17
Picture 23: List of the entity sets created in Hypermesh.....	17
Picture 24: Elements forming the contact entity set.....	18
Picture 25: Nodes forming the fixed entity set.....	18
Picture 26: Input for the static analysis in the .inp file.....	20
Picture 28: Assembly in Abaqus.....	22
Picture 29: Example of a tie constraint in Abaqus.....	22
Picture 30: Load set up in Abaqus.....	23
Picture 31: Boundary conditions set up.....	24
Picture 32: Mesh in Abaqus.....	24
Picture 33: Stress and displacement with a 250000N load.....	25
Picture 34: Peak stress.....	25
Picture 35: Time when elasticity first happens in the cross section bar.....	26
Picture 36: Stress in the rotational DOF.....	27
Picture 37: Stress and displacement with a 500000N load.....	28
Picture 38: Peak stress.....	28
Picture 39: Stress in the rotational DOF.....	29
Picture 40: Stress and displacement with a 1000000N load.....	29
Picture 41: Peak stress.....	30
Picture 42: Time when elasticity first happens in the cross section bar.....	30
Picture 43: Stress in the rotational DOF.....	31



Picture 44: Dynamic load.....	32
Picture 45: Stress and displacement with a dynamic load	32
Picture 46: peak stress with the dynamic load	33
Picture 47: Stress in the rotational DOF.....	34

FINITE ELEMENT STUDY OF A ROBOT

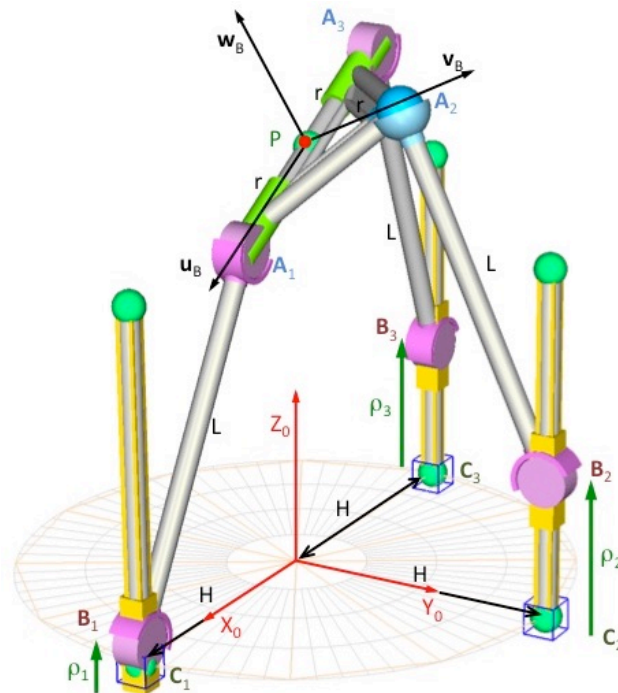
Introduction:

This project studies the previous work done by Jon Ander Sagarduy in the University of Basque Country. It studies the case of a three degree of freedom arm and its kinematical and structural analysis with the help of Finite Elements. To do so, there are three steps.

- First of all, the creation of the model in CAD. This is done with Catia v5, instead of Solid Works as it was done in the previous work.
- Second, is the mesh of the assembly with Hypermesh.
- Finally, the study of the different stresses and materials with Abaqus, with different movement laws.

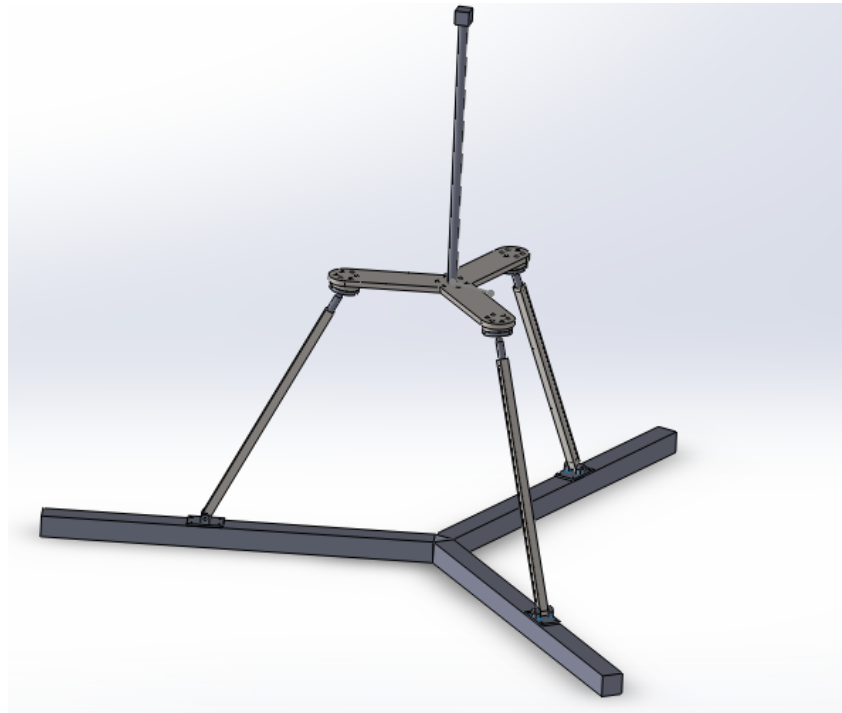
CAD model:

As it was explained before, the first thing that it has to be done is the creation of a CAD model in Catia v5. To do so, the student will take into account the drawings used in the bachelor's research project. The model in Solid works looks like this:



Picture 1: General model

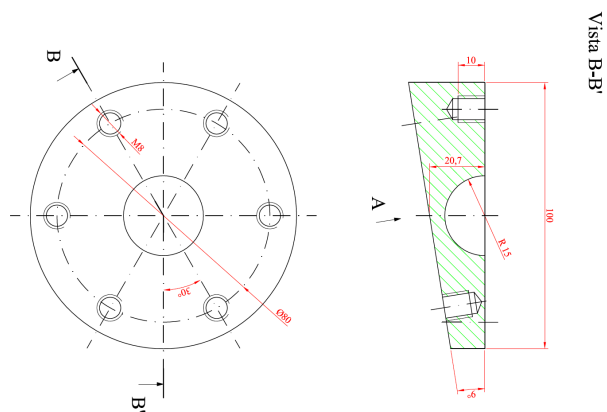
However, when doing this analysis another position of the fixed elements can be used. They can be fixed to the ground, forming a circumference separated from each other 120° :



Picture 2: Catya design

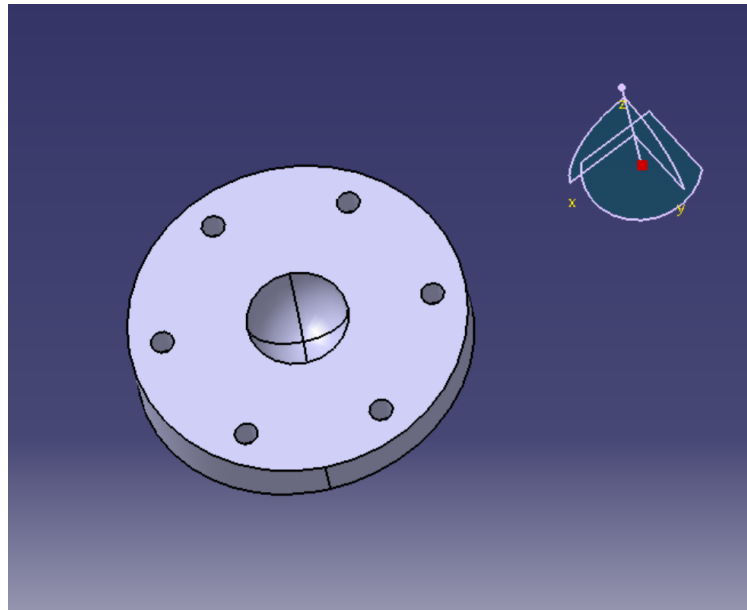
Taking into account these two position of the fixed part, the student must proceed to create the different elements in Catia v5.

1. The first part will be the lower accommodation of the sphere that will allow the spherical degree of freedom.



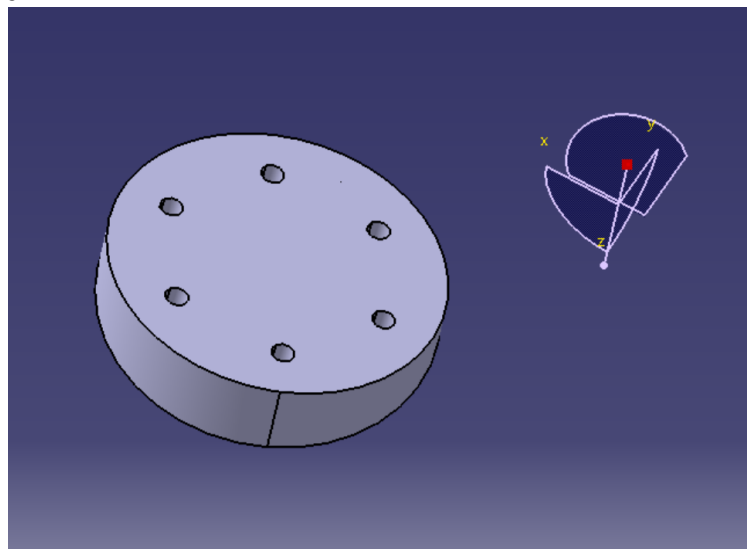
Picture 3: Sketch of the lower accommodation

Which in Catia will be:



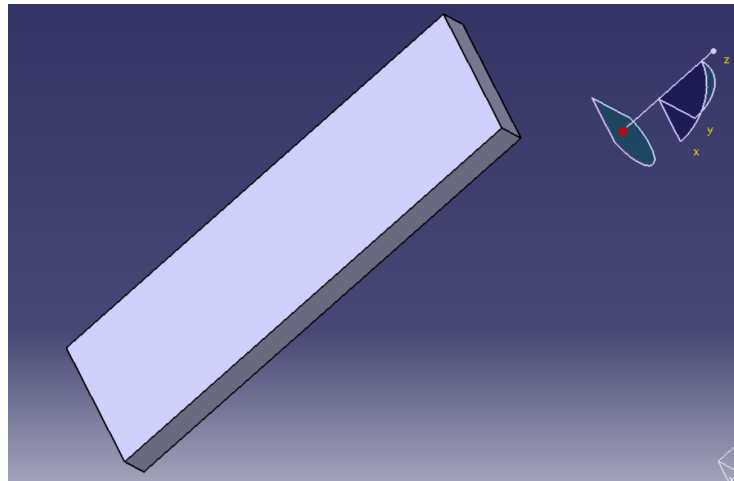
Picture 4: Front design of the lower accommodation

and the back:



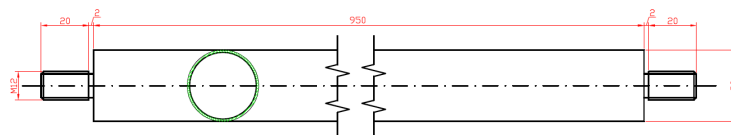
Picture 5: back design of the lower accommodation

2. The cap support has the following configuration:



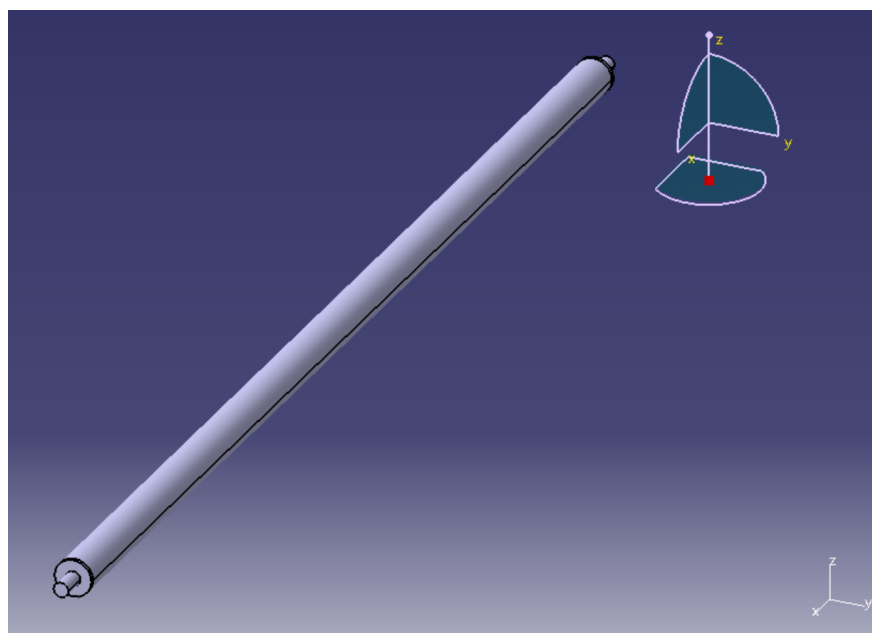
Picture 6: Catya design of the cap

3. The third element created in Catia will be the circular cross-section bar:



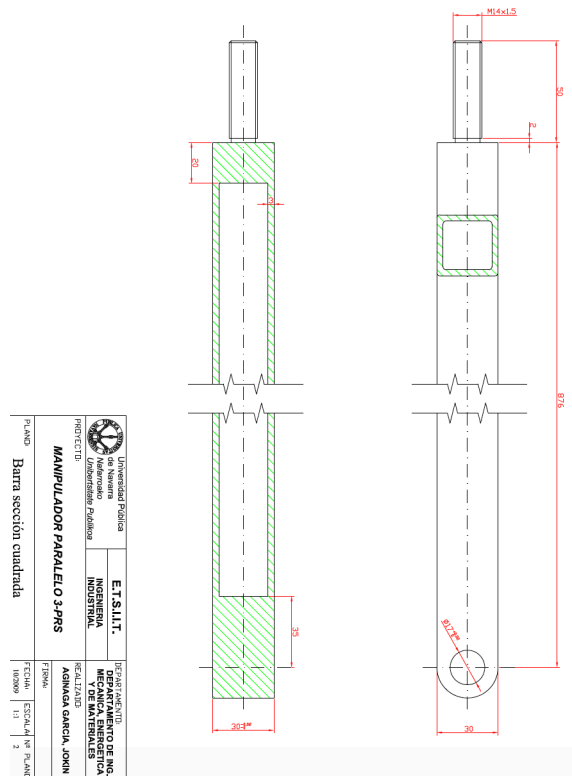
Picture 7: Sketch of the circular cross-section bar

which in Catia looks like:

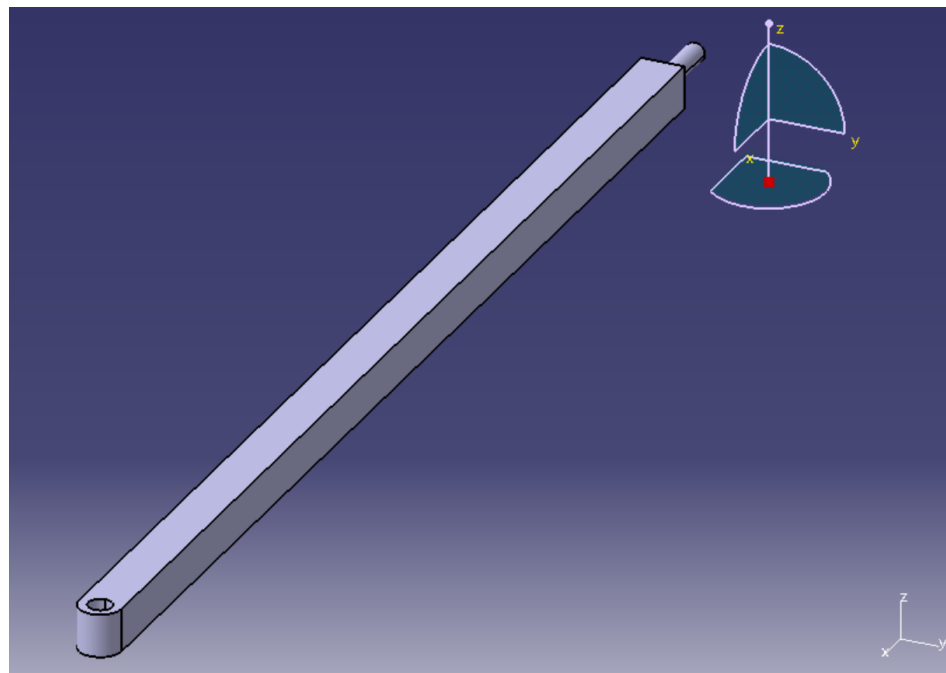


Picture 8: Catya design of the circular section bar

4. The fourth element done in Catia is the rectangular section bar:

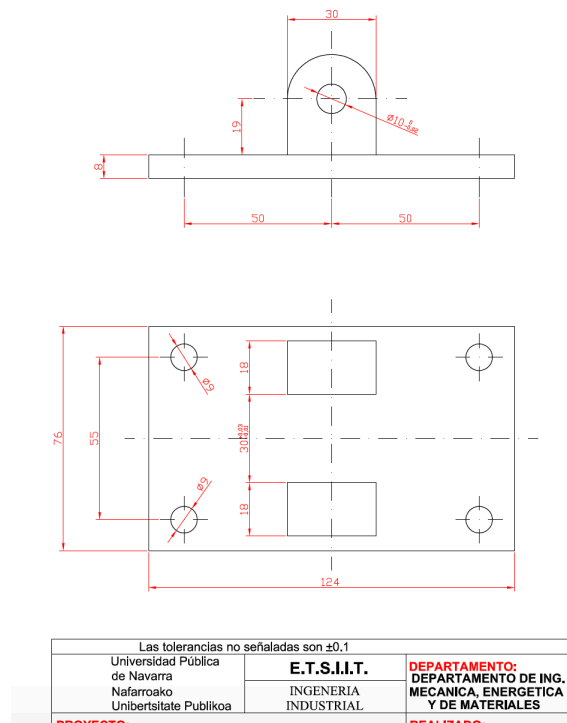


Picture 9: Sketch of the rectangular section bar



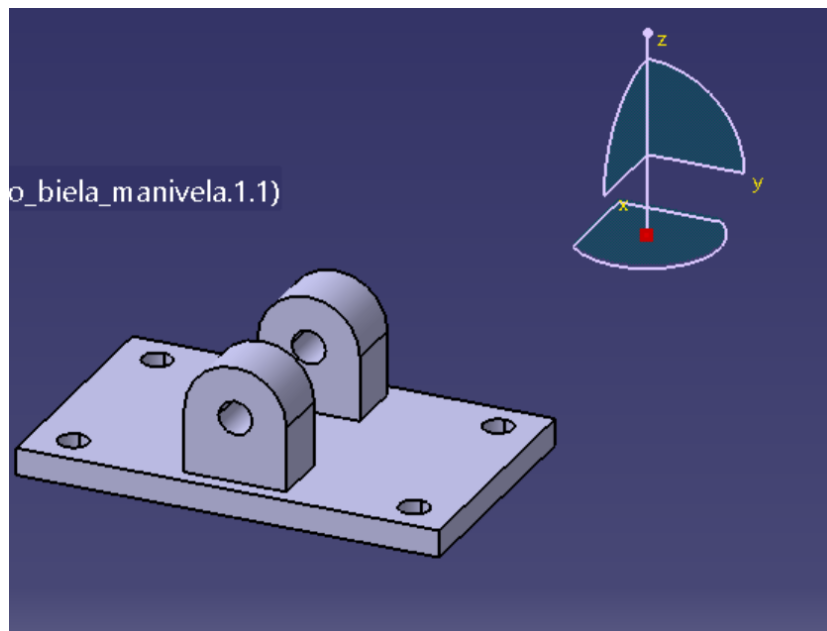
Picture 10: catya design of the rectangular section bar

5. The fifth element will be the component that is made of the crank rod crap:



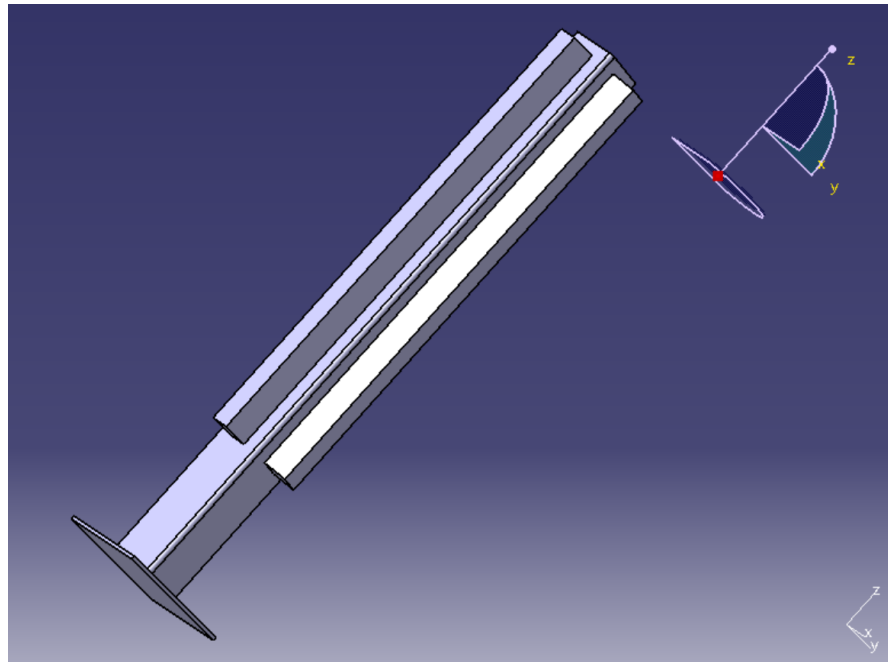
Picture 11: Sketch of the crank rod crap

which in Catia will look like:



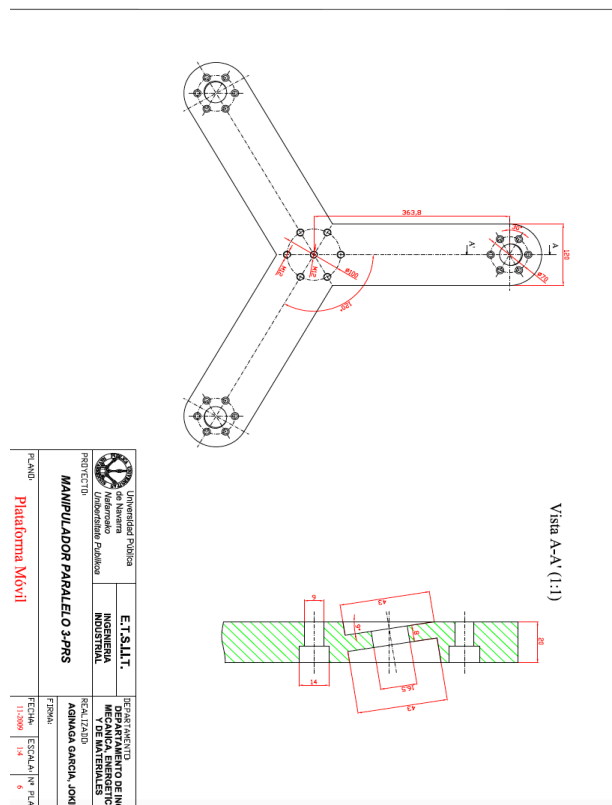
Picture 12: Catya design of the crank rod crap

6. The sixth element created is the fixed base which will work as the guide of the movement:

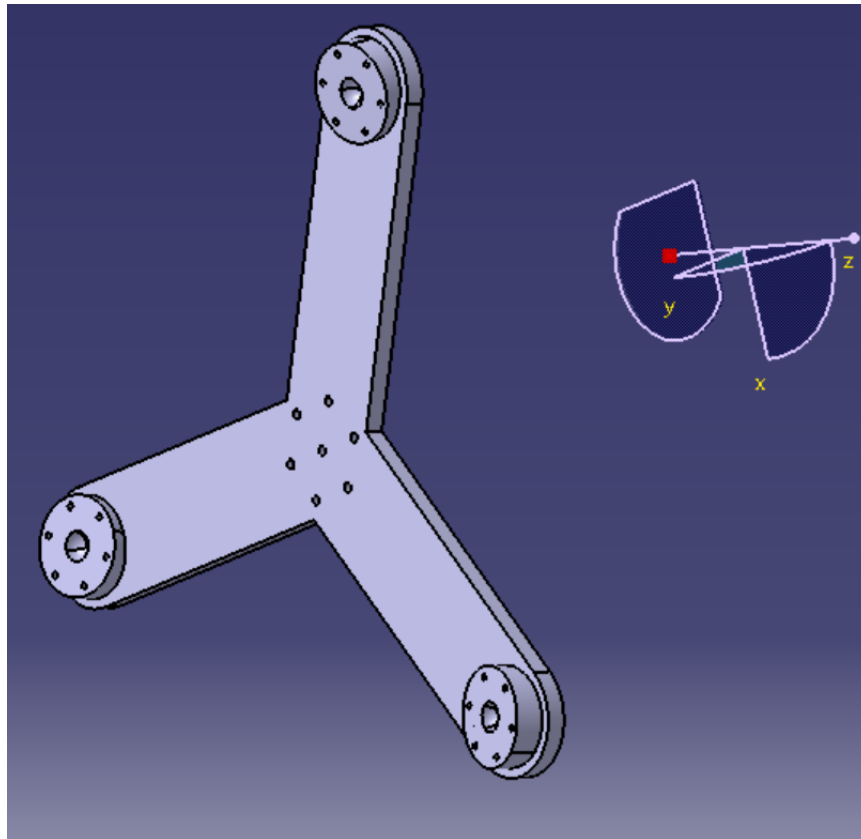


Picture 13: Catya design of the guides

- The seventh element is the platform, which will be movable, since it will be connected to the different bars that will allow its movement:

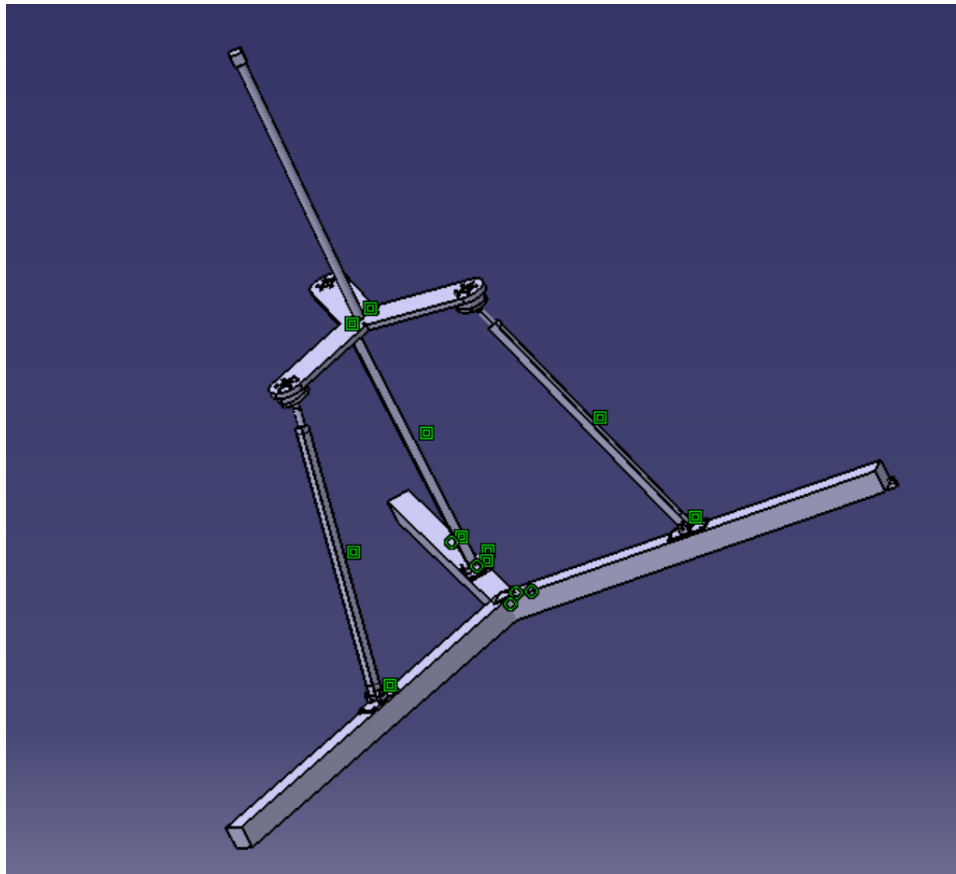


Picture 14: Sketch of the platform



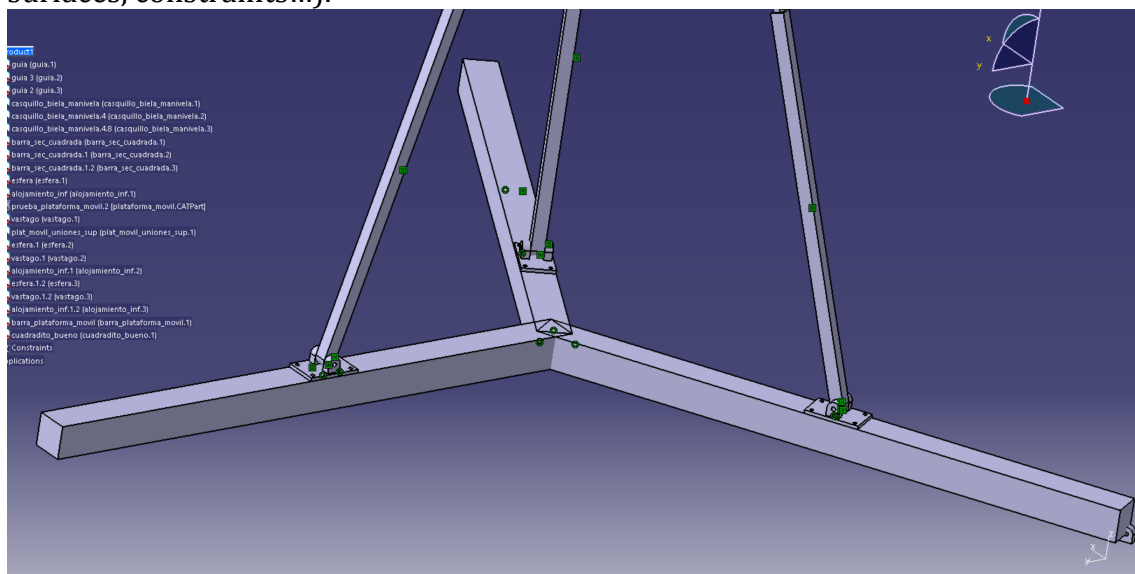
Picture 15: Catya design of the platform

Once all of the components have been described previously, it is time to make an assembly with all of them in order to have the final assembly. The assembly is done with Catya v.5 in the modulus of Assembly. The final appearance has the following form:



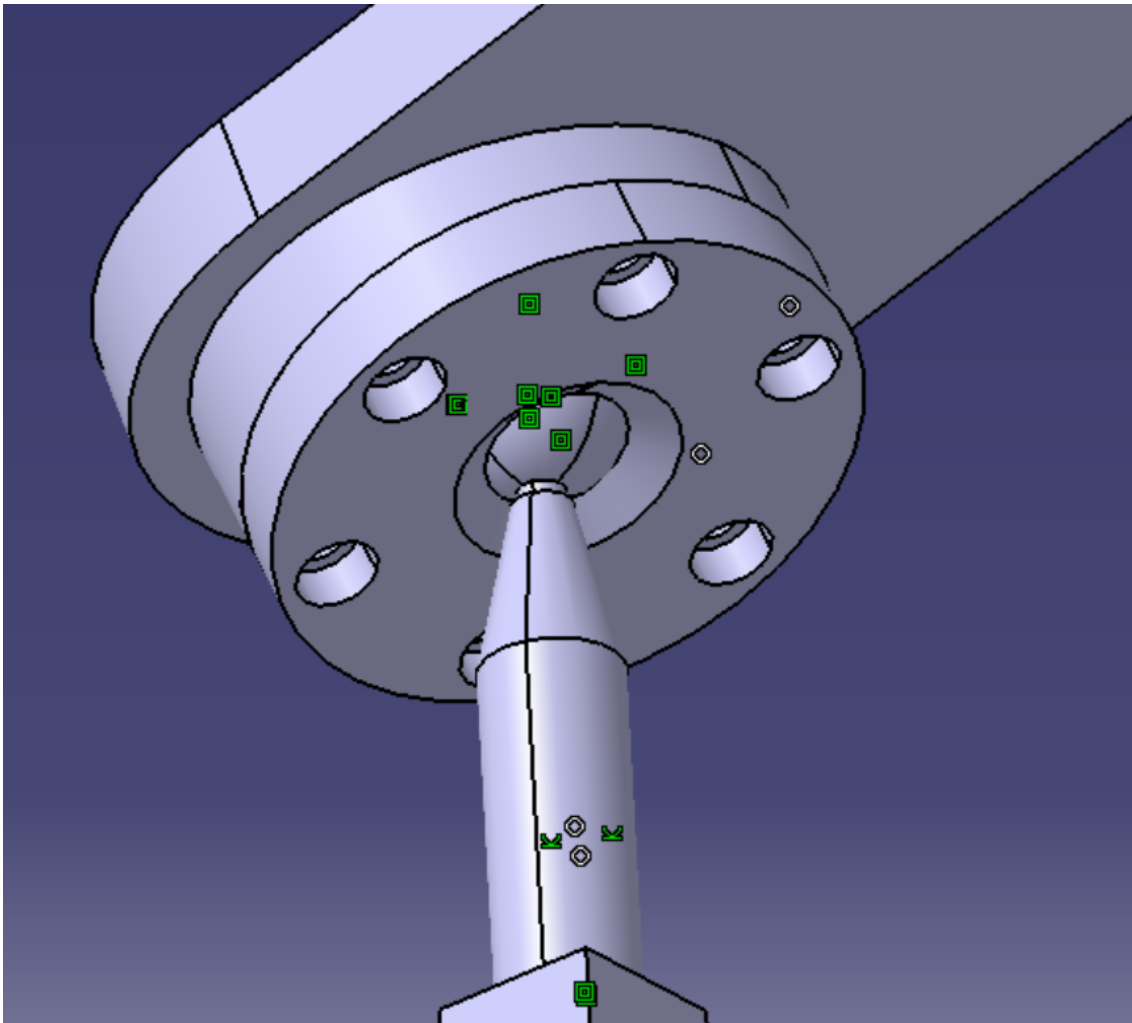
Picture 16: Catya design of the assembly of the different components of the structure

In this assembly all the necessary constraints have been defined (contact between surfaces, constraints...).



Picture 17: Position of the three guides

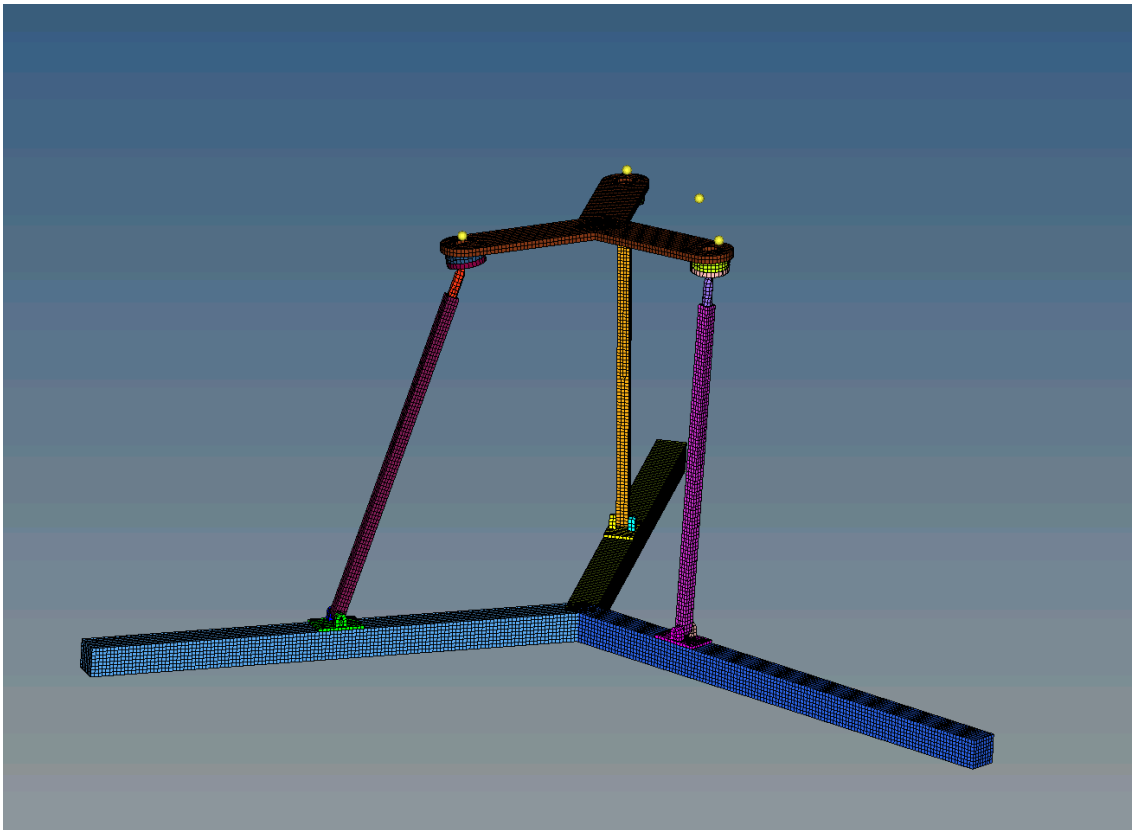
The picture above shows has the guides have been settled and fixed to the ground.



Picture 18: Assembly of the spherical DOF

The picture above shows how is done the union between the sphere, the upper and lower accommodations, the platform and the rectangular section bar.

Once the final assembly is done in Catya v.5, the next step will be to study the different analysis that can be carried out in this robotic arm: static and dynamic analysis. In order to do so, it is necessary to create a mesh. For that Hypermesh will be used. With an element size of 20mm the robot looks as follows:



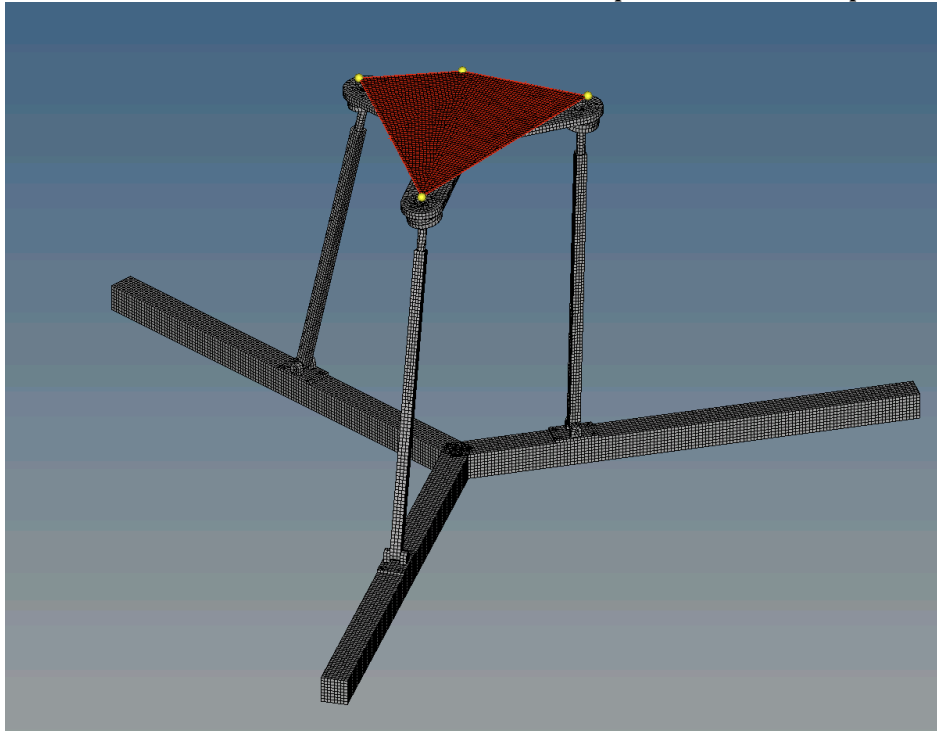
Picture 19: General 2D mesh in Hypermesh

Once all the components have been meshed, 3D elements will be created in order to study the different analysis. To do so, it is necessary to create as many 3D elements as 2D elements exist:

Component (54)		
alojamiento-inf-3D	38	■
alojamiento-inf1-3D	36	■
alojamiento-inf2-3D	37	■
alojamiento-sup-3D	39	■
alojamiento-sup1-3D	40	■
alojamiento-sup2-3D	41	■
alojamiento_inf	17	■
alojamiento_inf.1	23	■
alojamiento_inf.1.2	26	■
alojamiento_sup.1	20	■
alojamiento_sup.1.1	27	■
alojamiento_sup.1.2	28	■
barra-sec-cuadrada-3D	34	■
barra-sec-cuadrada1-3D	33	■
barra-sec-cuadrada2-3D	32	■
barra_sec_cuadrada	13	■
barra_sec_cuadrada.1	14	■
barra_sec_cuadrada.1.2	15	■
base-3D	42	■
base1-3D	43	■
base2-3D	44	■
base_casquillo_biela_manivela	4	■
base_casquillo_biela_manivela.2	7	■
base_casquillo_biela_manivela.2.6	10	■
enganche-3D	45	■
enganche1-3D	46	■
enganche2-3D	47	■
enganche3-3D	48	■
enganche4-3D	49	■
enganche5-3D	50	■
enganche_casquillo_biela_manivela	5	■
enganche_casquillo_biela_manivela.1	8	■
enganche_casquillo_biela_manivela.1.5	11	■
esfera	16	■
esfera.1	21	■
esfera.1.2	24	■
guia	1	■
guia 2	3	■
guia 3	2	■
guia-3D	29	■
guia2-3D	31	■
guia3-3D	30	■
plataforma-3D	51	■
prueba_plataforma_movil	19	■
Symmetry of enganche_casquillo_biela_manivela	6	■
Symmetry of enganche_casquillo_biela_manivela.3	9	■
Symmetry of enganche_casquillo_biela_manivela.3.7	12	■
vastago	18	■
vastago-3D	52	■
vastago.1	22	■
vastago.1.2	25	■
vastago1-3D	35	■
vastago2-3D	53	■
rigid	54	■

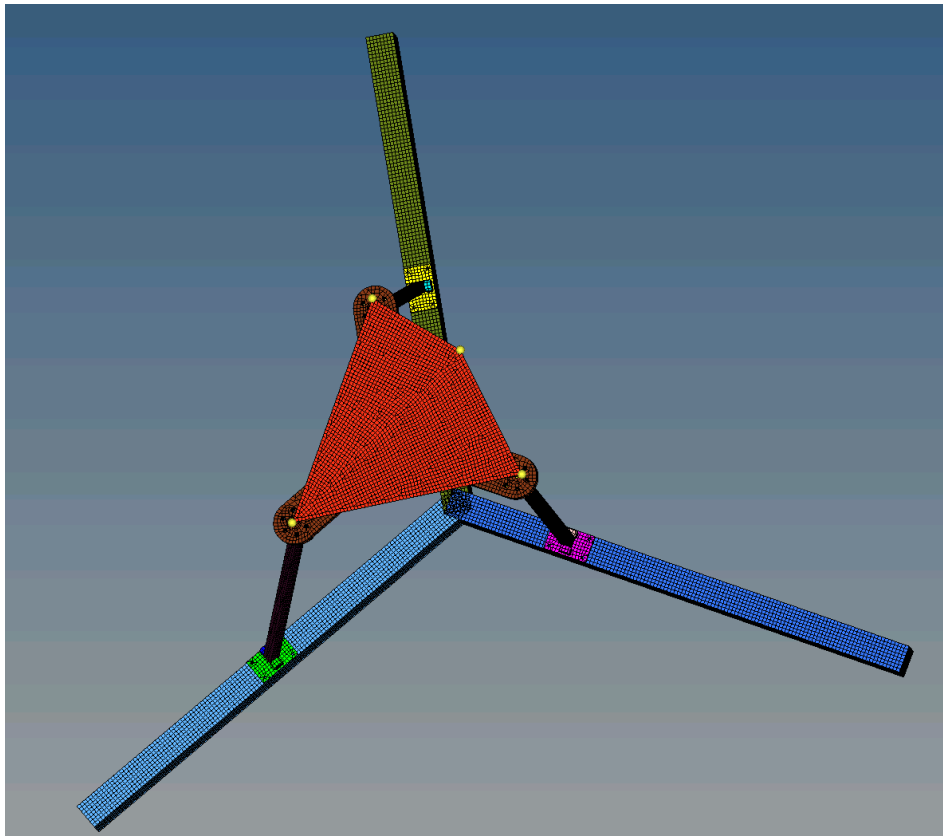
Picture 20: List of the elements that have been created in the Hypermesh file

A rigid has been created in order to simulate the displacement of the platform:



Picture 21: 3D mesh with the rigid

Or if it is seen from the upper part (with the 2D elements):



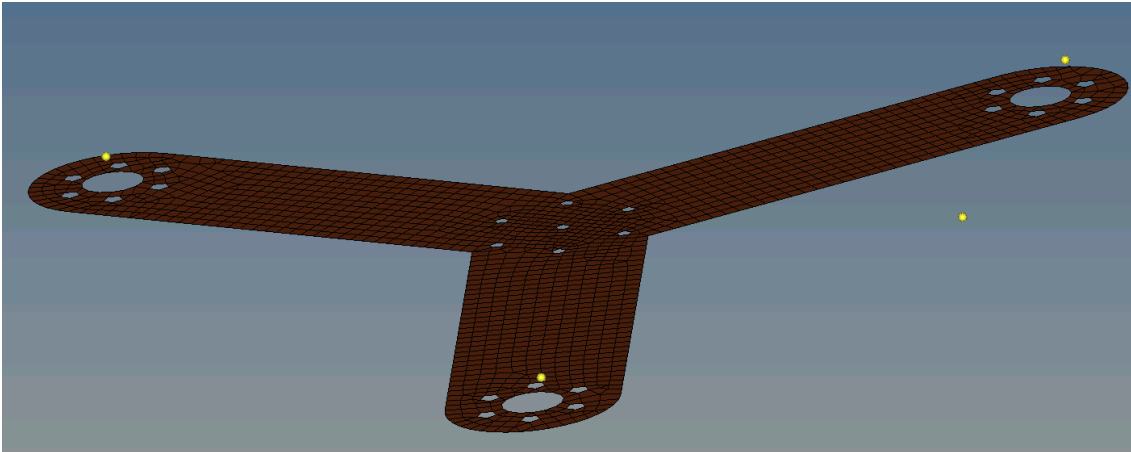
Picture 22: Upper view of the rigid

The following step in Hypermesh is to define the different entity sets that will take part in the simulation:

Set (3)	
contact	1
fixed	2
ref_n	3

Picture 23: List of the entity sets created in Hypermesh

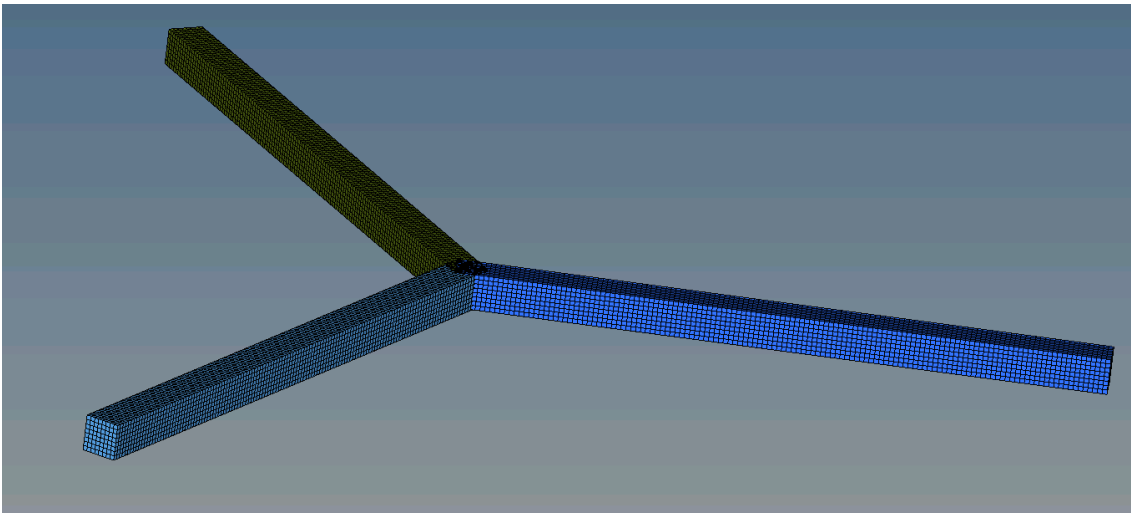
The first entity set defined will be the contact that refers to the elements that contact the rigid body:



Picture 24: Elements forming the contact entity set

As it can be seen from the picture above, the contact part will be the upper part of the platform, which is the part that contacts directly with the rigid body.

The next entity set defined is the fixed one. This entity set is defined in order to set fixed nodes and refer all the displacements towards them. In this case this entity set is set in the guides:



Picture 25: Nodes forming the fixed entity set

The last entity set defined is the reference node (ref_n). This reference node is defined in order to refer the boundary conditions of the different analysis. For the static analysis the reference node is located on the rigid body.

Once the Hypermesh is defined, it is time to create the .inp file that allows the static analysis. To do so, the Hypermesh model should be exported as a solver deck. Then, once it is exported, the next step will be to define the materials of each component. In this case, all components are made of the same material: Aluminum LM9. So in order to introduce this material, the material card must be created:

Material card: ALUMINIUM CASTING ALLOY LM9

Mechanical properties:

- ✓ Young's modulus. 71 GPa
- ✓ Yield stress: 190 MPa

- ✓ Ultimate strength: 255 MPa
- ✓ Elongation: 1%
- ✓ Poissons Ratio: 0.32

In order to get the elastic part, we use the data from the mechanic properties and we get three points:

1. Point A (ϵ_y, σ_y)
 - $\sigma_y=190$ MPa, since it is the one in the yield point.
 - $\epsilon_y=\sigma_y/E= 0.27\%$
2. Point B ($\epsilon_{ut}, \sigma_{ut}$)
 - $\epsilon_{ut}=1/2*\text{Elongation}=0.5\%$
 - $\sigma_{ut}=255$ MPa, since it is given in the table of mechanical properties.
3. Point C (ϵ_c, σ_c)
 - $\epsilon_u= 1\%$, since it is the point at the maximum elongation.
 - $\sigma_u= 396.31$ MPa, which can be calculated from the linear A-B line $y=ax+b$, with $a=28260.87$ and $b=113.7$.

	Eng. Strain	Eng. Stress (MPa)
Point A	0.0027	190
Point B	0.005	255
Point C	0.01	396.31

	True Strain	True Stress (MPa)
Point A	0.002696	190.513
Point B	0.004987	256.275
Point C	0.009950	400.273

	True Stress (MPa)	True Plastic Strain
Point A	190.513	0
Point B	256.275	0.00137
Point C	400.273	0.04312

ABAQUS material card:

```
*MATERIAL, NAME= ALUMINIUM
CASTING ALLOY LM9
*ELASTIC
71000, 0.32
*PLASTIC
190.513, 0
256.275, 0.00137
400.273, 0.04312
```

Once the material card of the elements is created it is time to set the boundary conditions of the analysis:

```
441747 *****
441748 ***** HISTORY DATA *****
441749 *****
441750 *STEP, NLGEOM, inc=500
441751 ***** ANALYSIS PROCEDURE *****
441752 *Static
441753 0.001, 1., 1E-10, 0.1
441754 ***** BOUNDARY CONDITIONS *****
441755 *BOUNDARY
441756 fixed,1,6,0.0
441757 ref_n,1,2,0.0
441758 ref_n,3,3,-50
441759 ref_n,4,6,0.0
441760 ***** OUTPUT REQUEST *****
441761 *NODE PRINT, NSET=ref_n,TOTALS=YES
441762 RF,
441763 *OUTPUT, FIELD
441764 *NODE OUTPUT
441765 U,RF
441766 *ELEMENT OUTPUT
```

Picture 26: Input for the static analysis in the .inp file

In the picture above it can be seen how the boundary conditions are defined. It is obvious that the displacement of the rigid is only done in the z-direction, towards the robot.

Static Analysis

Once the .inp file was created, the .odb file will be created thanks to Abaqus Command. However, when running it, some problems will arise. The most important one was that the constraints established in the Catya file will no longer continue in the Hypermesh nor the Abaqus model, so the model will have no constraints between elements at all. Giving this circumstance, it was decided that the Catya model will be directly imported in Abaqus CAE and from there on the static analysis will be done. In order to do so, first the material will have to be settled. In order to make it more real, all components will be made of the same material, Aluminum casting alloy LM9 (the properties and the material card are shown before).

Name: Material-1

Description:

Material Behaviors

Elastic

General Mechanical Thermal Electrical/Magnetic Other

Elastic

Type: Isotropic ▼ Suboptions

Use temperature-dependent data

Number of field variables:

Moduli time scale (for viscoelasticity): Long-term

No compression

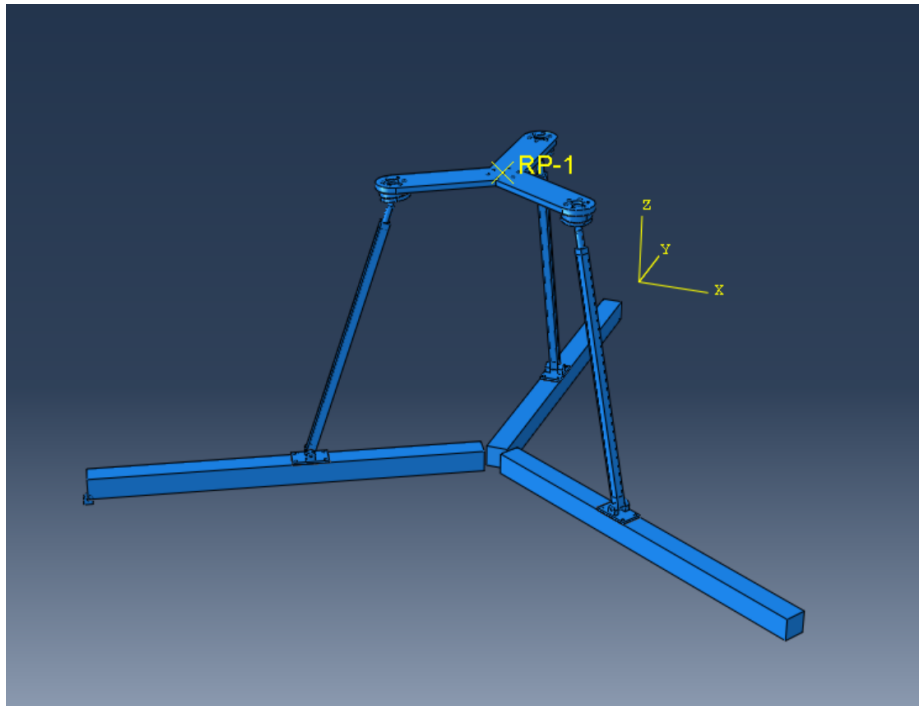
No tension

Data

	Young's Modulus	Poisson's Ratio
1	7100	0.32

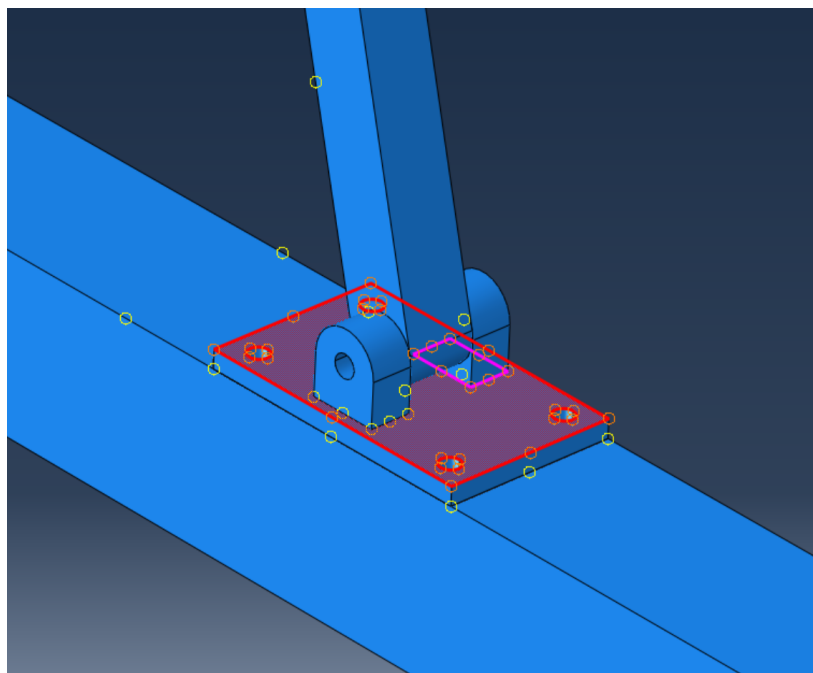
Picture 27: Material set up in Abaqus CAE

After giving each component the material card, the next step is to create the assembly in Abaqus, with all the components.



Picture 28: Assembly in Abaqus

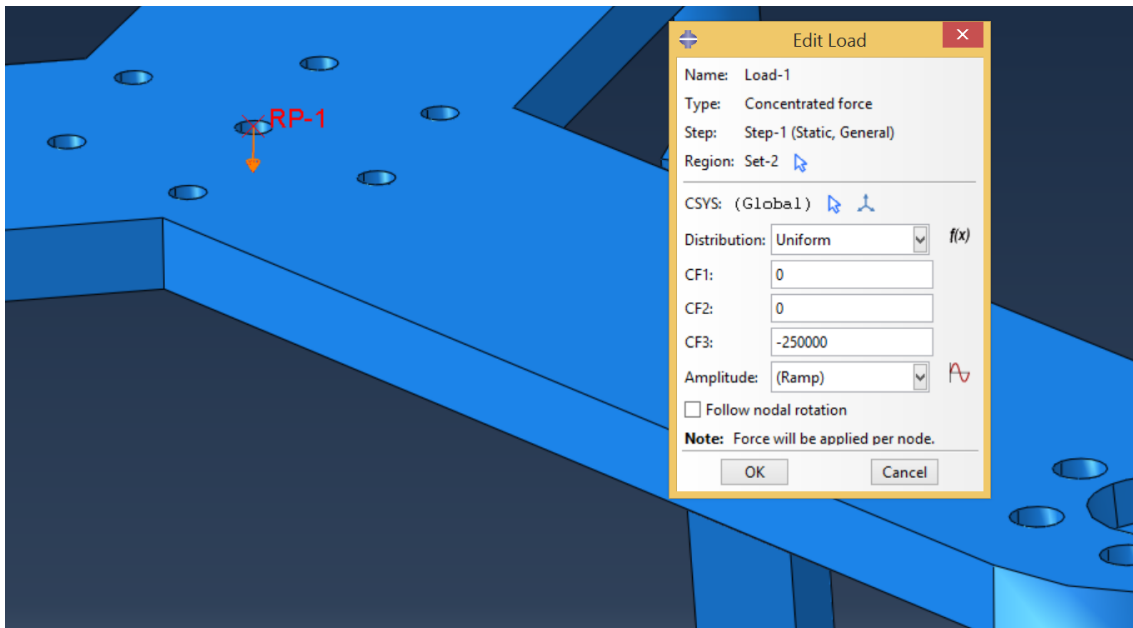
Once the assembly is created it is time to give all the constraints that couldn't be imported from Catya. This will be done with the "tie" command. So every component will have a tie constraint with every component it is in touch with. This is done this way in order to see how the assembly reacts to a particular case in which the structure is in any random way.



Picture 29: Example of a tie constraint in Abaqus

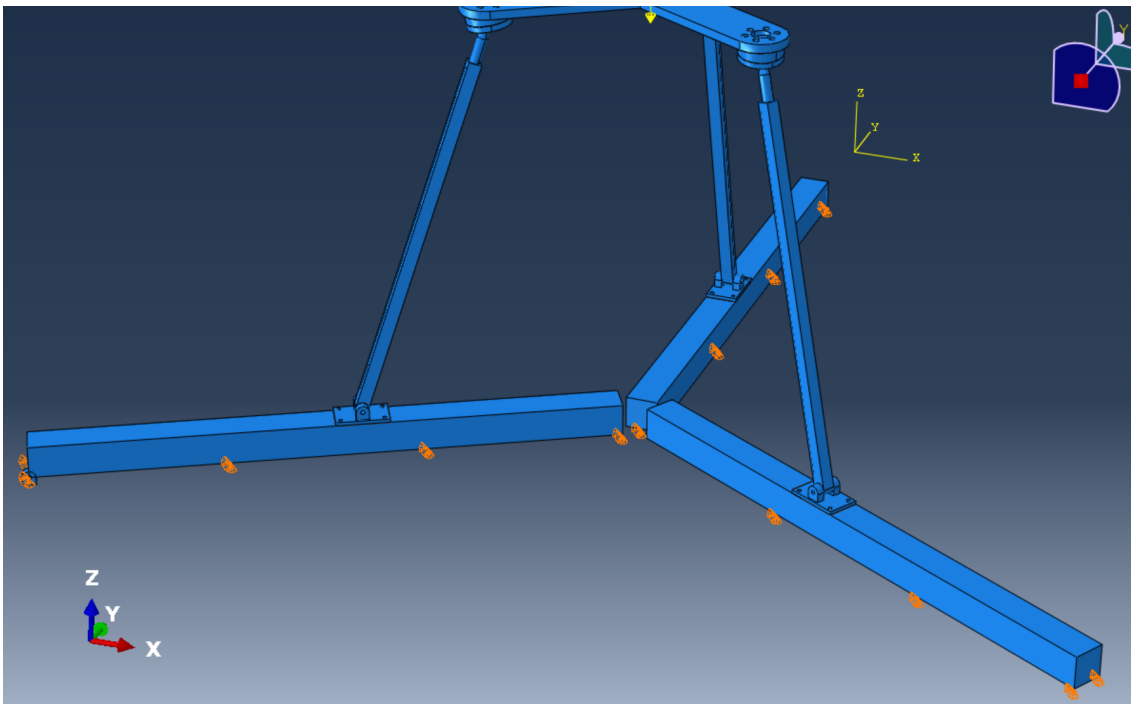
The picture above shows one of the constraints established in the structure. As it can clearly be seen, there is a tie constraint between the different components of the base of the kinetical crank rod crap. The master surface will be the upper surface of the base itself, whereas the slave surface will be the lower surface of the hitch.

After establishing all the constraints, it is time to set the load and the fixed parts of the structure, that is the boundary conditions. In order to set the load a reference point will be chosen and then with the coupling constraint, the reference point will be related to all the upper surface of the platform.



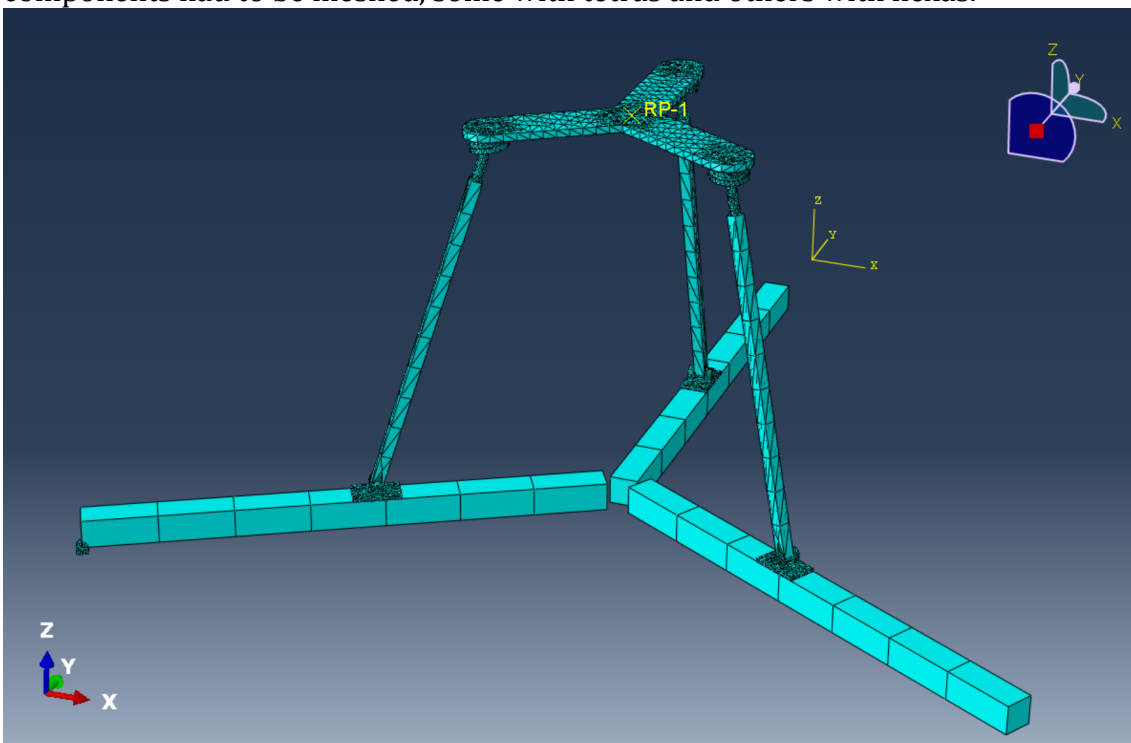
Picture 30: Load set up in Abaqus

The picture above shows where the load is applied in order to simulate the static analysis.



Picture 31: Boundary conditions set up

The final step before running the simulation will be the mesh. In this case all of the components had to be meshed, some with tetras and others with hexas.

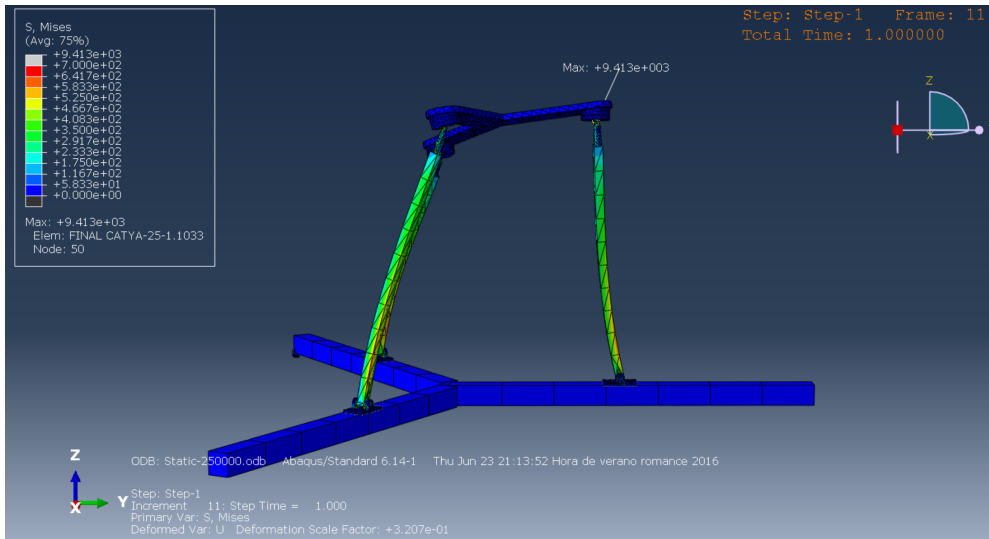


Picture 32: Mesh in Abaqus

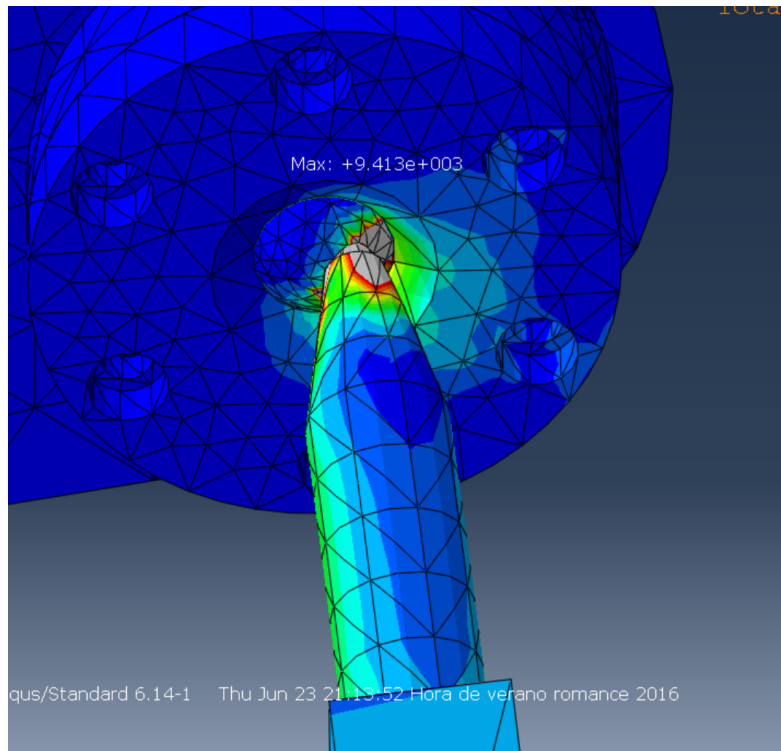
All these steps led to the running of the simulation with the “job” command.

In order to see how the structure reacted to different loads, 250000 N, 500000 N and 1000000 N. The results are shown below:

Results for the 250000N load:

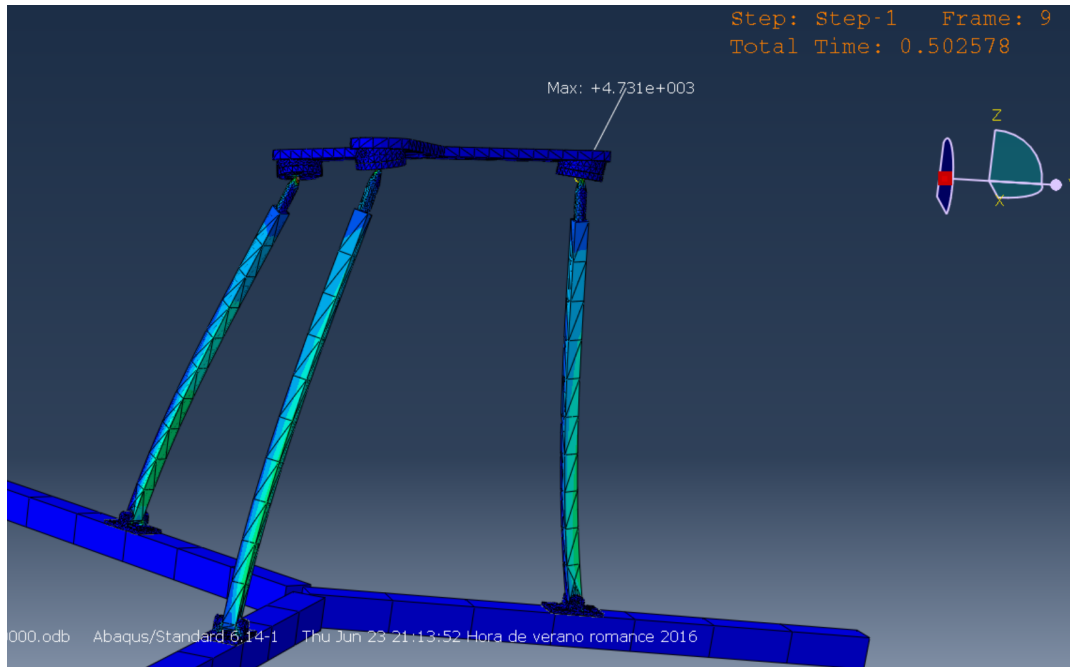


Picture 33: Stress and displacement with a 250000N load



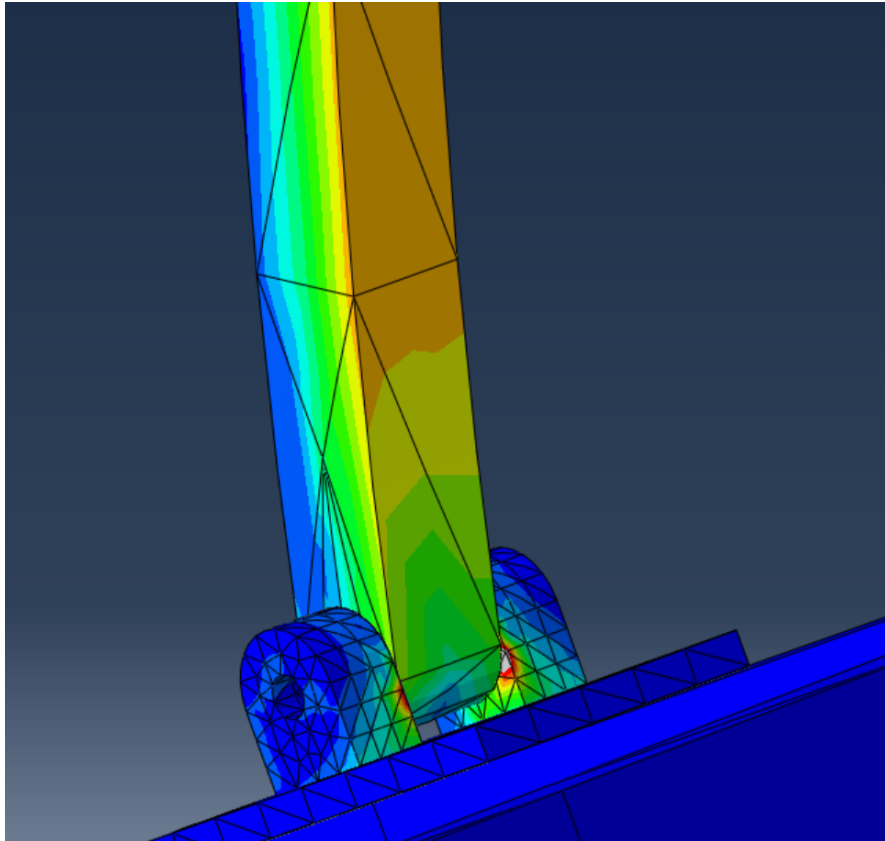
Picture 34: Peak stress

As it can be seen from the picture above the critical point lays in the sphere and with a 250000N load the peak stress is 9413 MPa, far higher than the elastic stress, so there will be elasticity at that point.



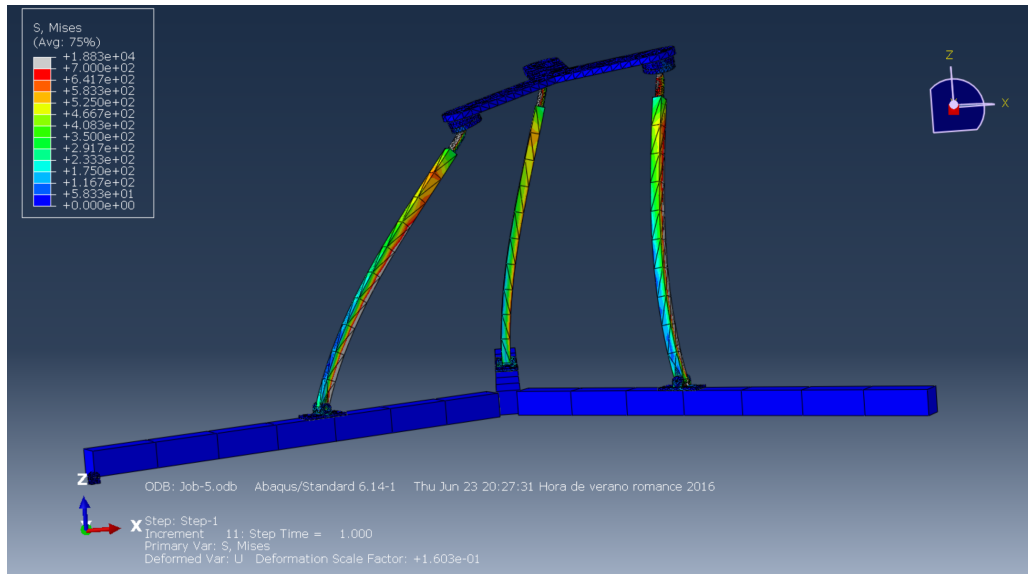
Picture 35: Time when elasticity first happens in the cross section bar

The phase in which elasticity happens is at the middle of the step.

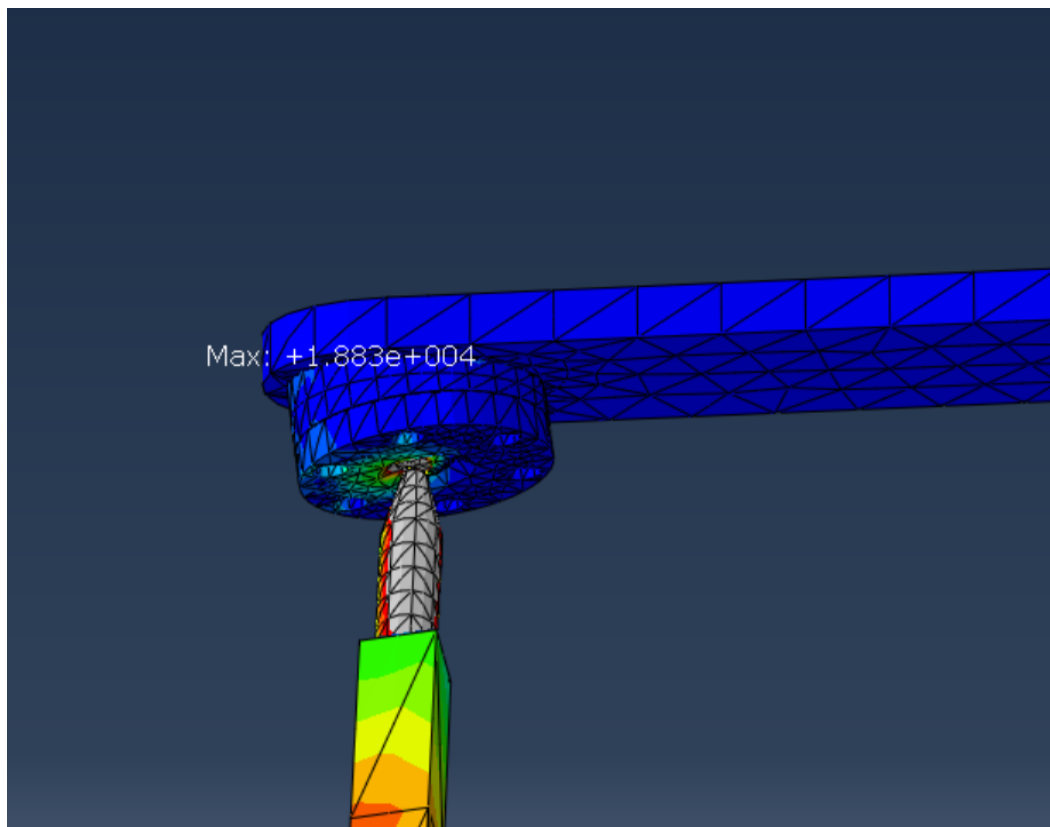


Picture 36: Stress in the rotational DOF

Results for the 500000N load:

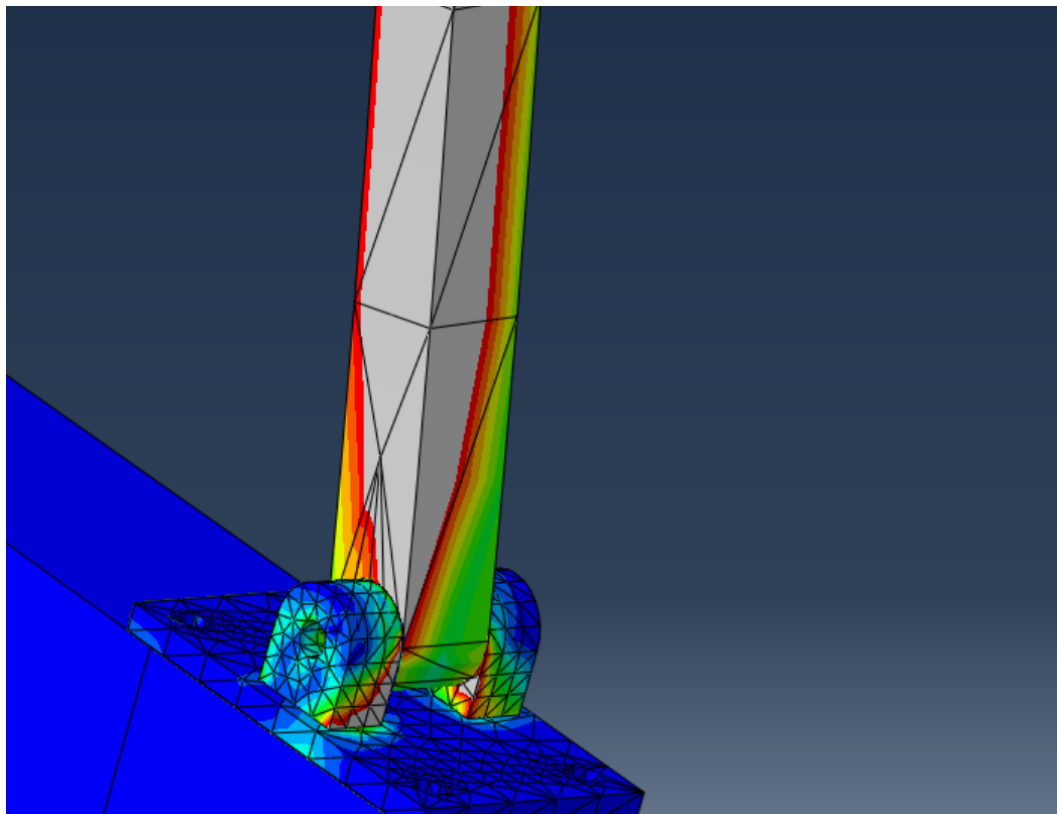


Picture 37: Stress and displacement with a 500000N load



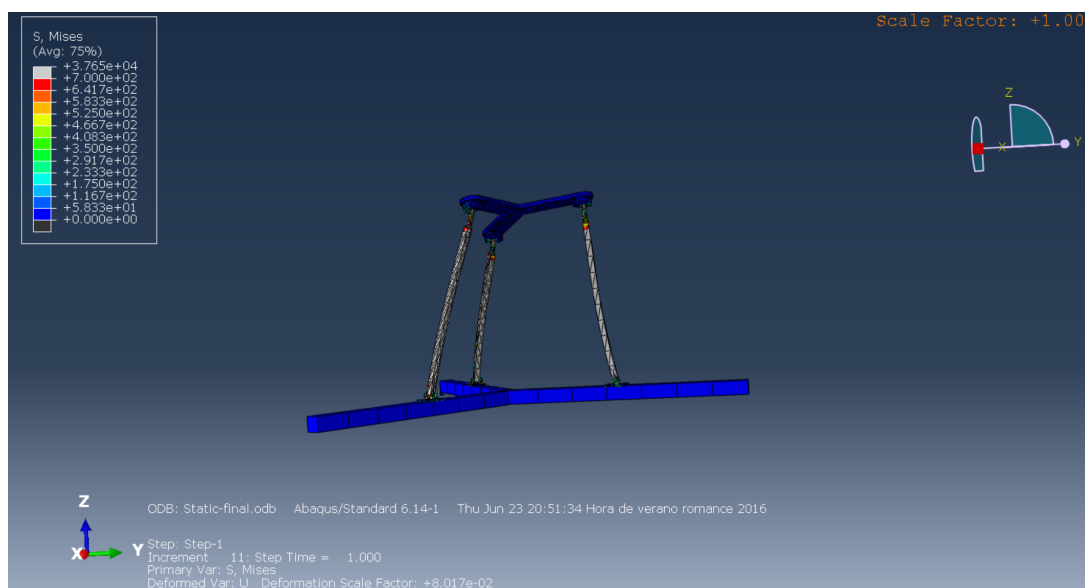
Picture 38: Peak stress

In the picture above, it can be seen that the peak stress for a 500000N load is higher than the one that happened for the 250000N load, since now it is 18830 MPa. Again at that point, elasticity will happen in the sphere and along the pivot.



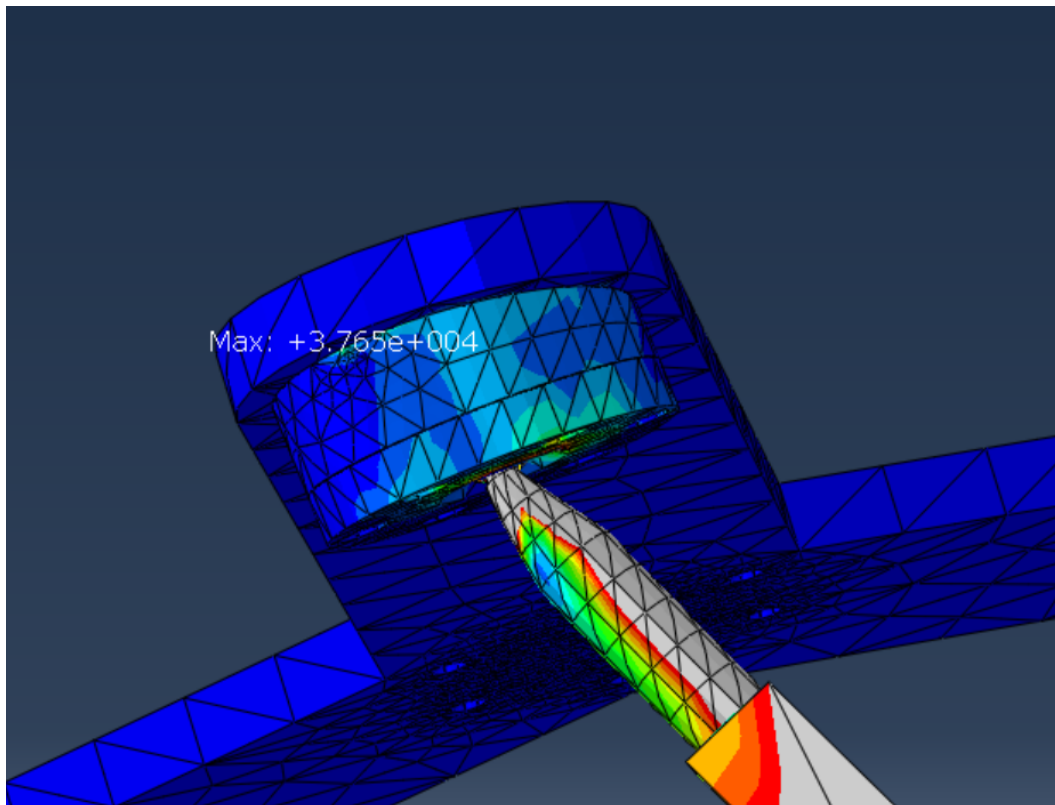
Picture 39: Stress in the rotational DOF

Results for the 1000000N load:



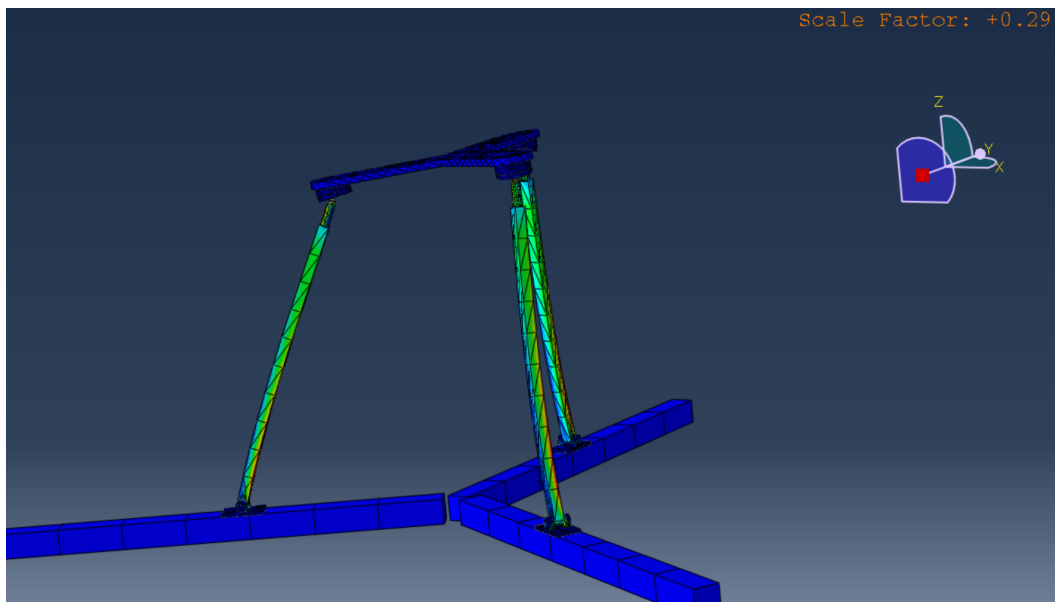
Picture 40: Stress and displacement with a 1000000N load

As the picture above shows, with a 1000000N load, most of the parts that will suffer the load will break (bars, spheres, pivots...).



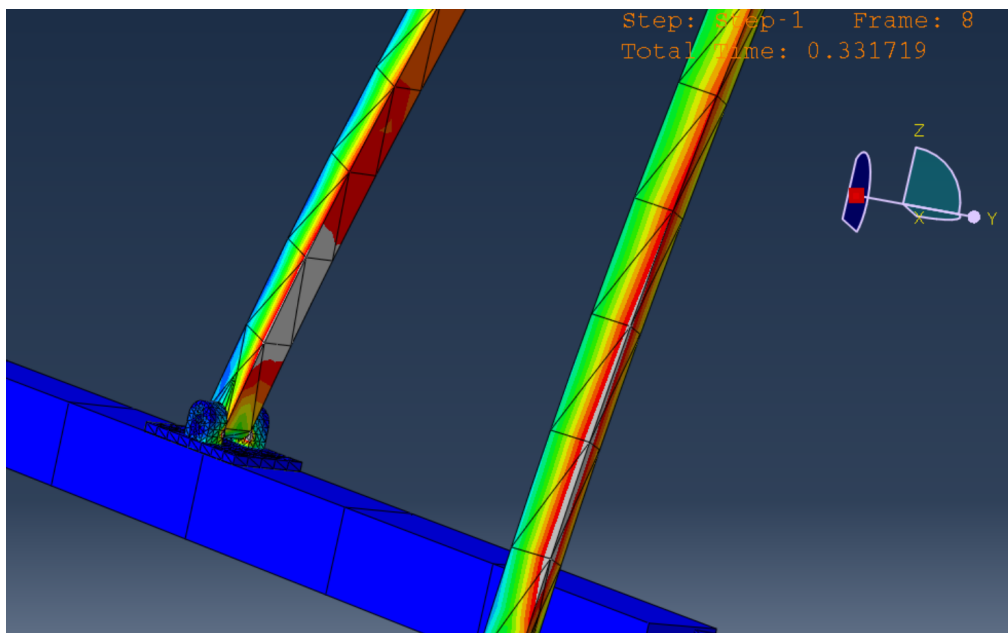
Picture 41: Peak stress

The 1000000N load shows a higher peak stress than the previous loads.



Picture 42: Time when elasticity first happens in the cross section bar

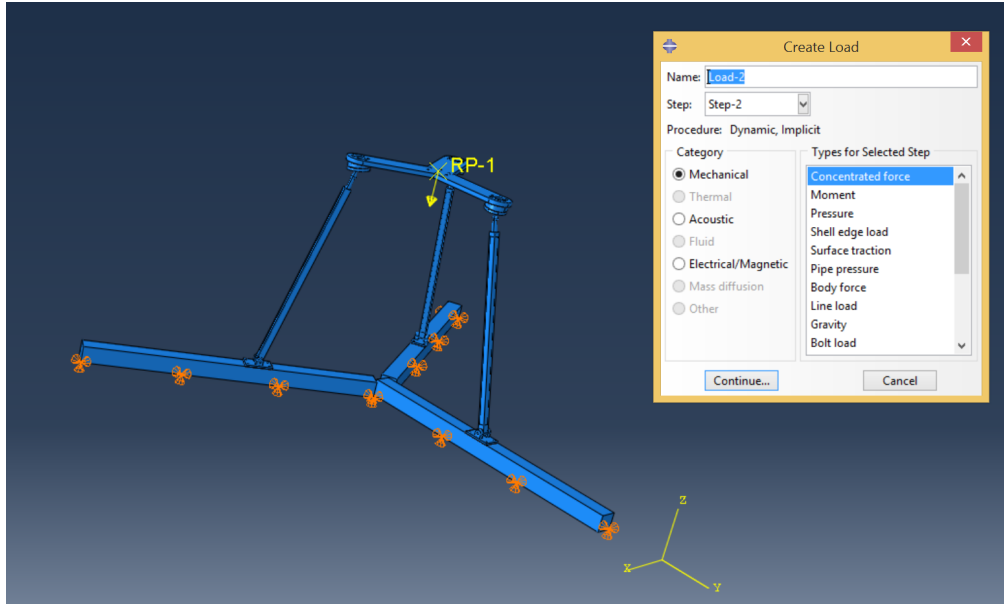
It also shows a sooner elastic appearance; in this case it happens at around the quarter of the step.



Picture 43: Stress in the rotational DOF

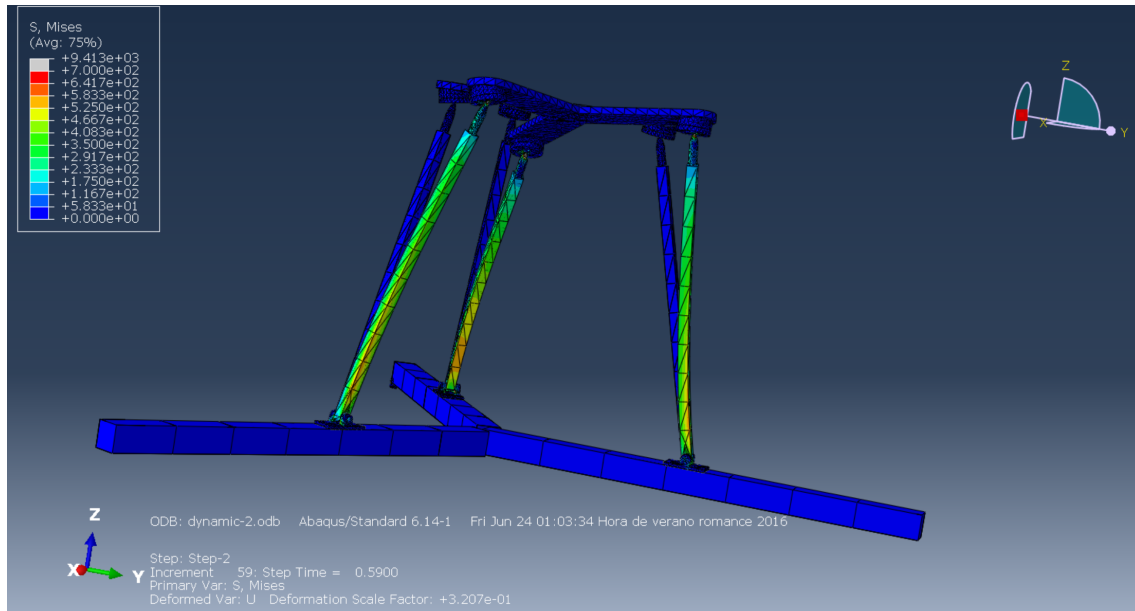
Dynamic Analysis

The next step in the research project is to carry out a dynamic analysis. To do this, a dynamic load will replace the static load. The constraints will be the same as the ones for the static analysis. With all these, the results of the dynamic analysis are:

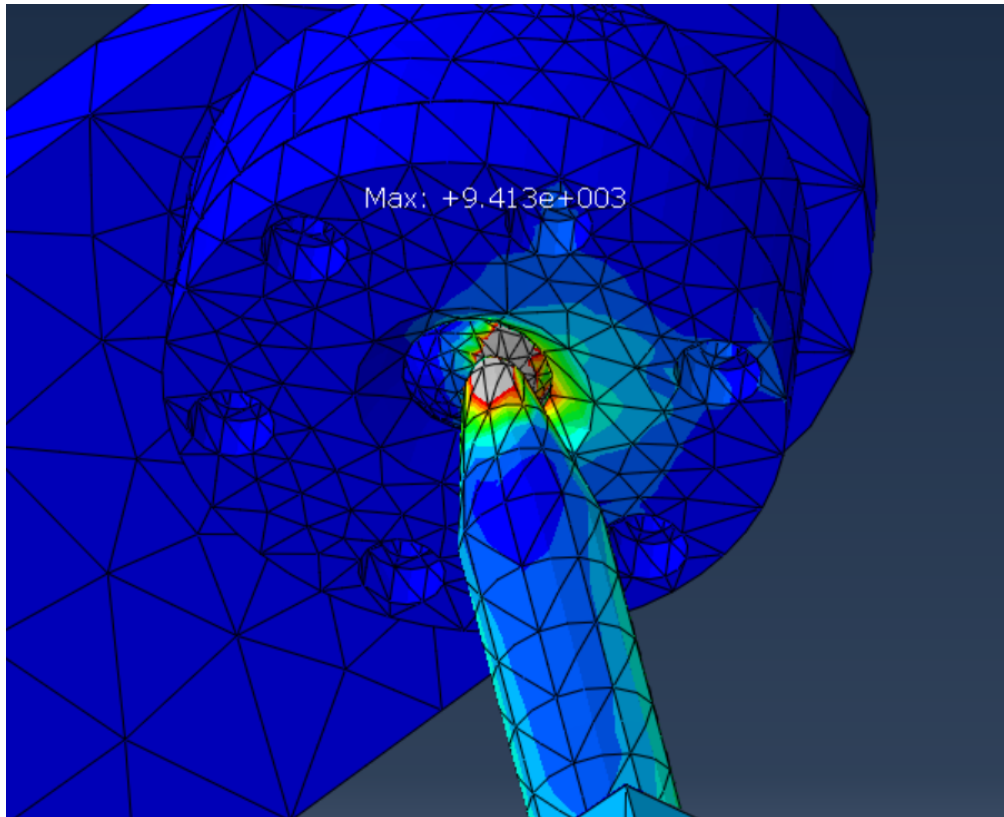


Picture 44: Dynamic load

As it can be seen from the picture above, the difference between the static and dynamic case, is that the step in this case follows a dynamic procedure.

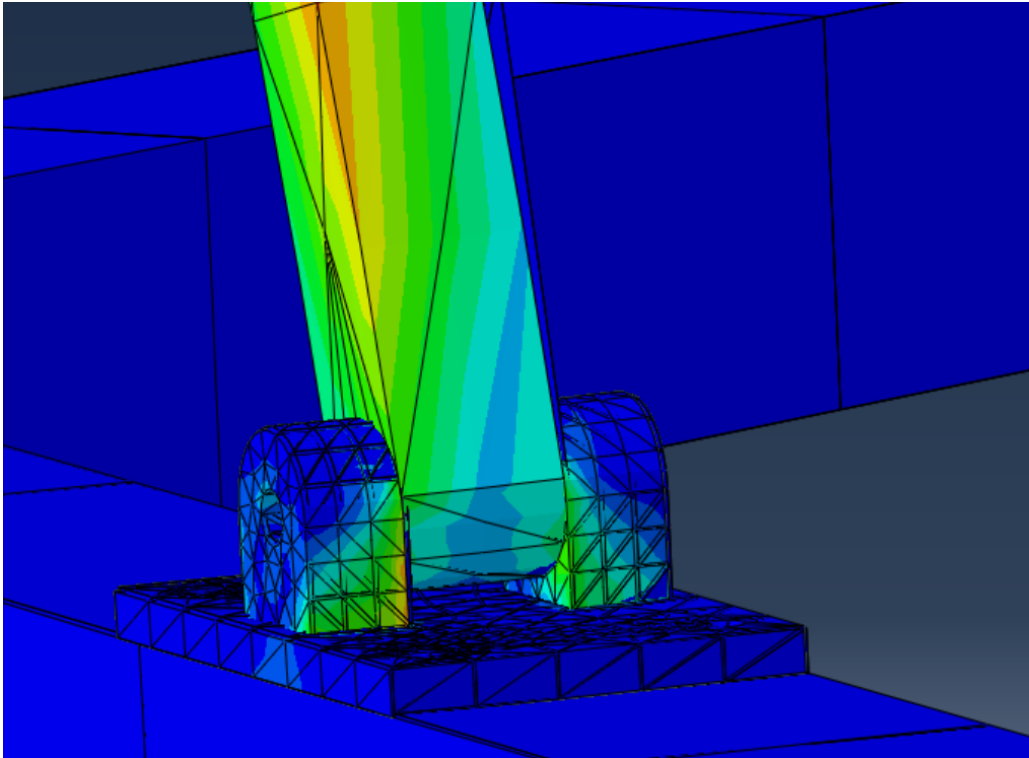


Picture 45: Stress and displacement with a dynamic load



Picture 46: peak stress with the dynamic load

Since it is the same load as the first static analysis, the peak stress is the same for both cases, however the displacement for each case is different being higher for the dynamic load.



Picture 47: Stress in the rotational DOF

Conclusions:

Different conclusions can be extracted from the work done during this research project.

For the static part, the very first and one of the most important conclusions of all the conducted analysis is the importance of a good model and prior planning. Without a previous careful insight on what is wanted to achieve and the way it is pursued is vital, maybe not so much in the first, initial simulations, but on those where an efficient use of resources (time being maybe the most important one) is of the utmost importance. It has been proven by experience that computational cost increases considerably when there has not been a good previous planning of the situation wanted to analyze. Especially in cases when the running can take hours and the final .odb file exceeds a length of 2 gigabytes, no error must occur (in where to apply forces, defining boundary conditions...)

The second most important one would be that the boundaries and constraints set in Catya have no use in Hypermesh and therefore in Abaqus. This can lead to the .odb file not working properly if not noticing this beforehand, since Abaqus will not consider any restriction that the Catya model has.

Also, that a good, systematic naming of the different files and tests can greatly help the simulation work, making it easier to sort the information and categorize the different results more comfortably.

Another noteworthy conclusion of the analysis conducted is that the simulation of Abaqus stops always when a certain stress level has been reached (well after the plasticity has been surpassed, so this still happens out of our area of interest and thus is not decisive).

Moreover, a conclusion made based on the results of the simulations is that the higher the load is the more peak stress the sphere and the rest of the structure holding the load suffer. This can end in most cases with a break due to fatigue or because elasticity is too high for the material.

Finally, as for the dynamic case, state that the displacement suffered when applying a dynamic step, are much higher than the ones in the static case and it is also of great importance that the dynamic analysis take more computational cost than the static ones.