

Stress analysis of a prosthetic implant with ABAQUS

Evaluation 1 | MNI 2

Kori DITMEYER-MOREAU 30/12/2014



Contents

1	Problem Context	2
1.1	Creation of 3D elements	2
1.2	Model characteristics	2
1.2.1	Material	2
1.2.2	Interface	3
1.2.3	Boundary condition	3
1.2.4	Loading	3
2	Creation of 3D FE models with 3 load cases	5
2.1	Creation of the model on ABAQUS	5
2.1.1	Importing the 3D elements:	5
2.1.2	Material	6
2.1.3	Assembly	7
2.1.4	Loading	7
2.1.5	Boundary conditions	8
2.1.6	Interfacial modelling	9
2.1.7	Meshing	9
2.1.8	Steps	10
2.1.9	Job	11
2.1.10	Results	12
3	Comparison of different properties changes	15
3.1	Comparison of linear and quadratic element types	15
3.1.1	Definition	15
3.1.2	ABAQUS modifications	15
3.1.3	Results	15
3.2	Investigation of material properties effects on stress	18
3.2.1	ABAQUS modifications	18
3.2.2	Results	19
3.3	Investigation of stress patterns for different interface modelling	21
3.3.1	ABAQUS modifications	21
3.3.2	Results	22
4	References	24

1 Problem Context

In this evaluation on ABAQUS, our aim is to model a prosthetic implant for a hip replacement. Indeed, prosthetic implants need to be firmly attached to the rest of the skeleton and thus we must look, with a model, at the deformations that can appear in the implant. In order to do this, we must first create the 3D elements and then enter the characteristics of the elements in ABAQUS.

1.1 Creation of 3D elements

Following the indications of the Figure 1, the different elements have been created in 3D with CATIA. Credit for M. G. LINT for the realization of the 3D elements.

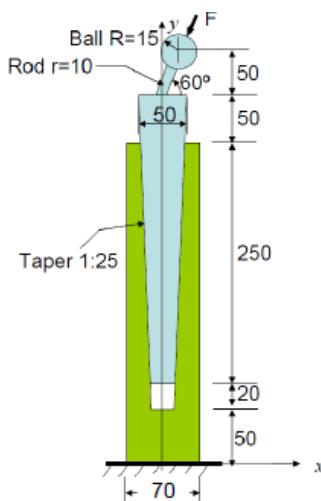


Figure 1: Elements size (Correira, 2015)

1.2 Model characteristics

1.2.1 Material

The implant and the femur must be modelled using specific material properties that will best describe the elements that we want to model. According to (Correira, 2015), we have the following material specifications:

- The implant is composed with cobalt-chromium alloy, an elastic isotropic material ($E = 110$ GPa and $\nu = 0.316$).
- The femur is composed of cortical bone that can be considered as elastic isotropic ($E = 11.5$ GPa and $\nu = 0.31$) or as elastic transversely isotropic with:

$$\begin{aligned} E_x &= E_y = 11.5 \text{ GPa}, E_z = 17 \text{ GPa}; \\ G_{xy} &= 3.6 \text{ GPa}, G_{xz} = G_{yz} = 3.3 \text{ GPa}; \\ \nu_{xy} &= 0.51, \nu_{xz} = \nu_{yz} = 0.31. \end{aligned}$$

1.2.2 Interface

The interface between the implant and the femur vary according to the quality of the patient bone. Hence for patients with a good bone quality, uncemented implants are chosen and for those who have poor bone quality, cemented implants are selected, limiting the risk of fracture during implant insertion. The choice of the interface influences the type of interface modeling. Hence for:

- Cemented interface, we chose a bounded contact type, that is in ABAQUS a tie constraint
- Uncemented interface, we chose a surface to surface contact type. In the Coulomb's friction model, we chose the coefficient of friction equal to $\mu=0.2$ in order the contact not to be frictionless.

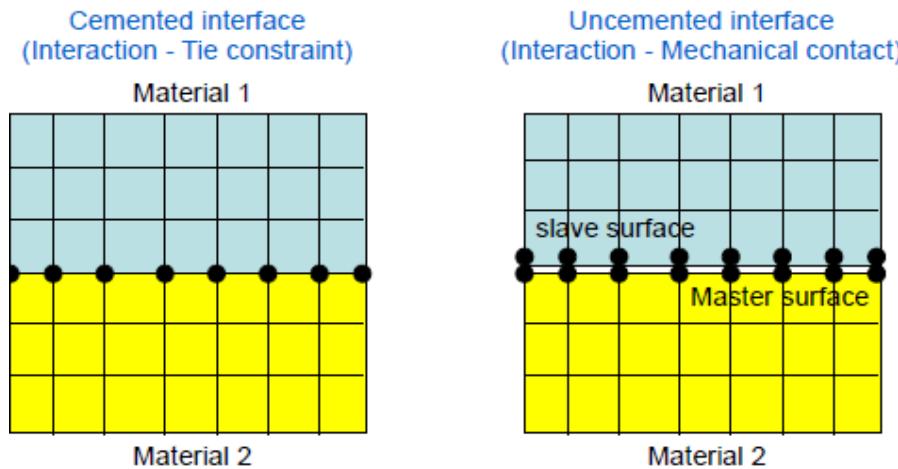


Figure 2: Difference of interfaces between implant and femur

1.2.3 Boundary condition

The lower part of the femur is assumed to be fully constrained, in order to not allow any displacement.

1.2.4 Loading

As the prosthetic implant is used also during walking, the hip contact force (F) vary during the time as well as the angle of orientation. For the rest of the study we will hence simulate the implant model for three loading cases, with three forces and angle chosen in the Figure 3 below. The three forces are shown with the red points.

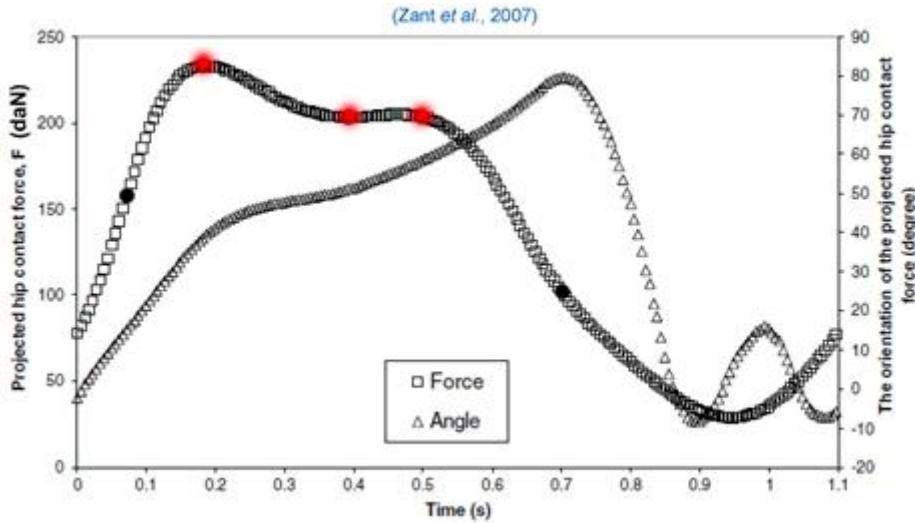


Figure 3: Force and direction variation during time (walking)

The values of the three loading cases chosen have been obtained with PlotDigitizer Software and the values are listed in the **Error! Reference source not found.** along with the values of the projection of the F force along the Z axis and the Y axis.

Loading Name	time(s)	F force (daN)	angle(°)	Fz(daN)	Fy(daN)
F1	0.18	234	39	1812	1476
F2	0.39	204	51	1288	1582
F3	0.50	204	58	1077	1733

Table 1: Values of the three force and angle chosen during time

In total, we will have 8 models to describe completely the possible combinations of solutions, as well as 8 jobs (see Figure 4 below).

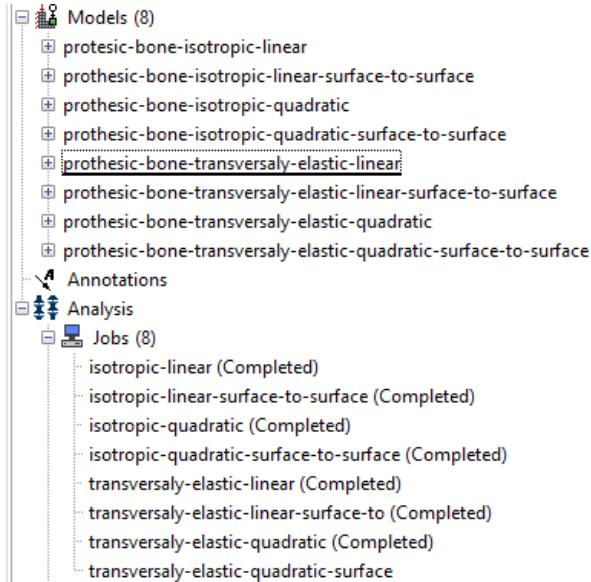


Figure 4: Models with their corresponding jobs

2 Creation of 3D FE models with 3 load cases

For this model, we have considered for the femur as an elastic isotropic material ($E = 11.5 \text{ GPa}$ and $\nu = 0.31$), with a cemented interface.

2.1 Creation of the model on ABAQUS

The creation of the model in ABAQUS is done according to the following steps:

2.1.1 Importing the 3D elements:

First, we must import the 3D elements created with CATIA. Hence, here we will have here two elements to import a femur and a prosthesis. We can see the elements imported in the Figure 5 below.

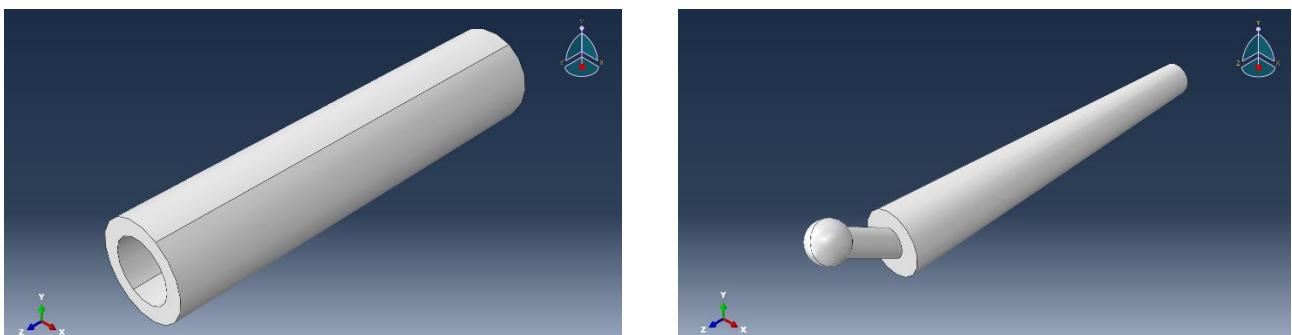


Figure 5: Femur and Prosthesis elements imported in ABAQUS

2.1.1.1 Orientation

When we import the elements, we must orientate them. Here we have chosen a global orientation for the both elements (see Figure 6).

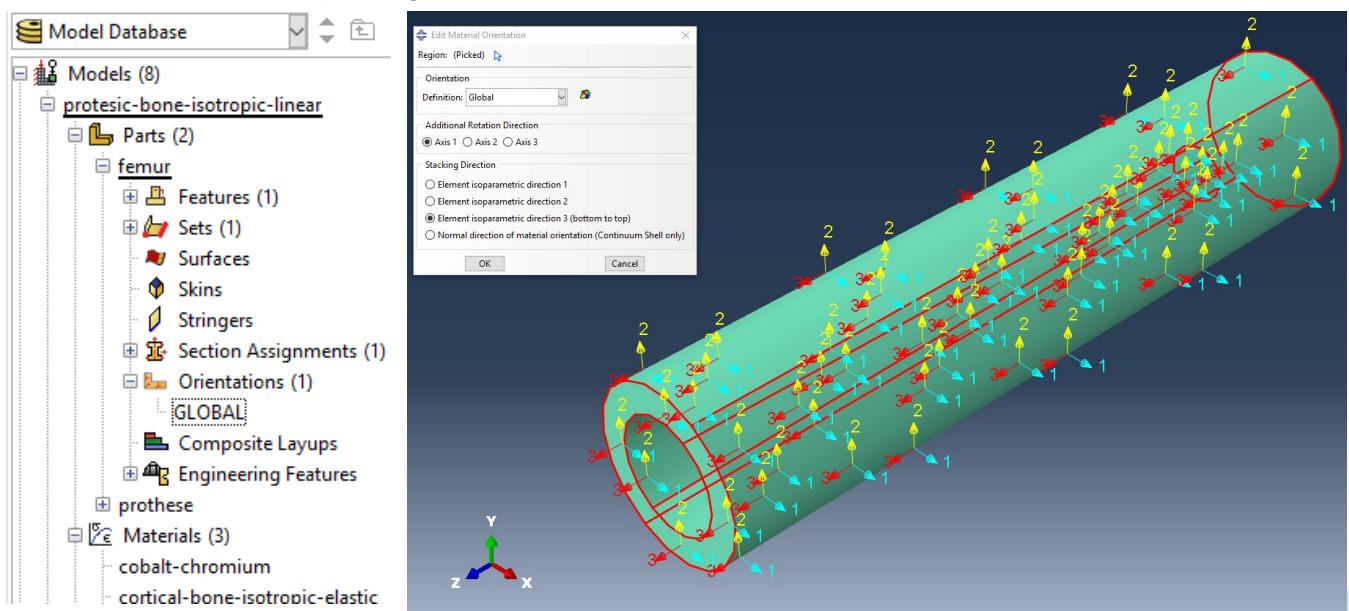


Figure 6: Orientation of parts

2.1.2 Material

For the material of the elements, we must first define the properties of the materials and then assign the properties to the elements.

2.1.2.1 *Definition of material properties*

We define here three materials types for the three different materials that we can have in this simulation:

- Cobalt chromium;
- Cortical bone, transversally isotropic elastic;
- Cortical bone, isotropic elastic.

In the Figure 7 below, we can see the material definition panel as well as the Hook's law coefficient used to describe the elastic behavior of the cobalt-chromium material. We have used the same method in order to define the cortical bone, isotropic elastic material. The values of the Young's Modulus are in MPa.

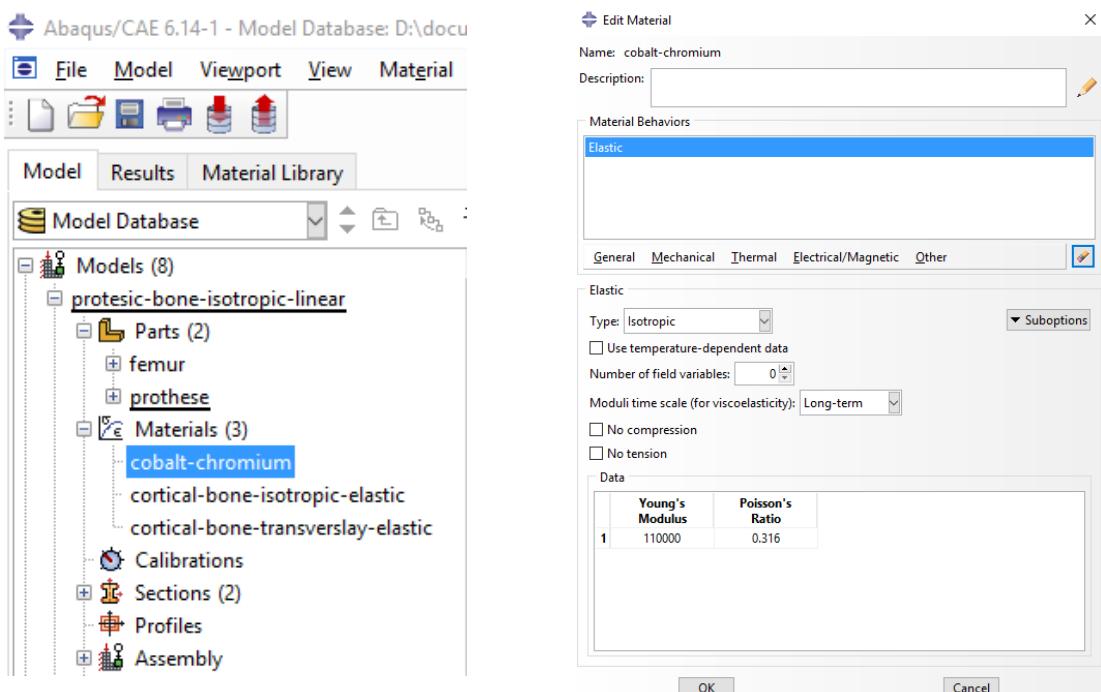


Figure 7: material properties definition

2.1.2.2 Assignment of material to the solid part

Once the materials have been created, we must assign these material to the specific parts. To do this, we must go in the section part, where we first create the section corresponding to the two elements and then we assign to the section the material corresponding with the element (Figure 8).

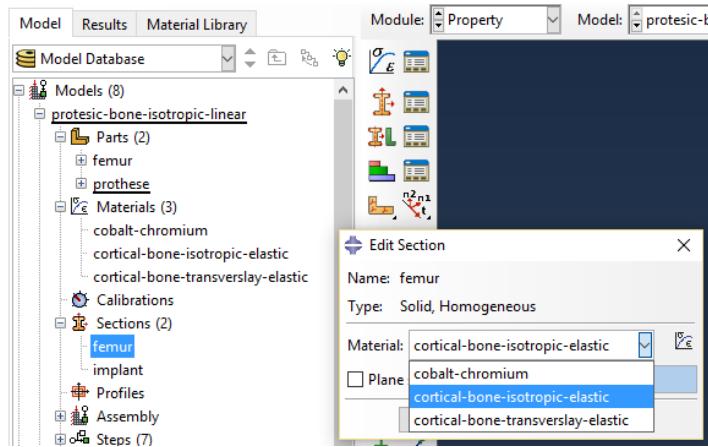


Figure 8: Assignment of material to a section

2.1.3 Assembly

To assemble the different parts into one element, we must create two instances corresponding to the two parts. The result is an element composed of the Femur and implant (Figure 9) .

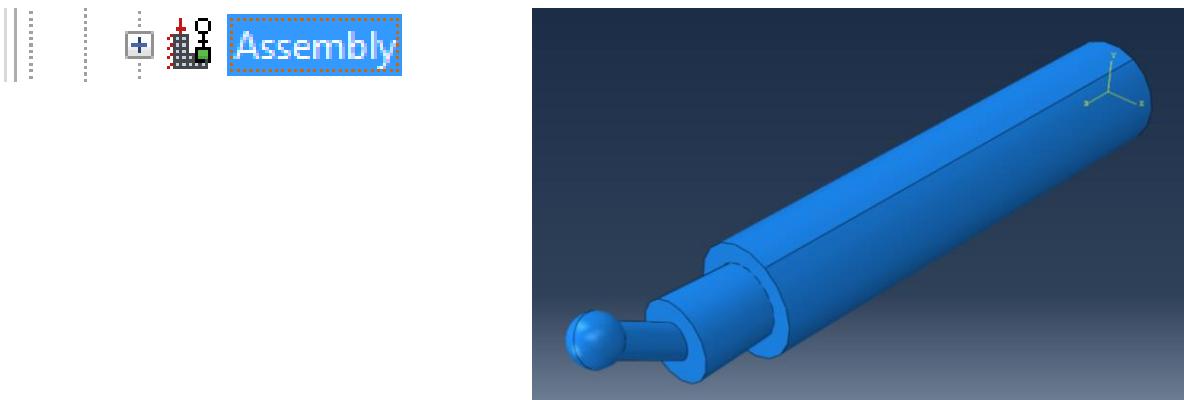


Figure 9: Model of Femur and implant assembled

2.1.4 Loading

In order to model force applied to the implant, we must create a loading force applied on the ball of the implant (Figure 10). This force is defined with the values that are listed in the Table 1page 3.

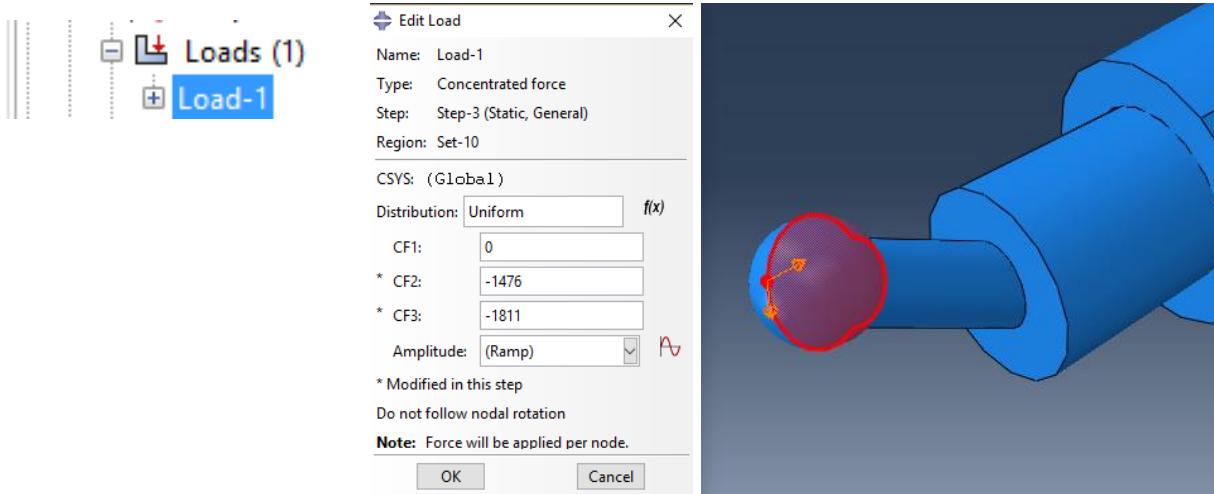


Figure 10: Loading force properties definition

2.1.5 Boundary conditions

In order to avoid any displacement of the femur during the implant interaction with the femur, we must constrain the femur at his bottom part. This is done by putting a ENCASTRE boundary condition at the bottom of the femur (Figure 11).

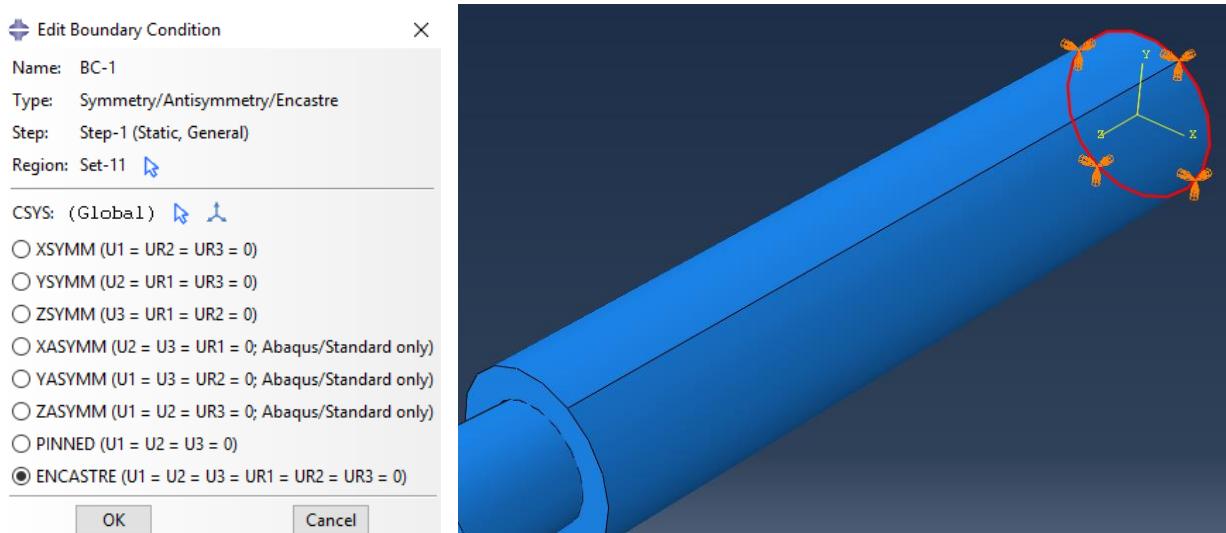


Figure 11: Encastre boundary condition

2.1.6 Interfacial modelling

To model the relation between the femur and the implant, we must in ABAQUS define an Interaction or a Constrain. Here we have chosen for the cemented model to implement a Tie constrain. Indeed, a Tie constrain ties two separate surfaces together so that there will no motion between them. To define the Tie constrain, we must first select the surfaces that will be in contact with each other (Master and slave surface).

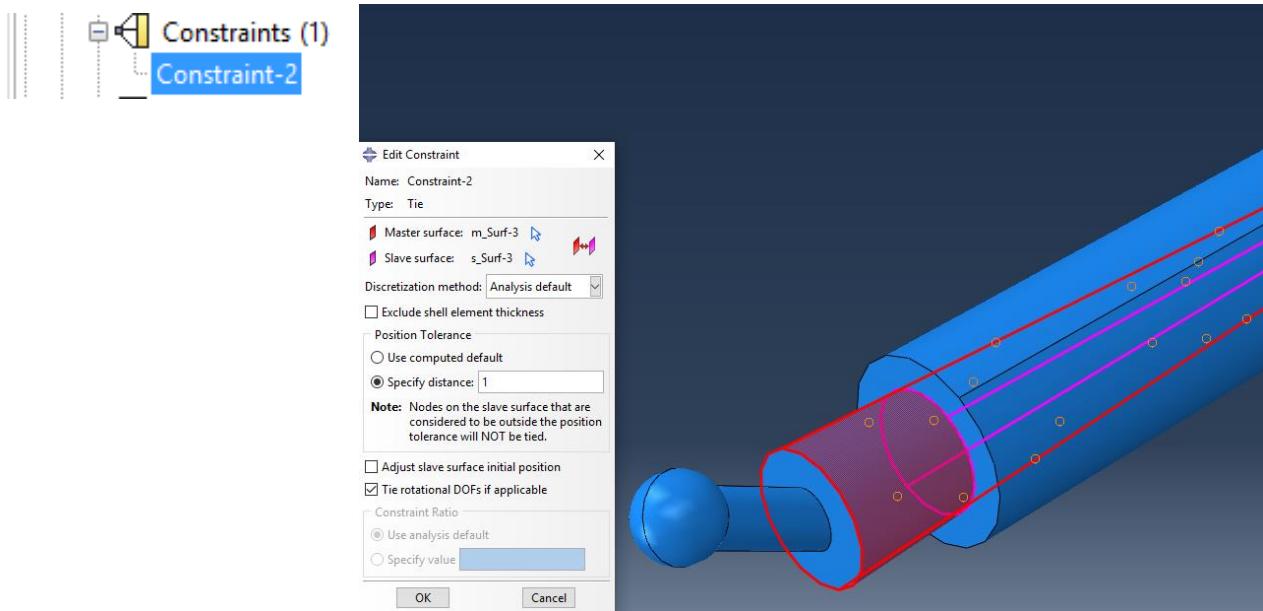


Figure 12: Tie constrain definition

2.1.7 Meshing

The next step is to mesh the femur and the implant, in order that the calculation could be done. In order to do this, the first step is to seed the femur and implant parts. The values used for the seeding are shown in the Figure 13. Once this is done, the second step is to define in the mesh control the element shape type. Here we have used a Tet shape that use exclusively tetrahedral elements (unstructured mesh). The third step is to assign an element type, in other words, to choose the geometric order of the mesh between quadratic and linear. The linear geometric order will result between 2 points this type of element in a linear deformation ($ax+b$) along element (Figure 14).

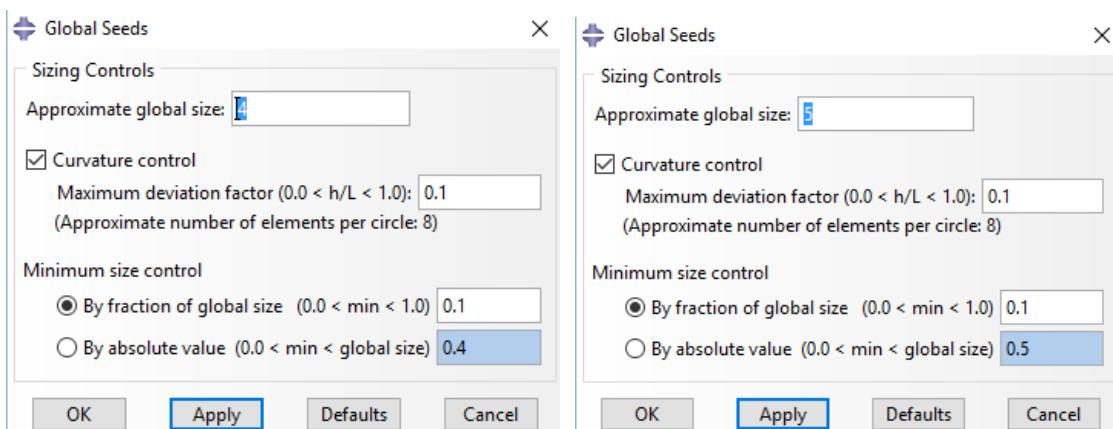


Figure 13: Global seed properties for the implant (left) and the femur (right)

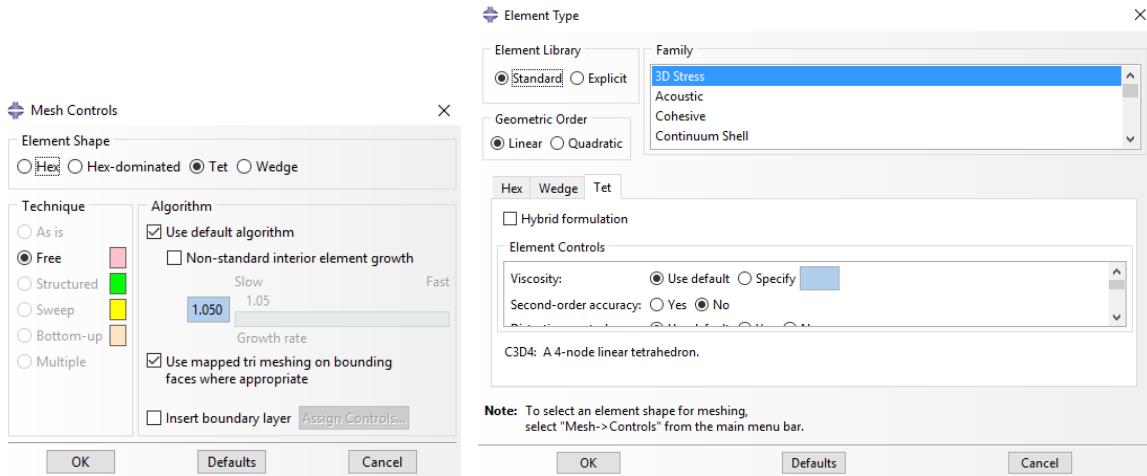


Figure 14: Definition of mesh element type and geometric order of calculus

The last step consists in meshing the two instance parts. The result of the meshing can be seen in the Figure 15 below.

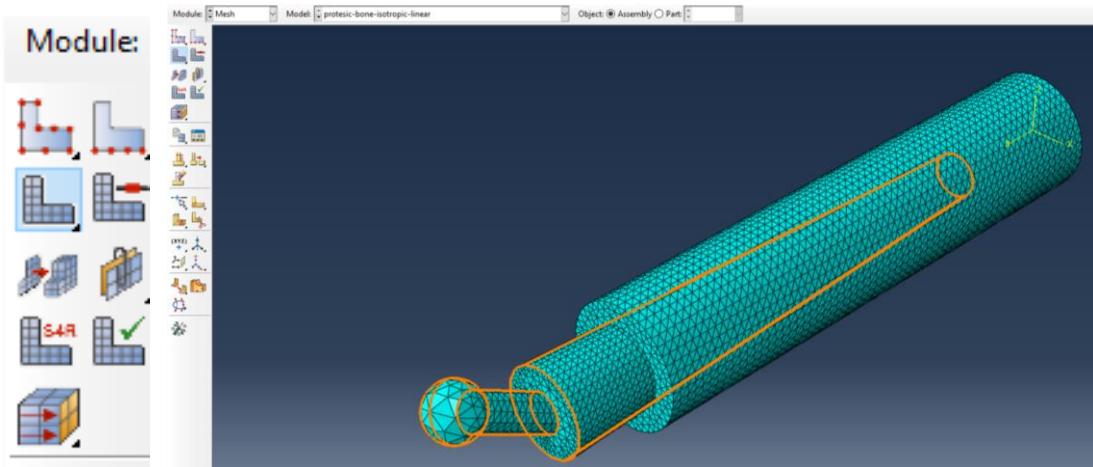


Figure 15: meshing of the two instance parts

2.1.8 Steps

In order to apply several values for the loading force, we must first create various steps, according to the number variation of the loading force. Here, we have chosen to make the loading force variate 6 times and thus we have created 6 steps. However, in the analysis, we will only analyze 3 of these steps. Each step must be define by the time duration and the increment size (see Figure 16 below). Here we have but the automatic mode and we have let the initial increment size to 1, nevertheless we will see in the § 3 that for quadratic elements this was a mistake.

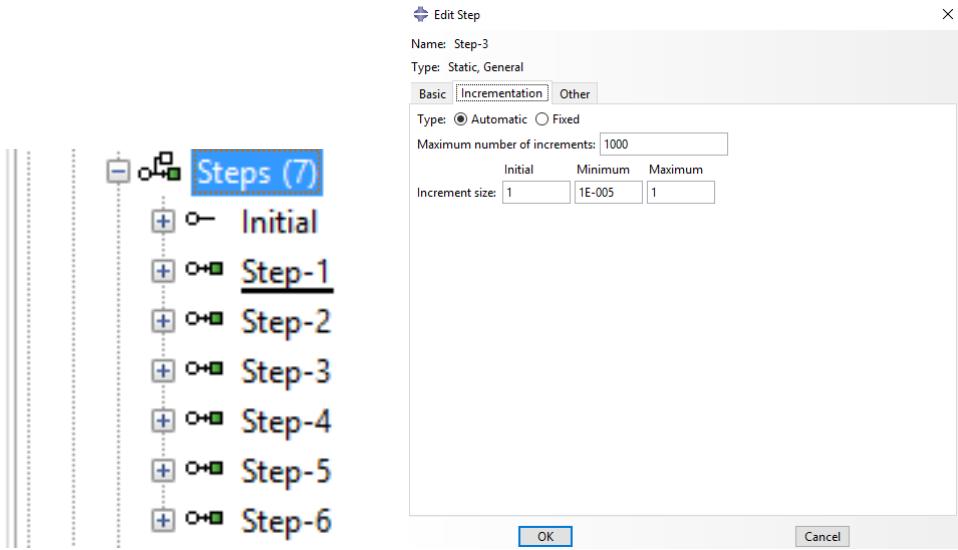


Figure 16: Steps setup

2.1.8.1 Loading Steps

For each steps we have to assign a new value of the loading force. We can see in the Figure 17 below the values entered, according to the Table 1 page 4.

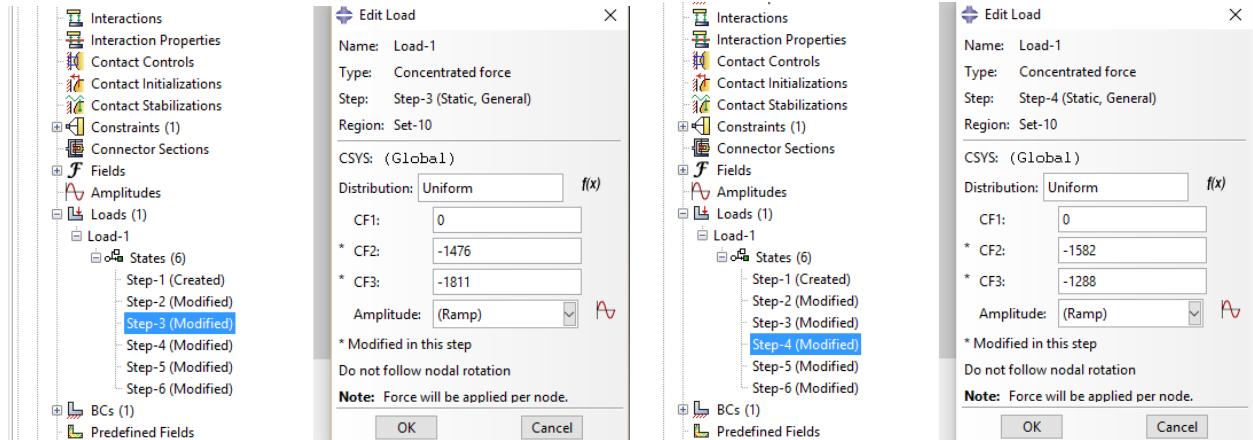


Figure 17: Loading steps

2.1.9 Job

For each different type of analysis that is demanded in the (Correira, 2015) paper, like to analyze the different bones materials, cemented or uncemented, linear or quadratic, we have created a specific job. In total we have run 8 different jobs and for each job 6 different steps. To run the simulation faster, we have used the parallelization mode with 8 processors and 1 GPU acceleration (Figure 18 below).

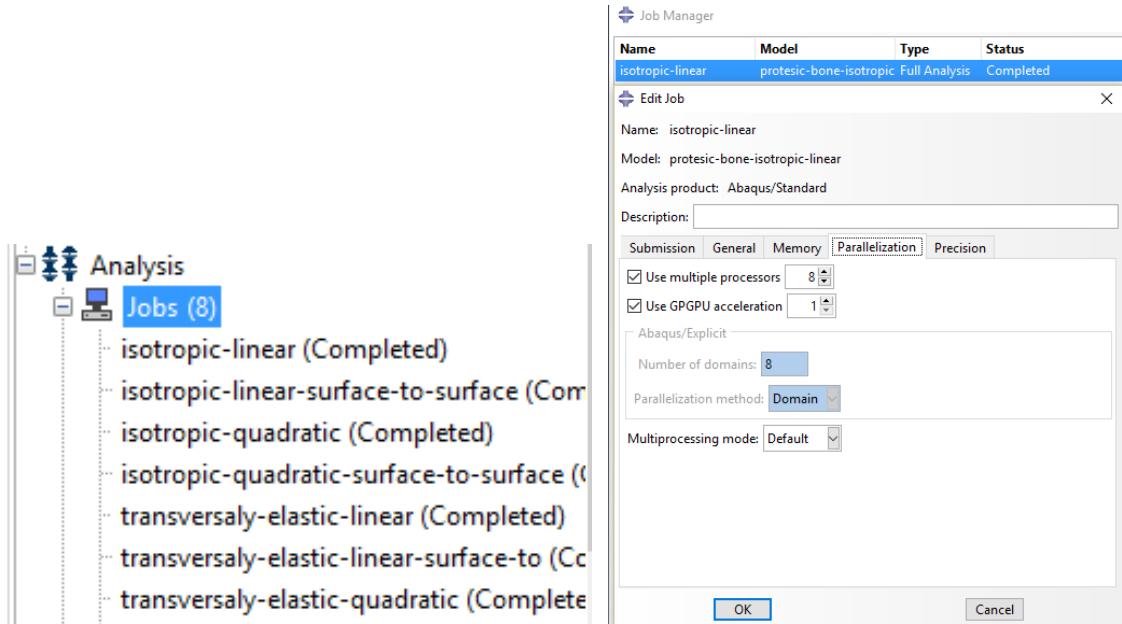


Figure 18: Jobs definition

2.1.10 Results

Once the job is completed, we can go in the result tab. For each job done, we have a different output database, with the 6 steps calculated. Each step has at least two frames with the initial value and the calculated value (Figure 19 below).

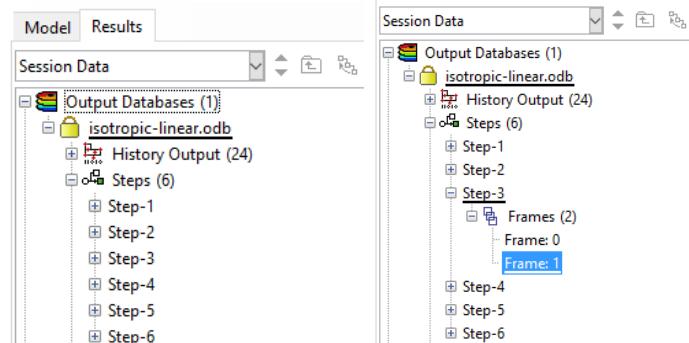


Figure 19: output database

To visualize the result, we have chosen to see the values of the Stress according to the Von Mises. The Von Mises stress is widely used by designers to check whether their design will withstand a given load condition. The deformation of the object will be according to the displacement U (see Figure 20).

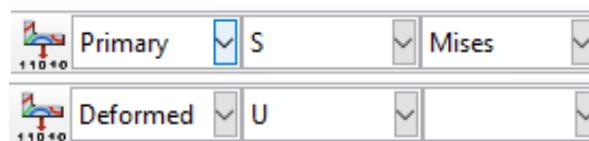


Figure 20: type of visualization

We can see in the figures below (Figure 22, Figure 23, Figure 24) the results of the simulation for the three forces described in the Table 1 for a Cemented interaction, linear elements and a isotropic elastic femur material.

For the first loading case which represent the highest value of the loading force that we have during a walking, the principal Stresses are located in the left bottom side of the Figure 22, which is the left part of the 'Rod'. There is also a Stress at the bottom part of the femur, were we have put a fully constrained boundary condition. We can see that the maximum value of the Von Mises stress (σ_v) is 33 MPa.

The most used failure criterion, is to check whether the Von Mises stress induced in the material exceeds the yield strength (point 2 of the Figure 21) of the material (for ductile material). Indeed, prior to the yield point the material will deform elastically (stress is proportional to strain (Hooke's law) and will return to its original shape when the applied stress is removed. Once the yield point is passed, some fraction of the deformation will be permanent and non-reversible. The failure condition can be simplified as $\sigma_v \geq \sigma_y$.

Here the yield strength of the Cobalt-Chromium alloy is $\sigma_y = 448 \text{ MPa}^1$. So here the prosthesis will not be deformed in a non-reversible way as we can see in the equation below:

$$\sigma_v = 33 \text{ MPa} < \sigma_y = 448 \text{ MPa}$$

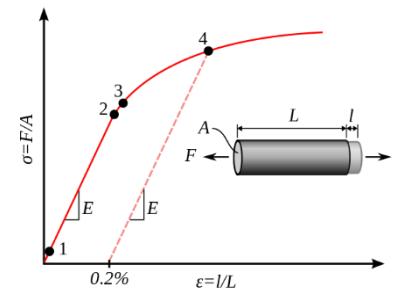


Figure 21: Stress-strain curves showing typical yield behavior

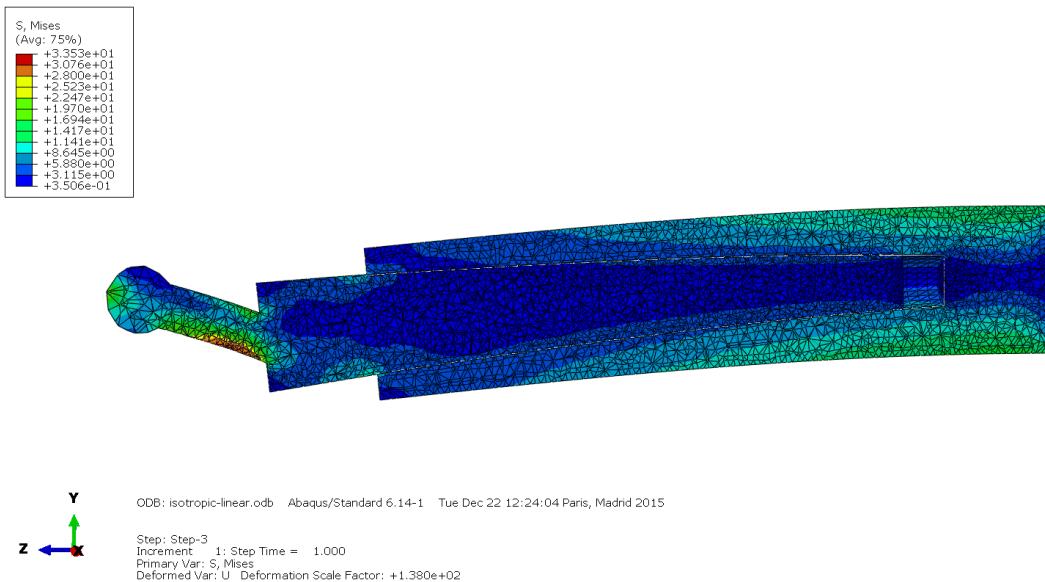


Figure 22: Simulation result for the loading case n°1 ($F_1 = 234 \text{ daN}$ - angle = 39)

¹ Data from <http://www.arcam.com/wp-content/uploads/Arcam-ASTM-F75-Cobalt-Chrome.pdf>

For the loading cases n°2 and n°3, the loading force is the same but inferior than the case n°1. Furthermore, the angle of the force is higher than the case n°1. Hence, we can see that the Stresses are not the same than in the case n°1. Indeed, the stresses appear mostly only on the 'Rod' and with the maximum value in a tiny part of the right side of the 'Rod'. Between the Figure 23and the Figure 24, the highest the angle is the highest the value of the Stress in the right side of the 'Rod' is. Here the maximum stress is $\sigma_v = 86 MPa$ that is below σ_y , but for the maximal angle during the walking of 80° this could may be led to a stress close to σ_y (the simulation was not done for this value).

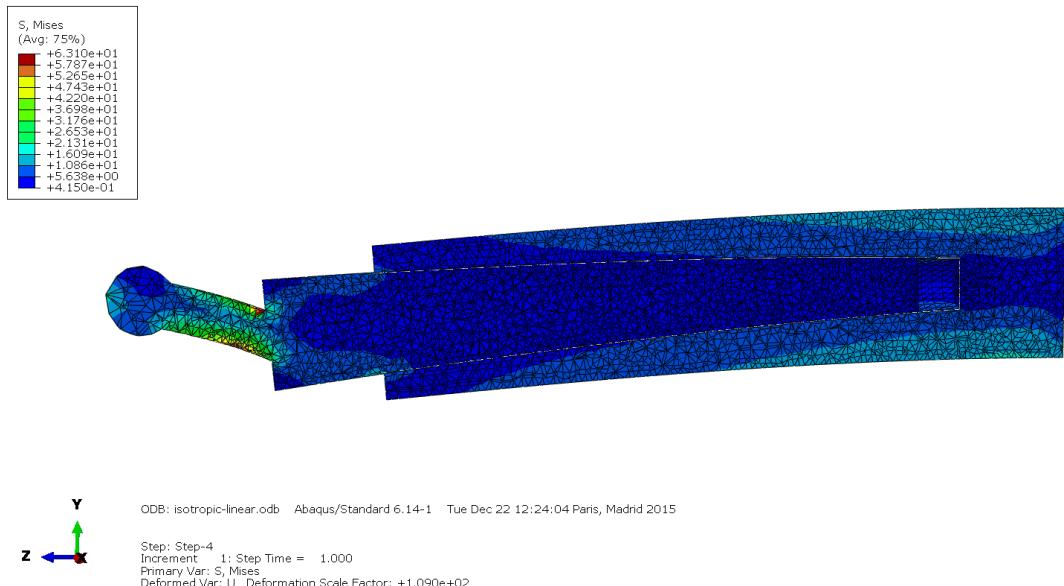


Figure 23: simulation result for the loading case n°2 ($F_2 = 204$ daN - angle = 51)

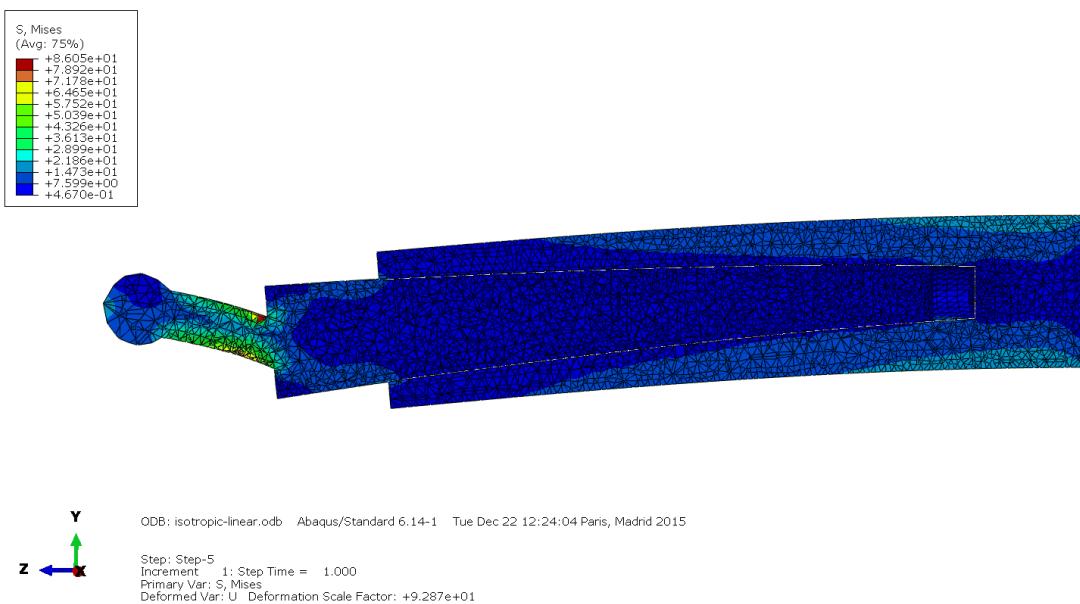


Figure 24: simulation result for the loading case n°3 ($F_3 = 204$ daN - angle = 58)

3 Comparison of different properties changes

3.1 Comparison of linear and quadratic element types

3.1.1 Definition

The quadratic element refers to quadratic shape function and it means that between 2 points the deformation along the edge will follow a quadratic function ($a*x^2+b*x+c$). On the other hand, the linear shape function will result in a linear deformation ($a*x+b$) between two points.

3.1.1.1 benefits and disadvantages

In case of large deformation, the quadratic element has more accurate answer. But these elements should not be used for the faces coming in contact. This will produce unrealistic jump in Contact Pressure values. In that case, it's better to refine the mesh and use linear elements. Furthermore, these quadratic elements we should modify the increment size in the time step panel to 0.1. Indeed, the calculation will not be able to be done in 1 time as the shape function between quadratic elements is not linear. To choose a smaller increment size is hence better.

3.1.2 ABAQUS modifications

In order to change the element type of the femur and the prosthesis, we must go to the interaction Module and in the element type option choose the Quadratic geometric order (see Figure 25 below).

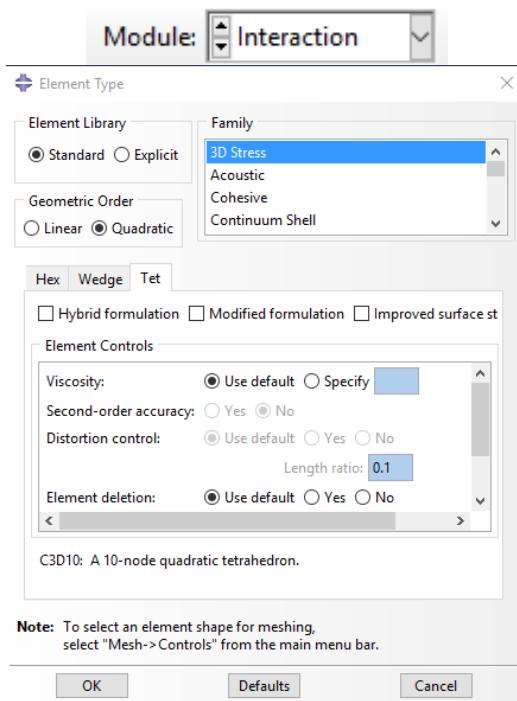


Figure 25: Geometric order

3.1.3 Results

For each steps and each type of interaction propriety (cemented/uncemented) and material (elastic isotropic/elastic transversely isotropic), we have run the calculation with linear and quadratic

elements. The results that are presented below, are not all the results obtain but the most interesting ones.

In the Figure 26 and the Figure 27, we have simulated our model (a cemented interaction, with an Elastic Isotropic femur material) using linear and quadratic elements. We can see that for the first force F1, we have not a huge difference between the linear and the quadratic modelling, however the quadratic model is a lot more accurate than the linear model. Indeed, in order to see all the values in the quadratic model, ABAQUS integrate the maximum values of the Stress in the automatic Stress scale. Thus with an automatic scale we see only the highest values. In the Figure 27, we can see the simulation with the automatic scale (bottom right side) and after the scale was change to exclude the representation of higher values than 33.53 Mpa like the Figure 26.

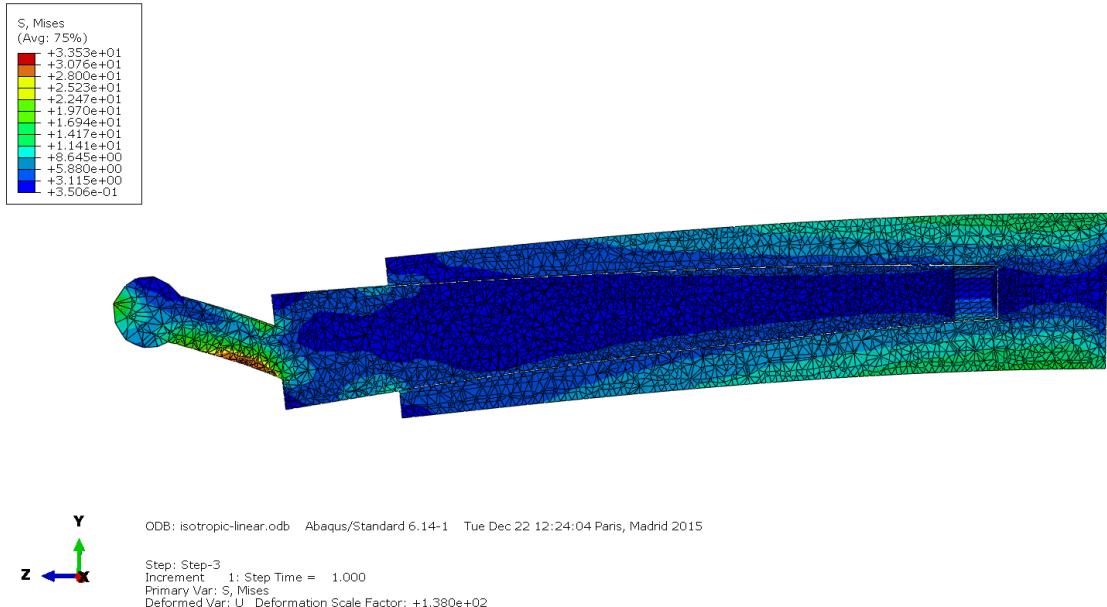


Figure 26: Simulation result : linear (Elastic Isotropic – Cemented - F1)

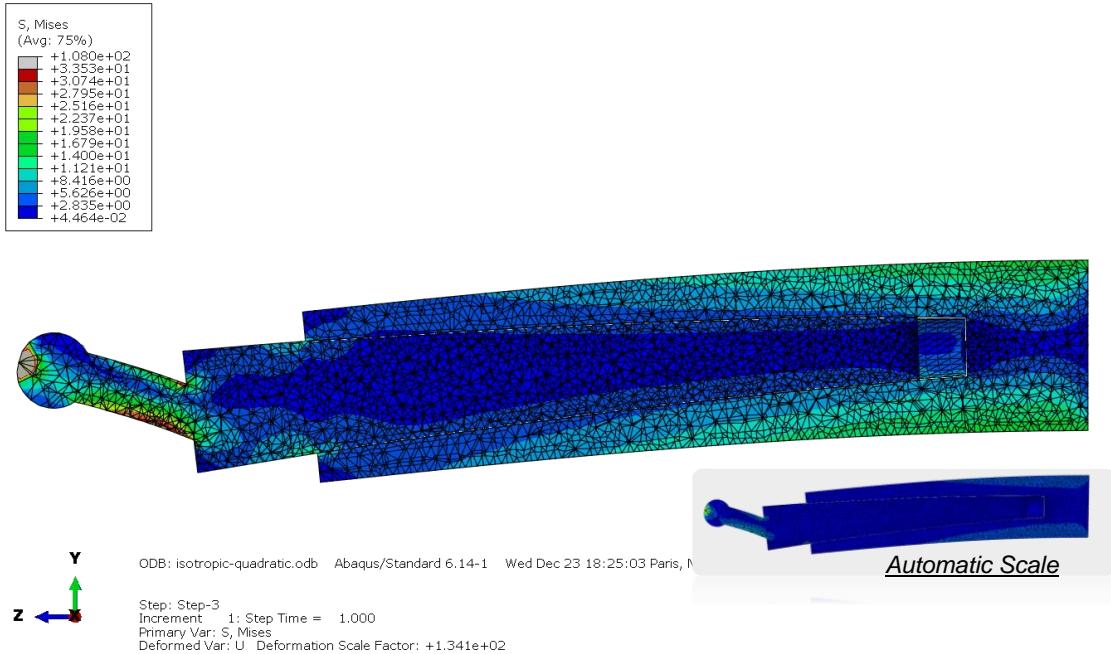


Figure 27: Simulation result : Quadratic (Elastic Isotropic – Cemented - F1)

If we compare the results for a Uncemented interface with a Transversely Elastic Isotropic femur, we can see that there is a strong difference between the linear and quadratic models. The linear and quadratic simulations present stresses along the interface between the femur and the prosthesis. This is due to the surface to surface contact. However, with the linear simulation (Figure 28), we have at the interface stresses on the prosthesis as well as on the femur part, unlike the quadratic simulation where we have stresses only in the prosthesis part (Figure 29). The reason why there is a difference is unknown for me.

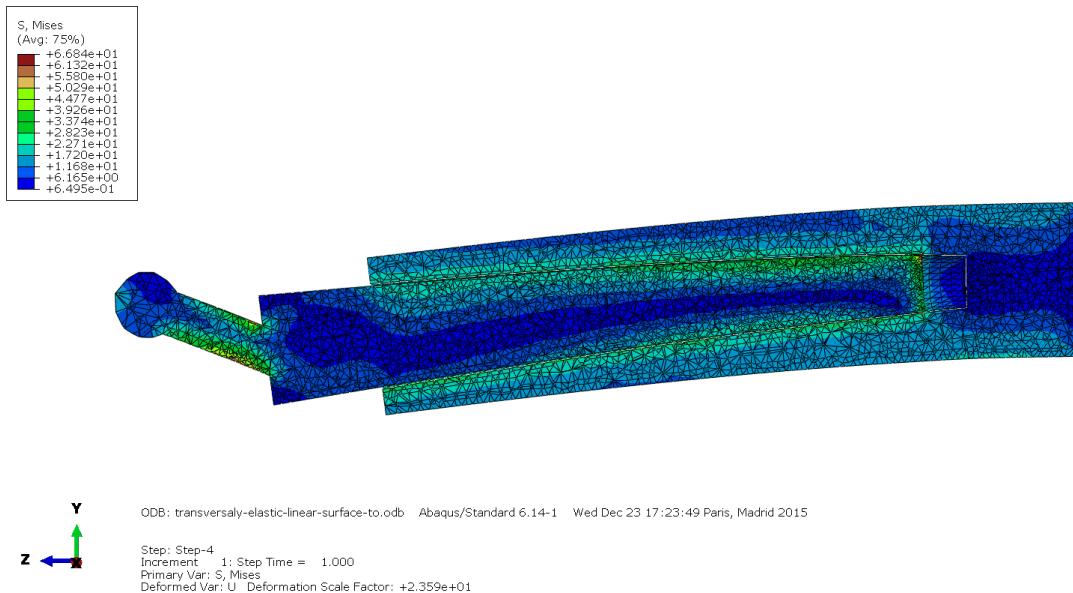


Figure 28: Simulation result : linear (Transversely Elastic Isotropic – Uncemented – F2)

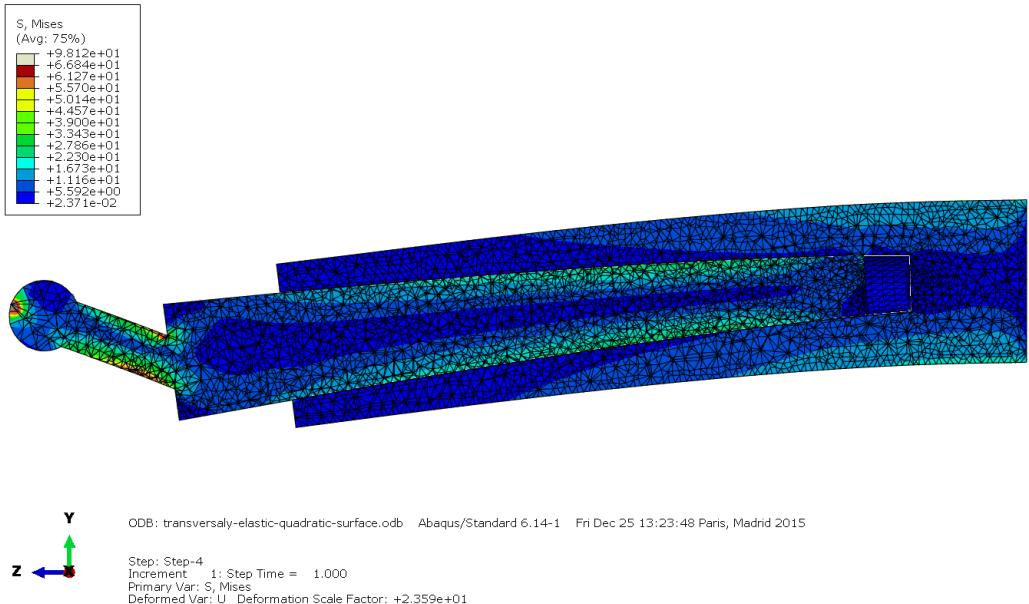


Figure 29: Simulation result : quadratic (Transversely Elastic Isotropic – Uncemented – F2)

3.2 Investigation of material properties effects on stress

3.2.1 ABAQUS modifications

In order to specify the transversely isotropic material for the cortical bone, we have to change in the Elastic material properties from Isotropic to Engineering Constants (see Figure 30 below). This will allow us not two write the entire values of the matrix, but only to write the coefficients needed in order to ABAQUS calculate automatically the matrix.

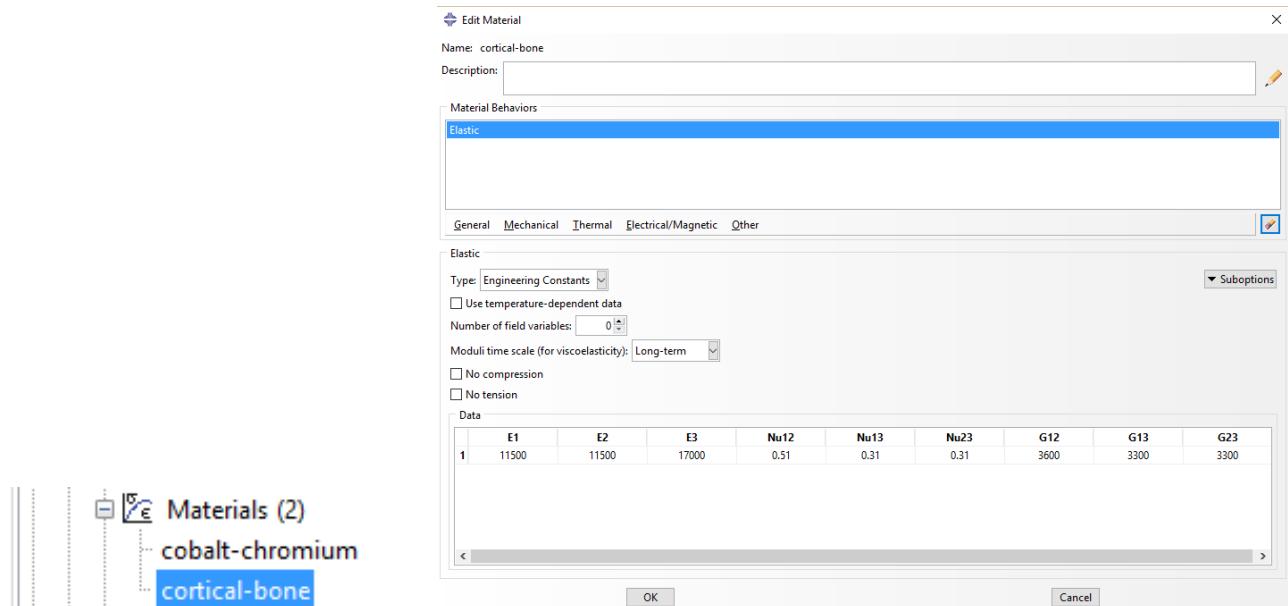


Figure 30: properties of elastic transversely isotropic material

3.2.2 Results

Here, we have to change the scale and the deformation ‘scale factor’ in order to see the changes between an Elastic Isotropic femur and a Transversely Elastic Isotropic femur otherwise the ‘Scale factor’ is chosen automatically by ABQUS and we can’t compare the deformation variations.

Here, we have to compare the simulations of a Uncemented interface with linear elements for two cases, one with an elastic Isotropic femur (Figure 31) and one with a transversely Elastic Isotropic femur (Figure 32). We have chosen a scale factor of 60 to amplify a lot the U displacement in order to see clearly the deformation. With the results below, we can see that the properties of the femur material have a large influence on the Stresses and on the displacement values. Hence, we see with the Transversely Elastic Isotropic material, the femur can absorb the loading force by deforming itself and thus we have not high values of stress. In contrary, with the Elastic Isotropic material, the loading force is not so well absorbed and the stresses values in the femur, at the interaction region with the prosthesis, are very high. Indeed, we have $\sigma_v = 253 \text{ MPa}$ and knowing that the Yield strength of the femur is about $\sigma_y = 188 \text{ MPa}$ ², the femur bone would be broken or deformed irreversibly. Hence, using linear elements with an Elastic Isotropic material is not a good idea.

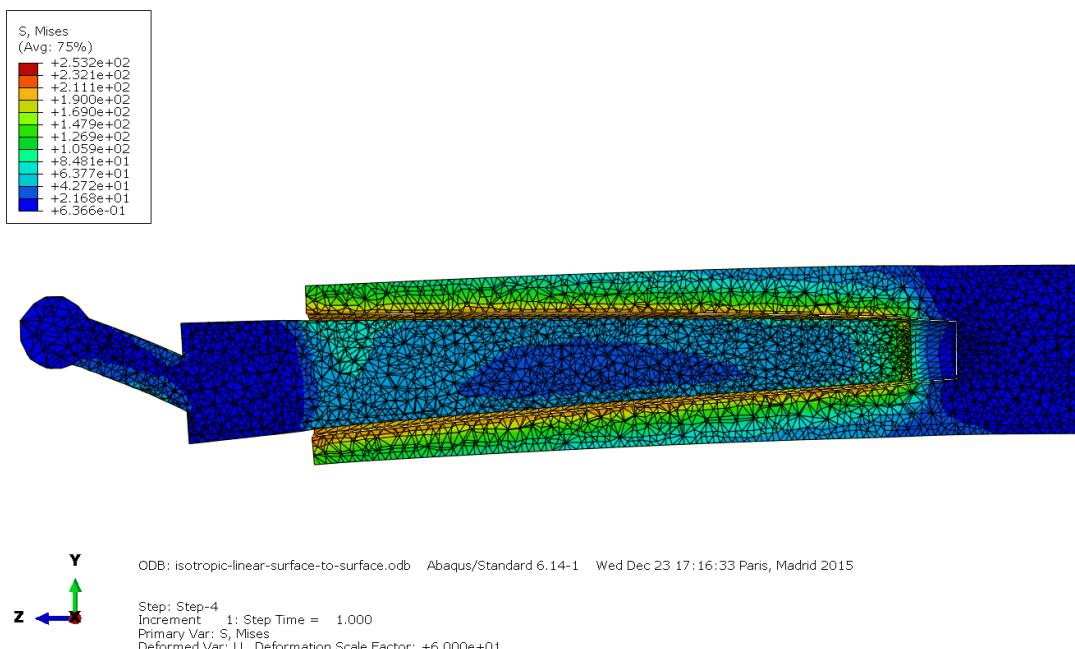


Figure 31: Simulation result : Elastic Isotropic (linear – Uncemented – F2)

² Data from : <http://onlinelibrary.wiley.com/enhanced/doi/10.1002/ar.10145/>

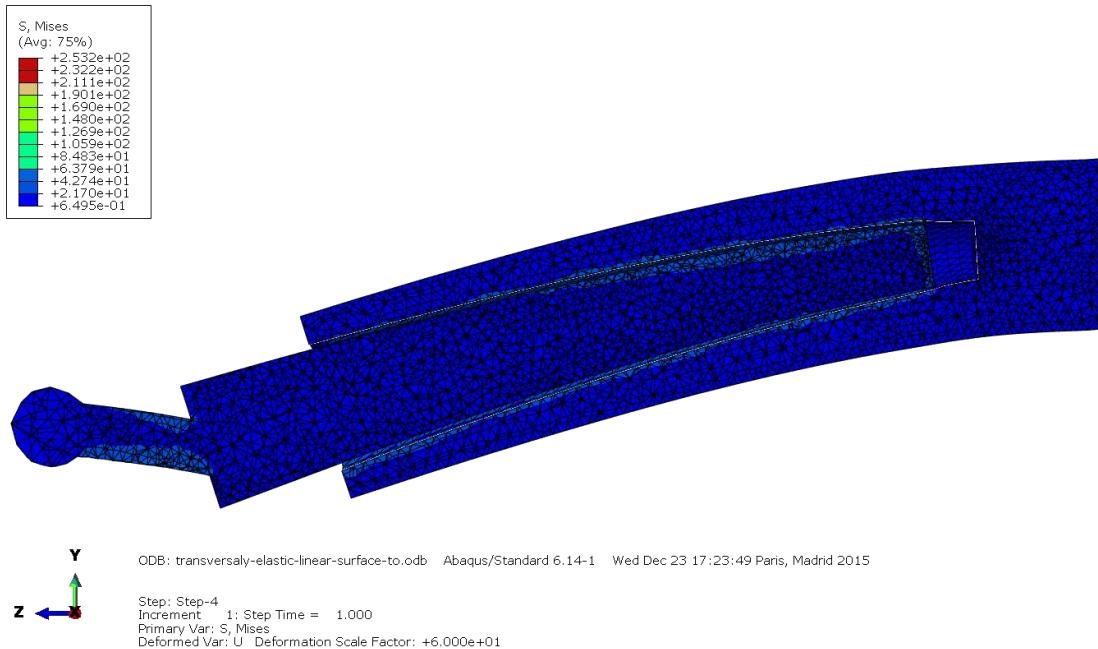


Figure 32: Simulation result : Transversely Elastic Isotropic (linear – Uncemented – F2)

If we change the configuration with quadratic (or even with linear) elements and a cemented interface, we can see the impact of the material type on the deformation between the Figure 33 and the Figure 34. We can see that with this configuration we don't obtain aberrant values like in the Figure 32.

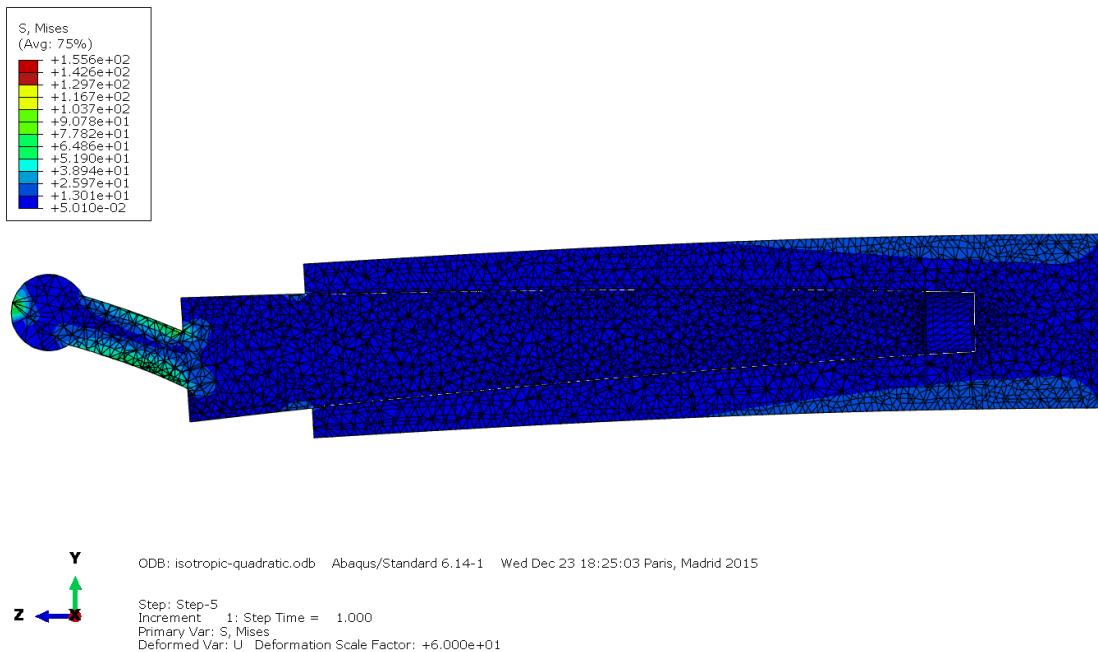


Figure 33: Simulation result : Elastic Isotropic (quadratic – Cemented – F3)

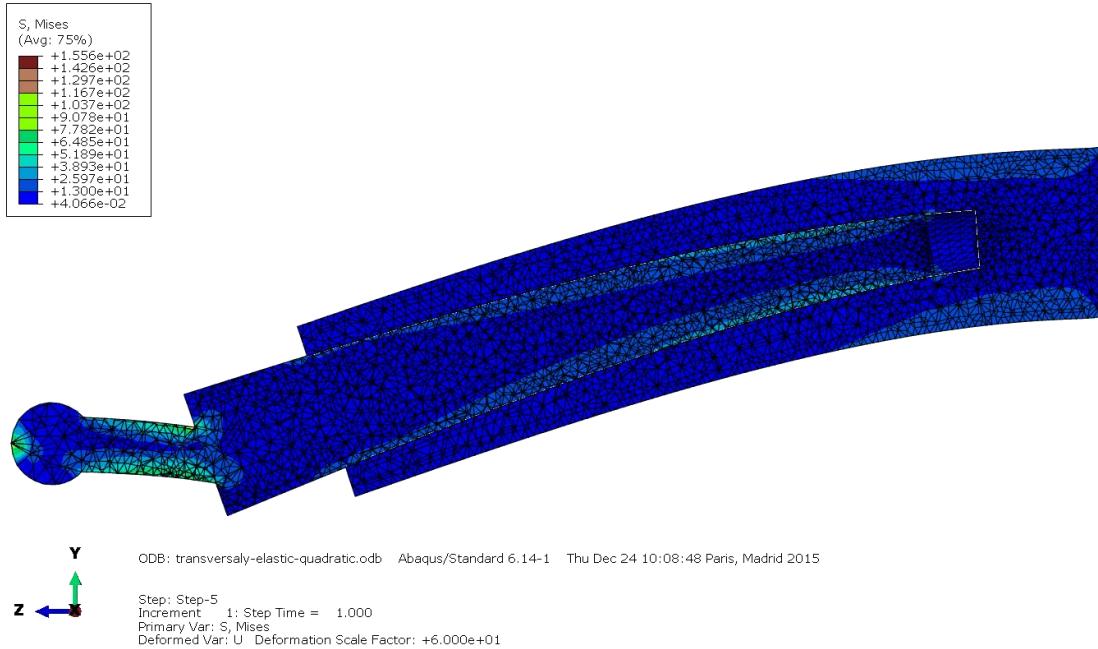


Figure 34: Simulation result : Transversely Elastic Isotropic (quadratic – Cemented – F3)

3.3 Investigation of stress patterns for different interface modelling

3.3.1 ABAQUS modifications

In order to change the type of contact between the femur and the prosthesis, from cemented (Tie) to uncemented (Surface-to-Surface) we have to delete the Tie constrain and create a new Interaction. We have to select the surface that will be in contact with each other and the master and the slave surface (see Figure 35 below).

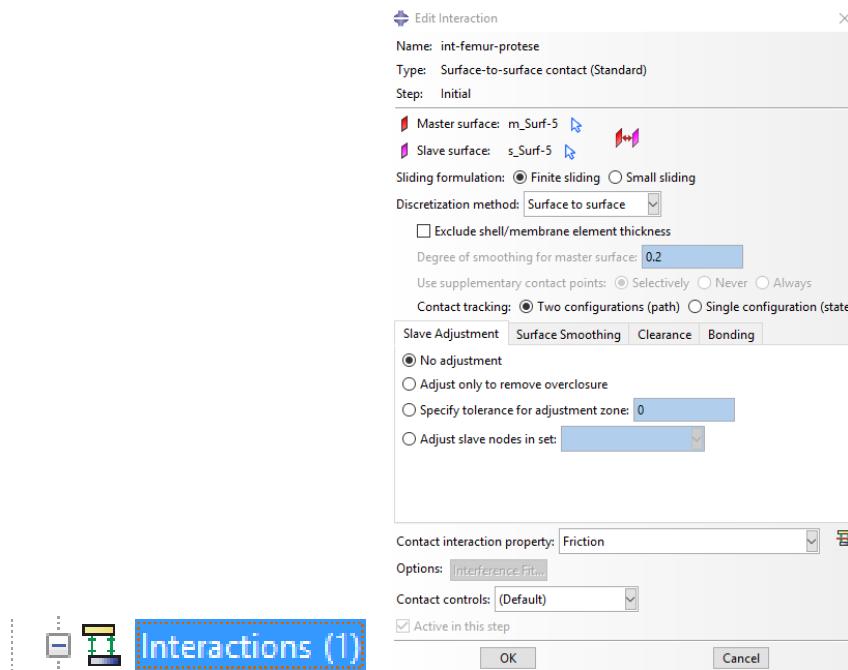


Figure 35: Surface to Surface interaction definition (uncemented)

3.3.2 Results

If we compare the results for a quadratic model with an Elastic Isotropic femur, we can see that there is no strong difference between the Cemented (Figure 36) and uncemented (Figure 37) models. Indeed, the values of Maximal stresses are the same (108 Mpa), however we can see that there is some difference at the interface, where there is, for the uncemented model, some stresses. The cemented implant, show less bone resorption and we see that there is no motion between the two parts of the model.

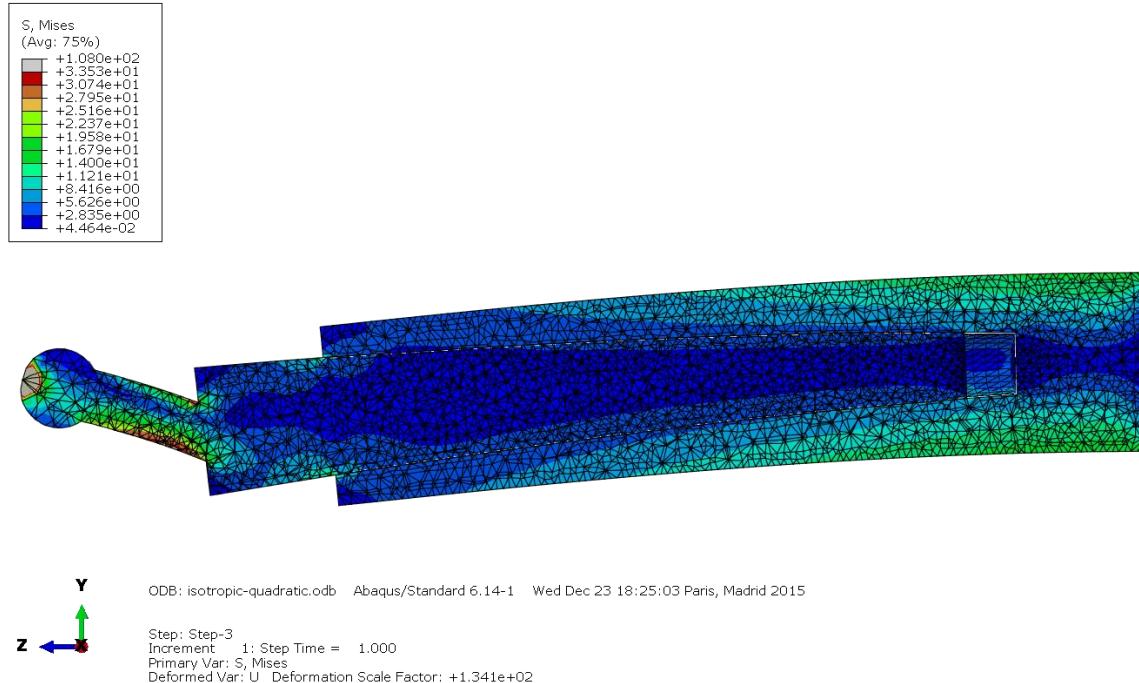


Figure 36: Simulation result : Cemented (quadratic - Elastic Isotropic — F1)

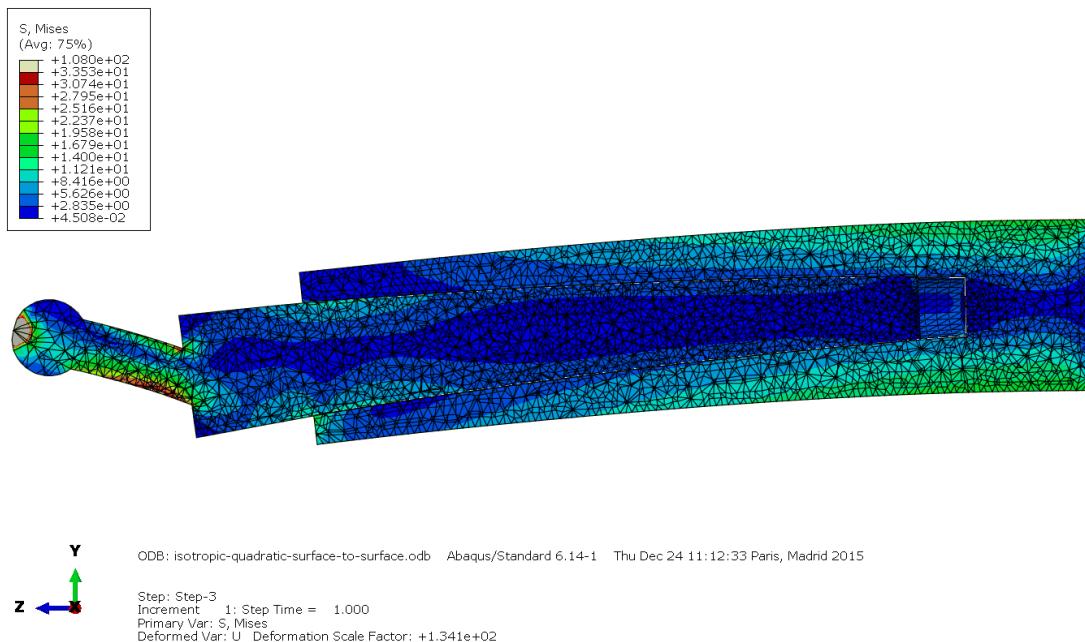


Figure 37: Simulation result : Uncemented (quadratic - Elastic Isotropic — F1)

If we compare the results for a **linear** model with an Elastic Isotropic femur, we can see that there is a strong difference between the Cemented and the Uncemented models. Indeed, we can see that there is a difference in the values of the stresses and in the locations of these ones. For example in the Figure 38 (cemented) the stresses at the interface are only located in the prosthesis part as in the Figure 39 (uncemented) they are located on both sides of the interface showing that a motion exist between the two parts of the implant.

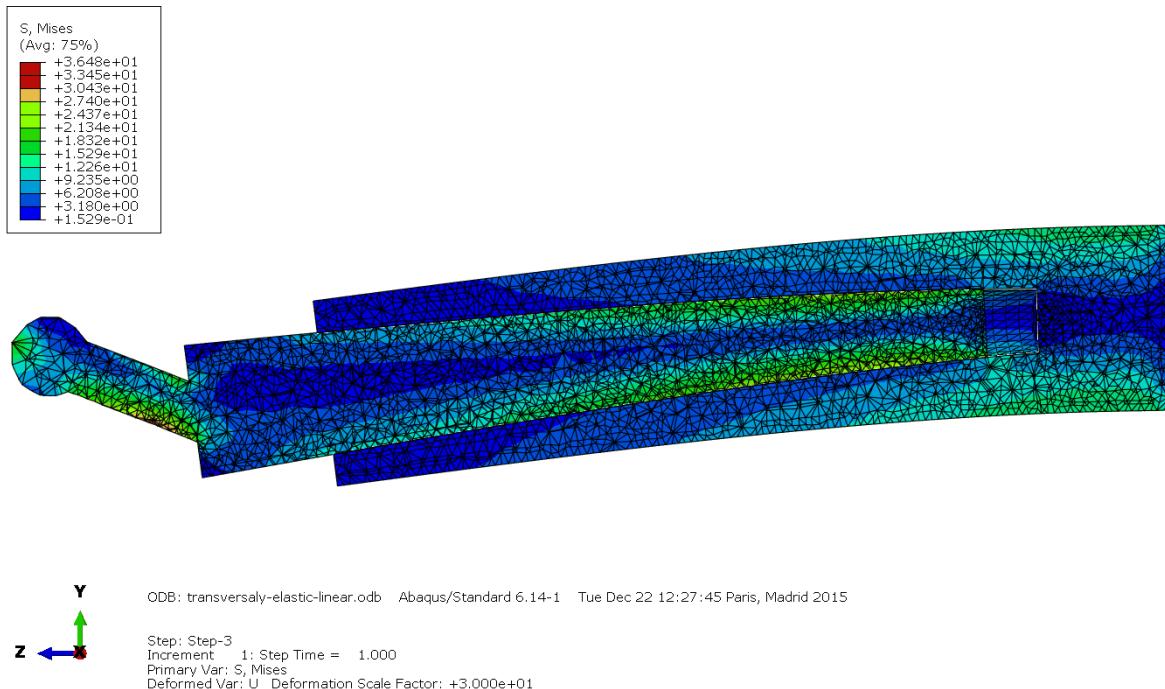


Figure 38: Simulation result : Cemented (linear - Transversely Elastic Isotropic — F1)

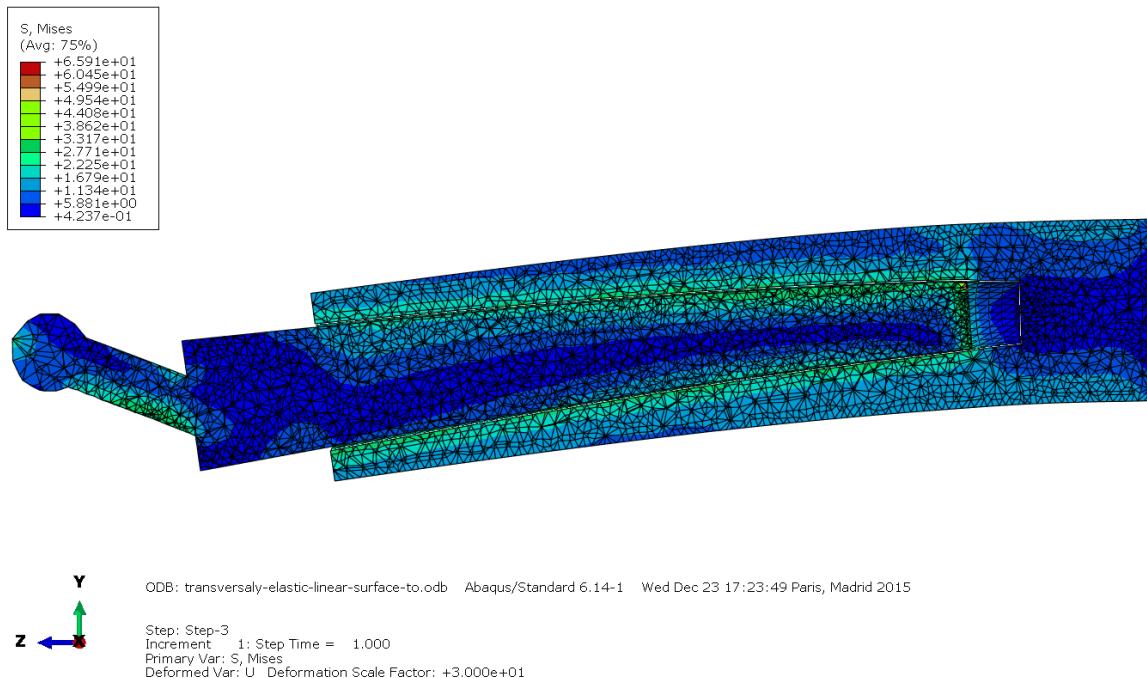


Figure 39: Simulation result : Uncemented (linear - Transversely Elastic Isotropic — F1)

4 References

Correira, J.P.M. 2015. *Evaluation 1 - Stress analysis of a prosthetic implant with ABAQUS.* Strasbourg : Université de Strasbourg, 2015.