POLITECNICO DI MILANO

School of biomechanics and biomaterials engineering

Finite Element Analysis (FEA) of Non-Modular Metallic Orthopedic Hip Femoral Stems

CONVERGENCE ANALYSIS

Laboratory of Computational Biomechanical Laboratory
Dott. Ing. **Jose Felix Rodriguez Matas**

Saverio D'Amico - 875160
Matteo Dell'Era - 875427

Accademic Year 2017/2018

SUMMARY

1 INTRODUCTION

1.1 Motivations and purposes

The aims of the proposed study is to verify the convergence of the finite element method by applying it to a computational stress analysis of a femoral stem of non-modular hip prosthesis in metal materials, and evaluating the most loaded areas. The Max principal and Mises stress data are extracted from the most charged areas for each discretization of the model.

1.2 Description of the device

The analyzed device is a femoral hip prosthesis of metal material used for motor rehabilitation in subjects where normal daily motor activity is impaired. This device is inserted into the femoral cavity, embedded with or without the aid of a cementitious material.

2 METHODS AND MATERIALS

2.1 Model of the device

For computational analysis, the device was modeled by a three-dimensional drawing in STEP format provided for this study. This is a standard format defined in ISO 10303-21 [1] that contains a set of rules for integration, presentation and data exchange between Computer-Aided Systems (CAD, CAM, CAE).

2.1.1 CAD Geometry

In the file described above is located the 3D CAD model of the device. No other changes have been made to the geometry of the model. The CAD display at the software interface appears in *Figure 1*.
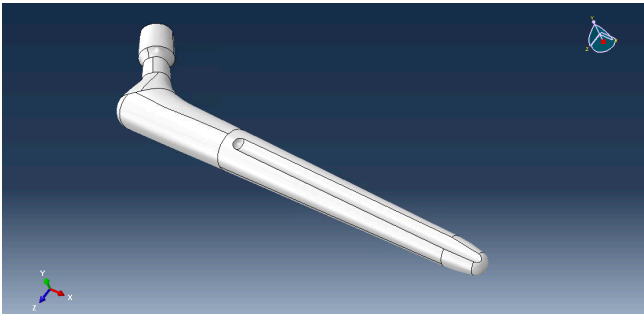

Figure 1

2.1.2 ISO and property rate

The study was carried out following the standards described in ISO 7206-4 [2] defining the orientation of the device in the space, the value and direction of application of the load and the boundary condition. Material properties such as Young's modulus for elasticity (E) and Poisson coefficient (v) are generally obtained from the material certification document. In

this study, the properties of the device will be applied to the model of a similar study previously performed [3] and are summarized in *Table 1*.

MATERIAL	MODULUS OF ELASTICITY	POISSON'S RATIO
Steel	113700	0.3

Table 1

Load application, device orientation in space and boundary condition are exhaustively described in the norm [2] and summarized in the following Table 2. All of these conditions are for a specific stem with a CT distance (distance between center of the head and the most distal point of the stem) between 120mm and 250 mm.
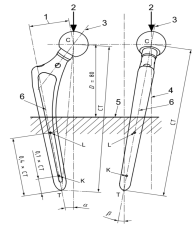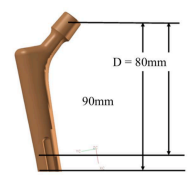
LOAD	A 2300 N of load is applied to point at the center of the maximum offset femoral head in an orientation defined	
ORIENTATION	One rotation in Y axis by 10°. One rotation in Z axis by -9°	
BOUNDARY CONDITION	The hip stem will first be cut at a distance from the center of the head as described in ISO 7206-4 (2010) with the worst-case head/neck offset. A second cut shall then be made 10 mm below the first cut. The hip stem shall be constrained in all directions on all faces distal to the second cut	

Table 2

2.2 Finite elements method (FEM)

2.2.1 Theoretical consideration

The finite element model (FEM) is used in this study to evaluate a numerical solution of a mathematical model that describes a physical problem. With this approach, the model of the device is divided into so many defined shape and size sub-elements on which to proceed by solving a partial differential equation system that is reduced to a system of algebraic equations [4]. It is important to emphasize how the application of this

method inevitably introduces errors and approximations for each application layer (modeling error, discretization error, solution error).

2.2.2 Application to the device
Software ABAQUS®
ABAQUS® proprietary software was used to perform the analysis of the finite elements on the model. The approach to the finite element model is described in the workflow in *Figure 2*.
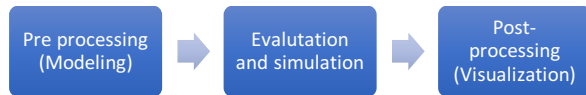

Figure 2

The software is used to run the process in *Table 3*.

1	Display device CAD geometry
2	Apply material property
3	Apply rotations according to ISO
4	Apply boundary condition
5	Apply the load
6	FEM to model
7	Effort simulation
8	Display stress gradient
9	Give Max Principle and Mises stress value

Table 3

Creating mesh
To apply the FEM method to the device, you must convert the model surface to a standard and homogeneous finite element set. Then create a grid called mesh. A type of quadratic tetrahedral element has been defined and subsequently applied by software, with triangular external surfaces, generated by default algorithm with option set to "free" and without the application of any boundary layer. Once the element type is defined, a series of uniform nodes, that will serve as the basis for mesh creation, are applied on the surface of the pattern. These points are called seed (*Figure 3*).
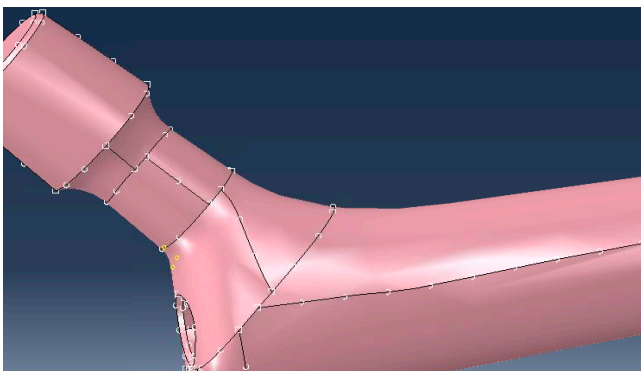

Figure 3 – Seeds in the surface of device

Defining the relative seed distance, you can create mesh on the surface with more or less grain. The surface of the device is then divided into several finite elements, consisting of several nodes. The software has a limitation of about 250000 nodes for simulation, so it is necessary to find a suitable strategy for creating ever dense mesh of elements and avoiding this limit. The strategy adopted was to select only few areas of interest for the creation of high density mesh, while keeping 3/3.5 seed values for the rest of the surface.

Areas of interest
The areas considered are the most loaded by a first qualitative analysis of a generic mesh with a relative seed distance of 1.5. Areas of interest are driver hole, potting level and the neck. (*Figure 4,5*)
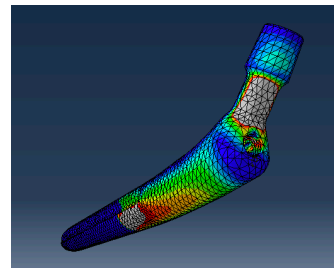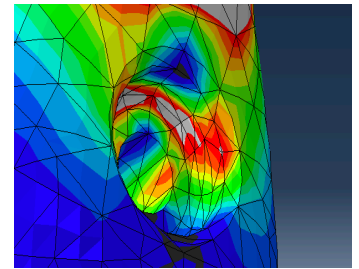

Figure 4 – Potting level and neck   Figure 5 – The driver hole

Exclusivity of areas of interest: Bias method
The Bias method was used to proceed with the exclusive high mesh creation in the areas under investigation. This method provides greater control over the application of the seed along the contours of the model. In fact, it is possible to apply a progressively greater or lesser distance value between a seed and its next (*Figure 6*).
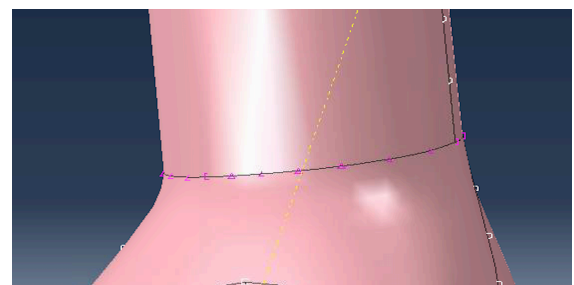

Figure 6

The criterion for perform the Bias method is to cover with a high-density mesh the direction where it is more obvious, from a qualitative analysis, a higher speed of propagation of the stress gradient. Once these directions were identified, was set out a criterion for applying the bias method for the three areas, schematized in the following *Figure 7*.
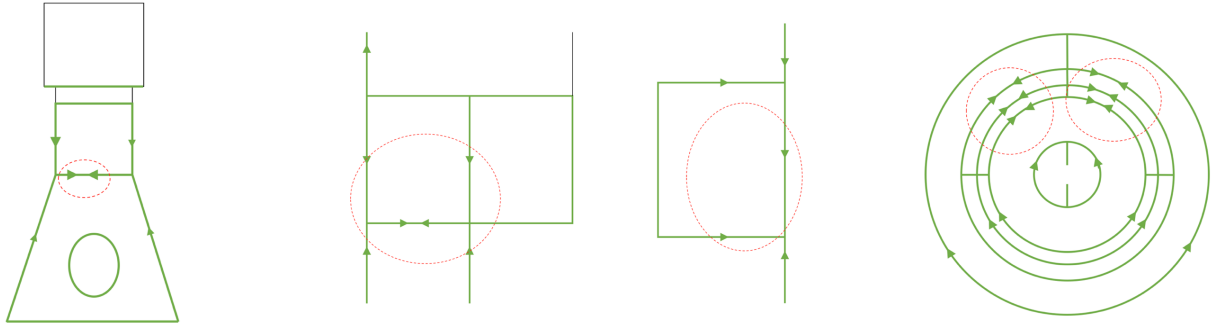
The value of the various seed, both linear and bias, does not have a default. An appropriate value was chosen every time, to avoid errors in the mesh. Provide a more detailed analysis of the errors and their approach in the following chapters.

Mesh development

As mentioned above, in order to carry out the convergence analysis, the model of the device is discretized in finite elements by smaller size, increasing the density of the mesh gradually. The seed value has been chosen to maintain an exponential growth as much as possible (*Figure 8*).
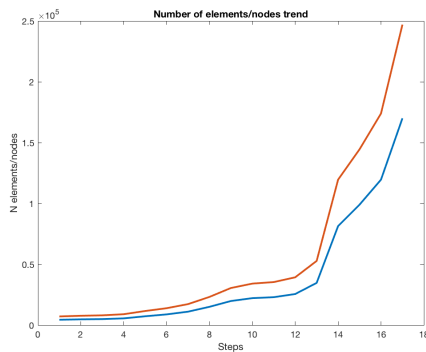

Figure 8

2.3 Data processing

2.3.1 Software output

The software allows the extraction of Max Principal and Mises stress parameters by manually selecting elements in the area of interest identified. For each selected tetrahedral element, are indicated the specific values for the gauss point, were the integral is approximate. All data is taken by a single operator on the same project file, for minimizing the interoperability variability that may occur, mainly in the choice of the area of interest.

To select the elements always in the same area, the graphic display of the stress has been changed in order to visualized every time the same stress area. In addition, the data is taken simultaneously on each area of interest, to avoid errors related to different mesh creations. The software mesh creation algorithm in fact is based on random variables, therefore, for each mesh created, all the data of interest are extracted at the same time.

2.3.2 Matlab® function

An ad hoc Matlab® function has been developed to automate the process of extracting interest data from each output files, for each area of interest. It performs the following functions in *Table 4*.

1	Open output files for every mesh value
2	Extract vector contains the stress value from output
3	Mean of all stress value for every element selected
4	Create stress vector value at every mesh value
5	Create vector for perceptual increase
6	Create number of elements vector
7	Plotting data

Table 4

3 RESULTS

The *Table 5* shows the minimum seed value and the number of elements and nodes created for each step of mesh creation, in addition, the stress values extracted from the output file of the software through the developed function. All the seed bias values used along each contour are not shown, only the smallest. This is because, as mentioned above, the bias values were chosen by adapting to the configuration that allowed the least number of errors in the mesh.

STEPS	MIN SEED VALUE	ELEMENTS NUMBER	NODES NUMBER	POTTING LEVEL		NECK		DRIVER HOLE	
				Max P.	Mises	Max P.	Mises	Max P.	Mises
1	8	4438	7160	134.41	135.08	284.32	260.13	121.68	133.09
2	7	4764	7710	148.35	148.93	296.87	261.28	131.12	139.73
3	6	4978	8052	148.86	149.41	295.72	265.99	134.46	145.86
4	5	5568	8984	148.98	149.16	280.18	246.70	123.48	133.31
5	4	7283	11616	148.54	148.74	317.43	284.69	125.63	139.14
6	3.5	8836	13915	148.56	148.73	293.04	261.16	123.95	140.45
7	3	11074	17278	148.39	148.84	314.32	295.47	129.05	140.02
8	2.5	15067	23205	150.45	150.38	332.70	295.38	134.88	149.44
9	1.5 (Bias)	19813	30520	159.60	158.78	345.04	318.61	126.79	146.82
10	1 (Bias)	22196	34138	160.52	158.39	363.36	323.47	130.65	148.01
11	0.8 (Bias)	22984	35399	161.71	158.39	366.31	328.05	145.88	156.88
12	0.5 (Bias)	25573	39322	158.86	158.88	397.22	361.27	138.02	156.13
13	0.25 (Bias)	34738	52794	166.62	165.61	414.44	379.16	159.84	173.70
14	0.125 (Bias)	81420	119596	167.13	167.07	418.21	384.71	164.50	177.63
15	0.1 (Bias)	99030	144525	167.53	167.24	413.50	381.77	167.68	180.17
16	0.08 (Bias)	119564	173912	167.91	167.68	421.55	388.22	168.25	180.02
17	0.05 (Bias)	170016	247002	167.94	167.58	424.33	391.78	169.99	183.38

Table 5 – All the data results from the analysis. When the bias starts, at the seed value of 1.5, the global seed value has been set to 3 until the value of 0.125. From 0.125 the global value has been set to 3.5 to avoid the nodes limit.

4 DATA ANALYSIS

4.1 Analysis of stress values

The following charts show the stress trends for each area of interest, depending on each step made and on the number of elements created.
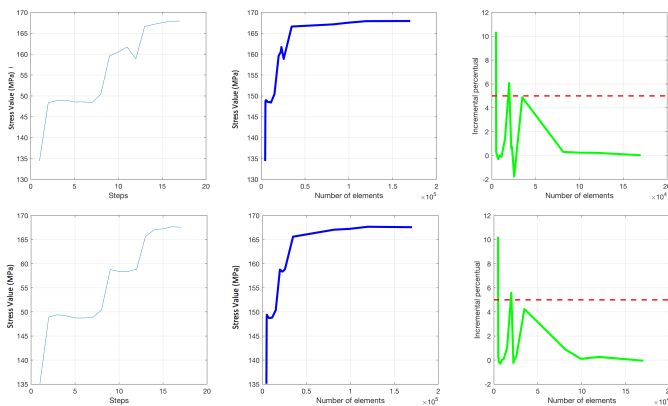

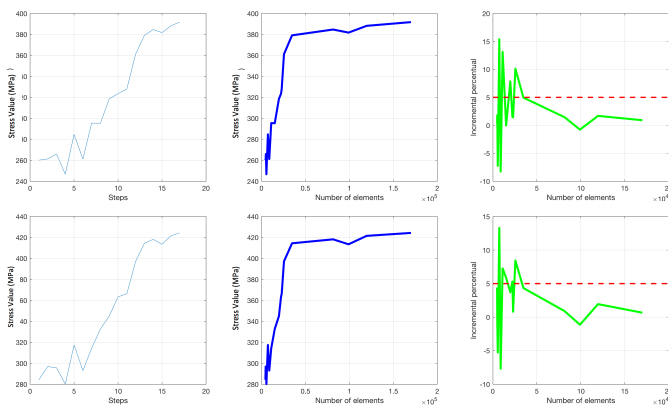*Figure 9 – Max principle and Mises stress value for POTTING LEVEL*

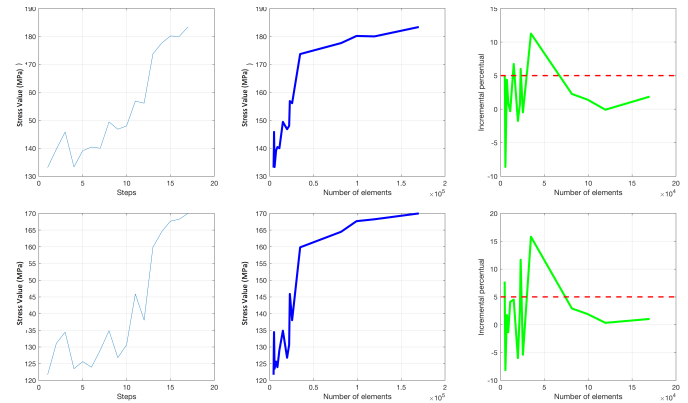*Figure 10 - Max principle and Mises stress value for NECK*

Figure 11 - Max principle and Mises stress value for DRIVER HOLE

4.2 Discussion

The results obtained, particularly by looking at the value of the stress respect to the growth of the number of elements, shows a clear convergent trend, despite several initial oscillations. The starting point was a very low-density mesh (seed 8), which clearly influenced the measurements, making the value of the average stress imprecise and unstable. This value is in fact mediated between the four vertices of each element. It is easy to understand how, in a larger element, the value is mediated around a large area, far from the actual value of the area of interest. Convergence is also evident from the graph showing the trend of the percentage.

4.4.1 Convergence criterion adopted

In order to pursue our purpose, and to establish the converging trend of the average of stress value in the areas of interest, a unique criterion was chosen. Is considered the percentage variation between a calculated average stress value (for every mesh value)

and its successor. The following algorithm is implemented in the Matlab® function.

$$\frac{data(i+1) - data(i)}{data(i)} < 0.05$$

Where $data(i)$ is the vector containing the average stress value for each mesh created. Despite the oscillations due to the errors by the choice of a low-density mesh or related to an untrustworthy choice of the area of interest, the overall trend of the percentage variation is to decrease gradually respect the increase of density mesh. For all areas of interest studied this percentage variation stays below the threshold, defined a priori. Therefore, according to the hypotheses advanced during the study, convergence is confirmed on each area of interest. It is noticed that in the potting level and the neck, percentage trend tends to zero for each iteration, the same thing does not apply to the driver hole, where the tendency of the stress has not reached a convincing stability, but it continues to increase. This may be due to the failure to reach actual real value. Probably going further, the tendency of the stress value would reach stability in the same way as the other two areas of interest. However, limited by the features of the software used in this study, it was not possible to proceed further. Anyhow, the stress value of the driver hole remains below the threshold.

4.3 Problems

Problems encountered during the mesh creation phase. The exclusivity high-density mesh creation in the three areas has inevitably led to a substantial difference in the size of the elements between the different parts of the model. In order to link all elements, approximations are introduced by the software, that in some cases produce mesh distortions causing alterations in the actual stress value. To solve this problem, and then obtain a mesh as little as possible distorted, seed values are selected, with or without Bias method, dynamically (while keeping the minimum following criterion described in 2.2.2.5) to search for the least distorted mesh configuration. It was also avoided to take data near the areas where are present, also minimal, distortions.

4.4 Conclusions

The stress value for each area of interest is consistent according to the referral study **[3]**. Also the convergence of FEM method, applied in this case to a non-modular metallic orthopedic hip femoral stems stress analysis, was proved according to the assumptions made in this study.

5 REFERENCES

[1] International Standard ISO 10303-21 (2010); STEP-File, data exchange.

[2] International Standard ISO 7206-4 (2010); Determination of endurance proprieties and performance of stemmed femoral components.

[3] Standard Practice for Finite Element Analysis (FEA) of Non-Modular Metallic Orthopedic Hip Femoral Stems.

[3] Course of computational biomechanics laboratory, lesson slide by Dott. Ing. J. F. R. Matas, 2017/2018