Structural Analysis and Finite Elements
CNST-H421

Project: Wrenche FEM Analysis

Electromechanical section

BERGAMASCHI Alexander Giovanni, RIIS Mehdi, VAN DELFT Julien

Pr. Peter BERKE, Antoine RANDOUR

Engineering Science

2020-2021 academic year

Table of contents

PART I - Project assignment

Introduction

As part of the course of *Structural Analysis and Finite Elements*, the study of forces acting on a wrench has to be done. To do so, the finite elements approach will be used. One will write a FE code in MATLAB which will provide a good understanding of the finite elements method. One will then use Salome-meca and compare the two software approaches.

NB: the units used in *Salome-meca* are Newton (N), metres (m) and Pascal (Pa). One can then see in the figures Pascal for strains and stresses unless something else is precised.

1   FE Model

1.1   Geometry and materials

The objective in this project is to design a wrench. The design of the wrench was not restrictive as any group could chose a wrench found in the home toolbox. A technical drawing of the wrench was drawn in *SolidWorks* (see fig. 1) so that the dimensions of the wrench were clearly shown to the reader.

Concerning the materials, the wrench is only made of chromium-vanadium steel. The Young's modulus (E) and Poisson's ratio (v) of this group of steel alloys are respectively around 190 GPa and 0.29. From the sources [1] and [2], the Yield strength is around 420 MPa and the Fatigue strength around 300 MPa.

Two different geometries were used for two different models, as it can be seen in Figure 2, a bad geometry with sharp corner and one more precise with a smoother transition between the head and shaft of the wrench.

1.2   Loads and boundary conditions

The loads and boundary conditions were chosen to reproduce the usual working parameters of a wrench that is used by an operator to screw or unscrew a nut or a bolt. The force is supposed to be distributed on the wrench's arm along an average human hand width of 80mm. A maximum force of 200 N or 2500 N/m was supposed to be enough to use the wrench for a large range of applications. This force is meant to simulate the load that can be developed by hand on a 3D wrench. Nevertheless, it is an assumption as discussed in section 1.5.

The wrench is supposed to be blocked by the bolt, therefore the boundary conditions were positioned in the inner profile of the wrench head, where the contact should happen. As will be discussed, two models were used, a bad one and a smooth one. Concerning the bad geometry model, the boundary conditions were applied along the whole upper and lower inner profile. Nonetheless, in the case of the good one, it was assumed that contact happens along two definite lengths of 7.5mm in the upper

and lower inner profile (see fig. 1), therefore it was imposed a zero displacement along it. However, this suppositions are an approximation which will be discussed further on.

1.3   Chosen mesh

In order to analyse the wrench, the 2D-mesh consists of triangles (TRIM3). To avoid enduring an excessive calculation time, one can build a first coarse mesh which indicates where the stresses and strains are the most important. Then, the mesh has to be refined in these zones until convergence is reached.

The following meshes were used:

- First mesh with coarse structure, few triangles

- Second mesh with finer elements in the found zones

- Latter mesh refined until convergence reached.

The mesh was refined where stresses and strains were more important until when higher refinement did not significantly change the previous result. In this particular case study, the mesh needed to be refined at the wrench head, near the inner profile corners where the stresses are quite important as explained in section 1.6.

1.4   Justification of the modelling approach

The importance of this study was to be able to correctly predict the behaviour of a wrench under operating conditions. It is therefore fundamental that all the assumptions made for the models allow this prediction. Having models as close as possible to reality would allow to obtain the most precise results. Nevertheless, it is well known that increasing the complexity of the model means higher conditions, more work, more computational power required, higher expenses. Consequently, it was essential to find a balance between these factors, the assumptions made in section1.5 try to find the best compromise.

Two models were analysed with three different meshes each as discussed in section 1.6. The purpose of using these two models is to see how the geometry has to be modelled compared to reality in order to be able to get converging stresses. Starting with coarse meshes and refining these in few steps, it was possible to reach a more precise model only where needed.

1.5   Assumptions and simplifications

A model of the wrench was developed to study its behaviour under strict assumptions. The model was simplified into a 2D problem in which all the applied forces, boundary conditions are within the xy-plane. This could be done because in reality the thickness of the wrench and its properties are

almost constant along the body; moreover the tool is made to work in certain conditions which do not contemplate forces or moments applied in the z direction. The strains and stresses along this third dimension can be approximated considering a constant thickness in the z-direction and through two distinct assumptions: the plane stress assumption, which sets σ_z equal to zero, or the plane strain assumption with ε_z equal to zero.

While with the code implemented in *MATLAB* it is possible to choose whether to be in plane strain or in plane stress giving a different input "key". With *Salome-meca* and also for PART II of this report, it was decided to analyse the plane stress case which is the best assumption to make with a wrench.

Another simplification of the model is its division in a finite number of TRIM3 elements, stresses and strains cannot assume a continuous trend as in the reality but these change in a discrete manner with a constant value by each triangle. However, this problem can be tackled increasing the number of elements; this should be done mainly where the stress and strain have a high gradient. In this way, the model approaches the real solution and the computations are reduced where not needed. This procedure will be described in the convergence study in section 1.7.

Furthermore, it is assumed that the applied load is distributed constantly along the length of the wrench. In reality, this load is applied by the operator's hand, therefore its distribution is not constant but it depends by the position of the fingers and other negligible factors.

Finally, the model assumes an unbreakable wrench as the material behaviour is assumed to be linear elastic. This is not a problem as the wrench is used with small loads, so in the elastic region and far from the plastic one. This assumption is valid.

It is important to note that despite all the restrictions and simplifications that were made to build the model, under normal working condition of a wrench, it can perfectly describe its behaviour with good approximation.

1.6 Presentation of increasing complexity FE models

Two models were analysed during the lab sessions, different geometries and different elements sizes were chosen.

First, a bad geometry model with sharp corners was made with three different mesh sizes computed. Due to the sharp edges and as expected, a problem occurred in the convergence study. Therefore the geometry was improved.

Then, a more precise model with smoother corners at the connection between the head and the shaft was made and three different meshes were used again for the new geometry. The convergence of this latter one is discussed in section 2.2.

1.7   Mesh convergence establishment

The convergence/divergence of all the parameters is something to keep an eye on. Indeed, after a certain number of elements, the values of stresses and strains should converge. In order to establish the convergence, each model has been refined quite a few times to be able to plot the values in *MATLAB* which gives a better visual insight of the situation. This procedure was followed and applied to the two models. The results are discussed in section 2.2.

1.8   Mesh refinement strategy

The aim of the refinement strategy is to obtain a meshing that gives the best compromise between a good representation of the stresses of the body and a low computational power to obtain these results. To make the convergence study, a coarse mesh was first used. Then, a very fine mesh over the whole wrench was computed but this latter didn't give better results than the coarse when looking far from the head. Thereupon, a coarse mesh was computed for the parts far from the head and the head-to-shaft edges. In fact, it was observed that the stress and strain gradients are mainly in the wrench head and near the head-to-shaft connection. To increase the complexity of the model, more refined meshes were used in those zones. It would have been better to decrease the element size only where it is needed and not in the whole head of the wrench. Unfortunately, it was not an easy procedure with *Salome-meca*. Finally, it was preferred to refine the whole head, including the head-to-shaft connection.

2   Numerical results

2.1   Salome-meca results

First, the coarse geometry is being studied in figure 3. It is seen that there are very localised stress concentrations in the wrench. Those concentrations are due to the geometry changes close to the sharp edges. Nevertheless, due to the sharp edges and boundary conditions, the finite elements close to those changes have stress singularities. This explains the form of those stresses ultra localised.

By looking at figure 4 with the better geometry, it is seen that there are stress concentrations at the end of the wrench arm. This explains by the change of angle performed by the arm to reach the head of the wrench. Indeed, by refining more and more the meshes, the stresses get localised on the edge of the arm's end. This confirms the first interpretation of the stresses locations as it gets closer to the change of geometry and there are no big stresses in the centre. One can also observe some big stress concentrations around the boundary conditions. Actually, the boundary conditions

restraining the points from moving lead to singularities in the finite elements close to those points as for the coarse/bad geometry. Nonetheless, those singularities remain local as stated by the *St. Venant*'s Principle [3] and this is showed in figures 4 and 5 by the stresses being huge in only few finite elements around the boundary condition lines.

2.2 Convergence results

Now, to prove the good quality of our meshes and of our calculations, the stress and strain values need to converge to a single finite value. As explained in the previous section, the finite elements may not converge for the points close to the stress singularities created by our finite elements model. Those singularities do not happen in real life of course but are due to the non linearity of the real wrench and the deformation of the bolt being screwed not taken into account. This verifies in figure 5 where the two elements close to the boundary conditions do not converge while the two elements close to the change of geometry are converging to a finite value. The stresses and strains for the third and fourth elements are all coming up nicely with some small changes around a fixed value. The von Mises stresses for those elements seem to converge to a value close to 230-240 MPa which is also the value found by the software *Solidworks* on the 3D-wrench in figure 1.

Moreover, for the coarse geometry, as the element close to the sharp edge sees a sudden non-linear change of geometry, it suffers from stress singularity. This means again that the element stresses should diverge and rise to infinity. It is shown in figure 6 and it proves the imperative need to consider a better geometry that got rid as much as possible of the sharp edges.

2.3 Comparison of the different models

2.3.1 First model

The first model can be seen in the upper part of figure 2. The geometry depicts well the wrench but it presents sharp corner in the connection between the head and the shaft. Different mesh sizes were computed for this geometry. First, a coarse mesh was used. Thanks to the computation made in *Salome-meca*, it was possible to identify the elements and so the zones where the stresses and the strains are higher. It is possible to observe in figure 3 that these zones are near the junction between the head and the shaft of the wrench. From material science, it is known that the stress distribution is higher near sharp corner and notches, this notion is reflected in the first model. Furthermore, it can be observed that at the trail, where there are no loads applied, the stresses are really low. This means that the stresses and strains that should be studied are within where the load is applied and where the boundary conditions are present. These results could be obtain formulating a trivial problem of machine design.

Nevertheless, this model does not well represent the real case. It is possible to see that the colour between two adjacent element changes from red to blue suddenly. There is no smooth transitions

between elements with high stress and other with low one. For this reason, the meshes were refined where the stress gradient is higher obtaining better results as can be seen in figure 3.

After each refinement, the convergence was checked as it is something crucial one wanted to reach. Unfortunately, after quite a lot of refinement no convergence was reached with this model as discussed in section 2.2. Another model had to be chosen to counteract the sharp edges and singularities causing divergence.

2.3.2   Second model

As a matter of fact, the previous model was not sufficient to correctly study the stresses present in the wrench. It was observed that the design does not represent the real geometry of the wrench and most important, as stated in section 2.2, it was observed that coarse elements close to sharp edges suffer from singularities.

Therefore, the design was improved in *Salome-meca*, making the geometry of the model more similar to the real one with smoother angle as can be seen in the lower part of figure 2. Then, the model was refined until convergence was reached (see fig. 4 and 5). With this geometry different mesh sizes were computed so as it was possible to observe better results compared to the first one, especially for the coarse mesh.

The main difference between the two models is the head-to-shaft smoothing which made the model parameters converge as expected. What could have been done is a third model with more precise zones of refinement. Unfortunately, time has passed too quickly for this to be done. Nonetheless, a *Solidworks* model was done in 3D to compare our final *Salome-meca* with it. The results were quite close as stated in figures 4 and 1.

3   Discussion

The studies performed on the wrench model showed that the stress concentrations were close to the edge of the wrench arm end and to the boundary condition lines. In order to study the behaviour of our real wrench, it is necessary to compared the found values with the material properties. First, by looking at the Yield strength and the maximal von Mises stresses in the wrench (excluding the singularities close to the bolt), it can be concluded that the applied load can be increased by a factor of 1.75 to reach the critical value where the wrench will start to plastic deform. Indeed, the maximal von Mises stress is equal to 240 MPa and the Yield strength is 420 MPa. In reality, the points close to the bolt have also big stress concentrations but those values cannot be estimated through our finite elements model due to the non-linearity not considered in the model. Nevertheless, the wrench has a bigger thickness for the head increasing the mechanical resistance in order to counter those stress concentrations.

Concerning the fatigue fracture, the same process applies with the fatigue strength of 300 MPa. This means that if this value is reached somewhere in the wrench, it will break after 100,000,000 cycles. By comparing it again with the maximal von Mises stress, one can conclude that the load can be increased by a factor $\frac{300}{240} = 1.25$. This means that a normal wrench should not suffer from fatigue rupture in normal conditions.

4   Conclusions

The procedure described in this report is a good method to obtain the stress and strain distribution of a wrench under working condition. However, it is important to keep in mind the fact that numerous assumptions have been made and a good comprehension of structural analysis is always needed to well interpret the numerical results. Moreover, it is important to always make sure to be in the working conditions defined for the model, otherwise unexpected or catastrophic failures might happen. As it will be explained in PART II, the results obtained with *Salome-meca* were tested and compared with a Matlab implementation, in this way it was possible to asses the correctness of the models described.

Finally, this project was really helpful in the learning and comprehension of the *Structural Analysis and Finite Elements* course material. Also, it helped in making first models with a finite elements software as it will be useful in few years for some of the electromechanical students. This is indeed added to the skills learned which is important for beginning in the world of work.

5 Work distribution within the group

Work part	Julien	Mehdi	Alexander
TRIM3	**X**	**X**	
Single FE verification		**X**	
Boundary conditions and reaction forces computation	**X**		
DPT	**X**	**X**	
Plate with a hole study in *Salome-meca*	**X**		
Wrench study in *Salome-meca*	**X**		
Convergence of the wrench results in *MATLAB*	**X**		
Report writing		**X**	**X**

Table 1: Work distribution

References

[1] MakeItFrom. Sae-aisi 6150 (g61500) chromium-vanadium steel. https://www.makeitfrom.com/material-properties/SAE-AISI-6150-G61500-Chromium-Vanadium-Steel, May 2020.

[2] boltport. Chromium vanadium steel aisi 6150. https://www.boltport.com/materials/chromium-vanadium-steel/aisi-6150/, 2017.

[3] M. Acin. Stress singularities, stress concentrations and mesh convergence. http://www.acin.net/2015/06/02/stress-singularities-stress-concentrations-and-mesh-convergence/, June 2015.

[4] Greg Todd. Stress concentrations at holes. http://www.fracturemechanics.org/hole.html.

6 Figures and tables

Here are listed the figures and tables used in the present part of the project.
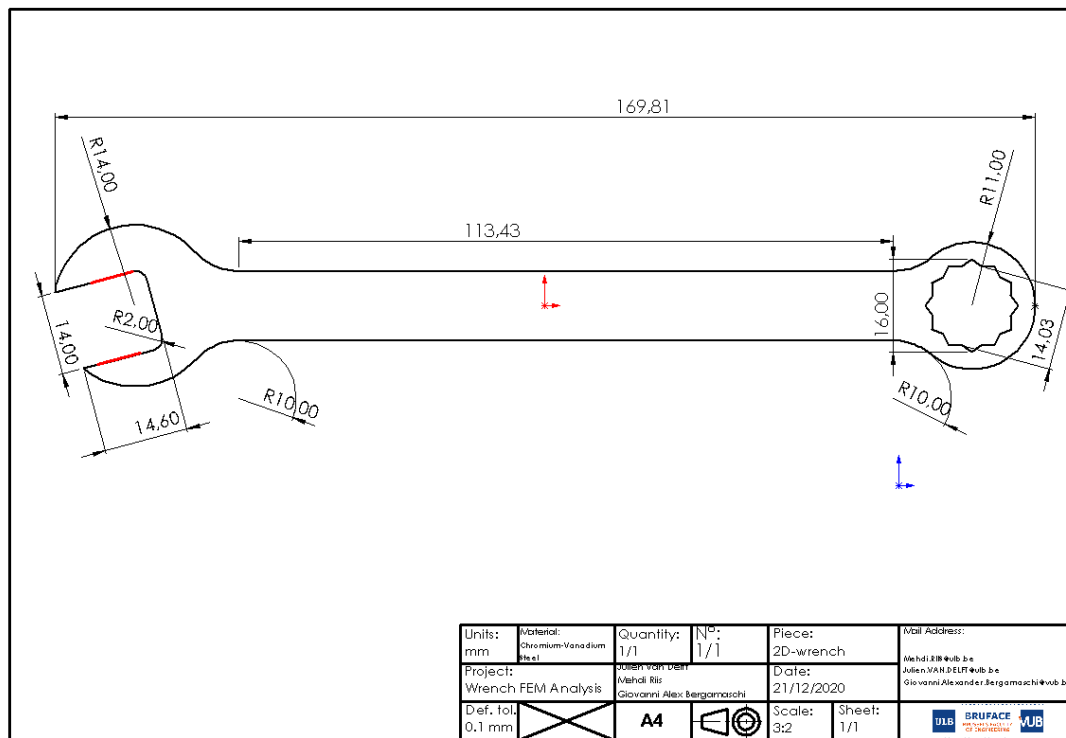
Figure 1: Technical drawing of the 2D-wrench
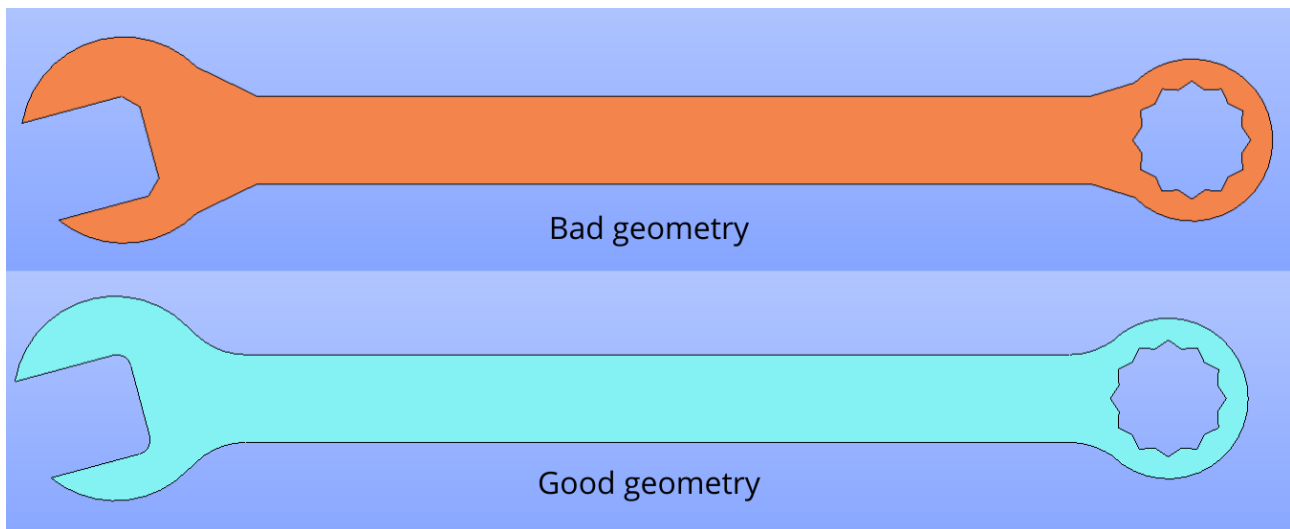
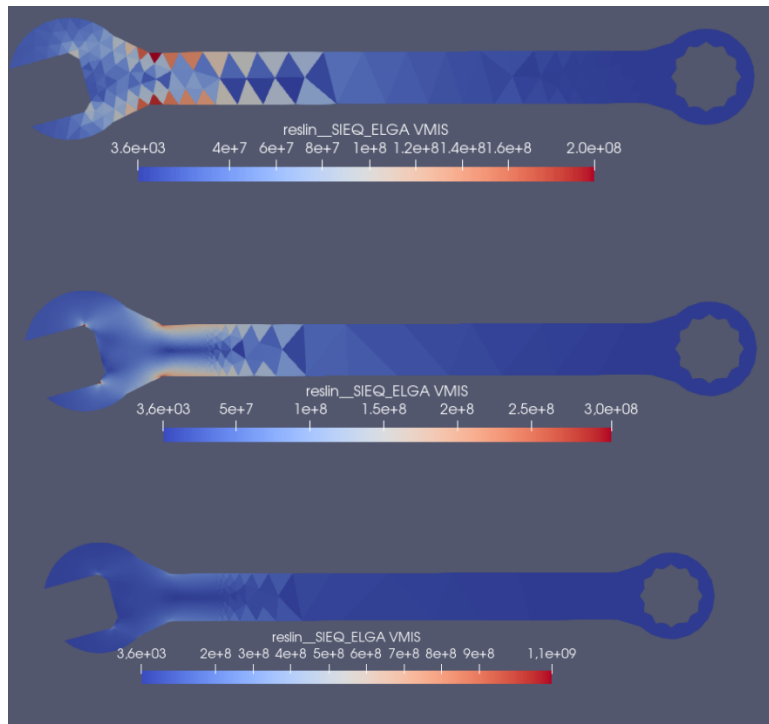Figure 2: Overview of the bad/coarse and good geometries

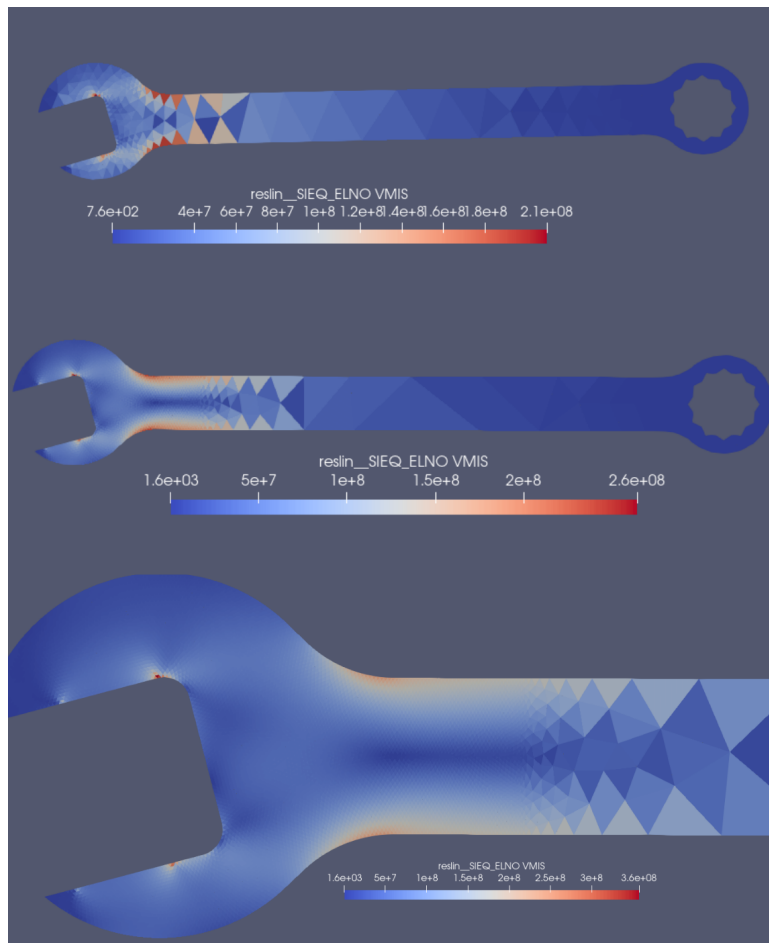Figure 3: Von Mises stresses of the bad geometry model for 3 different meshes

Figure 4: Von Mises stresses of the good geometry model for 3 different meshes
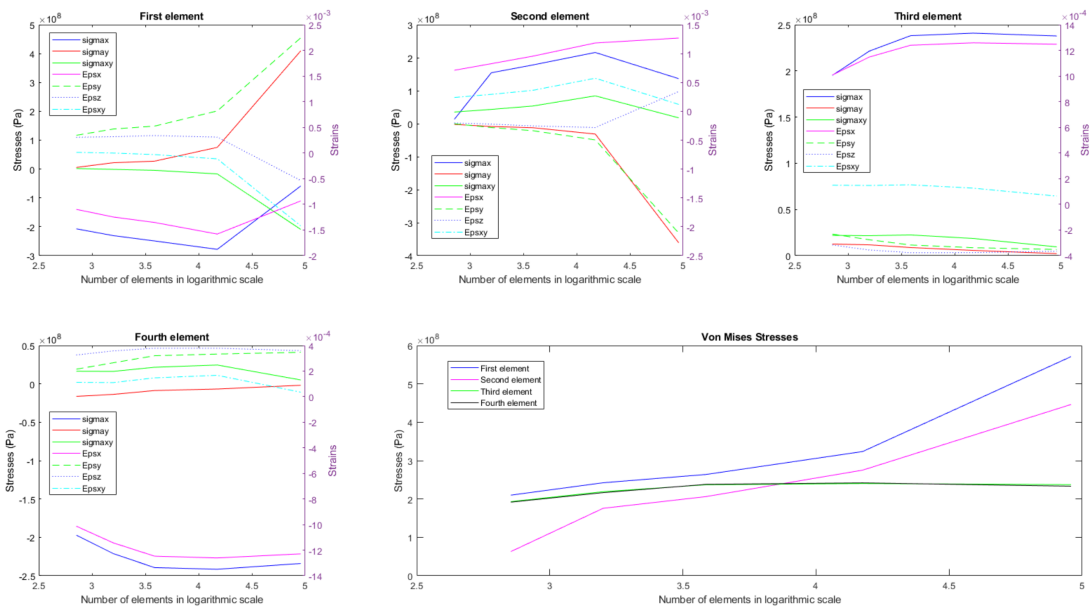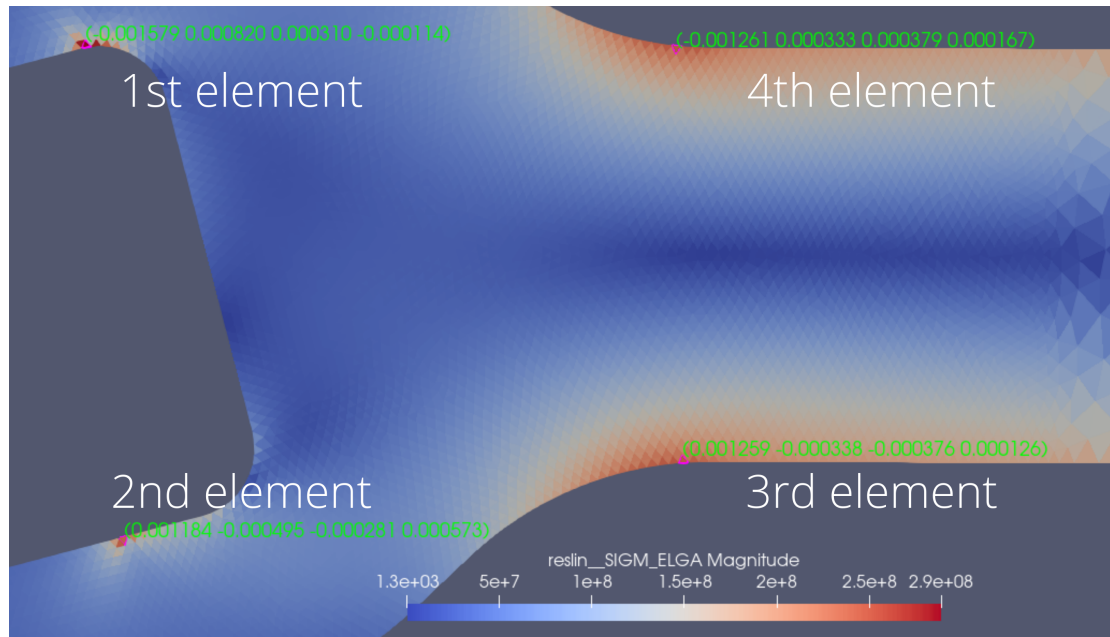
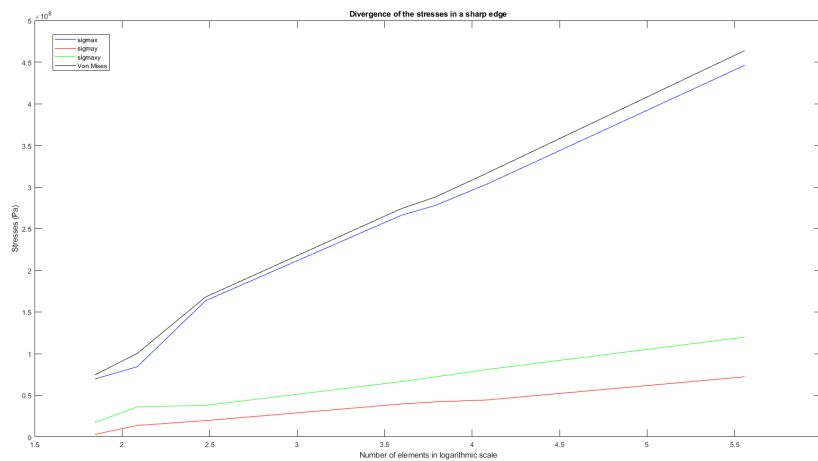Figure 5: Convergence plots of the stresses, strains and Von Mises stresses at 4 element locations

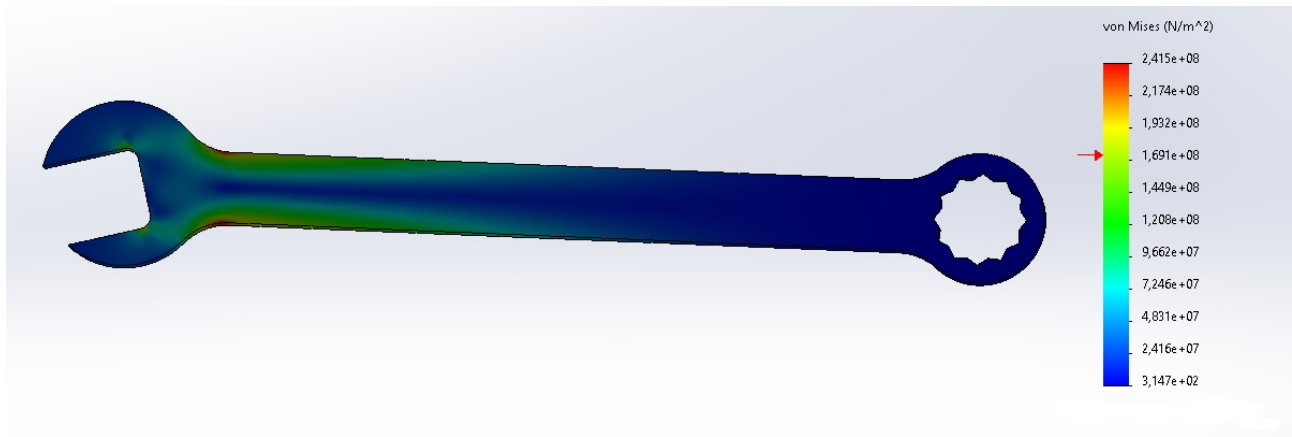Figure 6: Divergence plot of the stresses at the 3rd element in the bad geometry

Figure 7: Solidworks Von Mises stresses

PART II - MatLab implementation

7 FE code verification

7.1 Single Finite Element

The first test consists of analysing a single finite element (see fig. 8) to prove the effectiveness of the code used. To do so, the two left nodes were fixed while a displacement of -0.5 in the x-direction was applied to the right one.
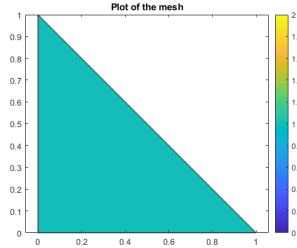

Figure 8: Finite Element studied

Figure 9: K Matrix

The aim of this test is not to use realistic values for E and ν but to see the exactness of the stresses and strains calculation. Then, to make this test easier, one sets the values of E and ν at respectively 1000 Pa and 0.3. As one assumes a plane stress configuration, the stress along z should be 0. Also, the displacement being applied along x, ε_y and γ_{xy} are indeed 0. It is indeed the case as can be seen in figure 10. Finally, one should get $\varepsilon_z \neq 0$. The figures show a perfect set of values of what one should get. Thus one can say the code calculates correctly the parameters discussed. Furthermore, as can be seen figure 9, the K matrix is a symmetric matrix.
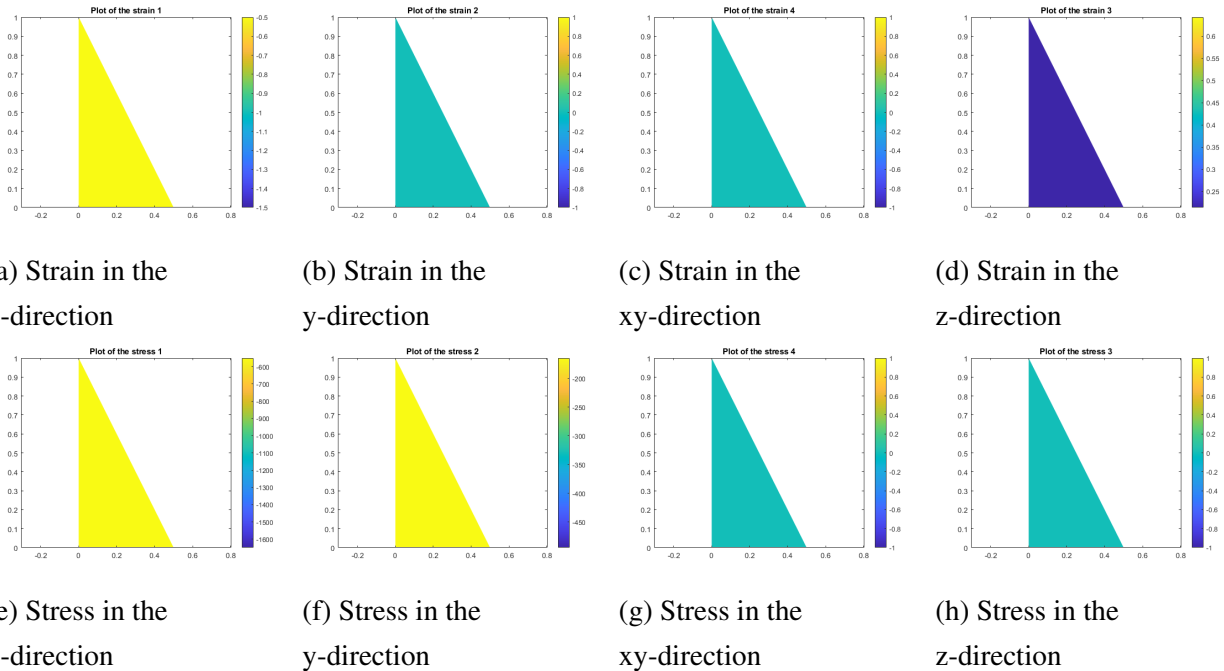

(a) Strain in the x-direction

(b) Strain in the y-direction

(c) Strain in the xy-direction

(d) Strain in the z-direction

(e) Stress in the x-direction

(f) Stress in the y-direction

(g) Stress in the xy-direction

(h) Stress in the z-direction

Figure 10: Single Finite Element Study

7.2 Displacement Patch Test

To verify the accuracy of our finite elements codes and calculations, some displacement patch tests were performed. The element studied is a square formed of 8 TRIM3 triangles (see fig. 11). The square has a length of 2 and is expanding in the x and y positive parts of the xy-plane. The rigid body (r-) modes and constant strain (c-) modes are checked during the displacement patch test. The parameter to look at here is the strain. As in the previous section the relations between stress and strain were already verified, it is not necessary to look at the stress here. Finally, the displacement is already indicated in the plots of the strains, so no need to plot it neither.
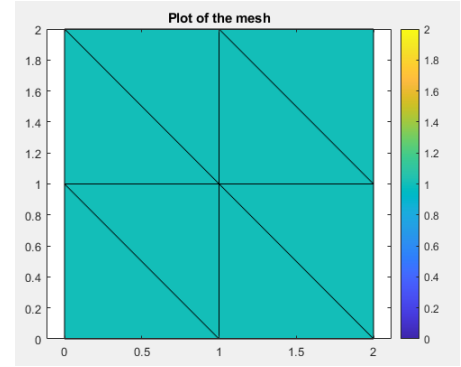

Figure 11: Element studied

To check the **r-modes**, no strains should be generated if the nodal displacement is a rigid body displacement. Applying a constant displacement on all the external nodes and observing the reaction of the internal node, one should see the element translating along the direction of the displacement with the internal node following the motion of the external nodes without causing any strain in the element. The figure 12 shows the strain for a constant displacement of 1 along x (fig. 12a), along y (fig. 12b) and for a rotation motion (fig. 12c). As it can be seen, the strain along x is equal to zero for the first figure, the one along y is also null for the second figure and the shear strain is equal to zero for the rotation. One can finally say that there is an agreement with the theoretical prediction.
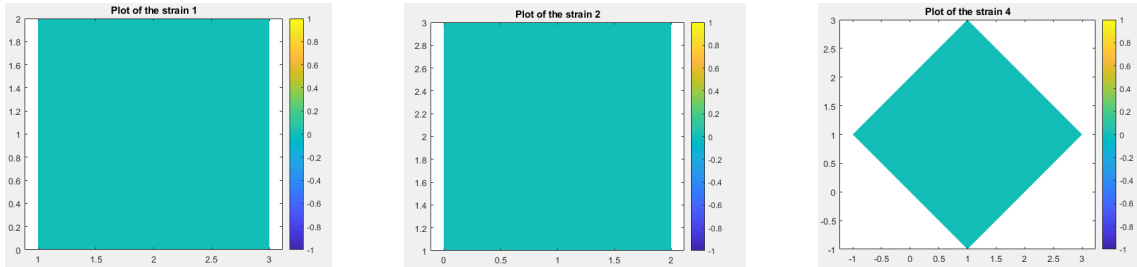
(a) Strain caused by the displacement along x

(b) Strain caused by the displacement along y

(c) Strain caused by the rotation in the xy-plane

Figure 12: r-modes verification

Now, for the **c-modes**, if the nodal displacements are linearly increasing displacements one should get constant strain field recorded. Opposite displacements are applied to the element. If the finite element code is correct, by imposing a displacement of 1 on the external nodes in 2 opposite directions and fixing the other displacements to zeros, the strain field is expected to be 1 in any point. Three tests need to be performed; the first one being the opposite displacements along x (fig. 13a), the second along y (fig. 13b) and the third a pure shear displacement (fig. 13c). One can see that a strain field of 1 is indeed found in each case and conclude that the c-modes were also verified.
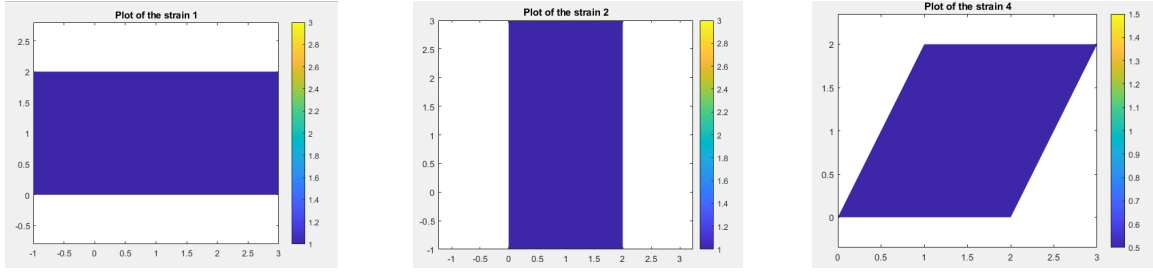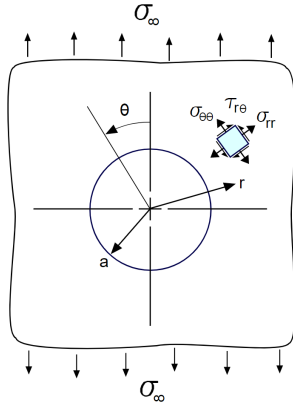
(a) Strain caused by opposite displacements along x

(b) Strain caused by opposite displacements along y

(c) Strain caused by a pure shear displacement

Figure 13: c-modes verification

7.3 Plate with a hole convergence study

The first thing to do for the convergence study is to find a possible closed form solution. As the tension applied is uniaxial and the plate is vast enough, a possible solution is the Kirsch's solution which leads to the following equations:

$$\sigma_r = \frac{\sigma_\infty}{2}\left(1 - \left(\frac{a}{r}\right)^2\right) + \frac{\sigma_\infty}{2}\left(1 - 4\left(\frac{a}{r}\right)^2 + 3\left(\frac{a}{r}\right)^4\right)\cos(2\theta)$$

$$\sigma_\theta = \frac{\sigma_\infty}{2}\left(1 + \left(\frac{a}{r}\right)^2\right) - \frac{\sigma_\infty}{2}\left(1 + 3\left(\frac{a}{r}\right)^4\right)\cos(2\theta)$$

$$\tau_{r\theta} = -\frac{\sigma_\infty}{2}\left(1 + 2\left(\frac{a}{r}\right)^2 - 3\left(\frac{a}{r}\right)^4\right)\sin(2\theta)$$

with a, the radius of the hole; σ_∞, the remote stress; r, the radial coordinate.

Young Module	Poisson	σ_∞	L	h	a	thickness
210 *GP*	0.3	88 N/mm^2	210*mm*	190*mm*	7*mm*	1*mm*

Figure 14: Plate with a hole Kirsch's solution [4]

At the hole, it means where $a = r$ and especially at $\theta = 90°$, one gets $\sigma_\theta = 3\sigma_\infty$, called the *Stress Concentration Factor*. One then knows that $\sigma_\infty = 264$ MPa. This latter value is not what one should get as the principal assumption is an infinite plate, which is not the case in reality. Nevertheless, it is a good approximation of what should appear on the plate. Also, the plate being plane symmetric, the study will be done with a quarter plate, making it easier.

The convergence study was done in *Salome-meca* and the results plotted in *MATLAB*. The first mesh used is a very coarse one. The h-refinement was performed in the corner where stresses were

15

the most significant. At each step, one can see in figure 15 that σ_∞ at $\theta = 90°$ converges around the expected value calculated earlier.

(a) Coarse mesh   (b) First refinement   (c) Second refinement
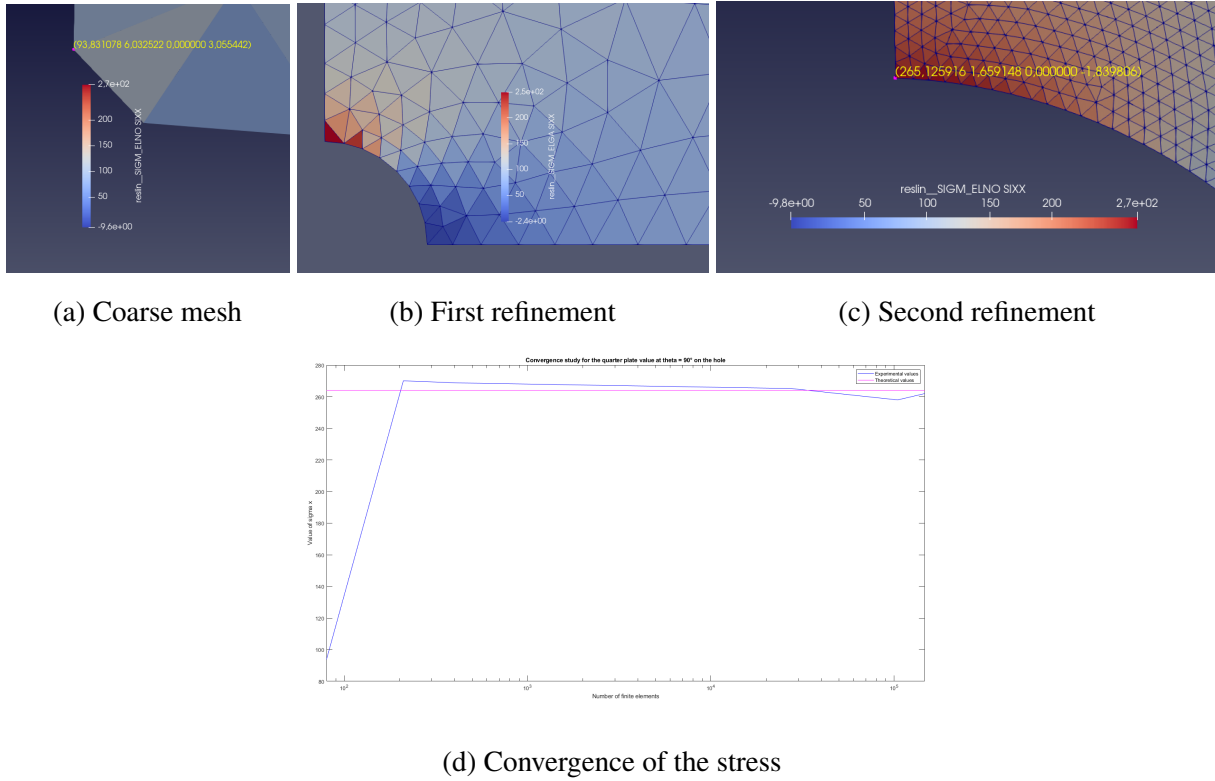

(d) Convergence of the stress

Figure 15: Convergence study

7.4 Comparison with the commercial code

To compare the two codes, the final mesh was preferred to get a clear view of the final results differences. One can see figure 16 that σ_x is quite similar from one plot to another. One difference is the peak value which is higher in *MATLAB*. Nevertheless, when one apply a zoom on the *MATLAB* plot, the means values are similar and the peaks are located at the inner profile. These peaks can be explained by the number of elements used and the non-converging boundary stresses as explained in section 2.2.
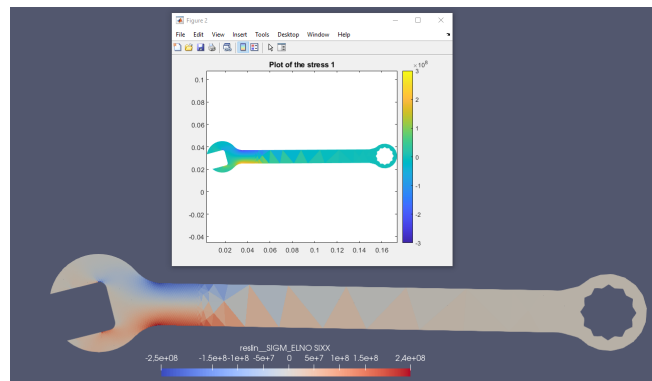

Figure 16: Comparison between *Salome-meca* and *MATLAB* stresses along x