Design of a torque-limiting spanner

Mecànica dels Medis Continus

Adrià Baró Biosca

Jorge Rodríguez Llorens

Ignacio Sánchez Zárate

Special thanks to AMARI METALS IBÉRICA, S.L.U.,
Alejandro Tallón and the laboratory staff for
contributing to the realization of this project, without
them the result would not have been the same.

Index

1. Introduction

The main objective of this project is to design a torque-limiting spanner using the finite element method (FEM). In this project, the spanner is intended to be used in industries where high quality bolts are not needed, such as low cost furnishing or DIY's. Nevertheless, it must comply with some important restrictions; minimum weight, high Young's modulus or the Spanish normative among others. In order to minimize weight, the form will have to be optimised and the most proper material must be chosen out of a wide range of them. Afterwards, according to the selection made, a prototype will be constructed, so that it can be tested in the last practice session.

To make the following parts more understandable, *Figure 1* illustrates the most important parts of a spanner:

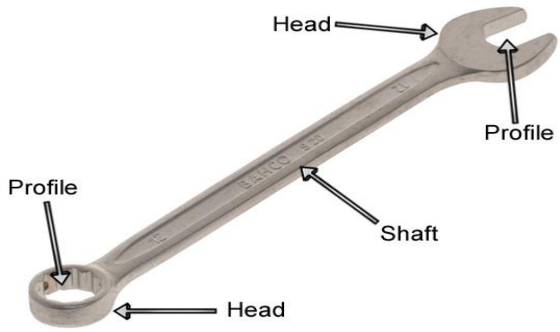


Figure 1. *Important parts of a spanner*

Finally, it is important to mention that the whole dissertation is going to be developed in English, because in our thoughts it is a great opportunity to practise a foreign language and to incorporate new, more technical vocabulary, which can be very useful in our future professional lives.

2. Investigation and preliminary design

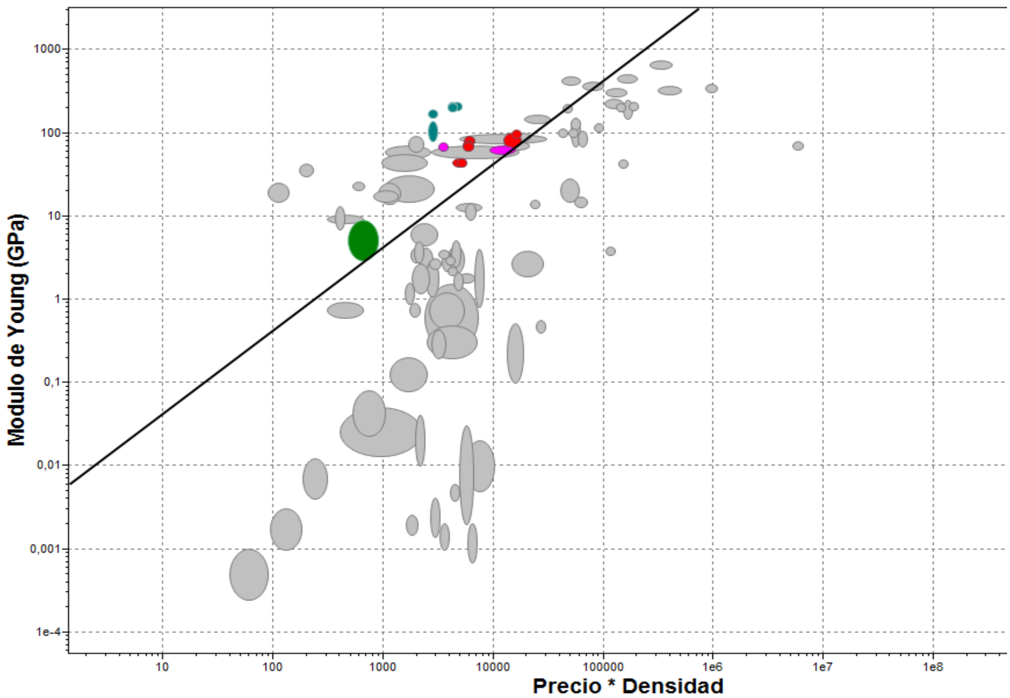
2.1 Basic function and regulations

A spanner is basically a steel hand tool with a handle carrying jaws or a hole of particular shape designed to provide grip and mechanical advantage in applying torque to turn objects, usually nuts or bolts. In our case the hexagonal nut metric is 14, so the spanners function is to be able to turn back the nut without deforming plastically. A safety factor must be designed, so that when nominal force is applied the material is still between 20%-30% away of reaching its elastic limit. Expressing this more mathematically: $1{,}2 \leq \gamma_{sec} \geq 1{,}3$. Being $\gamma_{sec} = \frac{M_{last}}{M_{nom}}$. This will ensure a proper performance of the spanner.

2.2 Material Selection

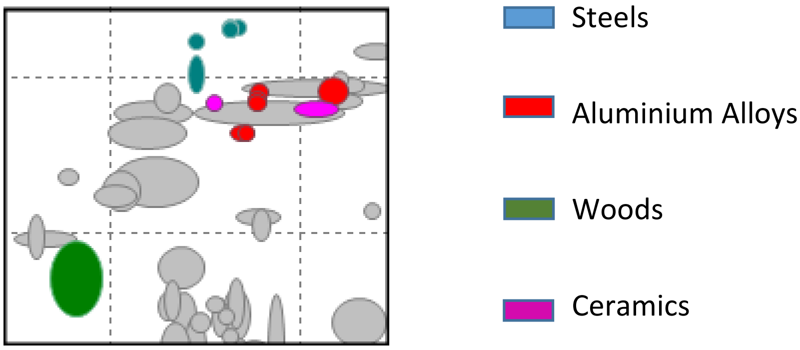
In order to choose the most appropriate material for the spanner, CES Edu Pack program will be used to start our search. This program allows the user to select between a wide range of materials, fixing a few solicitations. In our case, the material needs a high Young's Modulus, so that high stress can be applied without causing almost any strain. Another fact is that a cheap material is needed in order to accomplish the objective of spending a reasonable budget. This means that the factor $M = \frac{E}{price\ per\ volume}$ must be maximized. Applying logarithmic operations to both sides of the equation a straight-line equation is found: *log E = log price*volume + log M*, where log M works as a constant that can be adjusted by the user. The following graph is obtained, in logarithmic scale, where a line with slope=1 must be traced. In furtherance of maximizing the previous relation, the materials that lay over the line must be selected.

Before making any decision a new solicitation is going to be fixed: non-recyclable materials will be taken out from the possibilities showed on screen.



Graph 1. *CES EduPack graph: Young's modulus vs material price per volume unit*

As it can be seen, the coloured materials are the ones that maximize the chosen factor and accomplish with the recyclability solicitations. A lot of soft materials are discarded now.



Graph 2. *CES EduPack graph illustrating the final candidates*

After that, the candidates showed on screen are woods, ceramics (for example silicon carbide or aluminium nitride), aluminium alloys and a range of steels. Only isotropic materials will be chosen as it is easier to work with them. That means that woods are rejected from the candidates. In spite of the good mechanical properties of ceramics they are also discarded due to fabrication difficulties of the spanner.

Between all the candidates, only a few possible materials are left:

- Steels: low alloy steels, steels with low carbon content, grey cast iron and ductile cast iron
- Aluminium alloys

At the beginning, steel was elected as the first choice. But when the laboratory assay was executed, a high tensile elastic limit was obtained leading us to think that another material with less resistance should be chosen to make it easier to study its fracture. Then, an aluminium alloy is finally the best candidate for the purpose. After talking to some aluminum companies a piece of T651 aluminum alloy is the best option. No assay is needed due to a company report providing the most important mechanical properties. Some of these properties are showed in the chart below but the report is also available in the annex of this study.

Material	Young's Modulus (MPa)	Yield strength (MPa)	Density	Poisson's ratio
T651 Aluminium	72000	460	2,81 g/cm^3	0,33

Table 1. *Aluminium alloy mechanical properties*

2.3. Solicitations and characteristic values

In order to calculate the maximal force that the spanner must endure before it deforms plastically, firstly the nominal force has to be determined. To do so, each bolt has its own normalised tightening torque, which represents the momentum that the spanner has to apply to turn back the nut. The common bolt used nowadays is the DIN 931 ZN. According to the table of tightening torques that can be found in the annex, there are different tightening torques for each coefficient of friction (μ). In our case the bolt is zinc plated not only internally but also externally so μ lays between 0,12 and 0,20. For the nut μ=0,12 is chosen so the tightening torque is 48 Nm (Chart 3).

Figure 2. *DIN 931 ZN bolt*

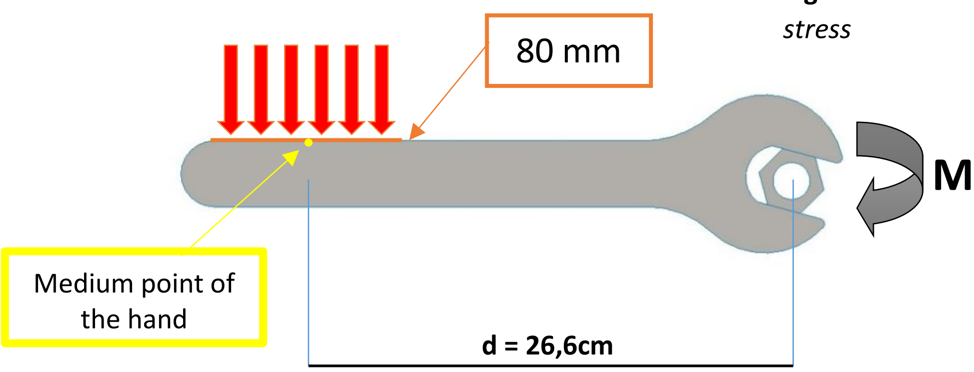
According to the kinetic momentum theorem applied on a fixed point C, the centre of the nut, the sum of all external moments equals 0:

$$\sum M_{ext}(C) = 0\,; Consequently\colon \sum F_{ext} \cdot d = 0$$

A media of the length of the user's hand is calculated as 80mm. That means that the surface of the hand in contact with the spanner is 80x10 mm^2. It is supposed that force is applied in the medium point of the contact surface with the hand of the user and the user holds the spanner as far as possible to minimize effort. Due to this, a nominal force is calculated as F = M/d = 48Nm/0,266m = 180 N. Nominal work stress can also be calculated. The pressure obtained below will be applied in the ANSYS program.

$$\sigma = \frac{F}{S} = \frac{180\ N}{800} = 0{,}225\ MPa$$

Figure 3. *Nominal force and stress*

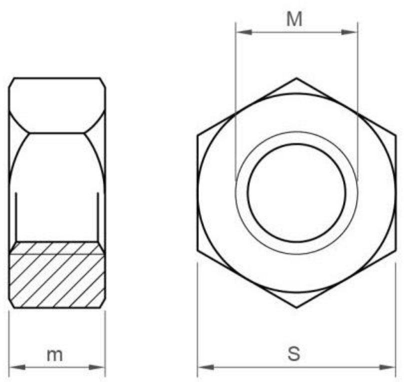


2.4. Preliminary design and prototype's construction technology

2.4.1. Preliminary design

For the design of our spanner SolidWorks' software will be utilised, that program allows to create solid models (CAD). However, before modelling the spanner using that software some simple sketches with the most relevant dimensions are drawn.

According to the metric given by the wording, the spanner needs to be used with 14M screws. So, the normalized dimensions of a 14M nut had to be investigated. The measurements of a 14M nut can be found if looking into some nuts' manufacturers, the relevant dimensions for the project are the distance between two parallel faces, the head's thickness of the nut and the length of one side.



m	*s*
11 mm	*22 mm*

Figure 4. *Normalized dimensions for 14M nut*

Keeping in mind previous dimensions, a wrench of 11 mm thickness and 22 mm profile is required. However, there are still a lot of unknown measurements, so looking for the common spanners in the market and combining some models; the final personal design can be properly developed.

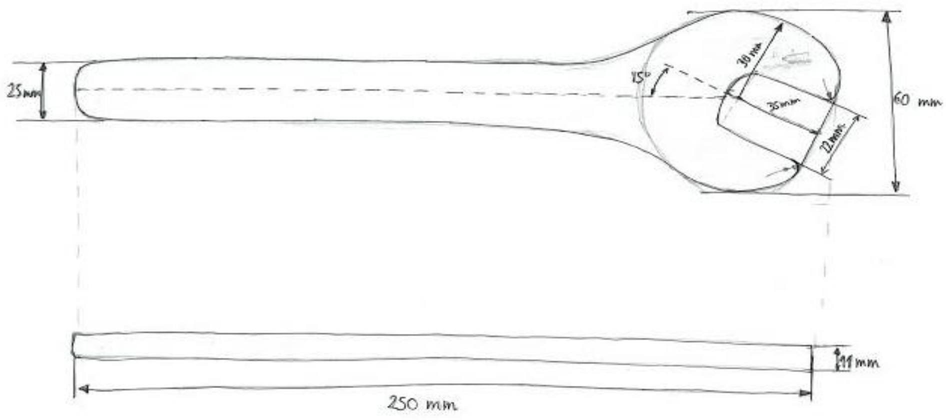


Figure 5. *Chosen sketch of our design, with the most important dimensions*

As it is shown in Figure 1, the head of the spanner presents 15º in relation to the shaft's handle, which provides more freedom of movement while working on highly confined spaces. The total length of the spanner is 25mm and the largest part of the head measures 60mm. Anyway, these measurements are not fixed yet so they will be modified to provide better specifications and properties.

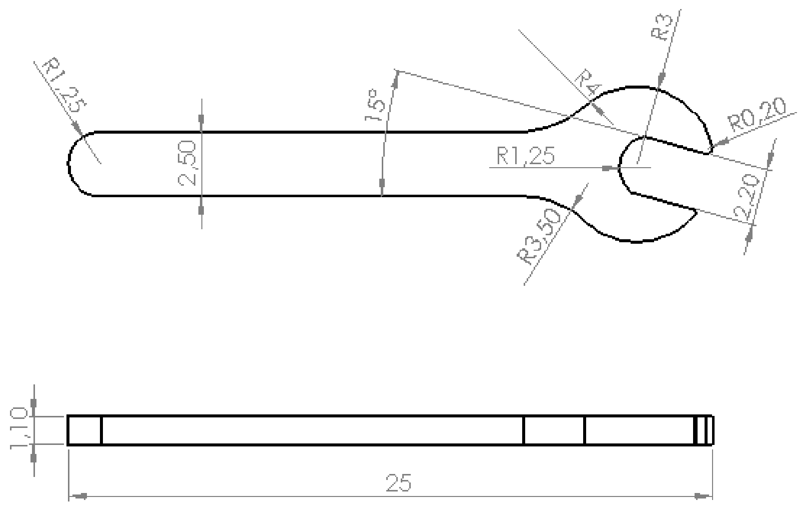


Figure 6. *Drawing of our spanner done with SolidWorks*

After that, the previous sketch will be modelled into the program. As it has been said before, some dimensions are not fixed or even specified so reasonable measures are going to be chosen and later be fixed accordingly.

2.4.2. Prototype's construction technology

As explained in previous sections, the material used for the prototype's construction is going to be aluminium. There are different methods for aluminium cutting, but the best ones to achieve a high accuracy and an excellent finish are the industrial cutting methods. However, these methods have an important inconvenient, the high cost of the machines used. After contacting several companies and professionals of the industrial cutting sector, water jet comes out to be the cleanest method and it works quite good while cutting aluminium. So, after opting for this method some previous companies were contacted and asked for an estimate price of the prototype.

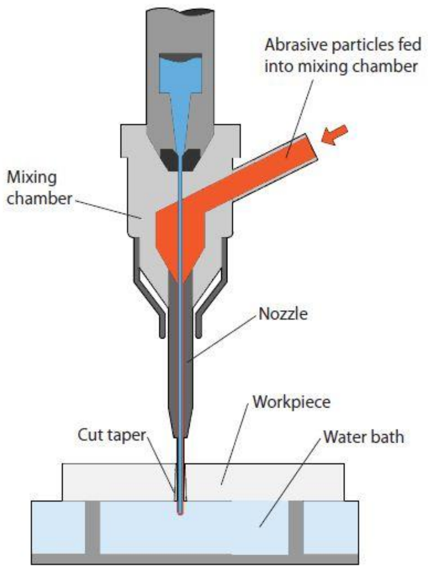


Figure 8. *Diagram of a water jet cutting machine*

Below, a short summary of how the water jet cutting works will be explained. The water jet focus a very thin stream of water mixed with abrasives at extraordinary pressures and very high speed into the sheet metal. This jet of water eats away at the metal and leaving a clean edge. Unlike the laser systems, the water jet doesn't melt the metal and is also very capable of cutting very thick sheets with high accuracy.

3. Finite element method

3.1. Model limitations and boundary conditions. Idealizations and numerical singularities.

The model is going to be analyzed and solved with ANSYS program, which has basically two resolution methods, p-method and h-method. P-method uses a fixed mesh and a polynomial of degree p to reach an approximate solution, the higher p the more precise the solution. On the other hand, h-method consists of adjusting the mesh size to obtain more precise results, which is more visually understandable and is therefore going to be used in this project.

Due to the constant thickness and symmetry of the spanner, the equality of the forces and restrictions along the z-axis and to reduce the computing costs, a flat shape with thickness has been considered. In terms of contour conditions, the following points have been fixed to avoid displacements of rigid body and indeterminate forms.

Points P and Q of *Figure 9* have been fixed. Point P has been fixed both x and y directions, because when force is applied the initial linear contact between the spanner and the nut becomes a punctual one in P and Q. However, to adjust the analysis to the real behavior, sliding in point Q is enabled, so only movement along y axis is restricted.

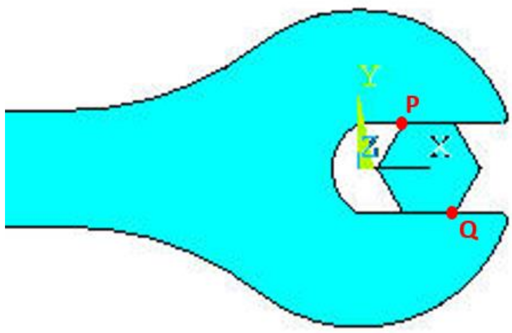


Figure 9. *Fixed points on ANSYS*

In fact, when applying the loads, punctual singularities will appear between the spanner and the nut in points P and Q. Nevertheless, according to the Saint-Venant's Principle, *"the difference between the effects of two different but statically equivalent loads becomes very small at sufficient large distance from application points"*.

In this case, the contact points are not expected to be in the failure region, so the previous idealization of the two contact points can be validated. This implies that in the stress analysis both points will present singularities since the area where the force is applied tends to zero. Inasmuch as both points are not representative the stresses next to these points will not be considered as real.

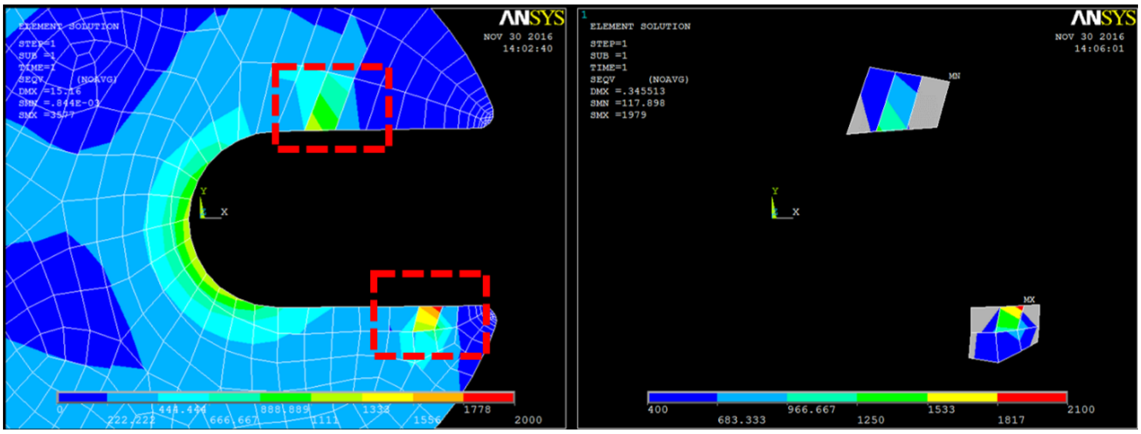


Figure 10. *Stresses next to the contact points*

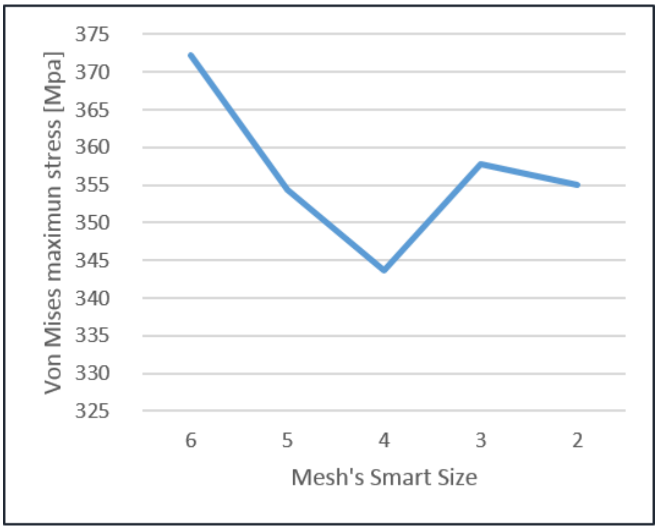
The element solution is studied because the nodal solution gives an arithmetic average of the result around, so the maximum stress is deleted and, therefore, important information gets lost.

3.2. Model characteristics finite element type, material models. Mesh.

It has been decided to mesh the area of the model with element Plane 183. Triangular elements eliminate a lot of information, so a square element has been chosen. Compared to the element plane 182, which is also squared, plane 183 has 8 nodes (4 on the vertexs+4 on center of the sides), whereas plane 182 only has 4 nodes. Because the geometry is very complex, with plane 182 information about parabolic displacements would be lost, and the solution would be less precise than with plane 183.

Regarding the material, its behavior has been defined as isotropic, linear and elastic and the values of the Young Modulus and the Poisson coefficient have been set at 72000 MPa and 0,33 respectively (more information about aluminum 7075 T-651 can be found in the annex).

Concerning the mesh size the analysis started with a smart size of 6. Then, an analysis was done in order to find the Von Mises maximum stress value. After that, mesh size was reduced to 5 and the same process was done. This analysis was carried out until the values of the maximum Von Mises stress were stabilized (meshing process at the annex). At this point, the meshing had enough precision to assure the reliability of the results.



Graph 3. *Von Mises maximum stress vs Mesh smart Size*

This graph shows that the stresses stabilize when arriving to mesh size 3 or 2. To reduce computing cost, a mesh's smart size of 3 is chosen.

3.3. Reliability of the results: critical points and mesh refinement

The refinement of the mesh is needed in the critical regions, where the highest concentrations of stresses can be found. In these areas the spanner would begin to deform plastically, therefore, these stresses need to be stabilized to achieve more accuracy. In the spanner this areas are the boundaries between the shaft and the head, refining this region ensures that the results converge to the real behavior which, consequently increases the precision of the calculations.

As it can be seen in *Figure 10*, the program has difficulties to calculate the correct stress in the critical region so a strange behavior is found is some elements.

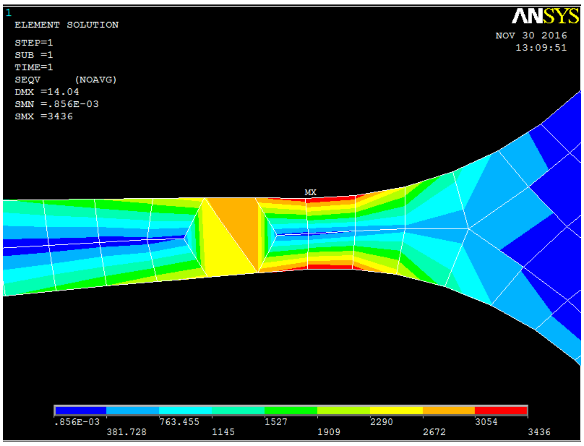


Figure 10. *Elements with no accurate results*

In order to achieve more accurate results a special meshing is done. An increase in precision is obtained with this refinement, which divides the critical zone in smaller elements. However, it raises the computation amount are carried out by the program.

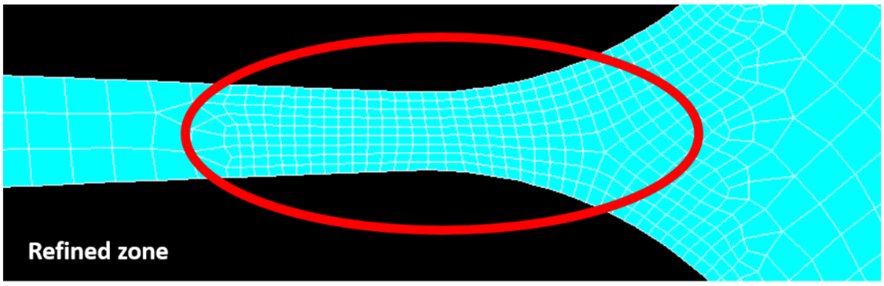


Figure 11. *Refined zone in our spanner*

As it can be seen in *figure 12*, when the solution is found with the refined mesh, stress concentrates in the elements on the edge of the spanner as it was expected. In the future, geometric changes in the shape of our spanner will facilitate its in this zone, when applying a determinate force.

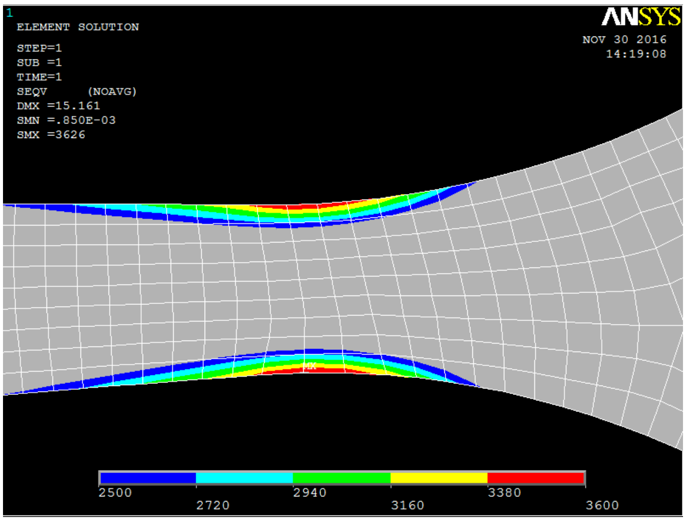


Figure 12. *Stress while applying the refined zone*

3.4. Validation of linearity hypothesis and smallness of displacements. Requirement of non-linear calculations?

A linear analysis has been carried out, since the Young’s Modulus is high and displacements are small enough to be considered infinitesimal.

The non-linear calculation considers the displacement of the point of application of the loads and consists of resolving it by an iterative method. Although knowing the Young's modulus it could be assumed that the linear hypothesis was valid, it has been certified with non-linear calculations, which show very similar results to those of linear analysis:

	Linear Calculation	Non-Linear calculation
Maximal displacement (mm)	1,519	1,509

The relative error is 0,6627%, so we can assume that the linear calculations are valid.

4. Results Analysis

4.1. Displacement fields. Maximal displacement under nominal force.

When applying the load, the piece deforms as it can be expected from a spanner of this characteristics. The head stays fixed, whereas the shaft displacements increase with the distance to the centre of the bolt, which can be observed in *Figure 13*. The maximum displacement is therefore achieved at the end of the shaft and is 1,519 mm.

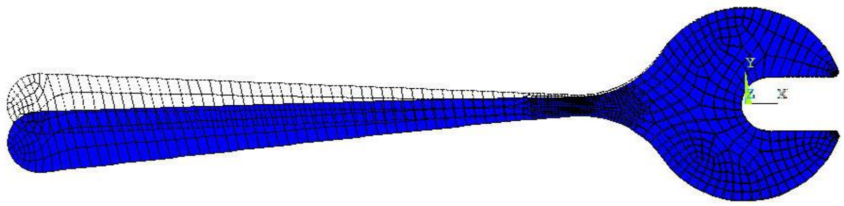


Figure 13. *Deformed shape on original shape.*

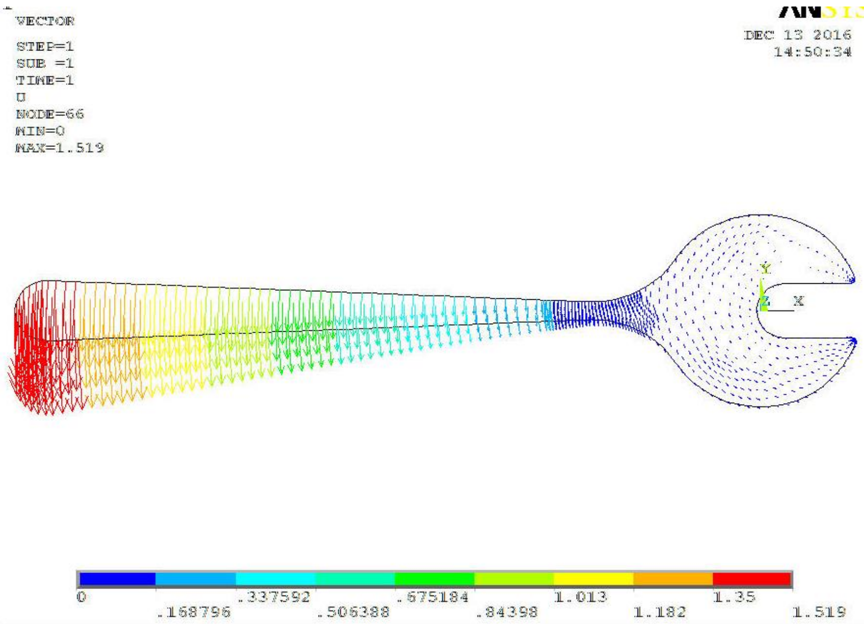


Figure 14. *Displacement fields*

Although it is a plane stress problem, displacements in the perpendicular direction of the piece do not have to be zero but this is not going to be treated. In this case deformations in this direction have been despised because they are quiet inferior to those produced in the working plane.

4.2. Stress distributions and principal axis

The following images show the principal axes stresses, arranged decreasingly ($\sigma_I > \sigma_{II} > \sigma_{III}$):

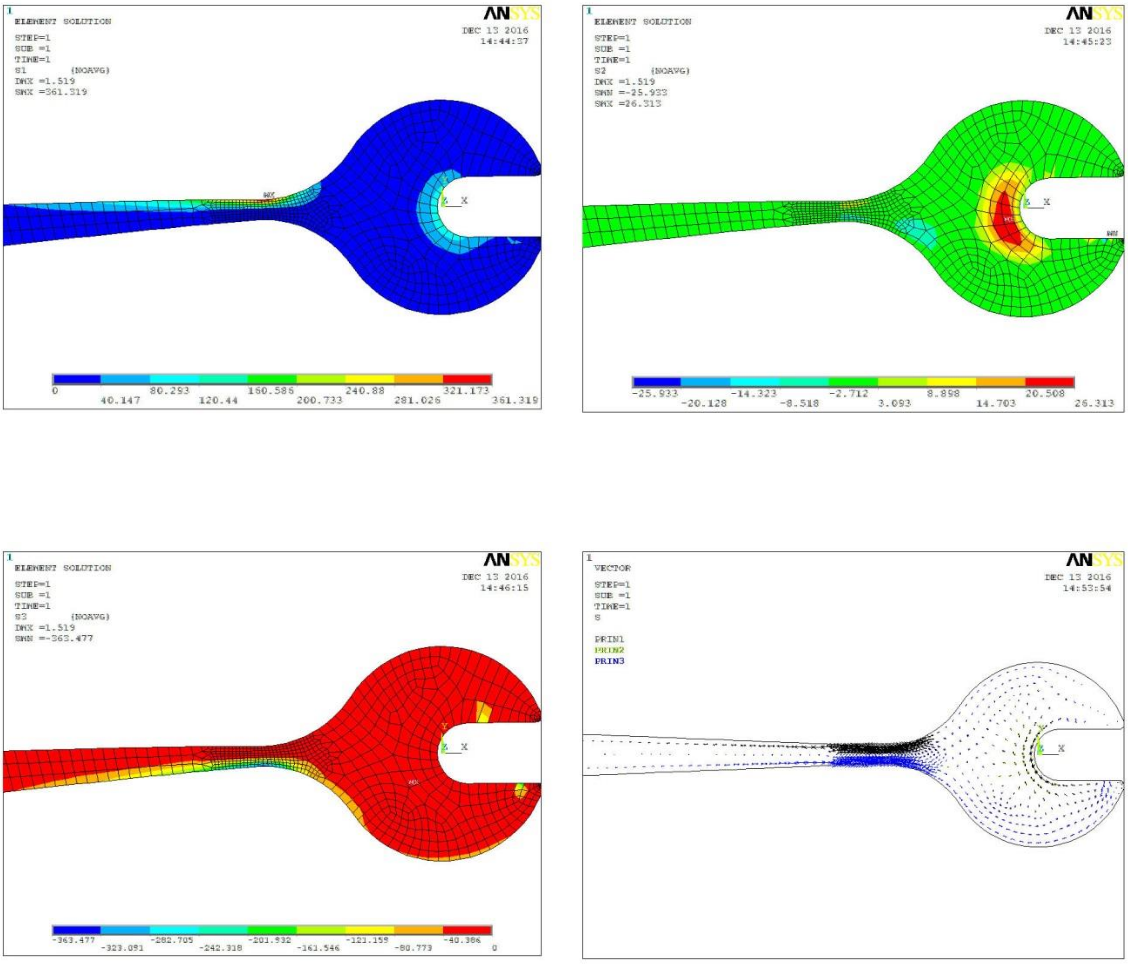


Figure 15. *Principal axes stresses*

Stresses in the first principal axis (σ_I) are related to traction, while those in the third (σ_{III}) are compressions. On the other hand, in the second principal axis stresses are rather low, but not equal to zero, whereas in the z-axis, they are indeed equal to zero (σ_z= 0). This phenomenon occurs because there is a specific stress ellipsoid for each element, and even though the second principal direction can in some elements coincide with z-axis, this is not a specific rule. As it can be observed in the top right figure, stresses (σ_{II}) in non-critical zones are in fact very close to zero, whereas in the centre of the head, for example, they are about 25MPa.

Finally, the fourth thumbnail illustrates where the greater stresses concentrations are located. Black colour shows the first principal axis, green the second and blue the third. It can be concluded that the upper part of the union between head and shaft works under traction, whilst the lower part works under compression.

4.3. Elastic failure criteria. Equivalent stress distributions.

One of the characteristic properties of aluminum is its ductility. For this reason, the Rankine theory that works for brittle materials cannot be utilized. Aluminum is a symmetric material, i.e. it has comparable values for both tensile and compressive yield. Both the criteria of maximum shear stress (Tresca-Guest) and the theory of Von Mises might be appropriate for the study of a material of this kind, but finally the Von Mises yield criterion for calculating equivalent stress associated with each point of the piece has been chosen, because it adjusts better to the real behavior of the material.

Von Mises theory pretends to predict failure by creeping seeing that is associated with the energy of change of form, but not of volume (since this is an elastic phenomenon essentially). The theory postulates that in state of stress, the material will fail when the distortion energy absorbed per unit volume is equal or greater than the limit associated with the failure of a uniaxial tensile test. Calculating equivalent stress from the energy of distortion, the following equation is obtained:

$$\sigma_{eq} = \sqrt{\frac{(\sigma_I - \sigma_{II})^2 - (\sigma_{II} - \sigma_{III})^2 - (\sigma_{III} - \sigma_I)^2}{2}}$$

where σ_i are the principal axes. This tension can be compared directly equivalent to the yield strength of aluminum.

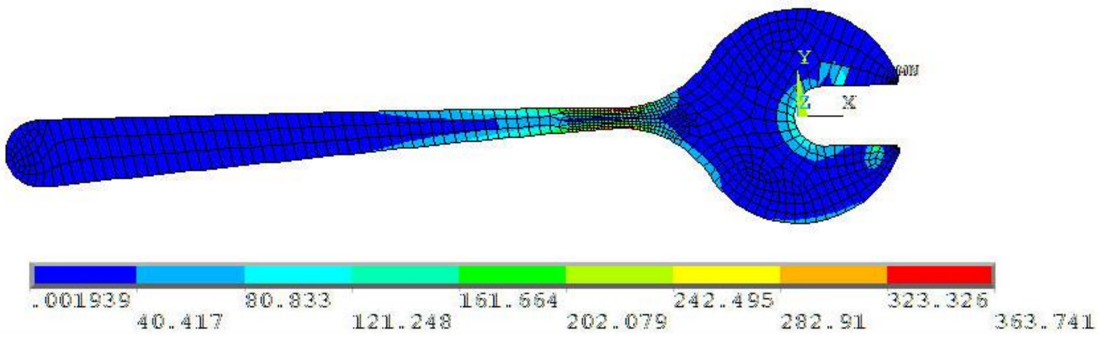


Figure 16. *Von Mises stresses*

4.4 Safety coefficient and failure load.

Due to project regulations, the safety factor has been set between 1.2 and 1.5. That is, the piece must withstand 1.2 times F_{nom} and should fail elastically before 1.5 times F_{nom}. Given that the yield stress is 460MPa, and that safety factor is the ratio between this and the maximum stress the spanner can tolerate:

$$\gamma_{sec} = \frac{\sigma_{el}}{\sigma_{max}}$$

It has been determined that the maximum stress the design can resist is among 353,8 and 383,3 MPa. These values have been used during the process of designing the piece to ensure that it meets the safety factor.

Once finished the corresponding calculations a maximum equivalent stress of 359,03 MPa has been obtained; i.e. the safety factor will be 1.281:

$$\gamma_{sec} = \frac{460}{359{,}03} = 1{,}281$$

Von Mises equivalent stresses have been considered for the calculation of the safety coefficient, because as being linear and isotropic it is equivalent to work with forces applied or with stresses affecting the material.

5. Design optimization, construction, test and conclusions

5.1. Process of design optimization

The spanner has suffered a continuous process of changes and modifications to create a piece with the desired specifications. The most relevant modifications that the spanner suffered, happened between design 1 and design 2: the head's angle has been suppressed because of the appearance of unwanted stresses, in unknown directions. Moreover, the length of the spanner has increased and the thickness of the shaft has been reduced. To sum up, all these changes have been carried out in order to avoid problems during the final analysis and to achieve a force that could be applied with our hands.

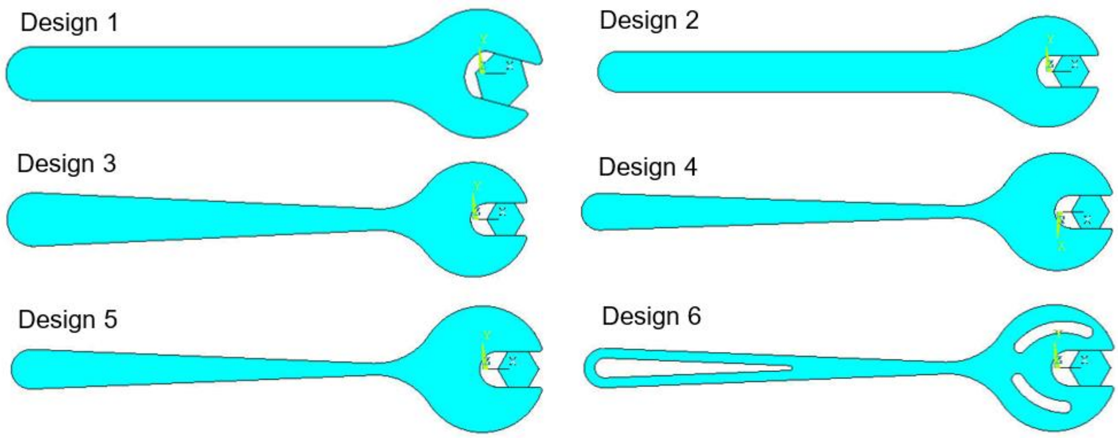


Figure 17. *Evolution of the spanner's design*

After the first process, and from design 2 to design 5, a cycle has been followed with the aim of finding a spanner with the final specifications; the cycle involves working with SolidWorks and ANSYS in a parallel way. The most outstanding modifications are the decreasing shape and the notable reduction of the spanner's shaft. These modifications have been done to concentrate the higher stresses into a wanted region so the spanner's failure could be controlled easily.

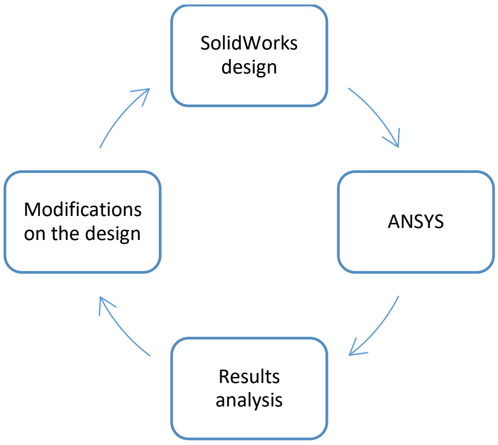


Figure 18. *Cycle followed to find the best design*

Once the design is found, the final step is optimizing it by reducing the weight without losing efficiency. So, an exhaustive analysis of stresses on the design 5 has been carried out to find the regions with smaller stresses. Finally, the regions with these smaller stresses have been emptied and another analysis with the final design (design 6) has been done to secure the non-appearance of new stresses there. The reduction of the weight has been of 20% compared with the design 6 and without meaning the loss of efficiency.

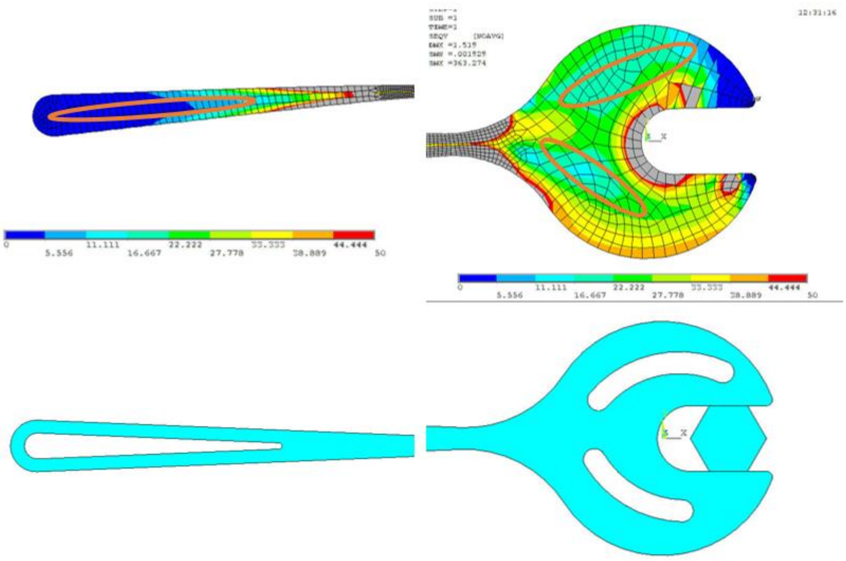


Figure 19. *Final optimization by reducing the weight*

5.2. Definitive results analysis

In this part, the whole finite element process followed is not explained because the working manner has been the same as the one in the section 3 (same mesh, boundary conditions, material model, loads applied …). As explained before, the Von Mises stress analysis has been done to make sure that the final optimization process has not affected the maximum stress or, if affected a little, the maximum stress still stays in the same region. The maximum obtained in this analysis is 359,03 MPa, whilst the maximum obtained before the optimization was 360,9 MPa. Consequently, the final safety parameter is $\gamma_{sec} = 1{,}281$ and therefore, the failure force to be applied will be $F_{nom} \cdot \gamma_{sec}$ = 180 · 1,281 = 230,6 N.

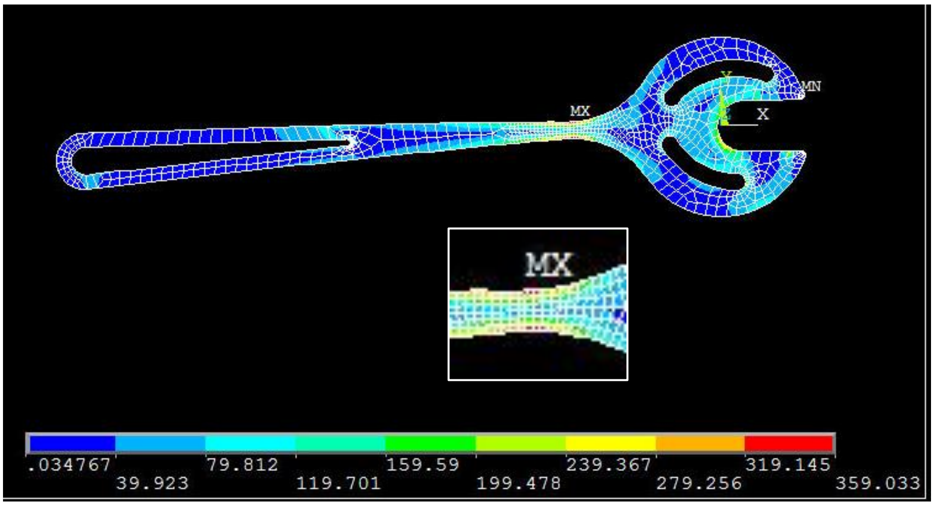


Figure 20. *Von Mises stress analysis*

Figure 20 shows the Von Mises stresses around the holes designed in the optimization process some stresses appear, but as being inferior than 120 MPa, they are not going to affect the final result.

Finally, a displacement analysis when applying 230,6 N has been carried out to make sure that the spanner's head is not going to open before the failure of the material. After seeing that this is not happening, the final design is being approved.

5.3. Definitive design, plan and constructive process

The final design, as it has been explained in previous parts, presents the most suitable shape to accomplish our goals; it concentrates the maximum stresses when and where it is wanted and it is also as light as possible.

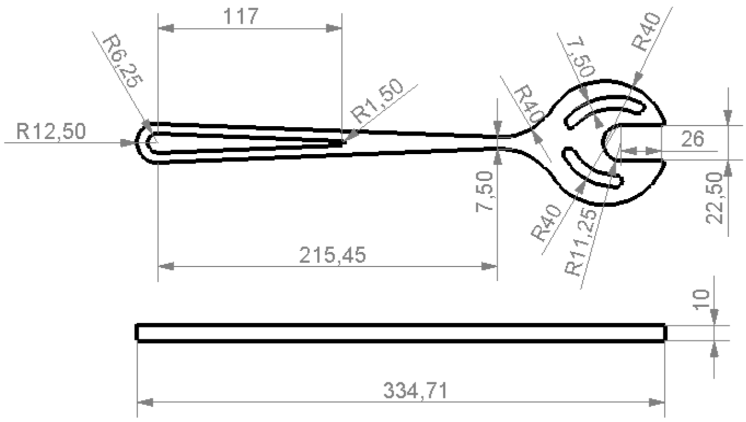


Figure 21. *Plan of the final design, in mm*

Figure 22. *Dihedral view of the final design, on SolidWorks*

The whole constructive process has been carried out in Molins de Rei (Barcelona) by AMARI METALS IBÉRICA, S.L.U. This company provided not only the aluminum plate of 10 mm but also the cutting of it using water jet cutting. As it was commented previously, this technique of cutting is perfect for our case and it is completely automatized, so after providing them with our CAD file the most laborious part of the process is done.

5.4. Budget, environmental impact and sustainability

The total budget spent on our spanner can be divided into the material's cost and the fabrication's cost. The fabrication of our spanner costs 55 €, although that cutting the material and the manpower supposes just 5€, getting the water jet cutting machine and entering the design into the computerized system cost 50€. In terms of the material costs, it can be considered almost 0 compared with the 55€ of the fabrication costs, buying a sheet of 100mmx340mm and 10 mm of thickness costs around 1€. Nevertheless, thanks to the kindness of AMARI METALS IBÉRICA, S.L.U., the final cost of the piece was 0€.

A lot of ecological concerns are observed in aluminum fabrication process. Obtaining the material from bauxite means lot of energy and residual waste thrown such as life-hazardous mugs or greenhouse effect gases. Killing the plants in the nearby is one of the effects of taking out from the earth the mineral and ground erosion could happen too. It also consumes huge amounts of energy because of the machines utilized on working on it (lasers or high industry equipment).

By the other way, aluminum is a recyclable material and there is a lot of stuff that can be done with it to make it sustainable without making it lose its properties. All aluminum products can be recycled just by wasting only a 10% of the energy that was needed in the first place to conform them. Less contamination is done too. For example, it is possible to build twenty aluminum recycled cans with the same energy needed to build a new one from the industry. At least, it can be said that our piece is sustainable if it is properly recycled.

5.5. Nominal and destructive test. Result correlation. Conclusions

A. Bibliography

Web: http://www.facom.com/uk/services/brochures/Brochures.html

Query date:15/11/2016

Web: http://ingemecanica.com/tutoriales/tornillos.html#seccion41

Query date: 15/11/2016

Web: https://www.upv.es/materiales/Fcm/Fcm11/ejercicios11_3.html

Query date: 17/11/2016

Web: http://www.chavesbao.com/#/product/2168-tuerca_hexagonal_sin_bano

Query date: 20/11/2016

Web: http://sheetmetal.me/tooling-terminology/how-to-cut-aluminum-sheet-metal/

Query date: 20/11/2016

Web: http://www.onlinemetals.com/

Query date: 30/11/2016

Web: http://www.enbuenasmanos.com/contaminacion-del-aluminio

Query date: 05/12/2016

B. Annex

Applicable Torque Value of Bolts , According to ISO898/1 , Tightening torque_N.m							
Thread / Tightening torque / Bolt grade	3.6	4.6	4.8	5.8	8.8	10.9	12.9
M1.6	0.05	0.065	0.086	0.11	0.17	0.24	0.29
M2	0.10	0.13	0.17	0.22	0.35	0.49	0.58
M2.2	0.13	0.17	0.23	0.29	0.46	0.64	0.77
M2.5	0.20	0.26	0.35	0.44	0.70	0.98	1.20
M3	0.35	0.46	0.61	0.77	1.20	1.70	2.10
M4	0.60	1.10	1.40	1.80	2.90	4.00	4.90
M5	0.60	2.20	2.95	3.60	5.70	8.10	9.70
M6	2.80	3.70	4.90	6.10	9.80	14.0	17.0
M8		8.90	10.50	15.0	24.0	33.0	40.0
M10		17.0	21.0	29.0	47.0	65.0	79.0
M12		30.0	36.0	51.0	81.0	114.0	136.0
M14		48	58	80	128	181	217
M16		74	88	123	197	277	333
M18		103	121	172	275	386	463
M20		144	170	240	385	541	649
M22		194	230	324	518	728	874
M24		249	295	416	665	935	1120
M27		360	435	600	961	1350	1620
M30		492	590	819	1310	1840	2210
M36		855	1030	1420	2280	3210	3850
M42		1360		2270	3640	5110	6140
M45		1690		2820	4510	6340	7610
M48		2040		3400	5450	7660	9190

Chart 2: *Tightening torques*

μ_G				Rosca	Rosca externa, tornillo								
Rosca				Material	Acero								
	Material			Recubrimiento	Óxido negro o fosfatado				Zincado (Zn6)		Cadmiado (Cd6)		Adhesivo
		Recubrim.		Fabricación	Conformado			Cortado	Conformado o cortado				
			Fabr.	Lubricac.	Seco	Engrasado	MoS_2	Engrasado	Seco	Engrasado	Seco	Engrasado	Seco
Rosca interna, tuerca	Acero	Ninguno	Cortado	Seco	0,12 a 0,18	0,10 a 0,16	0,08 a 0,12	0,10 a 0,16	–	0,10 a 0,18	–	0,08 a 0,14	0,16 a 0,25
		Zincado			0,10 a 0,16	–	–	–	0,12 a 0,20	0,10 a 0,18	–	–	0,14 a 0,25
		Cadmiado			0,08 a 0,14	–	–	–	–	–	0,12 a 0,16	0,12 a 0,14	–
	GG/GTS	Ninguno			–	0,10 a 0,18	–	0,10 a 0,18	–	0,10 a 0,18	–	0,08 a 0,16	–
	Al / Mg	Ninguno			–	0,08 a 0,20	–	–	–	–	–	–	–

Chart 3: *μ coeficients for torques*

ALUMINIO ZINC - EN AW-7075 / AlZnMgCu1.5

COMPOSICIÓN QUÍMICA (peso %)

Si	Fe	Cu	Mn	Mg	Cr	Zn	Ti
	max	1.2		2.1	0.18	5.1	max
0.4	0,5	2.0	0.3	2.9	0,28	6.1	0,20

PROPIEDADES FÍSICAS (valores nominales)

Densidad	2,81 g/cm^3
Módulo elástico	72000 MPa
Coeficiente de dilatación térmica lin. (20º-100ºC)	23,6 1/10^6 K
Conductividad térmica	134-175 W/mK
Intervalo de fusión	480-640 ºC

RADIOS DE PLEGADO

Estado\espesor	0.4<e<0.8mm	0.8<e<6mm	6<e<10mm
0	*0*	*1 a 2.5*	3.5
T6/T651	*4.5*	*5.5 a 8*	--

(Multiplicar el coeficiente por el espesor de la chapa)

RESISTENCIA MECÁNICA

Espesor (de... a)	Rm [MPa]	Rp0,2 [MPa]	A50 [%]	Dureza [HB]
0				
0.4 - 75mm	275	145	10-9	55
T6/T651				
0.4 – 300mm	525-360	460-220	6-1	157-104

(Norma EN 485-2)

CARACTERÍSTICAS Y APLICACIONES PRINCIPALES

La aleación de Aluminio 7075 que suministra AMARI, tienen una muy alta resistencia mecánica; resistencia media a la corrosión, buena aptitud para el forjado.

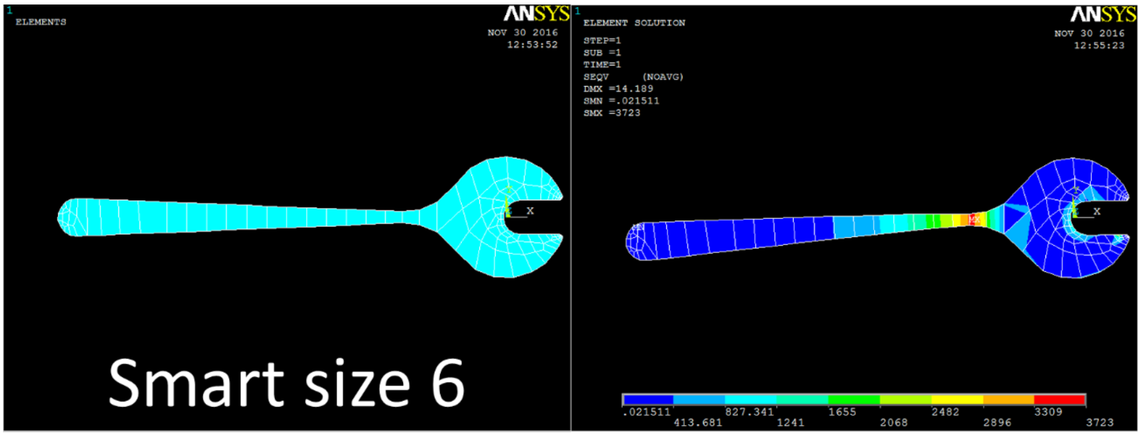
Por su elevada resistencia y apropiada mecanización, es una aleación adecuada para máquinas sometidas a elevadas tensiones mecánicas, moldes de soplado, matrices, herramientas, bicicletas y sus accesorios, industria del automóvil, piezas de avión, etc.

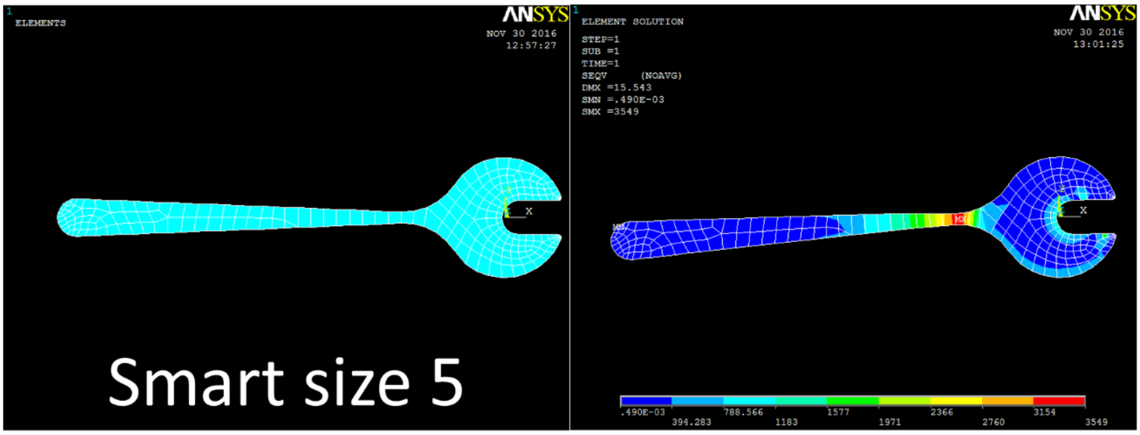
DOC 1: *Material report by AMARI*

Meshing process:

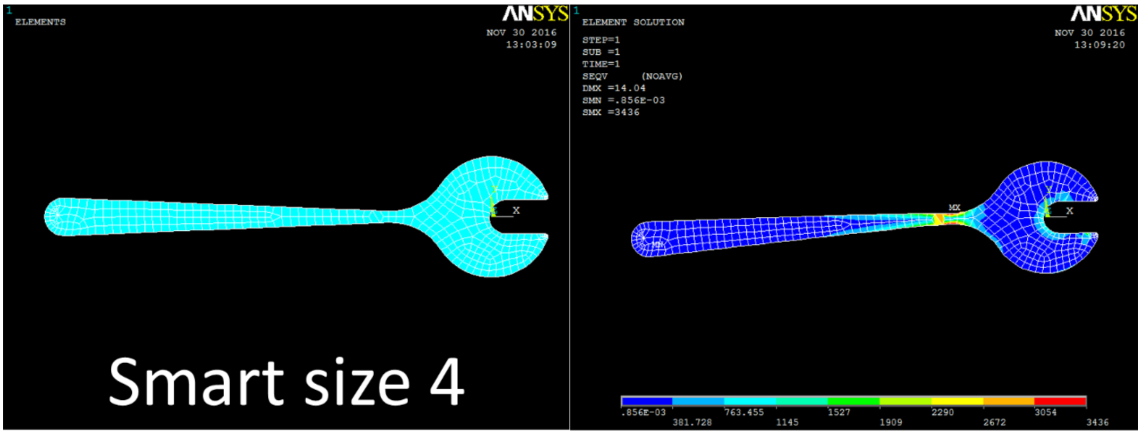
The meshing process can be seen in the following pictures where the images of the left show the meshing of the spanner and the ones in the right, the Von Mises stress solution. The smart size meshing tool was used in all the process.

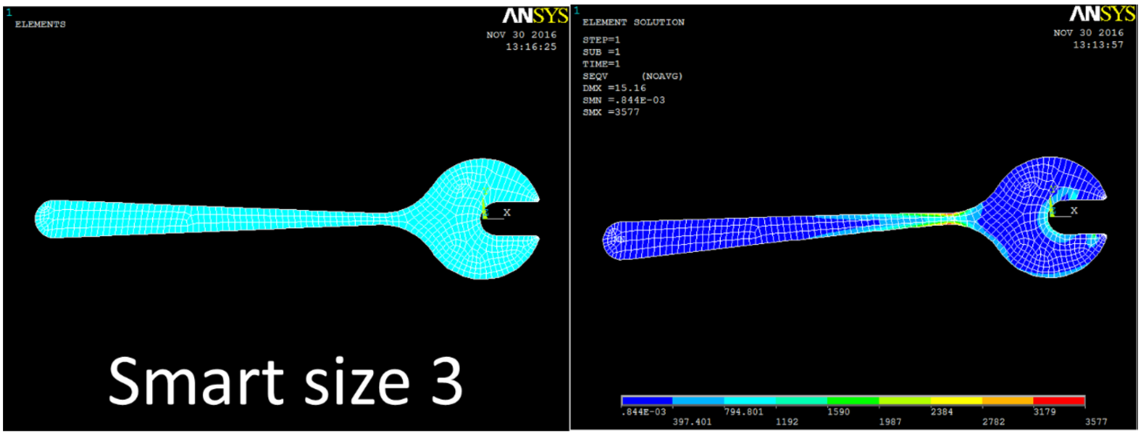
At the beginning, smart size 6 was chosen. The spanner was divided into elements which had a big size so the correct precision could not be reached as it can be seen in the picture below. The same thing happened with the 5 smart size. Taking a look at this meshing's pictures it can be observed that a big red element is showed on screen and the values of the maximum Von Mises stress are very different (a 5% of error is found between this meshes).

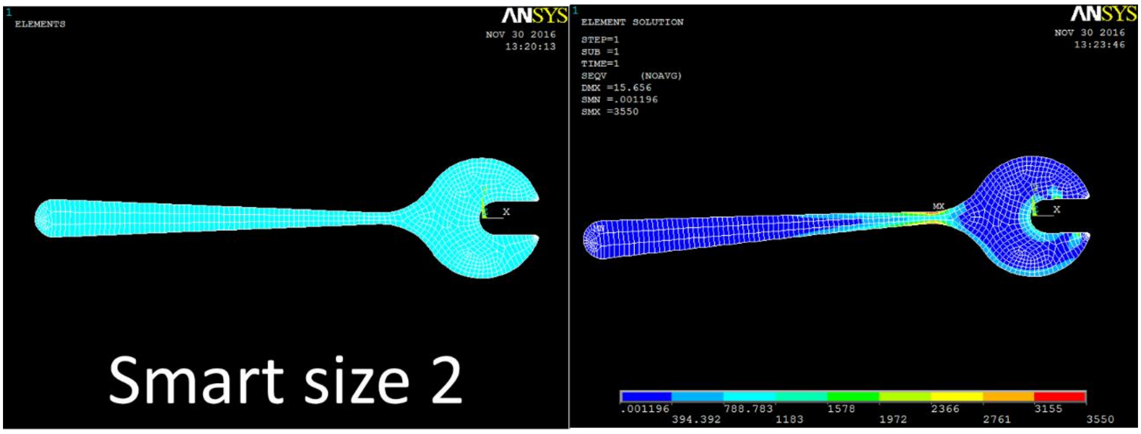




As it can be seen in the pictures below, when the smart size of the mesh is reduced, small elements are obtained in the meshing and the precision of the results increases. Stress slowly starts to concentrate at the edges of the spanner. And finally the maximum Von Mises stress is stabilized in the last two meshes (error is reduced to 1%). So it is proved that the results will be reliable when a two or three smart size of meshing is done. The 3 smart size will be chosen in this study to reduce the amount of calculations done by the computer.


ELEMENTS
ANSYS
NOV 30 2016
13:03:09
Smart size 4
ELEMENT SOLUTION
STEP=1
SUB =1
TIME=1
SEQV (NOAVG)
DMX =14.04
SMN =.856E-03
SMX =3436
ANSYS
NOV 30 2016
13:09:20
MX
.856E-03
381.728
763.455
1145
1527
1909
2290
2672
3054
3436


ELEMENTS
ANSYS
NOV 30 2016
13:16:25
Smart size 3
ELEMENT SOLUTION
STEP=1
SUB =1
TIME=1
SEQV (NOAVG)
DMX =15.16
SMN =.844E-03
SMX =3577
ANSYS
NOV 30 2016
13:13:57
.844E-03
397.401
794.801
1192
1590
1987
2384
2782
3179
3577


ELEMENTS
ANSYS
NOV 30 2016
13:20:13
Smart size 2
ELEMENT SOLUTION
STEP=1
SUB =1
TIME=1
SEQV (NOAVG)
DMX =15.656
SMN =.001196
SMX =3550
ANSYS
NOV 30 2016
13:23:46
MX
.001196
394.392
788.783
1183
1578
1972
2366
2761
3155
3550