DESIGN OF ACCESORIES FOR THE COPUPLING IN FORKLIFT TRUCKS: CRANE GIBS AND PALLET BOX LOCK

Ramon Miralbes

Hugo Malon

Luis Castejon

University of Zaragoza

Abstract

The aim of this paper is the development of a new design methodology based on the finite elements method applied to mechanical design projects, principally focused on accessories for forklift trucks accessories: pallet box lock and crane gibs.

So the developed methodology takes in care some aspects: mesh, load cases, material simulation, stiffness /strength design criterions, result analysis and application of some re-design methodologies: “try and error” and automatic computer design.

This methodology is applied in some existent pallet box lock and crane gibs (telescopic, tubular and closed box) designs. So it has been used numerical tools to analyze the structural safety of these elements and for their mechanical optimization. It has been tried to reduce the weight in the less critical zone and increasing the material quantity in other zones to obtain the weight and price optimized design.

It has been made some extensiometrical analysis in some of these elements with good results that allow the validation of the numerical methodology.

Keywords: *Crane gibs, FEM, Strength, Stiffness, Design.*

1. Introduction

This paper presents a new design methodology to design and optimize mechanical elements using finite elements. In the paper this methodology has been applied is some structural sets like crane gibs (closed box, tubular and closed box) and pallet box locks. Then it has taken in case some different criterions, like the strength, the stiffness, the weigh, regulations, manufacturing, etc.

Then, using only computational resources, the structural designs can be analyzed and redesigned without the need to build a real prototype; this imply a important economy and time save, and all the individual parts can be analyzed and redesigned separately.

Furthermore, the most unfavorable zones and the less uncharged zones can be obtained to optimize the material use and obtain an optimal design.

In this methodology there are established some different criterions to simulate the numerical behavior using finite elements; the materials, load, boundary conditions, load cases, the numerical simulation of the parts, the contacts, the simulation of the nonstructural parts, the welds, etc. are defined.

On the other hand, it has been analyzed the regulation for each machine and then the strength ad and the stiffness criterions are defined to establish the minimum reference value to accept the design.

It has been analyzed too and compared two different optimization methodologies: try and train and automatic numerical optimization.

At the end, the numerical results are compared with the experimental ones for some mechanical elements, using extensiometric gauges, to validate the numerical results and to obtain the numerical-experimental correlation.

2. Regulation

The regulation must be taken in care for the design of all type of machines. For these ones, the reference regulation is the UNE-EN 1726-1/1M: 2004 “Seguridad de las carretillas de manutención. Carretillas autopropulsadas de capacidad hasta 10000 Kg inclusive y tractores industriales con un esfuerzo de tracción al gancho hasta 20000 N inclusive.”

This regulation is adequate for the design of the pallet box locks but results insufficient for the design of the crane gibs because it is not so restrictive, so, like other author made (Miralbes, Castejon, 2008; Sun, Kleebager, 2003) it must be used another one: the UNE-58536: “Reglas para el cálculo de las estructuras de las grúas móviles de uso general”.

2.1. UNE-EN- 1726-1/1M

Regulation UNE-EN 1726-1/1M has been used for the design of the pallet box locks and for the fork lifts truck. This regulation is not excessively restrictive but it is adequate for these types of mechanical sets.

The part of the regulation that is used in this paper is:

“The structural components of the fork lift truck and their accessories must support a static load case during 15 minutes of 1.33Q1 and a 1.33Q2, , where:

- Q1 is the reference load at the standard elevation height and with the standard distance of the load center, like it is indicated in the capacity sheet.
- Q2 is the real maximum load at the maximum height like appears in the capacity sheet.

After and during the test, it must not appear any damage and any permanent deformation of the structure.”

This regulation takes only in consideration the useful transportable load and the own weight of all the parts of the structure, so the loads that must be included during the analysis are the own weight and the maximum load with a 1.33 scale factor. (In our case with a maximum load of 1000 Kg, it must be 1300 Kg).

2.2. UNE-EN- 58356

Regulation UNE-58356 establish some different load cases (see Table 1 of the regulation) and they are obtained combining some different load with a scale factor coefficient (table 5 of the regulation):

Main load:

- Own weigh (G)
- Service Load (F): is the weight of the useful load and the weight of the accessories to lift it (F0: pulleys, hook, cable, …).
- Dynamic effect that appears during the raising and descent (Φ). This scale factor is obtained with this equation:

$$\Phi = 1.1 + 0.13 \cdot V_h (m/s) \qquad (1)$$

- for Vh <1.5 m/s and for the rest Φ= 1.3

  Where Vh is the maximum elevation velocity for each element (cable, useful load, …)

- Forces due to the inertia effect of the crane gib: translation (T), rotation (S) and longitudinal displacement (L). These forces are apply separately with the own weight and with the service load.

Additional loads:

Wind loads in it most unfavorable position, during the service (Wi), obtained with the regulation UNE 53113 and out of service that specify the builder.

Special loads:

Static load case of the UNE 58-501 regulation (point 11.6). It is a rollover regulation which specifies that the structure must support 1.25 times the nominal load..

Then using the "partial security coefficients and limit stress" method, the load cases are this, using the scale factors of the table 5 of the regulation:

Standard load cases

- Case 1: standard load case without wind.

$$1.2 \cdot G + 1.35 \cdot \phi \cdot F + 1.5 \cdot T / S / L \qquad (2)$$

So it must be calculated separately with rotation, translation and longitudinal translation.

- Case: standard load case with wind

$$1.09 \cdot G + 1.2 \cdot \phi \cdot F + 1.35 \cdot T / S / L + 1.2 \cdot W_i \qquad (3)$$

Especial load cases

- Case 3: Out of service with wind

$$1.09 \cdot G + 1.2 \cdot F_0 + 1.35 \cdot T / S / L + 1.9 \cdot W_0 \qquad (4)$$

- Case 4: during the assembly with wind: this load case will not appear anytime for these structures.
- Case 5: static load case.

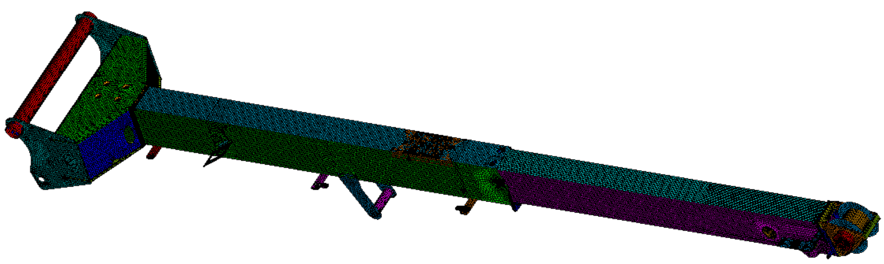
$$1.09 \cdot G + 1.09 \cdot F_0 + 1.09 \cdot 1.25 \cdot (F - F_0) \qquad (5)$$

Analyzing these load case it can be pointed that the UNE-1726 regulation load cases are the most unfavorable load cases.

3. Finite elements methodology

The finite elements method (F.E.M.) has some advantages for the analysis and the measurement of the elements that has the highest stresses and allows to modify easily the thickness and the materials, and it can be obtained the stress and the displacement maps(Z Zienkiewicz, Taylor, 2006). The tube and the closed box structures and the pallet box lock are simpler, so it will be analyzed the used methodology for the telescopic crane gib, and their principles can be used easily to the other structures (see Fig. 1). For these studies it has been used the FEM commercial program ABAQUS:

Figure 1: Finite elements model for the telescopic crane gib



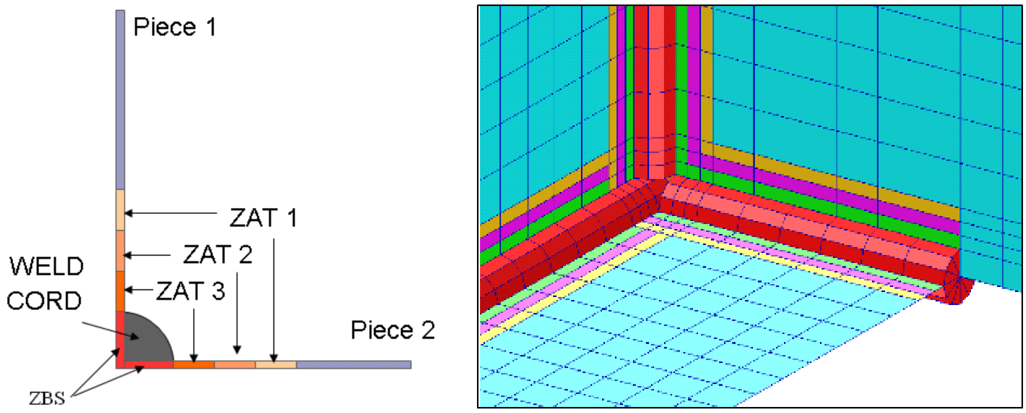
3.1. Materials

To model the material it has been used the stress-strain curve for each type of material and it has been supposed that we are always in the elastic zone, so the variables to introduce are the Young module, the density, the elastic coefficient and the poisson coefficient.

3.2. Weld models

Finite elements method allows simulating the welding (see figures 2 and 3). Like it can be observed, there are some different zones: the heat affected zone (HAZ), the under weld zone (UWZ) and the weld bead, and they must be all simulated. The properties of these zones can be obtained with a harness test, but usually the properties are better than the initial material, but they have worse fatigue behavior. Then, it can be used the initial elastic limit.

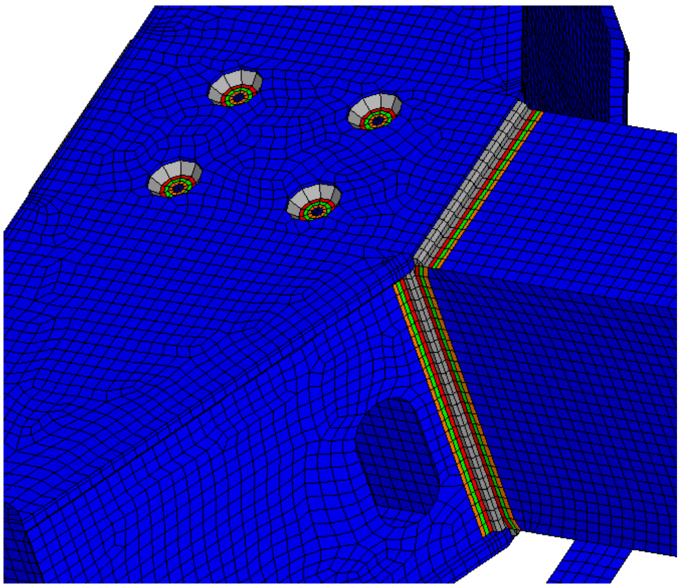
Fig. 2 y 3: Weld with HAZ and UWZ



Then if it is not necessary to do a fatigue analysis, the zones near the Weld cannot be simulated and it is only necessary to simulate the weld cordon with volumetric elements.

On the other hand, the shape of the welds depends on their type. In the case of a 90º weld it modellization is like the one that appear in the figure 4.

Fig. 4: 90º welding



3.3. Hydraulic cylinder

Telescopic crane jibs have inside a hydraulic cylinder that provides the longitudinal displacement. This element in their initial position and in their final position allows only to support longitudinal efforts, so to model it, it has been used a rigid beam with two spherical joints at the ends.

3.4. Non structural elements simulation

Usually crane gibs have some nonstructural parts that have a weight and it must be included in the model: hydraulic cylinder, valves, oil, etc.

These masses appear sometimes off-centered so they generate a torsion moment. To model the mass and the torsion, they are substituted to some forces and moments located at the joint zones.

4. Strength and stiffness criterions

To post-process the results it is necessary to establish some Strength and stiffness criterions and, using regulation UNE 1726, they are:

Combined limit stress (using Von Misses criterions):

$$\sigma_{V.M,\max} \leq \frac{Fluence\_\lim it}{1.11} \tag{6}$$

Buckling analysis:

$$\sigma_{compresion,\max} < 0.9 \cdot Bucklingl\_\lim it \tag{7}$$

Shear stress limit:

$$\tau_{\max} < \frac{Fluente\_\lim it}{1.92} \tag{8}$$

So these elements can be simulated using diverse structural and material resistance methodologies.

About the welding analysis, the UNE-58356 regulation allows calculating it, and it must be made like a zone without welds, but the security coefficient used is different and appears in table 1. In this case there are 4 different criterions to fulfill:

Table 1: security coefficient for the welds

Combined limit stress	Traction stress limit			Compression stress limit		Shear stress limit	
	Butt weld without preparation	Butt weld with preparation	Weld cord	Butt Weld	Weld cord	Butt Weld	Weld cord
1,11	1,11	1,25	1,57	1,11	1,39	1,57	1,92

5. Estimation of loads due to the various movements for crane jibs

The forces due to the inertia of the crane jibs are more difficult to calculate because each vehicle depends on the brakes installed, the hydraulic system, etc. (Miralbes, Castejon, 2008).

For this reason, a test with accelerometers is necessary to obtain the correct quantification of the forces due to inertia of the crane jib. The problem is that the carrying out of these trials is not always possible. The solution implemented is to adapt the equation of dynamic factors starting on the recommended maximum velocities in the European standard of cranes (UNE-58-507-77). The velocities established by the standard are shown in table 2

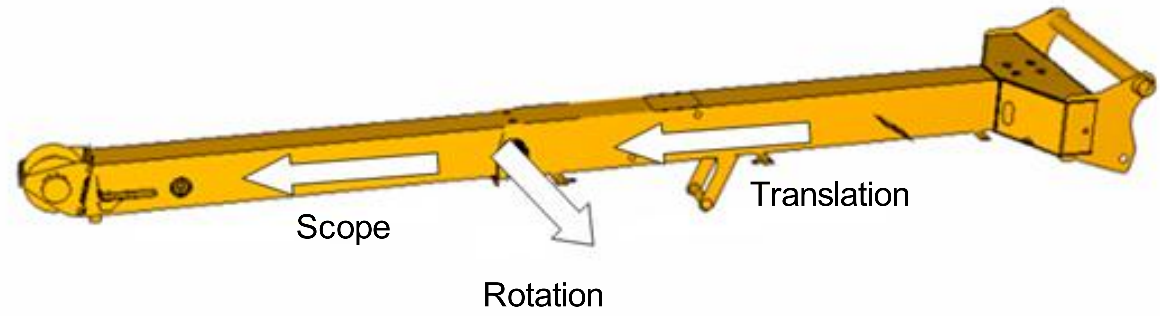
Table 2: Maximum admissible velocities for a crane

Maximum charge		Less than 10Tn	Between 10 and 15 Tn	Between 15 and 20 Tn	Between 20 and 30 Tn	Between 30 and 40 Tn	More than 40Tn
Translation velocity (m/s)		2.78	2.36	1.94	1.39	1.25	0.97
Rotation Velocity (rad/s)		0.37	0.31	0.25	0.25	0.21	0.21
Maximum Velocity in the end with a range of 16m		5.91	5.02	4.02	4.02	3.35	3.36
Maximum Charge	Less than 2.5Tn	Between 2.5 and 4 Tn	Between 4 and 6 Tn	Between 6 and 10 Tn	Between 10 and 15 Tn	Between 15 and 25 Tn	More than 40Tn
Range velocity (m/s)	0.62	0.45	0.31	0.23	0.15	0.10	0.08

After that, each movement is described separately:

- Translation: This movement is due to displacement of the forklift truck. In this movement, the maximum deceleration is produced in the stopping process. Numerical calculations are carried out by introducing acceleration in the longitudinal direction of the crane jib in both directions. For the realization of these calculations is necessary to know the maximum deceleration. This deceleration is calculated from the minimum stopping time, which is unknown. Thus a 1 second stopping time is estimated. Then, the maximum deceleration would be approximately of 2.8m/s2 that is a reasonable value.

Figure 5: Movements in a telescopic crane jib



- Rotation: This movement is produced in gyratory fork-lift truck when it tends to turn. The maximum speed is obtained at the end of the crane jib, when the machine has the maximum scope. In this case there are two types of tangential accelerations. First, the tangential acceleration due to centrifugal force, and second, the tangential acceleration which might arise from the acceleration or deceleration needed to achieve the rotation speed or to stop the crane jib, being the latter case in which the deceleration is higher. So:

$$a_{centrif} = \frac{v_{tra}^2}{r} \tag{9}$$

In the worst case the centrifugal acceleration can be 2.2m/s^2, although this value depends on the distance of each zone

$$a_{acel\_fren,\max} = \frac{v_{rot,\max}}{t_{stopping}} - a_{centrif} = \frac{\Omega_{rot} \cdot r_{range,\max}}{t_{stopping}} - \Omega_{rot}^2 \cdot r_{range,\max}$$

$$a_{acel\_rot,\max} = \frac{v_{rot,\max}}{t_{stopping}} \tag{10}$$

A maximum rotational deceleration of 3 m/s2 is obtained starting of a braking time of 2 seconds.

- Scope: This movement is due to displacement of the telescopic crane jib to be extended or collected. The speeds of this movement are very small, so that the decelerations or accelerations will be negligible compared to the rest of accelerations and the gravity.

6. Wind loads

Wind loads are calculated according to the standard UNE 53-113, which states that the force acting on the face on which the wind acts in normal conditions of use (wi) is:

$$P_{wind,ele} = P \cdot C_{f,element} \tag{11}$$

- P_{wind} is the wind pressure. In this study the wind pressure is 125 N/m2 (table 1 of the standard UNE 53-113)

- $Cf_{,element}$ is a parameter that depends on the shape of the element on which the air acts. In the cases analysed in this study all areas of exposure are closed box crane jibs, except for the lattices. The values used in the study were obtained from the table 2 of the standard UNE 53-113.

Wind load is applied evenly on the lateral surface of the jib in which the wind acts. The direction of this force is transverse to the crane jib.

These crane jib are not designed to work with storm winds, because with these machines is very dangerous to do manoeuvres in these environmental conditions. If necessary, it is calculated using a P value of 800 N/m2 for heights minor than 20m or of 1100N/m2 for heights between 20 and 100m.

7. Application to a telescopic crane jib

The application of the exposed loads is almost direct in telescopic crane jibs. The only drawback is the simulation of the friction pads.

Friction pads are used to allow relative movement of the boxes of the telescopic crane jibs, as well as to transmit the loads among themselves. These elements are attached to one of the two box beam of the telescopic crane jib. Its function is to prevent excessive friction between the two box beams that can damage the crane jib.

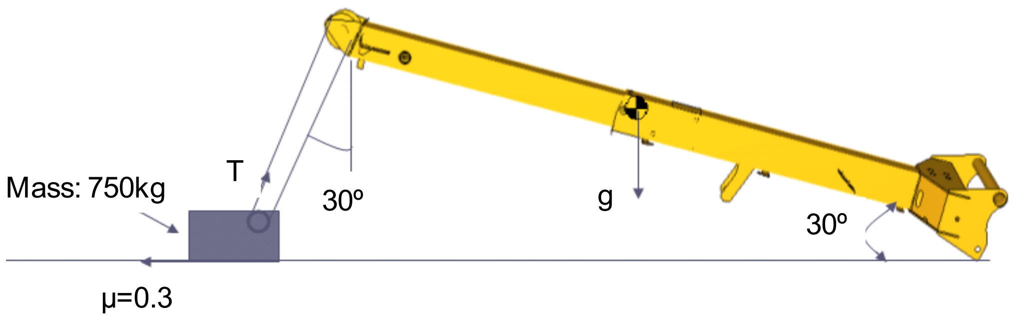
For this reason, these components are required in the model to achieve a proper sizing of the crane jib. The discretization of the friction pads allow to simulate the solicitations existing in a real manoeuvre in the box beams of the crane jib, as well as to obtain accurate results of stresses and strains

In the finite elements models, these components are discretized with volumetric elements. The bases of these pads are attached to the corresponding box beam. With respect to the other box beam, some frictional contacts have been established. These contacts allow obtaining a correct behaviour of the model.

For the particular case of a telescopic crane jib, due to its conditions of use, three additional load cases have been included in the analysis. These are:

- Additional Load case 1: Hypothesis of the load case 1, with an inclination angle of 30° in the telescopic crane jib and dragging the maximum capacity of load.

Fig. 6: Additional load case 1 for telescopic crane jib



This load case should not occur over the life-span of the crane jib, because this machine is not being originally designed to operate in this way. However, this additional load case has been introduced as a calculation of safety against this type of drag manoeuvre, as the following load case.

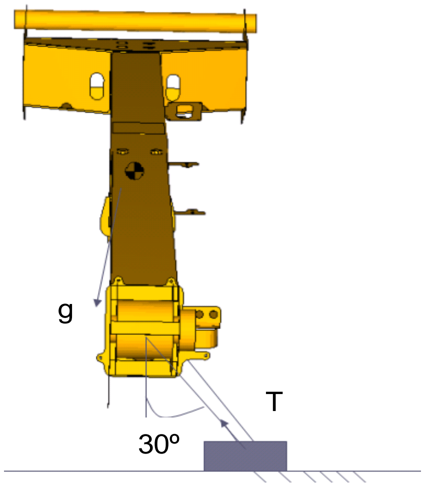
To make this calculation, the directions of gravity and applied forces have been modified in the finite element model. The value of the service load (P) is replaced by the value of T. The value of T is obtained from the following expression:

$$T = \frac{F \cdot \mu}{\cos(\alpha)} \tag{12}$$

where μ is the friction coefficient of the block with the land, which is considered 0.3, and α is the angle of the crane jib to the horizontal.

- Additional load case 2: Hypothesis of the load case 1, dragging laterally the maximum capacity of load with a 30º angle.

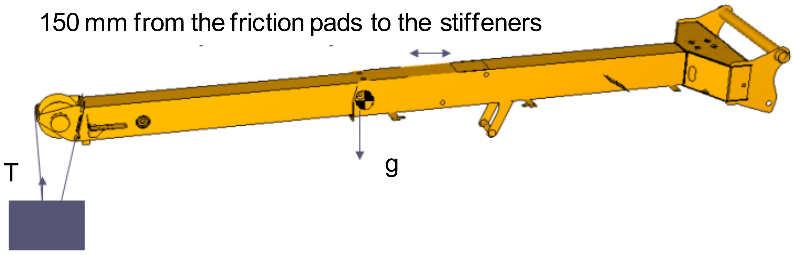
Fig. 7: Additional load case 2 for telescopic crane jib



This case is calculated similarly to load case 1. The only differences are variation of the application angle of the service force (F) and the value of this force, which is replaced by the tension value (T). This value is obtained analogously to the additional load case 1.

- Additional load case 3: Load case 1in middle scope.

Fig. 8: Additional load case 3 for telescopic crane jib



The telescopic crane jibs are designed to work only in two positions: totally spread and totally retract. But as in previous cases, an additional load case has been included as a security measure.

Note that these crane jibs have local stiffeners in the contact areas for maximum scope. These stiffeners are bears or over thickness. Therefore, in an intermediate load case of displacement of the box beam crane jib, these elements probably do not work. As a result, local stress concentrations can be generated, which must be quantified. Therefore, in this load case, an intermediate position of a box beam relative to the other box beam has been defined, but closer to the position of maximum scope of the crane jib;

Once the additional load cases, materials and contacts have been defined, the next step is to carry out the numerical calculation by means of the Finite Elements Method of the additional load cases. In the post-processing, Von Mises stresses and shear stresses must be analysed in every component of the crane jib, as has been specified in the section 4 of the article. Figure 9 and 10 show some of the overall results obtained

Fig. 9: Von Mises stress in the additional load case 1

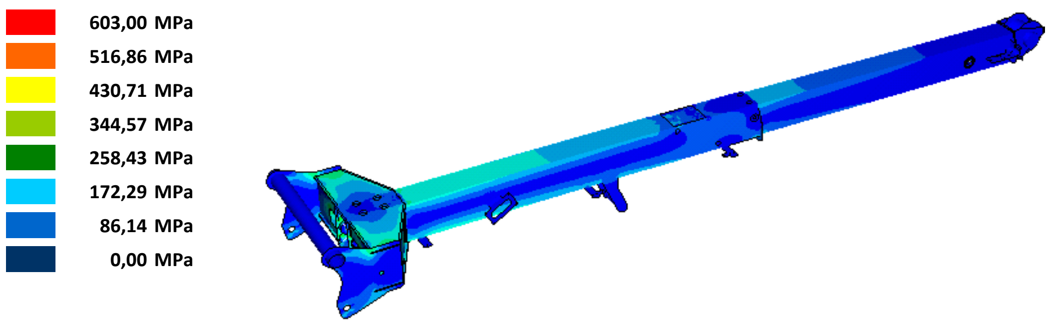
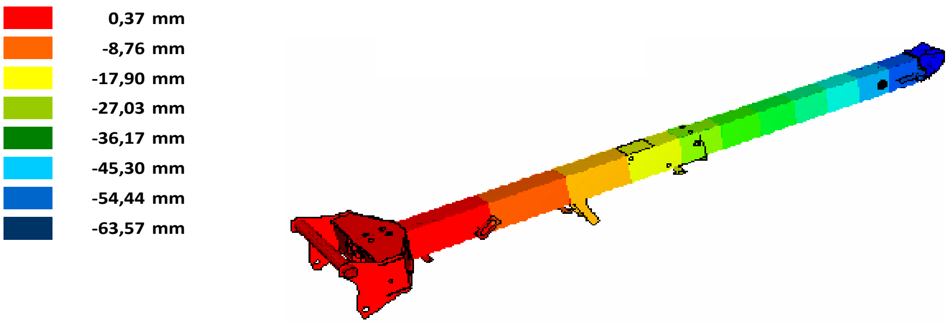


Fig.10: Vertical displacement in the additional load case 3



Method of trial and error has been applied in the redesign process.

Once the optimization process has finished, a telescopic crane jib model has been obtained which fulfil the stiffness and strength criteria specified by the standard and the company. The mass of the initial model and the optimized model is 498.2 and 434.7kg respectively, that supposes a 12% mass saving

In the optimization process have been applied the limits established in the equations 1, 2 and 3 for each of the components that make up the whole. Thus, in each of the pieces are examined the Von Mises stress, shear stress and the buckling. Then the safety factor of each component has been obtained starting the tensions, using the following equation.

$$SafetyFactor = \frac{\sigma_{e\_mat}}{\max(\sigma_{V.M.Max}, \tau_{\max}, \sigma_{comp.\max})} \tag{12}$$

If safety factor is higher than that established by the designers team (in our case of 1.11), these team reduce the thickness of the components or use materials with worse properties. In case that the safety factor is less, the designer team acts contrary, increasing the thickness or improving the quality of material used. This process is performed with all the pieces of the crane jib, taking into account the materials and the plate thickness available on the company. Once it is done, a new numerical calculation is carried out to analyse the behaviour of the modified components and to see if they fulfil the criteria. This process is repeated until to obtain the optimal thicknesses of all components of the machine

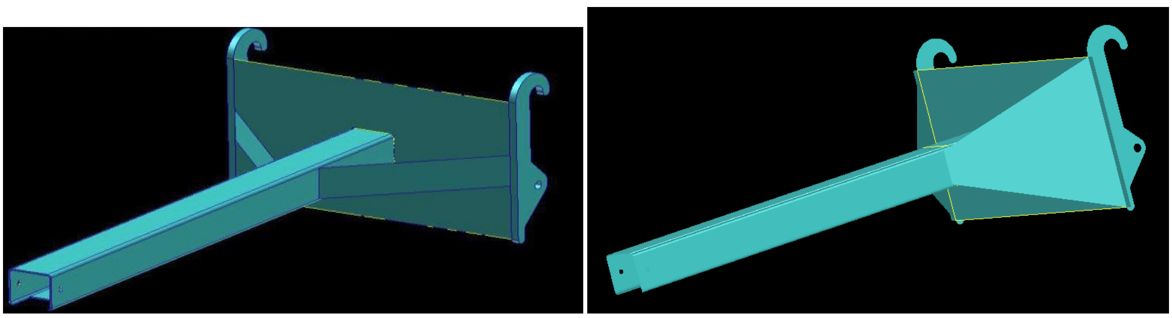
8. Application to a close box crane jib

In the case of closed box crane jibs is possible to make a previous pre dimension, starting the classical equations of mechanics of materials (Ortiz Berrocal, 1980) and the most common commercial square profiles. This process provides an initial starting profile. This profile should be calculated according to the load cases set out in section 2.2

In the case studied a crane jib has been designed with a capacity of 1,000 kg in a scope of 1,500 mm, so that the selected profile is a profile of 135x135x5 properly reinforced and modified.

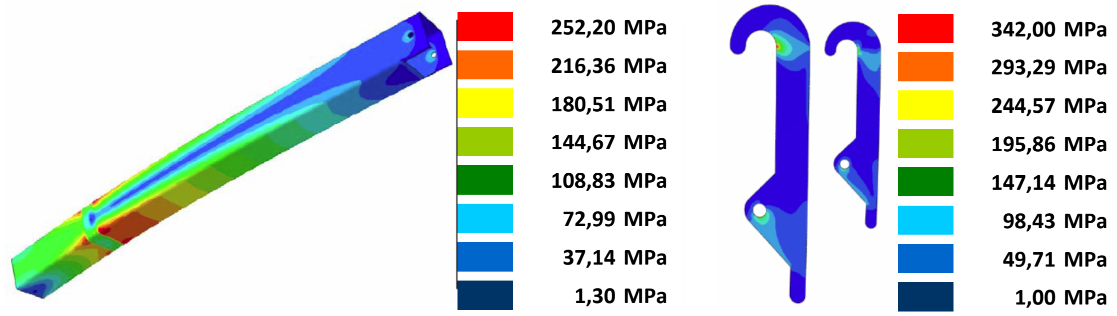
Subsequently a CAD model has been generated, and the design team has carried out a series of numerical calculations by means of the Finite Element Method and redesign. Figure 11 shows the process of redesigning the crane jib from the initial model to the optimized model.

Fig.11: Initial model (left) an optimized (right)



The Finite Elements Method has allowed obtaining the Von Mises stresses and safety factor for each piece of the crane jib. Figure 12 shows results for some components.

Fig.12: Von Mises stress of the closed profile (left) and the coupling plates (right) of the optimized model



The use of FEM has also allowed optimizing the shape and reduces the thickness some profiles, so that in the optimization process has been obtained a mass reduction of 29.5% in the optimized crane jib respect to the initial one. Table 3 shows the comparative mass of the initial and final model.

Table 3: Comparative of mass of the initial and optimized models of closed box crane jib

Mass (kg)	Tights	Towing hook	Profile	Rear wall	Collar	Total
Initial crane jib	7.87	13.65	28.8	29.5	-	79.82
Optimized crane jib	8.62	9.47	23.04	14.76	0.681	56.25

9. Application to a tubular crane gib

Like the previous case, it has been made a redesign for a tubular crane gib; these gibs are lighter but they have a more difficult and expensive manufacturing process. The load cases and the boundary conditions are identical to the previous case and the modeling and simulating process is quite similar, but in this case beam elements have been used. They can be analytically analyzed using classical resistance equations (Ortiz Berrocal, 1980).

Figure 13 show initial and final design and table 4 the weight analysis.

Fig.13: Initial model (left) and final one (right) for a tubular structure

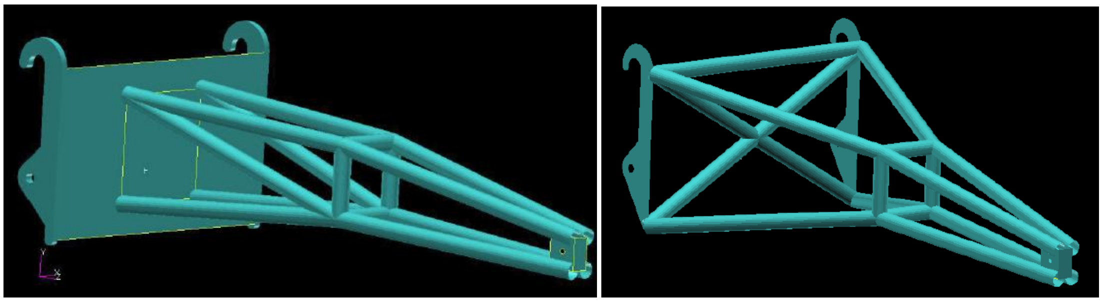


Table 4: initial and final weights for the tubular gib

Weight	Beamd	Hardness	End	Back sheet	Assemby
Initial	28,8	13,65	0,464	29,52	72,46
Final	22,55	14,28	0,929	4,2	41,93

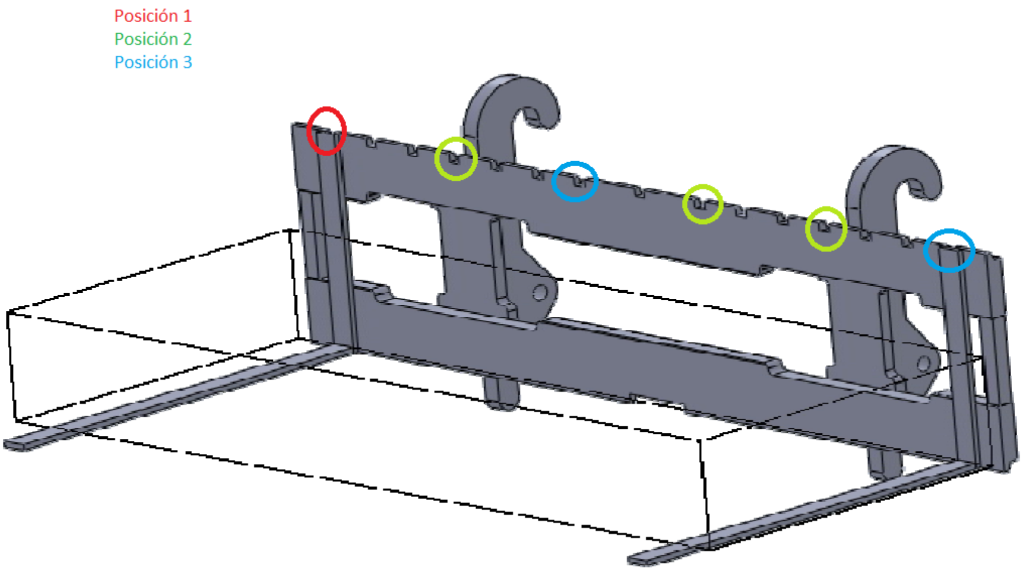
Main obtained conclusions are that it is possible to obtain a resistant assembly without the necessity to use bottom sheen and the beams must finish with a sheet to join them to the forklift truck. These sheets allow increasing the resistance of the assembly.

It has been obtained too that the diagonal bottom bars and not necessary and they can be deleted.

10. Pallet box lock application

For the pallet box lock, this element must be analyzed and designed to support the weight of the maximum portable load and nails; in this case the maximum load is 1.000 Kg. but it has been increases for the analysis to 1.330 Kg., like appear in the regulation. For the static load case there are three possible locations of the nails (see figure 14) and like appears in the point 2.1.

Fig. 14: possible positions of the load

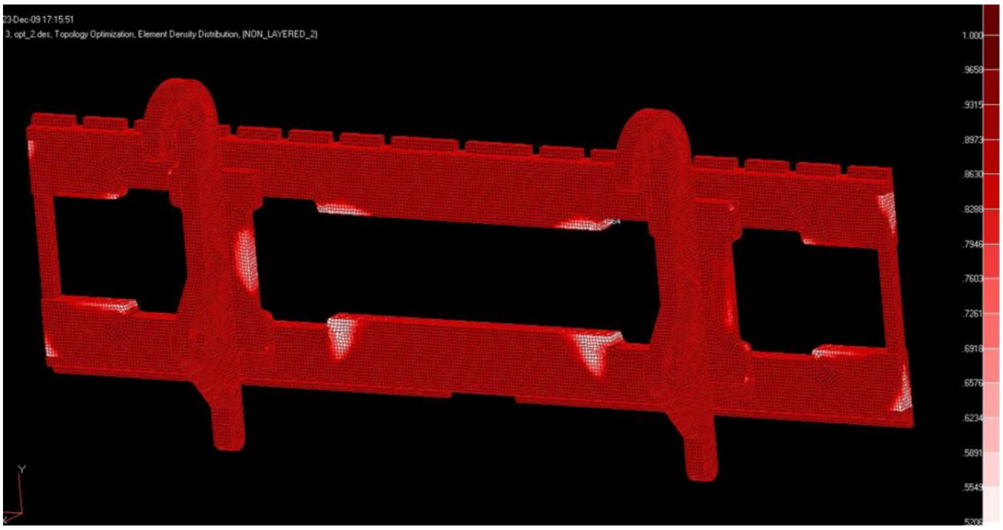


Nails have been not take in consideration because they are commercial element and it has been located the equivalent forces and moments in the contact zones to simplify the analysis.

It has been analyzed the stress and the displacements, and it has been made an automatic structural optimizations using the software MSC.Patran.Optimize.

Figure 15 shows the zones that can be optimized (in white) and that are low charged:

Fig. 15: zones that can be optimized and deleted



With the initial results, it has made some modifications:

- Delete 540 cm of the upper part of the bottom cross beam
- Increasing of 40 cm in the height of the lateral plates.

Table 4 shows the obtained results for the optimized model, with a weight reduction of 3.47 Kg and a final weight of 104.3 Kg.

Table 5: maximum Von Misses Stresses en each part of the pallet box lock

Piece	Material	Elastic Limit (MPa)	Weight (Kg.)	Von Mises Stress (MPa)	Security Coefficient
Upper transversal beam	St52	355	34.62	P1: 236 P2: 56.1 P3: 212	1.5
Botton Trensversal Beam	St53	355	31.5	P1: 223 P2: 24 P3: 183	1.59
Hook	St54	355	11.84 (each one)	P1: 196 P2: 75 P3: 219	1.62
Sheet	St55	355	5.54 (each one)	P1: 236 P2: 62.4 P3: 182	1.5
Box tube	St56	355	0.515 (each one)	P1: 123 P2: 21 P3: 94	1.54

11. Extensiometrical analysis

To verify the behavior of the mechanical assembly, to validate the numerical calculus methodology and to obtain the error of this numerical technic, it has been made an extensiometrical analysis with three uni-directional extensiometrical gauges (see figure 16) to obtain the strains in three characteristic points of the gib, previously selected.

Fig. 16: Extensiometrical gauges 1 (left) and 2 (right)

It has been made the fifth load case test of the point 3. The other load case is difficult to make experimentally because it is impossible to simulate the same wind loads. With only this load case it is possible to validate the used methodology ant the numerical results so, it is not necessary to do any additional test.

Like it can be observed in the figure 17, the gauges 1 and 3 support traction efforts and gauge 2 compression efforts. Figures 17 to 19 show that there is a maximum error of a 12% in the gauge 3; this is the lowest charged gauge so the error is higher for it.

Fig. 17: Comparative results for gauge 1

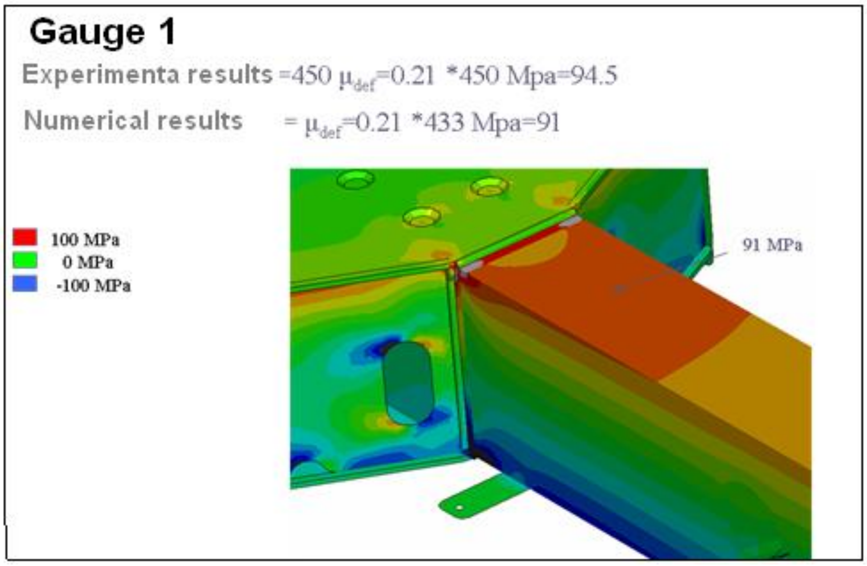


Fig. 18: Comparative results for gauge 2

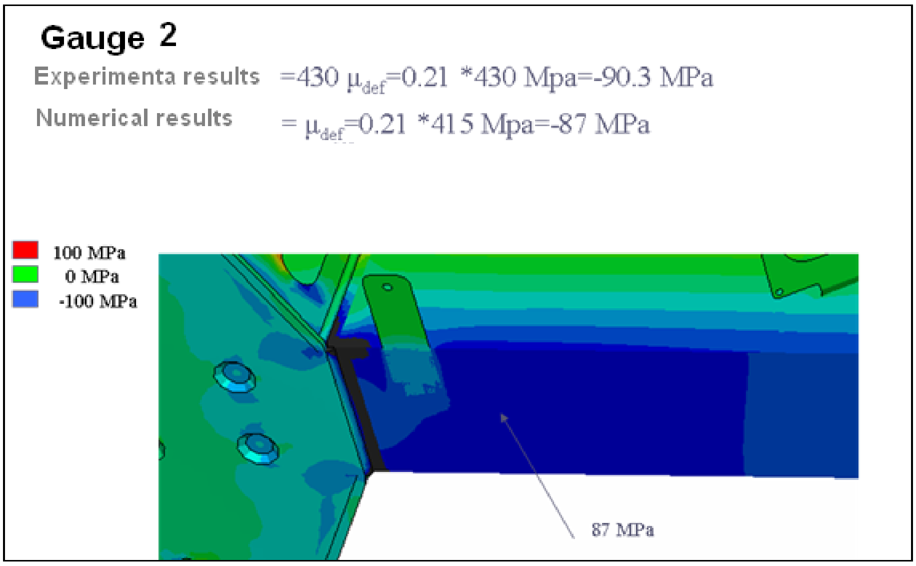


Fig. 19: Comparativa de resultados en la galga 3

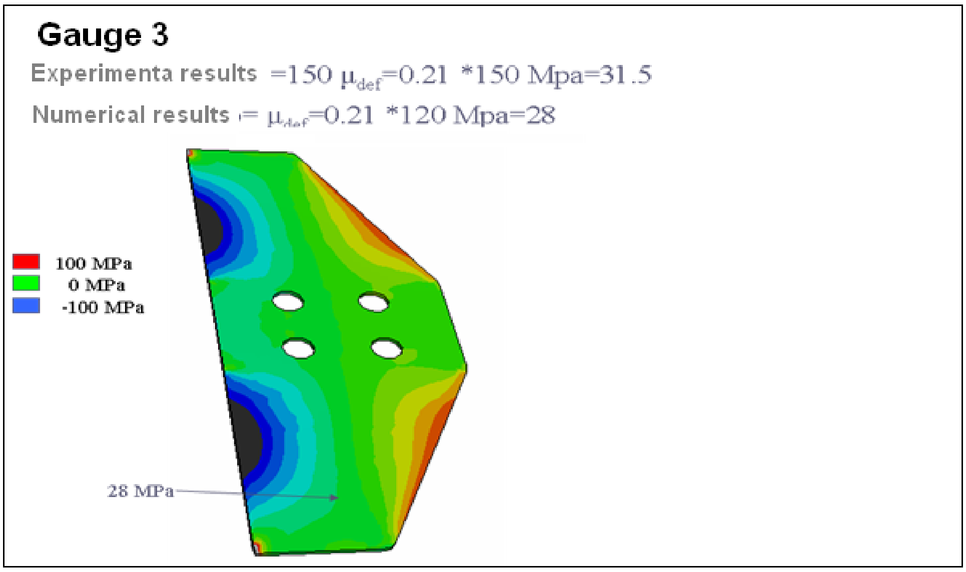
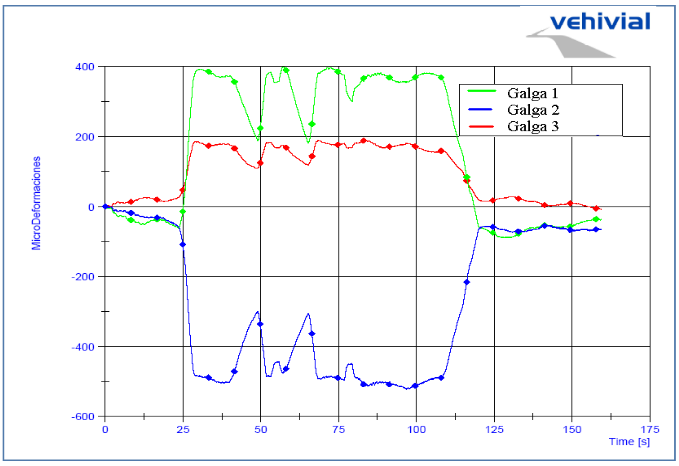


Figure 20: Micro-deformations vs. time for each gauge during the experimental test



12. Conclusions

Main conclusions are that, using this design methodology ant finite elements is possible to obtain lighter and with the same structural behaviour; it is possible too to analyse and design each part and each zone separately and quickly and easily it is possible to change the material, the geometry, the stiffness, etc., so the design process is easier, cheaper and it can be made in less time.

It has been observed too that the results of the extensiometrical analysis are quite similar to the numerical results for the same load case, so the methodology has been validated with experimental results.

About the optimization process, it has been applied two different methodologies: automatic optimization and "try and trial" one. Both are so good but the automatic one has the additional advantage that shows easily the zones that are susceptible to optimize and the program can to the optimization itself.

13. References

Miralbes R.; Castejon L.; "Design and Optimisation of Crane Jibs for Forklift Trucks". DYNA 2008 Vol. 83, no 5, pp. 282-290.

Ortiz Berrocal L.; "Resistencia de materiales". 3º Edicion. Escuela Técnica Superior de Ingenieros Industriales: Madrid, 1980, pp. 112-187.

Sun, G.F; Kleeberger, M.; "Dynamic responses of hydraulic mobile crane with consideration of the drive system". En "Mechanism And Machine Theory". 4º edidion. Oxford: Ed. Edelweisser. 2003. Vol. 38, no12, pp. 1489-1508.

Zienkiewicz O.C. y Taylor R.L., "El Método de los Elementos Finitos. Vol. 1, las bases". 8º Edición. Dover, Ed. Dover 2006. pp. 134-179.

Contact details (For further information, please contact)

Ramón Miralbés Buil. Dep. Ing . de Diseño y Fabricación. Ed. Torres Quevedo. C/ Maria de Luna s/n 50017 Zaragoza

Phone: +34/97676188

Fax: + 34/976762670

E-mail : miralbes@unizar.es

URL :