NUMERICAL MODELING, VALIDATION AND OPTIMIZATION OF FIXING DEVICES FOR STEERING MECHANISMS

Ana Paula C. S. Ferreira

apaula@utfpr.edu.br

Department of Mechanical Engineering, Federal University of Technology - Paraná (UTFPR)

Avenida Sete de Setembro, 3165, zip code 80230- 901,Curitiba, Paraná, Brazil

Ewald Carlos F. C. Machado

ewaldcarlos@gmail.com

JTEKT Automotiva do Brasil Ltda

Avenida Volkswagen Audi,1.200, zip code 83090-901,São José dos Pinhais Paraná, Brazil

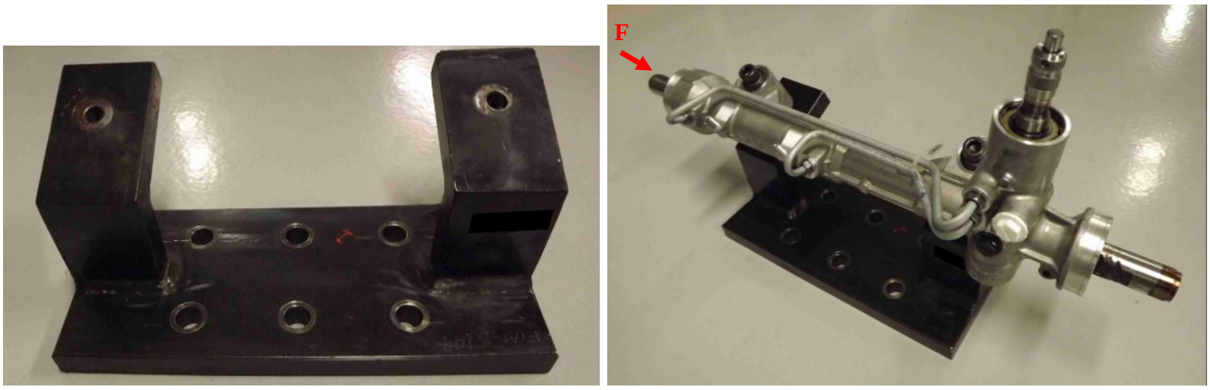
Abstract. *The laboratories of the automotive industry need to test the internal systems of vehicles to check strength and durability. An example is the steering mechanism, which is fastened at a device with high resistance, and subjected to loads of great intensity. These fixing devices are generally conceived without a proper structural analysis and optimization. This work presents a practical project where the goal is to verify if two existing devices are undersized or oversized. They are modeled by finite element method and the numerical model is validated with experimental data. Optimization is done in order to propose more efficient configurations for these devices.*

Keywords: *Non-destructive testing, Static analysis, Finite element method (FEM), Structural design, Optimization*

1 INTRODUCTION

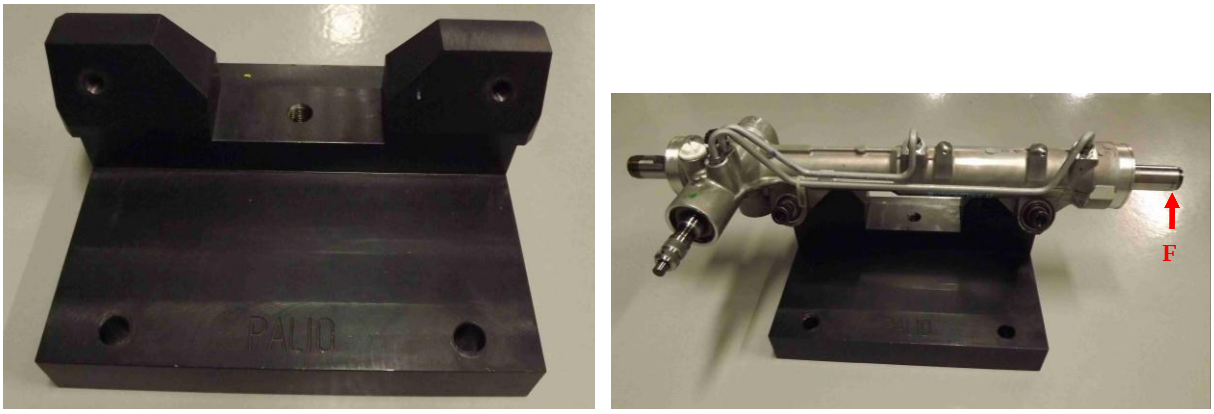
The Finite Element Method (FEM) nowadays is largely used in structural mechanics (Dill, 2012; Reddy, 2006). The improvements gotten in computational capability made possible expand the FEM applications. However, it is important to note that in all kinds of finite element analysis, the numerical model must properly represent the real structure. In order to check the validity of the finite element model, whenever possible, the FEM results are compared with experimental ones. Considering that the experiment is done in proper conditions (National Instruments, 2013), a good agreement among the numerical and experimental results indicates that the numerical model is reliable.

Some years ago, when the use of FEM was not so disseminated and available for the most industrial engineering areas, the structural projects were done with a combination of simplified analytical models and empirical knowledge from the engineers. The results are oversized or undersized structures and consequent loss of financial resources because of unexpected failure, rework or excess of material. In this context, this work has as main goal to check two existing structures regarding oversizing or undersizing. These devices are used to fasten automotive steering mechanisms in testing positions. They must support high loads during several tests. Figures 1 and 2 show the devices that are analyzed.



(a) Device 1 without the steering mechanisms (b) Device 1 with the steering mechanisms

Figure 1 - Device 1.



(a) Device 2 without the steering mechanisms (b) Device 2 with the steering mechanisms

Figure 2 - Device 2.

The analysis begins with experimental tests where loads are applied and deformations at strategic points are measured. These experimental data are stored for future comparison with numerical ones.

The finite element numerical model is done with Ansys® and in this stage the software capabilities are explored in order to keep the work practical aspect. Once the numerical model is considered acceptable, optimization (Arora, 2004; Vanderplaats, 1984) is done also with Ansys® in order to propose new devices.

2 METHODOLOGY

In this section the experiments and numerical model development are described. The experiments are done in order to validate the numerical model. The goal is to get a reliable numerical model.

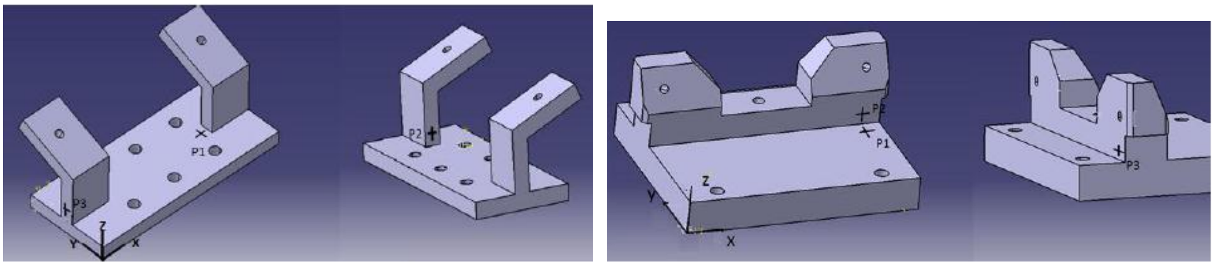
2.1 Experiments

The experiments are done in order to capture some deformations of the devices caused by the applied loads. Since the devices are robust and it is expected little deformation, it is convenient to use strain gages.

The loads that the steering mechanism should resist are standardized according to the project of the steering mechanism. Some fixing devices can test more than one steering mechanism, so the experimental load is the greater among the ones that can be applied into the fixing device. This load is applied by a hydraulic actuator. At device 1 the load direction is longitudinal to the steering mechanism (Fig. 1(b)) and the magnitude is 12kN. At device 2 the load direction is transversal to the steering mechanism (Fig. 2(b)) and the magnitude is also 12kN.

The deformations caused by the loads at the devices are measured by strain gages placed at strategic positions. These positions were defined by previous numerical simulation that detected the regions with great deformations.

The deformations are measured at points P1, P2 and P3 showed in Fig. 3(a) for device 1 and in Fig. 3(b) for device 2. For each point, the deformation in two directions is measured. The strain gages used are bidirectional from Kyowa and the acquisition hardware is the LMS SCM05.



(a) Device 1 (b) Device 2

Figure 3 - Representation of the strain gages positions.

Loads are applied by a hydraulic actuator associated to a load transducer. The load is gradually increased, starting with 1kN and reaching 12kN with steps of 1kN. For all load step, the deformation at each point in two directions is measured. It means that each device has a set of 72 data of stress-strain.

2.2 Numerical model validation

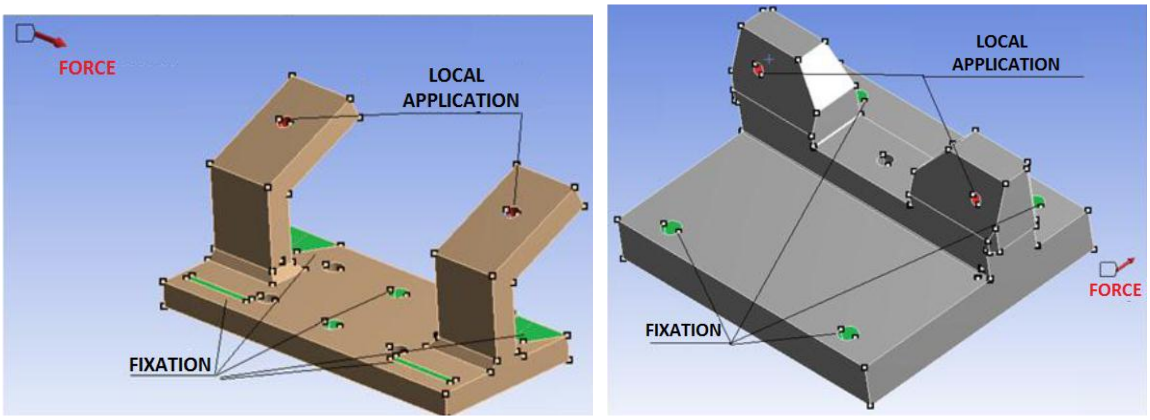
The devices are modeled at CATIA® and exported to Ansys®. The materials properties of the devices are in Table 1.

Table 1. Materials properties of the devices.

Properties	Device 1	Device 2
	SAE Steel 1010	**SAE Steel 1020**
Tensile strength (MPa)	360	380
Yield Strength (MPa)	210	220
Elastic Modulus (GPa)	190	200
Poisson’s Ratio	0.3	0.3

The Ansys® finite elements used are SOLID 187, CONTA 174 and TARGE 170 (Ansys® Workbench, 2011). The SOLID 187 is a tetrahedral element with quadratic displacement and 10-node. It has three degrees of freedom at each node (x, y and z translations). Finite elements CONTA174 and TARGE170 work together, the first one is a contact element and the second defines the surface where contact happens. The contact element and the face of the solid element with which it is connected must have the same geometric characteristics.

The loads are applied with the Remote Force function of Ansys®. It permits to apply a force at the same spatial position that it was applied in the experiment, but the steering mechanism does not need to be numerically modeled. Figure 4 shows the use of this function for the devices. It can be used as an alternative to the modeling a rigid part subjected to a force. A Remote Force is equivalent to a regular force on a face or a force on an edge, also some moment. Here the remote force application is defined at the faces of the screws holes. The spatial fixation of the devices is also defined at the faces of the screws holes, the ones located at the devices bases.



(a) Device 1 (b) Device 2

Figure 4 - Remote force Ansys® function.

The deformations were measured with PROBE function of Ansys®, which returns the value of the specified variable on the specified point. Figures 5 and 6 show a comparison between experimental and numerical results, for devices 1 and 2, respectively.

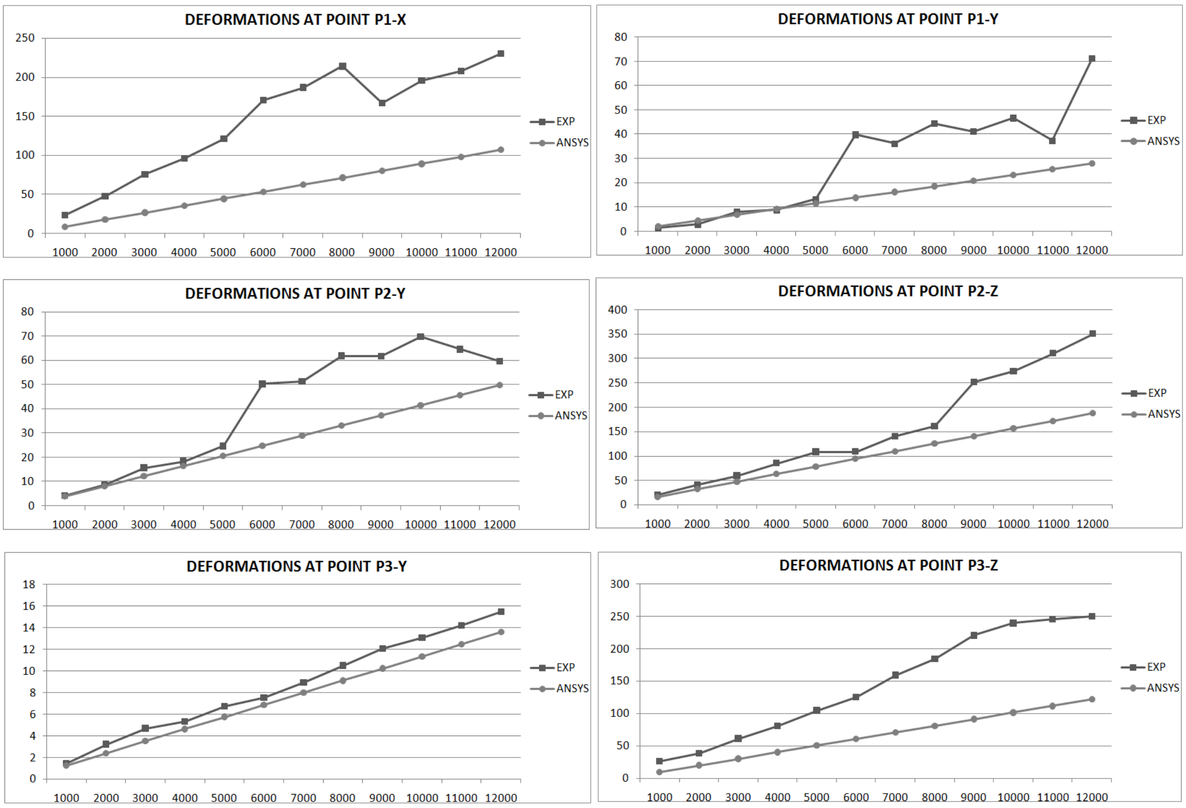


Figure 5 - Device 1: Numerical and experimental results.

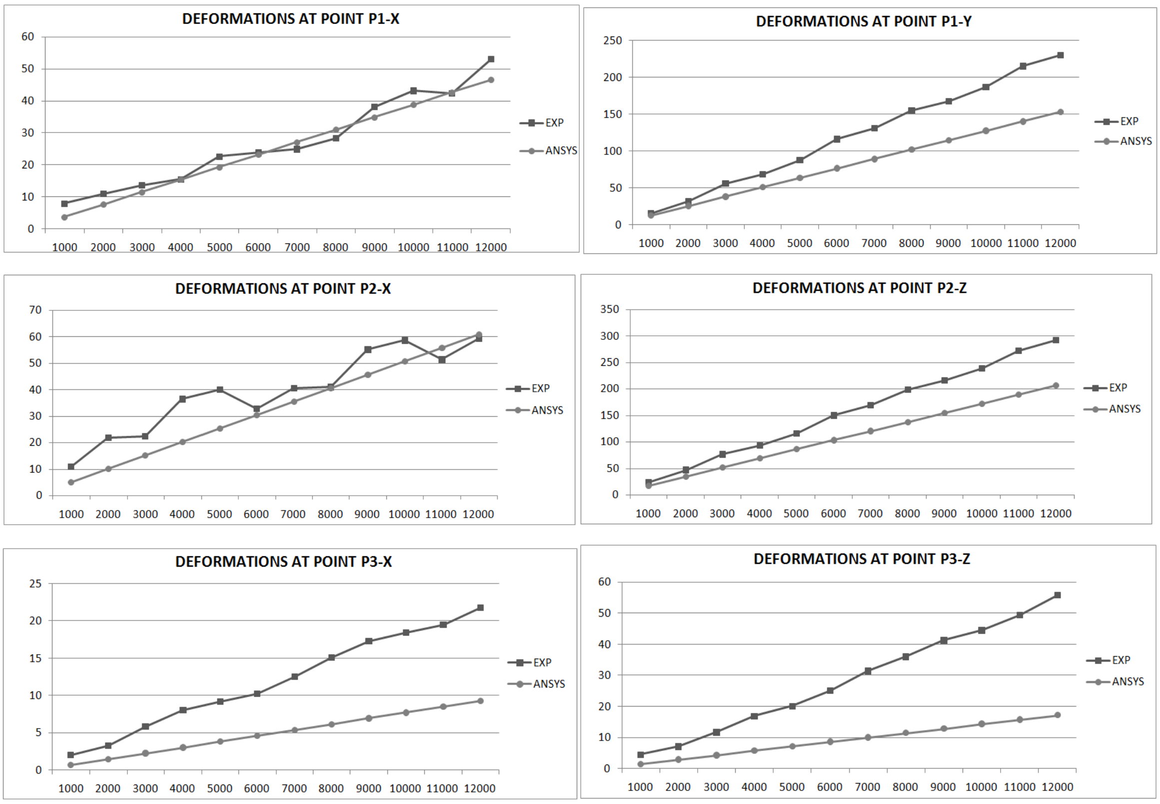
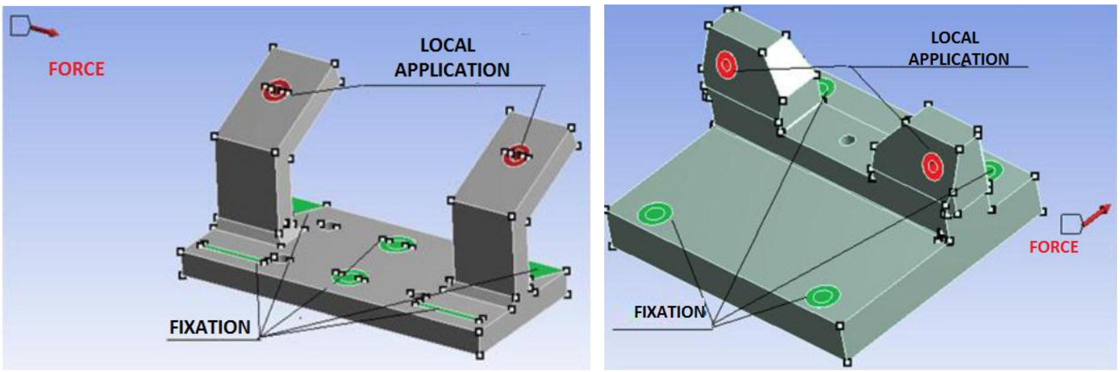


Figure 6 - Device 2: Numerical and experimental results

The positive aspect of these results is that the numerical and experimental results present the same tendency, however the numerical model has a linear load-deformation relationship in all cases and the same is not true for the experimental response. Furthermore, the relative error is too big indicating that the numerical model should be adjusted.

In order to get a better numerical representation the following model corrections were done:

- In the real model, the steering mechanism fixation is done with lock washer. This was included in the numerical model and is showed in Fig. 7.



(a) Device 1 (b) Device 2

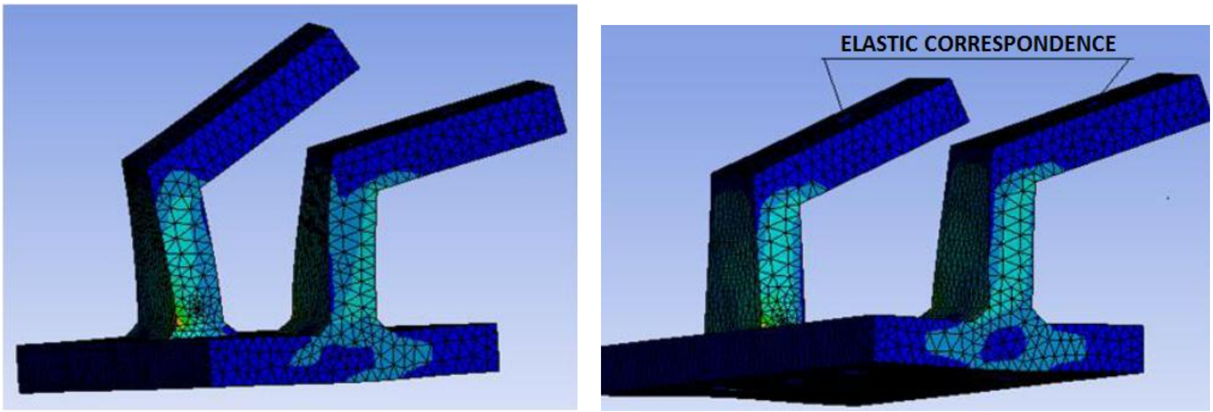
Figure 7 - Transmission of force between steering mechanism and the fixing device, after the adjustment of the numerical model.

- The material properties were altered according to Table 2, in order to consider some variations at the fabrication process.

Table 2. Material properties of the devices, after the adjustment of the numerical model.

Properties	Device 1	Device 2
	SAE Steel 1010	**SAE Steel 1020**
Tensile strength (MPa)	360	380
Yield Strength (MPa)	200	220
Elastic Modulus (GPa)	170	190
Poisson’s Ratio	0.25	0.3

- It is included an elastic correspondence in the interface of the device with the steering mechanism, in order to simulate the steering mechanism stiffness. Figure 8 shows a simulation without the elastic correspondence (a) and with it (b). It is possible to note that, with the elastic correspondence, the superior faces of the device are aligned.



(a) Without the elastic correspondence (b) With the elastic correspondence

Figure 8 - Device 1 numerical model.

- It is done a convergence analysis for the finite element mesh. The sizes of the elements are reduced from 10mm to 3mm, as shows Fig. 9. It is possible to note that the maximum stress response of the model stabilizes at 5mm, indicating that this is a suitable size for the elements.

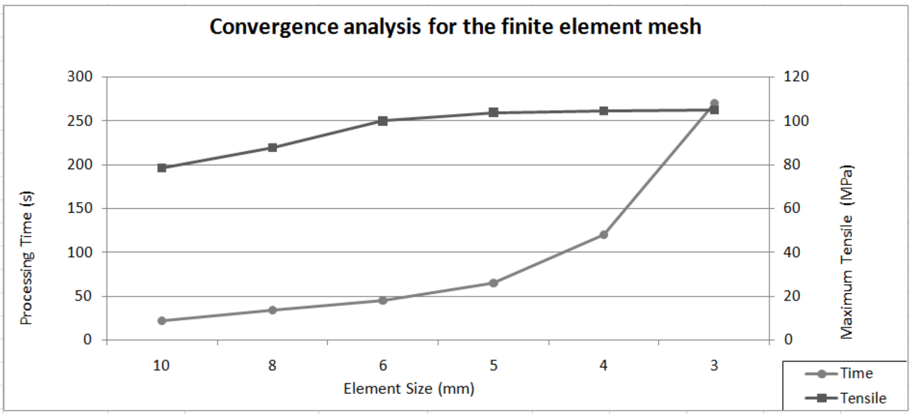


Figure 9 - Convergence analysis.

After these four adjustments in the numerical model, the simulations are repeated. The new results are in Figures 10 and 11.

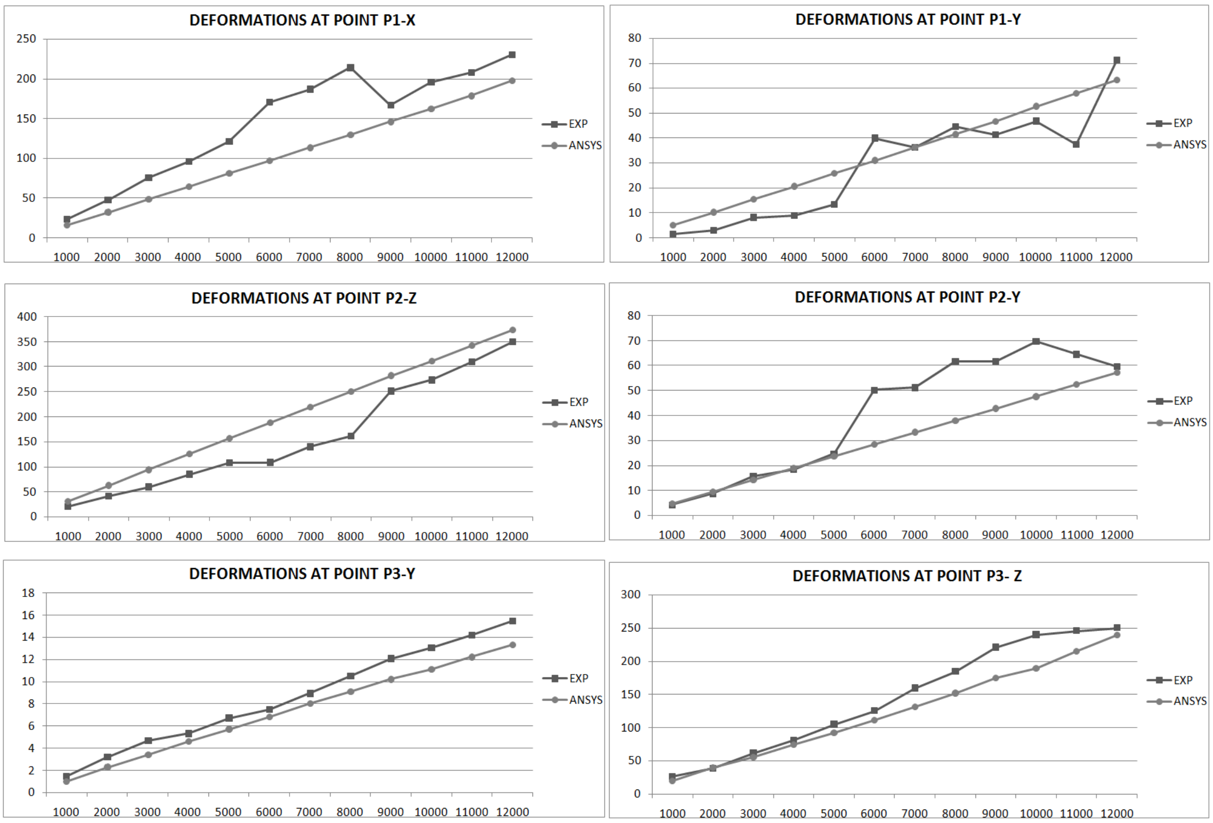


Figure 10 - Numerical and experimental results for device 1, after the adjustments in the numerical model.

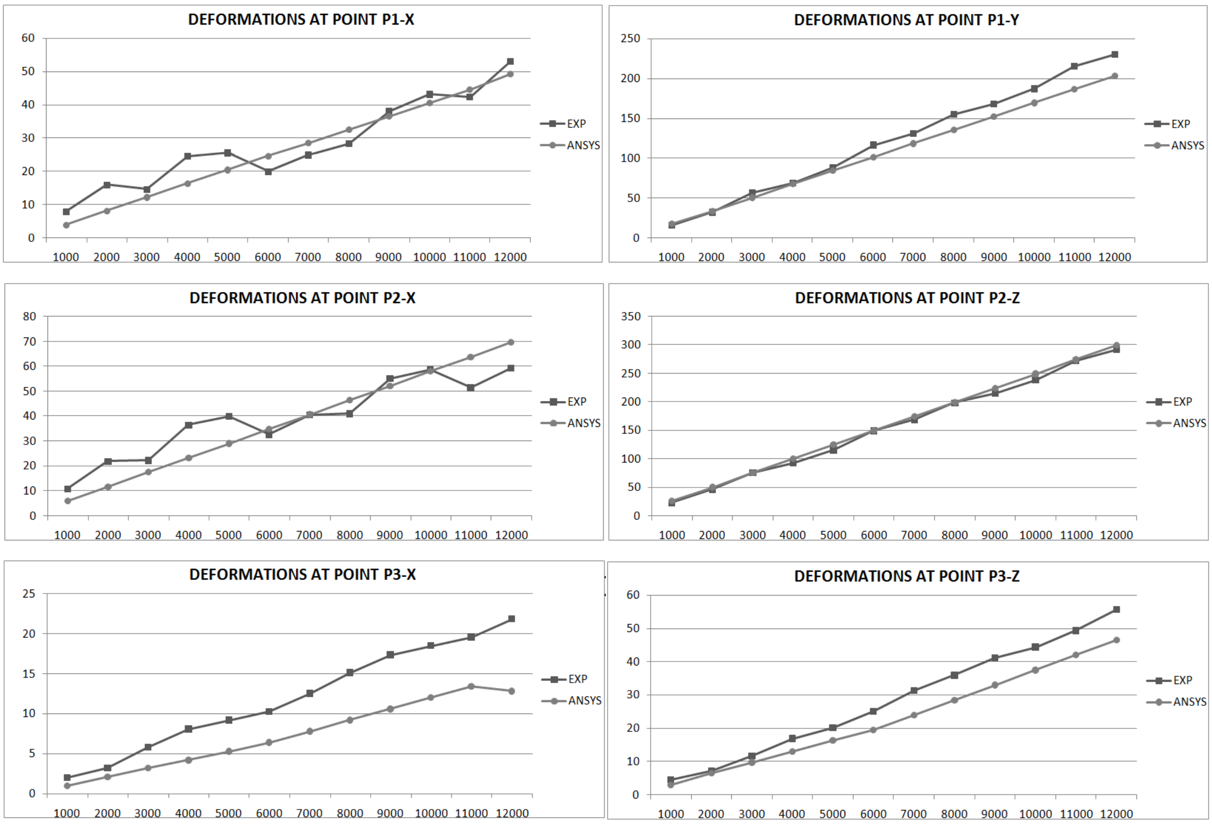


Figure 11 - Numerical and experimental results for device 2, after the adjustments in the numerical model..

Analyzing these figures it is possible to note that the numerical and experimental responses are in better agreement after the numerical model adjustments. The greater errors in deformation values at device 1, before model adjustment, were as follows: P1-X with 161.76%, P1-Y with 66.17% e P3-Z with 119.35%. After de model adjustments the errors in deformation values, in these directions, are reduced respectively to 56.19%, 38.59% e 14.38%. The greater errors in deformation values at device 2, before model adjustment, were as follows: P2-X with 63.33%, P3-X with 145.70% e P3-Z with 192.04%. After de model adjustments the errors in deformation values, in these directions, are reduced, respectively to 20.47%, 68.20% e 25.29%.

Considering that the devices were used in several and different tests, including cyclic ones, it is not possible simulate the wear that these devices have suffered. In this way, it is considered that the numerical models properly represent the devices and they are used for optimization.

3 RESULTS

This section presents the optimization formulation, setup and results. Once the numerical model is considered in good agreement with the real structure and reliable, it is used for optimization purpose. The optimization process proposes some modifications at the devices that make them more suitable to their applications.

3.1 Ansys® optimization formulation

The Ansys® internal optimizer, as any optimization process, requires that the objective function, design variables and constraints be defined. When the optimization problem has constraints, it is redefined by the optimizer, becoming unconstrained by the inclusion of penalty functions into the objective function.

An optimization problem with constraints can be defined by (Ansys Workbench, 2011; Vanderplaats, 1984):

$$\text{Minimize } F(\mathbf{X}) \tag{1}$$

Considering the following inequality, equality and side constraints, respectively:

$$\begin{aligned} & g_j(\mathbf{X}) \leq 0 \quad j = 1,\ \mathrm{n}_I \\ & h_k(\mathbf{X}) = 0 \quad k = 1,\ n_E \\ & X_i^L \leq X_i \leq X_i^U \quad i = 1,\ n_D \end{aligned} \tag{2}$$

where $\boldsymbol{X}$ is the vector o design variables, n_I and n_E are the number of inequality and equality constraints, respectively. X_i is a design variable with lower X_i^L and upper X_i^U bounds, n_D is the number of design variables.

The objective function may be linear or nonlinear, the equality and inequality constraints can exist or not, but side constraints must be defined. If the goal is maximize the objective function the optimizer makes $F(\mathbf{X}) = -F(\mathbf{X})$ and $g_j(\mathbf{X}) \geq 0$ is changed to $-g_j(\mathbf{X}) \leq 0$.

It is assumed that the set of design variables $\boldsymbol{X}$ is continuous between the lower and the upper bounds. The number of design variables must be reasonably small (i.e., less than 20). The optimizer has only one objective function. If it is necessary to use multiple objective functions, they must be combined into a single objective function, or optimized individually.

The original optimization problem with constraint is converted so that it can be solved as a series of unconstrained problems. The basic approach is

$$\text{Minimize } \Phi(\mathbf{X}) = F(\mathbf{X}) + P(\mathbf{X}) \tag{3}$$

where *Φ(X)* is called the pseudo-objective function. *P(X)* is the penalty term, which depends on the method being used.

In the Ansys® optimizer the penalty term *P(X)* is calculated using an iterative process called the augmented Lagrange multiplier (ALM) method. In this method, the unconstrained problems are solved by the Fletcher-Reeves (FR) method, which itself is an iterative process. The FR method uses a line search (LS) technique (yet another iterative process) to find a minimum in a given search direction.

In order to reduce the CPU time, the optimizer does not search for a global optimum. The assumption is that the number of minima within the space of the design variables is small. When this condition is met, a global optimum is generally found, but it is possible that it is a local minimum. One way to leave a local minimum is to restart the optimization process using as start point the optimum point found in a previous optimization.

It is possible to note that the Ansys® internal optimizer is proper to perform simple optimization tasks. If multicriteria optimization, or finding the global optimum are required, it will be necessary an external optimizer. The present partner of Ansys® is VisualDOC, but it is also possible to use Ansys® only to compute the values of the objective functions and develop an own algorithm for the optimization process.

3.2 Optimization setup

The optimization process requires the definition of the objective functions, constraints and design variables. The objective function for both devices is mass reduction. The constraint is the fatigue stress, since the devices are used for some tests with variable loads. It means that the ultimate stress S_{ut} is corrected by some factors, resulting in the fatigue stress S_e. The correction factors are presented in Eq. (4) (Norton, 2004).

$$S_e = 0.5 \times S_{ut} \times C_l \times C_s \times C_f \times C_t \times C_r \tag{4}$$

where,

S_{ut} – ultimate stress

C_l – loading factor. Used to considering the load kind. For only bending or torsion, it has unit value. For normal load, the value is 0.7.

C_s – size factor. It is defined in order to consider or not internal heterogeneity of the material. Eq. (5) is used for computing this factor for rectangular sections

$$C_{si} = 1.189\left(\sqrt{\frac{0.05bh}{0.0766}}\right)^{-0.097} \tag{5}$$

C_f – surface finish factor. Considers quality of the surface finishing. Eq. (6) is used for computing this factor for machining processes, since the devices have this kind of finishing.

$$C_f = 4.51 \times \left(S_{ut}\right)^{-0.265} \quad (6)$$

C_t – temperature factor. When the temperature is less than 450ºC, it is set equal to one.

C_r – reliability factor. It is related to statistical concepts of a large population of mechanical parts. Since in this work only two devices are considered, this factor is set equal to one.

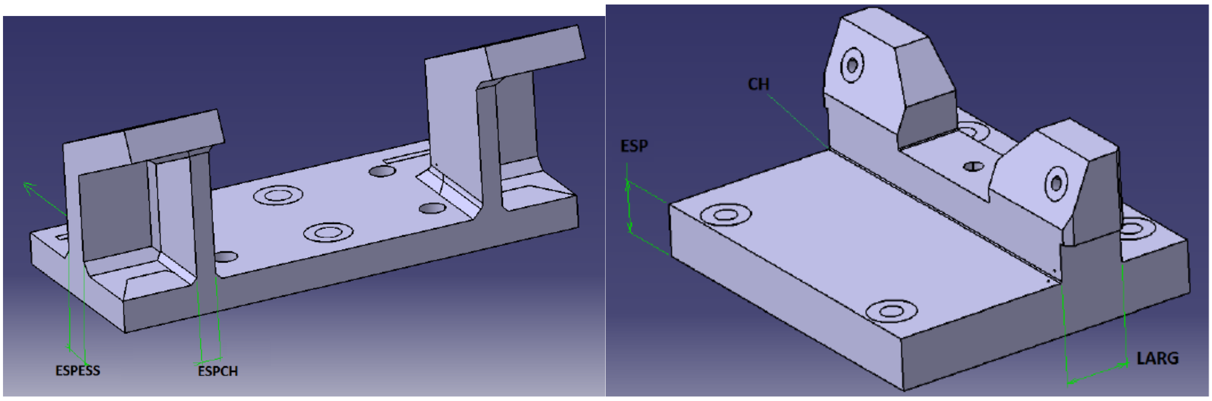
Table 3 shows the values of the correction factors for each device and the corresponding constraint value found.

Table 3. Correction factors and constraints values.

	Device 1	**Device 2**
C_l	1	1
Cs	0.832 (*b* = 20.7mm and *h* = 125mm)	0.769 (*b* = 50mm and *h* = 260mm)
C_f	0.948	0.935
C_t	1	1
C_r	0.897	0.897
Constraint	127 MPa	123 MPa

The design variables for device 1 are the thickness of the arms (ESPESS) and thickness of the reinforcements (ESPCH) presented in Fig. 12(a). This reinforcement is not present in the original device, but is included in order to reduce the stress in the device.

The design variables for device 2 are the thickness of the base (ESP), width of the tower (LARG), and the chamfer between them (CH), presented in Fig. 12(b).



(a) Device 1 (b) Device 2

Figure 12 - Design variables.

The maximum and minimum values for the design variables are presented in Table 4.

Table 4. Side constraints for the design variables.

	Design Variables	Min value (mm)	Max value (mm)
Device 1	Thickness of the arm	20	45
	Thickness of the reinforcement	5	15
Device 2	Thickness of the base	22	40
	Thickness of the tower	30	50
	chamfer	2	7

3.3 Optimization results

The optimization was done in order to reduce the devices mass, respecting the constraint of fatigue stress and the side constraints of the design variables.

The original and optimized configurations of device 1 are presented in Table 5. The optimization process led to three candidates to optimum points with resulting stress of about

127 MPa and mass of about 10.1 kg. The original device 1 has a stress of 184 MPa and mass of 9.5 kg. The optimization process suggested a mass increase of 0.6 kg but with a consequent stress decrease of 57 MPa.

Table 5. Optimization results for device 1.

	Design variables (mm)			
Device 1	**ESPESS**	**ESPCH**	**Stress (MPa)**	**Mass (kg)**
Original	20.7	-	184	9.5
Candidate A	20.6	14.1	127.2	10.5
Candidate B	23.1	12.4	126.8	10.6
Candidate C	27.4	8.9	127.0	10.7

Figure 13 shows the sensitivities analysis for device 1. This kind of analysis makes possible to verify the influence of each design variable in the objective function and in the constraint. Observing this figure it is possible to see that the dominant design variable is the thickness of the arm, for the objective function and for the stress constraint.

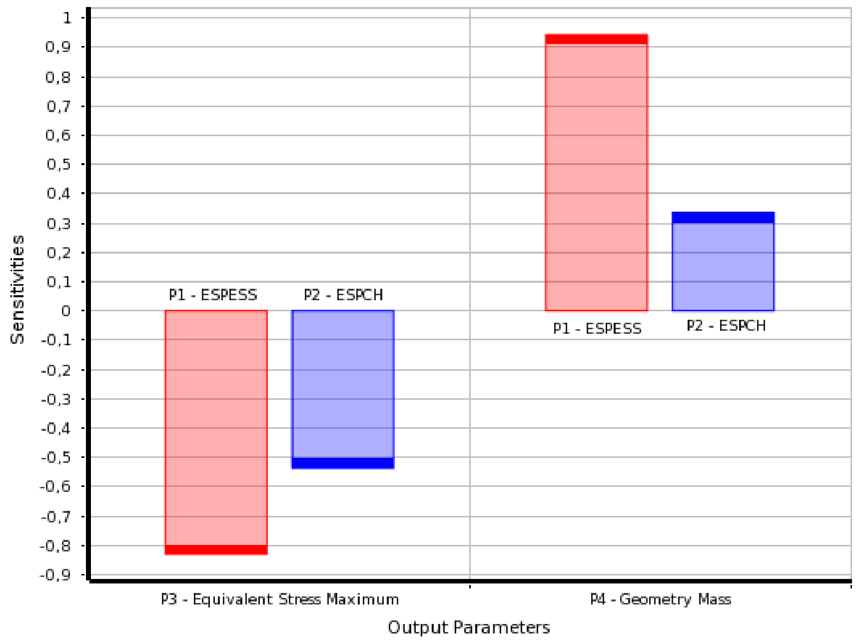


Figure 13 - Sensitive analysis for Device 1.

The original and optimized configurations of device 2 are presented in Table 6. The optimization process led to three candidates to optimum points with resulting stress of about 122 MPa and mass of about 18.5 kg. The original device 2 has a stress of 117 MPa and mass of 27.5 kg. The optimization process suggested a mass reduction of 9 kg and a consequent stress increasing of only 5 MPa.

Table 6. Optimization results for device 2.

	Design variables (mm)				
Device 1	**CH**	**ESP**	**LARG**	**Stress (MPa)**	**Mass (kg)**
Original	-	40	50	117	27.5
Candidate A	6.9	22.8	38.9	119.5	17.8
Candidate B	6.4	23.0	42.4	121.6	18.4
Candidate C	6.5	22.1	45.7	113.7	18.6

Figure 14 shows the sensitivities analysis for device 2. Observing this figure it is possible to see that the design variable that most influences the stress constraint is the tower thickness. However, the design variable that most influences the objective function is the base thickness.

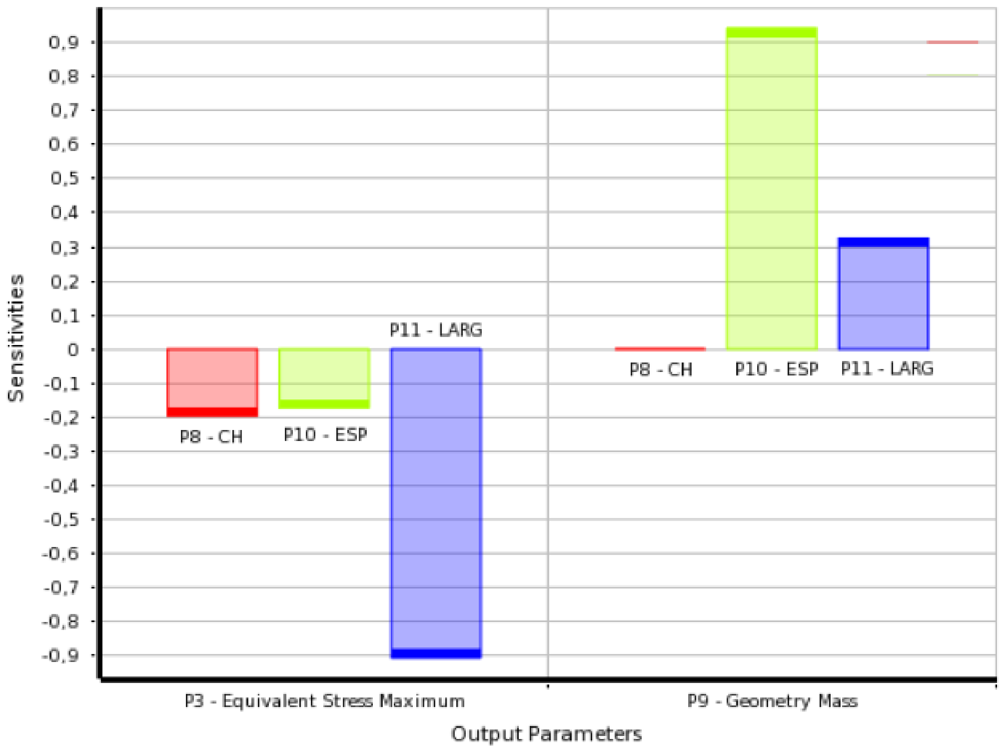


Figure 14 - Sensitive analysis for Device 2.

4 CONCLUSIONS

The global idea of the present work is to review an old design using modern engineering tools. Two fixing devices used for testing steering mechanisms are considered. The first task is to experimentally test these devices by applying loads of crescent magnitude and acquire the corresponding deformations at strategic points with strain gages. After that, numerical models of the devices are developed and simulated at the same loads of the experiment. The deformations are compared and the numerical model is adjusted in order to better represent the real one. As was expected, it is not possible a perfect adjust between numerical and experimental because the real devices were extensively used in various kinds of steering tests. However, numerical and experimental models presented the same tendency with good agreement in some points after the numerical model adjustment. This numerical model is considered acceptable and is used in optimization processes. The goal of the optimization is to find a lighter device that meets the constraint of fatigue stress and side constraints.

Device 1 has originally 9.5 kg and a maximum stress of 184 MPa. After de optimization, the device proposed has 10.7 kg but maximum stress of 127 MPa. Device 2 has originally 27.5 kg and a maximum stress of 117 MPa. After de optimization, the device proposed has 18.6 kg with maximum stress of 113 MPa. In both devices, the design review made possible a meaningful improvement.

The present work is a real example of the benefits that good tools and prepared team can bring to a corporation. Using relatively simple experiments and numerical simulation it is possible to improve results and save resources.

ACKNOWLEDGEMENTS

The authors want to thank to Federal University of Technology - Paraná (UTFPR) and JTEKT Automotiva do Brasil Ltda.

REFERENCES

Ansys Workbench, 2011. User’s guide version 14.0.

Arora, J. S., 2004. Introduction to optimum design, New York: Elsevier Academic Press.

Dill, H. E., 2012. The finite element method for mechanics of solids with ANSYS applications, New York, NY: Taylor & Francis.

National Struments. Measuring strain with strain gages. Available in: <http://www.ni.com/white-paper/3642/en>. Acces in 22/09/2013.

Norton, R. L., 2004. Projeto de máquinas: uma abordagem integrada, Bookman, Porto Alegre.

Reddy, J. N., 2006. An introduction to the finite element method. New York: McGraw-Hill Higher Education.

Vanderplaats, G. N., 1984. Numerical optimization techniques for engineering design: with applications, McGraw-Hill, New York.