ABSTRACT

This work presents the design nationalization procedure of an automotive rack housing. The main objective of the project was basically the product optimization, that was made by: material reduction; reduction of the number of components; manufacturing costs reduction; ease of assembly; and ease of manufacturing. The current design of the product, formed basically of a component made of steel bar and other one made in cast aluminum, in addition to bushings, inserts and other details was replaced by a new component completely built in cast aluminum alloy. The rack housing wall was reduced and ribs were added to it to increase its stiffness. Thus the total mass of the product was reduced in 13% and the total number of components was reduced in 21%. Structural calculations were made according internal procedures of the company both for the current design and for the proposed one. Quasi-static nonlinear structural analyses were made considering the elasto-plastic and the hyperelastic behavior of the materials, in order to check the possibility of failure by yielding, plastic collapse and ductile fracture. Besides, modal analyses were made in both designs in order to evaluate the dynamic behavior of the structures. It was guaranteed that the proposed design presents higher safety factors than the current design and higher stiffness too, allowing the structure to work free of resonances in the range of operation frequencies of the vehicle.

INTRODUCTION

Recent trends in automobile development activities for reduction of lead-time and cost have led to a current situation where Computer Aided Engineering (CAE) techniques are fully to skip conventional development steps for making and checking costly prototypes [1].

Many automakers now use a computer simulation instead of preparing costly prototypes to analyze the strength and the collision resistance of a vehicle body, for example [2].

Virtual engineering is defined as integrating geometric models and related engineering tools such as analysis, simulation, optimization, and decision making tools, etc., within a computer-generated environment that facilitates multidisciplinary collaborative product development. Virtual engineering shares many characteristics with software engineering, such as the ability to obtain many different results through different implementations. Interaction within the virtual environment should provide an easily understood interface, appropriate to the user's technical background and expertise, that enables the user to explore and discover unexpected but critical details about the system's behavior. Similarly, engineering tools and software should fit naturally into the environment and allow the user to maintain her or his focus on the engineering problem at hand. A key aim of virtual engineering is to engage the human capacity for complex evaluation [3].

Time to market is another important issue regarding to a product design. It is known that this time has been reduced year by year and engineers must design quickly and with minimal error.

Nishino [4], for example, modeled a static strength test using Finite Elements Method (FEM) in order to determine the load to failure through an inverse method (Figure 1).

Figure 1 - Maximum principal strain plotting for the static strength test performed by Nishino [4]

In that work, the correct material curve was measured and used in the Finite Elements Analysis (FEA). The same fixations of the test were represented, and the input load was measured until a strain level of 1.0% (material strength) was
Kubota [5] carried out an impact CAE simulation of a steering collision test. All the nonlinearities presents on this phenomenon (material, contact, finite strain and large displacements) were considered, and the experimental and numerical results were compared. Figure 2 shows the analysis model.

The actual drop impact test and the CAE simulation of the drop impact test showed good agreement [5].

Several examples of use of numerical tools during the steering system development and evaluation are available in the literature. Load to failure determination, noise reduction, mass minimization and kickback reduction are examples where CAE tools can be applied.

**MOTIVATION**

The motivation to start the development of a new configuration of this rack housing became when the company realized a great opportunity to prepare a design optimization of the current configuration of the product. That product is working inside acceptable levels of mechanical resistance, however a mass and manufacturing costs reduction could be possible.

**PRODUCT DESIGN MODIFICATIONS**

The current Rack Housing concept has one product formed basically of a component made of steel bar and other one made of cast aluminum. The proposal was to replace some components (for example mounting rubbers, supports, bushings) and this current configuration for one new component completely built in cast aluminum alloy.

After the decision to make this development, the product engineering team started to prepare a proposed 3D model based on the current steering gear. The current design was used as a baseline and the steel bar and the cast aluminum were replaced for only one cast aluminum component and some ribs were added to increase the stiffness of the part.

When the 3D model was concluded, the first Finite Elements Analysis (FEA) was performed. After analyzing the FEA results, the component was changed. At the first moment, some fillets were added on the housing and material was added in some critical points, locally increasing the thicknesses. Then, a new loop of FEA was conducted with the second proposed model.

With the final FEA results, a comparison between the results of the current and the proposal models was made. It was sawn that the proposal configuration is stronger than the current one. Besides, the natural frequencies are higher in the proposal model than in the current one. Another important point is that the total mass of the product was reduced in 13% and the total number of components was reduced in 21%. Figure 3 shows the current configuration and the second proposed model.

**NUMERICAL SIMULATIONS**

With the proposed configuration of the rack housing, virtual mechanical tests were performed. The same test procedures also were made with the current configuration, defined as the baseline. The following goals were established:
a) Increase the safety factor for overload failure

b) Increase the natural frequencies

Two types of structural tests were performed: static and dynamic. In the static analyses, the nonlinearities imposed by the mechanical behavior of the materials and the contact between the different parts were considered. In the dynamic tests free vibrations analyses were performed in order to determine the mode shapes and their respective natural frequency values.

### STATIC TESTS

For structural analyses the general equation of motion (showed below) is solved and simplifications are made in the equation depending on the type of the analysis (static, modal, transient, harmonic, and so on).

\[
[M][\ddot{x}] + [C][\dot{x}] + [K][x] = [F(t)]
\]

In the case of static analyses, the displacement is assumed to be quasi-static and the damping and inertial effects are unconsidered. So, the displacements are solved for in the matrix equation below:

\[
[K][x] = [F(t)]
\]

The constant \([K]\) represents structural stiffness. A linear structure obeys a linear relationship known as Hooke’s law and can be expressed simply as \(F = Ku\), where \(F\) is the applied force, \(u\) is the structural displacement which results from the applied force and \(K\) is the proportionality constant relating force to displacement. A common example of Hooke’s law is a simple spring (Figure 4).

![Figure 4 - Spring behavior and the Hooke’s law](image)

However, significant classes of structures do not have a linear relationship between force and displacement. Because a plot of force versus displacement for such structures is not a straight line, such structures are said to be nonlinear. The stiffness, \(K\), is no longer a constant, but rather becomes a variable function of applied load. This variable stiffness is referred to as \(K'\) or the tangent stiffness (Figure 5).

![Figure 5 - Nonlinear structural behavior and the tangent stiffness](image)

The most common example of a material nonlinearity is a ductile metal undergoing a tensile test beyond the elastic limit. Other examples of material nonlinearities include hyperelasticity, time dependent creep, viscoplasticity, and viscoelasticity. Other types of structural nonlinearities are situations that cause significant changes in stiffness. Typical reasons are:

- Large deflections which produce a dramatic enough change in shape to effect the structural stiffness and the direction of the loading. This category also includes applications where the very presence of a stress field influences the stiffness. A good example of both phenomenon acting together is a loaded fishing rod. A good example of stress stiffening without large deflection would be a guide wire or piano string in tension. As the tension changes, the stiffness also changes.

- Changing Status is another important category of structural nonlinear behavior. The most common changing status phenomenon is the contact relationship between two bodies (frictional contact or frictionless contact). When two or more bodies come into contact or separate as a result of applied loads, the stiffness of the assembly usually differs from the stiffnesses of the individual parts. This difference can often be quite dramatic.

- In many real world problems, all these sources of structural nonlinearity will operate together. One example is the large deflection impact of an automobile collision.

A Finite Elements (FE) Model was generated for each configuration of the rack housing using the commercial package ANSYS Mechanical [6]. The metallic materials, as laminated steel and cast aluminum, had their mechanical behavior represented by linear plasticity with isotropic hardening mathematical model [6]. That model allows the consideration of both elastic (recoverable) and plastic (permanent) deformations in the metallic components and the verification by yielding, plastic collapse and ductile fracture failure modes.

The parameters that define the constitutive model were defined from the minimal requirements from the standards DIN226 for the aluminum and ASTM A36 for the steel. Mechanical uniaxial tensile tests were performed with the
same materials used on the housing manufacturing and it was verified that their mechanical strength are significantly higher than the standard requirements. However, the standard minimum requirements were used in these analyses in order to obtain a higher safety factor. Figure 6 shows the curves and the parameters values for both aluminum and structural steel.

![Figure 6](image)

**Figure 6 - Material curves and parameters for aluminum and static structural using the linear plasticity model**

One of the current housing fixations is made by an elastomeric bushing pressed by a U-Type plate that is directly fixed on the chassis (Figure 7). That fixation is considerably flexible and was mathematically modeled to capture its influence in the housing stiffness.

![Figure 7](image)

**Figure 7 - Flexible fixation of the current rack housing**

To determine the mechanical behavior of the bushing’s material (Ethylene Propylene Diene Monomer - EPDM) a specimen was obtained from some part. On a macroscopic level, elastomers can undergo large elastic (reversible) deformations, anywhere on the order of 100-700%. This is due to the untwisting of cross-linked molecular chains. There is little volume change under applied stress since the deformation is related to straightening of chains. Hence, elastomers are nearly incompressible \( (v \approx 0.5) \) [7]. Because of the lack of conditions to perform shear, biaxial tensile, volumetric compression (machines and devices) and uniaxial tensile (dimensions of the test specimen), a brick with 22x20x10mm was made from some part and a compressive quasi-static loading was applied on it. No sliding between the specimen and the devices surfaces occurred and a longitudinal displacement of 3.61mm was applied. During the test, the reaction force caused by the compression was measured with a load cell and a Force x Deformation curve was generated.

In his work, Felhós [8] developed a mixed experimental-numerical procedure to determine the visco-hyperelastic properties of an EPDM rubber and investigated the rolling friction whereby a steel ball is rolling on a rubber plate.

Only an uniaxial compression test is not enough to characterize an elastomer. These materials have different stress x strain relations when subject to uniaxial tension, biaxial tension, uniaxial compression, volumetric compression and shear. Usually, in tension, the material softens then stiffens again. On the other hand, in compression, the response becomes quite stiff and the stress x strain relationship can be highly nonlinear (Figure 8).

![Figure 8](image)

**Figure 8 - Typical force x deformation relation under tension and compression of elastomers**

The numerical simulation of structural components needs suitable constitutive models and appropriated material parameters to describe its mechanical behavior. In order to determine the material parameters, experimental tests are performed in specimens and the experimental conditions are numerically reproduced in a simulation [9]. There is a great variety of material models available on literature, for example Mooney-Rivlin, Yeoh, Ogden, Gent, Arruda-Boyce, Blatz-Ko and Neo-Hookean.

The one chosen for this application was the Yeoh model, because of its numerical stability and easy convergence [10]. This model has three variations related to the number of inflection points in the curve (single curvature; one inflection point; two inflection points). The chosen model was the 2nd Order, therefore considering a curve with one inflection point (double curvature). A material is said to be hyperelastic if there exists an elastic potential function (or strain energy density function), which is a scalar function of one of the strain or deformation tensors, whose derivative with respect to a strain component determines the corresponding stress component [10]. The elastic potential function of the model chosen is shown below:

\[
W = c_{10}(I_1 - 3)^1 + c_{20}(I_1 - 3)^2 + \frac{1}{d_1}(J - 1)^2 + \frac{1}{d_2}(J - 1)^4
\]

Where \( W \) is the strain energy, \( I_1 \) is the first strain invariant, \( J \) is the volumetric ratio and the parameters \( c_{10}, c_{20}, d_1 \) and \( d_2 \) are respectively related to the incompressible strain, at the first and the second curvature, and to the compressibility, also at the first and the second curvature [10]. These are the four
scalar parameters that must be identified to define the material behavior. Due to the lack of data related to the material compressibility it was considered as fully incompressible ($v=0.5$). Consequently, $d_1$ and $d_2$ were assumed as zero ($d_1=d_2=0$).

A mixed numerical-experimental procedure has been used to determine the parameters $c_{10}$ and $c_{20}$. The mixed numerical-experimental procedure has been used in inverse modeling in several areas and it is widely applied in parameter identification procedures. Several experimental procedures can be set up to analyze the material mechanical behavior. Tensile tests are commonly used due to its simplicity. In Mahnken [11] a material characterization is presented for an axisymmetric tensile bar of ferritic steel. A sensitivity analysis is performed and the gradient based descent methods are used for the minimization of a least-square function. Other example of a possible approach for the inverse problem is found in Gavrus et al. [12] to characterize a thermal-visco-plastic model for Aluminum by means of resistance torque data of a torsion test.

The compression test performed with the EPDM specimen was numerically reproduced and an initial guess of the parameters was used. A Force x Deformation curve was obtained and compared to the same curve experimentally obtained. That parameters guess needs to be made several times until the difference between the numerical and the experimental values be minimized to an acceptable value. In order to automate the process an optimization routine was adopted using the commercial package modeFRONTIER [13] (Figure 9).

After the cost function minimization the following parameters values were identified: 0.01 MPa for $c_{10}$ and 0.025 MPa for $c_{20}$. The numerical and experimental curves, as well as the deformed shape of the specimen after the simulation can be seen in the Figure 10.

The optimization study was defined with two input parameters (design variables); one calculation procedure (compression test simulation); one output variable (numerical force curve); and a target vector (experimental force curve). Considering that the values of the abscissas are the same for both vectors (numerical and experimental), the cost function to be minimized corresponds to the sum of the squares of the differences between the vector values during each time instant at the whole test.

The chosen optimization algorithm is the SIMPLEX [14]. This method is widely used in nonlinear mono-objective optimization problems, the case of this study.

With the selected material model and the identified parameters, different stress x strain curves for uniaxial tension, biaxial tension and shear tests were generated (Figure 11).

**Boundary Conditions**

**Supports**

The fixations were modeled as rigid, not considering the flexibility added by bolts, bushes and washers. In the regions of contact with the bolts radial movements were fixed. In the base of the fixations, only the normal displacements were fixed (Figure 12, current model; Figure 13, proposed model). In the case of the current configuration, the base of the EPDM...
bushing had its movements fixed in the three directions at the region in contact with the chassis.

- With the **No Separation** contact, sliding is allowed only for short levels of deformation
- With the **Frictional** contact, sliding is allowed when the tangential force is higher than the frictional force ($\mu N$)

The coupling between the Housing and the Carter was modeled with a Bonded condition. On the other hand, both the contact between the U-Type Plate and the Bushing and the Carter and the Bushing were modeled using Rough condition.

### Loading

A load case considering the vehicle during a maneuver was applied on the models. The load scenario is composed by the compression force transferred by the steering arm and the torque generated by the driver through the steering wheel. This torque is decomposed and the resultant forces are directly applied on the housing (Figure 14).

### Results

The strain level was verified at each part for both configurations. The strain component evaluated was the maximum principal ($\varepsilon_1$). This is the strain responsible for crack opening, and because of this, it is considered the most critical one.

Figure 15 shows the contours of maximum principal strain for the current model. As one can see, strain values over the material ultimate strain limit (0.01mm/mm) were found. However, this region has influence of numerical singularity because of its lack of fillet’s representation. In addition, the considered material properties are considerably lower than the actual ones.
With the proposed model, however, no regions with maximum principal strain values over the material ultimate strain limit were found (Figure 16).

As the goal of these analyses was to compare the proposed model with the current one, only a decrease in the stress values is already sufficient to guarantee a structural strength improvement in the rack housing design.

**AVALIAÇÕES DINÂMICAS**

Free vibration analyses were performed with both configurations of the mechanism. In this case, the mass and stiffness matrices ([M] and [K]) are assumed as constant, and the natural circular frequencies (\( \omega_i \)) and the mode shapes (\( \phi_i \)) are calculated respectively as the eigenvalues and the eigenvectors of the expression below:

\[
([K] - \omega_i^2 [M])\{\phi_i\} = 0
\]

So, some assumptions are made in order to guarantee the linearity of the mass and stiffness matrices:

- Linear elastic material behavior is assumed
- Small deflection theory is used, and nonlinearities are not included
- Damping is not included (undamped natural frequencies)
- No excitation is assumed (free vibrations)
- The structure can be constrained or unconstrained

The first six vibration modes of each configuration were evaluated and the obtained results are shown in Figure 17. The same support conditions of the static analyses (Figure 12 and Figure 13) were used.

As one can see, the obtained values of natural frequencies for the proposed model are considerably higher than the obtained for the current model. This is due to the mass reduction and the stiffness increase.

**EXPERIMENTAL EVALUATIONS**

After the numerical calculations, experimental tests are performed with the chosen geometry in order to validate the design. Several tests are conducted in the company laboratories. These tests are described below. The current rack housing configuration was approved in all of those tests, and the proposed model will be evaluated after the first prototype manufacturing.

**DURABILITY TEST**

**Description:**

The test is conducted on the complete steering gear, including tie rods, which is held in a standard production body front-end or a device with similar stiffness using its original attachment points. The angular position of the tie rods corresponds to the vehicle as-installed position for the maximum permissible gross vehicle weight (heaviest engine). The steering
kinematics of the vehicle shall be simulated by means of the additional steering stroke.

Requirement:

With the exception of the thrust piece clearance, all parameters of the steering gear shall always comply with the drawing requirements after the durability test.

OVERLOAD TEST

Description:

The test is conducted on a core of the steering gear in oil-filled condition after it has been mounted in a rigid device using its original attachment points.

Working lines that meet the requirements of standard production shall be used to introduce the pressure; these are connected to a common connection of the test bench unit instead of the steering valve.

Requirement:

After loading the component with pressure \( p_1 \), no droplets must be visible.

IMPACT TEST

Description:

From height, a weight of mass is allowed to drop freely into the rack bar. It must be ensured that the falling movement is not impaired by friction between the weight and its guide. The impact work for the strength of the housing as well as for the rack and pinion strength shall be obtained from the performance specifications.

Requirement:

No starter cracks or fractures are permitted on the housing, the rack, or the pinion shafts. The mechanical steerability shall be guaranteed.

STATIC BREAKAGE

Description:

The test is conducted on a core of the steering gear, which is held in a standard-production body front-end or a device with similar stiffness using its original attachment points.

Requirement:

The test is conducted up until a drive torque at the pinion. After subsequent disassembly, the components must not exhibit any fractures or starter cracks.

RUNNING WEAR TEST

Description:

The test is conducted on a core of the steering gear, which is held in a standard-production body front-end or a device with similar stiffness using its original attachment points. The steering gear pinion is connected to the drive in such a way that forces cannot be applied to the steering gear in the axial or radial direction, except for the applied torque. The effective line of the load force runs through the center line of the rack, in which the pulling and thrusting forces are permitted to be applied over one side of the rack.

Requirement:

After testing, the mechanical and hydraulic function of the steering gear must be ensured.

COMMENTS AND CONCLUSIONS

CAE technology is very helpful during the product development. It allows the application on the steering gear of the actual loads that came from the vehicle during operation. The results of this study, for example, showed the strength and the dynamic behavior of the gear.

With the results provided by this study, a comparison with the vehicle conditions can be made. Besides, a checking of the need to make any design change (material, dimensional, etc) also can be performed before the prototyping. It reduces the costs and the time to market. Moreover, the natural frequencies, that are a very critical point, can be measured even before the physical model manufacturing.

Performing numerical simulations the engineer can have a better understanding of the physical phenomena, and consequently, define the points where a design modification is necessary to attend the requirements of the vehicle.

For the company, the possibility to make a mathematic simulation before start the prototype preparation is helpful to avoid to waste time. Other gains using numerical simulation are the design optimization and the product cost reduction. Especially for this work, the two objectives established during the start of the study were achieved: the mass of the steering
mechanism was reduced as well as the number of components, also reducing the stress level and increasing the system stiffness.

REFERENCES


CONTACT INFORMATION

Bentzion Pliacekos

Product Engineer

JTEKT Automotiva Brasil Ltda.

+55(41) 2102-8715

bentzion.pliacekos@jtekt.com.br

Antonio Carlos B. Guimarães Neto

Technical Support Analyst


+55(48) 3953-0057

antonio.guimaraes@esss.com.br

ACKNOWLEDGMENTS

The authors would like to thank all the product development team of JTEKT Automotiva Brasil Ltda., as well as the technical services team of ESSS – Engineering Simulation and Scientific Software.