2 nd Worldwide MSC Automotive Conference.
Dearborn, Michigan; October 9-11, 2000

Reducing the Weight of a Frontal Truck Axle Beam Using Experimental Test Procedures to Fine Tune FEA.

N. León *****, O. Martínez, P. Orta C., P. Adaya
ITESM, Campus Monterrey, Sucursal de Correos "J", Monterrey, N.L. Mexico
W. de Oliveira, M. Robledo
DIRONA S.A., Carr. Colombia, Escobedo, NL. Mexico

ABSTRACT

Following is a description of the methods used and the results obtained during an academic research project aimed to introduce in DIRONA the use of finite element methods to improve its product quality while reducing development time. Furthermore the objective of reducing material and manufacturing costs was followed while maintaining the stress levels under the values previously obtained with earlier products that had worked reliable during several years. For the finite element analysis, MSC.Patran/MSC.Nastran V. 8.0 was used. The FEA results obtained were experimentally validated through DIRONA standard experimental test procedures.

***** Corresponding author, tel. + 52 8 3284220, email: nleon@campus.mty.itesm.mx**,**

INTRODUCTION

DIRONA is a Mexican company that produces front and rear heavy truck axles and that is committed to excellence and competitiveness in the national and international markets. The following objectives are pursued in DIRONA´s product development process:

- Highest product quality and reliability
- Customer satisfaction
- Reduction of development time and costs

ITESM is a multi-campus educational system, unique in Latin America because of its academic excellence and research. DIRONA and ITESM joined efforts in order to introduce Hi-Tech CAD/CAM/CAE tools in the development and innovation process of new truck axles.

Following is an example of the use of MSC.Patran/MSC.Nastran finite element software during the development and innovation process of a front truck axles beam, where FEM techniques were used combined with experimental test procedures to reduce the weight of the beam while maintaining the stress levels at confident values. In this paper are described the methods used and the results obtained during an academic research aimed at introducing in DIRONA the use of finite element methods to improve its product quality while reducing development time.

The main objective in this academic research was to establish methods and ways of reducing material and manufacturing costs while maintaining the stress levels under the values previously obtained at earlier products that worked reliably during several years. DIRONA´s proposed engineering development process is shown in Figure 1.

PROBLEM DEFINITION: DIRONA´s product development process

DIRONA has manufactured front and rear heavy truck axles since many years in Mexico. 3D CAD tools are used during its design process, but the company did not earlier performed the whole new product development process. Now, DIRONA is aimed to introduce Hi-Tech CAD/CAM/CAE tools in order to perform its whole product development process. In this way DIRONA and ITESM joined efforts to introduce a product development process that reduces the development time and costs. For this purpose FEM techniques were used combined with experimental test procedures to validate and fine-tune the results of the FEM analysis.

The first step in this proposed engineering development process is to select the product to be improved or developed. During the development process a 3D parametric product model is created as basis for a family of parts. The advantage in making 3D parametric models is that it allows fast variations in the model geometry. DIRONA has been working with Pro/ENGINEER for five years and at the beginning also an academic program together with ITESM created the foundation for its introduction.

As shown in Figure 1, during this proposed engineering development process, the geometry of the 3D CAD model is imported in the FEA software for stress and strain analysis. After this step the product geometry is meshed using the automatic mesh generator in MSC.Patran. After mesh generation, the defined boundary conditions are applied to the model, which is then analyzed using MSC.Nastran. The results are post-processed using MSC.Patran and thoroughly analyzed by the design team to identify opportunities at the stress and strain levels. These results are used to formulate new product geometry with the 3D Parametric CAD software, which is analyzed until a satisfactory stress and strain map is obtained. The obtained FEA results are verified with a set of experimental test procedures.

Generally in product development process two basic paths can be followed to improve the product performance: one is the design of a new product concept (different geometry or function principles) the other alternative is the optimization of the existing product using different optimization strategies.

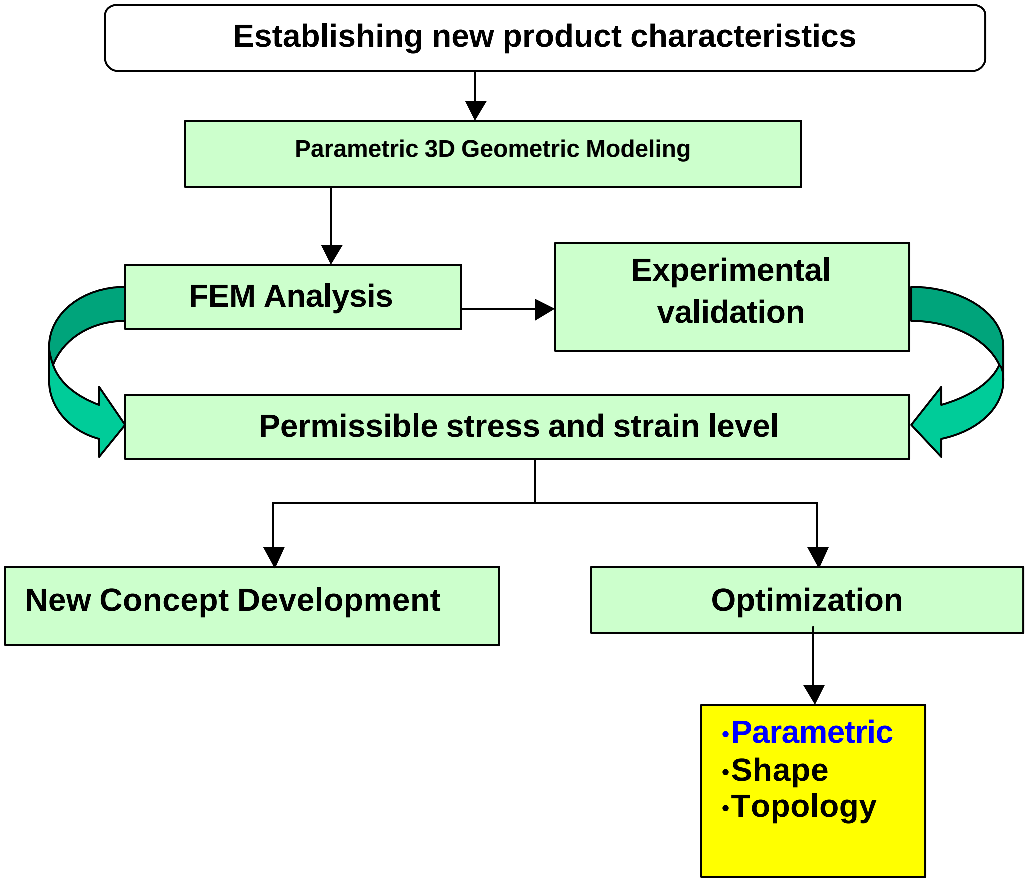

Figure 1. *Proposed basic engineering development process.*

During the optimization process the evaluation of the different alternative combination of product design parameter is carried out to achieve one or several objective functions. Generally the design parameters are subject to restrictions and boundary conditions that has to be previously described and defined. Furthermore a search strategy has to be defined to find the combination of design parameters that fulfills the objective functions.

Commonly, during product development existing limitations affect product performance, for example: natural physic laws, material properties, customer specifications, existing standards etc. These limits constraint the product design specifications, in achieving the desired objective functions. One important constraint are permissible stress and strain levels.

The optimization process involves following activities:

- Selection of variables that describe the design alternatives
- Selection of objective functions to be minimized or maximized.
- Establishment of restrictions, expressed in terms of design variables, which must be satisfied by any acceptable design.

Three different kind of optimization procedures exist:

- Parametric
- Shape
- Topology

In this case the optimization procedure to be applied would be the parametric one, but as there was not optimization software available for this project, instead of it an iterative parametric improvement procedure was used to improve the objective functions as part of the proposed engineering development process at DIRONA.

This iterative improvement procedure starts also with the definition of the restrictions and the parameters to be controlled. After each analysis modifications of the geometry are carried out based on experiment design (Figure 2). Here the influence of the design parameters changes is measured in order to identify the design sensitivity. These parameters usually are referred to as design variables, and they are used to represent the sectional dimensions of the beam.

CASE STUDY: FRONT AXLE BEAM, FOR THE VALIDATION OF THE PROPOSED ENGINEERING DEVELOPMENT PROCESS AT DIRONA.

The application of this process should allow the company to offer products with increased quality and reliability, but at lower cost. The families of parts integrated in the system of the front and rear axles, must have the same characteristics for each different lifting capacity. In this case the proposed engineering process is applied to the axle front beam.

The product was first modeled in 3D parametric CAD, (Figure 3). Before the FEM analysis, several simplifications of the model geometry have been made with the purpose of reducing the analysis time and model size. Figures 4 and 5 show the simplifications applied in this case. Furthermore, taking in accounts the part and loads symmetry, only one half was analyzed. This is compatible with the used DIRONA test procedure shown in Figure 6.

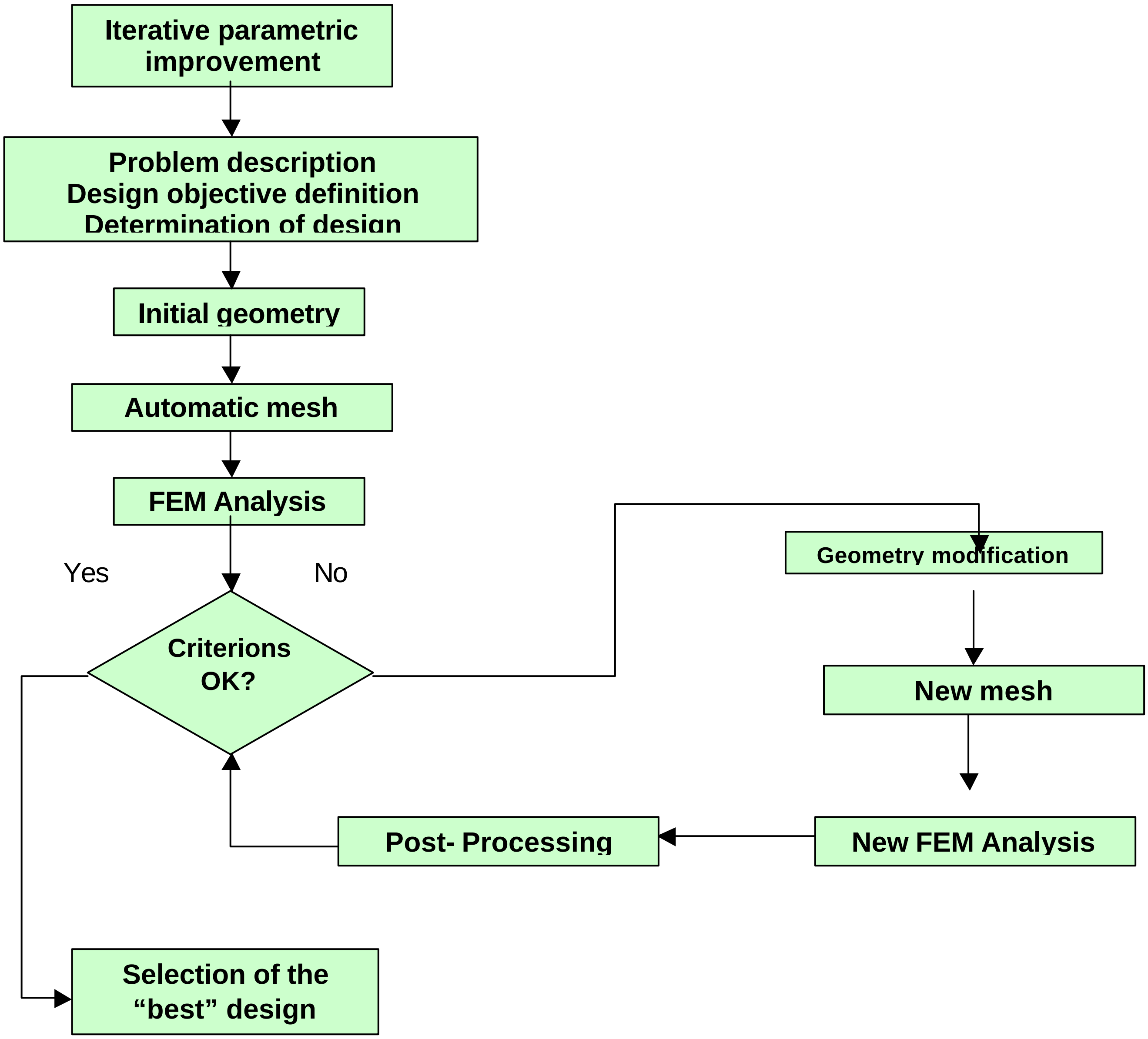

Figure 2. *Iterative parametric improvement at DIRONA´s engineering development process*

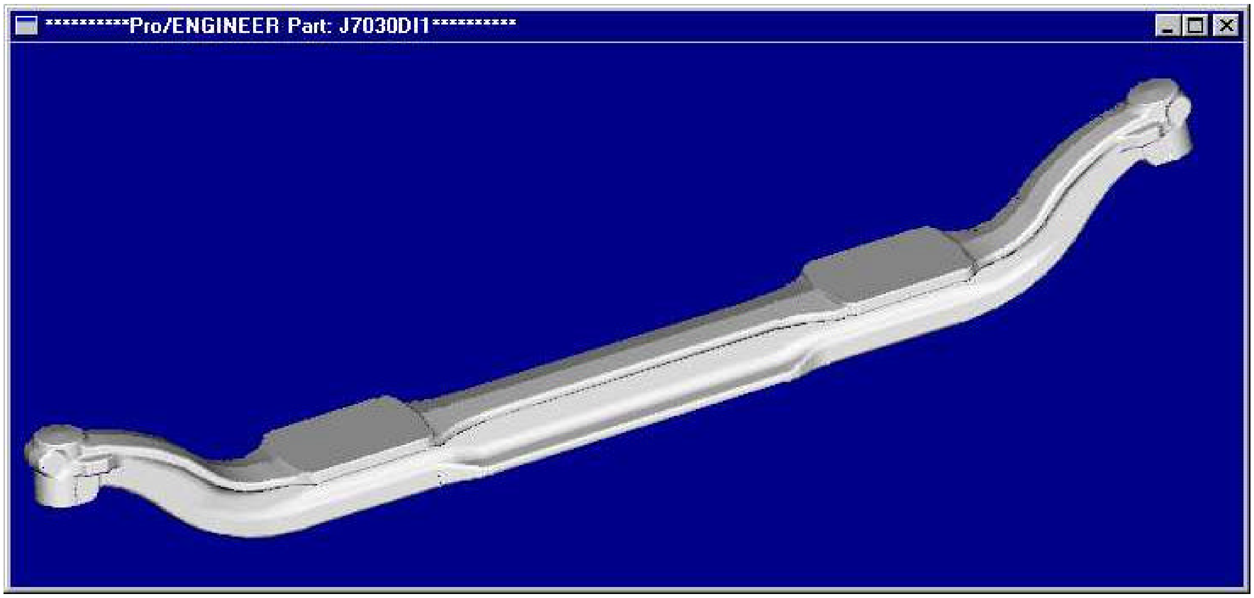

Figure 3. *3D Parametric CAD model of the front axle beam.*

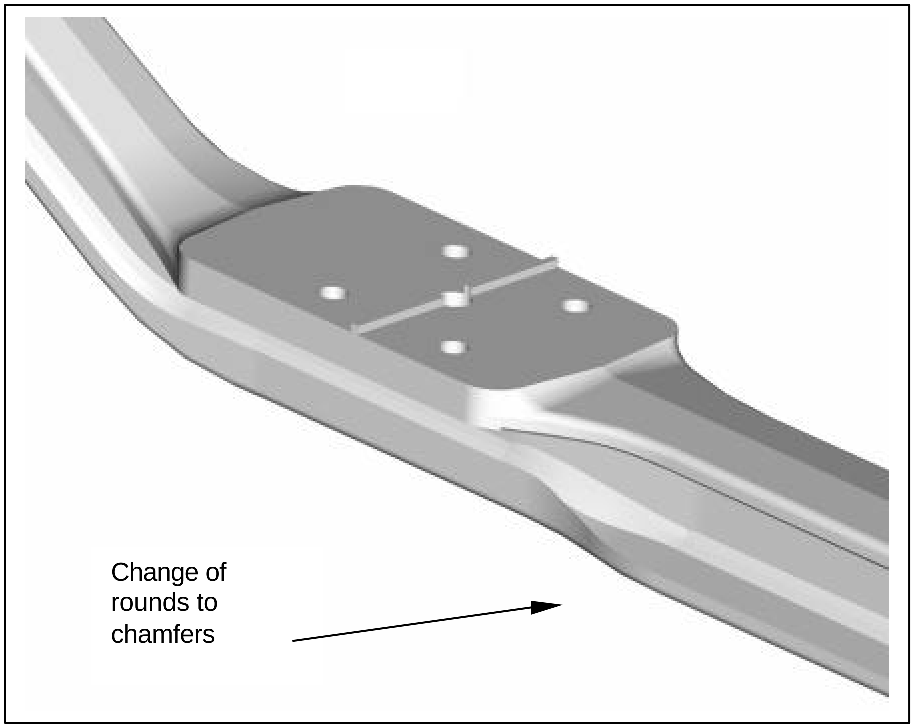

Figure 4. *Simplified CAD model*

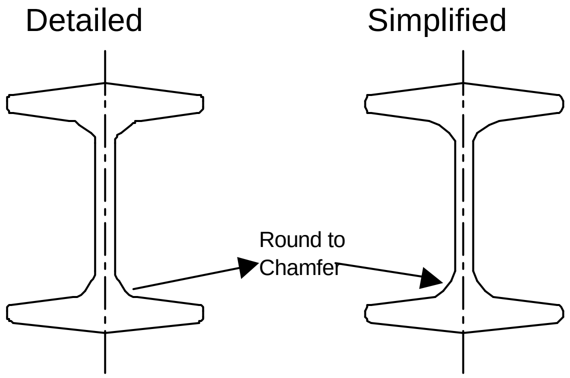

Figure 5. *Model simplification*

The next step was the definition of the boundary conditions, according to the Manager-Test Development manual. The purpose of this analysis was to validate the new design through the simulation of DIRONA´s test procedure (Figure 6).

- Combined vertical and torsion loads.

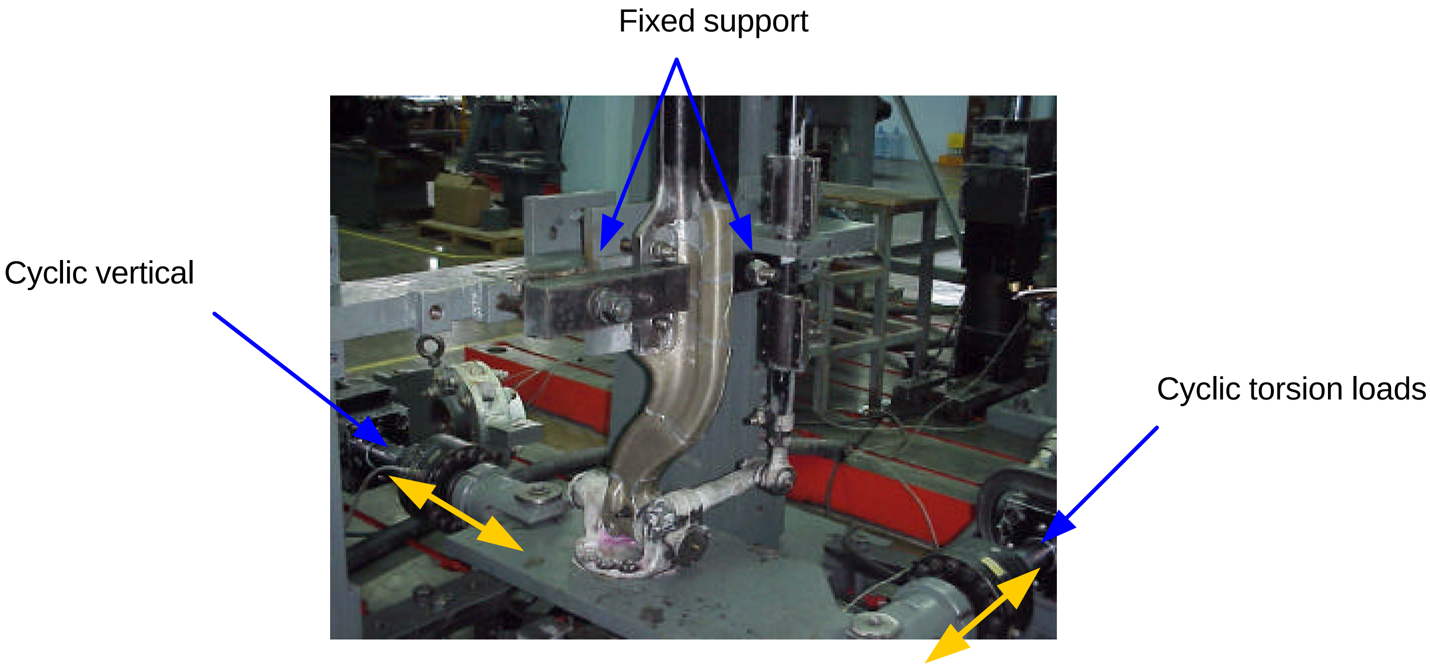

Figure 6. *View of the arrangement for the DIRONA test procedure.*

FEM ANALYSIS

The analyses were carried out using MSC.Patran ver 8.5 on Octane Silicon Graphics computers with 256 Mbytes RAM. The mesh was obtained through automatic mesh generation using lineal tetrahedron. After several sensitivity analyses, elements with average size of 10 mm were used. The first results are given in Table 1 and Figure 7.

In this test, the maximum shear stresses appear on the joint of the neck with the support bushing of the kingpin and near to the spring's base. These may be seen in fFigure 7 in the red circles. As this starting geometry had already been used in former designs, these values may be considered the maximum allowed shear stresses when using the same material. The weight reduction without surpassing these maximal values and, if possible, reducing those stress levels became therefore the intended improvement objectives.

Table 1. *Maximal shear stress values of the previously existing beam*

Torsion case (Stress levels)	105.17 % (1)
Weight	174.71lbs.

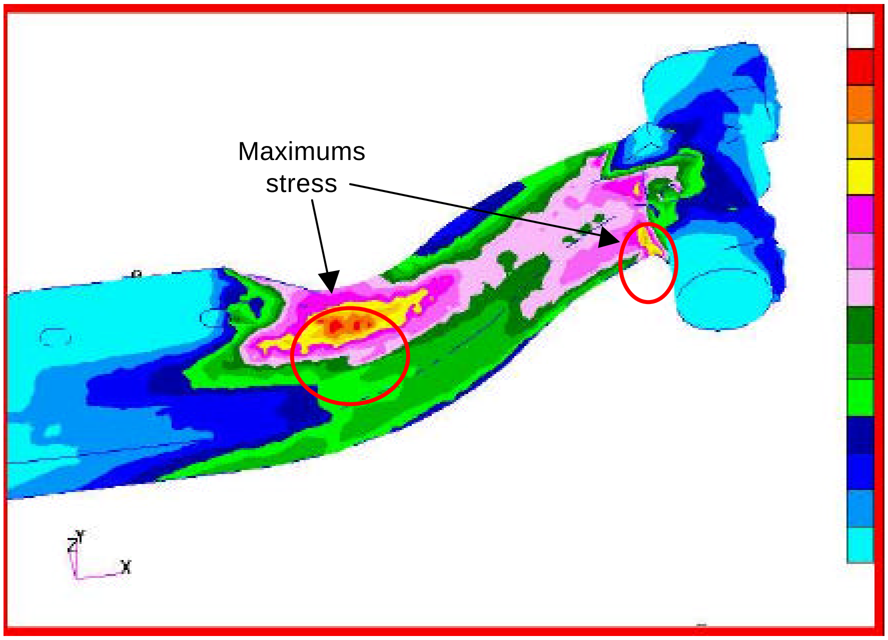

Figure 7. *Shear stress map of the previously existing beam*

---

(1) Stress values given in % of the maximal experimental measured value, which is set as 100 %

Table 2. *Demanded resources for the analysis with MSC.Nastran.*

Requirements	
Hard disk consumed	1.1 GBytes
Convergence	Yes
Order of the polynomial	1° order.
Elements number	183,907 elements

EXPERIMENTAL VALIDATION OF THE FEM ANALYSIS

At this point the results obtained by finite elements were verified experimentally. The stress measurement was carried out using PhotoStress®. PhotoStress is a widely used technique for accurately measuring surface strains to determine the shear stresses in a part or structure during static or dynamic testing. A special, strain-sensitive plastic coating is first bonded to the tested part. Then, as test or service loads are applied to the part, the coating is illuminated with polarized light from a reflection polariscope. When viewed through the polariscope, the coating displays the strains in a colorful, informative pattern, which immediately reveals the overall strain distribution and pinpoints highly strained areas. With an optical transducer (compensator) attached to the polariscope, quantitative stress analysis can be quickly and easily performed. Permanent records of the overall strain distribution can be made by photography or by video recording.

The shear stresses were measured with this method as following: the axle beam was first mounted in the test procedure stand, where vertical and torsion loads were gradually applied as shown in Figure 6. The colorful patterns displayed were recorded by photography (Figures 8 and 9).

Comparing Figure 8 and Figure 7, similarities may be visually perceived with the results obtained with the finite element analysis using MSC.Nastran. The post-processing of the results obtained with this experimental method and its quantitative comparison with the values calculated with FEM show a maximal variation of the 5,17%. This variation depends on the measurement point.

Figure 9 shows two crack initiation places, formed during the fatigue test of the axle beam appearing at the point where the maximal stress values may be seen at the stress map in Figure 7. These cracks appeared after the load cycles established by the test procedure were surpassed, but the test was continued until crack initiation.

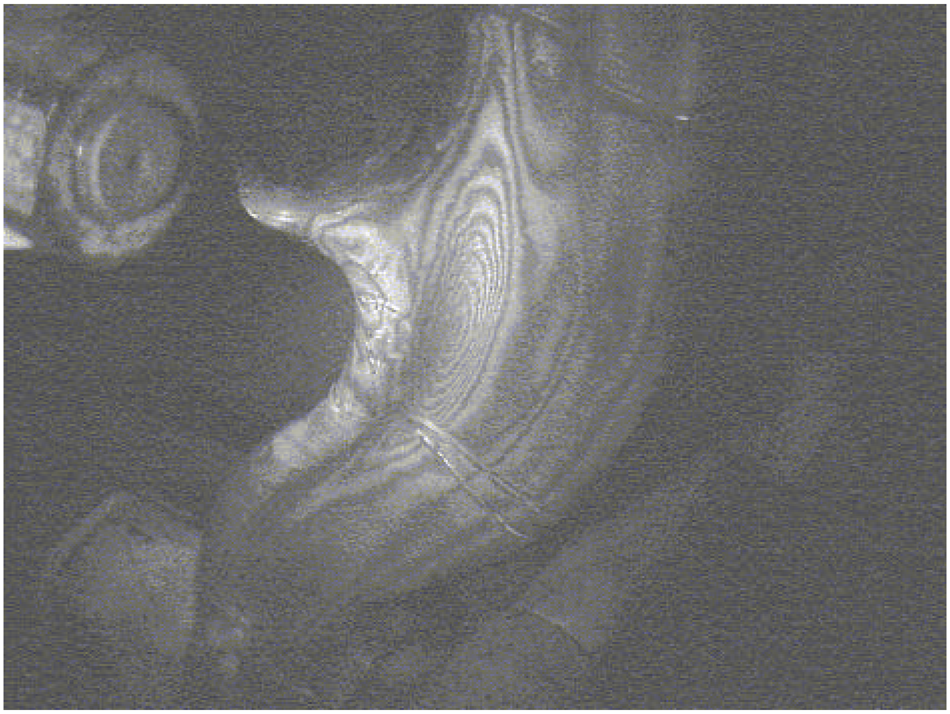

Figure 8. *Colorful pattern obtained with the PhotoStress method.*

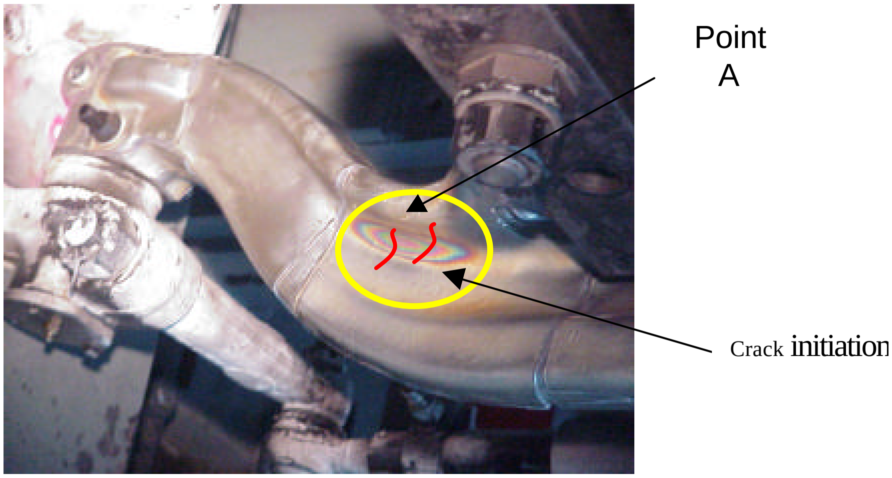

Figure 9. Crack initiation appearing at the places where maximal stresses were calculated

Table 3. *Agreement between FEM calculated and experimental measured values*

Point of measurement	Shear Stress Experimental result **t max (%)**	Shear Stress Analysis result **t max (%)**	Maximum difference
A	100%	105.17%	5.17%

PARAMETRIC IMPROVEMENT

As explained earlier an iterative parametric improvement procedure was applied to reduce the weight of the analyzed beam design. For this purpose, based on the resulting stress maps and on strength of materials science the design parameter were changed, strengthening the places where maximal loads appeared and reducing the sections of places where stresses were very low. Figure 10 shows the geometric features considered to be changed. After nine iterations, the weight was reduced 11.85 pounds, which means 6.8 % of the initial weight. Figure 11, shows a section and the variation applied. The black area shows places where material was removed and the dashed lines represent places were material was added.

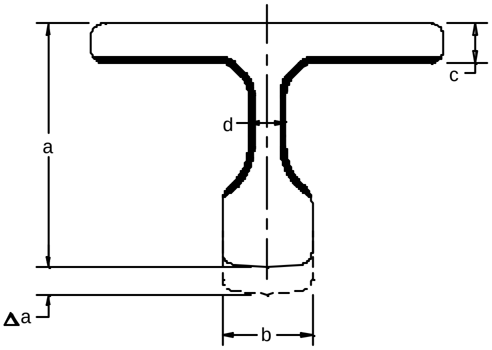

Figure 10. *Design Parameters.*

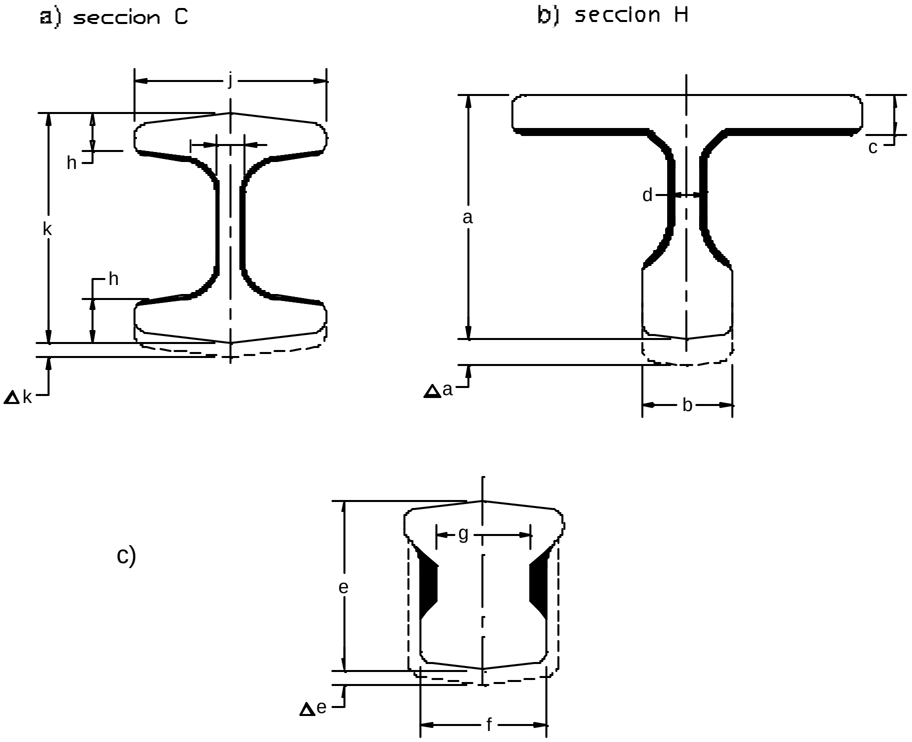

***Figure 11.**. Remove material.*

***Table 4.** Level stress of optimums beam (measure bases was experimental result).*

	Shear Stress Analysis result t **max (%)**	Weight Kg.
Actual case	105.17%	174.71 Kg.
Optimization case	103.05%	162.86 Kg.
Reduction	**2.12%**	**11.85 Kg.**

***Table 5**. Demanded resources for the analysis with MSC.Nastran.*

Requirements	
Hard disk used	0.9 Gbytes
Convergence	Yes
Order of the polynomial	1°
Elements number	183,907

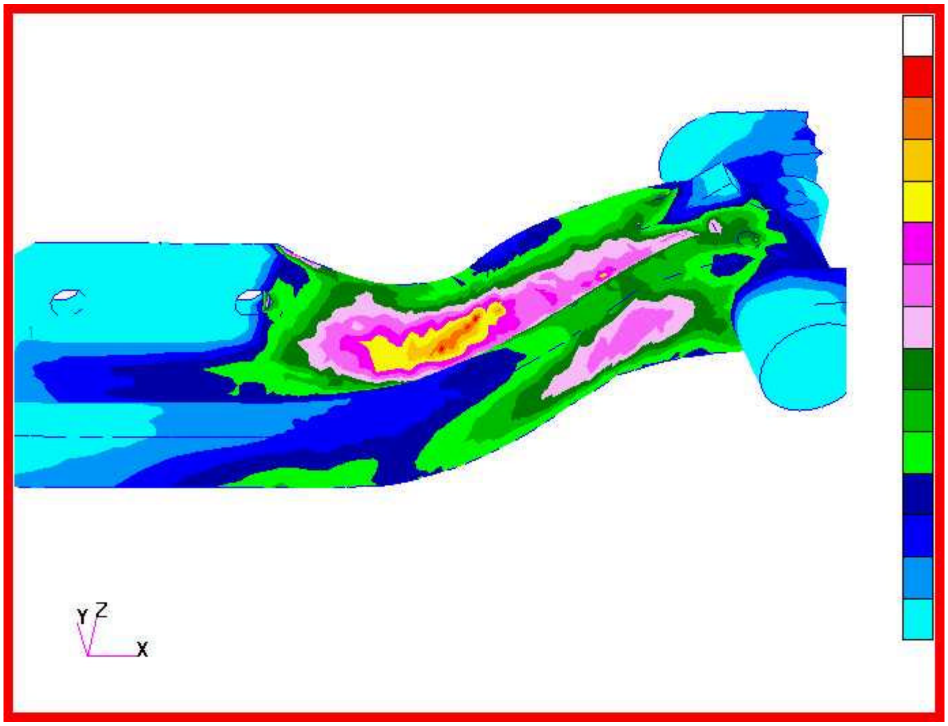

Figure 15. *View of the front axle beam with von Mises stresses.*

At this point further iterations did not show improvements in the objective function and it was decided to interrupt the iterative process.

CONCLUSIONS

The proposed engineering development process at DIRONA proved to be useful in reducing the development time and costs, while maintaining highest product quality and reliability. The introduction of the Hi-Tech CAD/CAM/CAE tools ass MSC.Nastran AND MSC.Patran, combined with the experimental test procedures at DIRONA, allows the experimental validation of FEA results at earlier stages of the product development process, avoiding time loss and rework. A next step will be the introduction of optimization procedures and software in the engineering development process to further increase weight reduction and analysis time.

REFERENCES

[1] Rainer, Gotthard Ph.: “Computer simulations as Parts of the Development Process”, International Symposium Engineering Center Graz, Austria, 25-26 July 1998.

[2] Petrin, H. Wiesler, B. “Application of FEM-Tools in the Engine Development Process”, 1st Worldwide MSC Automotive Conference, Munich, Germany; September 20-22. 1999.

[3] Papalambros Panos Y., Douglas J. Wilde., Principles of Optimal Design, Modeling and Computation. Cambridge University Press., 1988.

[4] MSC.Nastran User`s Manual, Version 66, The MacNeal-Schwendler Corporation, Los Angeles, CA, November 1998.