VON MISES STRESS SENSITIVITY ANALYSIS OF COMPONENTS SHAPED IN CATIA USING VBA PROGRAMMING

Hélio Guerrini Filho, guerrini@ita.br
José Antônio Hernandes, hernandes@ita.br
Rafael Thiago Luiz Ferreira, rthiago@ita.br
ITA - Instituto Tecnológico de Aeronáutica - Pça. Mal. Eduardo Gomes, 50 - 12228-900 São José dos Campos /SP, Brasil

Abstract. *The present work aims at the implementation of finite element sensitivity analysis for components modeled with the CAD/CAM/CAE software CATIA. Performing structural optimization with CATIA using the built-in first order optimization tools, though possible, has many limitations, from inefficiency to lack of convergence. However, the attractiveness for doing structural optimization inside the suite is high, since the user can benefit from its integrated environment. It would be very desirable that optimization could be done using more efficient tools, which could even be fully controlled by the user. With this idea in mind a specific code has been developed in VBA (Visual Basic for Applications) language with the purpose of computing finite element sensitivity analysis of structural responses of parts modeled with the suite. The finite difference sensitivities are carried out with respect to parameters defining component geometry. The paper presents the sensitivity of the von Mises stress with respect to independent modeling parameters to be used as design variables in a complex three dimensional part discretized with solid finite elements. The ability of performing user controlled structural response sensitivity analysis with CATIA opens new perspectives for future implementations of much more efficient structural optimization inside this integrated environment.*

Keywords: *sensitivity analysis, VBA programming for CATIA, structural optimization with CATIA*

1. INTRODUCTION

CATIA is well known as a powerful CAD tool, having the capability for the geometric modeling of virtually any part used in engineering design. It also has a reasonable capability for finite element structural analysis, including shell and solid elements. For instance, after the appreciation of finite element method (FEM) analysis responses, the part can be modified by the engineer with the aim of improving the satisfaction of design requirements. This typical trial and error approach is not efficient for designing because many time consuming stages of geometrical modeling and FEM analysis may be necessary in order to achieve an acceptable design, specially when design requirements are tight.

In a previous article by Hernandes $et\ al.$ (2008) CATIA was evaluated with respect to its efficiency for doing structural optimization by means of its own modules without help of any external programmed interface. In the same fashion, the work Ferreira and Hernandes (2007) evaluates the software efficiency in the optimization of isotropic sheet metal panels whose masses are minimized under fundamental natural frequencies constraints.

In the present work an effort is made to obtain the finite element structural sensitivity analysis of von Mises equivalent stress in some control points defined in the component modeled by CATIA. This sensitivity analysis may open new possibilities for the implementation of more efficient structural optimization tools in the future. To achieve this goal a module was coded using VBA programming language. It is based on von Mises stresses obtained from CATIA FEM analysis.

A new approach for calculating the finite differences sensitivities is suggested in such a way to smooth the values of von Mises equivalent stress aiming to stabilize the function values used with this purpose. This is necessary because in order to get the function derivatives the finite element component model must be remeshed and the new mesh can be different from the original one. Essentially the program perturbs some of the component independent parameters used as design variables, calculating the von Mises stress in the control points in the original and the perturbed designs. This process is repeated for the number of control points defined in the component.

Two examples are presented to illustrate the method. A simple tubular beam of rectangular section discretized with three-dimensional hexahedron finite element and a component with a more complex geometry, similar to some parts found commonly in the industrial environment, discretized with a three-dimensional tetrahedron finite element. In each one, control points are defined and parameterized in agreement with the geometry.

2. VON MISES STRESS SENSITIVITY ANALYSIS IN THE CONTROL POINTS

The filtering of the sensitivity information in topology optimization is highly efficient and ensure mesh-independence (Bendsøe and Sigmund, 2004). It is based on a weighted average of the element sensitivity in a fixed neighborhood or the filter area. The idea of filtering can be applied to other quantities defined over the finite element mesh, which we would like to make smoother or to have more independency from the mesh. In our case these are the von Mises stress in control points over the part discretized with finite elements. Therefore, the von Mises stress in a point located on the component shaped can be obtained from the weighted average of stresses from nodes in a fixed neighborhood. This suggests that the value of the filtered equivalent stress in the control point will be independent of the mesh.

The value of the von Mises stress σ_{pc} in a control point is determined by:

$$\sigma_{pc} = \frac{1}{\sum_{i=1}^{n} H_i} \sum_{i=1}^{n} H_i \sigma_{nd} \tag{1}$$

where σ_{nd} is the nodal stress, n is the number of nodes. The weight factor H_i is given by:

$$H_i = r_{min} - d_{k,i} \tag{2}$$

The radius r_{min} defines the filter area around the control point k and $d_{k,i}$ is the distance between node i and control point k. The operator H_i decays linearly with increasing distance $d_{k,i}$. If $d_{k,i} > r_{min}$ than the operator $H_i = 0$. Therefore the contribution of stresses given by each node decays linearly with distance. Supposing that the control point coincides with a node, then in the limit, when the radius becomes zero, the von Mises filtered stress would match the nodal von Mises stress.

The sensitivity of the von Mises stress in a control point with respect to a design variable is based on the finite differences method. The method requires two structural analysis. One for the initial stress state in the body with its original shape and another for the final stress state in the body with its perturbed shape due to a small change in the design variable. For the perturbed FE analysis a remesh of the body must be performed according to the current variable being perturbed.

Although it is possible to keep the mesh size constant, the new FE mesh will differ from the one of the original body. This means that it is not possible to base finite difference sensitivities directly on the nodal value of a von Mises stress and therefore we have the justification for using the proposed filtering scheme.

The determination of the stress σ_{pc} in the control point is straightforward. The strategy used here is one in which a certain number of FE nodes is stipulated and the minimum radius that contains this number of nodes is used in Eq.(1). This strategy seems to work better than fixing an arbitrary radius. Therefore, the r_{min} is determined by:

$$r_{min} = 1.2 \, d_{k,n+1} \tag{3}$$

where $d_{k,n+1}$ is the distance between the node $n+1$ more close of the control point k. The factor 1.2 is used as a margin to fix r_{min}.

The sensitivity analysis (Haftka and Gürdal, 1992) with respect to the design variables is given by:

$$\frac{\partial \sigma_{pc}}{\partial q_j} = \frac{\sigma'_{pc} - \sigma_{pc}}{\Delta q_j} \tag{4}$$

where σ_{pc} is the initial Von Mises Stress and σ'_{pc} is the final Von Mises Stress after the perturbation Δq_j of the design variable.

2.1 CONTROL POINTS

The control points are positioned in strategic locations where von Mises stress are high. When the body has one of its design variables perturbed, the position of the control point may change. Therefore the position of control points must be parameterized such that it follows the perturbed geometry. For example, considering the Fig.1, lets imagine a control point k defined over a segment between points A and B at position L_{p1} in the line r of length L_1, which represents the original body. In the perturbed body, the point k will assume a proportional location and in the direction of the variation ΔL_1, as in the Fig.1.

The new position of k is given by $L_{p2} = L_{p1}\frac{L_2}{L_1}$. If points k and A have coordinates s_1 and s_A, then the new coordinate s_2 of point k is given by:

$$s_2 = s_1 + \left((s_1 - s_A)\frac{L_2}{L_1} - (s_1 - s_A) \right) \tag{5}$$

where, $L_{p1} = (s_1 - s_A)$. Similar parametrization can be applied to control points defined over surfaces or volumes.

3. COMPONENTS SHAPED UNDER STUDY

Two components are discretized and analyzed by the finite element method.

3.1 COMPONENT 1

The component 1 is a tubular cantilever beam of square section made in aluminum. One vertical load of $F = 2500N$ is applied on the extremity, as shown in Fig.2.

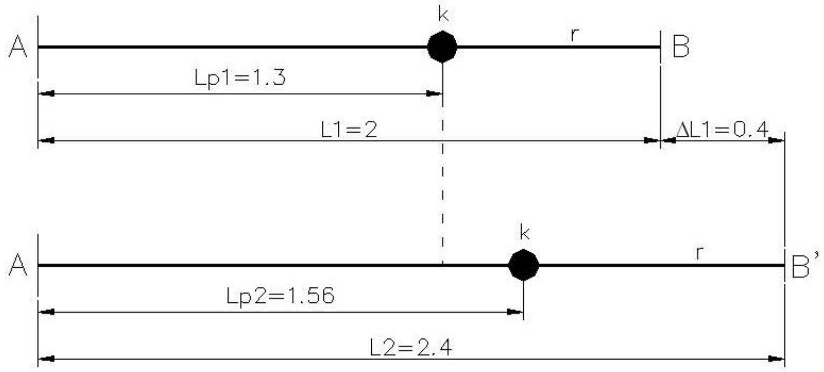

Figure 1. Displacement of the control point k.

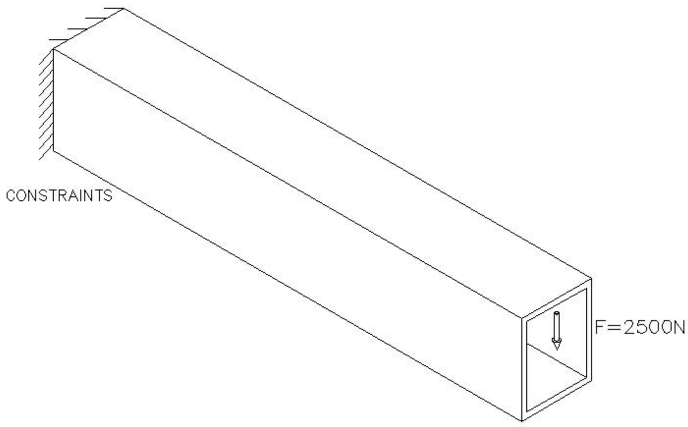

Figure 2. Component 1, load and constraints.

Two control points are defined where the von Mises stress will be computed. The point P_1 is an internal point close to the node where the von Mises stress is higher, the point P_2 is located in the beam web, as depicted in Fig.3.

The design variables T_1 and T_2 increase in the directions d_1 and d_2, respectively, when are perturbed. The relation between T_j for $j = 1, 2$ and points P_k is given by:

$$x_{2,k} = x_{1,k} + \left((x_{1,k} - x_A)\frac{T_2'}{T_2} - (x_{1,k} - x_A) \right) \qquad y_{2,k} = y_{1,k} \qquad z_{2,k} = z_{1,k} + \left((z_{1,k} - z_A)\frac{T_1'}{T_1} - (z_{1,k} - z_A) \right) \quad (6)$$

T_1' and T_2' are the values of the perturbed design variables. The beam is discretized in three-dimensional hexahedron finite element with eight nodes and maximum size equal $6mm$. This mesh is shown in Fig.4.

The finite element uniform meshes are generated by the *Advanced Meshing Tools* and analyzed with the *Generative Structural Analysis*, which are modules internal to CATIA (Dassault Sistèmes, 2005).

For control points P_1 and P_2 the radius corresponding to 10 neighbor FE nodes were established for the sensitivity analysis. Table 1 shows the results obtained for the sensitivity analysis of the von Mises stress for perturbations from 1% to 5% in the design variables.

Table 1. Results of Sensitivity Analysis, Component 1.

						For $\Delta T_j = 5\%T_j$		For $\Delta T_j = 3\%T_j$		For $\Delta T_j = 1\%T_j$	
P_k	T_j	Size of T_j (mm)	r_{min} (mm)	n	σ_{pc} (MPa)	σ'_{pc} (MPa)	$\frac{\partial \sigma_{pc}}{\partial T_j}$ $(\frac{MPa}{mm})$	σ'_{pc} (MPa)	$\frac{\partial \sigma_{pc}}{\partial T_j}$ $(\frac{MPa}{mm})$	σ'_{pc} (MPa)	$\frac{\partial \sigma_{pc}}{\partial q_j}$ $(\frac{MPa}{mm})$
P_1	T_1	6.0	9.230	10	26.660	26.300	-1.205	26.440	-1.216	26.590	-1.239
	T_2	6.0	9.23	10	26.660	26.465	-0.660	26.540	-0.652	26.630	-0.567
P_2	T_1	6.0	9.890	10	7.400	7.328	-0.234	7.356	-0.241	7.385	-0.246
	T_2	6.0	9.890	10	7.400	7.354	-0.153	7.385	-0.0848	7.395	-0.083

It can be seen in Tab.1 that the values of the derivatives seem to converge for the 5% to 1% perturbations. Certainly

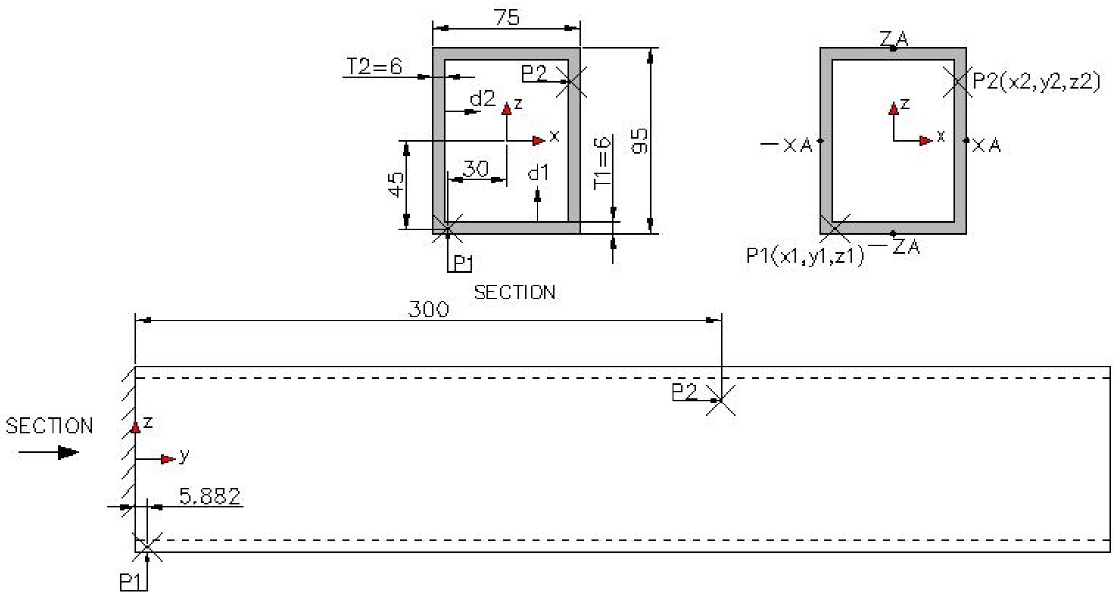

Figure 3. Component 1, 2D views with design variables and control points, units in (mm).

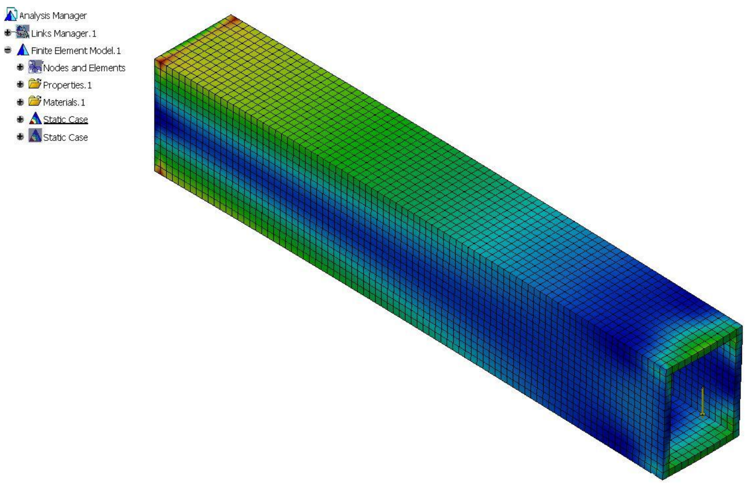

Figure 4. Component 1, showing its hexahedron mesh. The regions in red present the highest stresses.

this nice behavior is influenced by the good quality of the uniform FE mesh obtained with the hexahedron element.

3.2 COMPONENT 2

The component 2 has a much more complex geometry. The idea here is to have a part similar to those found in the industrial environment. Loads and displacement constraints are showed in Fig.5. Again, two control points are used. P_1 and P_2 are fixed in the neighborhood of maximum von Mises stress. There are six design variables, H_1, H_2, E_a and E_b which increase in directions d_1, d_2, d_a and d_b, respectively, when are perturbed. These variables and directions are indicated in Fig.6. The relation between the design variables and control points coordinates are as follows, for $k = 1, 2$:

$$x_{2,k} = x_{1,k} + \left((x_{1,k} - x_{A,k})\frac{E'_a}{E_a} - (x_{1,k} - x_{A,k})\right) \qquad x_{2,k} = x_{1,k} + \left((x_{1,k} - x_{A,k})\frac{E'_b}{E_b} - (x_{1,k} - x_{A,k})\right) \tag{7}$$

$$y_{2,k} = y_{1,k} + \left((y_{1,k} - y_{A,k})\frac{E'_a}{E_a} - (y_{1,k} - y_{A,k})\right) \qquad y_{2,k} = y_{1,k} + \left((y_{1,k} - y_{A,k})\frac{E'_b}{E_b} - (y_{1,k} - y_{A,k})\right) \tag{8}$$

$$z'_{2,k} = H_2 - (H_2 - z_{1,k})\left(\frac{H_2 - H'_1}{H_2 - H_1}\right) \tag{9}$$

$$z_{2,k} = (H_1') + (z_{2,k}' - 2H_1 + H_1')\left(\frac{H_1' - 2H_1 + H_2'}{H_1' - 2H_1 + H_2}\right) \tag{10}$$

where $x_{1,k}$, $y_{1,k}$, $z_{1,k}$ are the coordinates of the points P_1 and P_2 before perturbation and $x_{2,k}$, $y_{2,k}$, $z_{2,k}$ are the coordinates after perturbation. The coordinates $x_{A,k}$, $y_{A,k}$ and $z_{A,k} = z_{1,k}$ belong to points located in the edges of the part as can be seen from Fig.6. Parameters as H_1', H_2', E_a' and E_b' are the variables after perturbation. The Component2 is discretized in three-dimensional tetrahedron finite element with ten nodes and maximum size equal $6mm$. This mesh is shown in Fig.7.

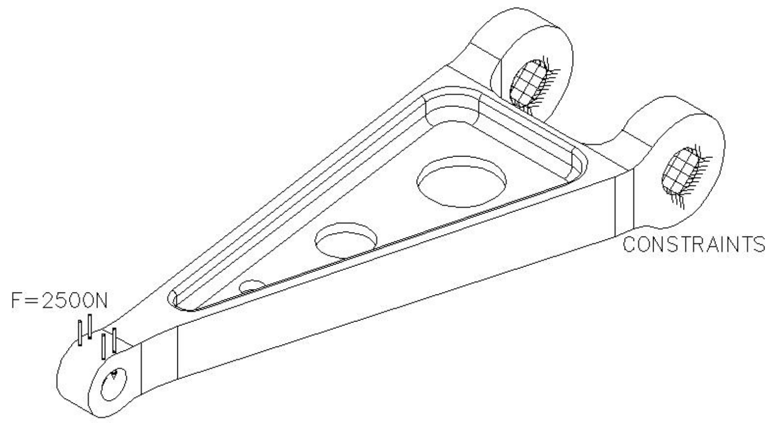

Figure 5. Component 2, load and constraints.

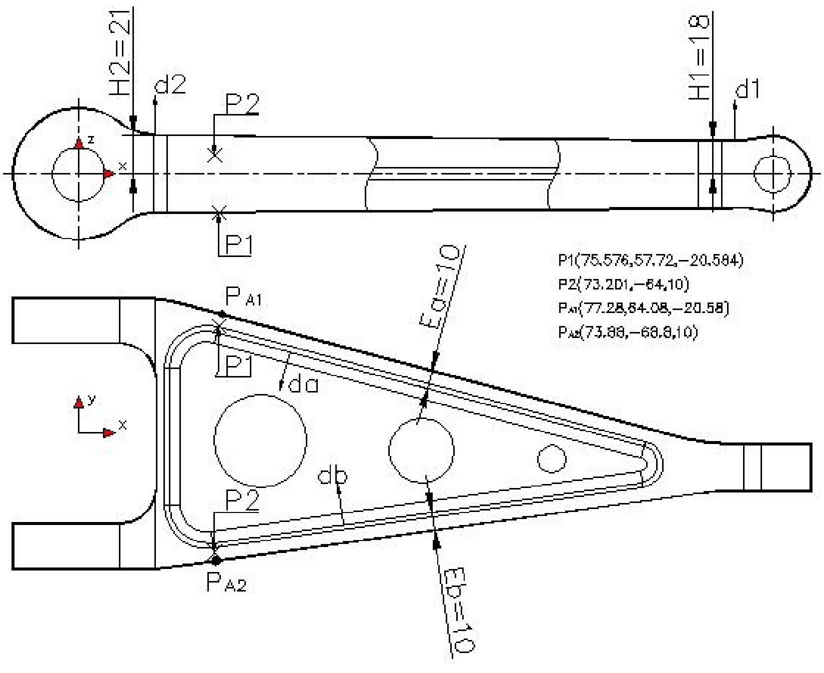

Figure 6. Component 2, 2D views with design variables and control points, units in (mm).

A first positive result from Tab.2 is that the signs of the derivatives are the same for the three perturbations of 5% to 1%. It can be seen in Tab.2 that the values of the derivatives for 5% and 3% are better related than the results of the 3% and 1%. If we assume that the tendency of change from the 5% to 3% is correct, it is hard to accept the changes obtained with the 1% perturbation, with higher jumps and tendency which for some variables are opposite to the one from 5% to 3%. Therefore it would wise to discard the results obtained with the 1% change.

4. CONCLUSIONS

The method proposed for the sensitivity analysis of von Mises stress worked satisfactorily. The von Mises stress sensitivities are calculated based on a weighted average of von Mises stress in a fixed neighborhood. This average was used to obtain the derivatives with respect to design variables by finite differences. Preliminary results for two problems show

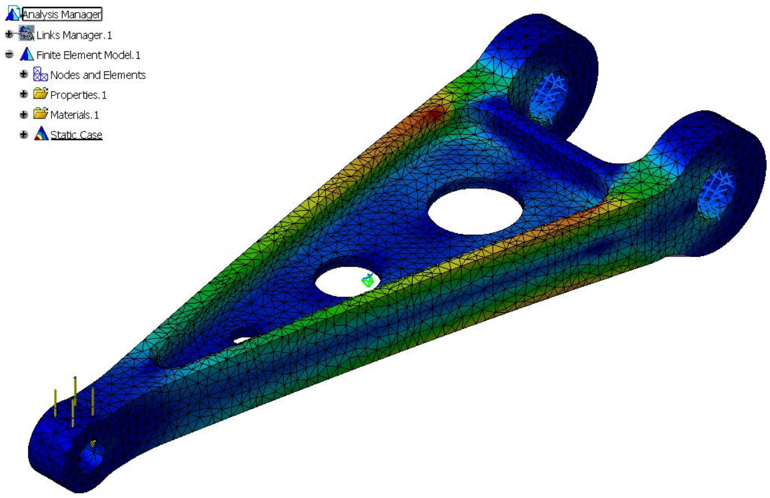

Figure 7. Component 2, showing its tetrahedron mesh. The regions in red present the highest stresses.

Table 2. Results of Sensitivity Analysis, Component 2.

						For $\Delta q_j = 5\% q_j$		For $\Delta q_j = 3\% q_j$		For $\Delta q_j = 1\% q_j$	
P_k	q_j	Size of q_j (mm)	r_{min} (mm)	n	σ_{pc} (MPa)	σ'_{pc} (MPa)	$\frac{\partial \sigma_{pc}}{\partial q_j}$ $(\frac{MPa}{mm})$	σ'_{pc} (MPa)	$\frac{\partial \sigma_{pc}}{\partial q_j}$ $(\frac{MPa}{mm})$	σ'_{pc} (MPa)	$\frac{\partial \sigma_{pc}}{\partial q_j}$ $(\frac{MPa}{mm})$
P_1	H_1	18.00	6.014	20	140.727	140.209	-0.576	140.265	-0.856	140.626	-0.561
	H_2	21.00	6.014	20	140.727	135.682	-4.804	137.303	-5.434	140.0132	-3.397
	Ea	10.00	6.014	20	140.727	138.358	-4.738	138.568	-7.194	140.340	-3.863
	Eb	10.00	6.014	20	140.727	139.498	-2.458	139.914	-2.707	140.061	-6.653
P_2	H_1	18.00	8.330	20	72.450	70.782	-1.853	71.404	-1.936	71.442	-5.601
	H_2	21.00	8.330	20	72.450	69.164	-3.130	68.956	-5.546	70.570	-8.953
	Ea	10.00	8.330	20	72.450	70.125	-4.651	71.271	-3.929	71.238	-12.115
	Eb	10.00	8.330	20	72.450	70.329	-4.241	69.656	-9.314	71.176	-12.738

that the derivatives seem to be stable for different perturbations from 1% to 5%. The results are specially encouraging for the second problem where a very complex mesh of parabolic tetrahedron finite elements was used to discretize a part with a complex shape. The method needs to be further tested and in the future incorporated to an experimental optimization code together with other appropriate techniques.

5. REFERENCES

Bendsøe, M.P and Sigmund, O., 2004,"Topology Optimization: Theory, Methods and Applications", 2nd ed., Springer-Verlag, Berlin, 365p.

Hernandes, J.A., Ferreira, R.T.L., Faria, A.R. and Meleiro, R.M., 2008, "Structural Optimization Using CATIA V5", Proceedings of the XVII SAE Brasil Congress, São Paulo / SP, Brasil.

Hernandes, J.A., Ferreira, R.T.L., 2007, "Structural Optimization of Reinforced Panels Using CATIA V5", Proceedings of the 19th International Congress of Mechanical Engineering, Brasília / DF, Brasil.

Haftka, R.T., Gürdal, Z., 1992, "Elements of Structural Optimization", 3rd Ed., Kluwer Academic Publ., Dordrecht, 481p.

Dassault Sistèmes, 2005, "CATIA V5R15 Generative Structural Analysis Documentation", 1089p.

6. Responsibility notice

The authors are the only responsible for the printed material included in this paper.