Application: AEF-A.10
Optimizing the solutions

KEY WORDS
Linear Static Analysis, Optimization, Linear Material, 2D Geometric Model, 2D Finite Element, Linear Finite Element, Element, Design Parameters, Status Parameters, Objective Function

CONTENT

A. PROBLEM DESCRIPTION
B. THE FEA MODEL
C. PREPROCESSING OF THE FEA MODEL
D. SOLVING THE FEA MODEL
E. POSTPROCESING OF THE RESULTS
F. PREPROCESSING OF THE OPTIMIZATION MODEL
G. SOLVING THE OPTIMIZATION MODEL
H. POSTPROCESING OF THE RESULTS
I. ANALYZING OF THE RESULTS
J. CONCLUSIONS

A. DESCRIEREA PROBLEMEI / PROBLEM DESCRIPTION

A.1 Introduction

In general, the FEA determines values of the output parameters (deformations, displacements, stresses), depending on the preliminary predefined model parameters. Some FEA have distinct optimization modules that for a preliminarily analyzed structure allow the determination of independent parameters, consequence of solving an optimization model that involves minimizing / maximizing some purpose functions while imposing restrictions of other dependent parameters (see Chapter F)

A.2 Application description

For the elaboration of the model of constructive optimization of the fixed bar and loaded with the force F from fig.a consider:

- predefined parameters: L, a, G;
- design parameters (input): D, H;
- state parameter (output): σ_{ech} (equivalent von Mises stress).

Optimization model (see Chapter F)

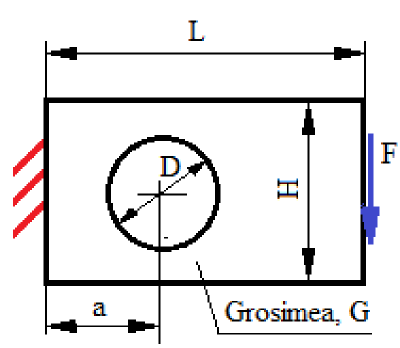
Restrictions:

$D_{min} < D < D_{min}$;

$H_{min} < H < H_{min}$;

$\sigma_{ech} < \sigma_a$ (the permissible stress imposed).

Objective function: Mass $\rightarrow$ min.



A.3 The application aim

This application presents, using finite element analysis, the algorithm for solving the problem of dimensional constructive optimization of the structure in the figure above. For preliminary FEA we consider: L = 50 mm, H = 40 mm, G = 10 mm, a = 20 mm. The values of the optimization model parameters are: D_{min} = 14 mm, D_{max} = 18 mm, H_{min} = 35 mm, H_{min} = 44 mm, σ_a = 140 Mpa.

B. THE FEA MODEL

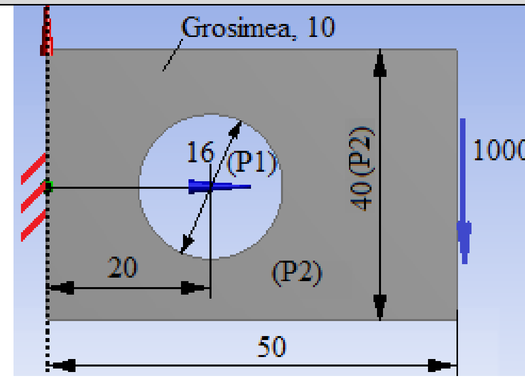
B.1 The model definition

For the analysis and optimization with FE, the following simplifying hypotheses are adopted:

- linear behavior of the material,
- adopting constraints associated with symmetry properties,
- external load by force distributed on the surface,
- the proposed problem is solved in two stages: structural analysis and optimization.

B.2 The analysis model description

Figure a shows the FEA and optimization model associated with the geometric plane model considered in the XY plane. The X-axis is the axis of symmetry of this model. In addition, the design parameters: hole diameter (P1) and width (P2) are also highlighted for optimization.



B.3 Characteristics of the material and the environment

The strength characteristics of E335 material for finite element analysis are:

- the modulus of longitudinal elasticity, E = 210000 N / mm^2;
- transverse contraction coefficient (Poisson), ν = 0.3.
- Average working temperature of the subassembly, $T_0 = 20^0$ C.

C. PREPROCESSING OF FEA MODEL

C.1 Activarea şi salvarea proiectului / Creating, setting and saving the project

Creating of the project

Toolbox: ↵⊟ Analysis Systems: ↵↵ Static Structural (the subproject window appears automatically - the name can be changed to *Optimization*).

Setting the problem type (3D)

A: ↳ Geometry → ↵Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↵Analysis Type, [select from list ↵▾, ↵3D] → [close the window, ↵✕].

Saving of the project

↵ Save As... → Save As, File name: [input name, AEF-A.10] → ↵ Save.

C.2 Modelling of material and environment characteristics

, Project Schematic: ↳ Engineering Data ✓ ◢ → ↵ Edit... → Outline of Schematic A2: Engineering Data:

↵ Structural Steel, Properties of Outline Row 3: Structural Steel: ⊟ Isotropic Elasticity → Young's Modulus, [select from list, C column(Unit) cu ↵▾, ↵MPa], [input in the box in column B (Unit) value, 210000] → ↵ Update Project → ↵ Return to Project (the other parameters remain the default).

C.3 Geometric modelling

C.3.1 Model loading, DesignModeler (DM)

↵↵ Geometry ? ◢ → ↵ New Geometry... → ANSYS Workbench: ↵ Millimeter, ↵ OK.

C.3.2 Sketch generation

Viewing default plane (XY)

DM, Tree Outline: ↲ Sketching → (Look at face/Plane/Schetch) , [automatically view of default plane XY Plane] ;

Generating of rectangle

↲ Draw → ↲ Rectangle → [the rectangular line is generated in quadrants I and II by marking ↲ the corner on the Y axis (the coincidence symbol C appears) and the release ↲ in the opposite corner in the other quadrant, fig. a].

Generating of circle

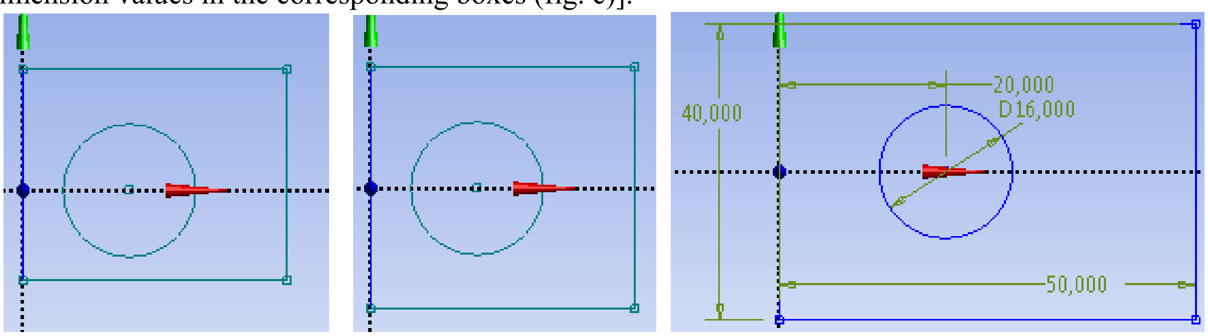
↲ Circle → [a circular line is generated by marking ↲ the center on the X axis (the coincidence symbol C appears) and the release ↲ on the contour, fig. a].

Symmetry constraint

↲ Modify → ↲ Constraints → Symmetry → [is activated / deactivated with ↲ option Ignore Axis (☑/☐]) → [is marked with ↲ the X axis of symmetry and with Ctrl + ↲ the horizontal lines] (the symmetrical model with respect to the X axis appears automatically, fig. b).

Dimensioning the sketch

↲ Dimensions → ↲ Semi-Automatic → [is activated with ↲ dimensions] → Details View, ⊟ **Dimensions: 4** : [enter the dimension values in the corresponding boxes (fig. c)].



a. ***b.*** ***c.***

C.3.3 Generating the solids object

DM: ↲ Extrude → Details View, ⊟ **Details of Extrude1**: Geometry , → ↲ Sketch1 → ↲ Apply; ☐ FD1, Depth (>0) , [input the depth, 10)]. ↲ Generate

C.3.4 Saving of geometric model

DM: ↲ (Save Project) → ↲ (Close).

C.4. Finite element modelling

C.4.1 Launching of the finite element modeling module

Launching of the finite element modeling module

, Project Schematic: Model → ↲ Edit... → [launching *Mechanical [ANSYS Multiphysics*].

Setting the unit of measure system

M: ↲ Units → ↲ Metric (mm, kg, N, s, mV, mA) (setting the UM system).

Setting the material characteristics

Outline: ↲ ⊞ Geometry → ↲ Solid → Details of "Solid", **Material** : ↲ Assignment → [select from list ↲, ↲ Structural Steel (default)];

C4.2 Meshing the model

Mesh → Generate Mesh (meshing with default parameters, fig. C.6,a).

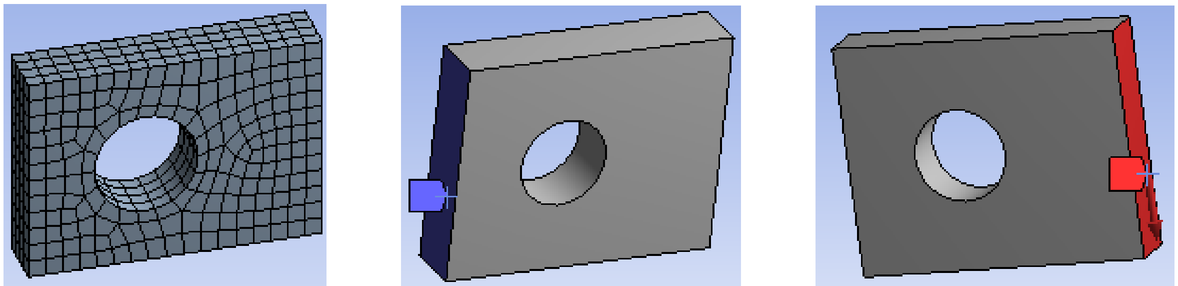
C.5 Supports and restraints modelling

Generation of embedding constraint

↲ **Static Structural (B5)** → ↲ Supports → ↲ Fixed Support; → ↲ (the line selection filter is activated) → [select left face (fig. C.6,b)];

C.6 Modelarea încărcării / *Loads modelling*

↵ Static Structural (B5) → ↵ Loads → ↵ Force ; → ↵ (the line selection filter is activated) → [select right face (fig. c)]; ,Details of "Force", ⊟ Scope: Geometry → [select right face (fig. b)] → ↵ Apply; ↵ Define By, [select from list ↵], ↵Components ; Y Component, [input Y value, -1000];



a. *b.* *c.*

D. SOLVING THE AEF MODEL

D.1 Setting the results

<u>*Selecting the total displacement*</u>

M, Outline: ↳ ⊞ Solution (A6) → ↵ Insert → ↵Deformation → ↵ Total.

<u>*Setting the equivalent stress*</u>

↳ Solution (A6) → ↵ Insert → ↵ Stress → ↵ Equivalent (von-Mises).

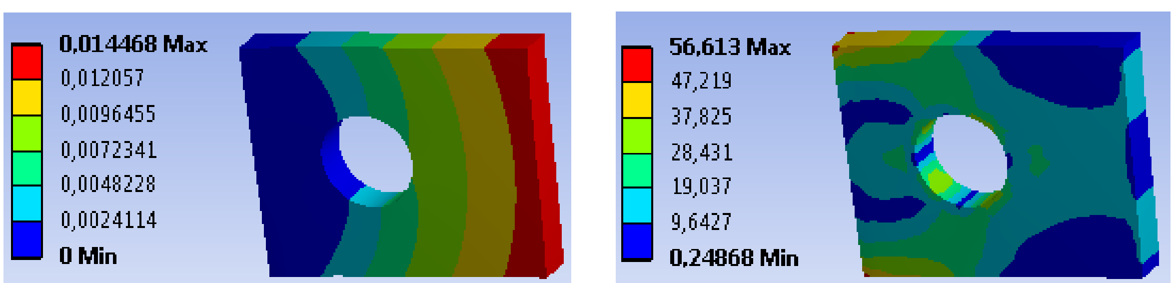
D.2 Lansarea modulului de rezolvare a modelului / Launching the solving module

↵ Solution (A6) → ↵ Solve.

E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement and equivalent stresses fields

M, Outline: ↵ ⊞ Solution (A6) → ↵ Total Deformation (fig. a); ↵ → ↵ Show Undeformed WireFrame (fig. a); ↵ Equivalent Stress (fig. b) Graph; ↵ Animation (view animation).



a. *b.*

F. PREPROCESSING OF THE OPTIMIZATION MODEL

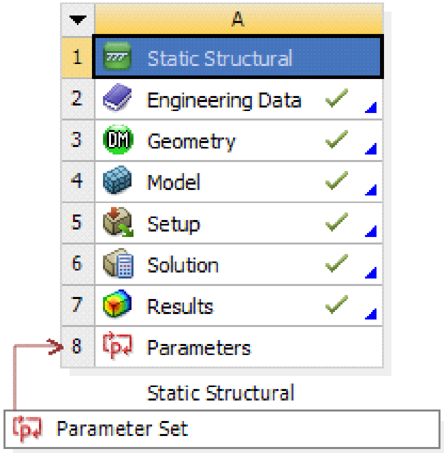
F.1 Setting input (design) and output (status) parameters

<u>*Setting input (design) parameters*</u>

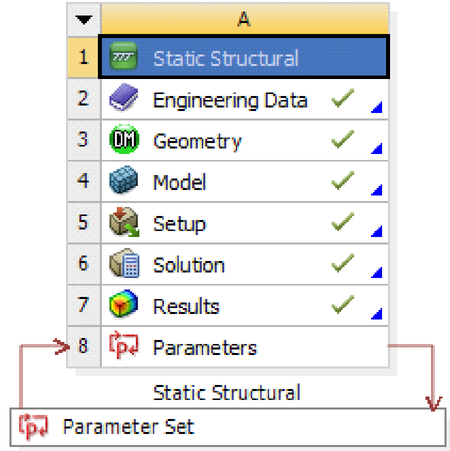
DM: ↵ Sketch1 → Details View, ⊟ Dimensions: 4: [activated with ↵ the button associated with the circle diameter, ↵ D D1] → A: Static Structural - DesignModeler, Parameter Name:, [input the name, *Diameter*], ↵ OK; [activated with ↵ the button associated with the rectangle width dimension, D L2] → A: Static Structural - DesignModeler, Parameter Name:, [input the name, *Width*], ↵ OK (↵ → Project Schematic : (the input parameter setting loop appears automatically, fig. a).

Setting output (status) parameters

Outline: → ↵ Geometry → ↵ Details of "Geometry", ⊞ Properties, [is activated with ↵ the button associated with the mass, P Mass]; ↵ Solution (A6) → ↵ Equivalent Stress → Details of "Equivalent Stress", ⊟ Results, [is activated with ↵ the button associated with the maximum equivalent voltage, P Maximum], ↵ OK (↵ → Project Schematic: (the output parameter setting loop appears automatically, fig. b).



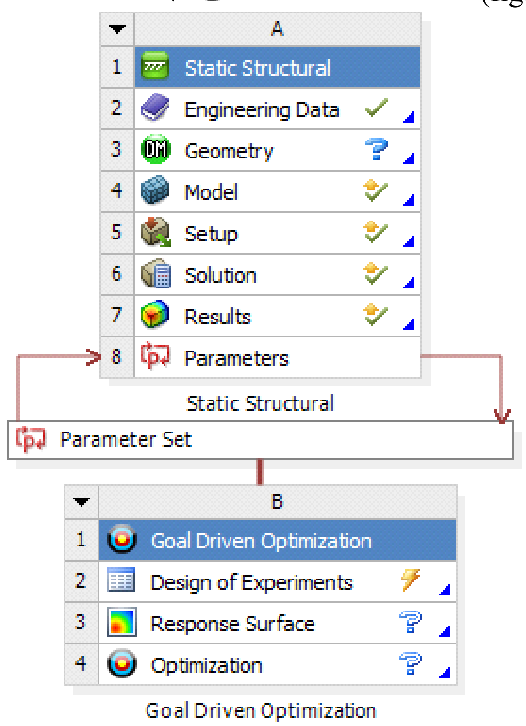
a.



b.

F.2 Launching the optimization module

↵ → ↵ ⊞ Design Exploration → ↵ Goal Driven Optimization (fig. a).



a.

F.3 Generating and visualizing feasible solutions

Generating feasible solutions

↵ Design of Experiments → Outline of Schematic B2: Design of Experiments: (↵ View → ↵ Outline, ↵ Properties, ↵ Table, ↵ Chart), ↵ P1 - Diametrul → Properties of Outline A5: P1: Lower Bound, [input the lower value, 14,4], Upper Bound [input the upper value, 17,6]; ↵ P2 - Latimea Properties of Outline A6: P2: Lower Bound, [input the lower value, 36], Upper Bound [input the upper value, 44]. ↵ Update, ANSYS Workbench → ↵ Yes (after processing the table with test variants appears, Table of Schematic B2:). ↵ Return to Project

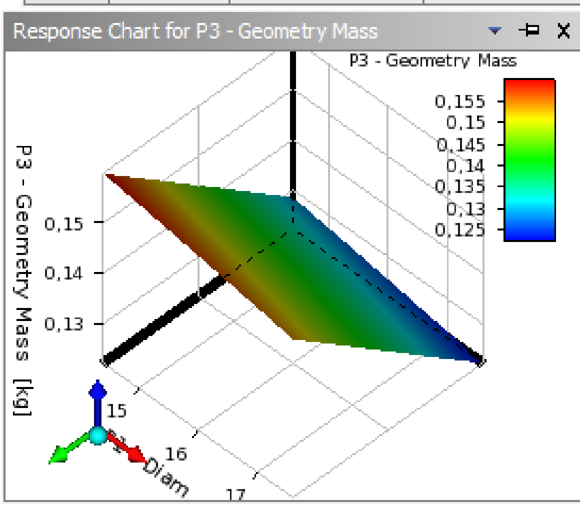
Visualization of feasible solutions

↵ Response Surface → Outline of Schematic B3: Response Surface, ↵ ⊟ Response Surface → ↵ Update. Outline of Schematic B3: Response Surface → ↵ Response, Properties of Outline A16: Response, Mode, [select from list ↵

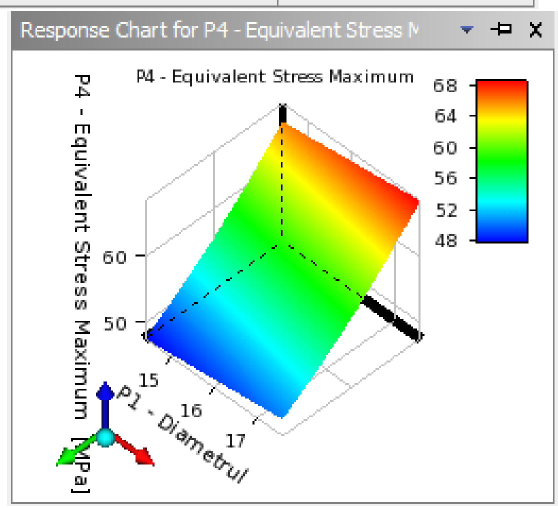
▼, ↵3D]; ⊟ Axes, X Axis, [select from list ↵▼, ↵P1 - Diametrul]; Y Axis, [select from list ↵▼, ↵P2 - Latimea]; Z Axis, [select from list ↵▼, ↵P3 - Geometry Mass] (appear the graph in fig.a); Z Axis, [select from list ↵▼, ↵P4 - Equivalent Stress Maximum] (appear the graph in fig.b).
↵ Return to Project.

Table of Schematic B2: Design of Experiments (Central Composite Design : Auto Defined)

	A	B	C	D	E
1	Name	P1 - Diametrul	P2 - Latimea	P3 - Geometry Mass (kg)	P4 - Equivalent Stress Maximum (MPa)
2	1	16	40	0,14122	56,613
3	2	14,4	40	0,14422	55,763
4	3	17,6	40	0,1379	58,223
5	4	16	36	0,12552	67,193
6	5	16	44	0,15692	49,085
7	6	14,4	36	0,12852	65,788
8	7	17,6	36	0,1222	68,589
9	8	14,4	44	0,15992	47,467
10	9	17,6	44	0,1536	50,114



a.



b.

G. SOLVING THE OPTIMIZATION MODEL

↵ → B: ↵ Optimization → Table of Schematic B4: Optimization, ⊟ Optimization Objectives: Objective, [select in column D from list ↵▼, ↵Minimize], [select in column E from list ↵▼, ↵Values <= Target]; Target Value, [input in column D the value limit, 140]. Properties of Outline A2: Optimization, ⊟ Optimization: Optimization Method, [select from list ↵▼, ↵NLPQL]. Outline of Schematic B4: Optimization → ↵⊟ Optimization. ↵ Update (appear in window Table of Schematic B4: Optimization lines from fig.a). ↵ Return to Project.

11	⊟ Candidate Points				
12	Candidate A	17,6	36	★★★ 0,1222	★★★ 68,565
13	Verification A			★★★ 0,1222	★★★ 68,589

a.

Obs. The NLPQL (Nonlinear Programming by Lagrangean Quadratic) method is based on the gradient algorithm for models with a single objective function and multiple constraints.

H. POST-PROCESSING OF RESULTS

H.1 Update the original model with the optimal design values

Entering the values of the optimal design parameters

↵ → ↵↵ Parameter Set → Table of Design Points: Current, [input in column B the optimal value, 17,6 (see the table above)], [input in column B the optimal value, 36 (see the table above)]. ↵ Update (the boxes in columns D and E are filled in automatically).

Table of Design Points

	A	B	C	D	E	F	G
1	Name	P1 - Diametrul	P2 - Latimea	P3 - Geometry Mass	P4 - Equivalent Stress Maximum	Exported	Note
2	Units			kg	MPa		
3	Current	17,6	36	0,1222	68,589		

a.

Checking the values of the optimal design parameters

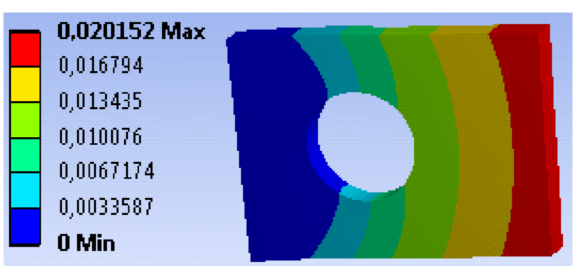
↵ → ↵ Geometry ✓ → ↵ Sketching (the values of parameters D1 (Diameter) and L2 (Width) updated with the optimal values are observed, 17.6 and 36 respectively).

Upgradare proiect

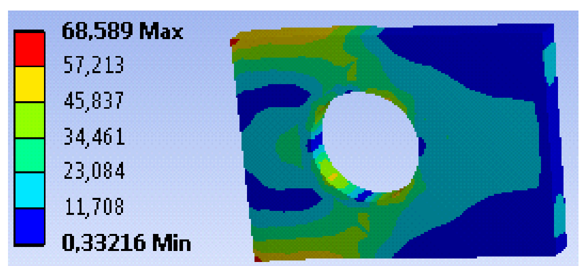
↵ Update Project.

H.2. Visualization of the field of displacements and equivalent post-optimized stresses

Outline: ↵⊞ ? Solution (A6) → ↵ ✓ Total Deformation (fig. a); ↵ → ↵ Show Undeformed WireFrame (fig. a); ↵ ✓ Equivalent Stress (fig. b) Graph; ↵ Animation (view animation).



a.



b.

I. ANALYSIS OF RESULTS

I.1 Interpretation of results

Following the analysis of the results obtained, as a result of the modeling and post-processing of the results (subchapters E and H), the following are highlighted:

- Following the deformation process of the non-optimized element (D = 16 mm, H = 40 mm) as a result of the action of the force F (subchapter A.2, fig. a) the maximum displacement is observed 0.0144468 mm (subchapter E. 2, Fig. a) in the area of the force action; the maximum equivalent stress has the value 56,614 MPa (subchapter E.2, fig. b) in the embedded area; the mass of the element is 141.22 g (subchapter F.3, Table of Schematic B2).
- Following the deformation process of the optimized element (D = 17.6 mm, H = 36 mm) as a result of the action of the force F (subchapter A.2, fig. a) the maximum displacement is observed 0.020152 mm

(subchapter. H.2, Fig. a) in the area of the force action; the maximum equivalent stress has the value 68.589 MPa (subchapter H.2, fig. b) in the embedded area; the mass of the element is 122.2 g (subchapter H.3, fig. a).

I.2 Design studies

The analysis of the above results shows the decrease of the element mass following the finite element solving of the optimization model; at the same time the increase of the maximum displacement (rigidity) is observed.
In order to optimize related to other design restrictions, it is necessary to modify the analysis model, re-adopt the design and status parameters and the objective function. Thus, it is necessary, after the modifications of the analysis and / or optimization model, to solve it by activating the commands ↵ Refresh Geometry; ↵ Solve. After solving the model, the results are reanalyzed and reinterpreted.

J. CONCLUSIONS

Modeling and analysis with finite elements in this paper were also carried out for didactic purposes following the initiation of the user with the main stages of development of a finite element optimization application in ANSYS Workbench, which emphasizes, above all, the modeling and analysis of a deformable element which is then dimensionally optimized.
The optimization model considered adopted involves the consideration of two geometric parameters as design variables, a state parameter (equivalent voltage) limited below the allowable value and the objective function that involves minimizing the mass of the element.
Following the solution of the finite element model of optimization, adopting the NLPQL method (Nonlinear Programming by Quadratic Lagrangean) which is based on the gradient algorithm for models with a single objective function and multiple constraints, the reduction of the element mass was obtained. maximum (but not exceeding the allowable value) and increasing the rigidity of the element.