

# An optimization approach by Finite Element Analysis, using Design of Experiments and Response Surfaces- a survey

D C Negrău<sup>1,2</sup>, G Grebenișan<sup>1</sup> C Gherghea<sup>1</sup>

<sup>1</sup> University of Oradea, Faculty of Managerial and Technological Engineering,  
Universității Street No. 1, Oradea, Romania

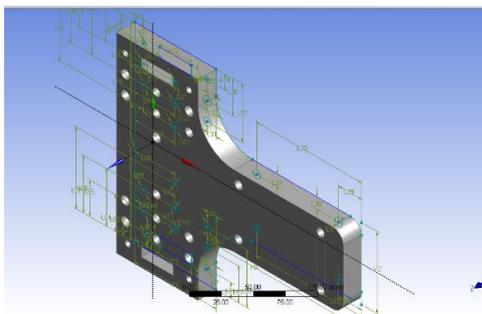
<sup>2</sup> [dan.negrau@yahoo.com](mailto:dan.negrau@yahoo.com)

**Abstract.** The deformation of an aluminum plate, called in this paper, a support plate, which is part of a bending plastic deformation device, is studied in this paper using the Finite Elements Method, respectively design points of the design by optimization using features such as Design of Experiments and Response Surfaces. The start of the analysis was determined by a functional requirement in the design and construction of a sophisticated bending device controlled by steel plates for the production of exhausts for cars. To optimize the project, it was necessary to analyze the behavior of the active material and components of the device. The analysis of material and piece behavior, in working conditions, has been developed by addressing the facilities offered by Ansys software. Following FEA analysis, optimizing the active components of the device has provided proven results in practice through the execution and operation of the device, according to the technology required by the product designer.

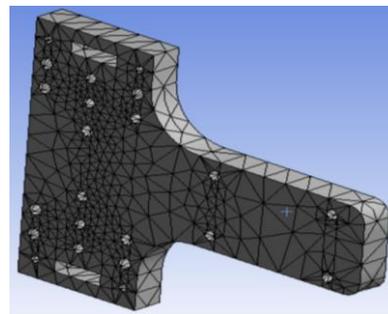
## 1. Introduction

In this research paper is presented the optimization of a support plate by modifying material and thickness, the plate which is part of a subassembly, the subassembly which is part of a sheet metal bending machine [1].

The support plate was built on the customer's technical prescriptions, so that it has the dimensions shown in figure 1 a). The material used is duraluminium, a material that meets the technological requirements, namely: it is easy to milling, low specific weight, insignificant cutting tool wear compared to the machining of a harder material (soft steel, stainless steel) and the production time is short [2].



a) 2D support plate



b) Mesh structure (9374 nodes, 5041 elements)

**Figure 1.** Geometric properties

## 2. Optimizing using Design Optimization Experiments (DOE)

The DesignXplorer module (from Design Optimization Exploration- DOE) proposes two optimization workflows: 1) - Response Area Optimization (RSO) based on DOE and Response Area; 2) Direct optimization. The behaviors of the optimization algorithm are identical for Response Area Optimization (RSO) and Direct Optimization (there are some exceptions). The default settings may differ for RSO or Direct Optimization. For RSO, output parameter outputs are retrieved in RS evaluations (very quickly). For Direct Optimization, the results of output parameters are found in the actual resolutions.

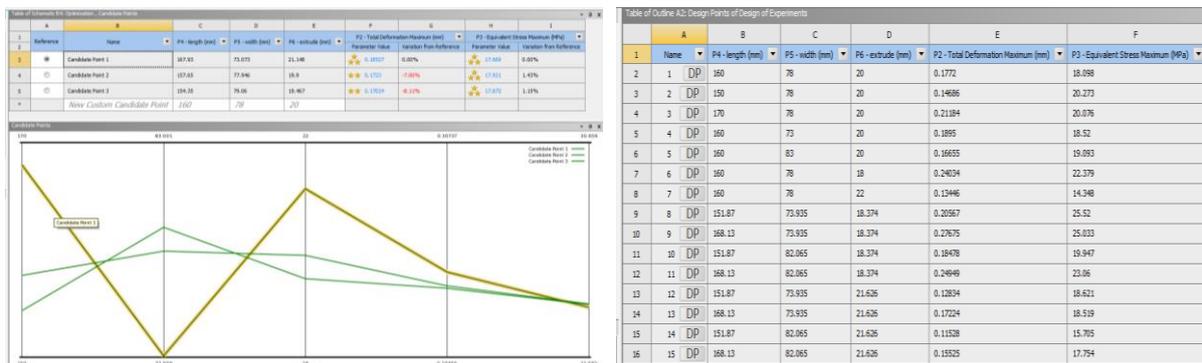
Work plan:

- A. Establishing the optimization area (Limits and types of parameters (manufactured / discrete values));
- B. Input Parameters Relationship (Eliminate Needlessness / Achievable Combinations of Input Values)
- C. Objectives and constraints (minimizing or maximizing some limits);
- D. Optimization methods (there are 6 optimization methods in DX:
  1. Screening (Shifted Hammersley) [implicit mode]
  2. MOGA (Multi-Objective Genetic Algorithm)
  3. NLPQL (Non-linear Programming by Quadratic Lagrangian)
  4. MISQP (Mixed-Integer Sequential Quadratic Programming Method)
  5. Adaptive Single-Objective(just for direct optimization)
  6. Adaptive Multiple-Objective (just for direct optimization)
- E. Convergence and post-processing

For all types of optimization, the history of each parameter is reported by a curve, the convergence criteria are also represented graphically.

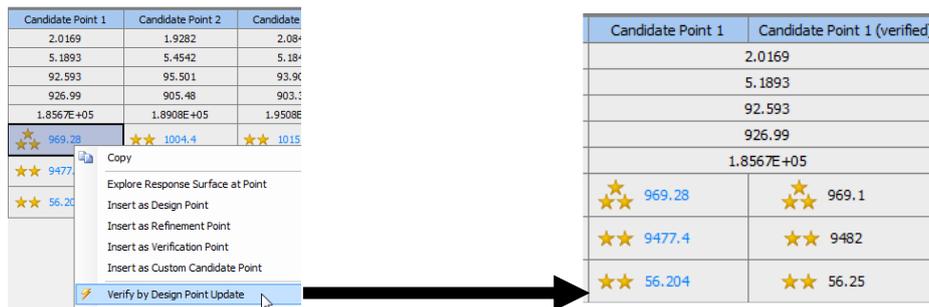
### 2.1 Optimization of design points (project) - proposed variants

Design points are generated according to the objectives and constraints of the system, depending on the user's settings. Out of the generated design points and for which the output parameter values were calculated, three points are generally selected that have the most convenient output parameter values, these points being candidate points. Among the candidate points, the user chooses the optimal option to continue the project. The design point data (figure 2), which has become the current point of the project, is inserted into the project input data, these being the key features on which the optimized project was created.



**Figure 2.** The candidate points and design points –follow in choosing as a Design point

Design points, verification (for optimizing the response area). RSO design points (figure 3) are in fact approximations of the response surface. The algorithm verifies the design points by creating and updating the output parameters with a "real solution" using the input points of the design points [3].



**Figure 3.** Design point in RSO

The output resolution parameter of the actual resolution is accordingly displayed below the response surface output values to allow easy comparison. If the results are not the same, it indicates that the response area is not entirely accurate in that area and refining or other adjustments may be necessary. It is possible to insert design points as well as a refinement point.

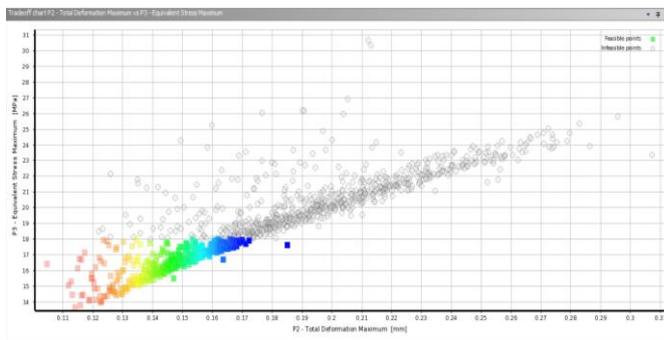
Optimization requirements:

- At least one of the output parameters should confirm an objective (Maximize, Minimize, Seek Target to run an optimization with MOGA, AMO, NLPQL, MISQP, and ASO methods.) If this is not done then the optimization problem is either undefined (without objective) or merely a problem of the satisfaction of the constraints (Objective set to  $> =$  Target or  $< =$  Target) When the problem is not defined, MOGA or NLPQL analyzes cannot run. AMO and MISQP.
- The Screening method does not depend on any parameter setting and can be used to perform preliminary design studies.
- External optimizations can be connected to DX via ACT Extensions. They can be written in IronPython, C++ or C#.

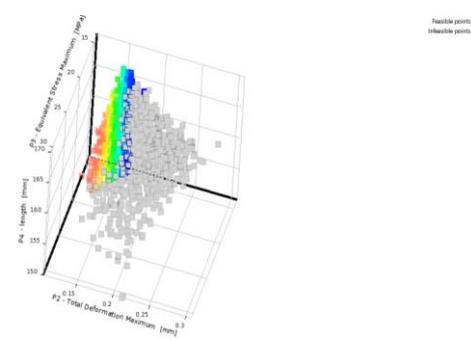
For example, an extension is available in the Client Portal to run MATLAB Optimization algorithms in DX.

### 3. Parametric optimization procedure

After the analysis with finite structural elements has been carried out, for the conditions imposed by the client, a parametric optimization project is carried out, starting from the setting of the optimization input parameters: 1)- the thickness of the support plate (20 mm); 2)-The acting force (N). Then adjust the optimization output parameters:1)- strength to deformation; 2)- the equivalent deformation stress (von Mises), whose values are determined according to the results of the static structural analysis, as well as the limit values as objectives of optimization [4, 5, 6].



a)-2d graph



b)-3D representation

**Figure 4.** The Tradeoff Graph-contains the Feasible Points(colored points) and Infeasible Points (grey points)

Figure 4 shows the Equivalent Stress and Total deformation graph, several points, a graph which is determined based on the input parameters: thickness, material, arm force.

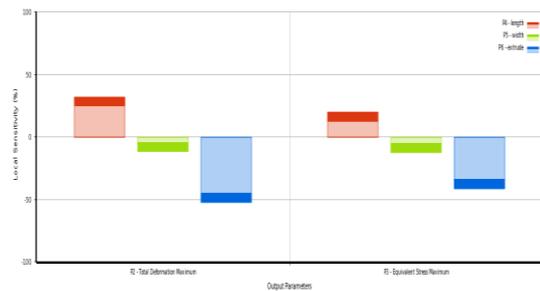
Based on the above input data, figure 4 shows the total deformation, equivalent stress for different material thicknesses chosen. By comparing the values of each plate thickness (20, 18, 16, 15, 14 [mm]), a graph provides information that the best thickness deformation values of 20 [mm] = 0.179 [mm], marked in the figure with the one star (\*).

#### 4. Support plate –steel

Since steel has better mechanical properties than aluminum, a more detailed analysis has been carried out on the steel plate. Figure 5 shows the influence of the change of thickness and material on Total Deformation, the best result is obtained in the case of a 16 mm thick plate.

Reference	Name	P4 - brat (mm)	P5 - g (mm)	P2 - Total Deformation Maximum (mm)	P3 - Equivalent Stress Maximum (MPa)	P6 - Geometry Mass (kg)
				Parameter Value	Variation From Reference	Parameter Value
*	Custom Candidate Point			X 0.097238	0.00 %	XX 5.5503
	Custom Candidate Point (specified) (DP 0)	115.42	15	X 0.1456	49.74 %	XX 3.5503
	Custom Candidate Point 1	115.42	16	X 0.083632	-13.99 %	XX 3.787
	Custom Candidate Point 2	115.42	14	X 0.12096	24.39 %	X 3.2136
	New Custom Candidate Point	115.42	15			

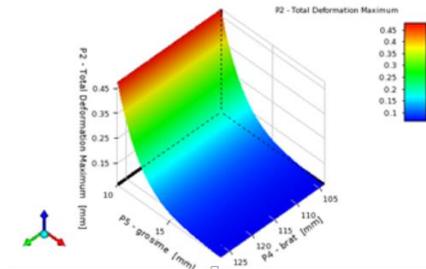
a)-Custom candidate points



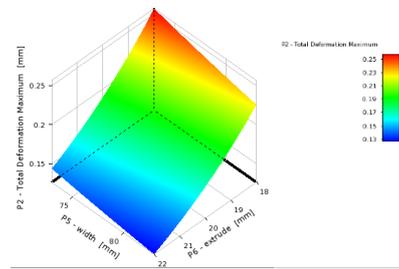
b)-the sensitivity of output parameters to the input parameters

**Figure 5** Influence of the change of thickness an material

The total deformation in the case of the slab thickness = 16 [mm], the deformation which is= 0.083 [mm]. The value that was intended to be reached to meet the requirements of the work, which require that the deformation the maximum elasticity is 0.083 mm, which will make it possible to assemble the protective parts around the exhaust properly.



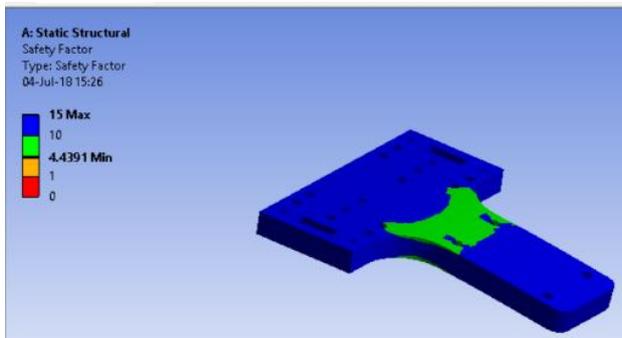
a)-thickness=16 [mm]



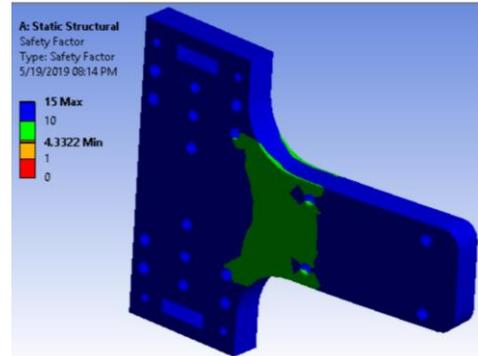
b)-thickness=20 [mm]

**Figure 6** Response surface,

Also, the safety factor is suitable = 4.4 (figure 7) so that the workpiece can be used without the problem that after several uses it will yield again elastically over the reachable limit = 0.1 mm.



a)-thickness=16 [mm] & structural steel



b)-thickness=20 [mm] & aluminium

**Figure 7.** Safety Factor

## 6. Conclusion

The safety factor of the assembly is given by the lowest safety factor determined for its components. From static structural analysis, FEA results in a minimum safety factor of 0.89 for the aluminum plate and 4.4 for the steel plate.

- have been achieved 1000 optimization templates for each input, output, and optimization objective was achieved, with a total of 5000 finite element analyzes to optimize the project.
- each analysis lasted approximately 2 seconds
- the three candidate points obtained have the following values (Figure 2 a))
- of those the 3 candidate point, Candidate Point 1 has the slightest variation from the reference (0.00%) and Candidate Point 2 has the worst variations compared to the reference
- for the optimal option that meets the optimization goals the values are:
  - o low-cost material
  - o the geometrical form is not complicated
  - o outside parameter P3 (*Total Deformation Maximum*)-0.09 mm
  - o outside parameter P4 (*Equivalent Stress Maximum*)- 21.48 MPa

Optimization was performed using three complementary methods: parameter correlation, linear optimization and response surface method, after the project was analyzed by the finite element method. Each of these phases of project implementation is determined by the results achieved. Thus, by correlating the parameters, the degree of influence of the input parameters on the output parameters was determined to decide which ones influence decisively or not the functioning and the degree of safety of the operation of the project. The degree of safety of the project is given by the degree of certainty regarding the quality of the product (the bent piece). This phase is viably linked to the finite element analysis of the perforated and bent punch. Thus, the lifetime and safety factor of the project are determining the quality of the obtained piece and the economic efficiency of the production.

Direct optimization takes into account the correlation of parameters, but does not use the results as a basis for optimization itself.

Optimization using response surfaces is a highly laborious method, but provides the most concise results, with optimization checkpoints performed on one or all three candidate points found in the optimization process.

Based on this analysis, the new device has been built, with a more robust construction that works mechanically at the desired parameters and a 40% lower cost of the semi-finished product.

## References:

- [1] D. C. Negrău, G. Grebenișan, C. Indre Experimental approach and finite element analysis of the behavior of a steel bending machine, 2019, IManE&E 2019
- [2] D C Negrău, *Finite Element Analysis and Solution Optimization for a bending device*, 2018, University of Oradea.
- [3] ANSYS 19, Workbench User's Guide <http://www.ansys.com> (accessed in April 2019)
- [4] Grebenișan Gavril, Nazzal Salem-*The Multi-Objective Genetic Algorithm Optimization, of a Superplastic Forming Process, Using ANSYS*, MATEC Web of Conferences, Volume 126, 2017, <https://doi.org/10.1051/mateconf/201712603003>.
- [5] Huei-H L, *Finite Element Simulations with ANSYS Workbench 17- Theory, Applications, Case Studies*, SDC Publications, 2017
- [6] Xiaolin Chen, Yijun Liu 2018 *Finite Element Modeling and Simulation with ANSYS Workbench*, Second Edition CRC Press DOI: 10.1201/9781351045872

## Acknowledgments

This research was developed under financing of S.C. Isrom Impex S.R.L Oradea, The Agreement # 17250/2017. Thanks to advisors and all support of the company Manager Eng. Binsleian Mihail,