**ANNALS of the ORADEA UNIVERSITY.**
**Fascicle of Management and Technological Engineering, Volume X (XX), 2011, NR2**

# WEIGHT REDUCTION IN STRUCTURES USING FINITE ELEMENTS AND MULTIOBJECTIVE GENETIC ALGORITHMS

**S. Sánchez-Caballero, M.A. Selles, R. Pla-Ferrando, E. Pérez-Bernabeu**
Universitat Politècnica de Valencia, Pl. Ferrándiz Carbonell, s/n, 03801 Alcoy, Spain
sasanca@dimm.upv.es

**Keywords**: Finite, element, FEM, genetic, algorithm, MOGA, NSGAII.

**Abstract:** The aim of this paper is to introduce a method to reduce the weight in structures which are subjected to multiple restrictions like deformation, max allowable stress, natural frequency, etc.… The method is shown through the analysis of an aluminum bracket, whose maximum stress and deformation is well defined. The analysis is done using the Structural and Design of Experiments modules of Ansys Workbench v12.1. As result of the method a weight reduction of 50,2% is achieved.

## 1. INTRODUCTION

Material cost is one of the most important in total cost of polymer based designs. Lack of raw materials will push this increasing trend in the future, so any reduction in weight is very significant.

There are many ways to achieve this reduction, but most of them can be classified into three categories[1]: sizing, geometrical and topological optimization.

Size optimization tries to find the optimum combination of the structure dimensions, while topological optimization looks for the most efficient geometry.

In geometrical optimization, the coordinates of joints in frameworks are variables to optimize. This has no use at this work.

Topological optimization tries to eliminate the lowest stressed elements of the mesh. There are many algorithms suitable; however most of them use artificial intelligence techniques. Between them Evolutionary Structure Optimization (ESO)[2] algorithms are the most used.

Both of them are very useful to designers but the first one does not change the geometry, only the dimension. It is also suitable for multiobjective optimization because variables are well defined. The last one gives a qualitative description about the way of reduce weight, based on stress levels, but not the solution itself.

The optimization algorithm employed was a multiobjective genetic algorithm (MOGA). More specifically the NSGA-II[3] algorithm. Traditional optimization methods like direct search, Optimally Criteria[4] methods or Non-Linear Programming (NLP) methods like NLP by Quadratic Lagrangian (NLPQL)[5] programming are not suitable because of its massive use of computation in first case or because it cannot handle multiple objectives.

The NSGA-II will predict, from a set of calculated points, center distributed in the variable design domain, not only the optimal variable combination but also its FEM results. Once the optimal is estimated, it has to be calculated to check that the optimal was really obtained. Here, this optimization method was implemented by means of the Goal Driven Optimization utility of Ansys Workbench.

## 2. EXPERIMENTAL

To show the optimization process a POM angle bracket was analyzed. The initial mass of the angle was 10,342 g.

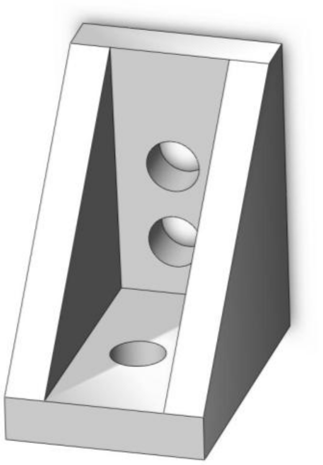

**Figure 1. Initial design.**

### 2.1. MATERIALS

The material used in this work was the POM BASF Ultraform® H2320 006 whose mechanical properties were obtained from Campus®.

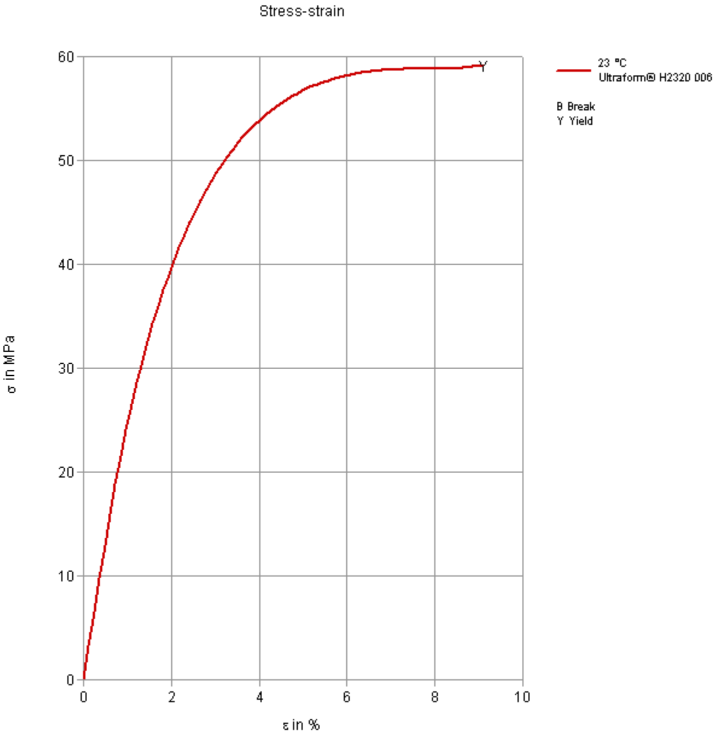

**Figure 2. True stress-strain curve.**

## 2.2. MODELING

The FEM model is composed of the POM angle, three flat washers and three screws. Washers and screw are made of steel.

Due to the difference of elastic modulus between POM and steel, it last can be considered ideally rigid. So the part made of steel where modeled like a rigid solid and the bracket like a elastic POM. Figure 3 shows the mesh for the angle and the rigid solids.

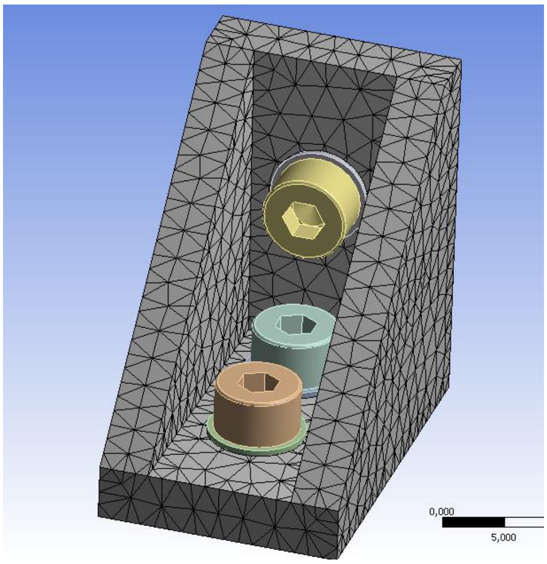

**Figure 3. Finite element model with rigid and elastic bodies**

## 2.3. LOADS AND CONSTRAINS

Once the model was built, a 100N load shown in the manufacturer catalogue, was applied in the single screw and while the other two were fixed in the thread. An only compression support was also applied on the basis, so the inner surface will only suffer compression stresses.

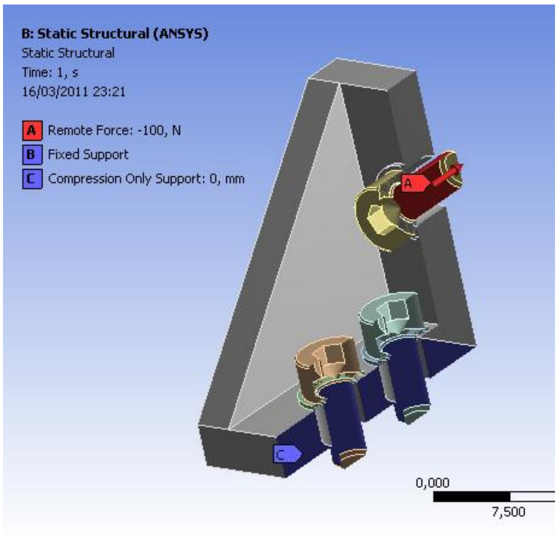

**Figure 4. Loads and constraints**

## 3. RESULTS AND DISCUSSION.

Once the whole model was done, it was solved and then stress and total deformation were obtained.

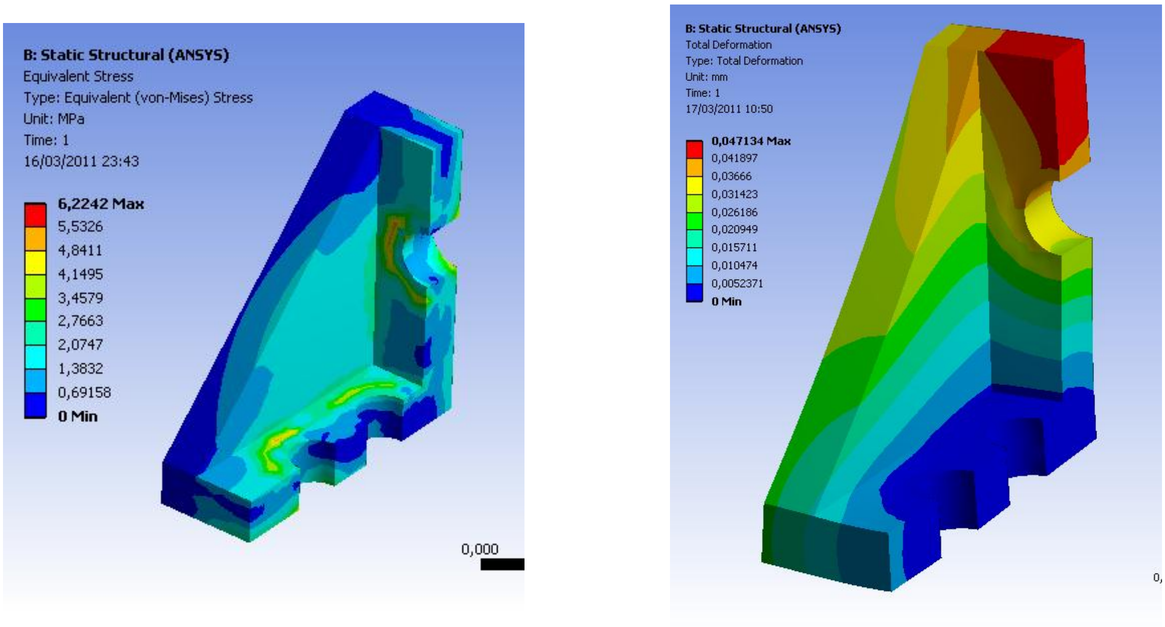

**Figure 5. Von Mises Stress and Total displacement. Initial design**

Then the input and output variables of the model were defined. The input ones were individual thickness of each plane and the output where the stress and total displacement.

Next the range per each input variable was defined and a set of points (15 in total), center distributed inside each variable domain, were generated. Then all these points where calculated generating a response surface.

Once the surface points were calculated, the optimization problem was defined as: minimize the angle mass, while de stress is lower than 55 MPa (next to yield point) and total deformation is lower than 0,5 mm.

Table1 summarizes the genetic algorithm parameters.

| | |
|---|---|
| Initial population | 100 |
| Individuals replaced per generation | 100 |
| Granulometry | 0,0098 |
| Maximum Allowable Pareto Percentage: | 70 |
| Maximun number of generations | 20 |
| Constraint handling (as goals /as hard constraints) | As goals |
| New population size | 100 |

**Table 2. Genetic algorithm parameters.**

From the calculated points, the minimization parameter and the constraints, a multiobjective genetic algorithm based on the NSGA-II algorithm estimated three candidates. Table 3 sumarizes the results.

| | Console thickness mm | 2 holes side thickness mm | 1 hole side thickness mm | Total deformation mm | Mass g | Max. Von Mises stress MPa |
|---|---|---|---|---|---|---|
| Candidate A | 1,8533 | 1,7931 | 2,0545 | 0,4945 | 5,2 | 34,0965 |
| Candidate B | 1,8629 | 1,7831 | 1,9047 | 0,5404 | 5,1 | 37,0082 |
| Candidate C | 2,4206 | 1,2157 | 1,6988 | 0,4914 | 5,2 | 34,2546 |

**Table 3. Algorithm results.**

From the three candidates, the Candidate A was chosen as final design because it has more uniform thickness than C. Candidate B violated total deformation constraint. This design is 50,2% ligher than the initial one without compromising its usefulness. Final design can be seen in Figures 6 and 7.

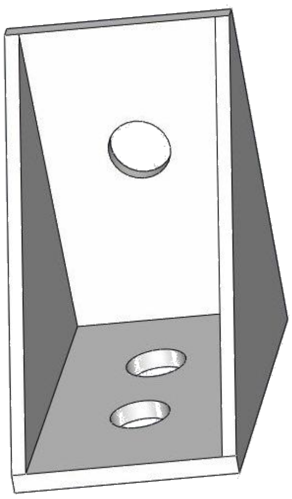

**Figure 6. Final design.**

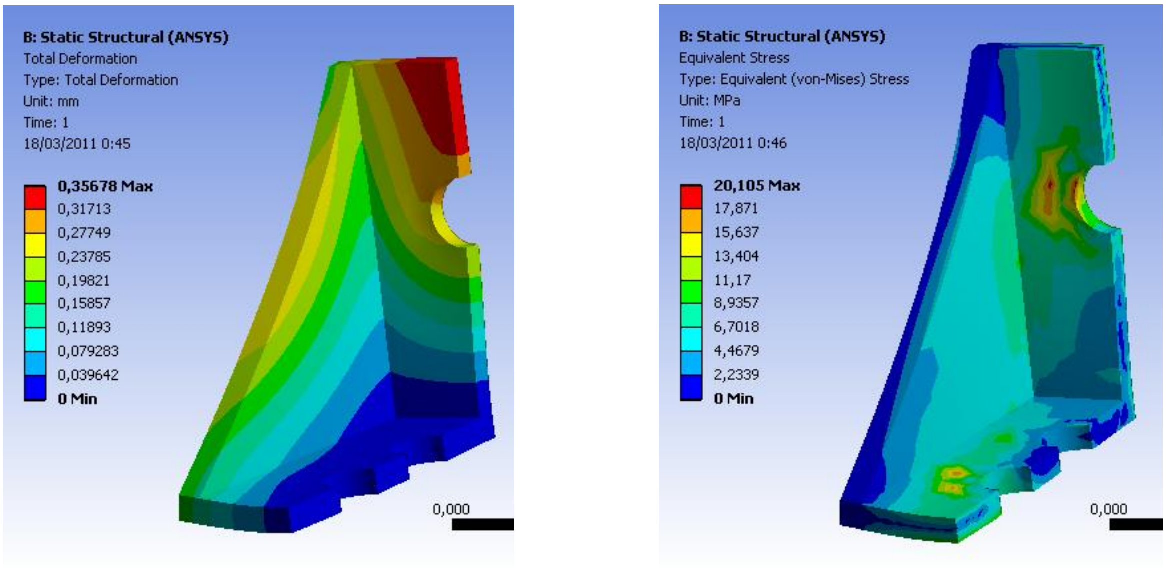

**Figure 7. Von Mises Stress and Total displacement. Final design**

Once the optimization was done, an ESO was run to find marginal reductions in weight. As is shown in Figure 8, it cannot be done any additional weight reduction because removing material from the console would compromise its rigidity.

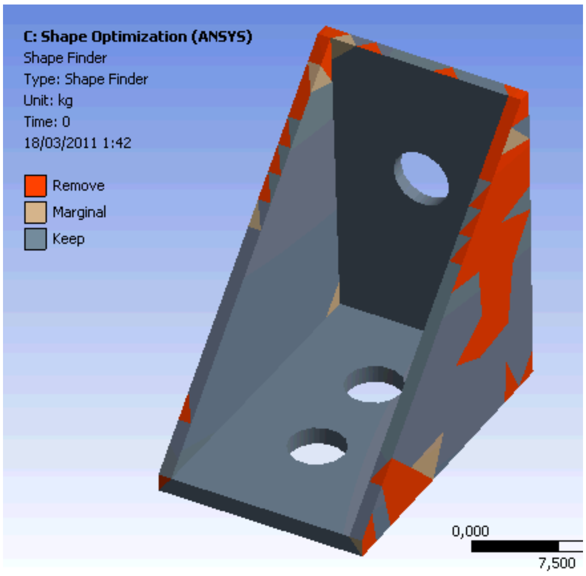

**Figure 8. ESO on Final design.**

## 4. CONCLUSIONS

Finally as conclusion to this paper some statements can be done:

1. The rigid idealization for screws and flat washers had a good behavior in simulations, simplifying significantly the computation.
2. As can be seen in Figure 7, the component safety was not committed with the weight reduction, so the main objective of this paper has been achieved.
3. From Figure 8, the component has an optimum design according to its constraints.

## 5. References

[1] G. I. N. Rozvany, "Layout theory for grid-type structures," *Topology Design of Structures,* vol. 227, pp. 251-272, 1993.
[2] Y. M. Xie and G. P. Steven, "A simple evolutionary procedure for structural optimization," *Computers and Structures,* vol. 49, pp. 885-896, 1993.
[3] K. Deb*, et al.*, "A fast and elitist multiobjective genetic algorithm: NSGA-II," *Evolutionary Computation, IEEE Transactions on,* vol. 6, pp. 182-197, 2002.
[4] W. Prager, "Optimality criteria in structural design," *Proc. Nat. Acad. Sci,* vol. 61, pp. 794-796, 1968.
[5] O. Exler and K. Schittkowski, "A trust region SQP algorithm for mixed-integer nonlinear programming," *Optimization Letters,* vol. 1, pp. 269-280, 2007.