11th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2007, Hammamet, Tunisia, 05-09 September, 2007.

# TOPOLOGY OPTIMIZATION OF LOWER TIE BAR BRACKET OF A CAR ENGINE

**Esmir Mehanović, Sakib Mustafagić**
**PS CIMOS TMD Ai**
**Gradačac**
**B&H**

**Sabahudin Ekinović, Edin Begović**
**University of Zenica**
**Faculty of Mechanical Engineering**
**Fakultetska 1, 72000 Zenica**
**B&H**

## ABSTRACT
*The paper presents some possibilities for shape design optimization that aiming to minimize mass of an analyzed part but also to uniforms distribution of a generated stresses. Here have been described computer optimization methods' providing guidelines for performing basic design and a way to upgrade existing designs. Applied topology optimization based on Finite Element Method application is a main topic of a paper. Here has been described an example of modeling and optimization of a car lower tie bar bracket.*
**Keywords:** Optimization, Finite Element Method, engine bracket

## 1. INTRODUCTION
In order to improve exploitation characteristics of a car, which is by the way a very complex machine structure with strong demands regarding energy consumption, comfortability, loading capacity, reliability, safety etc., it is necessary to find optimal shape and mass of its parts at the beginning of its creation, that is, in its design phase.
Of course, it is possible to design a new part's solution, from the scratch, but also it might be very useful to applied redesign of existing solutions. No matter what decision is done the most important step is to establish appropriate criterions of optimization which would enable enough space for application of obtained design in existing technological environment. Among these criterions are for example costs, ecological demands like decreasing emission of $CO_2$, sulphur, lead, then recycling and of course, costumer demands as a crucial point of all process. Only product that respects all above listed demands and much more, can be considered as a "well" designed.
In this time of all-present and fast-growing informatisation, designers, in performing their tasks, are supported by virtual reality created by modern tool -FEM software. Those tools make designing and testing easier, faster, and of course give opportunity to create versatile design solutions, but from the other side, rapid customer demands grow makes all those tools not only necessary but also very often not enough efficient for attaining new demands.
Optimization concept rely on mathematical description of events and activities very often bounded into various methods for finding minimum or maximum of an "objective" function, inside the defined boundary conditions like as limited exploitation conditions (deformations, vibrations, stresses etc.). However, it is important to know that solutions generated in such process are only among of possibilities that can acceptable solving of a problem with defined constraints.
Development of new commercial software for topology optimization is the first step to partial automatization of designing process in automobile parts production. Its effective application can significantly decrease designing time, reduce costs of a designed product, increase quality and machinability in manufacturing phase.
E-market today offer many software solutions for this purpose like OptiStruct, TOSCA, Nastran-Patran, GENESIS, FE-Design are.

This paper describes a car engine bracket development procedure done by sublimation of two approaches, one is by “conventional” designing methodology and second one by using HyperWorks Optistruct applications. Despite the limitations the last one have, like linear FEM model, no possibility to optimize global stress state etc., obtained solution converged to acceptable one more faster then in case of optimization done using “conventional” approach, underpinned only by designer knowledge and experience.

## 2. DESIGN AND NUMERICAL SIMULATION

Designing process starts with defining material, that is, material characteristics, product would be created of. In this case of engine bracket, selected material is aluminum alloy AlSi9Cu3, which is relatively cheep material with acceptable strength, temperature stability but also known as a brittle material (relative elongation about 1%) which hence imply sensitivity to dynamic load. Firstly, material mechanical characteristics (E, ν, σ-ε) have been determined using standard probe (ISO 6892) done (by casting) according to technology very similar to real production conditions. Results obtained by experimental testing of probes are shown in Figure 1.

**Rezultati**

| Nr | EMod N/mm² | Rp 0.1 N/mm² | Rp 0.2 N/mm² | Rp x - 1% N/mm² | Rm N/mm² | ε-F max % | RB N/mm² | ε-zruš % | S0 mm² |
|---|---|---|---|---|---|---|---|---|---|
| 1 | 75834,24 | 158,77 | 185,00 | - | 226,51 | 0,98 | 226,51 | 0,98 | 75,74 |
| 2 | 78562,57 | 166,03 | 190,68 | 236,63 | 236,86 | 0,99 | 236,86 | 0,99 | 74,82 |
| 3 | 80055,03 | 158,08 | 180,51 | 226,87 | 233,53 | 1,16 | 233,53 | 1,16 | 74,97 |

**Statistika**

| Serie n = 3 | EMod N/mm² | Rp 0.1 N/mm² | Rp 0.2 N/mm² | Rp x - 1% N/mm² | Rm N/mm² | ε-F max % | RB N/mm² | ε-zruš % | S0 mm² |
|---|---|---|---|---|---|---|---|---|---|
| $\bar{x}$ | 78150,62 | 160,96 | 185,39 | 231,75 | 232,30 | 1,04 | 232,30 | 1,04 | 75,17 |
| s | 2140,34 | 4,40 | 5,10 | 6,90 | 5,28 | 0,10 | 5,28 | 0,10 | 0,4944 |
| ν | 2,74 | 2,73 | 2,75 | 2,98 | 2,27 | 9,62 | 2,27 | 9,62 | 0,66 |

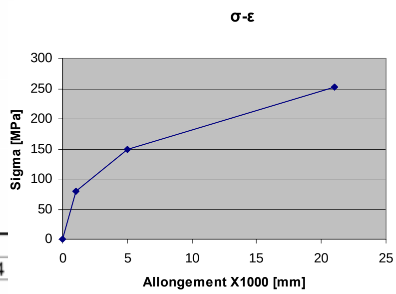


*Figure 1. Results of probes experimental testing*

These results were input to the next stage of designing process, stage of an initial models creation respecting other system components and limitations implied by those. Neither mass limitations nor process technological capability (casting) are not taken into consideration. Figure 2 represents the obtained model which is in accordance to defined requirements (acceptable stresses, vibration mode 1etc). The accordance has been tested by FEM analysis that tried to simulate as much as possible real, exploitation conditions.

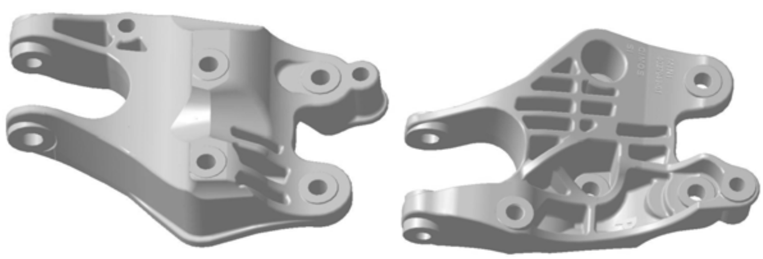

*Figure 2. Initial model of a bracket*

Now, when the initial model of a bracket was created its optimization can start. First step is to decrease mass of a part to upper level of 0.900 kg, but at the same time trying to keep the same or at least similar loading capacity and stiffness of a part. Anyway, mass and stiffness are factors with opposite influence which make this task more complicated.

In classical approach, conventional design method, optimization process has been carried according to the steps shown in Figure 3.

1. FEA calculations (model preparation, FEA model quality check, defining boundary conditions, obtained results interpretation).
2. Redesign (using designer's experience subtract materials from places with low stresses).
3. FEA calculations (evaluation of new results).
4. Final design (complete designing and technological definition of a part).

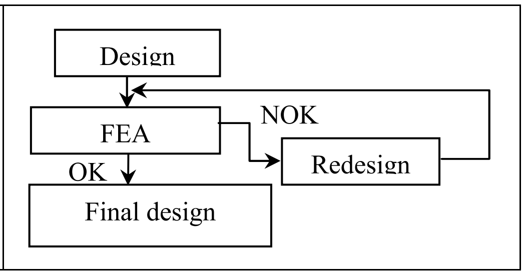


*Figure 3.* Steps in classical design process (method 1)

Optimization of initial model has been done by both approach. Figure 4 presents comparatively solutions generated by these two methods. Combining those methods a *compromise solution* has been found, then produced and finally tested in laboratory.

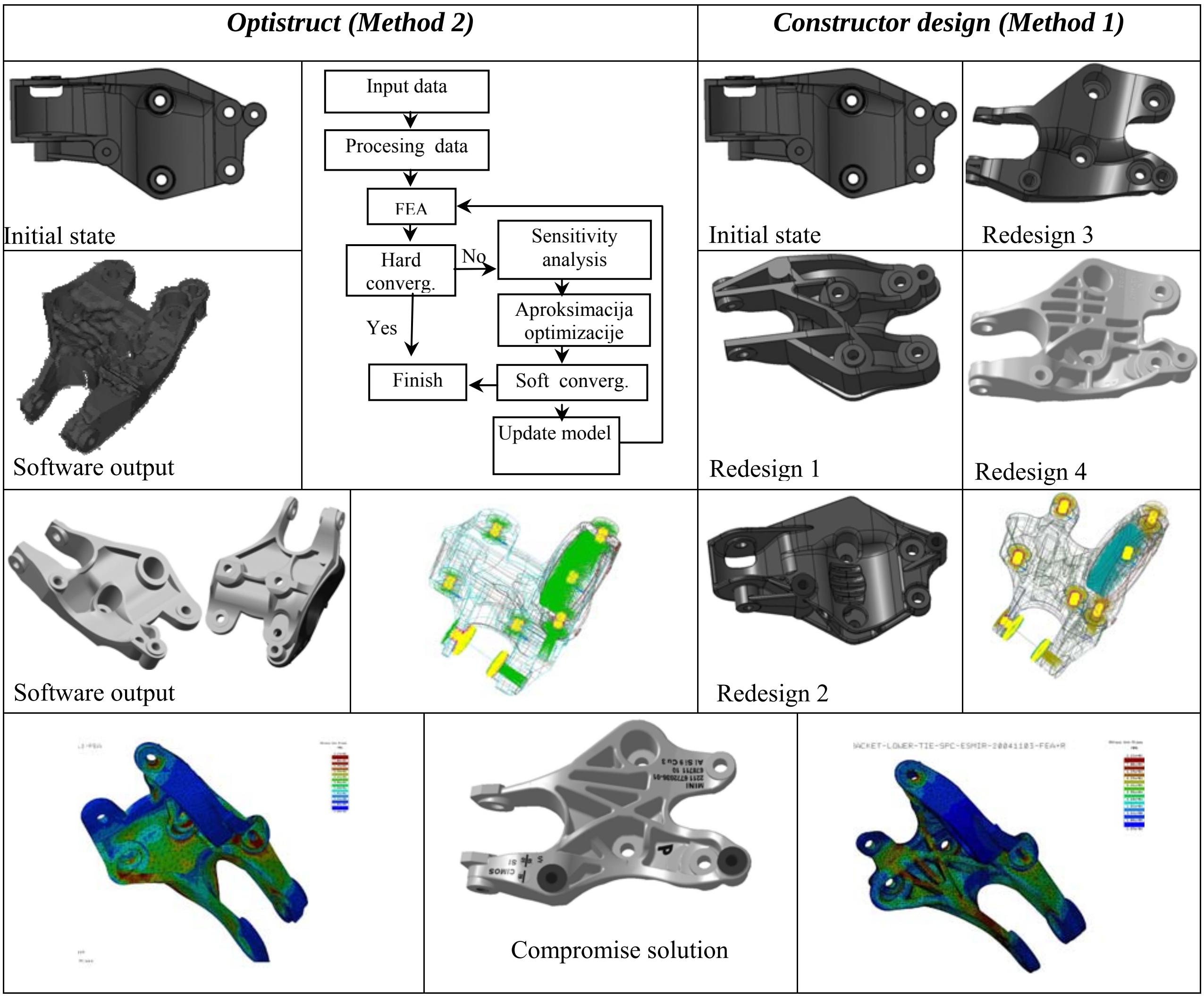


*Figure 4.* Comparison of optimization approaches

## 3. EXPERIMENTAL TESTING

Prototype testing has been carried out in CIMOS testing laboratory in Kopar, Slovenia. Purpose of testing has been to confirm results obtained by FEM analysis. Engine bracket was mounted on testing bench (figure 4.) and statically loaded in X axis direction. Moment lever length is 35 mm, testing speed 10 mm/min, and number of testing parts by one axis is 2. Figure 5. reveals results of this experimental testing of a real model.

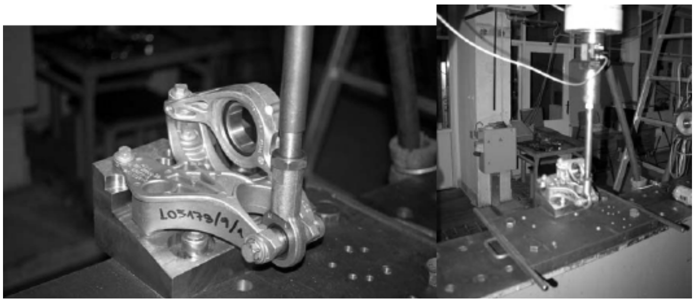

*Figure 4.The model testing*

| N | specimen number | load dir. | Minimal require-ments [kN] | result [kN] OK/ NOK |
|---|---|---|---|---|
| 1 | L05178/9/1 | +X | min 27,5 max 40,0 | 29,40 / OK |
| 2 | L05178/9/2 | +X | min 27,5 max 40,0 | 29,54 / OK |
| 3 | L05178/9/3 | -X | min 27,5 max 40,0 | -31,17 / OK |
| 4 | L05178/9/4 | -X | min 27,5 max 40,0 | -29,62 / OK |

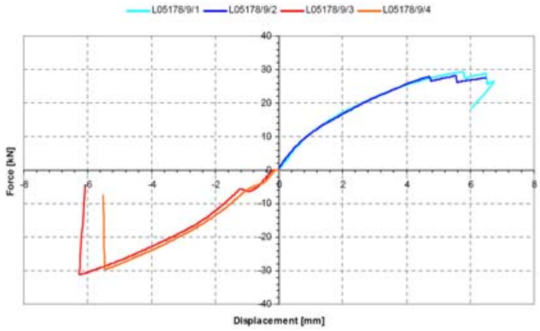


*Figure 5. Results of testing*

## 5. ANALYSIS OF RESULTS

Testing results on prototypes has confirmed predictions obtained by FEM analysis. Produced bracket is done with respecting exploitation requirements defined at the beginning of a process. Difference between results obtained by FEM analysis and measured on real part testing was 4-6 %, which is acceptable value that confirms efficiency of applied FEM model.

## 6. CONCLUSION

Simultaneous optimization in the bracket developing stage aimed to reveal some advantages and disadvantages of both used methods – designer's inventiveness versus software optimization. It can be concluded that first approach is characterized by advantages like: *lower investment in equipment*, *designer do not need to have "expert" knowledge about FEM method* etc., and disadvantages like are: *duration of a designing process can not be predicted; much more iteration (redesign) comparing to second method is needed*.

Second approach advantages are: *process is not time-consuming as a first one*, then *there is a possibility of simultaneous analysis of more then one solution for given problem*. Disadvantages are: *higher investment (software price, equipment etc.)*, then *some time need to be spent in educating designer to use software and also need knowledge and experience of a designer to implement 3D model into real environment*.

Obviously, both method have some advantages and disadvantages, so the decision which one to apply depend on requirements designed part has to fulfill and of course designer's abilities and "inclinations". Also, both methods ask for clearly defined input parameters like material characteristics, exploitation conditions and constraints. Hence, in order to reduce costs of the product development, decrease time to market time etc., sublimation of both processes might be used as very efficient designer's tool.

## 7. REFERENCES

[1] Spath D., Neithardt W., Bangert C.: Integration of Topology and Shape Optimization in the Design Process, CIRP Annals 2001

[2] MSC.Nastran™ Topology Optimization, http://www.mscsoftware.com