# COMPARATIVE STUDY FOR THE SIZE OPTIMIZATION OF A TAPERED PIPE CANTILEVER BEAM USING GENETIC ALGORITHMS

**Razvan CAZACU[1], Lucian GRAMA[2]**
[1] Industrial Engineering and Management Department, Petru Maior University, Nicolae Iorga Street, No. 1, Targu Mures, Romania, razvan.cazacu@ing.upm.ro
[2] Industrial Engineering and Management Department, Petru Maior University, Nicolae Iorga Street, No. 1, Targu Mures, Romania, lucian.grama@ing.upm.ro

***Abstract*—**This paper is intended as an investigation of the possibilities offered by coupling the scientific programming language MATLAB with the finite element package Abaqus in order to perform parametric structural optimization with genetic algorithms. To illustrate the methodology, a tapered pipe cantilever beam with a force applied at the free end was chosen and analyzed in 3 different modeling approaches: as a straight bar using 1D finite elements, as a shell using 2D finite elements and as a solid structure using 3D finite elements. The bar is modeled in the FEA software Abaqus and the scripts to modify it from code were written in the programming language Python. These scripts are called interactively and iteratively by the optimization algorithm written in MATLAB, in order to evaluate the fitness of the solutions. Besides proposing a methodology for shape optimization, this study also compares the differences in efficiency and results obtained by modeling a part with different types of finite elements.

***Keywords*—**shape optimization, tapered beam, genetic algorithm, Abaqus, MATLAB

## I. Introduction

STRUCTURAL optimization deals with finding optimum shapes for load bearing structures with respect to criteria like mass, stress or stiffness. It can be divided into 3 different but not always so distinct branches: topological optimization - aimed at finding optimal material distribution inside a feasible volume; shape optimization - for identifying optimum geometric features inside a domain with fixed topology; size optimization - dealing with finding optimum sections and sizes for parameterized structures.

Although topology optimization is responsible for the most part of the total objective minimization [1], size and shape optimizations give a structure the fine-tuning it needs to reach a true optimum shape, thus being equally important in the design process.

The three types of structural optimization are usually carried out separately, topology on one hand [1]-[3], size and shape on the other [4]. There are efforts to combine all optimization types in a single algorithm, with the obvious benefits of considering the interactions between the different types of parameters and a more streamlined optimization procedure, but these pose serious challenges for structures with plane and solid elements [5], being generally restricted to truss structures [6], [7].

According to the “no free lunch theorem” [8], there is no optimization technique better than all the others on all problems. However, as shown by the review on meta-heuristic methods [9], the genetic algorithms (GA) [10] and the particle swarm optimization (PSO) [11] are the most successfully applied ones in structural size and shape optimization, notable results being presented in [12] and [13]. There are several papers in the literature comparing the two most popular evolutionary computing methods, such as [14] and [15], the first of which concludes the best approach is to mix GA and PSO concepts in a single, unitary formulation.

In this paper we implement the GA described in [6] to perform a comparative study between 3 different physical models of the same structure. An important aim of the research is to establish a procedure for the connection of the programming language MATLAB used to implement the GA and the finite element analysis (FEA) software Abaqus used to evaluate the fitness of the candidate solutions, through the scripting language Python that Abaqus uses to expose its core architecture.

## II. The Problem and the Approach

### A. Problem Formulation and Parameterization

The problem chosen to illustrate the procedure of coupling optimization routines with FEA software and to highlight the differences between modeling structures with 1D, 2D and 3D finite elements is a tapered cantilever beam with pipe section, loaded with a concentrated force at the free end. The schematic view, as well as the magnitude of the force and the length of the beam are presented in Fig. 1. The structure is assumed to be made of steel, with the following material characteristics:

*1) Material density:* $\rho = 7.85 t/m^3$ ;

*2) Young modulus:* $E = 2.1 \cdot 10^5 N/mm^2$ ;

*3) Yield strength:* $R_{eH} = 200 N/mm^2$ .

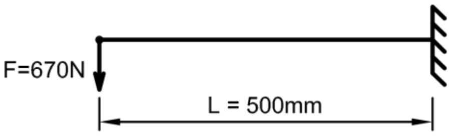


Fig. 1. Problem schematic view.

The optimization problem is formulated as the minimization of the mass (or volume) of the structure, under the constraint of maximum stress equal to the yield strength.

In the optimization process the material, the length and the boundary conditions are considered fixed, the optimization parameters being the sections of the beam at the two ends. These are the outer radii at the two ends (R1 and R2) and the wall thickness (t) as shown in Fig. 2.

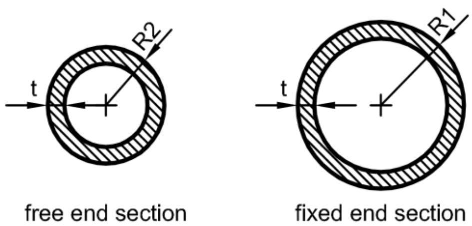


Fig. 2. Optimization parameters.

The three parameters are restricted to values between certain lower and upper values. These are listed below:

TABLE I
PARAMETERS AND THEIR BOUNDS

| Parameter Name | Parameter Description | Allowed Bounds (mm) |
|---|---|---|
| R1 | outer radius of the fixed end section | 10 .. 50 |
| R2 | outer radius of the free end section | 10 .. 50 |
| t | Pipe wall thickness | 1 .. 5 |

*B. FEA Modeling and Scripting*

As stated above, the cantilever beam has been modeled and studied in three different ways. The first is with 3rd order (Hermite interpolation polynomials) bar elements. We approximated the tapered cross-section with one varying in steps. Thus, the bar is divided into 10 segments of equal length, each having the cross-section obtained by interpolating between the sections at the two ends. This is done automatically with a script, as a function of the parameters, using (1):

$$\mathbf{R_i} = \mathbf{R_1} - (\mathbf{R_1} - \mathbf{R_2}) \cdot \frac{\mathbf{i-1}}{\mathbf{n-1}}, \Bigg| \begin{matrix} \mathbf{i = 1 \ldots n} \\ \mathbf{n = 10} \end{matrix} \quad (1)$$

The Abaqus model with 10 bar elements of stepped sections is shown in Fig. 3 below.

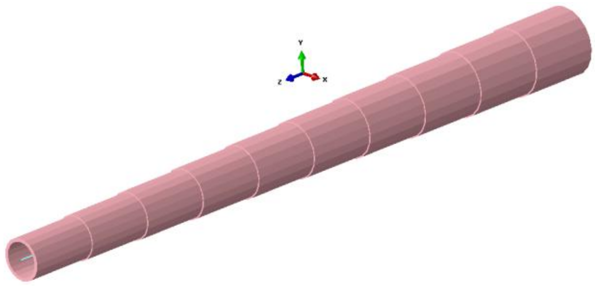

Fig. 3. The cantilever beam modeled with bar elements.

The second approach is to consider the pipe beam as a shell and model it as a space mesh of 2D elements (middle surface of the pipe), with a thickness of “t”. We used first order quadrilaterals, an example mesh being illustrated in Fig. 4:

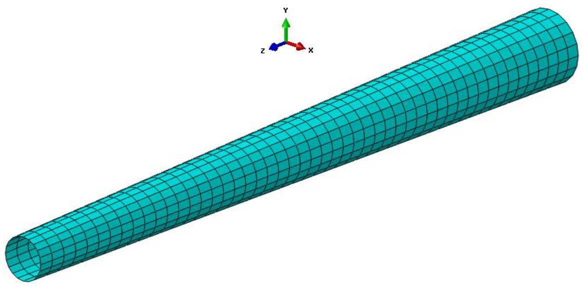

Fig. 4. The cantilever beam modeled with plane elements.

The third and final model considers 3D elements, first order hexahedrons and wedges that approximate the actual volume of the beam. An example of the mesh is shown in Fig. 5:

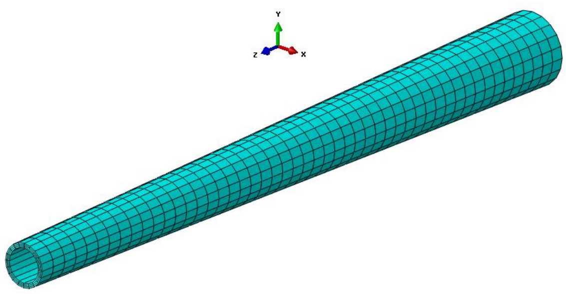

Fig. 5. The cantilever beam modeled with solid elements.

The modeling and analysis was done using the FEA software Abaqus, which has the advantage of allowing the scripting of the analysis and the automatic execution of the simulations in batch mode.

Each model has a Python script attached, which can be adjusted by changing the three optimization parameters. The script is responsible with modifying the parametric model, remeshing the part, start the analysis and write the requested results to an output text file.

This script is called in the command line by the optimization algorithm, which then reads the results from the output file.

*C. The Genetic Algorithm*

The optimization technique used in this research is the genetic algorithm found in MATLAB's Global Optimization toolbox, a brief description of the GA being given in [15]. The objective of the optimization is the total mass (or volume) under a constraint of maximum stress. The constraint is taken into account in the form of the parabolic penalty function introduced by the authors in [6]. This has the advantage of favoring individuals who violate the constraint condition by a small margin (likely to be close to some feasible optimum) and penalizes increasingly more drastic the individuals as they move away from the constraint limit.

The chromosome consists of only 3 real-value genes, corresponding to the three parameters (R1, R2, t) bounded as shown in Table I. The coding and decoding is done with the help of the Python scripts mentioned in the previous section.

The GA settings used are given in the list below:

1) *Number of generations: 51 (including the initial one);*
2) *Population size: 21 individuals;*
3) *Crossover method: single point (more appropriate for such a short chromosome)*
4) *Crossover fraction: 60%-80% of the new population is obtained by crossover from the current population;*
5) *Elitism: 1 individual is automatically passed to the new generation, ensuring we at least maintain our best solution found so far;*
6) *Mutation probability: 33%-50% (the numbers are indirectly proportional to the chromosome length [4]);*
7) *Selection: roulette wheel;*
8) *Fitness scaling: rank based (each individual receives a scaled fitness score indirectly proportional to its rank);*

## III. RESULTS

This section describes the optimization results, along with the settings used and the convergence history. The actual results can be compared with the theoretical ones obtained by using the Euler-Bernoulli beam theory. The maximum bending moment is obtained at the fixed end ($M_{max} = 0.335 kN \cdot m$) and is independent of the cross-sections if we ignore the self-weight of the beam.

The section of the free end does not influence the maximum bending moment, thus the optimum design with respect to the volume is obtained by taking the smaller possible value for the outer radius at that point. More, because a beam subjected to bending is more efficient as the material of the section is pushed further away from its center of gravity, the optimum thickness of the wall is the minimum allowable one.

With the maximum bending moment and wall thickness fixed, the only variable who's value influences the weight of the structure and its maximum stress is the outer radius at the fixed end. This can be calculated to give a maximum stress equal to the yield strength of the material, this being the value that gives the smaller volume while respecting the stress constraint.

Considering all the above, the theoretical optimum values for the parameters are summarized in TABLE II. However, caution must be taken when comparing these results with the actual ones given by the optimization routines, considering the simplifications the Euler-Bernoulli model introduces.

TABLE II
OPTIMUM THEORETICAL SOLUTION

| Parameter Name | Value |
|---|---|
| R1 | 23.83 mm |
| R2 | 10.00 mm |
| t | 1.00 mm |
| volume | 51570 $mm^3$ |
| mass | 0.405 kg |

*A. Formulation with bar elements*

For the formulation with bar elements, the comparison of results with the theoretical ones are the most relevant. Fig. 6 shows the evolution of the best individual over the 50 allowed generations. TABLE III lists the results of the optimum solution.

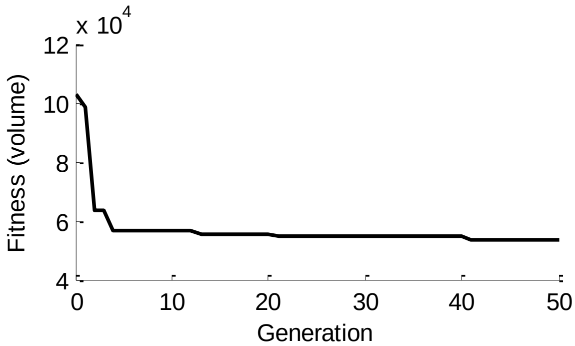


Fig. 6. Best individual score over generations for 1D model.

TABLE III
OPTIMUM FOR 1D SOLUTION

| Parameter Name | Value |
|---|---|
| R1 | 24.40 mm |
| R2 | 10.80 mm |
| t | 1.00 mm |
| max VonMises | 190.48 $N/mm^2$ |
| volume | 53715 $mm^3$ |
| mass | 0.422 kg |
| % above theoretical opt. | 4.2% |

*B. Formulation with plane elements*

Fig. 7 illustrates the evolution over the generations of the best individual. The best solution is found in the 33th generation.

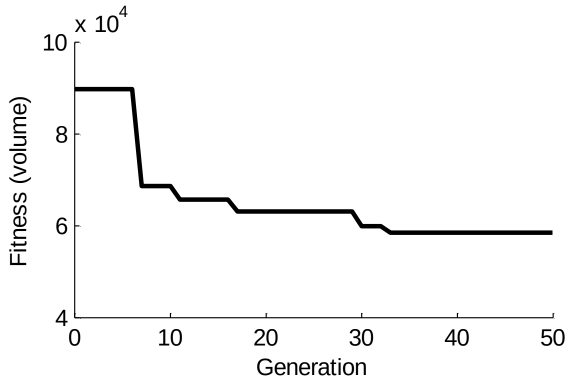


Fig. 7. Best individual score over generations for 2D model.

The data for the best final solution is synthetized in TABLE IV. It is worth noticing the width is close to the minimum, but the radius at the free end could be smaller.

TABLE IV
OPTIMUM FOR 2D SOLUTION

| Parameter Name | Value |
|---|---|
| R1 | 24.30 mm |
| R2 | 11.96 mm |
| t | 1.03 mm |
| max VonMises | 194.26 N/mm$^2$ |
| volume | 58473 mm$^3$ |
| mass | 0.459 kg |
| % above theoretical opt. | 13.3% |

### *C. Formulation with solid elements*

In the formulation with solid finite elements, the convergence of the solution is very fast, as can be observed in Fig. 8. The best one is achieved in less than 10 generations. As TABLE V shows, the solution found by the algorithm is very close to the real optimum, with *R1* and *t* at the minimum and the stress close to the limit.

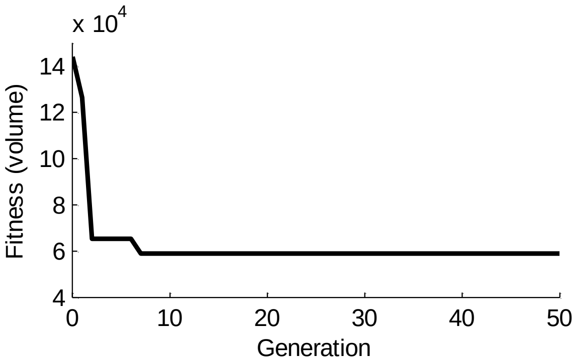


Fig. 8. Best individual score over generations for 3D model.

TABLE V
OPTIMUM FOR 3D SOLUTION

| Parameter Name | Value |
|---|---|
| R1 | 28.46 mm |
| R2 | 10.00 mm |
| t | 1.00 mm |
| max VonMises | 195.30 N/mm$^2$ |
| volume | 58882 mm$^3$ |
| mass | 0.462 kg |
| % above theoretical opt. | 14.1% |

## IV. CONCLUSIONS

As this research shows, the coupling of commercial programming languages and FEA software is a feasible approach to the size optimization of load carrying structures. As the results show, the genetic algorithm is capable of finding quickly solutions close to the absolute optimum, especially in the case of modeling the structure with solid elements. Although the difference in final mass of structure is the biggest in this case, the fact is due to the different simplifications in the 1D and 3D models rather than a less efficient optimization technique.

## REFERENCES

[1] L. Harzheim, G. Graf, "A review of optimization of cast parts using topology optimization - Part 1", *Structural and Multidisciplinary Optimization*, vol. 30, no. 6, 2005, pp. 491-497.
[2] Z.H. Zuo, Y.M. Xie, X. Huang, "Combining genetic algorithms with BESO for topology optimization", *Structural and Multidisciplinary Optimization*, vol. 38, no. 5, 2009, pp. 511-523.
[3] M.P. Bendsoe, O. Sigmund, *Topology Optimization: Theory, Methods and Applications*, Berlin: Springer-Verlag Berlin Heidelberg, 2003.
[4] J.E. Rodriguez, A.L. Medaglia, J.P. Casas, "Approximation to the optimum design of a motorcycle frame using finite element analysis and evolutionary algorithms", *IEEE Systems and Information Engineering Design Symposium*, 29 April, 2005, pp. 277-285.
[5] M. Zhou, N. Pagaldipti, H.L. Thomas, Y.K. Shyy, "An integrated approach to topology, sizing, and shape optimization", *Structural and Multidisciplinary Optimization*, vol. 26, no. 5, 2004, pp. 308-317.
[6] R. Cazacu, L. Grama, "Steel truss optimization using genetic algorithms and FEA", *Procedia Technology*, vol. 12, 2013, pp. 339–346.
[7] N. Noilublao, S. Bureerat, "Simultaneous topology, shape and sizing optimisation of a three-dimensional slender truss tower using multiobjective evolutionary algorithms", *Computers & Structures*, vol. 89, no. 23, 2011, pp. 2531-2538.
[8] D.H. Wolpert, W.G. Macready, "No Free Lunch Theorems for Optimization", *IEEE Transactions on Evolutionary Computation*, vol. 1, no. 1, 1997, pp. 67-82.
[9] S. Sanchez-Caballero, M.A. Selles, R. Pla-Ferrando, A.V. Martinez Sanz, M.A. Peydro, "Recent Advances in Structural Optimization", *Annals of the Oradea University, Fascicle MTE*, vol. 11, no. 1, 2012, pp. 2.118-2.127.
[10] J.H. Holland, *Adaptation in natural and artificial systems*, Michigan: University of Michigan Press, 1975.
[11] J. Kennedy, R. Eberhart, "Particle swarm optimization", *Proceedings of the IEEE International Conference on Neural Networks*, vol. 4, 1995, pp. 1942-1948.
[12] X. Tang, D.H. Bassir, W. Zhang, "Shape, sizing optimization and material selection based on mixed variables and genetic algorithm", *Optimization and Engineering*, vol. 12, no. 1-2, 2011, pp. 111-128.
[13] P.C. Fourie, A.A. Groenwold, "The particle swarm optimization algorithm in size and shape optimization", *Structural and Multidisciplinary Optimization*, vol. 23, no. 4, 2002, pp. 259-267.
[14] R.C. Eberhart, Y. Shi, "Comparison between genetic algorithms and particle swarm optimization", *Evolutionary Programming VII Lecture Notes in Computer Science*, vol. 1447, 1998, pp. 611-616.
[15] R. Cazacu, L. Grama, "Structural Optimization with Genetic Algorithms and Particle Swarm Optimization", *Proceedings of the Annual Session of Scientific Papers "IMT Oradea"*, vol. 12(22), no. 1, 2013, pp. 19-22.