STRUCTURAL OPTIMIZATION OF A WIND TURBINE TOWER INTERFACING MATLAB AND ANSYS

Author: Fernando Mendonça[1] *fgm1991@gmail.com.br*
Advisor(s): Fransisco Soeiro ; José Guilherme da Silva[1]
[1] State University of Rio de Janeiro

July 1, 2015

PPG-EM Seminars: season **2016**
www.ppg-em.uerj.br

Keywords: Optimization, tower, matlab, GA.

1 Introduction

Increasing energy consumption with simultaneous exhaustion of fossil fuels and rapidly progressing natural environment degradation caused countries to support renewable power industry. Current trends show that considerable technological progress related to the wind industry has been achieved over recent years.

Commercial wind turbine towers are about 100 m high and forces acting on them are very large in comparison to other types of towers. This causes the wind turbine tower to be expensive due to the large material consumption. Since wind power plants are manufactured in commercial scale even slightly change in the tower mass can lead to great savings. In accordance with the advance in size of turbines, tower optimization becomes more important, as towers account for 20–30% of the turbine cost.

This work aims to optimize the tower of a wind turbine REpower MM92 integrating the software MATLAB and ANSYS. The tower structure analysis will be carried out with ANSYS, which is a finite element software. The model was programmed in Ansys Parametric Design Language (APDL), taking into consideration the nacelle's weight and the wind loads. From the different optimization algorithms available in MATLAB, the Genetic Algorithms (GA) havs been selected to optimize the tower structure.

GA's can be described as search algorithms inspired from evolutionary biology, thus based on the mechanics of natural selection and natural genetics. They belong to a category of stochastic search methods, with an additional strength that randomized search is conducted in those regions of the design space which offer the most significant potential for gain.

2 Methodology

The wind turbine tower's structure was manufactured in steel and it has a conical shape. It is initially divided in three parts in order to make the assembly and the transportation easier. The diameter and thickness decrease from the base to the top of the tower.

The numerical model was created based on the Finite Element Method. It was programmed using the software ANSYS, wich became responsible for the static analysis during the process. Obtaining the maximum displacement and stress level was the analysis goal.

2.1 Structural optimization

A solution to an optimization problem specifies the values of the decision variables, and therefore also specifies the value of the objective function. This work aimed to optimize the volume by reducing the thickness of the tower, maintaining the diameter and height values. Two variables were adopted in this process:

- B: tower's base thickness
- C: tower's top thickness

The optimization was carried out under some structural restrictions. The constraints applied on the model followed the recommendation of Eurocode 3:

- Displacement restriction

$$\delta_{max} \leq 152cm \tag{1}$$

- Stress level restriction

$$\sigma_{max} \leq 255MPa \tag{2}$$

Finally the optimization problem can be stated as follows:

$$\begin{aligned} \underset{x}{\text{minimize}} \quad & f(x) \\ \text{subject to} \quad & g(x) \leq \delta_{max} \\ & h(x) \leq \sigma_{max} \end{aligned} \tag{3}$$

Where:

x - Project variables
$f(x)$ - Tower volume
$g(x)$ - Tower maximum displacement
$h(x)$ - Tower maximum stress level

2.2 MATLAB-ANSYS interface

The following are the points summarizing the genetic algorithm and the interfacing process of ANSYS and MATLAB:

1. Initially the algorithm creates a random population.
2. This new population then becomes the parameter for ANSYS.
3. ANSYS is then run in batch mode from MATLAB, generating the output files.
4. The output files are read in MATLAB and the constraint conditions are checked.
5. If the check is satisfied then the volume is accepted, else MATLAB returns an invalid value.
6. The algorithm then creates a new sequence of populations. In every step, the algorithm creates a new population by reproducing the individuals in the current generation.
7. Finally the algorithm stops when any one of the stopping criteria is met, wich in this work was the MATLAB's default stop criteria.

Figure (1) is a schematic illustration of the structural optimization using the genetic algorithm that is implemented in MATLAB.

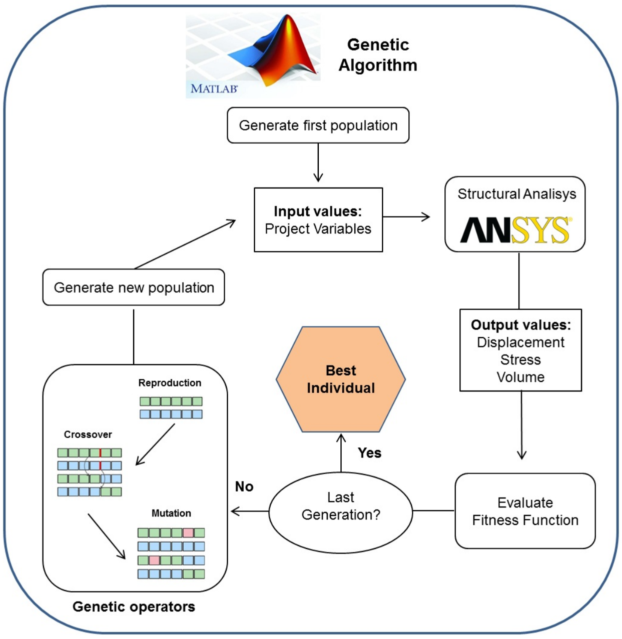

Figure 1: Genetic algorithm optimization interfacing MATLAB and ANSYS

3 Results and Discussion

The genetic algorithm searches for the different feasible thickness values inside the defined search that the space the fitness function is evaluated. The optimized values obtained by interfacing ANSYS and MATLAB using genetic algorithm are shown in Table1 and Table 2.

Table 1 shows that the optimization of the tower structure using the genetic algorithm generated a reduction of 53% of its volume. Thus, achieving less than the half of its original volume.

	Inicial value	Optimum value
B	$30mm$	$11,15mm$
C	$12mm$	$6,21mm$
Volume	$17,91m^2$	$8,41m^2$

Table 1: Optimum project variables values

Analysing the results in Table 2, it is possible to see that the maximum stress level supported by the tower in the end of the process represents 99.8% of the allowable value. It means that the optimum project variables achieved were satisfactory based on the optimization constraints.

	Optimum value	Allowable value
δ_{max}	$0,876m$	$1,52m$
σ_{max}	$254,6\ MPa$	$255\ MPa$

Table 2: Project restrictions values

4 Conclusions

The tower optimization using the genetic algorithm presented satisfactory results, providing a 53% volume reduction. Thus, applying this process in a large wind farm can be very profitable.

The optimization interfacing MATLAB and ANSYS proved to be an efficacious process. Since the interface demonstrated a high processing time, the projects need to be evaluated, however its behaviour proved to be very satisfactory when it comes to structural optimization.

5 Acknowledgments

The author thanks CAPES for the financial support.

References

[1] Bzdawka, K. - “Structural Analysis of a Wind Turbine Tower”, Poznań University of Technology, 2010.

[2] A. J. Chipperfield, P. J. Fleming, H. Pohlheim and C. M. Fonseca, "Genetic Algorithm Toolbox User's Guide", ACSE Research Report No. 512, University of Sheffield, 1994.

[3] CEN, EUROCODE 3: Design of Steel Structures, Parte 3-2, Tower, masts chimneys. European Commitee for Standardization, Brussels, 2004.

[4] Indu, K. R., Airin, M. G., “Optimization of Tapered Cantilever Beam Using Genetic Algorithm: Interfacing MATLAB and ANSYS”, IJIRSET, 2015, vol.4.

[5] A. Gauchía, B.L.; Boada, M.J.L.;V. Díaz “Integration of MATLAB and ANSYS for Advanced Analysis of Vehicle Structures”, Intech, 2014