Optimization of Mechanical Components of a High-Altitude Wind Generator

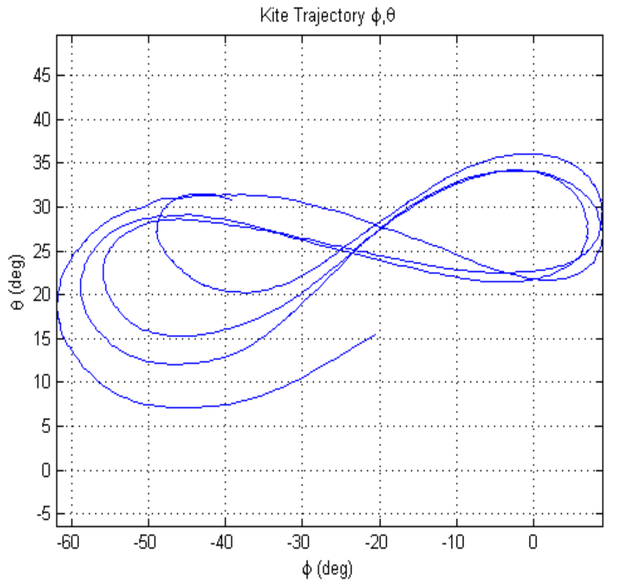
Dott. Ing. M. Picoco[a], Prof. Ing. G. Zavarise[b], Prof. Ing. G. Quaglia[a], Dott. Ing. G. Maneia[c]

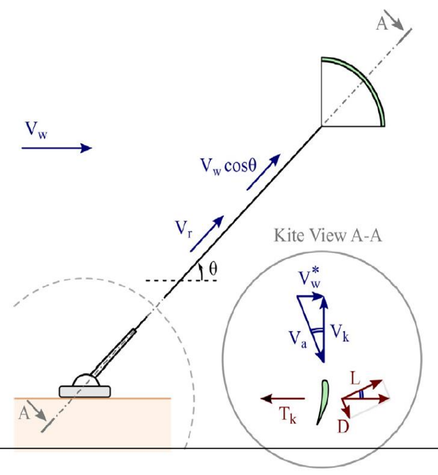
a *Department of Mechanical and Aerospace Engineering (DIMEAS), Politecnico of Turin, Italy*
b *Department of Structural, Geotechnical and Building Engineering (DISEG), Politecnico of Turin, Italy*
c *Kitenergy S.r.l., Turin, Italy*

Introduction

The aim of this work was to study a High-Altitude Wind Energy prototype behaviour under different load cycles, to find the currently oversized components and to redesign these parts using ANSYS Mechanical solver. The results were then subjected to a validation Finite Element Analysis. KE60 mkII prototype exploits a parachuting kite flying hundreds of meters above the ground. The ground station is composed by two rotating drums around which the cables are wrapped-up, a rotation system and a translation system.

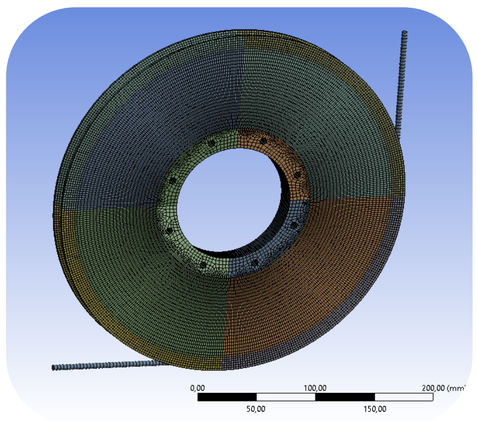
The graph alongside shows the behaviour of the kite during operating cicle of the machine. When the kite is located in one of the ends of the trajectory, it creates maximum load imbalance between the two cables and the load on the pulley wheel based inside the ground station is maximum. This load was used to analyze stress and strain on the pulley, in order to find the best shape with the lowest weight.
The disk, instead, is usually subjected to the same type of load, for emergency braking of drums during the power generation phase.



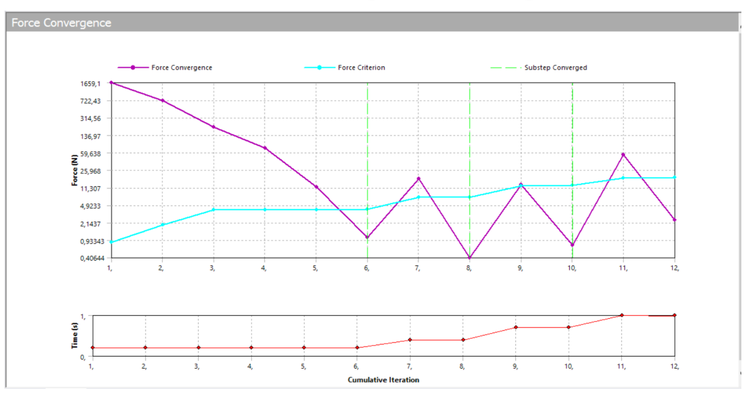


Pulley Wheel Model

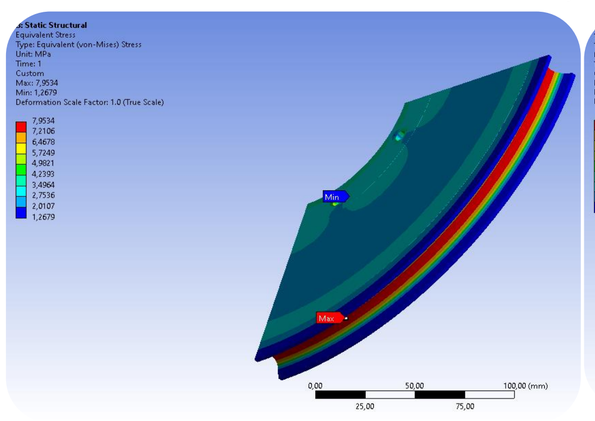
Model and Discretization

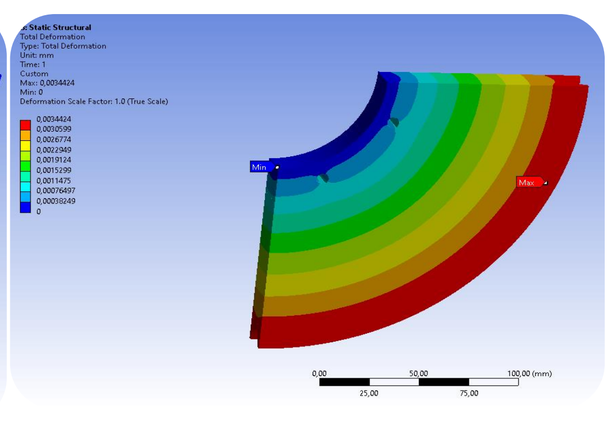


Force Convergence Diagram

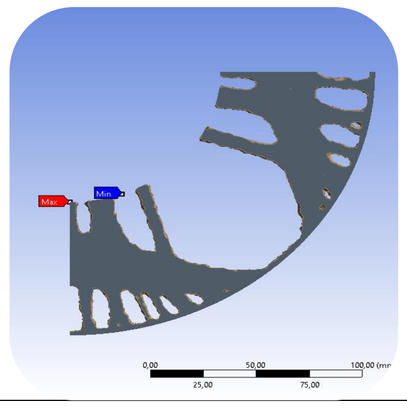


Static Structural Analysis Result: Stress and Strain

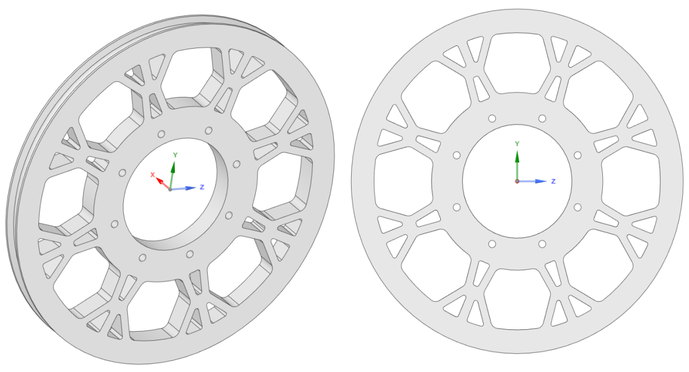




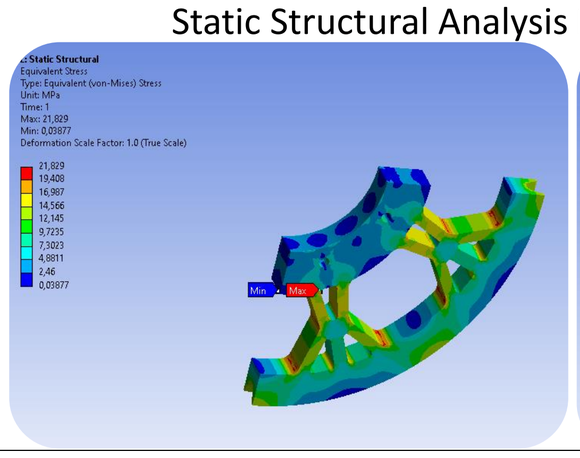
Topology Optimization Solution

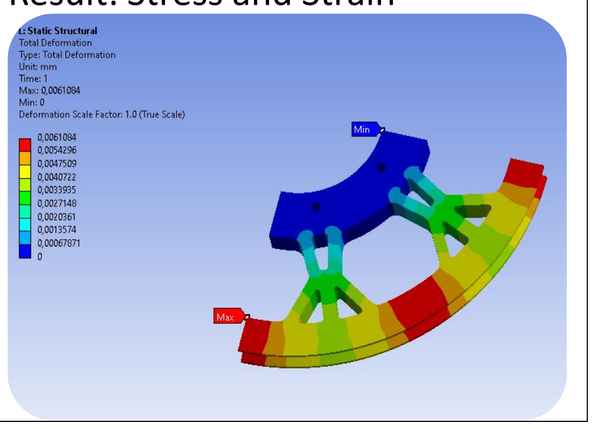


Shape Proposal



Static Structural Analysis Result: Stress and Strain

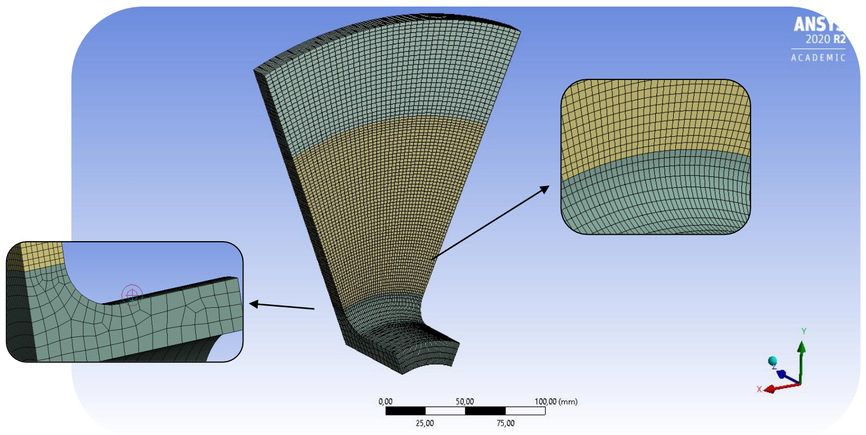




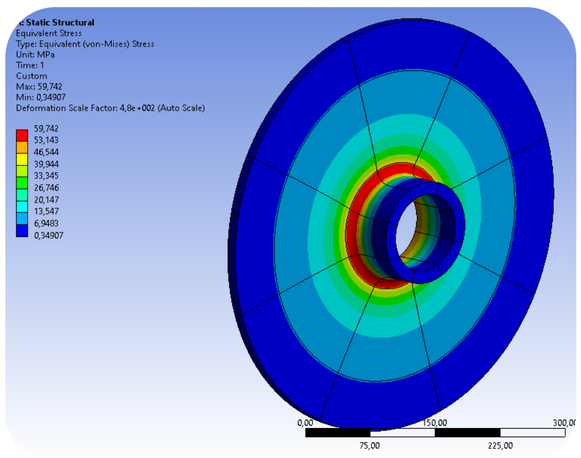
Disk Brake Model

Model

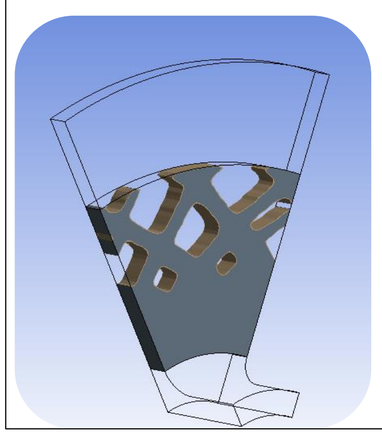
Discretization



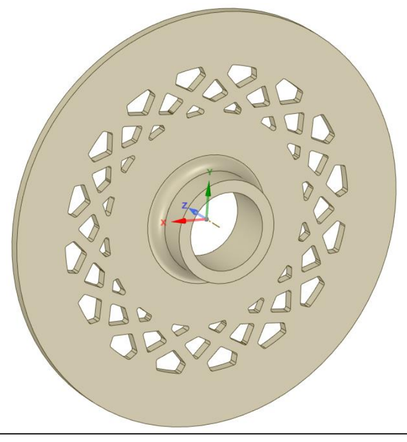
Von-Mises Stress



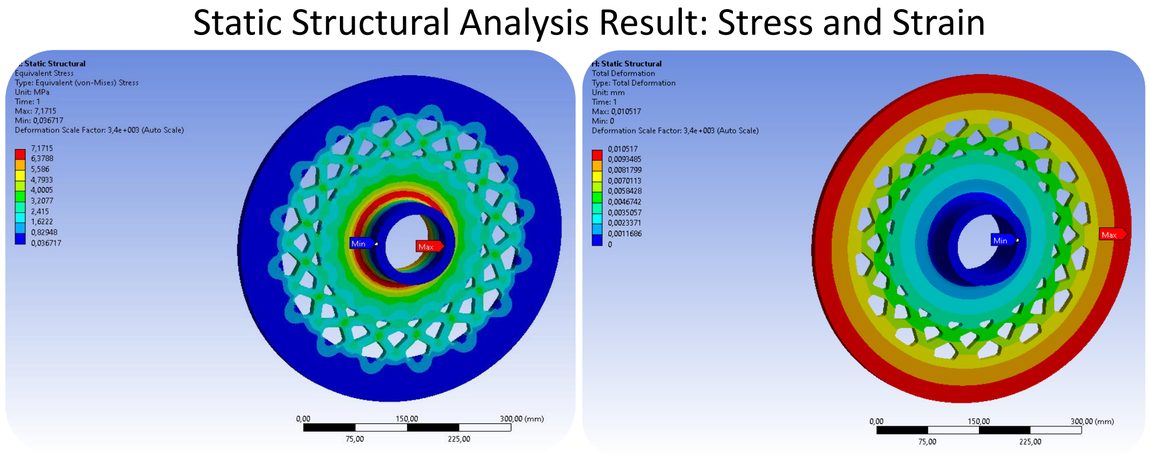
Topology Opt. Solution



Shape Proposal



Static Structural Analysis Result: Stress and Strain



Final Results

The solutions presented by the ANSYS solver have been analyzed and re-engineered, in order to obtain shapes that could be easily manufactured by metal cutting or metal laser sintering, but which still have the strength necessary to withstand the maximum loads to which the components are usually subjected.
The solver proved to be an excellent tool for analyzing and researching the best compromise between weight, strength and costs of materials.