

DEVELOPMENT OF AUTOMATED FINITE ELEMENT MODELS FOR LARGE WIND TURBINE BLADES

Mathijs Peeters¹ and Wim Van Paepegem¹

¹ Department of Mechanics of Materials and Structures, Ghent University
Technologiepark-Zwijnaarde 903, 9052 Zwijnaarde, Belgium

Email: mathijs.peeters@ugent.be

Email: Wim.VanPaepegem@UGent.be, web page: <http://www.composites.ugent.be/>

Keywords: Wind turbine blades, Finite elements, Automation

ABSTRACT

The design of wind turbine blades to date is done using design codes employing advanced analytic models. While these models are very good for performing many iteration steps in the design process, little is known about the exact stress distribution within the blade under different loads and conditions. Such conditions may include transportation, power production, emergency stops, etc. For this purpose a finite element model can be used. However, such a model is only as good as it is detailed. A real life wind turbine blade is typically built from sandwich materials, consisting of a foam or balsa core and glass or carbon fibre epoxy laminate on the top and bottom. Such a blade has a composite layup consisting of a large number of plies. As a consequence, adding all the detail of each ply to a finite element model can be a rigorous process.

In this work, a finite element model of a real life 50 metre long glass fibre epoxy composite wind turbine blade has been developed. The model employs shell elements with top offset positioned on the outer mould layer. For this purpose, a python script was built to automate the partitioning and layup creation process in the finite element code AbaqusTM. Based on an excel sheet, each ply is automatically added to the model, starting and ending at the exact specified millimetre. The use of shell elements results in a stepwise thickness transition of the laminate. This aligns with reality, since the local laminate thickness is the result of a discrete number of layers. As a consequence, each ply drop-off is accurately represented in the model.

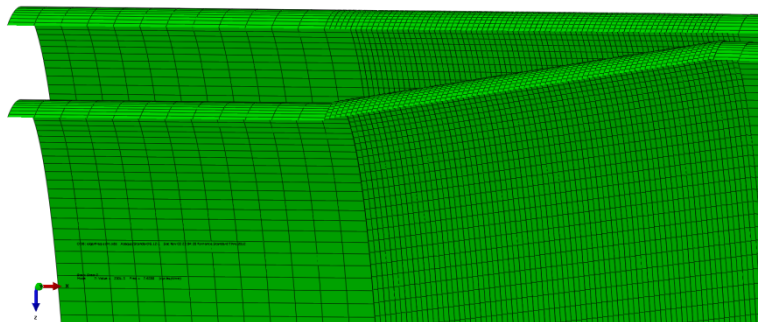


Figure 1: Rendered thickness of the shell elements at the root of the blade. Notice the smooth, stepwise thickness transition.

A modal analysis was conducted on the obtained model. The resulting Eigen frequencies were in agreement with those predicted in the design. The result of a computational fluid dynamics analysis was also applied to the model, under the form of a distributed pressure across the blade's surface.

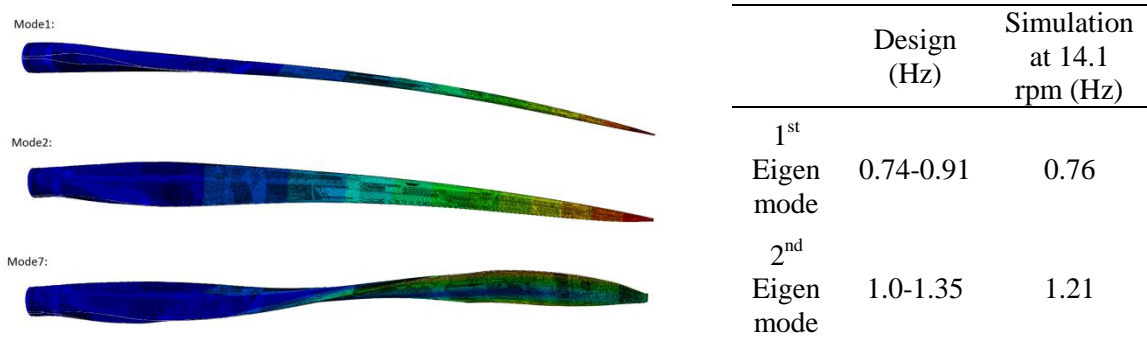


Figure 2: from top to bottom: 1) the first Eigen mode corresponding to a flap wise Eigen shape, 2) the second Eigen mode corresponding to an edgewise Eigen shape, 3) the first torsional Eigen mode.

Because the generation of the model is automated, the model is parametric and can hence be used for optimization purposes. This possibility was demonstrated using a genetic algorithm, using the tip displacement under a uniform surface traction load as fitness function.

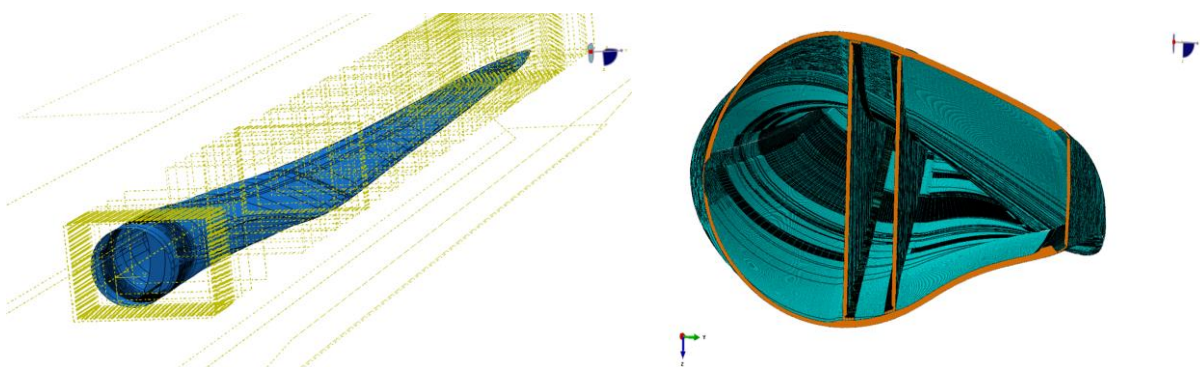


Figure 3: (left): FEM in the Abaqus/CAE pre-processor, each yellow plane performs an intersection. (Right): cross section of the blade mesh.

An important influence on the stiffness and the general response of the blade are the adhesive bonds. These are present between the shear webs and spar caps and at the leading and trailing edges. As these undergo a complex loading and shear effects are very large, solid elements should be employed to model the adhesive. The bonds are modelled using a multi-layer orphan mesh, which extrudes the shell elements of the laminate on the top and bottom of the shear webs which is connected to the adhesive. On each of the bonds, a tie constraint is applied to the outer surfaces to constrain them to the blade's shell or shear web. An option is used to set the distance between the outer surface of the adhesive layer and the blade's shell to zero before the simulation starts. This makes it possible to obtain a smooth, well connected adhesive.

In an effort to expand the flexibility of this type of model, to explore the possibility of other element types, such as continuum shell and solid elements and to make it possible to include other complex details of real life blades, such as the adhesive bonds at the leading and trailing edges, a script is under development which directly generates the mesh for a finite element model of a wind turbine blade starting from aerofoil data files.