**Aeroelastic Properties of Flexible Lifting Surfaces for Wind Turbines**

**Name:** Iñigo López Canalejo  
**E-Mail:** inigo.lopez@tum.de  
**Supervisor:** Prof. Dr.-Ing. Kai-Uwe Bletzinger  
**Chair of Structural Analysis**  
**Started:** 07/2017  
☑ ongoing ☐ finished

**Motivation**
Flexibility and adaptivity during flight can offer great potential in different domains of the aerodynamic of a wing [1]. Previous studies on flexible-membrane concepts suggested the possibility to improve the aerodynamic performance if some parameters such as the properties of the membrane, the pre-stress or the geometry are precisely chosen [2]. Due to its flexibility, the membrane can adapt itself to the incoming flow and produce an adequate camber, which influences the aerodynamic forces of the system. Lift can therefore be increased with the same amount of drag as for a rigid equivalent concept. The stall can be shifted to higher angles of attack and the effects of gusts could furthermore be alleviated.

**Goals**
Purposes of the ongoing project are: development of a coupled simulation methodology, experimental validation, required depth of modeling and modeling decisions and load adaptation. Modern methods for modeling flexible load-bearing structures, form finding and coupling strategies on the one hand and efficient flow simulations on the other hand are brought together and applied to the present problem in a coupled multi-field simulation and design environment. For the verification and validation of the methodology and the numerical results, data on aerodynamic performance, on flow fields and on structural deformation or adaptivity are obtained from wind tunnel tests. The advantages of a FlexWing in terms of lift characteristics, glide ratio and pitching moment characteristics are to be demonstrated.
Results
The in-house finite element based structural solver CARAT++ was used to solve the non-linear structural problem. In the current study, a predictor-corrector method using force control is used. The CFD TAU-Code from the German Aerospace Center (DLR) Institute of Aerodynamics and Flow Technology is used to perform the fluid simulations. Using the finite volume method, the TAU-Code solves the three-dimensional compressible steady or unsteady Reynolds-Averaged Navier-Stokes (URANS) equations. The TAU-Code is also based on the dual-grid approach where, during the pre-processing, a second grid is constructed on the basis of the initial one according to the cell-vertex grid metric.

Fig. 1 shows the structural mesh and the 3D C-type grid, which is used for the simulation of the flow and the mesh used for the structure.

Fig. 1: Meshes used for the FSI computations.

The in-house open source coupling tool EMPIRE is used to couple the structural solver (CARAT++) with the fluid solver (TAU).

Fig. 2: Lift coefficient at Re = 280,000.

Fig. 2 shows that the numerical FSI computations present the same behavior as the experimental results. That is, higher lift and a smooth and delayed stall compared to the rigid computations.

Outlook
In the future, the transient deformation behaviour will be studied. Also, the reduction of load peaks by means of passive contour adaptation will be investigated.

References