USING OPEN SOURCE SOFTWARE FOR SOLVING AEROLASTICITY CASE FOR WIND TURBINE BLADE

Sergei V. Strijhak[1], Georgy A. Shcheglov[2], Pavel S. Lukashin[1,2] and Valeriia G. Melnikova[1,2]

[1] Ivannikov Institute for System Programming of the RAS,
25, Alexander Solzhenitsyn st., Moscow, 109004, Russia
strijhak@yandex.ru, www.ispras.ru/en/

[2] Bauman Moscow State Technical University
5/1, 2-nd Baumanskaya st., Moscow. 105005, Russia
shcheglov_ga@bmstu.ru, www.bmstu.ru/en/

Key Words: *aeroelasticity, wind blade; solver, OpenFOAM, Code_Aster, finite-volume method, finite-element method, mesh, turbulence model, static, dynamic, frequency.*

Due to the development of Wind Energy and construction of new wind farms in Europe, China, India and Russia, there is a need for the solution of application-oriented problems and development of effective methods for calculation of wind turbine's elements. One of the directions for computational continuous mechanics is linked with problems in aeroelasticity. The possibility of solving one of the problem in aeroelasticity using a complex program approach on the basis of open source software OpenFOAM, Code_Aster is shown in this work.

The famous test case with the blade for a NREL 5-MW Reference Wind Turbine, with 61.5 meters long, was selected. The techniques of solving problem for a static and dynamic aeroelasticity in which calculation of flow of the blade with a subsonic air flow is done in OpenFOAM library (solvers simpleFOAM and pimpleFOAM with the Spalart-Allmares turbulence model) are considered. The calculation of the intense deformed status of the wind turbine blade is done in Code_Aster code.

The flowcharts for 3 different approaches for solving problems of aeroelasticity, examples of scripts and command files for data transfer between two codes in the course of calculation are provided. The control-volume mesh consisting the hexahedral elements, the total number is about 400000 cells – 2 million cells, for calculation of flow around the blade with RANS, URANS, DES methods is constructed in OpenFOAM.

The finite-element mesh consisting of triangular shell elements of first order, the total number is about 10000, for calculation of the intense deformed status is constructed in Salome-Meca code. The results of calculation are provided in the form of fields for pressure and velocities; graphics for residuals of pressure, velocity, viscosity; projections of aerodynamic force from time; diagrams of displacement and stress; the frequencies of external force, the values of pressure for points on the surfaces and displacement of the tip of the blade from time. The simulations were run using resources of HPC cluster UniHUB of ISPRAS. The project was supported by RFBR (grant No. 17-07-01391, 17-08-01468).