COUPLED CFD-STRUCTURAL ANALYSIS OF SOLAR PANEL PARKING SPOT

Jakub Jakubec
Juraj Paulech
Vladimír Kutiš
Emil Mojto
Vladimír Goga

Slovak University of Technology in Bratislava
Faculty of Electrical Engineering and Information Technology
Ilkovičova 3, 812 19 Bratislava, Slovak Republic
e-mail: jakub.jakubec@stuba.sk

Abstract

Nowadays, the issue of the impact of weather conditions on the mechanical structures is utilized by appropriate engineering using computer simulation softwares. One of them is software ANSYS which solves the multiphysics problems.

Key words: fluid analysis, structural analysis, CFD, ANSYS.

INTRODUCTION

Wind force acting on the structure causes its deformation, which is automatically accompanied by the emergence of supporting parts mechanical stress. Considering such problems it is optimal to start with the simulation of airflow which defines the external pressure effects on the structure and causes its subsequent deformation. These problems are solved by interconnection of the fluid (CFD) and structural analysis. CFD—structural analysis of parking spot structure with solar cells for recharging electric vehicles (Fig. 1) will be presented.

GEOMETRY OF THE COMPUTATIONAL MODEL

Geometry of model is based on real solar panel parking spot [1], Fig. 1. The parking spot is designed for two cars – especially electric vehicles, due to possibility for charging the batteries during parking period. Main dimensions of the parking spot are: length 3.8 m, width 5 m, height 3.2 m [1]. Solar panel itself includes 12 solar cell units, its dimensions are length 3.96 m, width 4.92 m and thickness 10 cm. Inclination of the panel is 18 degrees (detached to the horizontal plane). Solar panel was modeled as one continuous part considering some simplifications. Mechanical material properties of the solar panel were simplified according to fractions of individual components from that the panel consists of (solar cells, steel reinforcement, bottom metal cover and top transparent cover material, air), so artificial Young modulus of the solar panel was 3.5 GPa, Poisson ratio 0.3 and artificial density 350 kg.m⁻³. Geometry of the solar panel for parking spot was created in software Solid Edge [2] and is shown in Fig. 2.

Fig. 1 Solar panel parking spot [1]

Fig. 2 Geometry of the parking spot created in software Solid Edge

MESH OF THE MODEL

We studied influence of direction and strength of the wind on our construction. Under these conditions, coupled CFD – structural analysis is needed. So we needed to create two different finite meshes: for CFD analysis of the air that flow off the solar panel parking spot, and structural mesh for the construction of the parking spot itself. CFD mesh was created in software ANSYS ICEM CFD [3]. Air region was considered as a box with dimensions 50×25×15 m (length×width×height). As it is in all cases of CFD studies based on Finite Volume Elements (FVM) method, all near-wall regions were modeled in detail. The CFD mesh consists of more than 3.5 million elements. Detail of the mesh is shown in Fig. 3.
The construction of parking spot was meshed in ANSYS Workbench [3]. High number of solid elements was needed because of thin-walled profile of the construction and because of high demands on mesh quality in the regions where maximum mechanical stress are expected. Total number of solid elements for all three supporting parts is more than 570 000. Detail of the construction’s mesh is shown in Fig. 4.

**BOUNDARY CONDITIONS AND SOLUTION PROCESS**

The analysis was performed in ANSYS Workbench environment as one-way coupled CFD–structural analysis. The mesh of air region was interconnected with the structural mesh by general connection.

Horizontal directions of the wind (Fig. 5) and two wind intensities were considered: a) 120 km.h\(^{-1}\), b) 150 km.h\(^{-1}\) (considered as limiting value under our climatic conditions). The construction was fixed to the ground. Effect of solar panel parking spot self-weight was also considered.

The task was solved using cluster computer: 15×CPU@4.0 GHz, 7 GB of RAM occupied, and solution of one case took 10 hours.

**CONCLUSION**

Results for mechanical stress and deformation are shown in Fig. 6 and Fig. 7, respectively. Maximum von Mises stress value 55 MPa is under Yield stress value for chosen construction steel.

It should be noted that the task was solved as static problem, while in real terms the wind speed changes in time.

**References**

[1] Infinite Energy: Electropor, Kinslea Works, Kingston Road, Leatherhead, Surrey. KT22 7LE. UK.


[3] ANSYS Swanson Analysis System, Inc., 201 Johnson Road, Houston, PA 15342/1300, USA.

**Acknowledgement**

This work was supported by Grant Agency VEGA, project No. 1/0534/12, Grant Agency KEGA, project No. 015STU-4/2012, Slovak Research and Development Agency under the contracts APVV-0450-10 and APVV-0246-12.