Aerodynamic Analysis by Computational Fluids Dynamics in Building: Case Study

Authors : Javier Navarro Garcia, Narciso Vazquez Carretero

Abstract : Eurocode 1, part 1-4, wind actions, includes in its article 1.5 the possibility of using numerical calculation methods to obtain information on the loads acting on a building. On the other hand, the analysis using computational fluids dynamics (CFD) in aerospace, aeronautical, and industrial applications is already in widespread use. The application of techniques based on CFD analysis on the building to study its aerodynamic behavior now opens a whole alternative field of possibilities for civil engineering and architecture; optimization of the results with respect to those obtained by applying the regulations, the possibility of obtaining information on pressures, speeds at any point of the model for each moment, the analysis of turbulence and the possibility of modeling any geometry or configuration. The present work compares the results obtained on a building, with respect to its aerodynamic behavior, from a mathematical model based on the analysis by CFD with the results obtained by applying Eurocode1, part1-4, wind actions. It is verified that the results obtained by CFD techniques suppose an optimization of the wind action that acts on the building with respect to the wind action obtained by applying the Eurocode1, part 1-4, wind actions. In order to carry out this verification, a 45m high square base truncated pyramid building has been taken. The mathematical model on CFD, based on finite volumes, has been calculated using the FLUENT commercial computer application using a scale-resolving simulation (SRS) type large eddy simulation (LES) turbulence model for an atmospheric boundary layer wind with turbulent component in the direction of the flow.