Summary of doctoral thesis

The doctoral dissertation focuses on wind loading on structures, using a cooling tower as a case study. It begins with a literature review on wind-flow characteristics and on wind–structure interaction. Review consist of design calculation methods under various standards, including Eurocode, ASCE 7-10, and others. The work also reviews methodologies for wind-tunnel testing, including techniques for generating the Atmospheric Boundary Layer (ABL). Selected measurement methods for determining wind speed and wind pressure are described. The thesis shows fundamental laws and equations of fluid dynamics and presents selected turbulence models.

The dissertation consists of tree main parts: experimental part with wind tunnel testing, numerical CFD analysis of wind tunnel experiment and main CFD calculation of full-scale model of chimney cooling tower. Whole research is performed for the geometry of catenoidal cooling, based on structure located in Opole Power Plant.

The laboratory investigations were conducted at the Silesian University of Technology, with assistance from experts of the Institute of Thermal Technology. The cooling-tower model was tested in a wind tunnel with a 2.8 m-long and 1,2×1,2m width test section. A 1:400 scale model was used, with a diameter of approximately 20 cm. Examined flow velocities varies from 2.5 to 18.5 m/s due to the wind tunnel's technical capabilities. Generation of the Atmospheric Boundary Layer (ABL) was neglected. The model was placed on a prepared table. This avoided exposing the model to flow gradient in the near-wall regions of the tunnel. Particle Image Velocimetry (PIV) was used for measurements of selected velocity fields behind model. The obtained images were processed into velocity fields using the Dynamic Studio software.

The biggest of the dissertation comprises numerical (CFD) calculations that replicate the wind-tunnel experiment. In the preliminary analyses, simple models were developed for canonical cases (a cylinder, flow in an empty tunnel). In the main computations, the geometry of wind tunnel with cooling tower inside was used. A series of sensitivity studies was performed on key parameters, including the mesh and the assumed time-step size. Numerically computed velocity fields were compared with experimental data to validate the model. Based on the simulations, lift and drag forces were assembled, as well as pressure distributions on the surface of shell. The calculations showed that the tunnel geometry has a significant influence on the obtained quantities, and the test results cannot be transferred directly to the real structure. The cooling-tower model was also compared with a cylinder model constructed under the same assumptions to demonstrate the differences.

In the final and most important part, computations were performed for a full-scale cooling-tower model with a height of 185 m. Drag, lift, and the pressure distribution on the shell surface were determined from the simulations. A mathematical description of the distribution was attempted, using a Fourier series and linear interpolation. The results show that the flow around the cooling tower differs

noticeably from the flow around a cylinder. They also show that wind acts on the tower differently than on the cylinder.

ANSYS Fluent was used to perform the simulations. Large-model computations were executed on the Ares cluster in the PL-Grid network. Result interpretation and approximation were created with Wolfram Mathematica. Additionally, a dedicated console application in C# was written for the statistical analysis of large amount, of results from Fluent outputs.

Maciej Wiśniowski

29.09.2025