CADFEM Consulting

Wind Tunnel Shape Optimization Using ANSYS CFD

Based on the simulation results, optimal design of a small-scale wind tunnel was obtained

Contact person:

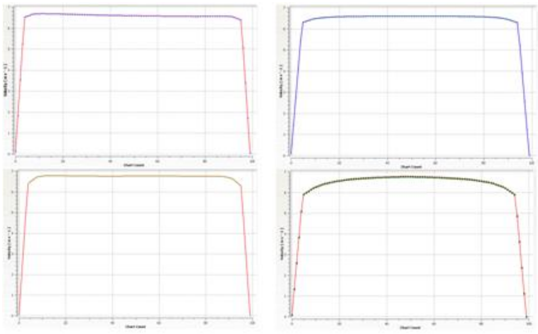
Andrey Krylov
Tel: +7 (495) 644-06-08
E-Mail: andreyk@cadfem-cis.ru

Goal and Subject of Simulation

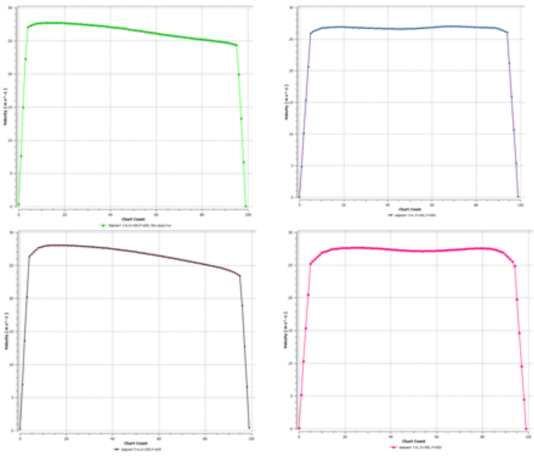
The main objective of the CFD simulation was to optimize the design of a small-scale closed circuit wind tunnel (customer – Moscow State University of Civil Engineering). The purpose was to get maximum uniformity of velocity profiles in front of the tunnel operating area. The designed wind tunnel is to be used in measuring wind loads applied to new models of buildings and engineering structures.

ANSYS FLUENT12.1 was used in the process of analysis. The simulation was performed on the Cray CX1 high-performance supercomputer.

Basic results

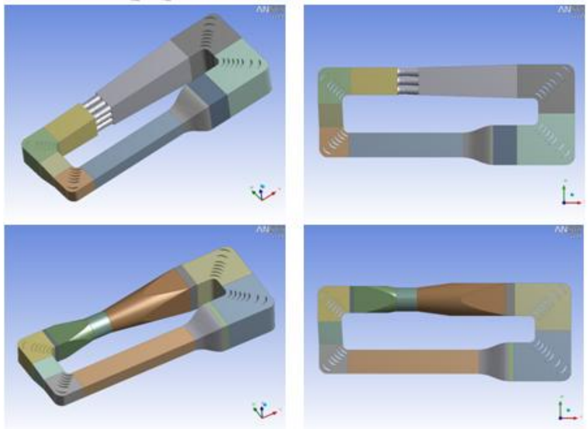


Velocity profiles in horizontal and vertical sections, distance – 2 meters from the nozzle, flow rate – 100 m3/s.

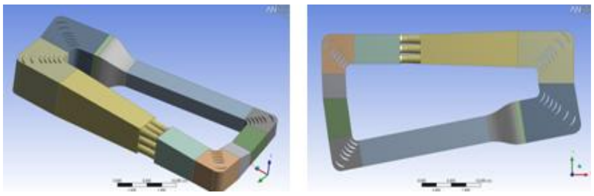


Velocity profiles in horizontal and vertical sections, distance – 15 meters from the nozzle, flow rate – 100 m3/s.

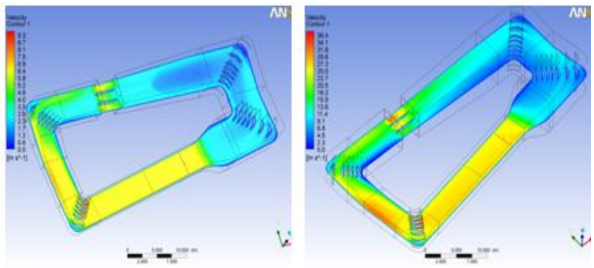
Construction elements: honeycomb, 9-block fan, cooled blade.



Studied alternative designs: with a chamber oversized and with a mono-fan



Optimized design of the wind tunnel has 10 degree opening to the symmetry plane



Velocity profiles in horizontal section of a wind tunnel for minimum flow rate 100 m3/s and nominal flow rate 300 m3/s.

The material is published upon authorization of Moscow State University of Civil Engineering.