WIND TUNNEL SIMULATION OF LOW RISE BUILDINGS WITH AN ARTIFICIAL WIND TUNNEL USING COMPUTATIONAL FLUID DYNAMICS SOFTWARE

T. C. Dodanwala[1], H. M. Ariyasena[1], K. N. D. P. Silva[1], P. A. S. Karunarathne[1], A .A. Hashmi[1], N. S. Fernando[1], R. S. Mallawaarachchi[2]

[1]Department of Civil & Infrastructure Engineering, Faculty of Engineering, South Asian Institute of Technology and Medicine (SAITM), Sri Lanka.
[2]Senior Lecturer, Department of Civil & Infrastructure Engineering, Faculty of Engineering, South Asian Institute of Technology and Medicine (SAITM), Sri Lanka,
E-mail: rajeev.saitm@gmail.com

ABSTRACT

It can be seen from literature and from past events over the last 5-10 years, a significant development in the frequency of heavy wind situations in a more localized manner over many parts of the country causing lots of damage to the properties, infrastructure facilities and human lives. Although much attention is paid in the design stage of high rise buildings for the adverse impacts from possible cyclonic and heavy wind situations in the country, less or no attention is paid in constructing low-to medium rise buildings especially in inlands, either due to lack of awareness on disaster resistant construction techniques or assuming that there is no such vulnerability. The 1978 Sri Lanka Cyclone (04B) on 23rd November, 1978 is the worst ever cyclone in the Sri Lankan history. It was an eye-opener for academics, researchers and many other parties involved in the construction sector. Hence, research were conducted, wind zonation maps were prepared and guidelines were established for better and safe houses low to medium rise buildings.

The British Standard Code of CP 3: Chapter V: Part 2: 1972 is widely used in designing buildings for wind loads since last 2-3 decades. Because of the complex nature of the wind behavior around objects, the actual wind loads acting on a building can never be predicted from code based calculations, although there are factors in the design codes to consider the topography (S_1) and ground roughness, building shape and height above ground of a building (S_2). Such detailed design parameters can only be obtained by conducting a wind tunnel simulation of a physical scaled down model of the building together with its surrounding. Since it is financially not viable to conduct such studies for low to medium rise building, an attempt was made in this research to create an artificial wind tunnel using Autodesk Computational Fluid Dynamics (CFD) software Autodesk Flow design. The surface pressure values obtained from the computer model was validated with manual calculations based on CP 3: Chapter V: Part 2: 1972. A physical scaled down model of a single story house with gable walls and a pitched roof will be modelled and tested in a small boundary layer wind tunnel in future for further validation of the surface pressure coefficients obtained from the computer model.

***Key words*:** Heavy wind situations, Wind tunnel simulations, Autodesk Flow Design