Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 190 (2017) 377 – 384

Procedia Engineering

www.elsevier.com/locate/procedia

Structural and Physical Aspects of Construction Engineering

Experimental Validation of Computer Fluid Dynamics Simulation aimed on Pressure Distribution on Gable Roof of Low-rise Building

Juraj Králik[a,*], Lenka Konečná[b], Dagmar Lavrinčíková[a]

[a]Slovak Univesity of Technology in Bratislava, Faculty of Architecture, Institude of Constraction in Architecture and Engineering Structures, Námestie Slobody 19, 812 45 Bratislava, Slovakia
[b]Slovak Univesity of Technology in Bratislava, Faculty of Civil Engineering, Department of Structural Mechanics, Radlinského 11, 810 05 Bratislava, Slovakia

Abstract

Modeling of air flow over buildings belongs to a challenge that was accepted by several authors. There are couple reasons for it, from which the most frequent is simulation in order to obtain pressure or external pressure coefficients distributions. These simulations are always a balance between accuracy and computer needs and consumed time. Aim of this contribution is to validate Computer Fluid Dynamics (CFD) simulation by experimental measurement in Boundary Layer Wind Tunnel (BLWT). A low-rise building with gable roof will be examined and results will be compared to wind tunnel test. For the purpose of this simulation were chosen Delayed Detached Eddy Simulation (DDES) and Scale-Adaptive Simulation (SAS) turbulence models. It is observed that the DDES model failed in predicting pressures around the roof ridge. The average deviations on the gable roof from the BLWT measurements are 37.2 % for DDES and 26.7 % for SAS model.

Peer-review under responsibility of the organizing committee of SPACE 2016

Keywords: experiment; Ansys; airflow; presure; presure coeficients; gable roof

1. Introduction

As Computer Fluid Dynamic (CFD) software develops, problems of fluid dynamics becoming interesting for more engineers. CFD is a handy tool capable of reasonable predicting of air-flows and in this article will be used to predict pressure distribution on simple rectangular low-rise building with gable roof. Setting up CFD simulation using table

* Corresponding author. Tel.: +421-903-951-403.
E-mail address: kralik@fa.stuba.sk

Peer-review under responsibility of the organizing committee of SPACE 2016
doi:10.1016/j.proeng.2017.05.352

PC is always a balance between accuracy, consumed time and computational demands needed for simulation. And at the same time there's no universal turbulent model, in fact there are several turbulence models which are being offered by several commercial or non-commercial software. There are three turbulent flow simulation methods Reynolds Averaged Navier-Stokes Simulations (RANS), Scale Resolving Simulations (SRS) and Direct Numerical Simulation (DNS). For the purpose of this analysis was used commercial software package ANSYS Fluent R16.2.

Some experiments in wind tunnel were carried out by Ho, where he was in his work, [1], was describing the background of the project, the basic models, testing configurations, the wind simulation, the standard archival format for distribution of the data, and a basic analysis of the data. He claims that the data obtained within his study are consistent with the expected aerodynamic behavior and comparisons with full scale data show that the wind tunnel tests match the full-scale reasonably well, but cannot reproduce the largest of the peak point suctions near roof edges.

The air flow around isolated gable-roof buildings with different roof pitches was investigated by wind tunnel experiments and Computational Fluid Dynamics (CFD) simulations based on steady Reynolds Averaged Naviere Stokes equations (RANS) model by Tominaga, [2]. He used three different pitches, 3:10 (16.7°), 5:10 (26.5°), and 7.5:10 (36.9°) and furthermore he tested four turbulence models, namely, the standard k-ε, the RNG k-ε, the realizable k-ε, and the k-ω SST model. He observed large difference in the flow patterns is between the pitches, this implies that the flow pattern around a building with a pitched roof changes critically at a roof angle of around 20°.

Another research based on wind tunnel experiment was done by Krejsa, [3], where he in his work was studding aeroelastic behavior of bridge deck under influence of the wind. His experiments show that the traffic can have influence on the stability of the bridge. And the traffic situated leeward influenced the heave damping, perhaps due to different reattachment of the flow at the bridge deck, creating the vertical negative net damping force. On the other hand the bridge was mostly sensitive to torsional flutter, while setup without traffic was the most unstable. Thus, such influences cannot be neglected, when analyzing the instability of a bridge loaded by the wind.

A low-rise building was constructed near Shanghai Pudong International Airport by East China Sea to study the characteristics of wind field and wind pressure on the roof of the building. The remarkable feature of the test building is that the roof pitch can be adjusted range from 0° to 30°. This analysis was done by Xu, [4], where he in his work was analyzing wind pressures on gable roof with different roof pitches (0°, 10° and 20°) and then comparison was done with a wind tunnel test on a rigid model of 1:30 scale. His field measurement was consistent with that by wind tunnel test. Furthermore he analyzed the probability distributions of fluctuating pressures, which agrees well with Gaussian processes when the skewness is larger than -0.5, while having better agreement with Gamma distribution when the value of skewness is between -1 and -0.5.

Aim of this contribution is to compare experiment in Boundary Layer Wind Tunnel (BLWT) with CFD simulation. As experiment object was chosen a low-rise building with gable roof with roof pitch of 20°. Experiment was focused on obtaining pressure values from pressure taps in selected locations on the gable roof. During this experiment were also measured velocity profiles. CFD final volume element model was created with respect to y+ value, with limit to accuracy set by number of polyhedral elements to around $2 \cdot 10^6$ (a table PC task).

Nomenclature

z_0	terrain roughness
z_{ref}	reference high
u_{ref}	reference velocity
u_{fric}	friction wind velocity
κ	Von Karman constant
k	turbulent kinetic energy
ε	turbulence dissipation rate
ω	specific turbulence dissipation rate
C_μ	model constant