

INFLUENCE OF TOPOGRAPHY ON WIND PRESSURES IN TANKS USING CFD

O.A. FALCINELLI[†], S.A. ELASKAR^{†,‡} and L.A. GODOY^{†,‡}

[†]*Faculty of Exact, Physical and Natural Sciences, National University of Córdoba
P. O. Box 916, Córdoba 5000, Argentina. lgodoy@com.uncor.edu*

[‡]*CONICET*

Abstract — The influence of topography on wind pressures acting on structures has been of interest to the civil engineering community for some time; however, because of the complexity of the problem, only few cases have been solved. The evaluation of pressures in tanks located in hills is one of those complex problems and has not been addressed in the literature. This paper presents a computational fluid dynamics simulation of the problem, in which the domain including the hill and the tank is discretized using finite elements. The results show that the actual location of the tank with respect to a hill has a significant influence on the pressures, so that tanks located at the top of a hill undergo severe increases in pressure coefficients and also changes in pressure distributions around the tank.

Keywords— CFD, finite elements, tanks, topographic effects, wind pressures

I. INTRODUCTION

This paper addresses the computational modeling of wind pressures acting on the cylindrical part and on the roof of storage tanks, which are placed at different locations with respect to a hill. Wind pressures on above ground steel storage tanks may be evaluated by means of boundary layer wind tunnel (BLWT) simulations, or by computational techniques using computational fluid dynamics (CFD). Both methodologies have experimented considerable improvements over the past few years, so that advances in computing power, electronic instrumentation and computer-based rapid prototyping have improved the efficiency of BLWT simulations, whereas CFD models have benefited from computing power and improved software. There are still several advantages in using BLWT over CFD simulations, especially whenever the flow is dominated by turbulence, as in the present case. However, BLWT facilities are not available in most academic laboratories (even less so in developing countries) and those that have the facilities tend to be overloaded. Thus, CFD models are a useful tool to obtain estimates of pressures in structural configurations with reasonable costs. This paper addresses such situation, in which the influence of topographic effects on wind loaded tanks is of interest.

The importance of taking into account the influence of topography to estimate wind pressures has been recognized for some time in the context of civil engineering structures. Current codes of practice, such as the

ASCE 7 (2006) provisions, include factors to account for the location of a structure with respect to hills and escarpments. Topographic factors relate wind velocities in open terrain with velocities at specific locations in a geographic accident (Jackson and Hunt, 1975; Lemelin *et al.*, 1988); however, considering the way in which they are formulated, those factors appear to be independent of the specific structure under study. For more complex situations, the ASCE document recommends the use of engineering judgment, expert advice, or wind tunnel studies. This work explores an alternative in which a computer simulation of the problem is carried out using computational fluid dynamics and the finite element method.

There are several CFD and wind tunnel studies performed on different hill types as well as on isolated tanks, but not the two taken in conjunction. Regarding wind loads on tanks, the most important reference known to the authors is the study in three parts reported by Macdonald *et al.* (1988, 1990a, 1990b), in which wind tunnel measurements were carried out for isolated tanks, and also for groups of tanks. Macdonald and co-workers also discussed the problems faced in representing Reynolds numbers using similitude theory for low-rise structures. Another class of open top tanks was addressed in the literature by Holroyd (1983). The flow over hills has been studied by several researchers using CFD, including Bergeles (1985), Taylor (1998), Kim *et al.* (2000), Lun *et al.* (2003) and Bitsuamlak *et al.* (2006). Most researchers model a two dimensional domain; however, a three-dimensional volume is needed in the present case because the presence of the tank in the hill destroys the plane condition.

Section II in this work contains the general fluid dynamics formulation of the problem. The computational model to simulate the air flow around a structure which is located in a hill, based on a general-purpose finite element code, is explained in Section III. The model is validated in Section IV by comparison with the results of two benchmarks originally solved by other authors. Results of the flow for an isolated tank located on a horizontal plane are given in Section V; whereas Section VI reports new results that include both a hill and the structure.

II. BASIC FLUID DYNAMICS FORMULATION

The planetary boundary layer near ground is simulated in this paper under the assumptions of stationary mean flow, viscous, incompressible, isothermal, and turbulent