Tunnelling and Underground Space Technology 73 (2018) 222–235

Contents lists available at ScienceDirect

Tunnelling and Underground Space Technology

journal homepage: www.elsevier.com/locate/tust

Study on numerical modeling of jet fans

Aleksander Król[a,*], Małgorzata Król[b]

[a] Faculty of Transport, Silesian University of Technology, Poland
[b] Faculty of Energy and Environmental Engineering, Silesian University of Technology, Poland

ABSTRACT

An overview of ways of jet fan modeling with the use of Ansys Fluent methods is presented. The theoretical models were built and validated basing on the experimental measurements made by Giesen et al. (2011). The results obtained with the all models were compared with this experimental data and next the detailed discussion was provided. A special attention was paid to the turbulence modeling, the range and the shape of the jet produced by the fan.

Ansys Fluent supports three suitable approaches allowing for accurate mapping of the axial fan operation, which differ in the degree of simplification of the reality. The first approach involves modeling a fan as a volume source of momentum. All three momentum components can be set, so the fan axis can be freely oriented in the space. The second approach involves modeling a fan with known characteristic. A fan is here a 3D object, but the fan rotor is a plane surface of zero thickness. The model is more accurate, but requires a set of parameters, which should be determined by the measurements. In the third approach a fan is also modeled as a 3D object, in this case the rotor has the real spatial dimensions. This model is capable to calculate all fan features by itself, but requires more computational resources. Apart from the above the fourth fan model as a velocity inlet is willingly used.

The selection of the model is dependent on data available and required minuteness of the fan representation and operation.

1. Introduction

The proper operation of road tunnels requires well functioning ventilation systems. The most often applied for short or medium long tunnels is the longitudinal ventilation system. Operation of the longitudinal system involves the use of jet fans. The fans should ensure relevant parameters of the air inside a tunnel during the normal tunnel operation (even in traffic congestion). They should also provide the conditions which enable the evacuation in case of fire. After the fire extinction the system should be able to clean up the tunnel of the combustion products.

The operation of jet fans in a road tunnel depends on many parameters. Their function is affected by the proximity of tunnel walls, its slope, its eventual curvature or interaction between the smoke plume and the tunnel ceiling (in case of fire). The operation of a longitudinal ventilation system and fans work are also vulnerable to ambient conditions as wind or the temperature difference at portals resulting from insolation.

The crucial role of ventilation systems to ensure the safety of use of road tunnels led to many scientific researches on proper operation of ventilation systems (Wang et al., 2017; Cascetta et al., 2016; Jin et al., 2017; Mei et al., 2017; Król et al., 2017). The studies concern either the influence of different parameters as tunnel inclination or fans arrangement on efficiency of ventilation system or just the optimization of fans operation. The mentioned analyses are often done for reduced scale models (Wang et al., 2017; Mei et al., 2017; Yao et al., 2016; Tanaka et al., 2016; Kashef et al., 2013; Tang et al., 2016). Such studies done in real tunnels in current use are much less often due to difficulties with coordination of many services work and necessary temporary traffic turn off (Memorial Tunnel Fire Ventilation Test, 1955; Król et al., 2017). The most extensive research on the subject of jet fan in a large enclosure has been made by van Oerle (van Oerle et al., 1999). The studies on fans work or ventilation systems operation are conducted by the use of Computational Fluid Dynamics (CFD) methods as well (Ang et al., 2016; Yu et al., 2016; Colella et al., 2011; Lin et al., 2014; Eftekharian et al., 2014; Musto and Rotondo, 2014, 2015; Cascetta et al., 2016).

Research on the development of the stream generated by jet fans has been carried out for many years. In the initial period they were based mainly on real measurements and theoretical considerations. Forstall

* Corresponding author at: Faculty of Transport, Silesian University of Technology, Krasińskiego Street 8, 40-019 Katowice, Poland.
E-mail address: aleksander.krol@polsl.pl (A. Król).

https://doi.org/10.1016/j.tust.2017.12.024
Received 4 October 2017; Received in revised form 11 December 2017; Accepted 24 December 2017
0886-7798/

Nomenclature

d_{out}	outlet diameter
G	turbulence generation rate
h	thickness of the toroid region swept by the blades in the axial direction
I	turbulence intensity
k	turbulence kinetic energy
l	length of the fan
$\Delta P(Q)$	pressure rise cross the fan for a given axial flow rate Q
r	local radial distance from the fan axis
R_h	radius of the fan hub
R_{ij}	Reynolds viscous stress tensor
R_{ip}	radius of a point on the fan blade based on the inflection point ratio
R_t	radius of the fan blade tip
S_a	axial volume unit force
S_r, S_t	radial and tangential unit forces
U_r, U_t	radial and tangential velocity components
u, u_i	velocity, velocity components
u', u_i'	fluctuant component of velocity
u_{out}	outlet velocity
$u(x)$	velocity distribution
U, U_i	averaged velocity, averaged velocity component
U_ϕ	local tangential velocity
W_{fan}	fan power
Γ	diffusion coefficient components
ε	kinetic energy dissipation rate
μ_T	turbulent viscosity
ρ	density
ω	specific kinetic energy dissipation rate
Ω_{oper}	fan operating angular velocity

stated that the dependence of the mean velocity in the fan axis to the average velocity in the remaining volume is the basic value determining the range and distribution of the jet (Forstall and Shapiro, 1951). The jet behavior in isothermal conditions was well recognized regarding both the structure and mechanisms (Ricou and Spalding, 1961; Rajaratnam and Pani, 1974; Rajaratnam, 1976; Launder and Rodi, 1983). Durao in his experiments compared the range of free jet and wall jet (Durao and Whitelaw, 1973). While Champagne studied the velocity profiles in the fully developed jet region (Champagne and Wygnanski, 1971). A lot of researchers concentrated their study on the far-field of the free jet also (Wygnanski and Fiedler, 1969; Rodi, 1975; George, 1989; Dowling and Dimotakis, 1990). Nowadays more and more frequent tests of free jet and wall jet are carried out using numerical methods.

CFD methods allow for multivariate analyses and give the opportunity to examine many models of a system, what is of big importance when optimizing a system or device. There are two software packages most willingly used in such analyses: Ansys Fluent and Fire Dynamics Simulator (FDS) (Kim et al., 2008; Ko and Hadjisophocleous, 2013; Ang et al., 2016; Brzezińska and Sompolinski, 2016; Cascetta et al., 2016; Musto and Rotondo, 2015; Yu et al., 2016). Other programs are also widely used for modeling of jet fans. STAR-CCM software is a program with a broad range of engineering applications (Hamzehloo and Aleiferis, 2016; Zhu et al., 2014). Available turbulence models are here LES model, Reynolds Stress Model and DES model (DES is a combination of the LES model in the basic volume and the RANS model in the boundary layer). Another program for modeling of flows generated by jet fans is PHOENICS. This program allows for a wide selection of turbulence models too: in addition to commonly used turbulence models also LVEL Turbulence Model (useful for conjugate-heat-transfer problems) and Multi-Fluid Model (useful for turbulent mixing and chemical reaction) can be used (Huanga et al., 2015; Barbato et al., 2014). Jasmine is one more program used for discussed purposes. It implements the RANS turbulence model (Kumar et al., 2010).

However, the use of such highly specialized software does not guarantee the correctness of obtained results. The great number of available physical models and detailed solutions, which can be applied (it particularly concerns Ansys Fluent) allows for high degree of flexibility of being built models of systems or devices. It is a convenient and powerful way of building models, but on the other hand the models need to be validated by the measurements obtained in real experiments. It is sometimes difficult or even impossible, that is why it is essential to fully understand the ways of modeling the devices which are the components of ventilation system in a road tunnel and the phenomena which occur. One should be aware at the model building stage that the selection of a particular sub-model entails consequences influencing the calculated final data.

If the analysis regards jet fans, which are the most important components of a longitudinal ventilation system, the Ansys Fluent package supports four ways of their modeling. A jet fan can be mapped in the model as:

- A pair of inlet and outlet surfaces.
- A volume with assumed momentum source.
- A single zero-thickness surface with fan boundary condition (it models the fan rotor), there are three sub-models available.
- A finite volume in which fan rotor is modeled, there are also three sub-models available.

These models are more widely described in the paper. As it will be shown there are many options and what is more important each model will give different results.

The model, which often appears in CFD analyses of longitudinal ventilation system is inlet-outlet model. Here an object, with the surfaces of types inlet and outlet is introduced and the required outlet velocity is directly set (Wang et al., 2012; Eftekharian et al., 2014; Enright, 2014; Lin et al., 2014).

The momentum source model is also willingly applied (Musto and Rotondo, 2014, 2015; Se et al., 2012). It is the simplest, realistic model available in Ansys Fluent. It generates the velocity distribution and models the pollutant transport, e.g. demonstrates the movement of fire gases through the fan, as in real conditions.

Model of fan as a surface boundary condition is much less often applied. The need of using such relatively complex model arises when the structure of airflow just behind of the fan outlet becomes important. For instance Moonen et al. used it (with fan curve defined) for modeling of the flow conditions in a wind tunnel (Moonen et al., 2006).

The last mentioned model, a rotor of finite dimensions is extremely rarely used. It is due to its complexity. However, this model is regarded as the best reproduction of a jet fan, it is comparable with detailed models with individual blades imaging (Ansys Fluent, 2013).

The essence of the numerical studies is the most reliable representation of examined devices and processes ongoing when the devices work. The operation of a jet fan produces a jet, which range and the spatial distribution of velocity and turbulence are of great importance for the efficiency of the system of longitudinal ventilation and smoke removal. The selection of a jet fan model is obviously dependent on examined phenomenon, but after the literature review it seems that it is often done randomly, without deeper considerations. When analyzing a jet distribution at a considerable distance from a fan, the simple model can be applied. However, when examining the influence of inclination of deflector blades on the shape of generated jet, the most reliable modeling of jet fan seems to be necessary.

The source area of the jet is strongly turbulent, which additionally