CFD simulation of aerodynamic resistance in underground spaces ventilation

I. Diego, S. Torno & J. Toraño
GIMOC, Mining Engineering and Civil Works Research Group, Oviedo School of Mines, University of Oviedo, Independencia 13, 33004 Oviedo, Spain

Abstract

The main problems once someone gets deep inside an underground space are the lack of light and air. If the underground space is long enough or there are pollutants or heat sources inside the space the need for a ventilation system becomes essential. In the particular case of road tunnels, the need to dissipate engines fumes also raises the need to account for the high risk present of fire inside the tunnels. It seems clear that whatever the underground space is designed for, a ventilation system must be present during the construction and subsequent operative phase of the installation. A state of the art tool to calculate the ventilation system is computational fluid dynamics (CFD). The air in the underground space is discretized in finite volumes and mathematical methods are used to obtain the pressure and velocity fields all over the domain, capturing and analyzing all possible flow details, no matter the geometry if the mesh is fine enough. One of the main goals in classical ventilation calculations is to obtain the overall pressure drop of the air as it passes through the cavity, as this will guide the sizing and selection of the installation fan(s). This paper uses the commercial CFD code Ansys CFX 10.0 to calculate the pressure losses in a tunnel installation, comparing the results with the classical frictional losses calculations procedure. These simulations will guide a methodology of using CFD to calculate sections of the underground ventilation or even, if mesh sizes and computer means are big enough, to fully calculate ventilation over all domains. These studies have been carried out in the framework of the Research Project CTM2005-00187/TECNO, “Prediction models and prevention systems in the particle atmospheric contamination in an industrial environment” of the Spanish National R+D Plan of the Ministry of Education and Science, 2004–2007 period.
Keywords: CFD, turbulence, tunnel resistance, pressure drop.

WIT Transactions on the Built Environment, Vol 102,
www.witpress.com, ISSN 1743-3509 (on-line)
doi:10.2495/US080131

1 Introduction

The research group of the Mining Engineering and Civil Works of the University of Oviedo, based in the School of Mines of Oviedo (Spain) is developing a research project granted by the Spanish Ministry of Education and Science. One of the project goals is to develop and study dust emission factors in several industrial situations, from quarry blasts to ship unloading or loading facilities. The researches and simulations have been very successful in several fields obtaining interesting results in the case of dust coming from quarry blasts, both using computational fluid dynamics (CFD) [1] and classical dispersion methods, [2]. The research group has also experienced simulating air flows in underground spaces ventilated through auxiliary systems [3].

CFD seems to be the perfect tool to calculate air flows in underground spaces, both the velocity fields [4, 5] and the pressure fields [6, 7], with immediate applications in HVAC design [8] and security regarding fires and smoke [9].

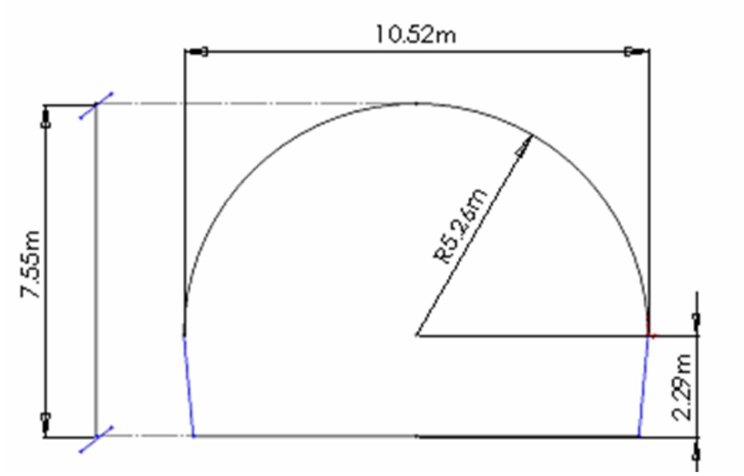

Figure 1: Tunnel section.

The next step to be taken is to translate the dust dispersion calculations to underground environments. In order to check the accuracy of the CFD simulations the pressure drop calculated by the CFD methods will be compared to the results of the classical frictional methods based in Darcy-Weisbach formulations. An assumption is made: if the pressure drop calculation is correctly developed by the CFD model then its accuracy would be proved and then the calculated velocity/pressure fields will be used as a base for multiphase simulations involving species other than air.

In this paper a section of 500 meters of a tunnel, Figure 1, will be simulated and the flow through it simulated. Its cross section is 67.07 square meters and its perimeter is 31.22 meters.

2 Theoretical head losses

To calculate the head losses created by turbulent flows in non circular ducts the modified Darcy-Weisbach equation [10] is used:

$$\Delta_{p,f} = \frac{1}{2} \cdot \frac{f \cdot L \cdot V^2 \cdot \rho}{D_H} \tag{1}$$

where
$\Delta p,f$: fictional head loss (Pa)
V: velocity of the air flow (m/s)
f: friction factor
ρ: air density (kg/m^3)
D: hydraulic diameter of the duct (m)
L: length of the duct (m)

In order to calculate f there are several empirical equations that relates f to Reynolds number and to the roughness of the wall. There will be used the Colebrook-White one [10]:

$$\frac{1}{\sqrt{f}} = -2\log\left(\frac{\varepsilon_r}{3.7} + \frac{2.51}{Re\sqrt{f}}\right) \tag{2}$$

where
f: friction factor
ε_r: relative roughness = ε/D.
Re: Reynolds number

$$Re = \frac{v \cdot D_H}{\upsilon} \tag{3}$$

where
v: velocity (m/s)
D_H: hydraulic diameter (m)
n: cinematic viscosity (m^2/s)

The values of the tunnel roughness are taken from [11] as reflected in table 1.

3 CFD simulation

The equations that govern the fluid flow are the Navier Stokes, eqn (4), which relates the velocity and pressure fields as well as density; the continuity equation, eqn (5), that express the mass conservation; and finally the energy equation, eqn (6), which relates also the temperature fields. These expressions create a system of differential equations that can only be solved, in the vast majority of cases, by numerical methods.

$$\rho \frac{D\overline{V}}{Dt} = -\overline{\nabla} p + \rho \overline{g} + \mu \nabla^2 \overline{V} \tag{4}$$

$$\frac{D\rho}{Dt} + \rho\overline{\nabla} \bullet \overline{V} = 0 \tag{5}$$

$$\rho\frac{D\tilde{u}}{Dt} = K\nabla^2 T - p\overline{\nabla} \bullet \overline{V} \tag{6}$$

where

ρ: Density

∇ $\bullet$: divergence operator

U: velocity vector

p: pressure

μ: viscosity

T: Temperature

∇: Gradient

t: Time

V: velocity

g: gravity

$\tilde{u}$: specific heat

K: conductivity

Table 1: Tunnel roughness.

Tunnel support system	**Roughness (mm)**
Tunnel lining by shotcrete	
Very smooth surface	0.3
Medium conditions	2.5
Rough surface	9
Tunnel lining by concrete	2.5
Tunnel excavated in rock and no lining	
roughness surface	100
high roughness surface	200
irregular surface	300

In turbulent flow there have also to be solved additional equations that allow the calculation of the velocity and pressure fields in all the domains, the so called "turbulence models". Taking into account just the RANS (Reynolds Averaged Navier Stokes) they vary from the more or less simple where just one equation is added to the calculations, as the Spalart-Allmaras model, to the two equations models, k-epsilon or even seven equations, Shear Stress Transport (SST) models. This paper will show calculations that use k-epsilon models and scalable wall transport functions, which simplify the meshes used in the vicinity of the walls. All the calculations will be made using commercial code Ansys CFX 10.0. The domain to be calculated is divided in finite volumes where the former equations will be solved by linear methods. A base in this methodology is to demonstrate the independency of the results to the type and size of the mesh.

The mesh is done in three dimensions in a non structured way, using tetrahedral, prismatic or pyramidal elements developed using software ICEM CFD 10.0 starting from parametric geometrical models developed in

SolidWorks. Tetrahedrons and pyramids cover the most part of the domain, but the prisms are used to gather calculation nodes in the vicinity of the wall, where the velocity gradient is high. Figure 2 shows a cross section and a longitudinal section of the mesh, whereas figure 3 shows details of prism layers close to wall and ground.

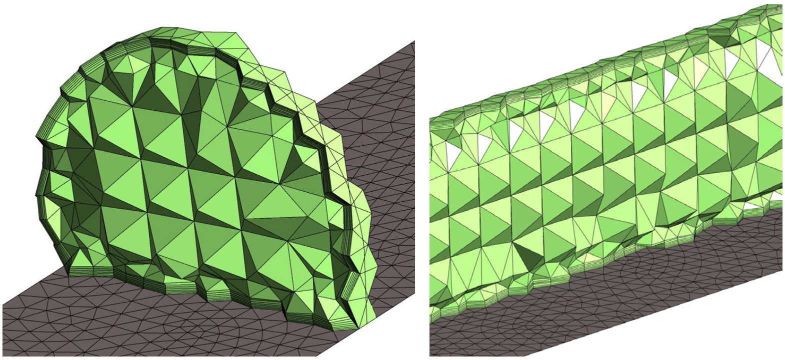

Figure 2: Cross section (left) and longitudinal section (right) of mesh.

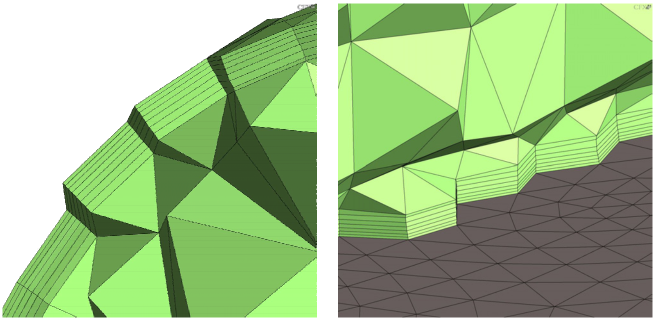

Figure 3: Details of mesh in the vicinity of the walls and ground.

Once the calculations are made and the model adequately converges the post processing of the data files will allow the analysis of the pressure and velocity fields. The CFD head losses will be calculated following the Bernoulli equation (7) as can be found in any Fluid Mechanics book [10]:

$$E_1 = E_2 + h_p = \frac{V_1^2}{2g} + \frac{P_1}{\gamma} = \frac{V_2^2}{2g} + \frac{P_2}{\gamma} + h_p \rightarrow h_p = E_1 - E_2 \qquad (7)$$

where
E: energy (m)
V: flow velocity (m/s)
P: pressure (Pa)
g: gravity (m/s^2)
γ : specific weight (N/m^3)
h_p: head loss (m)

Quantities using a subscript “1” refer to a point in the centre of the tunnel at the entrance. A quantity with a subscript “2” refers to a point at the output.

4 Comparison: theoretical vs. CFD

Dozens of simulations have been done in the 500 m long tunnel domain. Although a condition of symmetry can be applied in the centre of the tunnel, thus allowing the use of half the number of meshing elements, the complete section geometry has been used in order to test the behaviour of the software in this 500 metres section.

Now there will be shown some of the results organized as per the mesh where they were calculated. Figure 4 shows the 4 meshes initially developed in order to accomplish the mesh independency study. Mesh 1 is a fine mesh composed only by tetrahedrons (“tetras”) and pyramids, using mainly tetras of 0.97 m edges. This creates approximately 400.000 elements in the mesh (400k) all over the tunnel domain. As approximately each element involves 1 Kb of RAM total amount of RAM needed in the calculation will be in the 400 Mb ranges, which is easily accessible for any nowadays computer.

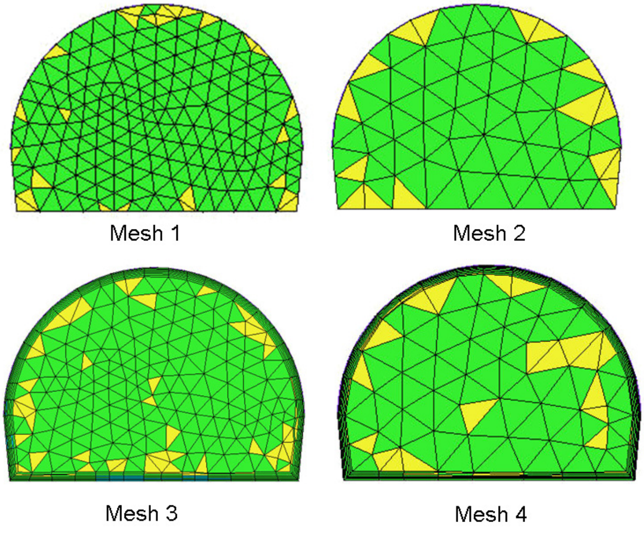

Figure 4: Different meshes used.

Mesh 2 has tetras of 1.2 meters of edge, thus obtaining less resolution, 225k elements. Mesh 3 adds to mesh 1 a layer of prisms around the tunnel walls and ground, creating 400k tetras and around 500k prisms. Memory and calculation times get increased. Mesh 4 adds prisms to mesh 2, obtaining 225k in tetras and 200k in prisms, lump sum of 425k elements. In all cases models obtained can be used in affordable time in computers with at least 1 Gb of RAM with no need to parallel processing.

Table 2: Mesh characteristics.

Mesh	**1**	**2**	**3**	**4**
Elements	477,793	253,328	918,338	426,224
Tetrahedral	409,826	226,697	397,425	225,932
Thickness prism layer (m)	N/A	N/A	0.292	0.292
Volume tetra (m^3)	0.106	0.17	0.109	0.192
Edge of tetra (m)	0.97	1.13	0.98	1.18

Each one of the meshes served to several calculations at several air velocities (between 0.2 and 8 m/s, ranges legally established in Spain by the underground works standards [12]) and all the roughness values included in Table 1 in case of walls, but maintaining null the roughness of the ground. This guide to several pair of data (CFD calculation, theoretical calculation as referred in point 2 of this paper) that can be compared in graphics as the ones in Figure 5 and 6. A regression line is fitted to each group of points. Following table 3 includes the regression line for each mesh (y=vertical coordinate=CFD calculation; x=horizontal line=Theoretical calculation).

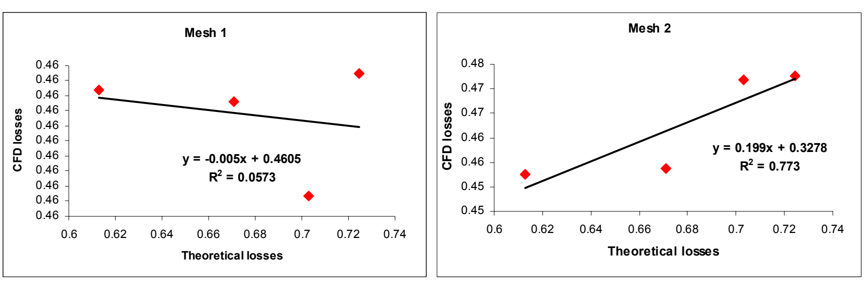

Figure 5: Comparisons 1 and 2.

The adjustment in case of mesh 1 is quite bad, with no correlation among several velocities and roughness. In case of mesh 2 results are better with correlation factors over 0.7 but with a line quite far from the ideal y=x fit.

Once prisms are included the results gets much better, with optimal regression factors over 0.99 in both mesh 3 and mesh 4. Seems clear that in this kind of simulations the use of prisms is a must. Mesh 3 and mesh 4 gives similar regression lines in $y=0.325 \cdot x$. This is, for any air velocity or wall roughness the losses obtained using CFD are the 32.5% of the losses obtained through the Darcy equation. This variation should come from the fact that the Darcy equation considers only one friction factor for both the walls and the ground of the tunnel, while CFD can take different friction factors and only walls friction factor has been considered.

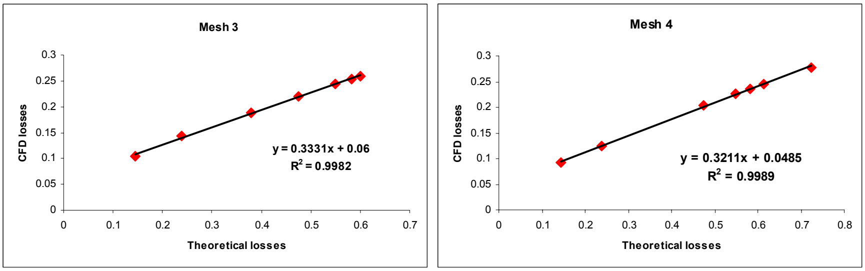

Figure 6: Comparisons 3 and 4.

Table 3: Error adjustment lines and regression factors.

Mesh	Regression equation	R^2
1	-0.005x+0.4605	0.0573
2	0.199x+0.3278	0.773
3	0.3331x+0.06	0.9982
4	0.3211x+0.0048	0.9989

If now there is considered the ground roughness, and only working with mesh 3, there will be obtained different adjustments as are shown in table 4. The regression line gets proximal to y=x, always with very good regression factors.

As the ground roughness grows so does the slope of the adjustment line. This can be explained through the fact that the head losses must grow as does the friction factor.

Table 4: Adjustment for different ground roughness.

Ground roughness	Adjustment line	R^2
Null	0.3211x+0.0048	0.9989
10 mm	0.5039x+0.0386	0.9993
Same as wall	0.6555x+0.0041	0.9991

5 Conclusions

Alter the calculations referred above first conclusion is that the calculation of pressure drops in tunnels requires tetrahedral meshes that include 12 tetras in the distance between the floor and the ceiling of the tunnel (what means in a 67 m^2 section tetras of 1.18 m of edge). The use of prisms in the neighbourhood of the walls is a must.

Using this two modelling advices there is obtained independency of the CFD results to the mesh used, which is mandatory when dealing with this sort of calculations. Hybrid meshes (by "hybrid" we mean concurrent use of tetras and prisms) of at least 400k elements give the same results as the meshes of 900k elements. The CFD calculated values differ from the theoretical calculations in a 68% value when considering no roughness of the ground. But the regression factor of the adjustment line of the comparison between both values (see figures 5 and 6) is almost 1, what lead us to say that calculations are right.

The roughness of the ground will affect calculations, as the CFD pressure drop vs. calculated pressure drop ratio will grow as the ground roughness do. If the same roughness is used in the ground and in the walls (which fits better with the concept of hydraulic diameter) a ratio of 0.65 is obtained. More simulations are being conducted in order to improve and explain this results.

Acknowledgements

We want to acknowledge the support from the Spanish Ministry of Science and Education that granted these researches through the project CTM2005-00187/TECNO, "Prediction models and prevention systems in the particle atmospheric contamination in an industrial environment".

References

[1] J. Toraño et al - A CFD Lagrangian particle model to analyze the dust dispersion problem in quarries blasts. Capítulo del libro «Computational methods in Multiphase Flow IV». 2007. ISBN: 9781-84564-079-8

[2] J. Toraño et al, "Contamination by particulated material in blasts: analysis, application and adaptation of the existent calculation formulas and software". Environmental Health Risk III, pp. 209-219, (2004)Sd

[3] J. Toraño et al,"Computational Fluid dynamic (CFD) use in the simulation of the death end ventilation in tunnel and galleries. Advances in Fluid Mechanics VI, pp 113-122.

[4] K.W. Moloney, I.S. Lowndes and G.K. Hargrave, "Analysis of flow patterns in drivages with auxiliary ventilation", Trans. Instn Min. Metall. 108, pp 17-26 (1999).

[5] S.A. Silvester, I.S. Lowndes and S.W. Kingman, "The ventilation of an underground crushing plant", Mining Technology (Trans. Inst. Min. Metall. A), Vol. 113, pp. 201-214 (2004)

[6] Achieving Energy Efficiency for Underground Mine Ventilation Systems. A.J. Basu, D. Datta. Web link. www.miningmiair.com/M%26I%2520-%2520MEMO-Sudbury%25202005.pdf, accessed 01/04/07.

[7] Meyer, CF. Determining the friction factors for underground colliery board and pillar workings. Safety in Mine Research Advisory Committee, Col 465, May, 1998, pp 1-84

[8] Ke, Ming-Tsun et al, Numerical simulation for optimizing the design of subway environmental control system. Building and Environment 37 (2002) 1139 – 1152. Elsevier.

[9] Gao, P.Z. et al. Large eddy simulations for studying tunnel smoke ventilation. Tunnelling and Underground Space Technology 19 (2004) 577–586
[10] Mechanics of Fluids 3rd. Merle C. Potter. David C. Wiggert. Ed. Brooks/Cole. (2001)
[11] Hacar, F. La rugosidad en túneles sin revestimiento en relación con la ventilación. Ingeotúneles Libro 3. Pp 455-491. ETSI Minas Madrid. (2000)
[12] Spanish Standard RGNBSM, ITC 05.0.01