4th International Congress
of Croatian Society of Mechanics
September, 18-20, 2003
Bizovac, Croatia

4th ICCSM

MODELING OF DISPERSION AND DIFFUSION OF POLLUTANTS FROM INDUSTRIAL CHIMNEY STACKS IN RIJEKA REFINERY

Z. Mrša and M. Čavrak

Keywords: *Atmospheric modelling, air pollution, ground concentrations, plume rise*

1. Introduction

Air pollution affects human health, ecosystems and materials in variety of ways, and deserves appropriate attention. The atmosphere can act as a means for transporting local pollution emissions to other locations, even long distances away and to other media.

Environmental fluid mechanics is used to simulate the influence of pollutants to environment. Although it's not yet fully explained and usually every simulation in that field leads to discrepancy from a real physical system, it is necessary to have at least some air pollution dispersion scenarios to assess the influence of the humane born emissions.

Here we present the model and simulate dispersion of pollutants from two chimneystacks situated in Rijeka's refinery and calculate ground-level concentrations of hazardous effluents (p.e. SO2, CO2) on a complex terrain around Bakar's bay. Importance of conveying such a simulation is to ensure that pollutants are well dispersed and to see if they are within the regulatory levels.

Many simplified models (based on Gaussian plume model and Briggs plume rise equations) exist for fast calculation of pollutant concentrations, but they are not accurate enough. Their validity is superimposed just on the flat or linearly growing terrain with smooth roughness. Those limitations are the main reason that we adopted full 3D turbulence model for the better simulations on a complex terrain.

2. Problem definition

In order to perform the simulation we needed to define geometry for a domain of consideration, mesh it and finally define boundary conditions. IDEAS CAD software was used to make geometry based on terrain topology (Figure 1.) and to generate unstructured tetrahedral mesh. Geometry was made as a 'box model' (in atmospheric modelling one of the most used one [3]) which bottom side was replaced with terrain-shaped surface and it represent a control volume for the finite volume method, a numerical method we used.

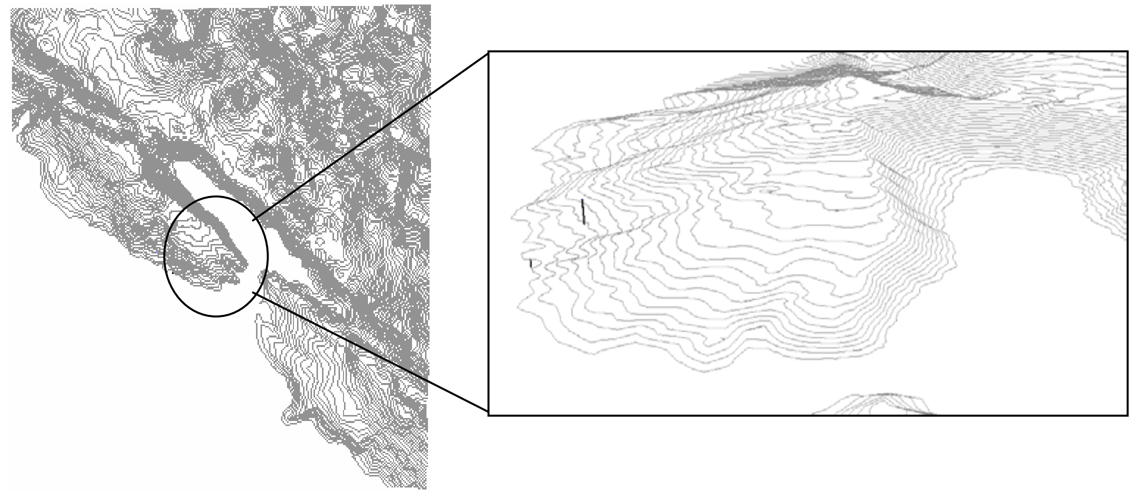

Figure 1. Topology and the location of chimneys

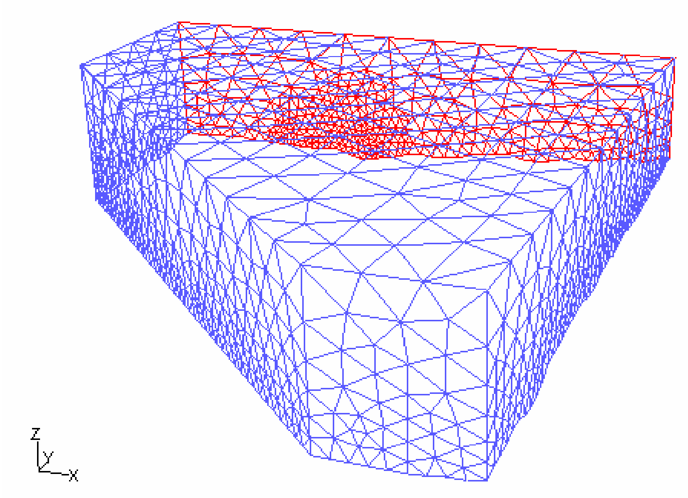

Figure 2. Box model

2.1 Mesh generation

Unstructured tetrahedral mesh that consists of 250000 elements was generated. Distribution and size of triangular elements on surfaces and tetrahedral elements inside the volume depended upon the range of terrain's and chimneystacks' dimensions. Those dimensions ranges from 3 meters on the top of a chimney up to 12 km long boundaries (Figure 3., Figure 4.). From the Figure 4. it is evident that on one hand main purpose of making such a course mesh away from the chimneys was to concentrate out attention on the effects of pollution and the assessment of ground concentrations downwind from the chimneys, while on the other hand it was the mesh generator that constraint us in obtaining more finer mesh. Nevertheless, the results have shown that we never needed finer mesh because of reliability of results that solver gave us and the phenomena it caught. It is to be mentioned that it is accustomed not to make the finer mesh on the beginning of simulation but to adapt the mesh as soon as the first results from the courser mesh are calculated.

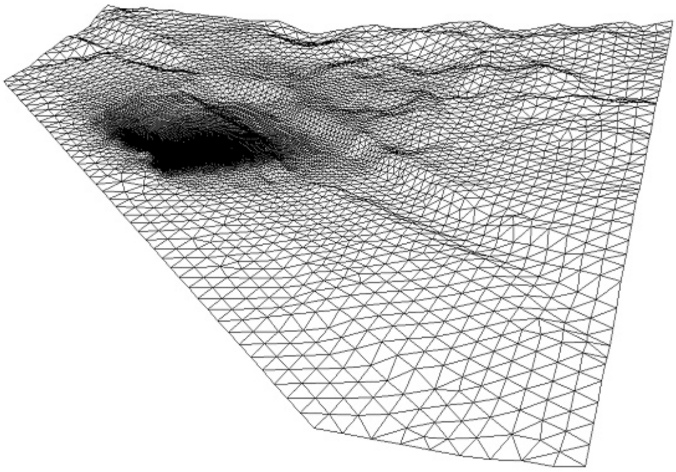

Figure 3 Meshed terrain

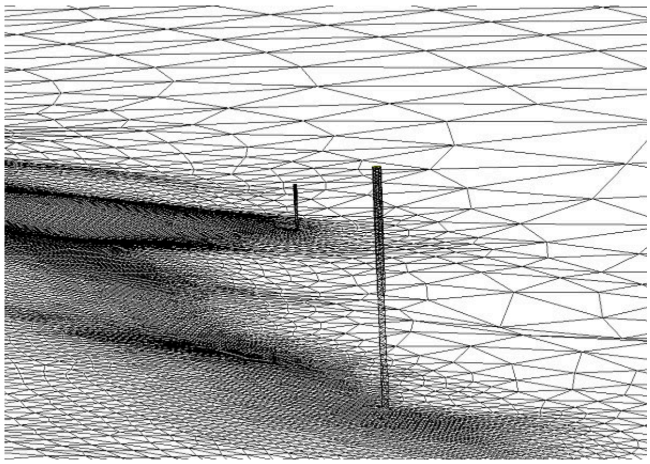

Figure 4 Meshed chimneys

2.2 Boundary conditions

In all commercial CFD software the boundary conditions are given on geometry surfaces that bound the domain. We used FLUENT as a solver and postprocessor. In order for FLUENT to understand the problem we are dealing with, boundary conditions had to be given. Five different south wind profiles each with reference values of 2.5, 5 and 10 ms^{-1} were used as air velocity inlets (Figure 5. and 6.). It is to be mentioned that no time varying profiles were used but a steady-state meteorological condition was chosen from regulatory model's (ISC3 and SCREEN3D) dispersion algorithms provided by EPA (Environmental Protection Agency). Species mass fractions needed for effluents' velocity inlets were determined by calculating combustion products in vessels and furnaces connected to chimneys and using already measured species' emissions [1,2]. Because fluid properties can't be determined for outlets, an 'outflow boundary condition' was used, approximating zero downwind gradients of all unknowns. Terrain and stacks were defined as wall boundary condition with relative roughness of 0.5 for rural terrain. K-ε turbulence model was used to predict turbulent behaviour in flow field.

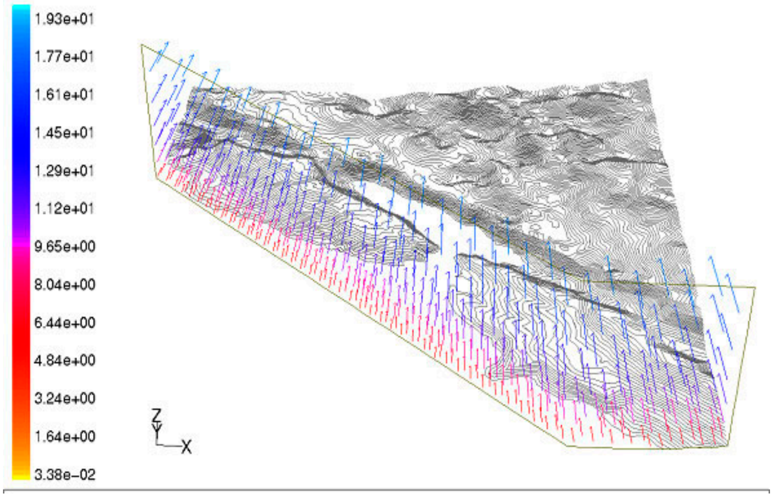

Figure 5 South wind profile as velocity inlet

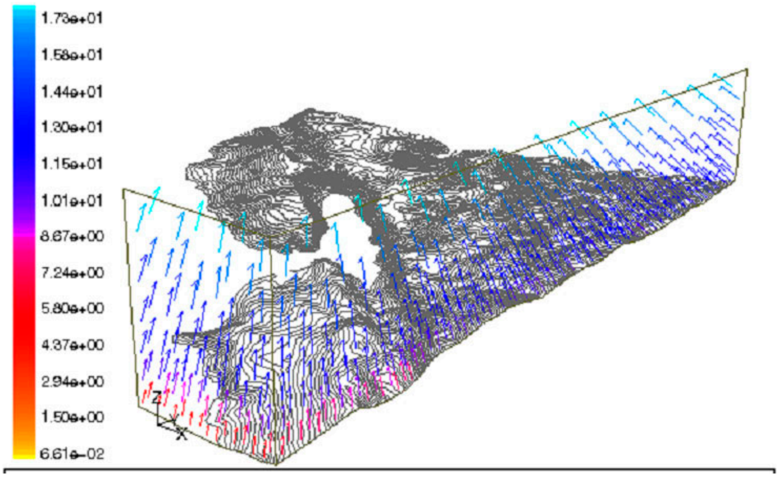

Figure 6 Northwest wind profile as velocity inlet

3. Results and discussion

Whenever dealing with air pollution we have to be careful with validating obtained data. Problems rise with terrain complexity and real-time meteorology. If meteorology is missing then it is not possible to evaluate emissions in any way. This work offers just an insight in how pollutants disperse with respect to the topology of terrain. Figure 7 shows ground concentration of SO2 for the south wind. It is seen from the picture that the smoke plume is leaded with the flow field and so curving the plume to the left so that it passes right through the town of Bakar.

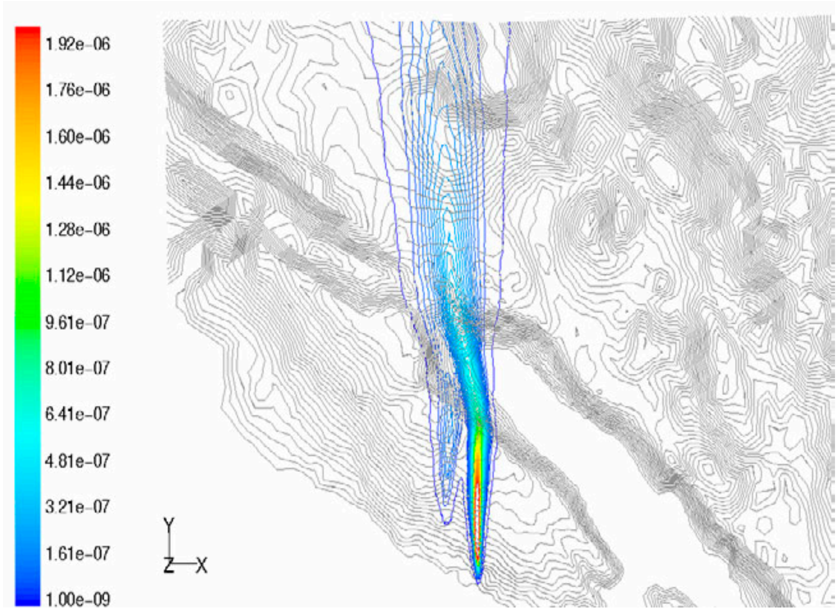

Figure 7. Ground concentrations of SO2 (kg/m^3) for South wind

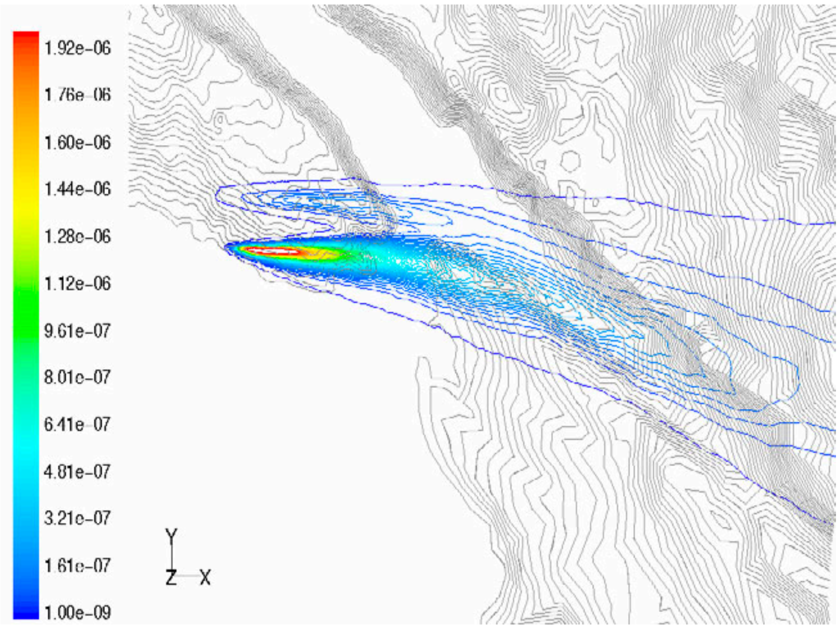

Figure 7. Ground concentrations of SO2 (kg/m^3) for West wind

Figure 7. shows yet another wind direction and dominance of a flow field to a plume advection. The only way how to analyse those results is by locating the maximum ground concentration and to compare them with Clean Air Act regulatory levels. For example, the case from Figure 6. gives us the maximum concentration of SO2 and it is 3060 μg/m^3. The regulatory levels prescribe 350 μg/m^3 of SO2. First of all those maximums are located inside the walls of refinery not influencing the urban area of Bakar bay. Moreover, where the chimneystacks are located the terrain goes up in the direction of the wind and so the plume hits the ground sooner then in other simulations like in Figure 7. where it has enough space to disperse before hitting the ground. The other pollutants are in reasonable amounts (far lesser) then the regulatory levels for all the simulation made. The only difference in ground concentrations is if we compare different wind speeds. It means that for different ratios of jet and crosswind momentum fluxes the plumes differ and so it is with the ground

concentrations. If the plume is higher then jet momentum flux is dominating and the ground concentration are lower and vice verse.

Figure 8. Counter-rotating vortex pair

The other important issue we have covered is the phenomenon we caught. It is the existence of a wake downwind of the stack that has a popular nickname "horseshoes" (Figure 8). This counter-rotating vortex pair is product of shear effects of bent jet and it contribute for the enlargement of entrainment area for effluents to disperse in crosswind flow [5,6]. To be sure that we covered the effects on the smaller areas, such as this one, we made an comparison with experimental data [4]. Data that are available from that article were supplied on the flat terrain and with LDA (Laser Doppler Anemometer). We took the data from the same location and it is 1.6 diameter downwind from the chimneys, close enough to overpass the influence of slightly elevating terrain, and on the three height levels from the stack (z/d = -2, 0, +2). On Figure 9 we present the x velocity component (u) and z velocity components (w), respectively.

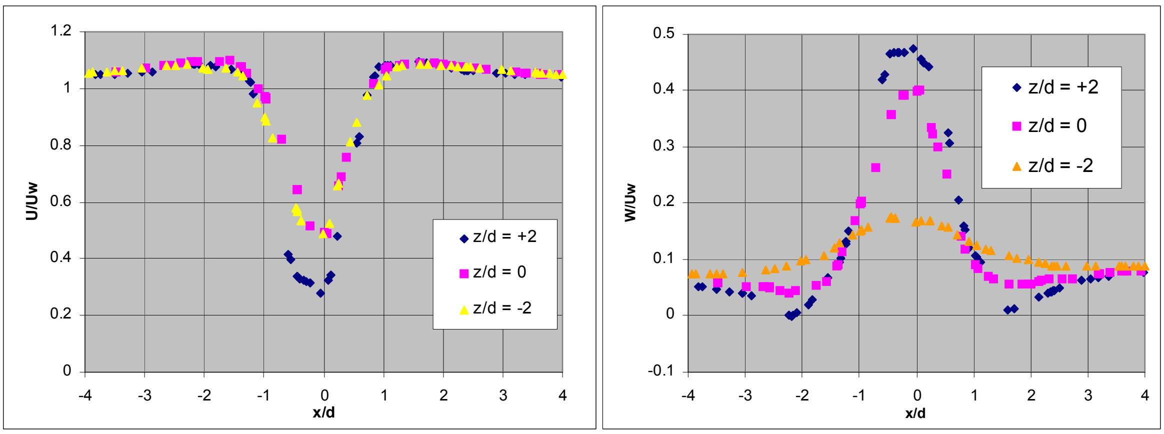

Figure 9. x- and z-component velocities

Those two diagrams shows the natural behaviour of the rising plume and almost perfectly coincide with the experimental data [4]. As the height levels rise, vertical component (W) grows while the horizontal, downwind component (U) decays proportionally. It is so until the crosswind fully entrains the jet giving the dominations to the horizontal velocity.

4. Conclusion

Despite the lack of true meteorological boundary and initial conditions as it is accustomed in the field of atmospheric modelling, we obtained some useful and significant results. Flow structures and vorticity in the near wake region of the elevated round jet in the crosswind was observed. This vortex pair decay in downwind direction quite fast (5-10 chimney diameters) regarding the speed of the wind. Usually if terrain is flat and the wind has the uniform profile those vortices could extend their existence up to 50 diameters downwind as lot of publications suggest. Furthermore the domination of crosswind momentum flux encourage the vortices to decay faster and to change into the crosswind behaviour.

Ground concentrations of emissions from the chimneys have been investigated. The contour plots from figures shown before offer the insight on how flow field determines the path in which concentrations will be deployed on the ground. We haven't performed simulations for mainly jet dominated flows but we can guess that the plume will rise more and that concentrations measured on the ground need to be much lower.

5. Recommendations and future research needs

In order to obtain real initial and boundary conditions for given domain prior to defining emissions of effluent it is necessary to perform some kind of meteorological pre-processor that will based on real measured values (wind velocity, pressure, temperature, humidity, etc.) make a scenario of a flow field for a given number of days (at most three days). The best and the most used one is MM5 (Mesoscale Meteorological Model version no. 5) of UCAR U.S. It also assume better resolution of terrain vegetation or land use for a better determination of roughness coefficients. For such an analyses Linux cluster is very much needed. So this is our next step.

References

[1] ANT – Laboratorij za analitiku i toksikologiju d.o.o., Zapisnik br. 99016-1E o mjerenjima i analizama emisija onečišćujućih tvari u poduzeću INA-Industrija nafte d.d. Rafinerija nafte Rijeka, Kostrena – pogon energane, studeni 1998. god.

[2] ANT – Laboratorij za analitiku i toksikologiju d.o.o., Zapisnik br. 99016-3E o mjerenjima i analizama emisija onečišćujućih tvari u poduzeću INA-Industrija nafte d.d. Rafinerija nafte Rijeka, Kostrena – FCC POSTROJENJE, prosinac 1998. god.

[3] Westbrook, James A., Air Dispersion Models: Tools to Assess Impacts from Pollution Sources, American Bar Association, Natural Resources & Environment, Vol 13, No. 4, 1999. god.

[4] Huang, R.F., Hsieh, R.H., "An experimental study of elevated round jets deflected in a crosswind", Experimental Thermal and Fluid Science, No. 27., 2002, pp 77-86

[5] Zhang, X., Ghoinem, A. F., A computer model for the rise and dispersion of winf-blown, buoyancy-driven plumes – II. Linearly stratified atmosphere, United states Departement of Commerce, Technology Administration, National Institute of Standards and Technology, NIST-GCR-93-637, 1993.

[6] Zhang, Xiaoming & Ghoinem, Ahmed F., A computer model for the rise and dispersion of winf-blown, buoyancy-driven plumes – I. Neutrally stratified atmosphere, Pegamon Press Ltd., *Atmospheric Environment*, Vol. 27A, No. 15, str. 2295-2311, 1993.

Prof. dr. sc. Zoran Mrša, dipl. Ing.
University of Rijeka, Technical faculty, Department for fluid mechanics and computational engineering, Vukovarska 58, Rijeka, Croatia, +385 (0)51 651-500, +385 (0)51 651-290, mrsa@riteh.hr
Marko Čavrak, dipl. ing.
University of Rijeka, Technical faculty, Department for fluid mechanics and computational engineering, Vukovarska 58, Rijeka, Croatia, +385 (0)51 651-554, +385 (0)51 651-290, mcavrak@riteh.hr