Geophysical Research Abstracts
Vol. 14, EGU2012-1507, 2012
EGU General Assembly 2012

Modeling and simulation of a liquid jet in view of CO_2 atmospheric dispersion

N. Herzog and C. Egbers
Germany (Nicoleta.Herzog@tu-cottbus.de)

epsconf ae

Department of Aerodynamics and Fluid Mechanics,
Brandenburg University of Technology Cottbus, Germany

Modeling and simulation of a liquid jet in view of CO_2 atmospheric dispersion

Dr. rer. nat. Nicoleta Herzog, Prof. Dr.-Ing. Christoph Egbers

January 6, 2012

In the frame of the CCS-technology voluminous CO_2 emissions from large power plants to geological storage sites is transported through pipelines. Potential risk during the transportation process are leakages or even breaches of the pipeline, where CO_2 can escape into the air. Failures can be caused by corrosion or externally construction works. Compared with air CO_2 has a higher density, thus the formed clouds will accumulate in topographic lows presenting a potential risk for humans and animals. When CO_2 is stored underground several potential leakage pathways exist as injection well failures or ground movement. At the moment, several uncertainties and knowledge gaps exist with regard to the dispersion behavior and the modeling of CO_2 released into the atmosphere, CO_2 threshold values and effects of CO_2 leakages at different distances from the pipeline, [**?**].

In the scientific community, work is mainly focused on the means of leak detection and on measuring the total amount of leaked gas or liquid. As it leaks from a high pressure facility, a gas or supercritical fluid can involve a free jet release with a velocity of up to the speed of sound which will entrain many times its own volume of ambient air. In the study [**?**] attention was raised on the calculation of the release speed for high-pressure pipelines/transportation facilities, as a mean of early dispersion of the gas, particularly considering the case of CO_2 within CCS projects. A CFD model was used in [**?**] to simulate a high-speed CO_2 release with specified velocities with the aim of evaluating the effect of initial gas dispersion on the downwind length reached by toxic concentrations of the pollutant. But the used CFD model for CO_2 dispersion considers a released without momentum (either plume or instantaneous release). The models described in [**?**] give a good basis for the modeling of outflow and dispersion of CO_2 released from a high pressure pipeline. However, additional modeling of the CO_2 jet direction and momentum and solid CO_2 formation, sublimation and dispersion will be necessary. An important contribution to the uncertainty is the influence of the jet direction (horizontal, upwards, downwards) and momentum (free jet or impinging jet) on the harmful CO_2 concentration distance.

In the numerical treatment of this task, the first step of our study was to benchmark the turbulence characteristics of a liquid jet injected through a circular nozzle hole into a stagnant gas, based on the results of [**?**]. Simulation results of the jetting two-phase flow will be presented in a large range of the jet release speed.

References

[1] J. Koornneef, M. Spruijt, M. Molag, A. Ramirez, A. Faaij and W. Turkenburg: *Uncertainties in risk assessment of CO_2 pipelines*, Energy Procedia, 1, 1587-1594, (2009)

[2] A. Mazzoldi, T. Hill, J. Colls: *A Consideration of the jet-mixing effect when modelling CO_2 emissions from high pressure CO_2 transportation facilities*, Energy Procedia, 1, 1571-1578, (2009)

[3] A. Mazzoldi, T. Hill, J. Colls: *Assessing the risk for CO_2 transportation within CCS projects, CFD modelling*, International Journal of Greenhouse Gas Control, 5, 816-825, (2011)

[4] M. Molag, C. Dam: *Modelling of accidental releases from a high pressure CO_2 pipelines*, Energy Procedia, 4, 2301-2307, (2011)

[5] A. Mansour and N. Chigier: *Turbulence characteristics in cylindrical liquid jets*, Phys. Fluids, 6(10), (1994)