CFD ANALYSIS FOR RISK ANALYSIS IN URBAN ENVIRONMENTS – TILBURG CITY CASE STUDY

Corina Hulsbosch-Dam, Andreas Mack, Sjoerd van Ratingen, Nils Rosmuller, Inge Trijssenaar

Urban Environment and Safety, TNO, Utrecht, the Netherlands

Abstract:
For risk analysis studies, relatively simple dispersion models are generally applied, such as Gaussian dispersion and dense gas dispersion models. For rail transport risk analyses in the Netherlands, fixed consequence distances are applied for various standard scenarios of hazardous materials releases. The advantage is that the results are uniform and are relatively independent of the party performing the calculations, which facilitates the decision making. The drawback of this method, however, is that the results are mostly (very) conservative and the implementation of safety measures cannot be taken into account.

The use of CFD enables to account for local topography of the buildings in a city and additionally enables the investigation of safety measures. This paper describes the use of CFD for a hazardous material release in Tilburg city. The first scenario follows from the standard scenarios for rail transport in the Netherlands as given in guidelines for risk analysis as well as the RBM-II software. The paper describes how the CFD model is set-up for an urban environment and the results are compared to the standard risk analysis scenario. The advantages and disadvantages of both CFD and standard risk analysis methods are discussed. A second scenario is a dense gas release in the same area, which is not part of the standard scenarios. The CFD results for this scenario are compared to wind tunnel experiments.

Key words: *Computational Fluid Dynamics, risk analysis, urban environment, wind tunnel, ammonia*

INTRODUCTION

In the Netherlands standard ways of calculating risks is prescribed for transportation by rail, road and water. The calculated risks are Societal Risk (*groepsrisico*, GR) and Locational Risk (*plaatsgebonden risico*, PR). The former indicates the frequency of a number of lethalities for a certain railway section and gives a guide value. The latter calculates the probability of lethality of a single individual at a certain location as a consequence of the transport of dangerous goods. The used risks are based on standard calculations, with set effect distances.

This method is a (very) conservative way of working. The effects of buildings can not be taken into account. Also, introducing measures does not influence the result of the calculated risks. For these situations CFD calculations that give 3D information could be very useful. In the presented study the societal risks calculated by the standard method RBM-II (*RisicoBerekeningsMethodiek* = Risk Calculation Method) are compared to the societal risks calculated by CFD. The location for this comparison is the area around the train station in the city of Tilburg.

COMPARISON RBM-II AND CFD

Introduction to RBM-II and scenario selection

In the RBM-II method all transports of dangerous goods through a certain spot are included. The goods are classed into several categories and for every category a representative substance is chosen. For the representative substances possible accidental release scenarios are defined and for each of these scenarios the effect distances are given. Further information that is needed to obtain a total risk is the amount of goods transported, population density, meteorological data, and failure probabilities.

As rail tracks frequently pass through or close to the densely populated city centres high societal risks are expected at these locations. Indeed, in several locations of the basic network for rail transport (*Basisnet Spoor*) the guide value for the societal risk is exceeded. One of these locations is in the city of Tilburg. The guide value is exceeded by a factor of 4.3. [Minister IenM, 2011] For this reason the location of Tilburg is chosen.

One of the substances that is transported along this track is ammonia. For this substance several release scenarios are defined in RBM-II; the (semi)continuous release is taken for the comparison. The numbers defining this release scenario are shown in Table 1. The results from RBM-II for this scenario are the dimensions of clouds with a certain level of lethalities. These numbers are reported in Table 2.

Table 1 Source definition in RBM-II

Storage		
Volume	89	m^3
Mass	50 000	kg
Pressure	616 257	N/m^2
Temperature	282	K
Release		
Diameter	0.075	m
Duration	667	s
Mass flow	75.01	kg/s
Rain out fraction	0.6859	-
Source strength	23.56	kg/s
Vapour mass fraction	0.4364	-

CFD case set-up

The numbers from Table 1 are also used as input for the CFD calculations. The calculations were performed with Ansys-Fluent 14.0. The release is in southward direction, with a northern wind in stability class D5. The release is a two-phase release with 75μm diameter droplets.

The standard kε-model is used for turbulence modelling. The droplets are described by the Discrete Phase Model (DPM).

The resulting ammonia cloud calculated with CFD is shown in Figure 1. Figure 1(a) and (c) show the concentration contours, the lowest value plotted is the 1% lethality contour (2.30 10^{-3} mass fraction ammonia). The lowest concentration in the right hand side of the figure ((b) and (d)) is 1 10^{-7} mass fraction ammonia. From these contour plots the effect of the buildings is clearly visible. In Figure 1(a) also the 1% lethality contour as it is calculated with RBM-II is indicated in red.

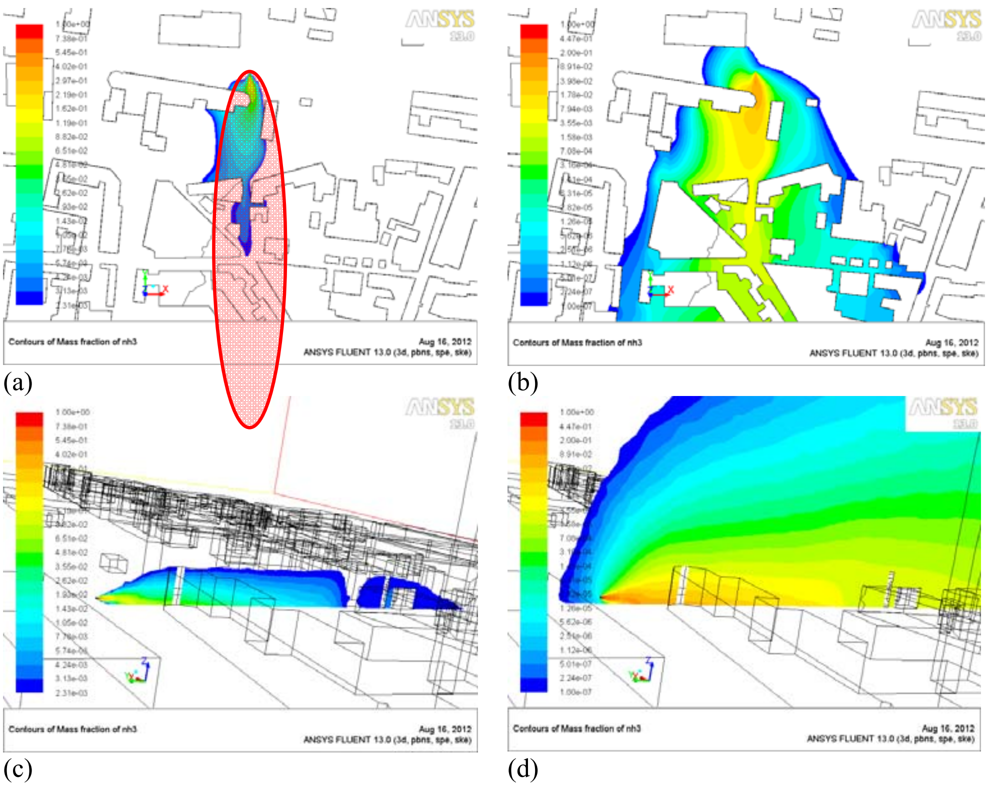


Figure 1 Ammonia concentration contours calculated with CFD. The red oval shape in (a) shows the clouds as it is calculated with RMB-II. (a) and (b) show top views with different minimum concentrations, (c) and (d) show a cross section through the jet symmetry plane.

Consequences for risk analysis

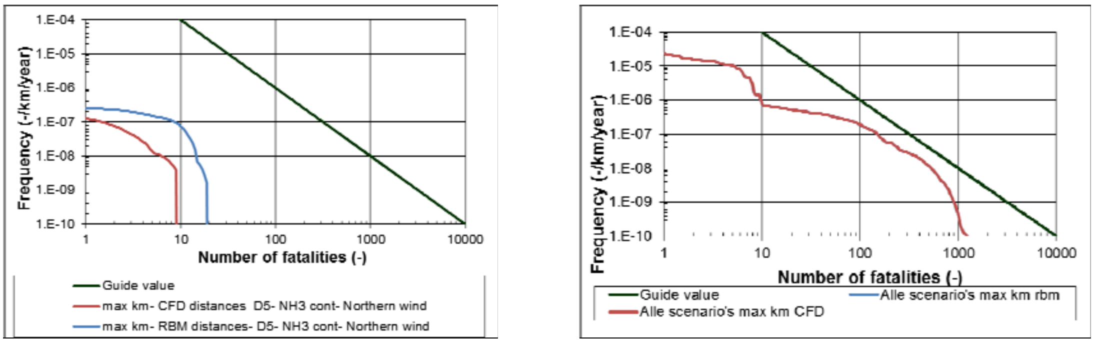
Figure 1(a) already shows the extends of the clouds calculated by CFD and RBM-II. The numbers describing the cloud are given in Table 2. Clearly the cloud obtained with CFD is smaller than the cloud obtained with RBM-II. This has consequences for the calculated societal risk. Figure 2(a) shows the fN-curve for this single scenario. An fN-curve shows the probability or frequency that at the chosen location a number of casualties will fall due to the (planned) transport of dangerous goods. The blue (RBM-II) curve is higher than the red (CFD) curve and both curves for this single scenario are far below the guide value (black line). For the total fN-curve all release scenarios for all stability classes, wind direction and substances classes are summed. The result is the red line in Figure 2(b). This value is closer to the guide value. Also the total fN-curve is higher than the curve for the single scenario of semi-continuous ammonia release. It shows that the chosen scenario has only a (small) contribution to the total risk at the chosen location.

Another point of attention is that the calculated total fN-curve does not exceed the guide value, which is in contrast with what is reported in [brief minister2011]. This can be caused by differences in used population and a different method of adding the different scenarios.

For the total risk the replacement of 1 single scenario has no big influence. However for detailed information on the chosen scenario CFD is very useful. To really see an influence of CFD calculations on the total risk, more scenarios should be calculated with CFD.

Table 2 Overview of length, width and off-set of ammonia cloud for RBM-II and CFD. As a standard these values are taken at 1m height.

	RBM-II			CFD		
Lethality (%)	Length (m)	Width (m)	Off-set (m)	Length (m)	Width (m)	Off-set (m)
1	453	99	0	204	73	2
10	340	75	0	165	62	2
25	281	62	0	139	55	2
50	211	45	0	111	47	2
75	174	37	0	92	38	2
90	135	28	0	78	29	2
99	75	16	0	58	21	2



(a) (b)

Figure 2 (a) fN-curve for (semi) continuous ammonia release near Tilburg station; (b) total fN curve for all release scenarios.

DENSE GAS IN CFD AND WIND TUNNEL

Scenario description

For the same location in Tilburg a second study is performed. A dense gas (CO_2) is released in the wind tunnel with a low velocity and turbulence background wind field. This scenario is not part of the standard scenarios in RBM-II. Here it serves as a validation case to compare CFD results to wind tunnel measurements.

For this dense gas dispersion case, the reference wind velocity at 10 cm height is 0.3 m/s. The boundary layer profile in the wind tunnel for the present case can be described by:

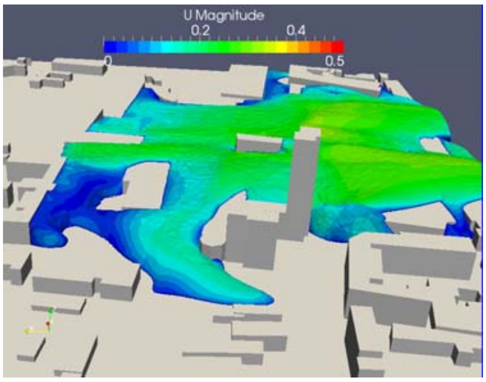
$$u(z) = 0.3(z/0.1)^{0.26} \quad (1)$$

With $u*$ chosen at $u*=0.258$ and a roughness height of 0.0017m.

The continuous release of CO_2 was fixed at the measured value of 485 l/s at ambient temperature. The release mechanism is a 70 mm diameter porous surface. The computation was performed in wind tunnel scale with openFoam applying the kε-turbulence closure modeling buoyancy effects. The computational grid consists of $1.2 \cdot 10^6$ points.

Comparison CFD and wind tunnel


(a)



(b)

Figure 3 Qualitative comparison between wind tunnel experiment (a) and CFD calculation (b). The wind direction is from left to right for both figures.

The flow topology in the wind tunnel and the CFD solution are shown in Figure 4. As can be seen clearly, the global dispersion pattern is comparable. The dense gas is travelling upstream but then is locked at the leeside of the buildings.

In Figure 4 the molar fractions of CO_2 are plotted on the surface. Once again the upwind and lateral dispersion and the effects of the buildings to lock the dense gas can be seen clearly. A comparison between the experimental and numerical data at the 16 measurement locations (located at ground level) is given in Figure 5.

The lateral dispersion is in good agreement between the wind tunnel and numerical data. Close to the release, the numerical solution indicates about 20% higher values than the experiments. This can be explained due to the fact the release in the numerical solution is forced to steady state but clear unsteady behavior is observed in the experiments.

Further downstream, the maximum calculated concentrations are higher (maximum factor 2-4) compared with the experiments. The sensor positions 8 and 9 are located in areas with strong concentration gradients, due to this the measured data is very sensitive. At sensor locations 14 and 15, the numerical solution also shows higher values than the experiments. The measurement times are only 2 minuts, which might be too short to reach steady state behaviour at these distances from the source.

Nevertheless, the flow topology is in very good agreement with the experimental data and also the gradients in the molar concentrations due to the interaction of the dense gass with the built environment, are reproduced well.

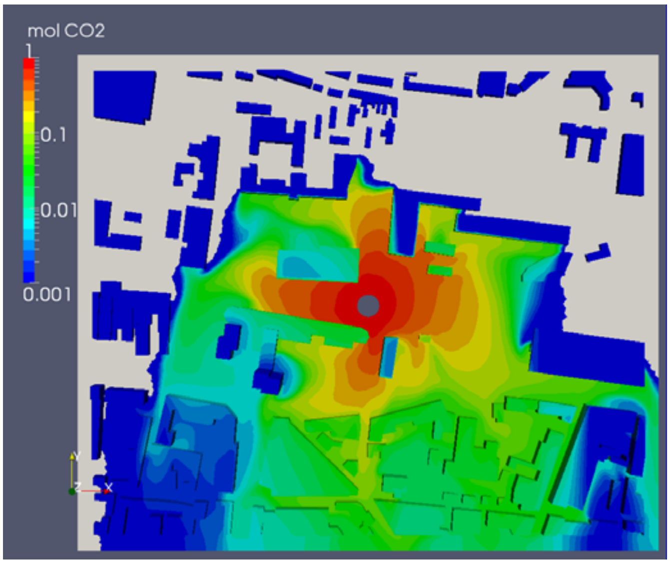


Figure 4 Dense gas CFD CO_2 molar fraction iso-contours at ground. (The wind direction is from top to bottom.)

Figure 5 CO_2 molar fraction (%) for the wind tunnel experiment (WT) and the CFD calculations. The blue circle indicates the release.

CONCLUSIONS

When comparing the effect contours of CFD and RBM-II for a single release scenario different effect distances are found. CFD is able to take the effect of buildings into account, reducing the effect distances significantly. The resulting differences are interesting for land-use planning and also to calculate the effect of measures taken to minimize the exposure of the public.

The final part of the paper shows a good agreement between a CFD calculation and a wind tunnel experiment for a dense gas release in the built environment. This gives confidence in using CFD as a tool for the situations mentioned in the paragraph above, including the effect of the built environment and possible safety measures accurately.

REFERENCES

Minister IenM, 2011: www.rijksoverheid.nl/documenten-en-publicaties/brieven/2011/08/15/basisnet-spoor-bijlage-4b-groepsrisico-s-zuid-nederland-basisnet-spoor-2011.html