CHEMICAL ENGINEERING TRANSACTIONS

VOL. 31, 2013

Guest Editors: Eddy De Rademaeker, Bruno Fabiano, Simberto Senni Buratti
Copyright © 2013, AIDIC Servizi S.r.l.,
ISBN 978-88-95608-22-8; **ISSN** 1974-9791

A publication of
AIDIC
The Italian Association
of Chemical Engineering
Online at: www.aidic.it/cet

DOI: 10.3303/CET1331035

Onshore Explosion Studies –Benefits of CFD-Modelling

Olav Roald Hansen[a], Scott Davis[b], Filippo Gavelli[b] and Prankul Middha[c]

[a]GL Noble Denton, Sandslihaugen 10, NO-5254 Bergen, Norway
[b]GexCon US, 4833 Rugby Avenue, Suite 100, Bethesda MD, 20814
[c]GexCon AS, Box 6015 Postterminalen, NO-5892 Bergen, Norway
Olav.Roald.Hansen@nobledenton.com

Integral or phenomenological consequence models are extensively used for explosion and dispersion studies at onshore petrochemical facilities. These models will generally ignore the influence of the geometry of the facility on the ventilation and flow patterns, the generation of flammable gas clouds, and any subsequent explosions. Another significant weakness of these models is the inability to handle dense vapour cloud dispersion in low wind conditions. Risk and consequence studies performed according to API-RP 752 or Seveso-II are mostly referred to as worst-case assessments. In reality these represent some kind of a probabilistic assessment as only “maximum credible” release scenarios are considered. Typically non-conservative gas cloud sizes are predicted, deflagration-to-detonation transitions (DDT) potential is ignored, and the ability to predict the effect of mitigation is limited when using such tools. Computational Fluid Dynamics (CFD), on the other hand, can incorporate the 3D geometry of the facility and how it influences dispersion and explosion processes. In the decades after the Piper-Alpha explosion and fire (1988), the need for CFD-based consequence models was recognized for offshore oil and gas facilities, and standards like ISO 13702 (1999) and ISO19901:3 (2010) were introduced. This approach has significantly reduced explosion risk on offshore platforms today. In contrast, authorities seldom require CFD-based consequence studies for onshore plants, but instead accept tools that ignore geometry and important physics, and are unable to provide guidance on optimal mitigation methods. Numerous major explosion accidents on onshore facilities in recent years indicate that the explosion risk has not improved over the past couple decades. There are reasons to believe that the conservative attitude against the use of improved consequence modelling (CFD) is part of the explanation to the poor safety performance in recent years. This article will discuss the situation within industrial process safety and consequence calculations. The potential and benefits of using CFD for explosion studies on onshore facilities will be illustrated with examples.

1. Introduction

Empirical or integral models, used frequently for onshore risk and consequence studies, are unable to account for the geometry of the facility as well as topography and vegetation surrounding a site. The geometry will have a major influence on the ventilation and flow patterns through the facility, which affect the size and shape of a potential flammable gas cloud. For example, clouds may develop upwind of the release location due to jet momentum, wake effects or density of the vapour. The most popular dispersion models, assuming a Gaussian profile downwind, ignore these effects. If the purpose of a flammable gas dispersion study is to predict the hazard distance to lower flammable limit (LFL), integral model predictions will in most cases be higher than or equal to what will be seen in reality, despite the weaknesses mentioned above. If however the goal is to identify risk “drivers” for a given facility (e.g., predict flammable vapour clouds from realistic releases), these models may yield inaccurate results. Idealized gas plumes predicted by Gaussian models may sometimes underestimate the hazardous gas cloud volume within a congested plant by 1-2 orders of magnitude for a given release rate, or incorrectly identify a risk driver because the 3D facility details were not considered during the release. Further, risk and consequence studies according to API-RP 752 (2009) or Seveso-II (2002) are often referred to as worst-case assessments. This may be far from reality for various reasons. According to API-RP-752 only “credible”

Please cite this article as: Hansen O.R., Davis S.G., Gavelli F. and Middha P., 2013, Onshore explosion studies – benefits of cfd-modelling, Chemical Engineering Transactions, 31, 205-210 DOI: 10.3303/CET1331035

release scenarios, with a “reasonable probability of occurrence” shall be considered. There is a wide spread in how this is interpreted, and release rates assessed are generally much smaller than any maximum possible release rate. One typical “credible worst-case” choice may be a release rate based on an assumed hole size of 20% of pipe diameter, i.e. 4% of pipe cross-sectional area. A real worst-case scenario can therefore for many systems have a release rate one order of magnitude higher.

Empirical blast prediction methods will need to assume or estimate explosion source strength. The BST-method, see Pierorazio (2005), estimates flame speeds based on gas reactivity and rough qualitative classification of congestion level (low, medium, high) and confinement of the geometry. For the TNO Multi-energy method, e.g. van den Berg (1985), the user simply assumes explosion strength for a given type of scenario, which may sometimes be prescribed by authorities or agreed within the industry. These correlations and guidelines are based on experiments of limited scales, and may severely underestimate potential explosion strength for large-scale accident scenarios. A widely used assumption in onshore blast studies is that only gas cloud energy inside one congested unit of a plant will contribute to the blast waves, and often only a fraction of this is assumed (using a yield factor, e.g. 20%). From large-scale gas cloud explosion experiments, e.g. MERGE, Mercx (1994), and BFETS, Selby and Burgan (1998), tests, one can conclude that for strong explosions (> 1 barg) one will have to use a yield factor of 100% of the gas inside a unit, to be able to predict far-field blast. One assumption taken when ignoring gas clouds outside one congestion unit is that no deflagration-to-detonation-transition (DDT) will take place, so that the flames will slow down when leaving the congested unit. If a DDT would occur, the flammable gas cloud within and around the facility (potentially orders of magnitude larger) could contribute to the blast wave energy. Accident investigations of Buncefield, BMIIB (2009), Sunrise, Ontario Fire Marshal (2010) and Jaipur accidents, Johnson (2011), all concluded that DDT had occurred. It is worthwhile to notice that the recent API-RP 752 (2009) standard does not discuss the possibility of DDT at all.

Based on the methodology above, blast studies will provide hazard distances to a given pressure level or blast loads to dimension control rooms and other buildings on the site. This hazard distance is typically assumed from the centre of a congested unit of the facility. If the gas cloud extends far outside these congested areas of the facility and the explosion either undergoes DDT or the vapour cloud fills other congested areas like vegetation surrounding the plant, the energy contributing to blast waves may take place much closer to the control rooms, buildings or property lines. If a plant is surrounded by low-lying vegetation it may be particularly exposed, especially during dense gas releases in low winds. The philosophy used for predicting the blast loads may for such cases be wrong and non-conservative.

Finally, one main reason why empirical/integral models should be replaced by more accurate prediction methods is that the integral models are not able to predict the potential benefit of various mitigation measures. The current widespread use of such models may have impeded the development of “inherently” safer designs and risk reducing solutions within the industry. To summarize, the main problems with empirical/integral consequence studies are:

- Ignoring geometry effects can result in the incorrect identification of relevant risk drivers and also lead to significant under prediction of relevant vapour cloud sizes
- Selected “credible” leak rates may be arbitrary and not representative for worst-case
- Blast source strength assumption may be arbitrary and non-conservative for real scale accidents
- Potentially very non-conservative methods to estimate blast energy based on cloud size/congestion
- Potential for deflagration-to-detonation transition (DDT) is usually ignored
- “Epicenter” of blast energy not in centre, but potentially outside unit closer to targets
- Limited ability to evaluate the effects of safer designs or mitigation measures, impeding improvements towards “inherently” safer designs.

2. The use of Computational Fluid Dynamics (CFD) for risk and consequence studies

Over the past three decades, 3D Computational Fluid Dynamics (CFD) models have been developed and are gradually becoming more popular. With CFD, one can potentially model the detailed interaction between physical/chemical processes and geometry. For consequence modelling, there may be a need to consider a large number of scenarios as the results will be direction dependent and may change significantly when scenario parameters are varied, e.g. wind and release parameters, ignition location relative to cloud and geometry. The validity of CFD predictions may vary among models and modellers, see e.g. Venetsanos et al. (2009). As CFD simulations can be computationally expensive, a challenge when doing a risk study will be to perform the necessary simulations and obtain predictions of acceptable validity without spending too much time and resources.

Within explosion risk and consequence modelling the offshore oil and gas industry has been a front-runner in developing and applying advanced CFD methods. The development of the FLACS CFD tool for explosions was initiated in 1980, with support from a consortium of major oil and gas companies. To achieve good efficiency and precision when simulating dispersion and explosions in complex petrochemical facilities, a distributed porosity concept and sub-grid modelling have been implemented, see Hjertager (1985). To ensure good data to validate the models against, there were extensive test programs in parallel to the modelling, with the full-scale experiments carried out as a part of the JIPs in the 1990s at GL Noble Denton Spadeadam test site being the most relevant; see Selby and Burgan (1998).

Since 1995, CFD has been extensively applied for safety studies on offshore oil and gas platforms and FPSOs. Industry standards like NORSOK Z-013 (2001, 2010), ISO 13702 (1999) and ISO 19901:3 (2010) provide a clear message that details matter, and that explosion studies shall be performed in a way so that geometry and scenario effects can be properly taken into account and the effect of mitigation measures predicted. It is also concluded that worst-case design will often have a prohibitive cost and that probabilistic studies with tolerance criteria (e.g. once every 10,000 years) for potential escalation are recommended. In such probabilistic explosion studies, several hundred ventilation, dispersion and explosion CFD-simulations are typically carried out to estimate the risk level and a design accidental load. Some oil and gas companies are applying similar approaches for safety studies on onshore facilities. According to NFPA-59A, there are similar requirements for dispersion modelling for LNG siting studies. While NFPA-59A and US regulations prescribe a deterministic approach to establish hazard distances, it is acknowledged that geometry and scenario details and quality of modelling matter. As validity and performance of consequence models (including CFD) can vary substantially, a model evaluation protocol (MEP) has been established, see Ivings et al.(2008). According to the MEP 33 defined experiments must be simulated and the performance evaluated, see e.g. Hansen (2010a). Thereafter the model performance as well as the model itself shall be properly documented and reviewed before authorities would conclude under what circumstances a model can be applied. In October 2011, the CFD software FLACS was approved by US DOT to be used for LNG siting studies according to NFPA-59A.

3. Should use of CFD be a preferred approach for explosion studies at onshore facilities?

There are some important differences between petrochemical installations offshore and onshore which can explain much of the historical differences in safety study approaches. Many of the offshore installations are far from the shore in deep waters and harsh climates, which make the operations and repairs quite expensive and will increase the risk for workers during major accidents as evacuation and rescue operations can be complicated. The cost of downtime for an offshore installation can be high. To keep people safe, it is essential to prevent that an accident scenario escalates. The elements described above may be important reasons for offshore industry to use CFD for risk and consequence studies.

At an onshore petrochemical facility, the fraction of workers surviving major accidents may be higher and the repair costs typically lower than offshore. With the simplified assumptions on blast potential from a unit seen e.g. in API-RP-752, there may be no need for any advanced CFD-based calculation methods. Regardless, there is a tendency of increasing population density around chemical sites combined with a gradually higher litigation costs due to pollution or 3rd party injuries or fatalities. The Macondo accident and spill (BOEMRE, 2011), even if offshore, demonstrated the potential for extreme litigation costs. One should expect that this would convince risk exposed industries to apply the best possible approaches and methods when doing safety studies. With improved computer power advanced CFD methods are getting better and faster. For more than a decade a typical offshore explosion risk study has included several hundred CFD simulations. There is thus no good reason why this should not be possible for an onshore risk study. To facilitate an extensive use of CFD for onshore studies some challenges that need focus are:

- A reasonably detailed 3D geometry model is required for a CFD study
- The CFD-tool must have an acceptable validity, and user dependency should be low
- There is a need for a proper risk methodology
- Incident and accident statistics must be available
- Mitigation methods should be an integral part of methodology
- Authorities, standards and industry will have to facilitate a change towards better methods

3.1 A detailed 3D geometry model required for CFD

The geometry layout including piping, equipment, tanks, decks, walls and vegetation will be very important for modelling ventilation, dispersion, and in particular explosions. Accurate consequence predictions when evaluating hazard scenarios will therefore depend on a precise 3D model. Many offshore oil and gas projects are developed in the last two decade or two, and good CAD geometry models exist. Onshore petrochemical facilities are typically older and larger, and CAD models may not exist or be very basic.

For the generation of a 3D geometry model for a CFD study, there are several feasible approaches. A representative congestion method (RCM), see Hansen (2010b) and Figure 1, will represent only the major objects (walls, decks, major equipment) in detail, while the remaining objects (piping, structural beams, vessels, smaller equipment) are represented in a simplified way. This model can be made with very limited manual effort. It is not an accurate representation of the geometry, but can be used with reasonable precision for screening studies. Hansen et al. (2010b, 2012) demonstrated that a simplified RCM geometry with the same object surface area as in the real geometry, may give comparable explosion pressures.

If a coarse CAD model or manually created model exists for a facility a methodology called Anticipated Congestion Method (ACM) can be used to add congestion to represent finer piping and equipment that are not modelled in detail. This approach is routinely applied on offshore explosion studies, and dedicated object libraries exist in order to facilitate the process. Another potentially attractive method is to develop a 3D model from laser-scan. The interpretation of laser-scan results into a 3D geometry database is currently too expensive to be widely deployed for CFD-studies onshore. When more cost-efficient, automatic approaches are developed, this method can become feasible for safety studies.

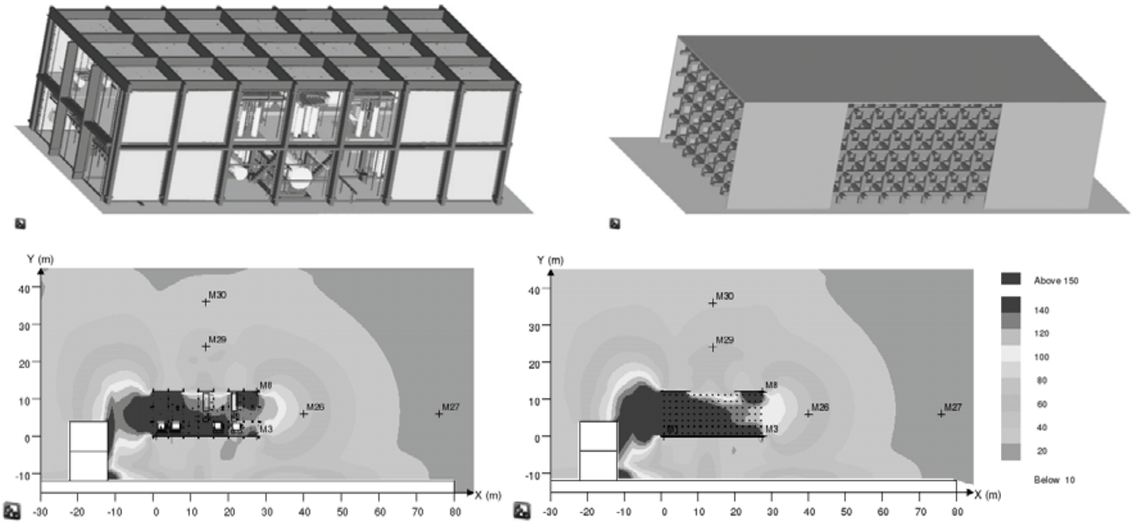

Figure 1: To develop a proper 3D model can be a major challenge for onshore CFD studies. An example of Representative Congestion Method approach is shown above, in which actual geometry (left) is represented by a simplified geometry (right). The use of RCM geometry may give consequence predictions with good accuracy, see comparison of external blast predictions (kPa) in lower pictures.

3.2 Validated models and users

It is well-known that simulation results can vary significantly among different CFD-tools and even different users of the same tool. When a CFD-tool is to be used for prediction of hazards and dimensioning in safety studies, such a scatter may be unacceptable. A possible way to reduce this problem will be to qualify dispersion and explosion models, for instance with a MEP-concept with benchmark experiments and evaluations like defined for LNG. To limit the problem with user dependency for FLACS, GexCon has developed quite detailed validation-based user guidelines and removed the ability of the user to modify important input parameters, such as turbulence models.

3.3 Methodology for onshore risk and consequence studies

For offshore explosion studies a main focus will be to limit the explosion loads so that barriers preventing escalation remain intact, while onshore studies have focussed on limiting the blast at control rooms and buildings. Evidence of DDTs in recent major explosion accidents should change the aim of the studies, and also the onshore industry should aim at design and mitigation to limit the explosion severity to prevent DDT and escalation. If DDT is prevented, separation gaps between units (or water curtains) may be used to control flame speeds into other units. A study could evaluate mitigation measures (fences or water curtains) to prevent flammable gas migration into potential dangerous neighbouring areas or dense vegetation, see Figure 2.

It will be important to establish a transparent and consistent risk methodology. It should be possible for different safety consultants to follow this and obtain comparable results for the same facility. The procedure could be based on the risk of exceeding a certain design accidental load (DAL) or likelihood for DDT, or it could be a deterministic worst-case analysis. The procedure should be flexible with regard to mitigation solutions to establish “inherently” safer designs. A CFD explosion risk or consequence study could include the following elements:

- Establishment of best possible 3D geometry (RCM, CAD+ACM or laser-scan based)
- Ventilation & dispersion study to establish ignited cloud sizes (wind, leak and ignition statistics)
- Explosion simulations to evaluate DDT potential and safety gap performance
- Blast calculations to generate load distribution on targets, structural assessment of some cases
- Evaluation of mitigation measures (change of design, soft barriers, water deluge, ESD/BD)

Statistical data may be lacking or have major uncertainties. If critical, the methodology should be adjusted to reflect this and choices should be made to limit the impact of uncertainties.

3.4 Authorities and industry should jointly embrace and help develop new risk methodology

While most of the above may be implemented for offshore oil and gas platforms, through standards like ISO 13702 and ISO19901:3, the use of CFD is not encouraged much for onshore plants. Regulatory bodies and industry could cooperate to develop a concept as described above. The regulator could require that companies perform more accurate risk and consequence studies which would ensure that the industry would get improved methods for estimating explosion risk and the effect of mitigation options. 2D/integral models could still be used within the new model framework, but in a way such that the output from these should always be conservative (over-predicting the overpressures or gas cloud sizes). This way one could choose to reduce conservatism by applying more detailed and accurate modelling tools.

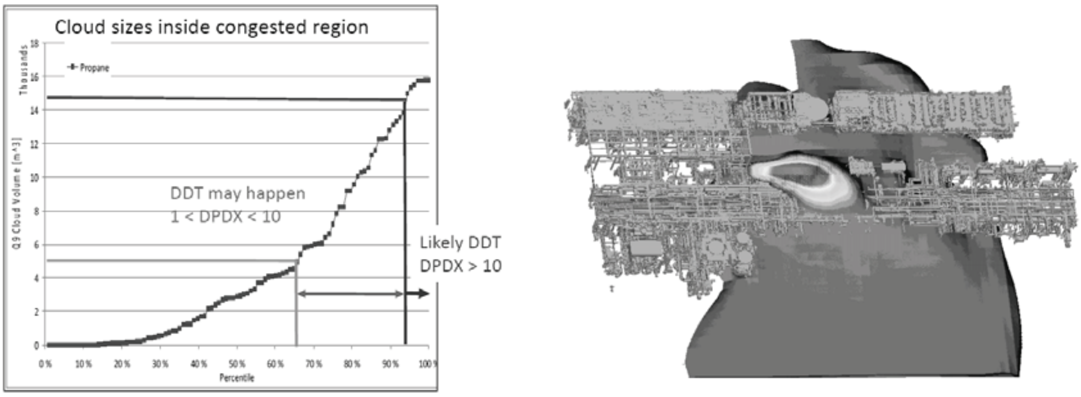

Figure 2: A CFD-based risk and consequence study for an onshore facility could focus on identifying potential (worst-case or probabilistic) for gas cloud build-up in various parts of a facility and evaluate blast consequences from these clouds. Left graph shows example of predicted cloud size distribution among more than 100 severe dispersion scenarios at a facility, with indication of expected DDT potential based on an explosion study. There may often be a greater mitigation potential in efforts reducing gas cloud sizes inside units than to mitigate explosions once gas clouds are generated, and to study these effects in proper detail there is a need for CFD-modelling. Right picture shows an example of a CFD-simulation, in which the dense gas cloud falls to the ground before being dispersed by the wind.

4. Conclusions

There is a need for improved methods and harmonization in the field of onshore safety studies. Methods and consequence tools frequently used have severe limitations and may have a marginal impact preventing major accidents. When tools and methods are unable to describe the phenomena, it is hard to evaluate mitigation and optimise safety measures. This paper describes arguments, gives examples and proposes a methodology to support the increased use of more accurate, CFD-based methods for risk and consequence studies in the onshore industry, similar to what has been done in the oil and gas industry.

References

API-RP 752, 2009: Management of Hazards Associated with Location of Process Plant Permanent Buildings, the American Petroleum Institute

BMIIB, 2009, Buncefield Major Incident Investigation Board (2009), the Buncefield Final Report.

BOEMRE, The Bureau of Ocean Energy Management, Regulation and Enforcement, 2011, Report regarding the causes of the April 20, 2010, Macondo well blowout,

FLACS software, www.gexcon.com, last visited Jan 14, 2013.

Hansen, O. R., Gavelli, F., Ichard, M., & Davis, S. G., 2010a, Validation of FLACS for vapor dispersion from LNG spills: model evaluation protocol. J. of Loss Prev. in the Process Industries, 23(6), 857-877.

Hansen, O.R., Hinze, P., Engel, D. & Davis, S., 2010b, Using computational fluid dynamics (CFD) for blast wave predictions. Journal of Loss Prevention in the Process Industries, 23: 885-906.

Hansen, O.R., Gavelli, F., Davis, S.G. & Middha, P., 2012, Equivalent cloud methods used for explosion risk and consequence studies. In press: J. of Loss Prev. in the Proc. Ind., doi:10.1016/j.jlp.2012.07.006

Hjertager, B. H., 1985, Computer simulation of turbulent reactive gas dynamics. Journal of Modelling Identification and Control, 5, 211-236.

ISO 13702, 1999, Petroleum and natural gas industries - Control and mitigation of fires and explosions on offshore production installations - Requirements and guidelines, the Int. Org. for Standardization

ISO 19901-3, 2010, Petroleum and natural gas industries - Specific requirements for offshore structures - Part 3: Topsides structure, the Int. Org. for Standardization

Ivings, M. J., Jagger, S. F., Lea, C. J., & Webber, D. M., 2007, Evaluating vapour dispersion models for safety analysis of LNG facilities. The Fire Protection Research Foundation.

Johnson D.M., 2011, Characteristics of the Vapour Cloud Explosion Incident at the IOC Terminal in Jaipur, 29th October 2009, GL Noble Denton report 11510, August 2011

Mercx, W. P. M., 1994, Modelling and experimental research into gas explosions. Overall report of the MERGE project, CEC contract: STEP-CT-0111 (SSMA).

NFPA-59A, 2009, Standard for the Production, Storage, and Handling of Liquefied Natural Gas (LNG), NFPA, 1 Batterymarch Park Quincy, MA 02169-7471, USA

NORSOK Z-013, 2010, Risk and emergency preparedness analysis. Norsok standard. Available from Standard Norge, Postboks 242, N-1326 Lysaker, Norway.

Ontario Fire Marshal, 2010, Sunrise Propane explosion investigation report, inv. no: 212-007-2008

Pierorazio, A.J., Thomas, J.K., Baker, Q.A and Ketchum, D.E 2005, An update to the Baker-Strehlow-Tang vapor cloud explosion prediction methodology flame speed table, Process Safety Progress, Volume 24, Issue 1, pages 59-65.

Selby, C., & Burgan, B., 1998, Blast and fire engineering for topside structures, phase 2, final summary report. UK: Steel Construction Institute. Publication number 253.

Seveso-II Directive 96/82/EC on the control of major-accident hazards involving dangerous substances, The European Commission

van den Berg, A.C., 1985, “The Multi-Energy Method: A Framework for Vapour Cloud Explosion Blast Prediction,” Journal of Hazardous Materials, Vol. 12, pp. 1-10.

Venetsanos A.G., Papanikolaou E., Delichatsios M., Garcia J., Hansen O.R., Heitsch M., Huser A., Jahn W., Jordan T., Lacome J-M., Ledin H.S., Makarov D., Middha P., Studer E., Tchouvelev A.V., Teodorczyk A., Verbecke F., Van der Voort M.M., 2009, An Inter-Comparison Exercise On the Capabilities of CFD Models to Predict the Short and Long Term Distribution and Mixing of Hydrogen in a Garage, International Journal of Hydrogen Energy, 34(14), 5912-5923.