CHEMICAL ENGINEERING TRANSACTIONS

VOL. 86, 2021

A publication of AIDIC
The Italian Association of Chemical Engineering
Online at www.cetjournal.it

Guest Editors: Sauro Pierucci, Jiří Jaromír Klemeš

ISBN 978-88-95608-84-6; **ISSN** 2283-9216

DOI: 10.3303/CET2186091

Ground Influence on High-Pressure Methane Jets: Different Concentration Clouds Scenarios

Cristian Colombini[a,b], Marco Pontiggia[b], Giovanni Uguccioni[b], Renato Rota[a], Valentina Busini[a,*]

[a] Politecnico di Milano - Department of Chemistry, Materials and Chemical Engineering "Giulio Natta", Via Mancinelli 7, 20131, Milano, Italy
[b] EOGHR (Oil & Gas HSE & Reliability) - RINA Consulting S.p.A., Via Gran S. Bernardo Palazzo R, 20089 Rozzano, Italy
valentina.busini@polimi.it

Because of their relevant consequences (in particular, associated with domino effect), accidental high-pressure flammable gas releases are one of the major hazards in the industrial safety framework.
It is likely that the accidental loss of containment can involve obstacles that, as a matter of fact, are present in any process facility. As obstacle, flat surfaces (e.g., walls, ground, etc.), equipment (e.g., tanks, pipes, etc.) or structures can be counted.
Focusing on the scenario of an accidental high-pressure unignited methane jet interacting with an obstacle, this work investigates how the proximity to the ground influences the jet cloud extent when considering different concentrations of methane in air. Varying the height above the ground of the source term, the effect of the ground was systematically studied through an extensive Computational Fluid Dynamics analysis. Thanks to the sensitivity analysis performed, the main achievement is the demonstration that methane releases observed at different concentrations in air, from sources at different pressures and outflowing from accidental holes of different sizes are similarly influenced by the ground presence.
The conclusion of the present work is that, the assessment of the hazardous area extent of the flammable release at any concentration of interest can be evaluated exploit an analytical model specifically derived, providing a useful alternative of practical precision to more expensive CFD computations.
This way, for this specific accidental scenario, delineating the area involved within the flammability limits become easier and faster.

1. Introduction

Nowadays, the number of infrastructures and facilities where methane is handled in liquified form is constantly increasing. However, for most of the end user (*e.g.*, power station), methane has to be vaporized and, commonly, it is at high-pressure conditions. However, it is still frequent to handle methane in gaseous form all along the production and usage chain. Therefore, among the safety implications to be considered in risk analysis activities for plants involving methane, both liquid and gaseous accidental Loss of Containment (LOC) have to be accounted. High-pressure jets are hazards of paramount importance in the process safety (Liao et al., 2018), mostly when flammable compounds are considered: in case of either an early or a late ignition, the consequences can be relevant. As well documented by Casal and collaborators (2012), a jet fire or a flash fire can be intended as a major accident initiator.
Focusing on high-pressure jets, works available in literature mainly concerned the free jet situation (that is to say, a jet in unconfined environment). For this reason, nowadays the overall structure of a high-pressure free jet is well known (Franquet et al., 2015) and, the acquired knowledge helped researchers of process safety field in developing simplified mathematical models that, with low computation demand, permit to quickly estimate key physical variables characterizing the accidental scenario. Thanks to this, quantitative assessments of gaseous high-pressure releases are performed, to date, exploiting such practical tools.

Paper Received: 12 September 2020; Revised: 24 February 2021; Accepted: 28 April 2021
Please cite this article as: Colombini C., Pontiggia M., Uguccioni G., Rota R., Busini V., 2021, Ground Influence on High-pressure Methane Jets: Different Concentration Clouds Scenarios, Chemical Engineering Transactions, 86, 541-546 DOI:10.3303/CET2186091

The involvement of either the ground or a piece of equipment placed in the vicinity can be expected to be the most probable accidental scenario in an industrial environment. It is in this more probable situation that, troubles using the simple tools occur: as well demonstrated, when a jet impacts an obstacle, its behavior significantly changes (Colombini and Busini, 2019a; Colombini and Busini, 2019b). Thus, to model this accidental scenario, the useful mathematical models developed for free jet scenarios fail (Pontiggia et al., 2014). To properly investigate this type of situation, only Computational Fluid Dynamics (CFD) analysis can be reliably performed (Batt et al., 2016). However, CFD is not yet drawbacks free: time consumption and expertise needed limit the daily use of CFD to perform risk assessments (Zuliani et al., 2016).
As obstacle, the ground should be counted as possible alternative (Colombini et al., 2020a). In fact, when involved, the increase of the damage area is expected (Hall et al., 2017). In literature, influence of a flat surface (horizontally and vertically oriented) was investigated, both numerically (Angers et al., 2011; Benard et al., 2016; Colombini et al., 2020a; Colombini et al., 2020b) and experimentally (Hall et al., 2017).
Focusing on the scenario of an accidental high-pressure unignited methane jet interacting with an obstacle, this work investigates how the proximity to the ground influences the jet cloud extent. The aim was to highlight how such influence varies when considering different concentrations of methane in air for different release situations. In particular, the main objective was to verify the possibility to extent the applicability range of the procedure proposed by Colombini and collaborators (2020a) (that, by hand calculations, enables safety analysts to quantitatively estimate the hazardous distance reached by the jet cloud in such a kind of scenario) to methane releases evaluated at concentrations levels different from LFL (i.e., at LFL/2 and UFL). Varying the height above the ground of the source term, the effect of the ground was systematically studied through a Computational Fluid Dynamics analysis. Thanks to the sensitivity analysis performed, the main achievement is the demonstration that methane releases observed at different concentrations in air, from sources at different pressures and outflowing from accidental holes of different sizes can be, within the range investigated, assessed using the same analytical model specifically derived, providing a useful alternative of practical precision to more expensive CFD computations. This way, for this specific accidental scenario, delineating the area involved within the flammability limits become easier and faster.

2. Materials and methods

As noted in the previous Section, it can be easily assumed that, when in gaseous form, methane is at high-pressure. Moreover, in certain applications, methane is handled at very high-pressure, hundred times the ambient one. In such situation, an upstream pressure larger than the critical threshold to achieve chocked conditions is noticed (Cameron and Raman, 2005), and a supercritical release is expected. In this case, specific numerical measures have to be considered aiming to correctly perform the simulation of complex phenomena such as shock waves formation and Mach disk establishment (Franquet et al., 2015). However, since the present work is targeting a safety analysis, the far field zone of the jet is of primary importance. Therefore, a poor representation of the early stage of the jet is acceptable as long as the effects of these phenomena are taken into account on the jet development. Known as Equivalent Diameter Models (EDM) (Franquet et al., 2015), in literature many are the available approaches to model the jet source term. Although they are based on different assumptions, all of them have been specifically designed to substitute the real high-pressure source with a fictitious one, characterized to be in pressure equilibrium with the environment. In the present work, the well-known model of Birch and collaborators (1984) was used.

Table 1: Equations of the Birch and collaborators (1984) model defining the pseudo-source characteristics of the methane jet inlet. Further details are provided in the work of Birch and collaborators (1984).

Pseudo-source characteristics	Equation
Equivalent diameter	$d_{ps} = d\sqrt{C_D\left(\frac{p}{p_{amb}}\right)\left(\frac{2}{\gamma+1}\right)^{\frac{(\gamma+1)}{2(\gamma-1)}}}$
Mass flow rate	$\dot{m}_{ps} = \rho_{ps} \cdot A_{ps} \cdot v_{ps}$
Total temperature	$T_{TOT,ps} = T_{ps} + \frac{v_{ps}^2}{2\,C_p}$

Table 1 provides the equations of the Equivalent Diameter Model of Birch and collaborators (1984) used to compute the equivalent source specifics. In the Equations of the Table, d_{ps} is the resulting diameter of the pseudo-source, d is the actual orifice diameter, C_D is the discharge coefficient, p is the storage pressure, p_{amb} is the environmental pressure, γ is the specific heat ratio, Cp is the methane heat capacity, $\dot{m}_{ps}$, v_{ps}, ρ_{ps}, T_{ps}, $T_{TOT,ps}$ and A_{ps} are the resulting mass flow rate, velocity, density, static temperature, total temperature and area extension of the pseudo-source, respectively.

Since the accidental scenario is outdoor located, realistic wind conditions were considered. As open field situation, a velocity profile perpendicular to the ground and in accordance with the atmospheric class 5*D* of the Pasquill's categories was supplied to the solver through a User Defined Function (UDF) (Pontiggia et al., 2014). To perform the CFD analysis, the numerical solver Fluent was used. To obtain a good quality representation of the flow field keeping as low as possible the computational costs, the RANS approach (Reynolds Averaged Navier Stokes equations) was adopted. Given the advantages provided, the k-ω SST turbulence model was chosen (Menter, 1993). For all the modeled release situations, a spill from a storage tank (or a pipeline) was guessed, assuming to be constant in time (*i.e.*, steady state condition). The characteristics defining the modeled release situations are reported in Table 2. A representative environmental temperature equal to 300 K was considered.

Table 2: Characteristics defining the modelled situations in terms of upstream source pressure (p), upstream source temperature (T), actual source diameter (d), height of the source above ground (h) and observed methane concentration in air (c). Results of runs 14-26, 44-47, 56-59, 68-71 are taken from the work of Colombini and collaborators (2020a).

Run	p [bar]	T [K]	d [m]	h [m]	c
1-13	65	278	0.0254	0.14, 0.43, 0.72, 1.02, 1.31, 1.60, 1.90, 2.18, 2.47 2.77, 3.06, 3.35, 4	LFL/2
14-26	65	278	0.0254	0.14, 0.43, 0.72, 1.02, 1.31, 1.60, 1.90, 2.18, 2.47 2.77, 3.06, 3.35, 4	LFL
27-39	65	278	0.0254	0.14, 0.43, 0.72, 1.02, 1.31, 1.60, 1.90, 2.18, 2.47 2.77, 3.06, 3.35, 4	UFL
40-43	30	278	0.01907	0.22, 0.52, 0.96, 1.93	LFL/2
44-47	30	278	0.01907	0.22, 0.52, 0.96, 1.93	LFL
48-51	30	278	0.01907	0.22, 0.52, 0.96, 1.93	UFL
52-55	85	278	0.0127	0.25, 0.58, 1.08, 2.16	LFL/2
56-59	85	278	0.0127	0.25, 0.58, 1.08, 2.16	LFL
60-63	85	278	0.0127	0.25, 0.58, 1.08, 2.16	UFL
64-67	120	278	0.0127	0.29, 0.69, 1.28, 2.57	LFL/2
68-71	120	278	0.0127	0.29, 0.69, 1.28, 2.57	LFL
72-75	120	278	0.0127	0.29, 0.69, 1.28, 2.57	UFL

As fluid volume, a rectangular box of different sizes was built for each of the simulations performed. In fact, computational domain dimensions were properly sized in order to avoid any interference with the boundaries but, at the same time, avoiding a useless waste of computational resources. The ground was modeled as a *wall* boundary condition type, with a roughness height equal to 0.01 m, simulating a concrete surface. The wind conditions included are the one already discussed; considering a realistic wind profile developed perpendicularly to the ground, with an intensity of 5 m/s at 10 m from the ground. To this end, all the inlet boundary conditions (i.e., back to the methane inlet, lateral and top boundaries of the box domain) were set accordingly. A vertical planar symmetry in correspondence of the jet axis was used. For what concerns the fluid volume discretization, a full unstructured tetrahedral grid was built. Ranging between 7 and 9 million of cells, the most widely adopted quality criteria (i.e., orthogonal quality and skewness) were always satisfied. Moreover, also the grid independence of the results was checked and positively achieved.

Further information on methodological aspects and CFD model settings which this work is based on are elsewhere extensively covered and detailed (Colombini et al., 2020a).

3. Results and discussion

To investigate the influence that the ground has on the jet behavior, the height of the source above the ground (h) was systematically varied. As well detailed elsewhere, the principal effect of the ground influencing the jet parallel to it is the enhancement of the jet cloud, involving a larger area (Colombini et al., 2020a). Plotting the

axial Maximum Extent (ME) of the jet cloud at the considered concentration versus the height of the source above the ground (h) it is possible to quantify the ground effect. For the runs defined in Table 2, Figure 1 shows this dependence.
Aiming to highlight how the ground influence varies when considering different concentrations of methane in air for different release situations, from the results depicted in Figure 1, unfortunately, it is not possible a clear and effective comparison suggesting useful considerations. However, we can note that:

- Independently by which set of runs is considered, the results trends are similar to each other
- Independently by which set of runs is considered, the smaller the height of the source above the ground, the longer the jet cloud maximum extent results to be (in particular, the increase is almost linear)
- Independently by which set of runs is considered, above to a certain height of the source the jet cloud maximum extent remains practically unvaried
- The greater the pressure, the longer the jet cloud maximum extent results to be
- The greater the source diameter, the longer the jet cloud maximum extent results to be
- The greater the observed concentration, the shorter the jet cloud maximum extent results to be

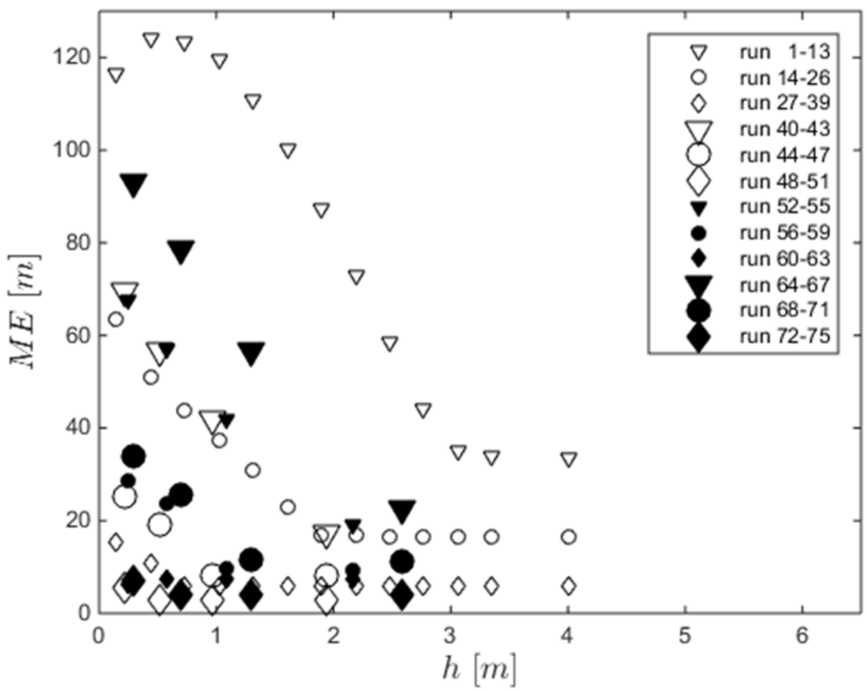

Figure 1: Results of runs listed in Table 2 are plotted in terms of ME versus h. Results of runs 14-26, 44-47, 56-59, 68-71 are taken from the work of Colombini and collaborators (2020a).

From the analysis of the results shown in Figure 1, we can see that these are in accordance with the physics that characterizes the jet development in such a situation (*i.e.*, the Coanda effect (Colombini et al., 2020a)) and that the results agree with previous ones obtained from the Authors (Colombini et al., 2020a; Colombini et al., 2020b). Considering Figure 1, results of runs 1-13 show that, for small values of h, the jet extent is no longer increased, leading to jet clouds of similar maximum length. This occurrence can be due to the fact that, for large horizontal distances from the source reached by the jet cloud, the momentum loss of the jet is such that the ground influence becomes less relevant. For this reason, despite different heights above ground, its influence appears to be similar.
To effectively show which is the dependency of the ME upon only the distance of the source from the ground (h), a proper space that allowed to offset the different methane concentrations in air observed, the different source pressures and orifice diameters considered was defined. To offset both upstream pressure and orifice diameter effect, plane defined in Figure 1 was manipulated as follows:

- y axis was normalized by dividing for the correspondent ME of the free jet (ME_{FJ})
- x axis was divided by the correspondent equivalent source diameter (d_{ps})

This normalization works because both ME_{FJ} and d_{ps} depend on the upstream pressure and the orifice diameter; as previously detailed, the greater the upstream pressure or the orifice diameter, the greater the jet cloud maximum extent results to be, highlighting this dependence; while, as reported in Table 1, d_{ps} depends on both the upstream pressure and the orifice diameter. To offset the effect of the observed methane concentration level, only the x axis required a further manipulation since both ME and ME_{FJ} already depend on the concentration observed. To this end, the ratio c/LFL, where c is a concentration value at which the methane jet cloud can be observed (in this case, LFL/2, LFL and UFL) and LFL is the reference methane concentration chosen, was used to perform the scaling. From the analysis of the results rearranged in the dimensionless space of Figure 2, we can see that the layout of the results appears to be very similar to the one seen in the dimensional space (Figure 1). In particular, once the differences of the sources characteristics and concentration observed are offset, the results well overlap, indicating a similar ground influence for all the runs considered.

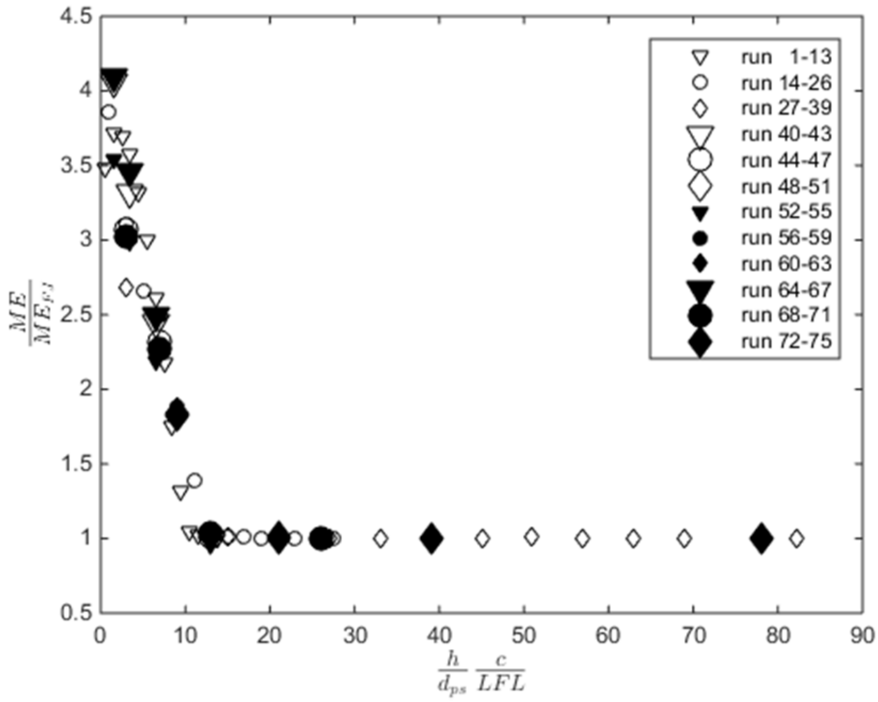

Figure 2: Results of runs listed in Table 2 are plotted in the dimensionless space specifically defined to offset both different upstream pressures, different source diameters and different methane concentrations observed. Results of runs 14-26, 44-47, 56-59, 68-71 are taken from the work of Colombini and collaborators (2020a).

4. Conclusions

Focusing on the scenario of an accidental high-pressure unignited methane jet interacting with an obstacle, this work investigated how the proximity to the ground influences the jet cloud extent. The aim was to highlight how such influence varies when considering different concentrations of methane in air for different release situations (in terms of upstream pressure and orifice diameter).

With regards to the results shown, the results present a good overlap when compared on the dimensionless space, leading to conclude that the ground similarly influences the jets in all the scenarios considered.

Because to offset the different concentrations effect the LFL was used as reference value, the procedure proposed by Colombini and collaborators (2020a) can be quantitatively applied to evaluate methane jets extent of different concentrations without the need of any correction.

Finally, it should be stressed that these conclusions are expected to be valid inside the parameters window investigated. The use of detailed CFD simulations should be always considered both for confirming the estimated values, as well as for obtaining more reliable estimation in scenarios characterized by parameter values outside the investigated window.

References

Angers B., Hourri A., Bénard, P., Tchouvelev A., 2011, Numerical Investigation of a Vertical Surface on the Flammable Extent of Hydrogen and Methane Vertical Jets, International Journal of Hydrogen Energy, 36(3), 2567-72.

Batt R., Gant S., Lacome J., Truchot B., 2016, Modelling of Stably-Stratified Atmospheric Boundary Layers with Commercial CFD Software for use in Risk Assessment, Chemical Engineering Transactions, 48, 61-66 DOI:10.3303/CET1648011.

Bénard P., Hourri A., Angers B., Tchouvelev A., 2016, Adjacent Surface Effect on the Flammable Cloud of Hydrogen and Methane Jets: Numerical Investigation and Engineering Correlations, International Journal of Hydrogen Energy, 41, 18654-662.

Birch A.D., Brown D.R., Dodson M.G., Swaffield F., 1984, The Structure and Concentration Decay of High-Pressure Jets of Natural Gas, Combustion Science and Technology, 36, 249-261.

Cameron, I., Raman, R., 2005. Process System Risk Management, first ed. Elsevier Amsterdam., The Netherlands.

Casal, J., Gómez-Mares, M., Muñoz, M., Palacios, A., 2012, Jet fires: A ‘minor’ fire hazard? Chem. Eng. Trans. 26, 13–20. DOI: 10.3303/CET1226003.

Colombini, C., Busini, V., 2019a, Obstacle Influence on High-Pressure Jets based on Computational Fluid Dynamics Simulations. Chemical Engineering Transactions, 77, 811–816. https://doi.org/10.3303/CET1977136.

Colombini, C., Busini, V., 2019b, High-Pressure Methane Jet: Analysis of the Jet-Obstcle Interaction. Proceeding of the 29th European Safety and Reliability Conference.

Colombini, C., Martani, A., Rota, R., Busini, V., 2020a, Ground Influence on High-Pressure Methane Jets: Practical Tools for Risk Assessment, Journal of Loss Prevention in the Process Industries 67, 104240.

Colombini, C., Carlini, L., Rota, R., Busini, V., 2020b, Ground Interaction on High-Pressure Jets: Effect on Different Substances. Chem. Eng. Trans. 82, 367-372. https://doi.org/10.3303/CET2082062

Franquet, E., Perrier, V., Gibout, S., Bruel, P., 2015, Free underexpanded jets in a quiescent medium: A review. Prog. Aerosp. Sci. 77, 25–53. DOI: 10.1016/j.paerosci.2015.06.006.

Hall J.E., Hooker P., O'Sullivan L., Angers B., Hourri A., Bernard P., 2017, Flammability Profiles Associated with High-Pressure Hydrogen Jets Released in Close Proximity to Surfaces, international journal of hydrogen energy, 42, 7413-21.

Liao, N., Huang, K., Chen, L., Wang, Z, Wu, J., Zhang, F., 2018, Numerical simulation of gas dispersion during cold venting of natural gas pipelines. Adv. Mech. Eng. 10, 1–14. DOI: 10.1177/1687814018755244.

Menter, F.R., 1993, Zonal Two Equation kω Turbulence Models for Aerodynamic Flows. 24th Fluid Dynamics Conference.

Pontiggia M., Busini V., Ronzoni M., Uguccioni G., Rota R., 2014, Effect of Large Obstacles on High Momentum Jets Dispersion, Chemical Engineering Transactions, 36, 523-528 DOI: 10.3303/CET1436088.

Zuliani C., De Lorenzi C., Ditali S., 2016, Application of CFD Simulation to Safety Problems – Challenges and Experience Including a Comparative Analysis of Hot Plume Dispersion from a Ground Flare, Chemical Engineering Transactions, 53, 79-84. DOI: 10.3303/CET1653014.