Modelling of a supersonic accidental release in Oil&Gas offshore: characterisation of a source box

A. Carpignano*
T. Corti*
A.C. Uggenti*
R. Gerboni*

* Politecnico di Torino DENERG,
Torino, Italy

The Oil&Gas offshore installations context is characterised by limited and congested spaces and, because of this reason, consequences assessment, which plays a fundamental role in the definition of safety barriers, poses complex modelling challenges.
During the early stages of the platform design, mono-dimensional or semi-empirical models are mostly used in industrial applications because of their simple and rapid implementation; nonetheless, they do not gather the peculiar geometry of spaces and their lack of accuracy often brings to risk overestimation, hence materials and economical wastes. Besides, the safety solutions adopted in the conventional industrial installations, such as the increase of distances among critical equipment, are not viable in the offshore context. Computational Fluid Dynamics (CFD) models can accurately simulate accidents evolutions but require a computational effort incompatible with the early design phase time schedule. This work proposes a hybrid solution targeted to be a compromise between the empirical and the CFD models, splitting the accident evolution in two parts, of which the first one is replaced by a "source box" suitably chosen while the second one is CFD modelled in order to reduce the computational time while maintaining a good accuracy.
The purpose of this work is the description of the source box and its characterization through the main variables involved in the release phenomenon and their value ranges.
A case study is also presented to validate the representativeness of the chosen model.
Keywords: *CFD, supersonic releases, dispersion, source box, offshore platforms, risk analysis.*

Modellazione CFD di un rilascio supersonico accidentale in ambito Oil&Gas offshore: caratterizzazione di una source box. *Il contesto delle installazioni Oil&Gas offshore è caratterizzato da spazi limitati e congestionati e, anche per questa ragione, l'analisi delle conseguenze, che gioca un ruolo fondamentale per la definizione delle barriere di sicurezza, propone sfide modellistiche complesse.*
Durante le prime fasi della progettazione delle piattaforme, così come in molti altri ambiti industriali, vengono applicati modelli semi-empirici o modelli monodimensionali, anche grazie alla loro semplicità e rapidità di implementazione; tuttavia, questi modelli non gestiscono bene la geometria degli spazi e la loro scarsa accuratezza spesso porta a sovrastime dei rischi e, quindi, a sprechi di materiale ed economici. Inoltre, le soluzioni per garantire la sicurezza adottate in ambiti industriali convenzionali, come l'aumento delle distanze fra apparecchiature critiche, non sono applicabili in ambito offshore. I modelli CFD (Computational Fluid Dynamics) possono, invece, simulare accuratamente l'evoluzione degli incidenti e la geometria del contesto, ma richiedono un impegno computazionale incompatibile con i tempi delle prime fasi di progetto.
Questo lavoro propone una soluzione ibrida che individua un compromesso fra i modelli empirici e CFD, dividendo il fenomeno dei rilasci accidentali in due parti, delle quali la prima è sostituita da una "source box" accuratamente scelta dall'analista, mentre la seconda parte viene modellata in CFD. Ciò consente di ridurre molto l'onere computazionale, pur mantenendo una accuratezza accettabile.
Lo scopo di questo lavoro è fornire una descrizione della source box e una sua caratterizzazione attraverso le variabili più importanti nell'evoluzione del fenomeno di rilascio e i relativi range di valori.
Viene, inoltre, presentato un caso studio per validare la bontà del modello scelto.
Parole chiave: *CFD, rilasci supersonici, dispersione, source box, piattaforme offshore, analisi di rischio.*

1. Introduction

Risk analysis aims at identifying the critical issues of a system, estimating the damages resulting from accidents that may occur during the construction and management of a system and defining the improvements and the barriers necessary to prevent and/or mitigate those consequences. The risk is defined as a function of the frequency (evaluated through probabilistic analyses) and the damages of the potential hazardous scenario (evaluated by both qualitative and quantitative methods).

In the Oil&Gas (O&G) sector, the necessity of continuously ensuring the operators safety and the environment safeguard, notwithstanding activities that are hazardous due to the constant presence of pressurized, flammable and toxic substances, makes the field of consequences evaluation of crucial importance since the very preliminary design of plants.

Offshore platforms, although various in size and deployment, have common structural features. They are composed of decks, hosting different equipment dedicated to the steps of the hydrocarbon extraction process and premises for personnel accommodation, and they are separated via floors which can be grated or laminated. This difference, of course, greatly influences the dispersion behaviour in case of a release. The most frequent equipment that constitute the internal structure of each deck are vessels dedicated to separation, compression and other steps of the process and pipelines: both kinds of components can operate under pressure that can reach up to 100bar values. The area of each deck can be of some hundreds of square meters, and the height of each floor usually ranges between 3 and 6 meters, which clarifies how congested offshore O&G installations are. With their complex geometry and their customisation to specific production objectives and peculiarities, it is difficult to take advantages from risk and consequences analyses performed for other installations. However, in general, the possible accidents that can originate on a platform, and for which a consequence analysis is preliminary required, are ruptures in any of the pressurised components: these can provoke an immediate formation of a supersonic under-expanded jet and the probable jet interaction with some of internal platform items.

The main O&G consequences assessment methodologies are based on semi-empirical or CFD models, each one with its advantages and drawbacks. Nowadays, the empirical models are the most used in the market (Zio and Pedroni, 2012), mainly because of their simple and rapid implementation due to the geometry and physical phenomena approximations; unfortunately, it can result in consequences overestimation, and therefore in overprotected structures and waste of materials and money. On the other hand, CFD models (Davis *et al.*, 2013) are capable to take into account complex geometries and phenomena neglected by the parametric models, but their computational cost cannot allow an extensive CFD analysis utilization in the platform risk assessment during the design and construction phases; therefore, only few scenarios are typically analysed, or, anyway, less scenarios than those that can be selected using the Quantitative Risk Analysis (QRA). The CFD simulations may indeed take so long that results become available when the key design choices are already made. Consequently, CFD sees its role reduced to a final verification. Nowadays its importance is growing thanks to its major capability and effectiveness and the steady increase of computing power.

This work proposes a development of the hybrid solution proposed at Politecnico di Torino and presented by Uggenti *et al.* (2016) targeted to be a compromise between the empirical and the CFD models: this solution splits the accident evolution in two parts, of which the first one is replaced by a “source box” suitably chosen while the second one is CFD modelled in order to reduce the computational time while maintaining a good accuracy. The aim of this work is the description of the source box characterization.

To fulfil the objectives of the work, the release of flammable substances (e.g. natural gas) from a high-pressure storage has been selected as case study among the reference accidents for an O&G rig.

2. Methodology

2.1. CFD models

Several CFD models exist and may be deemed suitable for the simulations required by the present study. They are divided into three classes: Reynolds Averaged Navier Stokes (RANS), Large Eddy Simulation (LES) and Detached Eddy Simulation (DES).

Among them, RANS models are based on the idea that every instantaneous flow parameter of the Navier Stokes (NS) equations can be decomposed into a time averaged part and a turbulent fluctuating part, if the flow is statistically stationary. For example, the velocity can be expressed as in (1):

$$u(x,t) = \overline{U}(x) + u'(x,t) \qquad (1)$$

The most common RANS classes are the two equation models k-ε and k-ω, which integrate the NS equation with the kinetic energy transport equation k, the turbulent dissipation ε or the specific turbulent dissipation ω. The k-ε model well represents the round jet and the free stream region, as shown in Testa *et al.* (2013), meanwhile the k-ω well represents the jet impingement. Menter (1993) proposed the SST k-ω model which switches between the k-ε and k-ω according to free stream or near wall regions. Yin *et al.* (2013) demonstrated that the SST k-ω performs better than k-ε models in jet impingement simulations. Also other authors (Kim *et al.*, 2006) (Chougule *et al.*, 2011) supported the use of SST k-ω.

LES are space averaged NS equation with Favre approximation. LES models divide the dominion into grid and sub-grid regions with a filter function that separates flow larger eddies (which will be resolved because anisotropic) and smaller eddies (which will be only modelled, because isotropic) of the flow. The model is simple but shows some issues in complex turbulent flows simulation, it is not valid near walls and presents a high computational cost.

DES combines the favourable aspects of RANS (efficiency and accurate calculation at boundary layers) and LES (accuracy in highly separated flows) (Spalart and Schur, 1997), thus a temporal and spatial decomposition. The model is efficient and accurate but it needs high computational resources and it is still under development.

The peculiarities of these models and their suitability for the application to the offshore context have been discussed in a previous work (Impalà, 2016). From the outcomes of the study, for this work, the SST k-ω model was chosen as best fit for the simulation requirements of the phenomenon inside the source box.

2.2. The source box concept

The hybrid model (Uggenti *et al.*, 2016) considers separately the release and the dispersion phase, however only the latter is simulated with CFD at the time of the consequences assessment. The first phase, instead, is considered to evolve within the source box (Fig. 1) and as such, it is studied again with CFD: the results in terms of concentrations and velocities are, then, calculated with the model and defined for the points that lay on the source box surface. The va-

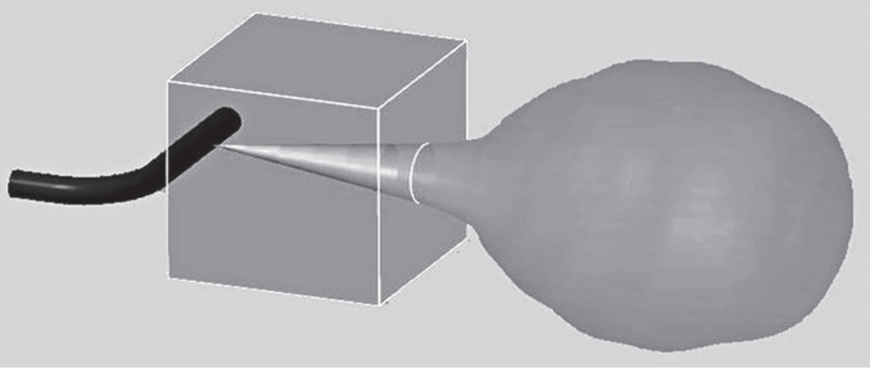

Fig. 1. The source box concept for the simulation of the supersonic release phase.
Il concetto di source box per la simulazione della fase supersonica di rilascio.

lues of concentration and velocity of the flux on the surface of the source box are calculated according to the input conditions, such as the release pressure and the geometry. Solving the whole problem with one single CFD model would pose considerable convergence problems as the velocities involved in two main phases (supersonic release and dispersion) have different orders of magnitude.

The idea is to create a sort of catalogue of source boxes, which the analyst can browse to choose the most suitable one according to the environmental and accidental characteristics of the case study, without the need to re-do the CFD analysis of the release phase.

The source box is a ready-for-use tool of which all the main characteristics are known: the domain, the modelling and the mathematics of the phenomenon, the variables which influence the phenomenon within the source box and their ranges.

Previous studies (Guasco, 2015), (Pederiva, 2015) demonstrated the possibility to represent the release phase into a source box and each author analysed a different jet impingement against an obstacle. The present work is specifically dedicated to the source box definition and characterisation.

2.3. *The source box model*

The source box has dimensions that allow to host the release phase, which means that it hosts the space within which the velocity of the jet decreases from the supersonic state to a value that is comparable to the wind speed.

It is possible to define this space according to the Stephens hypothesis (Stephens, 2002): the author has shown that after a length equal to 10 X_{Mach}, the expansion phase is closed. X_{Mach} is the distance between the source and the point where the first Mach disk appears and it depends from the storage pressure and from the release diameter:

$$X_{Mach} = 0.645 \cdot d_{release} \sqrt{\frac{P_{storage}}{P_{ambient}}} \qquad (2)$$

The source box dimension calculated in this way allows to ensure that the transition from the supersonic state to the subsonic one of a free jet happens within the box: this is even more true for the case studies that pertain to the O&G rigs, where jets usually encounter obstacles during their evolution. These real case jets are slowed down within the dominion of the source box defined for the free jet and the transition of speed is clear.

In (Uggenti *et al.*, 2016) this hypothesis has been tested comparing calculated results obtained from a CFD simulation in a 10 X_{Mach} dominion with experimental data available in (Novembre *et al.*, 2006) for a natural gas release. The two datasets were in good accordance.

2.4. *Variables and their ranges*

In order to describe the phenomenon inside the source box, it is necessary to identify the most influencing variables. The set of source boxes that compose the catalogue is defined according to the values of the most influencing variables. Four variables have been identified to classify the source boxes: release pressure, dimension ratio, rupture diameter and jet direction.

The release pressure is probably the most important factor due to its influence on jet characteristics and domain dimensions. The pressure at which a flow exits from the rupture determines also the fact it is subsonic or supersonic and the limit is represented by the critical conditions, as defined in (3):

$$\frac{p_0}{p_{crit}} = \left(\frac{2}{\gamma+1}\right)^{\frac{\gamma}{\gamma-1}}$$

where p_{crit} is the critical pressure, p_0 is the ambient pressure and γ is the specific heat ratio of the released substance. The release pressure influences the release speed in the case of a subsonic jet and it determines the critical conditions of a supersonic jet; furthermore, when the jet is choked, the release pressure determines the substance density and, therefore, the mass flow rate. The source box dimensions are ten times X_{Mach}, which is directly proportional to the square root of the release pressure. With this domain assumption, when the release pressure increases, the source box dimensions increase too. An issue related to the release pressure is its range: on a platform, pressure in pipes or other containers can be up to 100 bar. In this study, a pressure step of 5 bar is assumed reliable for the source box analysis: the pressure values analysed in this work will be multiple of 5 in the range 5-100 bar.

There are mainly two geometrical parameters in the source box setting: the dimension of the obstacle and the distance between obstacle and source. If these parameters are taken into account separately, the possible combinations are too many and this would not only increase the difficulty during the source boxes description, but, above all, it would hamper the source box choice, which the analyst will do for the case under study. In order to ease the proper source box selection, it is possible to classify the obstacle size and its distance from the source with a consideration on the impinging jet phenomenon. It is possible to imagine to have a jet release with constant characteristics and its impingement on a variable size obstacle in front of it. When the jet size, definable with its diameter d_{jet}, is bigger than the obstacle size, definable as the diameter $d_{cylinder}$ for a cylinder or the width L_{plate} for a plate, the jet overtakes the obstacle with only some turbulences due to the impingement. In the opposite case, when the jet finds an obstacle with a dimension that is bigger than d_{jet}, the impingement blocks the stream which will not move anymore along the axis, but towards a normal direction. When the dimensions are comparable, the impinging jet can have different behaviours, which depend also on the type of obstacle. In the case of a cylinder, the flow can lose momentum but continues around the cylinder body due to the Coanda effect for which streamlines of a supersonic flow move along a curved surface without detachment. Shock waves are not observed in this case because of low Mach numbers (less than 2) reached by the flow. In the case of a flat plate, the decrease of momentum is higher due to a bigger opposing face area and flow streamlines do not continue on the same direction of the centreline.

In order to move from phenomenological observations to values, it is possible to define the dimensions ratio A between jet and obstacle as:

$$A_{cyl} = \frac{d_{jet}}{d_{cylinder}}$$

in the case of the cylinder (4)

$$A_{plate} = \frac{d_{jet}}{L_{plate}}$$

in the case of the flat plate. (5)

The dimensions of the obstacle are referred to the face area, which, for a plate is its area, but for a cylinder it is the surface projected on the plane normal to the centreline. These dimensions represent the surface that the jet "sees" at the impingement. In the case the size of the jet is smaller than the obstacle, A tends to zero and the obstacle is like an infinite plate on which the flow stops. In the case the size of the jet is comparable to the obstacle, A is equal to 1. In the case the size of the jet is bigger than the obstacle, A tends to infinite and the obstacle influences the jet like a needle in a river flow. Classifying the geometry of the source box in this way is an approximation as there is a series of infinite possibilities, but this method allows simplifying the approach of the source boxes study in a reliable way as the extreme possibilities are considered.

The size of the rupture for risk analysis is standardized (OGP, 2010). For this study, only the smallest values are considered, chosen among the most probable: 5mm, 30 mm and 100mm. From its definition, the source box dimensions are directly proportional to the rupture size.

The jet direction can be upward, downward, horizontal or one of the infinite diagonal directions between the horizontal and the vertical positions. Even if the horizontal direction is the most studied, the vertical direction should be taken into account, too: the released substance can move toward the upper or the lower deck and have a different dispersion pattern with respect to the horizontal release. In the source box model presented in this work, the upward, downward and horizontal directions are addressed.

2.5. Resulting source box catalogue dimension

The source box characterisation, as defined in the previous paragraphs, leads to an important result that is the evaluation of the catalogue dimension. Having defined the range of variability or the type of values that each characterising parameter of the source box can assume, it is possible, in the end, to calculate the number of their possible combinations that is the number of possible source boxes the analyst may choose among when facing a real life offshore platform consequences analysis.

In details, Table 1 recaps the ranges and values of each parameter: 20 values are chosen for the release pressure, 3 values for the rupture diameter, 3 values for jet direction and 3 values for dimension ratio. With this categorization, a total of 540 source boxes for each obstacle are identified and should be analysed. These combinations should be foreseen for the two selected types of obstacle.

The viability of this model has been tested choosing a set of ten source boxes for which the dimensional factor A, rupture diameter and jet direction where defined and a range of ten values of pressure where considered. Besides, the ten source boxes where analysed for both kinds of obstacles (leading to a total of 20 source boxes analysed).

2. Case study

The case study is based on the release of CH_4 from a rupture in a pressure pipeline or vessel. Figure 2 shows the source box for the case of

Tab. 1. Source box parameters and their ranges of variability.
Parametri caratterizzanti della source box e possibili valori che questi possono assumere.

Parameter	Range or values
Pressure	5-100, step 5 bar = 20 values
Dimensional factor A	0
	1
	Infinite
Rupture Diameters (mm)	5
	30
	100
Jet direction	Upward
	Downward
	Horizontal

a cylindrical obstacle. The case study was performed via a series of simulations with variable release pressures and fixed dimension ratio, jet direction and rupture diameter.

The release pressure affects the domain dimensions, therefore at every run the source box and the mesh change. The jet direction is assumed horizontal, the rupture dimeter is d_{exit} = 30mm and the dimension ratio is A = 1, meaning the obstacle dimension is similar to the jet dimension at the impingement. In order to characterise the obstacle, a standardized pipe DN250 is considered with an outer diameter d_{out} = 273mm. The same value is used for the plate width, while its thickness is 100mm. The range of pressure values is 35-80 bar with a step of 5 bar, so that 10 values of pressure are considered for performing the simulations.

In order to check the accuracy of the simulations, the grid independence analysis was performed.

The number of runs for this series is 10 for each obstacle and the results are obtained as function of the release pressure. The output value of these simulations is the CH_4 concentration averaged on the surface according to (6):

$$\frac{1}{A}\int \Phi dA = \frac{1}{A}\sum_{i=1}^{n}\Phi_i \left|A_i\right| \quad (6)$$

where Φ_i is the value of the variables on i-cell surface, A_i is the i-cell surface and A is the total area of the SB surface.

Figures 3 and Figures 4 show an exemplary pattern of CH_4 molar concentration on the planes (horizontal and vertical) passing through the centre of the inlet: it is possible to see how the jet behaves during and after the impingement against the cylinder or the flat plate. In the case of the cylinder, the jet overtakes the obstacle and shows the Coanda effect.

In the case of the flat plate, the obstacle does not allow the jet to continue in its direction, but it diverts it on the other directions perpendicularly to the jet axis.

In Figure 4, the effect of the obstacle at the impingement zone is shown from the point of view of the plane normal to that of Figure 3.

From these figures, it is possible to understand that, in the case of the cylinder, the source box faces with a significant concentration are North, Up and Down, while in the case of the flat plate these are East, West, Up and Down.

Table 2 reports the values of CH_4

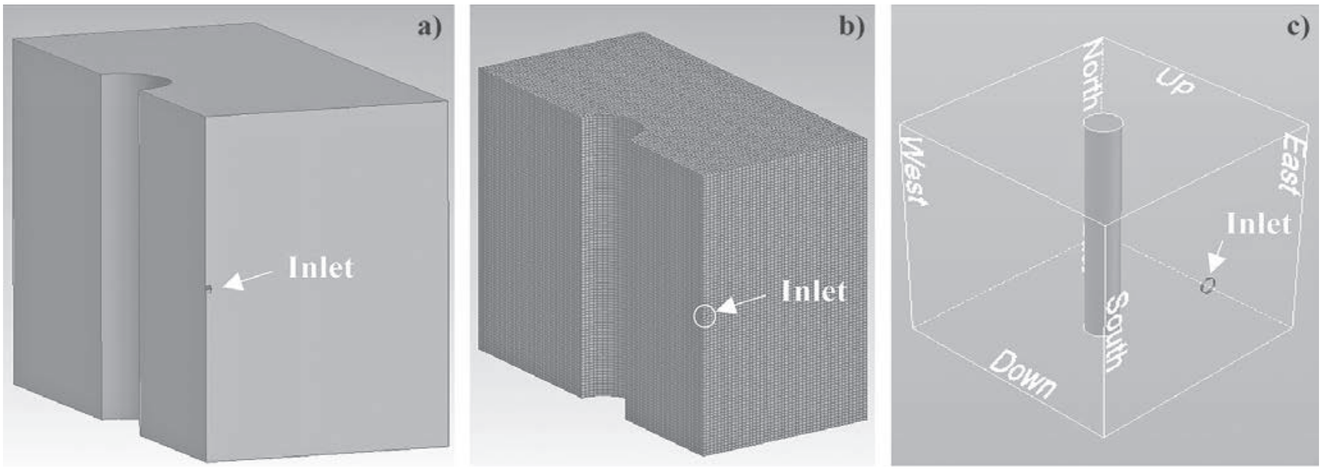

Fig. 2. Source box as domain for CFD simulation of a jet release through an inlet that is represented by a rupture in a pressure pipeline or vessel. a) Half domain with cylinder obstacle and symmetry plane passing through its axis. b) Hexagonal mesh of the source box. c) Nomenclature of the source box faces as reference for the simulation results.
Source box come dominio per la simulazione in CFD di un rilascio di un getto attraverso un inlet rappresentato da un foro di rottura in un tubo o un serbatoio in pressione. a) Semidominio con ostacolo cilindrico e piano di simmetria passante per il suo asse. b) Meshatura della source box. c) Nomenclatura delle facce utilizzata per riferire i risultati della simulazione.

concentration on the source box faces, calculated making use of the model chosen according to the considerations reported in the Methodology paragraph. It is possible to notice a decreasing trend in molar fraction in both obstacle shape cases when the pressure release rises. Furthermore, in both cases, Down and Up faces have almost the same CH_4 concentration.

This is due to the fact these faces are symmetric with respect to the release direction after the impingement, there is no wind interaction that could modify the direction of the jet after the impingement and the gravity effect in this release phase is negligible.

At this point is possible to make some considerations:

- the effect of pressure on the domain size is higher than its effect on released gas transportation, in fact:
  - in case the source box dimensions were the same, the concentration should have increased with pressure while,
  - if the pressure were constant, the concentration should have decreased with the source box dimensions
  - as in this case, when the pressure release increases, the

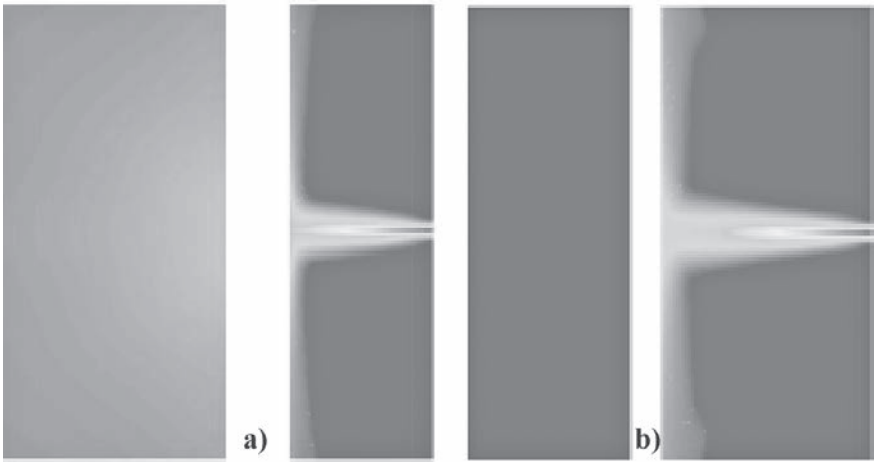

Fig. 3. Lateral view of a supersonic jet impingement against a) a cylinder and b) a flat plate.
Vista laterale dell'impatto supersonico di un getto contro a) un ostacolo cilindrico e b) una piastra.

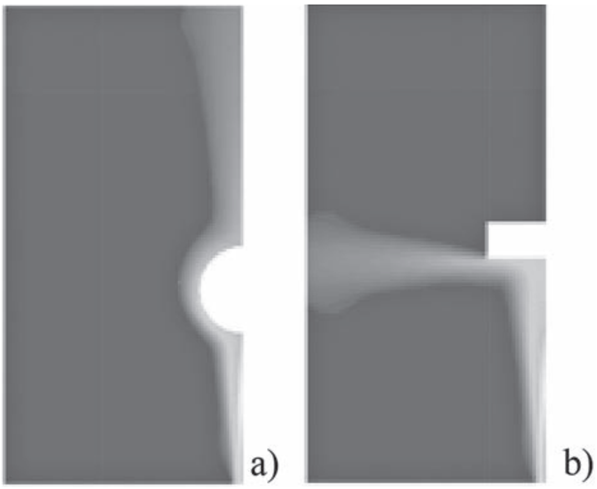

Fig. 4. Top view of a jet impingement against a) a cylinder and b) a flat plate.
Vista dall'alto dell'impatto di un getto contro a) un ostacolo cilindrico e b) una piastra.

Tab. 2. CH_4 molar fraction on the source box faces in case of flat plate or cylindrical obstacle jet impingement.
Frazione molare di CH_4 sulle facce della source box al variare delle pressioni di rilascio nel caso di ostacolo piastra o ostacolo cilindrico.

	Faces – Flat plate obstacle			Faces – Cylinder obstacle		
Pressure [bar]	Down	East	Up	Down	North	Up
35	0.00852	0.00978	0.00852	0.00853	0.0156	0.0085
40	0.00803	0.00978	0.00807	0.00802	0.0144	0.00805
45	0.00717	0.00858	0.00714	0.00756	0.0137	0.00758
50	0.00697	0.00864	0.007	0.00725	0.0133	0.00727
55	0.00668	0.00783	0.00668	0.00674	0.0129	0.00674
60	0.00635	0.00783	0.00637	0.00654	0.013	0.00654
65	0.00602	0.00738	0.00607	0.00651	0.0126	0.00652
70	0.00594	0.00722	0.00594	0.00614	0.012	0.00614
75	0.00586	0.00716	0.00587	0.00537	0.0104	0.00538
80	0.00558	0.00681	0.00551	0.00534	0.0102	0.00534

  concentration on the surface decreases, it can be concluded that the prevailing effect is that of the pressure over the source box dimensions.
- the model shows the same behaviour with both kinds of obstacle and throughout the range of pressures.

3. Conclusions

This study allowed to approach the definition of source boxes with the setting of the domain, the CFD model, influencing variables and their ranges.

It can be considered the first step towards the preparation of a source box catalogue where the most relevant cases are collected.

The positive aspects of the source box are:
- flexibility, as this approach overtakes the problem of case sensitivity
- ease of use, as the source box is a given black box depending on a few input conditions
- expandability, as it is possible to increase the number of cases increasing the variables or their ranges of values.

A critical aspect of this model is represented by the way high pressure releases and large rupture diameters are dealt with, as the source box domain assumes so large dimensions that they become comparable to the entire platform size and the problem description becomes unrealistic. A fixed dimension for the source box domain may be necessary in these cases, and this is matter for future works.

In future works it will also be necessary to look for correlations between the input variables and the flow and concentration parameters at source box faces in order to produce the previously discussed catalogue without the need to perform all the case by case simulations identified in this work.

References

Chougule, N.K., Parishwad, G.V., Gore, P.R., Pagnis, S., Sapali, S.N., 2011. *CFD analysis of Multi-jet air Impingement on flat plate*. Proc. World Congress on Engineering, London, UK, July 6-8.

Corti, T., 2016. *CFD modelling of accidental events in Oil&Gas environment: definition of a source box*. MSc Thesis – Politecnico di Torino.

Davis, S., Gavelli, F., Hansena, O., Middhac, P., 2013. *Onshore Explosion Studies – Benefits of CFD Modelling*. Chemical Engineering Transactions, v. 13, pp. 205-273.

Guasco, S., 2015. *Towards the CFD simulation of accidents on off-shore platforms: dispersion of a turbulent jet hitting a cylinder*. MSc Thesis – Politecnico di Torino.

Impalà, M., 2016. *Models and Tools for the simulation of exhaust dispersion in Oil&Gas offshore platforms*. MSc Thesis – Politecnico di Torino.

International Association of Oil&Gas Producers, 2010. *Risk assessment Data Directory*. Report 434-1 – Process release frequencies.

Kim, E., Park, S.H., Kwon, J.H., 2004. *Numerical Study on* $k - \omega$ *Turbulence Models for Supersonic Impinging Jet Flow Field*. Computational Fluid Dynamics. Springer, 573-578.

Menter, F.R., 1993,. Zonal two equation k-turbulence models for aerodynamic flows. NASA Technical Memorandum 103975.

Novembre, N., Podenzani, F., Colombo, E., 2006. *Numerical study for accidental gas releases from high pressure pipelines*. Proc. European Conference on Computational Fluid Dynamics, Egmond aan Zee, The Netherlands, September 5-8.

Pederiva, E., 2015. *Towards the CFD simulation of accidents on off-shore platforms: dispersion of a turbulent jet hitting a flat plate*. MSc Thesis – Politecnico di Torino.

Spalart, P.R., Allmaras, S.R., 1992. *A one-equation turbulence model for aerodynamic flows*. Proc. AIAA 30th Aerospace Sciences Meeting and Exhibit, Reno, (NV), 6-9 January.

Spalart, P.R., Shur M., 1997. *On the Sensitization of Turbulence Models to Rotation and Curvature*. Aerosp Sci Technol 1, 297-302.

Stephens, M.J., Leewis, K., Moore, D.K., 2002. *A model for sizing high consequence areas associated with natural gas pipelines*. Proc. 4th International Pipeline Conference.

Testa, E., Giammusso, C., Bruno, M., Maggiore, P., 2013. *Fluid dynamic analysis of pollutants' dispersion behind an aircraft engine during idling*. Air Qual Atmos Health 6, 367-383.

Uggenti, A.C., Carpignano, A. , Savoldi, L., Zanino, R. , Ganci, F., 2016. *Perspective and criticalities of CFD modelling for the analysis of Oil&Gas offshore accident scenarios*, Proc. ESREL 2016, Glasgow, UK, 25-29 September.

Yin, J.L., Wang, D.Z., Cheng, H., Gu, W.G., 2013. *Assessment of RANS to predict flows with large streamline curvature*. IOP Conference Series: Materials Science and Engineering. Vol. 52. No. 2.

Zio, E., Pedroni, N., 2012. *Risk Analysis-Uncertainty characterization in risk analysis for decision making practice*. Les cahiers de la sécurité industrielle.

Acknowledgment

This paper would not have been possible without the sponsorship of the Ministry of Economic Development's Directorate General for Safety – National Mining Office for Hydrocarbons and Georesources. In particular, the authors would like to express their gratitude to the people who have supported this work along all the stages of its realization