Development and experimental validation of a novel CFD approach for the simulation of high-pressure accidental gas releases

PhD Candidate: Alberto Moscatello (Energy Department, Politecnico di Torino)

Academic Tutor: Andrea Carpignano (Energy Department, Politecnico di Torino)

Summary

Current regulations impose strict constraints on the safety of risk-relevant facilities as process plants involving flammable and/or toxic substances, Oil & Gas plants and nuclear plants. For all the aforementioned cases, a Quantitative Risk Assessment (QRA) or Probabilistic Risk Assessment (PRA) is mandatory to demonstrate the achievement of the required safety level. One of the main steps of the **QRA/PRA** is the consequences assessment of the critical accidental scenarios coming from the preliminary hazard's identification analysis. The simulation of these major hazards events is required to estimate the associated consequences effects. The state-of-practice for the consequences analysis relies on the use of integral models or empirical models based on massive measurements and few theoretical principles. The real geometry is neglected, and the physical phenomenon is roughly approximated, permitting the simulations of a huge number of accidental scenarios in a short time and with a low computational effort. Due to the extremely simplified assumptions, these models tend to highly overestimate the damage areas, especially when the complexity of the geometry plays a key role in the development of a hazardous scenario (e.g., influencing a gas cloud dispersion), leading to an oversizing of the structures and the safety barriers with a consequent money and resource waste. This is the case of offshore Oil & Gas platforms, where a reduced amount of space is available for the equipment, and nuclear plants since most of the equipment is confined in the containment building for safety reasons. More accurate approaches, like computational fluid dynamics (CFD) ones, can be employed to accurately predict the accident evolution, but they still have too high computational cost. For this reason, they are used only for the verification of few critical scenarios, whereas they cannot provide timely results to drive the plant design.

The present work proposes a CFD approach able to guarantee a suitable accuracy-computational cost trade-off, investigating the feasibility of an effective CFD-QRA integration. The model, developed in ANSYS Fluent, is targeted for the simulation of high-pressure gas releases in large, congested geometries. These scenarios lead to the formation of underexpanded jets, that are likely to impinge part of the equipment near the leak point, hence it is crucial to consider the gas-geometry interactions also within the first centimeters. In literature, several CFD-based methodologies are available to model such scenarios, but most of the works propose the use of empirical correlations to account for the initial jet expansion, thus neglecting possible impingements in the nearfield zone of the jet.

The basic idea of the model here proposed, is the splitting of the accidental scenario in **two-steps**: the supersonic compressible jet near the leak source, defined here as the *release,* and the incompressible subsonic *dispersion* in the plant. The first part is simulated in a small domain, the Source Box (SB), sized according to the criterion of assuring negligible compressibility effects at its outlet boundaries. Its results are then used as gas source boundary conditions for the second part, the gas dispersion simulation in the plant geometry. The approach is called **Source Box Accident Model (SBAM)** and allows to simplify the overall simulation process, since the complexity of the initial underexpanded jet (*1st step*) can be addressed separately, in an *ad hoc* domain. Furthermore, the possibility to get the SB results in few seconds, realizing a Reduced Order Model (ROM) reproducing its behavior, was investigated at a preliminary stage giving promising results. As a consequence, a strong computational cost saving is

expected, since the dispersion simulation (*2nd step*) is largely faster and easier than the SB one as discussed in the results.

Once the SBAM main features, applicability range and novelties are introduced, a **numerical benchmark analysis** and the model **experimental validation** are presented. For these purposes, SBAM is applied to several case studies involving the accidental high-pressure methane release on an offshore platform.

The *benchmark* CFD simulation is based on the state-of-the-art practices for ANSYS Fluent, and does not imply the splitting of the phenomenon, indeed, both the release and the dispersion are modelled in the plant geometry. Comparing this last simulation with SBAM on the same case study, results show a good agreement as small relative errors are obtained on several risk-oriented parameters, e.g. the flammable volume. The comparison involves also the implementation features and simulation time, hinting that the splitting of the phenomenon can represent a strong advantage in terms of computational effort and simulation implementation.

The *experimental validation* concerns the reproduction of several high-pressure methane releases on a platform under wind condition. The SEASTAR WT, an open cycle subsonic wind tunnel, was employed to reproduce the wind field. A 1/10 scaled platform mockup, equipped with gas sensors, was realized to represent the real environment. Several case studies, obtained changing the wind direction, the release pressure and the release position and direction, are considered. To assure the fluid dynamic similarity between the real scale and the model scale, a **scaling procedure** tailored on gaseous critical flows under wind conditions was developed. This methodology is based on the preservation of a dimensionless parameter, the dimensionless discharge momentum flux, which includes the wind velocity and the gas release parameters. Predicted and observed values of concentrations at sensors locations are at first directly compared to analyze the qualitative gas distribution in the domain. Secondly, a more robust methodology for gas dispersion models validation is employed to verify the SBAM performance. This last approach uses some statistical parameters to account for random and systematic errors and to quantify the correlation between the two sets of data. From the analysis of the results, it can be deduced that the model predictions are in good agreement with the measured values, hinting similar gas distributions with a slight overestimation tendency. Basing on the statistical parameters criteria, a preliminary validation of SBAM is achieved, since its predictions result acceptable in most of the test cases.

To conclude, the **main outcomes** of this study are presented. The work offers a new perspective on the accident simulation methodologies related to the QRA for congested layout plants, proposing a CFD approach to accurate model high-pressure gas impinging jets without a prohibitive computational effort. Furthermore, to overcome the lack of suitable methodologies for the realization of scaled experiments involving critical gaseous flows under wind conditions, a novel scaling approach is presented. This procedure allows to relate the scaling of fluid dynamic parameters, the wind velocity and the gas release pressure, to the geometry scaling factor. Finally, a relevant contribution was given to the wind tunnel calibration as well as to the gas sensing system optimization, and useful insights on the realization of high-pressure gas releases experiments under wind conditions are provided.