11th. World Congress on Computational Mechanics (WCCM XI)
5th. European Conference on Computational Mechanics (ECCM V)
6th. European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

CFD SIMULATION OF LNG SPILLAGE

R. Marcer[1], B. Yerly[1], L. Pomié[2], B. Lequime[2], Mathieu Rivot[2], Eric de Carvalho[2] and F. Baillou[3]

[1] Principia, La Ciotat France, richard.marcer@principia.fr
[2] Technip, Paris France, lpomie@technip.com
[3] Total, Paris France, francoise.baillou@total.com

Key Words: *LNG, cryogenic fluid, CFD simulation, multi-phase flow, vaporization.*

LNG (Liquid Natural Gas) accidental cryogenic spillage is one of the major safety concerns of Floating LNG facilities-FLNG. Superstructures such as topsides, deck and hull shall be properly protected against these risks.
Stakes of Quantitative Risk analyses and safety studies performed in FLNG detailed engineering require our knowledge and modelling capabilities to be improved. The following complex physical phenomena which occur during cryogenic LNG releases from topsides leak point down to the sea water contact, need especially to be investigate in order to better quantify the extent of these phenomena and evaluate protection requirements:

1) LNG jet fragmentation (liquid jet breakup), vaporization, rainout.
2) LNG pool formation, spreading and coalescence, heat transfer to the LNG pool and vaporization on solid structures leading to LNG vapour dispersion.
3) FLNG deck overtopping, drip and fall along hull vertical structures.
4) LNG pool formation on seawater and spreading.
5) LNG pool vaporization and dispersion from sea level.

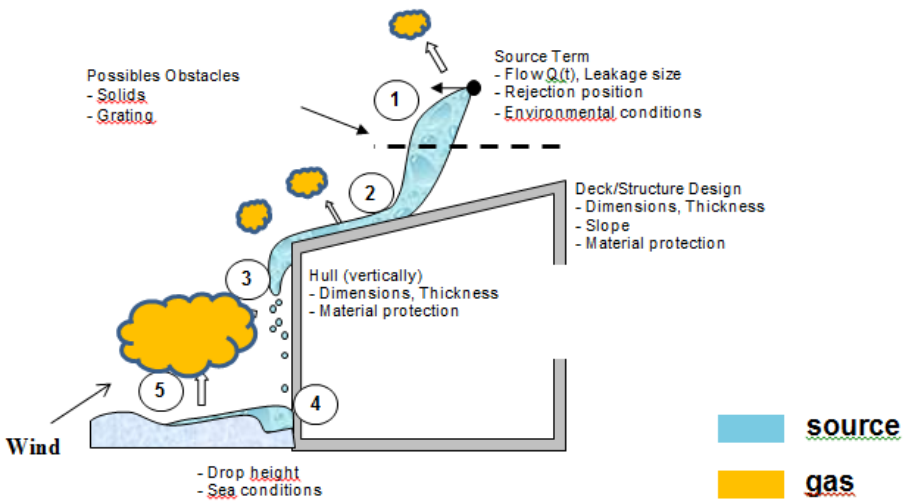

Figure 1: Sketch of the different mechanisms from the jet leak in FLNG topsides (1) down to the pool spreading/vaporization on sea water (4) and vapour dispersion (5)

The objective of the project is to develop a 3D CFD model able to sequentially simulate all the physics of these phenomena. For this purpose, the CFD software EOLE of Principia is considered. It is a URANS code using coupled VOF and mixture models to simulate all dynamic/thermodynamic processes of LNG multi-phases flows, more specifically:

- LNG/water, LNG/air and water/air interfaces mechanisms (VOF model including surface tension)
- Two-phase dispersed LNG droplets in continuous air phase (mixture model)
- LNG vaporization (vaporization/condensation source term in VOF and mixture models)

Academic validations have been performed following a step-by-step approach allowing to investigate the different mechanisms separately:

- Jet mechanisms (mechanical fragmentation – thermodynamic flashing – rainout).
- LNG pool spreading and vaporization on solid substrate.
- LNG pool spreading and vaporisation on water.

The paper will present the theoretical description of the numerical model and examples of academic validation results.

This work has been carried out in the frame of the French CITEPH-83-2012 project supported by TECHNIP and TOTAL.

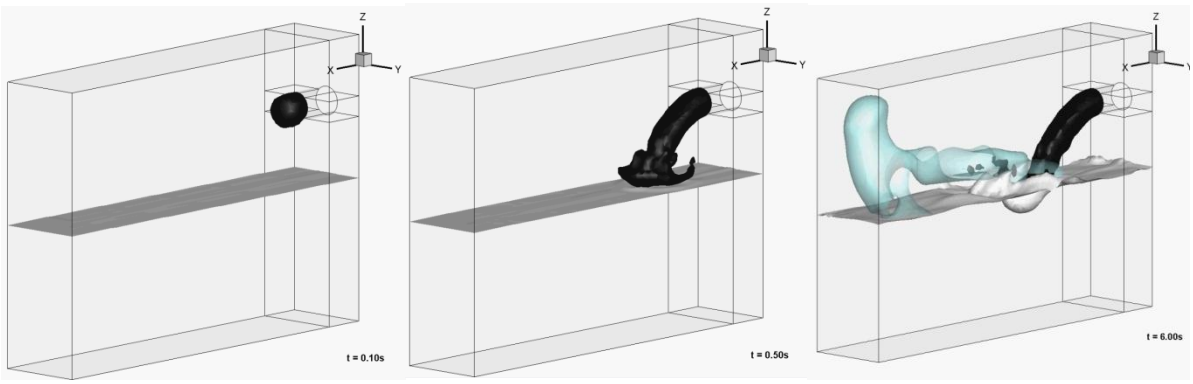

Figure 2: LNG spillage (in black) on water (in grey) and issued vapour dispersion (in blue)