CFD Modelling of LNG Pool Fires.

Leszek Rudniak[1*], Eugeniusz Molga[1]

1 Department of Chemical and Process Engineering, Warsaw University of Technology, Warynskiego 1, 00-645 Warsaw, Poland;

**Corresponding author: Leszek.Rudniak@pw.edu.pl*

Highlights

- SAS turbulence and Eddy Dissipation model effectively applied in modelling LNG pool fire.
- Discrete Ordinates radiation thermal model used for risk assessment.
- The results from numerical simulations were in agreement with the experimental data.

1. Introduction

There is an increased risk of uncontrolled LNG leakage from equipment that transports and stores this product. This is a consequence of the increasing global demand for natural gas in the last two decades. As a result of such leakage of liquid gas from a faulty installation, a pool fire may occur. The pool fire is most often the result of the ignition of the released evaporation of liquid flammable substances, which form a spillage on the surface of the substrate. An example of a surface fire may be the leakage of volatile liquids, liquefied gases (e.g. LNG) from apparatus, tanks or pipelines. As a result of pool fire, thermal radiation and toxic combustion products are generated that pose a threat to the surrounding populations. Therefore, an important factor is the assessment of a safe distance ensuring that there are no irreversible health effects, property and environmental damage.

2. Methods

In the presented work, the CFD model of pool fire has been applied to numerical simulations of methane combustion and risk assessment of such an event. The implemented CFD model in ANSYS Fluent v. 18.2 [1] software was based on the concept of infinitely fast combustion reactions (Eddy Dissipation Model, one step methane combustion reaction) proposed by Magnussen and Hjertager [2]. The gas flow was modelled using the SAS turbulence model proposed by Menter and Egorov [3]. The usage of SAS turbulence model was due to the superior accuracy of SAS results relative to URANS simulation and less hardware and CPU requirements in case of LES models. The numerical simulation of the LNG surface fire also included the radiation mechanism (Discrete Ordinates model). A 300 m diameter and 300 m height domain contained 1.1 million hexahedral elements. In calculations, it was assumed that the surface area of liquid LNG was 81 m what corresponded to the experiment performed by National Sandia Laboratories [4]. The evaporation rate of methane was 0.016 kg/m^2s. Numerical simulations were carried out both in the absence of wind and for different values of wind speed. Steady state velocity field as an initial condition was taken in transient simulations. The time step for the transient calculations was 0.001 s. the numerical simulations were executed in parallel mode using 16 cores in order to speed up the calculations

3. Results and discussion

In this study, the results obtained in the form of temperature profiles, concentration of reactants and products of combustion reaction, intensity of thermal radiation were used to analyse the risk assessment of pool fire due to the leakage of LNG which could occur in industrial installations. As an example, the temperature profile has been shown in Fig. 1, in case of absence of wind.

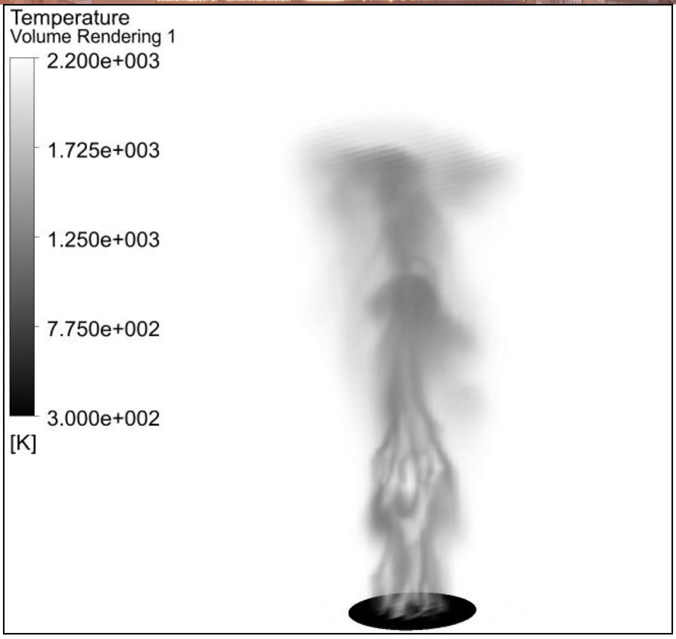

Figure 1. The temperature profile at t=14.6 s.

4. Conclusions

In this work, the transient simulations of LNG fire were conducted using a CFD model implemented in ANSYS Fluent solver v. 18.2. The results obtained from numerical simulation (e.g. fire height) were consistent with the experimental data published by Sandia National Laboratories in 2011. The intensity of thermal radiation were used to determine of a safe distance from a fire source where the radiation dose did not exceed 2.5 kW/m^2 as the minimum dose that causes pain after 60 s [5]. In the future, CFD modelling is also planned to perform a numerical simulation of pool fire on a particular industrial site adjacent to residential and public buildings in order to analyse the risk assessment of pool fire due to the leakage of LNG.

References

[1] ANSYS Documentation, 2017.: http://ansyshelp.ansys.com.
[2] B.F. Magnussen, B.H. Hjertager, On Mathematcal Modeling of Turbulent Combustion with Special Emphasis on Soot Formation and Combustion. 16th Symp. (Int.) on Combustion (1976). Comb. Inst., Pittsburg, Pennsylvania, 1976, pp.719-729.
[3] Menter F., Egorov W., The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 1: Theory and Model Description. Flow Turbulence Combust., 2010, 85, 113-138. DOI 10.1007/s10494-010-9264-5.
[4] T. Blanc, P. Helmick, R. Jensen, Luketa A., The Phoenix series large Scale LNG Pool Fire Experiments. SANDIA REPORT, 2011, Sandia National Laboratories.
[5] J.H. Klote, Smoke control, SFPE Handbook of Fire Protection Engineering, 1995.

Keywords
CFD; Entrainment Velocity; LNG; Pool Fire.

Acknowledgments

The project funded within the project EVARIS by the National Centre for Research and Development (Poland) under the agreement DOB-BIO7/09.03/2015.