

LNG-Engine Safety: Design of Protective Measures Using CFD

Kees van Wingerden*, Nicolas Salaun

Gexcon AS, Fantofvegen 38, 5072 Bergen, Norway
kees@gexcon.com

Nowadays LNG has become a common fuel for fuelling ship engines and similarly for the generation of power. Large gas engines are used for the propulsion of the ships and heating of the buildings. These engines are provided with big exhaust pipes which can be long and often contain a turbo charger or/and a silencer and/or a heat exchanger/boiler and a catalyser. In case of a release of an unburnt LNG-air mixture into the exhaust due to ignition failure subsequent ignition of the resulting cloud of natural gas, inert gases and air in the exhaust may occur. The resulting explosion may cause severe damage to the exhaust and the aforementioned equipment items installed in the exhaust pipe.

Protection is possible by installing explosion vents. The paper addresses how to design this protection measure using the dedicated CFD-tool FLACS. Using the FLACS tool one can describe both the process of gas cloud-build-up after an ignition failure, the subsequent ignition of the cloud, the explosion development (strongly dependent on the presence of turbulence inducing elements such as a boiler, silencer, etc.) and the resulting loading of the exhaust pipe.

The paper addresses scenario choice, the effect of inerting gases (as present in fuel gas (mixtures of inert gases and methane)) on explosion development and how to optimise explosion venting: number of bursting discs, their size and their location.

A secondary hazard is the release of fuel gas (mixture of methane and nitrogen) may be released in a powerhouse. A powerhouse is an enclosed building, which contains the gas engines, potentially an alternator, the gas supply system and a fluid flow system all for power generation.

In case of a release from the fuel gas system into the powerhouse, gas cloud build-up may result and upon ignition cause a damaging explosion.

Using the FLACS CFD-tool the effect of several scenario parameters can be investigated such as the effect of different degrees of mechanical ventilation, release rate and direction, ignition source location and protection measures. The paper presents an example of such a study.

1. Introduction

LNG as a fuel is getting more and more common. Examples include LNG-fuelled ships such as ferries and heating of high-rise buildings. The use of LNG as a fuel also implies introduction of related hazards, which need assessing, and which need where necessary to be counteracted. The current paper addresses two aspects: the possibility of explosions in gas engine exhausts and the possibility and mitigation of hazards associated with gas releases into the building containing the gas engine, sometimes referred to as the powerhouse.

May 2005 Annex VI of the IMO International Convention for the Prevention of Pollution from Ships (MARPOL, 2005) entered into force limiting the sulphur content of marine fuels on a global basis to 4.5%. In January 2012, this limit was further reduced to 3.5 % sulphur. In certain areas, the so-called emission control areas, limits are even considerably lower. Since January 2015, a maximum sulphur content of 0.1 % is maintained in these areas. The emission control areas are the Baltic Sea, the North Sea, North America (US and Canada) and the U.S. Caribbean. In this context LNG is often chosen as an alternative to Diesel.

Similarly, emission from power plants has been regulated (Directive 2001/80/EC, 2001) limiting among other pollutants the emission limits of SO₂. Again, this has caused an interest in using LNG as a fuel instead of coal and other sulphur containing fuels.

2. Limiting the consequences of gas explosions in exhaust pipes

To burn the LNG large gas engines are used. The exhaust gases generated by the combustion are transported away using big exhaust pipes. These exhausts often contain a heat exchanger to optimise the use of the energy generated, a catalyser to limit environmental exposure and a silencer to reduce sound levels.

Failure of ignition in one or more of the engine cylinders may result in the release of natural gas in the exhaust pipe possibly causing an explosion in the exhaust upon ignition. The strength of the explosion depends on the duration of the release into the exhaust pipe, the number of cylinders failing, the composition of the gas, the geometry of the exhaust (diameter, presence of bends and restrictions, etc.), the presence of obstructions such as a silencer or a heat exchanger and the process conditions: temperature and flow speed through the exhaust at the moment of ignition.

To limit the consequences of such exhaust pipe explosions explosion vents can be installed. The resulting reduced explosion overpressures will be dependent on the size of the vent(s), the number of vents and their location and the properties of the explosion venting device.

Simple methods such as e.g. the guidance given in NFPA 68 (National Fire Protection Association, 2013) will not be able to predict the consequences of explosion in exhaust pipes accurately. Models using Computational Fluid Dynamics (CFD) allow for implementation of physical sub-models and therefore a possibility to describe the processes occurring during explosions in detail. The CFD-based tool FLACS (FLame ACceleration Simulator) is a dedicated tool developed for exactly this purpose (Arntzen, 1998).

FLACS has been developed primarily aimed at simulating the dispersion of flammable gas in process areas, and subsequent explosions of gas-air mixtures. FLACS solves the compressible conservation equations for mass, momentum, enthalpy, and mass fraction of species on a 3-D Cartesian grid using a finite volume method. Ever since then the tool has continuously been improved. This development has been going hand in hand with validation against available experimental databases.

FLACS has been used to predict the consequences of explosions in exhaust pipes on several occasions. The validation work that has been performed has generally been performed in geometries not directly relevant for exhaust pipes (Skjold et al, 2013). Nevertheless, an experimental programme have been performed devoted to explosions in exhaust pipes and the results of FLACS-simulations were compared to the experimental results obtained. This work is however confidential and cannot be presented. Figure 1 shows the experimental set-up used for these experiments.

The first step when performing a study to investigate the consequences of an exhaust pipe configuration is to represent the geometry in all its details. An example is shown in Figure 2. The geometry would also contain bursting disc(s) or similar to limit the consequences of the internal explosions including their or its location.

The second step is to choose scenarios to investigate. Since FLACS allows for representing scenarios directly one can consider looking into ignition failure in one or several cylinders, dispersion of this gas into the exhaust mixing with the burned gas from other cylinders in which combustion did take place (once fully mixed the gas may not be flammable anymore, but a flammable mixture may be present in the initial stages of dispersion), duration of the release and the moment of ignition. Also the location of ignition can be a variable although the most likely ignition source location is the engine itself. The tool takes into account the effect of temperature on the explosion limits and the effect of diluents and temperature on the local combustion rate. An example of the development of total flammable fuel injected into the exhaust in time is shown in Figure 3.

The pressure development inside the exhaust pipe is monitored on any location of interest. Examples of pressure time histories are shown in Figure 4.

Based on the results considering several scenarios, the maximum overpressure generated compared to the structural strength of the exhaust pipe consequence reducing measures can be taken by increasing the number of vents, increasing the vent size and/or relocating vents.



Figure 1: Experimental set-up used for investigating the course of explosions in pipes provided with vents and bends representing exhaust pipes. The diameter of the pipe is 700 mm, the maximum length 36 m.

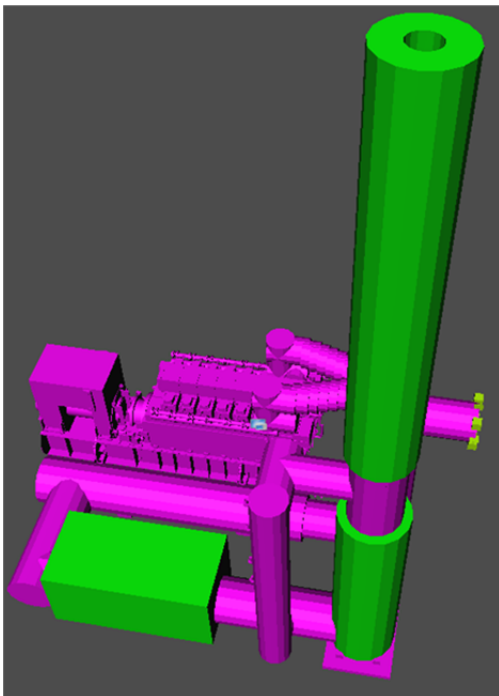


Figure 2: Example of an implemented exhaust system including a catalyst, a boiler and a silencer.

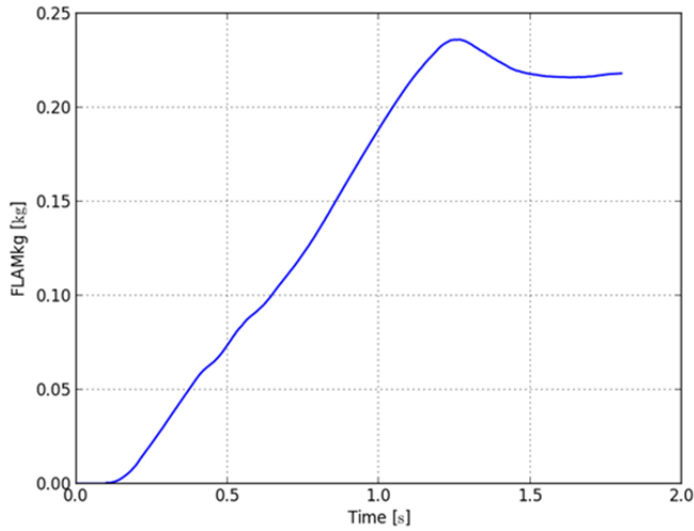


Figure 3: Example of a time history of the total flammable fuel in the simulation domain.

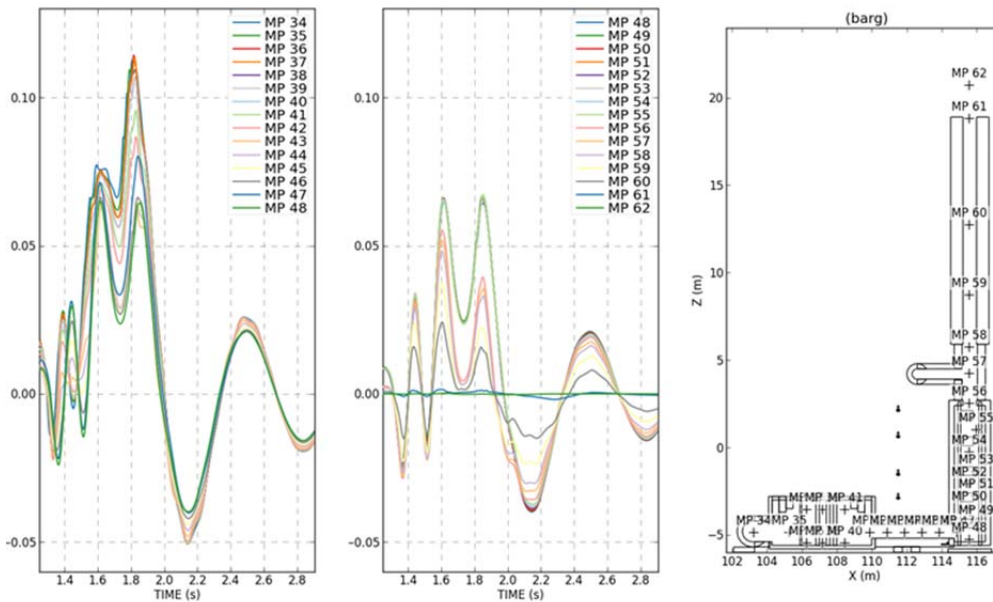


Figure 4: Examples of pressure-time histories measured in the exhaust of a gas engine.

Simulations are performed to investigate the effect of the vent layout on the maximum explosion load inside the exhaust until a configuration is found resulting in acceptable explosion loads.

3. Assessing the consequences of explosions in powerhouses

A leak of LNG or natural gas into a powerhouse can cause serious consequences as well. The jet release can mix with the air in the powerhouse creating a flammable gas cloud its size depending on the release rate, geometrical aspects and the ventilation of the powerhouse building. An ignition of this cloud can result in damage to the powerhouse, the equipment inside and injuries to people. To investigate the hazards CFD simulations can be performed using FLACS according to the following steps:

The first step is the representation of the powerhouse geometry and specification of the main parameters influencing the flow under operating configurations such as ventilation in/outlets and hot surfaces (causing buoyancy).

Next the ventilation conditions prevailing in the powerhouse are simulated. The ventilation flows in the building are the starting point for the dispersion simulations. Several accident scenarios are looked into during the dispersion simulations considering leaks of various sizes (leak rates) and leak directions. These simulations give an idea of the gas cloud build up potential. Explosion simulations for these clouds give the explosion potential.

Figure 5 gives a moment of cloud build-up in a powerhouse containing multiple engines.

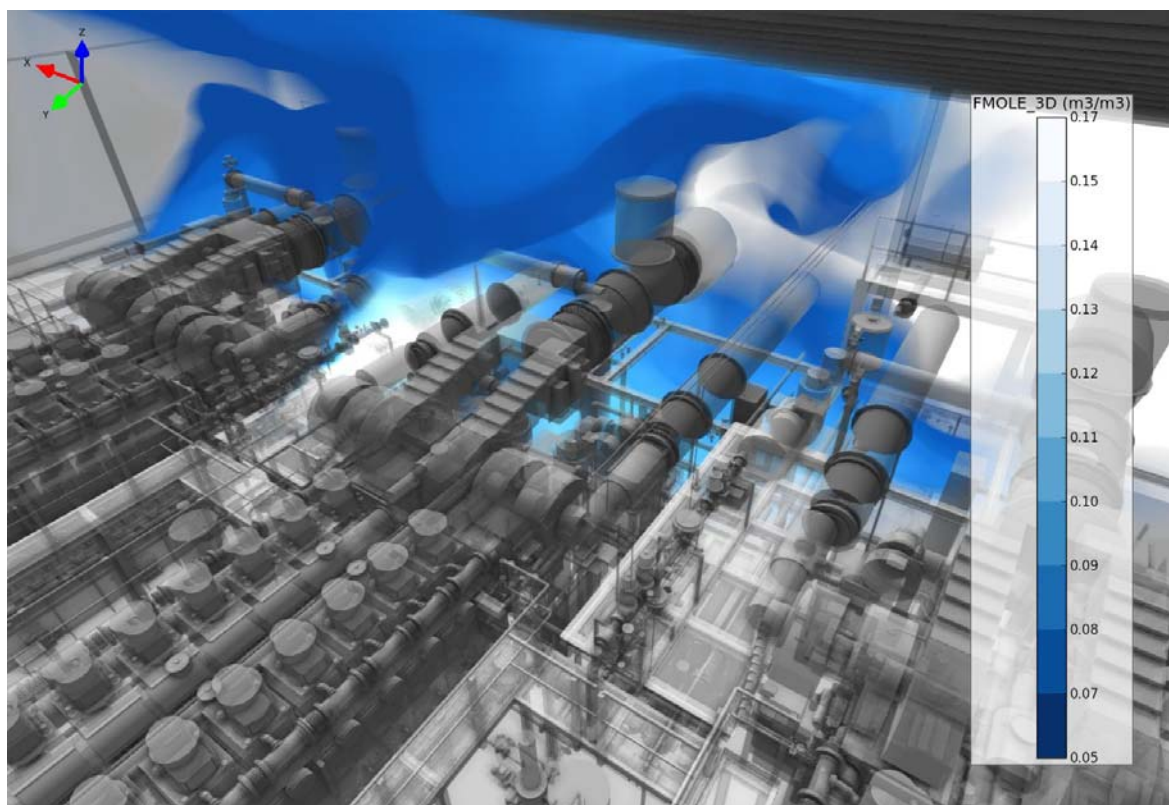


Figure 5: Examples of a moment of gas cloud build-up in a powerhouse containing multiple gas engines.

The dispersion simulations would allow for judging a gas detector system layout if available. Gas detectors can be used to undertake mitigative actions such as closing valves limiting the duration of a release. The dispersion study can show whether the gas detector layout is able to detect all or at least close to all leaks resulting in flammable clouds that can cause damage. The detection of these leaks shall be sufficiently early allowing for taking mitigative actions. The dispersion study also allows for studying the efficiency of these mitigative actions.

The dispersion study would also allow for getting an idea of the distribution of gas clouds that can occur in the powerhouse depending on the release rate and direction. An example is shown in Figure 6.

The efficiency of the ventilation system can be studied based on the size of clouds vs the release rate, the time it takes to dilute short duration releases and identifying areas with low ventilation rates (long durations of presence of flammable gas in case of a release).

The explosion study can show the level of explosion loads that can be expected for the various clouds resulting from the dispersion study. It would allow for designing control room walls and windows as well as other safety critical equipment. The explosion study can be extended to consider mitigative actions such as explosion venting to limit loading on walls and especially the aforementioned control room walls and windows. Special attention should be given to piping connected to walls potentially resulting in pipe rupture upon wall movement or failure and an ensuing escalation.

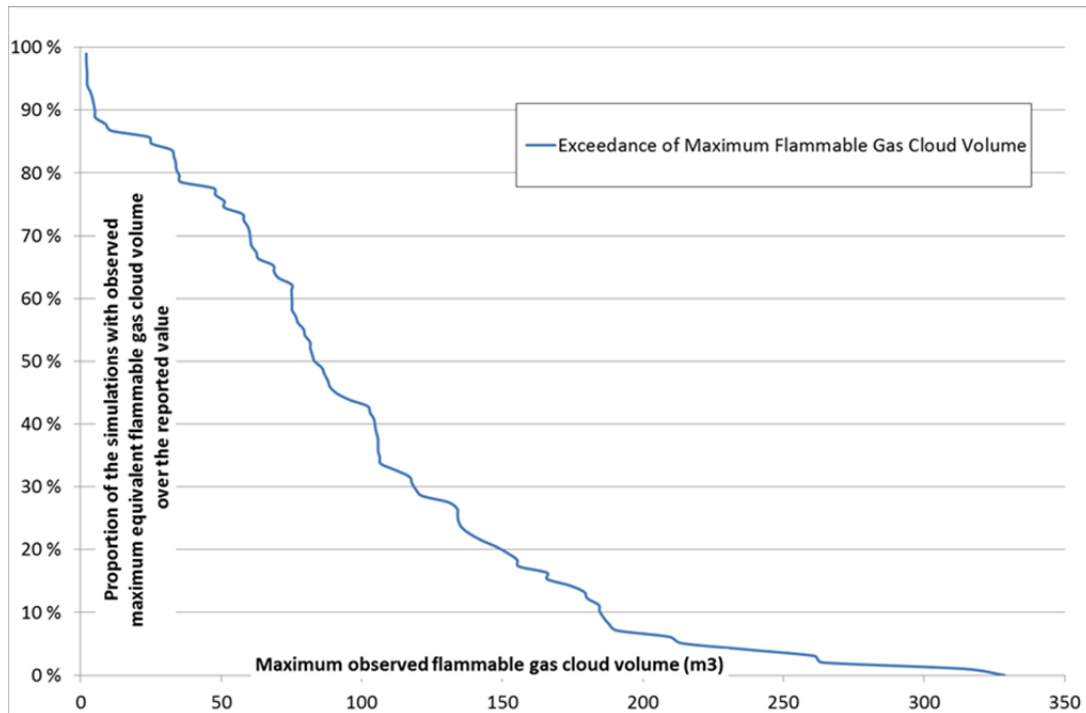


Figure 6: Examples of flammable gas exceedance curve showing the distribution of gas clouds from a CFD-dispersion study.

4. Conclusions

The loads due to explosions in exhaust systems of LNG-fuelled gas engines can be estimated using the CFD-tool FLACS. Realistic conditions (temperature, flow, presence of obstructions (silencer, boiler, catalyser, etc.) and bends, diluents including combustion gases) can be taken into account. Validation tests performed under atmospheric conditions in a 700 mm pipe provided with vents and a bend were performed to verify FLACS simulations for the same geometry.

Similarly the hazards of releases of LNG or natural gas into rooms where gas engines are located can be assessed using the CFD-tool FLACS. This includes an evaluation of the ventilation, gas detector layout and an assessment of explosion loads on the walls and roof of the room and if available on the walls and windows of a neighbouring control room. Reduction of explosion loads by introducing vent openings and panels can be investigated.

References

- Arntzen, B.J. (1998). Modelling of turbulence and combustion for simulation of gas explosions in complex geometries. PhD Thesis, NTNU, Trondheim, Norway (ISBN 82-471-0358-3).
- Directive 2001/80/EC on the limitation of emissions of certain pollutants into the air from large combustion plants, 2001
- International Convention for the Prevention of Pollution from Ships (MARPOL), Annex VI - Prevention of Air Pollution from Ships, May 2005
- National Fire Protection Association, Standard on Explosion Protection by Deflagration Venting, NFPA 68, 2013
- Skjold, T., Pedersen, H.H., Bernard, L., Ichard, M., Middha, P., Narasimhamurthy, V.D., Landvik, T., Lea, T & Pesch, L. (2013). A matter of life and death: validating, qualifying and documenting models for simulating flow-related accident scenarios in the process industry. Fourteenth International Symposium on Loss Prevention and Safety Promotion in the Process Industries, Florence, 12-15 May 2013, published in Chemical Engineering Transactions, 31: 187-192