



LH₂CRAFT - SAFE AND EFFICIENT MARINE TRANSPORTATION OF LIQUID HYDROGEN

CALL: HORIZON-JTI-CLEANH2-2022-2

PROJECT NUMBER: 101111972

QRA REPORTS WITH FIRE/EXPLOSION SIMULATION AND STORAGE TANK INTEGRITY

Work package	Related task(s)	Dissemination level	Document nature	Estimated delivery date
WP8	T8.4	Public	Document, report	31.05.2026

Lead author(s)	Contributing partner(s)	Reviewer(s)
Mujeeb Ahmed Mughadar (UOS) Hayoung Jang (UOS) Injun Yang (UOS) Lin Yang (UOS) Insik Hwang (UOS)	(KSOE) , (HYD)	Vladislav Sidorenkov-Duprez (BV) Antony Jopen (TWI) Xiaofei Cui (TWI)

DOCUMENT HISTORY

Version	Date	Author Editor Reviewer	Description
v0.1	31/07/2025	Hayoung Jang (UOS) Injun Yang (UOS) Lin Yang (UOS) Insik Hwang (UOS)	Initial version
V0.2	29/04/2026	Mujeeb Ahmed Mughadar (UOS) Hayoung Jang (UOS) Injun Yang (UOS) Lin Yang (UOS) Insik Hwang (UOS)	Updated version
V0.2	06/05/2026	Vladislav Sidorenkov-Duprez (BV)	Review comments
	15/05/2026	Antony Jopen (TWI) Xiaofei Cui (TWI)	Review comments
	22/05/2026	Vladislav Sidorenkov-Duprez (BV)	Additional review comments
V1.0	28/05/2026	Mujeeb Ahmed Mughadar (UOS) Hayoung Jang (UOS) Injun Yang (UOS) Lin Yang (UOS) Insik Hwang (UOS)	Final version

TABLE OF CONTENTS

Document History	2
1. Introduction	9
2. Scenario #1 – Fire analysis at vent mast no. 3 & 4	13
2.1. Background.....	13
2.2. Model Validation	13
2.3. Model Scenarios and Conditions	14
2.4. Model Setup	16
2.5. Simulation Results	18
3. Scenario #2 – Explosion analysis at cargo compressor room.....	20
3.1. Background.....	20
3.2. Model Validation	20
3.3. Model Setup	21
3.4. Model Scenarios and Conditions	23
3.5. Simulation Results	23
4. Scenario #3 – Jet Fire analysis at pressure relief valve	27
4.1. Background.....	27
4.2. Model Validation	27
4.3. Model Scenarios and Conditions	27
4.4. Model Setup	28
4.5. Simulation Results	29
5. Scenario #4 – LH2 spill analysis	31
5.1. Background.....	31
5.2. Model Validation	32
5.3. Model Scenarios and Conditions	36
5.4. Model Setup	38
5.4.1. Computational domain.....	38
5.4.2. BOUNDARY CONDITIONS	38
5.5. Simulation Results	39

6.	Scenario #5 – Dispersion analysis at vent mast no.3 & 4.....	46
6.1.	Background.....	46
6.2.	Model Validation	46
6.3.	Model Scenarios and Conditions	50
6.4.	Model Setup	51
6.4.1.	Computational domain.....	51
6.4.2.	Boundary conditions.....	52
6.5.	Simulation Results	53
7.	Scenario #6 – Dispersion analysis at vent mast no.4.....	56
7.1.	Background.....	56
7.2.	Model Validation	56
7.3.	Model Scenarios and Conditions	56
7.4.	Model Setup	57
7.4.1.	Computational domain.....	57
7.4.2.	Boundary conditions.....	58
7.5.	Simulation Results	58
8.	Scenario #7 – Dispersion analysis from Cargo compressor room ...	60
8.1.	Background.....	60
8.2.	Model Validation	60
8.3.	Model Scenarios and Conditions	60
8.4.	Model Setup	62
8.4.1.	Computational domain.....	62
8.4.2.	Boundary conditions.....	63
8.5.	Simulation Results	64
9.	Scenario #8 - Safety zones for cargo operation	65
9.1.	Background.....	65
9.2.	Model Validation	65
9.2.1.	Storage of liquid hydrogen	65
9.2.2.	Potential cargo operation location	66
9.3.	Model Scenarios and Conditions	69

9.3.1. Cargo operation scenario	69
9.3.2. Risk frequency	69
9.3.3. Leakage quantity	71
9.4. Simulation Results	72
9.4.1. Loading operation	72
9.4.2. Unloading operation.....	74
10. Conclusion	75
11. Bibliography	79
12. ANNEX: Demonstrator ship's cargo P&ID	81

LIST OF FIGURES

Figure 1. Illustration of the envisaged LH2 carrier modelled by KSOE.	10
Figure 2. Fire events near the vent mast at no.3 vent mast.	13
Figure 3. Comparison between the experimental flame length data and CFD simulation results.	14
Figure 4. Location of accommodation and vent masts and wind direction around the hull.	14
Figure 5. Computational domain for hydrogen jet-fire analysis from vent mast no. 3.	17
Figure 6. Computational domain for hydrogen jet-fire analysis from vent mast no. 3 and no. 4.	17
Figure 7. Hydrogen Jet-Fire Simulation Results under Varying Wind Speeds: (a) 1 m/s, (b) 6 m/s, and (c) 10 m/s. Flame behaviour and direction are visualised to assess thermal impact on surrounding structures.	18
Figure 8. Hydrogen Jet-Fire Simulation Results under Varying Wind Speeds for Dual Venting: (a) 1 m/s, (b) 6 m/s, and (c) 10 m/s. Flame behaviour and direction are visualised to assess thermal impact on surrounding structures. (d) Time series of flame length for different wind speeds.	19
Figure 9. Explosion event near cargo compressor room.	20
Figure 10. Pressure-time histories for different grid resolutions (5.0 cm, 10.0 cm, and 20.0 cm) in Hydrogen explosion simulation.	21
Figure 11. Simulation setup for hydrogen explosion in cargo compressor room: grid configuration and ventilation openings.	22
Figure 12. Simulation Setup for Hydrogen Explosion in Cargo Compressor Room: Grid Configuration, Ignition Source, and Monitoring Points.	23
Figure 13. Pressure-Time Histories at MP1 (on the centre-facing wall surface) for Different Hydrogen Concentrations (5%, 10%, and 18%) within Cargo Compressor Room.	24
Figure 14. Temperature-Time Histories at MP1 (on the centre-facing wall surface) for Different Hydrogen Concentrations (5%, 10%, and 18%) within Cargo Compressor Room	24
Figure 15. Temperature evolution in a domain containing 18% hydrogen concentration at selected time steps. (a) $t = 0$ s, (b) $t = 0.20$ s, (c) $t = 0.25$ s, (d) $t = 0.27$ s, (e) $t = 0.30$ s, (f) $t = 0.33$ s, (g) $t = 0.53$ s, (h) $t = 0.95$ s)	26
Figure 16. jet-fire event near cargo compressor room.	27
Figure 17. Cargo area schematic showing the leak point and its relation to Vent Masts and the cargo compressor room.	28
Figure 18. Simulation setup for hydrogen jet fire near cargo compressor room: grid configuration and leak point.	29
Figure 19. Total heat flux distribution on the cargo compressor room wall at $t \approx 7$ s for a 2 mm hydrogen leak located 0.5 m from the structure: (a) full-scale legend, (b) capped at 60 kW/m^2	29
Figure 20. Total heat flux distribution on the cargo compressor room wall at $t \approx 6$ s for a 2 mm hydrogen leak located 2 m from the structure.	30
Figure 21. Liquid hydrogen (LH2) spill occurs due to tank damage.	31
Figure 22. (a) Illustration of the boundary used in the based-line simulations; (b) the grid domain used in the base-lined simulations[8].	32
Figure 23. (a) Contours of constant hydrogen concentration deduced from temperature data from towers 2, 5 and 8; Test 6, time = 21.33 s [7] (b) The hydrogen gas cloud concentration contours after 20 s predicted in the referenced study [8] (c) Hydrogen concentration distribution at $t=20$ s from this validation simulation.	34
Figure 24. Comparison of hydrogen concentration results for NASA test 6 with referenced work and our simulation results for tower 2.	35
Figure 25. Hydrogen concentration of monitor point T2 for different gird levels.	36

Figure 26. A real spill accident caused by collision and tank damage [11].....	37
Figure 27. Computational domain for pool spill simulation.....	38
Figure 28. 3D plot for hydrogen gas cloud concentration contour development after LH ₂ pool spill.....	40
Figure 29. Horizontal cross-section contour plots after LH ₂ pool spill at times 40 s, 80 s, 110 s, and 140 s.....	41
Figure 30. Vertical cross-section contour plots after LH ₂ pool spill at times 40 s, 80 s, 110 s, and 140 s. ...	41
Figure 31. Top-down view of liquid hydrogen (LH ₂) pool depth.	42
Figure 32. Side view (X-Z plane) of the LH ₂ pool depth.....	43
Figure 33. Temporal evolution of hydrogen mole fraction at MP4 under varying wind speeds.....	44
Figure 34. Distribution of hydrogen vapour cloud in the X–Z plane at 100 s for different wind speeds.....	44
Figure 35. Temporal evolution of hydrogen mole fraction at MP5 under varying pool radius: 0.2 m, 0.4 m, 0.6 m, 0.8 m (mass flow rate: 92.14 kg/s, 368.56 kg/s, 829.26 kg/s, 1474.23 kg/s)	45
Figure 36. Distribution of hydrogen vapour cloud in the X–Z plane at 90 s for different pool radius: 0.2 m, 0.4 m, 0.6 m, 0.8 m (mass flow rate: 92.14 kg/s, 368.56 kg/s, 829.26 kg/s, 1474.23 kg/s)	45
Figure 37. Dispersion event at vent mast No.3 & 4.	46
Figure 38. Sketch of the experimental setup for the distribution experiments. [12].....	47
Figure 39. Vertical hydrogen concentration profiles along the axis of the release for experiments with all nozzles at different release rates. [12]	48
Figure 40. CFD model performance measures, geometric mean bias, MG, and geometric variance, VG. .	49
Figure 41. Computational domain for vent mast simulation.	51
Figure 42. 2D and 3D Dispersion Profiles of Hydrogen Gas Released from No.3 and No.4 Vent Masts: (A) Horizontal dispersion length, (B) Distance between the cloud and the accommodation structure, and (C) Distance between the cloud and the funnel.....	54
Figure 43. Dispersion event at vent mast No.4.....	56
Figure 44. Computational domain for vent mast simulation.	57
Figure 45. 2D and 3D dispersion profiles of hydrogen gas released from no.4 vent masts: (a) horizontal dispersion length, (b) distance between the cloud and the accommodation structure, and (c) distance between the cloud and the funnel.	59
Figure 46. Dispersion event at cargo compressor room.....	60
Figure 47. Configuration of the cargo compressor room and ventilation fan arrangement on board.	61
Figure 48. Computational domain for hydrogen release simulation.	63
Figure 49. 2D and 3D Dispersion Profiles of Hydrogen Gas Released from ventilation fan.	64
Figure 50. Overview of scenario #8.	65
Figure 51. Potential cargo operation location and wind rose (Port of Rotterdam).....	67
Figure 52. Potential cargo operation location and wind rose (Port of Luderitz).....	68
Figure 53. Fuel leakage event tree.....	71
Figure 54. Simulation results for loading operation.....	72
Figure 55. FN Curve result for loading operation.....	73
Figure 56. Simulation results for unloading operation.....	74
Figure 57. FN Curve result for unloading operation.....	74

LIST OF TABLES

Table 1. Summary of QRA scenarios identified from the HAZID study.	11
Table 2. Numerical simulation cases for scenario no.1, including number of vent openings, wind direction, and wind speed.....	15
Table 3. Case description for scenario 2 including volumetric concentration of hydrogen and ignition location.	23
Table 4. Case description for scenario 3 including leak size, direction and distance to wall.	28
Table 5. Parameters for LH ₂ spill simulation.	33
Table 6. Information of grid levels.	35
Table 7. Parameters for LH ₂ spill simulation.	37
Table 8. Boundary conditions for LH ₂ pool spill simulation.....	39
Table 9. Release scenarios investigated in the experiments. [12]	47
Table 10. Input Parameters for Vent Mast Simulation.....	50
Table 11. Computational domain for core and stretched grid regions.	52
Table 12. Boundary conditions for hydrogen dispersion simulation.....	53
Table 13. Summary of Hydrogen Dispersion Distances under Different Leak and Wind Conditions.	55
Table 14. Computational domain for core and stretched grid regions.....	58
Table 15. Summary of hydrogen dispersion distances under different leak and wind conditions.....	59
Table 16. Input Parameters for hydrogen dispersion simulation.	62
Table 17. Computational domain for core and stretched grid regions.....	63
Table 18. Summary of DSB risk tolerance criteria [14].....	66
Table 19. Population density with weighting factor (Port of Rotterdam).....	67
Table 20. Population density with weighting factor (Port of Luderitz).....	68
Table 21. Frequency of leakage	70
Table 22. Leak frequency per operation.....	70
Table 23. Fuel leaked quantity per operation.....	72
Table 24. Detailed range for simulation results (loading operation).....	73
Table 25. Detailed range for simulation results (unloading operation).....	74

1. INTRODUCTION

Hydrogen is an energy carrier with unique advantages and challenges when used as a fuel in maritime applications. Due to its extremely low molecular weight and high diffusivity, hydrogen disperses rapidly in the atmosphere. However, these same properties, combined with its wide flammability and detonation limits, pose significant safety concerns in confined or semi-confined environments such as ships.

One of the most critical hazards associated with hydrogen is its exceptionally wide flammability range. Hydrogen forms a flammable mixture with air between 4% (LFL, Lower Flammability Limit) and 75% (UFL, Upper Flammability Limit) by volume. Within this range, the released hydrogen-air mixture can sustain combustion and may lead to explosion under appropriate conditions. Furthermore, the detonation limit—the range in which a mixture can detonate rather than merely burn—lies between approximately 18% and 59%, highlighting the severity of risk in specific concentration bands.

Given these characteristics, accurate prediction of hydrogen dispersion behaviour is paramount. Onboard a liquefied Hydrogen (LH₂) carrier, venting events—whether due to normal boil-off gas handling or emergency release—must be carefully assessed to prevent hydrogen from accumulating in flammable concentrations near the accommodation area or ventilation intakes. If hydrogen accumulates within the flammable range (4–74% by volume) and an ignition source is present, it can lead to fire or even explosion, depending on the spatial confinement and energy of ignition.

Therefore, it is essential not only to understand where and how hydrogen disperses, but also to evaluate the fire and explosion hazards that could result from transient gas clouds. Flammable mixtures can form rapidly and persist depending on wind conditions, geometry, and release rate—posing serious risks to life, ship structures, and critical safety systems.

To support safe ship design and operation, computational fluid dynamics (CFD) modelling is employed not only to simulate hydrogen dispersion under representative release scenarios, but also to analyse the consequences of potential fire and explosion events. Dispersion simulations help predict the extent, duration, and concentration of flammable clouds, while fire and explosion simulations provide insight into thermal radiation profiles, flame propagation, and explosion overpressure loads in and around critical areas.

Together, these simulations enable a comprehensive assessment of hydrogen-related risks—from accumulation to ignition and consequence—supporting informed decisions on the placement of vent masts, optimization of escape routes, and design of fire and gas detection systems. The integrated results are essential for establishing safe separation distances, defining shutdown and alarm thresholds, and ensuring regulatory compliance for hydrogen-fuelled ship operations.

For this research, a full-scale LH₂ carrier was carefully selected as the reference vessel, as depicted clearly in Figure 1. The reference vessel has a length overall (LOA) of 299.06 m and a length between perpendiculars (LBP) of 291.00 m. Its breadth is 46.40 m, with a moulded depth of 26.50 m and a trunk depth of 35.50 m. These principal dimensions support the arrangement of hydrogen cargo systems and associated onboard infrastructure for the safe and efficient transport of liquid hydrogen (LH₂).



Figure 1. Illustration of the envisaged LH2 carrier modelled by KSOE.

The vessel under investigation is classified as a Gas Carrier of Type 2G as per IGC Code, a specialised categorisation explicitly indicating its suitability and enhanced safety features tailored for the carriage of liquefied gases. Specifically engineered to store and transport LH₂, this vessel utilises advanced membrane-type containment tanks, renowned for their superior insulation capabilities and structural integrity under extreme cryogenic conditions. The design specification mandates that these tanks maintain the LH₂ cargo at an exceptionally low temperature of approximately -253°C, reflecting hydrogen's boiling point under atmospheric pressure conditions.

In 2024, a HAZID (Hazard Identification) study was conducted in the LH2CRAFT project as part of the safety assessment for the LH₂ carrier. The findings are documented in Deliverable 8.2 'HAZID and HAZOP Report'. The sessions took place over three days in a hybrid workshop format held in Trieste, Italy, from June 11th to 13th. As an outcome of the study, eight QRA (Quantitative Risk Assessment) scenarios were identified. Among them, three scenarios required fire and explosion simulations, three focused on hydrogen dispersion analysis, one addressed LH₂ spill behaviour, and one was aimed at defining safety zones around critical release points. As described in Table 1, these scenarios provided the basis for targeted CFD modelling to support risk evaluation and inform design decisions.

The fire, explosion, and dispersion modelling of hydrogen released from the vent mast was conducted using FLACS (FLame ACceleration Simulator), developed by Gexcon (Norway), Version FLACS-CFD v24.1. FLACS is a specialised CFD software that has been extensively developed and widely applied in industrial safety engineering, particularly for analysing gas-related hazards. While it is well known for its accuracy in gas dispersion simulations, FLACS is also robust in modelling flammable cloud ignition, fire dynamics, and explosion overpressure in complex geometries—making it highly suitable for hydrogen safety studies.

Due to hydrogen's high diffusivity and broad flammability range, as well as its high reactivity and low ignition energy, FLACS enables detailed evaluation of both temporal and spatial variation in gas concentration and the consequential effects of ignition, such as flame acceleration and pressure wave propagation. This makes it an effective tool for simulating vent mast release scenarios involving potential fire or explosion events.

The governing equations used in FLACS are based on the Reynolds-Averaged Navier-Stokes (RANS) framework. The standard k-ε turbulence model is applied to represent turbulent transport, and the Finite Volume Method (FVM) is used for spatial discretisation. Combustion and explosion modelling are

supported by in-built sub-models that account for flame propagation, energy release, and confinement effects.

Table 1. Summary of QRA scenarios identified from the HAZID study.

Scenario no.	Identified QRA scenario from HAZID study	Specific description
Scenario #1	Evaluate the effects of fire and explosion events on LH ₂ tank during FERA and/or QRA Study	When a jet fire occurs at No.3 vent mast, and the wind is blowing from bow to stern
Scenario #2	Evaluate the effects of fire and explosion events on LH ₂ tank	To identify the explosion pressure of LH ₂ within cargo gear locker, ensuring its structural integrity and safety
Scenario #3	Evaluate the impact of fire and explosion events in and through the pressure relief system	When a jet fire occurs in the No.3 pressure relief system
Scenario #4	Develop a dedicated study and dispersion analysis considering accidental scenarios such as grounding, collision, tank damage	When an LH ₂ spill occurs due to tank damage
Scenario #5	Develop a dedicated dispersion study for hydrogen release from the vent mast to avoid that potential flammable / explosive / asphyxiant atmosphere can impair the escape routes and a safe ship abandonment	To identify hazardous zones and determine the optimal position for the vent mast, minimising LH ₂ concentration around the accommodation area. (Determine the release range when both No.3 & 4 vent masts release simultaneously)
Scenario #6	Ensure that the location of ventilation system inlets serving all ship's spaces is checked in the next steps of the risk analysis to verify that the propagation of flammable / explosive / asphyxiant atmosphere within the accommodation, service and machinery spaces is excluded	Confirm whether the hydrogen released from No.3 or No.4 vent mast forms a sufficient flammable hydrogen cloud in the direction of the residential area
Scenario #7	Consider taking into due consideration any possible release of LH ₂ on the weather deck	Identify the concentration of hydrogen emitted from the ventilation fan, originating from the cargo compressor room due to leakage
Scenario #8	Define security and safety zones for cargo operation (loading and unloading)	Establish designated safety and security areas during cargo loading and unloading operations

In parallel, to support the defining of safety zones, this simulation was conducted using DNV-Phast 9.0, a widely used consequence analysis tool. DNV-Phast enables efficient estimation of flammable gas cloud dispersion extents, thermal radiation contours, and explosion damage radii under conservative assumptions. While not as detailed in geometry as FLACS, DNV-Phast provides rapid screening of credible leak scenarios and was instrumental in quantifying exclusion zones around vessel locations based on regulatory thresholds and escalation criteria.

Together, the use of both FLACS and DNV-Phast allowed for a comprehensive risk-based approach to modelling hydrogen behaviour—from dispersion and ignition to consequence assessment and safety zone definition—supporting robust design and operational safety of the LH₂ carrier.

2. SCENARIO #1 – FIRE ANALYSIS AT VENT MAST NO. 3 & 4

2.1.BACKGROUND

This study aims to evaluate the potential impact of a jet fire incident originating from Vent Mast No.3 under specific wind conditions, particularly when the wind is blowing from bow to stern (See Figure 2). Given the close proximity between Vent Mast No.3 and Vent Mast No.4, it is important to assess whether flame impingement or elevated thermal radiation could affect the surrounding area, including adjacent equipment.

The primary focus of this analysis is to determine:

- The extent and direction of flame spread under the given wind conditions.
- The temperature distribution in the vicinity of the fire, especially around Vent Mast No.4.

This evaluation will help identify potential interference between the vent masts during a fire scenario and support decisions related to fire safety design and risk mitigation.

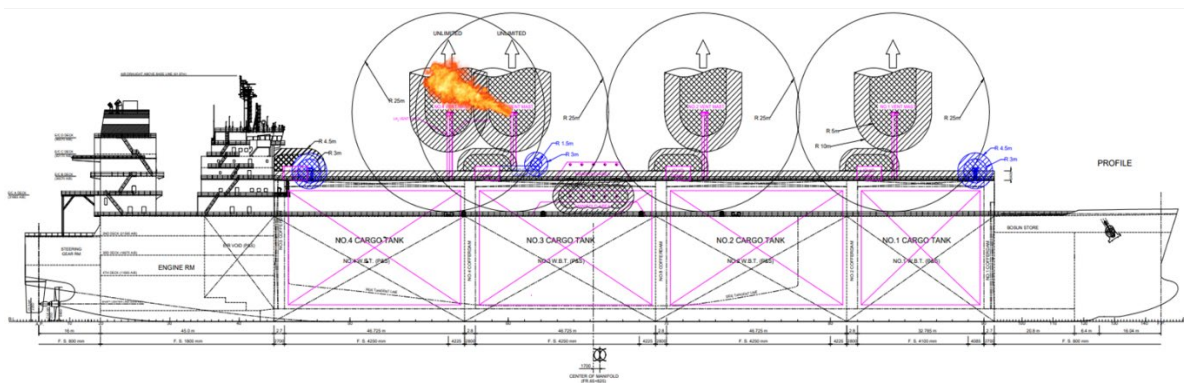


Figure 2. Fire events near the vent mast at no.3 vent mast.

2.2.MODEL VALIDATION

To ensure the credibility of the CFD simulations conducted using the FLACS software, a validation exercise was carried out based on experimental data presented [\[1\]](#).

The referenced study investigates the hazards associated with hydrogen jet diffusion flames generated by high-pressure gas releases through circular nozzles of varying diameters (0.1 to 4 mm) and release pressures (0.01 to 40 MPa gauge). Key parameters such as flame length, flame width, and blow-off limits were experimentally determined and correlated with nozzle geometry and operating conditions.

Figure 3 presents a comparison between the experimental flame length data and the CFD simulation results obtained using FLACS, for a nozzle diameter of 0.004 m [\[1\]](#). The flame length is plotted against release pressure. The experimental data are shown for nozzle diameters of 0.002 m and 0.004 m, while the simulation results correspond to the 0.004 m case.

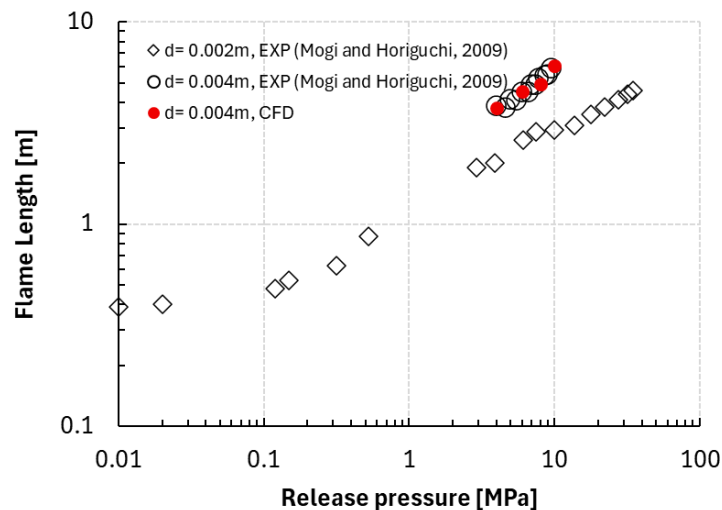


Figure 3. Comparison between the experimental flame length data and CFD simulation results.

The FLACS model demonstrates a good level of agreement with the experimental measurements, particularly in capturing the trend of increasing flame length with higher release pressures. This supports the validity of the simulation approach for modelling hydrogen jet fires under high-pressure conditions. Details of the numerical set-up used in the simulations will be provided in the following chapter.

2.3.MODEL SCENARIOS AND CONDITIONS

To investigate the dispersion behaviour of hydrogen released from vent masts on a vessel, a series of CFD simulations were conducted based on a representative ship configuration. The computational domain includes key areas such as the accommodation block and two vent masts (No. 3 and No. 4), which are considered as potential hydrogen release points (See Figure 4).

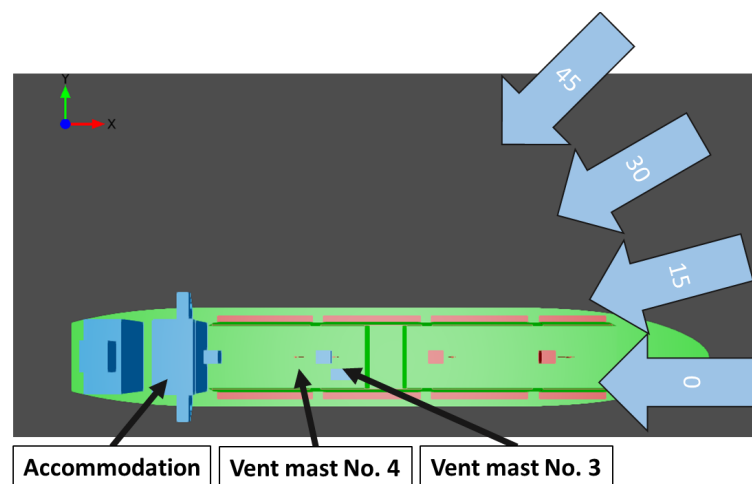


Figure 4. Location of accommodation and vent masts and wind direction around the hull.

Hydrogen is assumed to be released vertically from the vent masts when the pressure relief valve (PRV) is activated. The release is modelled as a continuous jet fire, representing a worst-case scenario in which overpressure leads to PRV opening and subsequent ignition of the released hydrogen. This simulation is

not intended for dispersion analysis but rather focuses on the thermal and safety implications of jet-fire events.

The simulation cases were selected based on conditions expected to produce the most severe consequences, including venting configurations and wind alignment that could direct the flame toward critical areas such as the accommodation block. Buoyancy effects due to the low molecular weight of hydrogen are included, and the flame behaviour is modelled using appropriate combustion and radiation models to capture the thermal impact on surrounding structures.

The initial set of cases, summarised in Table 2, considers both single and dual venting configurations under a 0° wind direction with a wind speed of 1, 6, and 10 m/s. These conditions were identified as critical through preliminary screening and are used to evaluate the potential for high thermal loads and flame impingement.

Table 2. Numerical simulation cases for scenario no.1, including number of vent openings, wind direction, and wind speed.

Case	Number of vent openings	Wind direction [deg]	Wind speed [m/s]
S#1-S-1	Single at No.3	0	1
S#1-S-2	Single at No.3	0	6
S#1-S-3	Single at No.3	0	10
S#1-D-1	Dual at No.3, 4	0	1
S#1-D-2	Dual at No.3, 4	0	6
S#1-D-3	Dual at No.3, 4	0	10

To calculate the release rate Q , the minimum required discharge flow rate of hydrogen gas from a membrane tank is determined in accordance with the International Code of the Construction and Equipment of Ships Carrying Liquefied Gases in Bulk (IGC Code). The release rate Q is then calculated using the following equation:

$$Q = FGA^{0.82} \quad (1)$$

where F represents the fire exposure factor, G is the gas factor, and A is the external surface area of the tank. For membrane tanks, the fire exposure factor F is 0.1.

The gas factor G is defined as:

$$G = \frac{12.4}{LD} \sqrt{\frac{ZT}{M}} \quad (2)$$

where T is the absolute temperature at relieving conditions, representing the thermodynamic state of hydrogen during the release. It is specified as 53.15 K, L is the latent heat of vaporisation at relieving conditions, representing the amount of energy required to convert a unit mass of hydrogen from liquid to vapor without a temperature change. In this context, it is specified as 445.65 kJ/kg., and M is the molecular mass of hydrogen, which affects its velocity and energy characteristics during vaporisation. The value used is 2.016 kg/kmol., Z is the compressibility factor, which corrects for deviations from ideal gas behaviour. The value used is 1.

The term D is a coefficient related to the ratio of specific heats, given by:

$$D = \sqrt{k \left(\frac{2}{k+1} \right)^{\frac{k+1}{k-1}}} \quad (3)$$

where k is the ratio of specific heats at relieving conditions, with a value of 1.383.
The external surface area of the tank A is 7137.15 m².

Finally, the mass flow rate of hydrogen gas from the membrane tank is determined as:

$$\dot{m} = \rho Q \quad (4)$$

where ρ represents the density of hydrogen gas at relieving conditions. This equation ensures that the pressure relief system is appropriately sized to handle emergency venting scenarios, maintaining the safety of the liquefied hydrogen containment system.

In this case, the release rate Q has been calculated as 30.27 m³/s. To convert this into a mass flow rate, the density of hydrogen gas must be known. However, hydrogen gas density varies significantly depending on temperature.

Assuming standard conditions (273.15 K), the density of hydrogen is approximately 0.08988 kg/m³, resulting in a mass flow rate of:

$$\dot{m} = 0.08988 \times 30.27 = 2.72 \text{ kg/s} \quad (5)$$

2.4.MODEL SETUP

The numerical simulations were conducted using the FLACS software to evaluate hydrogen jet-fire behaviour under a representative jet-fire venting scenario, where hydrogen is released through pressure relief valves (PRVs) and ignited shortly after. This scenario reflects a worst-case condition involving sustained flame formation following overpressure events.

Two types of computational domains were defined: one for single venting (from Vent Mast No.3) and another for dual venting (from both Vent Mast No.3 and No.4). A structured mesh was applied, with finer grids concentrated around the vent openings and in regions where the hydrogen flame was expected to develop. The mesh gradually coarsened away from these critical areas to balance accuracy and computational efficiency.

For the single venting case, the total number of grids was 370,548. The minimum grid size near the vent was 0.12 m, and the maximum grid size reached 2 m. The domain size in the x-direction was set to 55 meters, ensuring sufficient upstream distance for wind development and adequate downstream space to fully capture the extent of the hydrogen flame. This configuration allows for accurate representation of both the inflow boundary layer and the potential flame trajectory in the wake of the vent mast. The other domain dimensions were 40 m (y-direction) and 57 m (z-direction), with the z-direction starting from the open deck level. Vent Mast No.3 was positioned at the centreline in the y-direction.

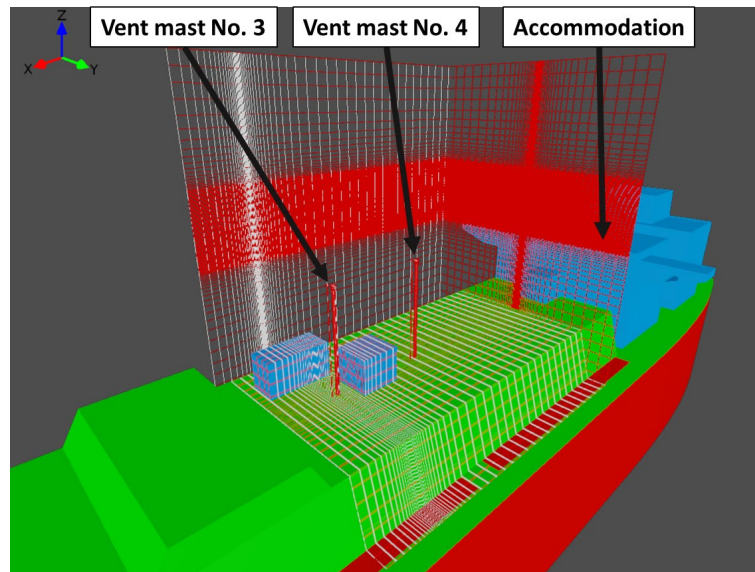


Figure 5. Computational domain for hydrogen jet-fire analysis from vent mast no. 3.

For the dual venting case, the total number of control volumes was 266,175 cells. The minimum grid size remained at 0.12 m, while the maximum grid size increased to approximately 5 m. All other domain settings were consistent with the single venting configuration (see Figure 6).

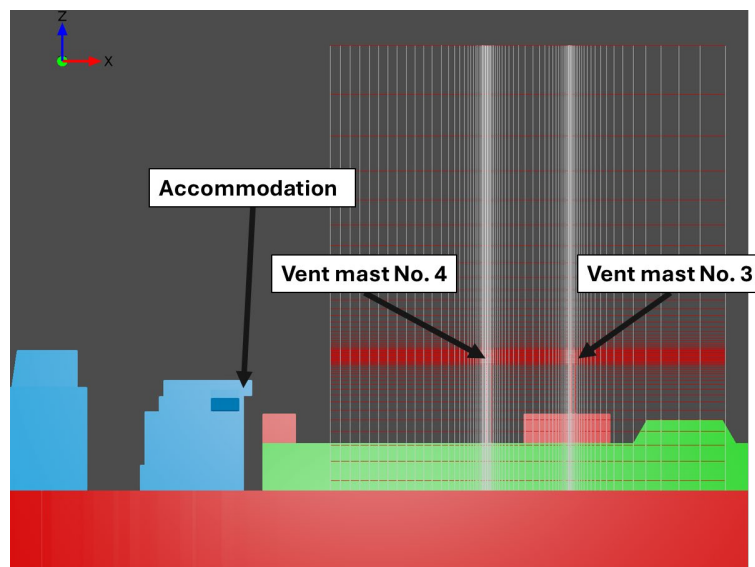


Figure 6. Computational domain for hydrogen jet-fire analysis from vent mast no. 3 and no. 4.

Hydrogen was released at a mass flow rate of 2.72 kg/s, and ignition was assumed to occur 1 second after the start of venting. This ignition timing was selected to simulate a delayed ignition scenario that leads to a sustained jet fire.

From the simulation results, the flame size and direction could be clearly identified, along with their potential impact on surrounding structures. In particular, the analysis enabled the assessment of thermal exposure to the accommodation block, providing critical insights into the vulnerability of habitable areas under jet-fire conditions.

2.5.SIMULATION RESULTS

Hydrogen jet-fire simulations were performed under varying wind conditions to assess flame behaviour and its potential impact on surrounding structures. The cases analysed include single venting from Vent Mast No.3 with wind speeds of 1 m/s, 6 m/s, and 10 m/s, corresponding to Cases S#1-S-1, S#1-S-2, and S#1-S-3, respectively.

As shown in Figure 7, the flame characteristics are significantly influenced by wind speed. At low wind speed (1 m/s), the flame rises vertically with minimal horizontal displacement, resulting in concentrated thermal exposure near the vent mast. As wind speed increases to 6 m/s and 10 m/s, the flame is increasingly deflected downstream, extending horizontally and interacting with the ship's superstructure.

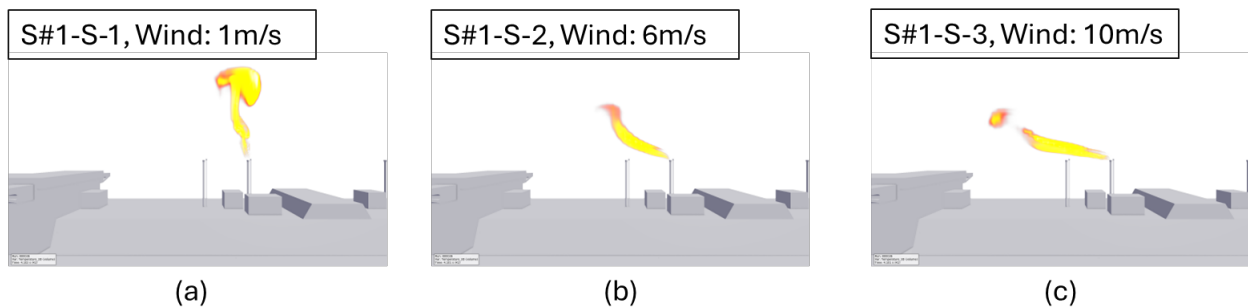


Figure 7. Hydrogen Jet-Fire Simulation Results under Varying Wind Speeds: (a) 1 m/s, (b) 6 m/s, and (c) 10 m/s. Flame behaviour and direction are visualised to assess thermal impact on surrounding structures.

In the highest wind condition (10 m/s), the flame exhibits pronounced bending and elongation. However, in the case of single vent opening, the flame does not reach the accommodation block, indicating that thermal exposure to habitable areas remains limited under this configuration. This result highlights the importance of venting configuration in determining flame reach and potential hazard zones.

Following the single venting analysis, additional simulations were conducted for the dual venting configuration, where hydrogen is released simultaneously from Vent Mast No.3 and No.4. The objective was to examine how the presence of multiple release points influences flame behaviour under varying wind conditions.

Figure 8 illustrates the flame development for wind speeds of 1 m/s, 6 m/s, and 10 m/s. At 1 m/s, the flames from both vents initially behave independently. Notably, flame formation is observed only at Vent Mast No.3 immediately after ignition. However, approximately 12 seconds later, flame propagation is also observed at Vent Mast No.4. This delayed ignition is attributed to thermal radiation and heat transfer from the initial flame, which raises the temperature around Vent Mast No.4 sufficiently to support jet-fire development. The result demonstrates that even under low wind conditions, thermal interaction between venting points can lead to secondary flame formation.

As wind speed increases to 6 m/s, the flames begin to merge and are deflected downstream, forming a broader and more coherent flame structure. Under the highest wind condition (10 m/s), the combined flame exhibits significant horizontal extension and directional bending, with increased thermal footprint across the deck area.

Compared to the single venting case, the dual venting configuration results in a wider flame base and greater overall flame length, especially under strong wind conditions. While the flame in the single venting

scenario remained confined and did not reach the accommodation block, the dual venting case shows a more expansive flame trajectory. However, even in the most severe condition (10 m/s), the flame tip does not directly impinge on the accommodation block, suggesting that the habitable zone remains outside the immediate thermal hazard range.

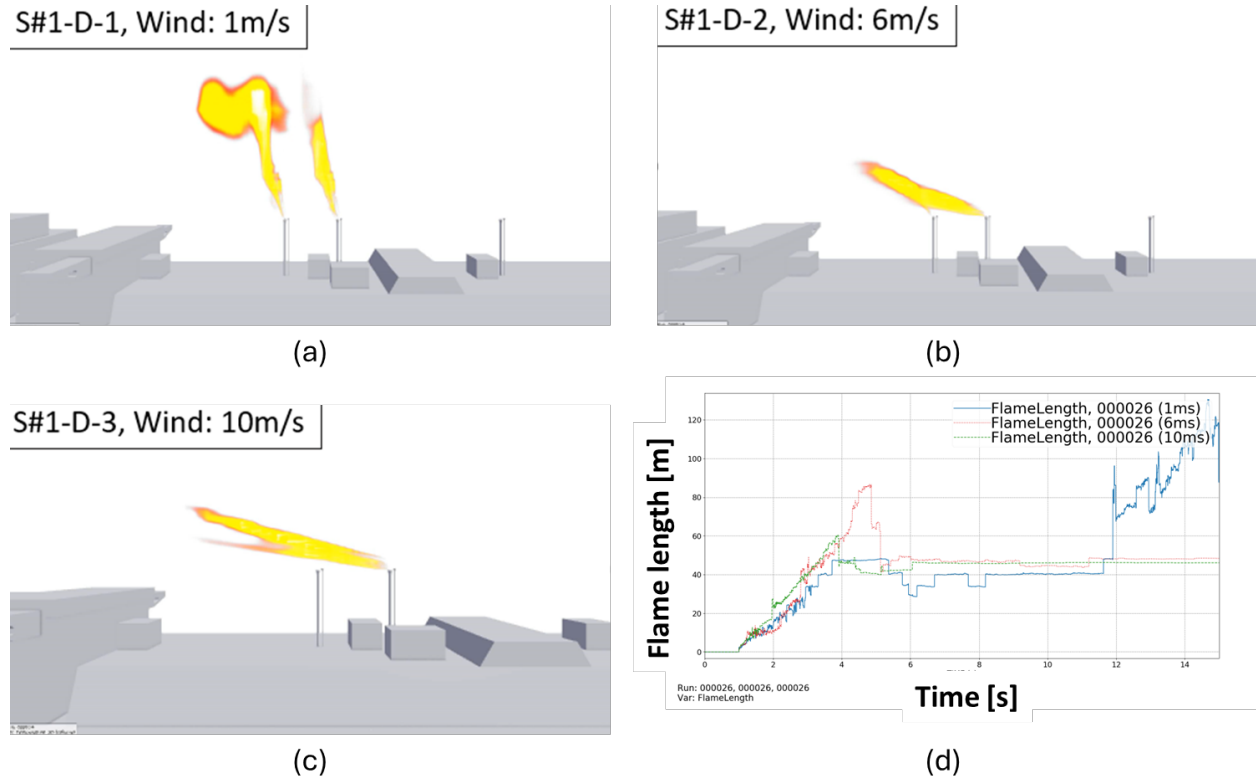


Figure 8. Hydrogen Jet-Fire Simulation Results under Varying Wind Speeds for Dual Venting: (a) 1 m/s, (b) 6 m/s, and (c) 10 m/s. Flame behaviour and direction are visualised to assess thermal impact on surrounding structures. (d) Time series of flame length for different wind speeds.

Figure 8(d) presents the time trace of flame length for each wind condition, highlighting the dynamic response of the flame to wind-induced momentum and turbulence. The dual venting setup consistently produces longer flames than the single venting case, emphasising the importance of vent configuration in fire hazard assessments.

These findings underscore the need for careful consideration of vent layout, ignition timing, and wind exposure in the design of hydrogen safety systems, particularly in scenarios involving simultaneous PRV activation and potential flame interaction.

3. SCENARIO #2 – EXPLOSION ANALYSIS AT CARGO COMPRESSOR ROOM

3.1.BACKGROUND

This scenario investigates the consequences of a fire and explosion event originating within the cargo compressor room, with a focus on its potential impact on the open deck area (see Figure 9). The explosion is assumed to have initiated inside the cargo compressor room, and the top side of the compartment is configured with an opening to represent a ventilation inlet and outlet. Those openings allow for the release of explosion pressure and serve as a location for pressure measurement.

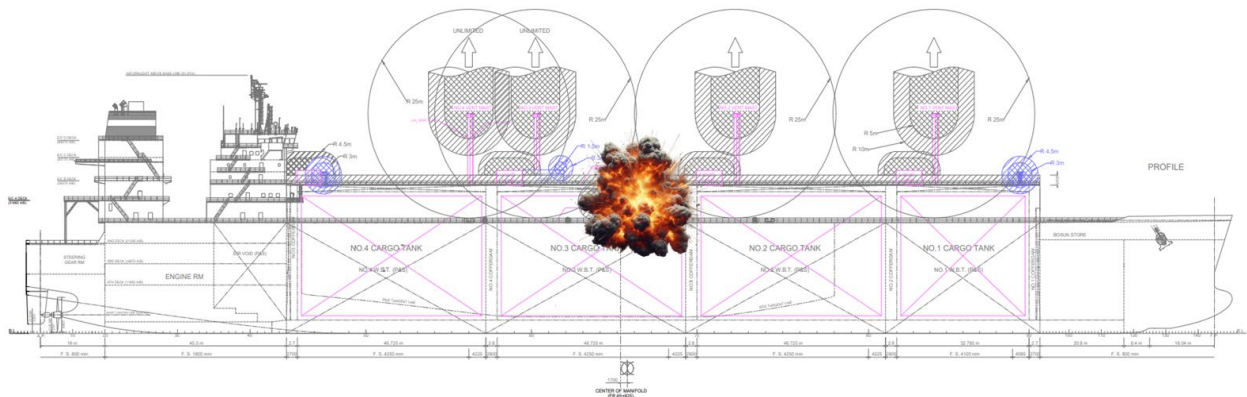


Figure 9. Explosion event near cargo compressor room.

The purpose of this study is to evaluate the effects of fire and explosion events on the surrounding structures, particularly the LH₂ tank, as part of a Fire and Explosion Risk Assessment (FERA) and/or Quantitative Risk Assessment (QRA). The analysis aims to identify the peak explosion pressure generated within the cargo compressor room and assess whether the resulting pressure wave could compromise the structural integrity of nearby components.

In particular, the spatial propagation of explosion pressure from the cargo compressor room toward the open deck area is examined to determine the potential for secondary accidents. This includes verifying whether the pressure levels are sufficient to affect the LH₂ tank or other critical systems located on the deck, thereby informing safety design and mitigation strategies.

3.2.MODEL VALIDATION

To ensure the reliability of the simulation results for the explosion scenario within the cargo compressor room, model validation was conducted by referencing experimental data and prior numerical studies.

Experimental results were used as a benchmark [2]. Their study investigated vented explosion overpressures in a room-sized enclosure using hydrogen–air mixtures and compared them with propane and methane mixtures.

The experiments were performed in a 64 m³ chamber (4.6 × 4.6 × 3.0 m) with vent areas of 2.7 or 5.4 m² and various ignition locations. The study provided detailed pressure-time histories and flame arrival data, which were used to develop and validate a three-dimensional gas dynamic model using Large Eddy

Simulation (LES) in OpenFOAM V2412. These results offer a robust reference for evaluating hydrogen combustion behaviour in confined and vented environments.

Further validation was supported by previous work that involved CFD simulations of vented hydrogen deflagration in a medium-scale enclosure [3]. Their study compared transient pressure and flame speed predictions against experimental data from FM Global, also conducted in a 64 m³ enclosure filled with 18% v/v hydrogen–air mixture. The CFD results showed satisfactory agreement with both empirical correlations and NFPA 68 vent sizing standards, reinforcing the credibility of CFD as a tool for explosion safety assessment.

In the present study, simulations were conducted using the FLACS software. A grid sensitivity analysis was performed to determine the optimal mesh resolution for accurate pressure prediction (see Figure 10). It was found that a grid size of 5 cm produced results most consistent with experimental data, particularly in terms of peak overpressure and flame propagation. This finding aligns with previous validation studies and confirms that the selected mesh resolution is appropriate for capturing key explosion dynamics.

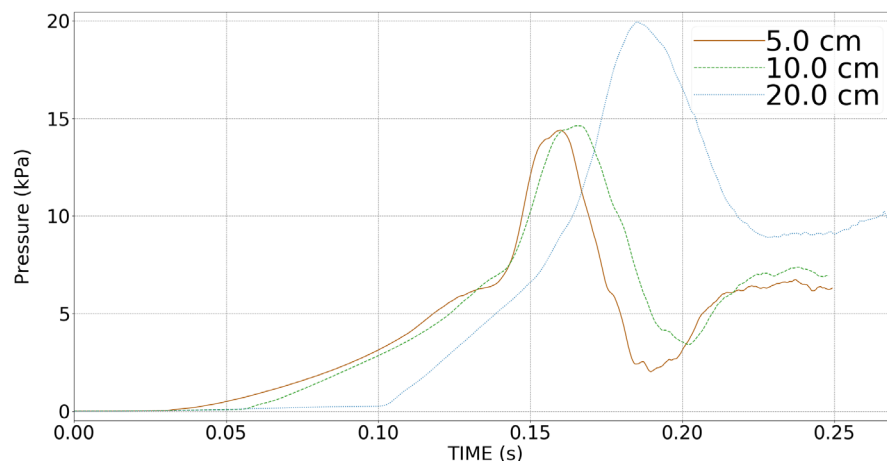


Figure 10. Pressure-time histories for different grid resolutions (5.0 cm, 10.0 cm, and 20.0 cm) in Hydrogen explosion simulation.

Importantly, the simulation results obtained using FLACS showed no significant deviation from the trends and magnitudes reported in the referenced experimental and numerical studies. This consistency reinforces the credibility of the current modelling approach and supports its application in explosion safety assessments for onboard hydrogen systems.

3.3.MODEL SETUP

The simulation was carried out using the FLACS software to model a hydrogen explosion scenario within the cargo compressor room (see Figure 11). The cargo compressor room is assumed to be empty without considering the presence of different equipment and pipelines. A constant minimum grid size of 5.0 cm was applied to fully cover the cargo compressor room, ensuring sufficient resolution for capturing flame propagation and pressure dynamics.

This approach is consistent with previous studies, which demonstrated that finer mesh resolution improves the accuracy of explosion pressure predictions [3].

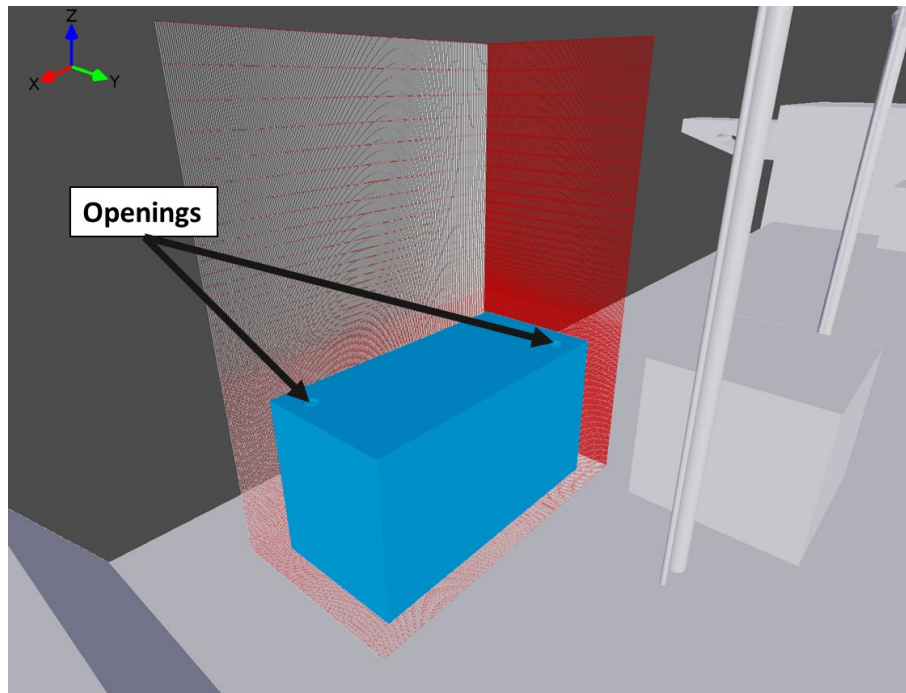


Figure 11. Simulation setup for hydrogen explosion in cargo compressor room: grid configuration and ventilation openings.

Ignition was assumed to occur at time = 0 seconds, with the centre of the cargo compressor room selected as the ignition location. This configuration represents a typical internal deflagration scenario and allows for consistent comparison with validated experimental data.

The openings defined in the model represent the locations of mechanical ventilation fans and natural opening, which serve as vent paths for explosion pressure relief. These openings are critical in determining the pressure release characteristics and external flame development.

To ensure that the vertical propagation of the explosion and associated pressure wave was adequately captured, the domain size in the height z-direction was extended to 2.5 times the height of the cargo compressor room. As a result, the final domain dimensions were set to 12 m in the x-direction, 6.5 m in the y-direction, and 16.5 m in the z-direction. This extended domain allows for accurate assessment of pressure effects on the open deck area and surrounding structures.

In order to evaluate the dynamic response of the system, monitoring points were placed at the openings and on the centre-facing wall surfaces (see Figure 12). These locations were selected to measure pressure and temperature during the explosion event, enabling detailed analysis of the thermal and mechanical loads acting on critical areas of the structure.

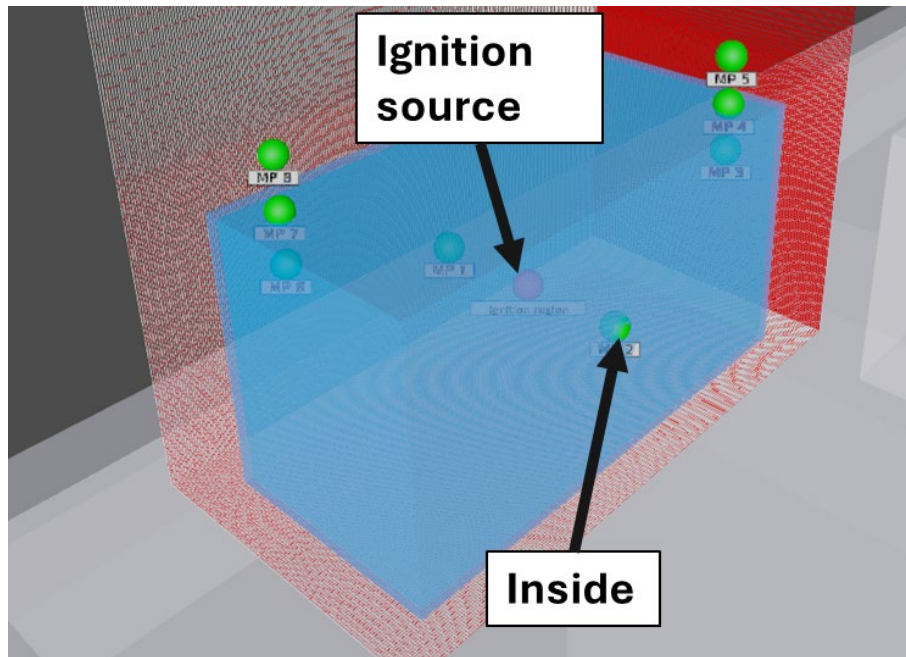


Figure 12. Simulation Setup for Hydrogen Explosion in Cargo Compressor Room: Grid Configuration, Ignition Source, and Monitoring Points.

3.4.MODEL SCENARIOS AND CONDITIONS

The simulation scenarios were developed to assess the effects of hydrogen explosions within the cargo compressor room under varying hydrogen concentrations. Three cases were considered, each with a different volumetric concentration of hydrogen: 5%, 10%, and 18%, representing a range of possible gas accumulation conditions (see Table 3). In all cases, ignition was assumed to occur at the centre of the cargo compressor room, reflecting a representative internal ignition scenario.

Table 3. Case description for scenario 2 including volumetric concentration of hydrogen and ignition location.

Case	Volumetric concentration of hydrogen [%]	Ignition location
S#2-C-1	5	Centre
S#2-C-2	10	Centre
S#2-C-3	18	Centre

3.5.SIMULATION RESULTS

The simulation examined the pressure development resulting from hydrogen explosions within the cargo compressor room, with measurements taken at MP1, located at the centre of the room. The analysis focused on three hydrogen–air mixture concentrations: 5%, 10%, and 18% by volume.

The results, illustrated in Figure 13, show a clear correlation between hydrogen concentration and explosion severity. As the volumetric percentage of hydrogen increases, both the peak pressure and the rate of pressure rise increase significantly. For the 5% mixture, the explosion produced a relatively low peak pressure of approximately 0.005 bar, indicating a mild deflagration. In contrast, the 10% mixture resulted

in a peak pressure of around 0.5 bar, while the 18% mixture generated a much more intense explosion, reaching a peak pressure of approximately 4.5 bar.

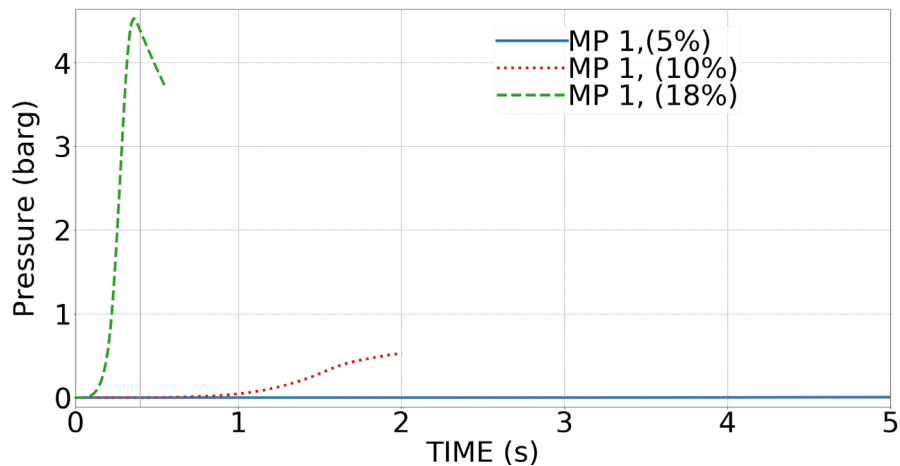


Figure 13. Pressure-Time Histories at MP1 (on the centre-facing wall surface) for Different Hydrogen Concentrations (5%, 10%, and 18%) within Cargo Compressor Room.

The time required to reach peak pressure also decreased with increasing hydrogen concentration. The 18% case exhibited a rapid pressure escalation immediately following ignition, suggesting highly reactive combustion dynamics. This behaviour highlights the increased risk associated with higher hydrogen concentrations, particularly in confined spaces.

The pressure-time curves further reveal that the initial pressure rise becomes steeper as the concentration increases, indicating faster flame propagation and more energetic combustion. These findings underscore the importance of controlling hydrogen accumulation via gas detection with automatic shutdown, ensuring adequate ventilation as well as increasing the walls and ceiling openings in enclosed environments to mitigate explosion hazards.

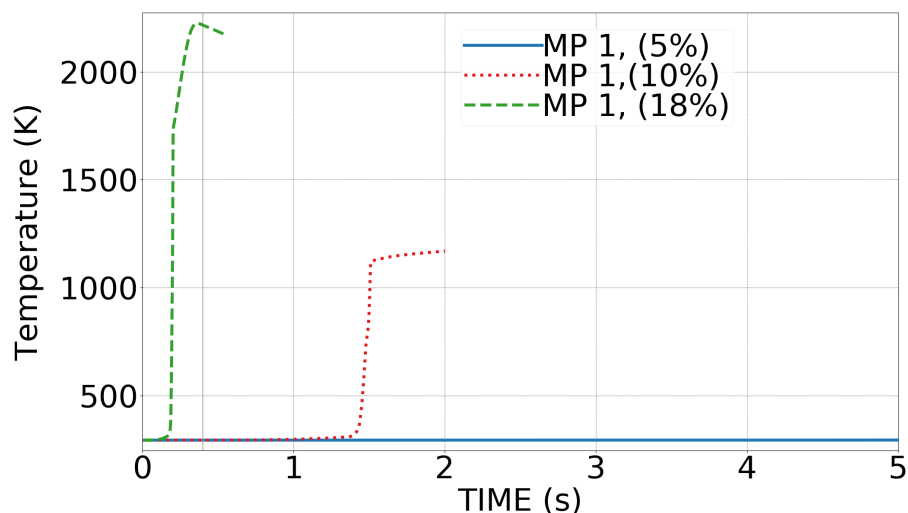


Figure 14. Temperature-Time Histories at MP1 (on the centre-facing wall surface) for Different Hydrogen Concentrations (5%, 10%, and 18%) within Cargo Compressor Room

Figure 14 presents the temporal progression of temperature for MP 1 at three different concentrations: 5%, 10%, and 18%. The simulation results reveal a strong dependence of thermal behaviour on concentration.

At a concentration of 5% (blue solid line), the temperature remains nearly constant at approximately 294 K, indicating the absence of significant exothermic reactions. In contrast, the 10% concentration case (red dotted line) exhibits a delayed temperature rise, reaching a peak of approximately 1169 K slightly after 1 second, suggesting the onset of moderate thermal activity. The most distinct behaviour is observed at 18% concentration (green dashed line), where the temperature rapidly escalates to a peak of 2224 K almost immediately after ignition, indicating a highly reactive or potentially explosive regime.

These findings underscore the critical influence of concentration on the thermal response of MP 1, with higher concentrations leading to significantly more intense and rapid temperature rises.

Figure 15 illustrates the progression of temperature distribution following the ignition of a hydrogen-air mixture with 18% hydrogen concentration. Initially, no thermal activity is observed (a), but localised heating begins at (b). A significant temperature rise and flame propagation are evident from (c) to (h), with the combustion front expanding and intensifying over time. The sequence highlights the dynamic nature of hydrogen combustion and the rapid development of high-temperature regions within a few seconds.

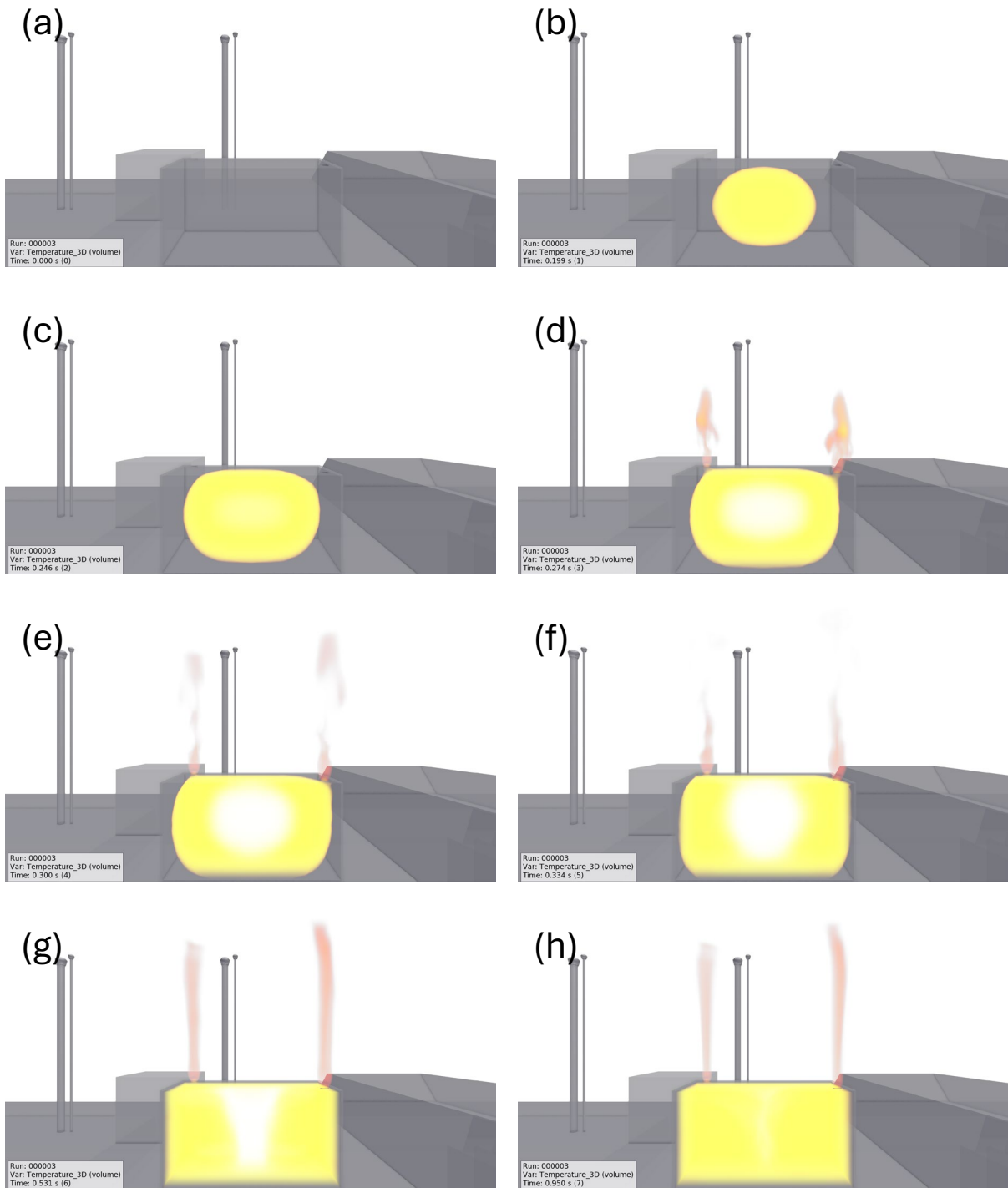


Figure 15. Temperature evolution in a domain containing 18% hydrogen concentration at selected time steps.

(a) $t = 0$ s, (b) $t = 0.20$ s, (c) $t = 0.25$ s, (d) $t = 0.27$ s, (e) $t = 0.30$ s, (f) $t = 0.33$ s, (g) $t = 0.53$ s, (h) $t = 0.95$ s)

4. SCENARIO #3 – JET FIRE ANALYSIS AT PRESSURE RELIEF VALVE

4.1.BACKGROUND

In liquefied hydrogen (LH₂) carriers, pressure relief valves (PRVs) are essential components for maintaining system safety under overpressure conditions. However, mechanical failure or improper sealing in the PRV or adjacent piping may lead to unintended hydrogen leakage. When such a release occurs in the presence of an ignition source, it can result in a jet fire, characterised by high-velocity, high-temperature flame propagation (see Figure 16).

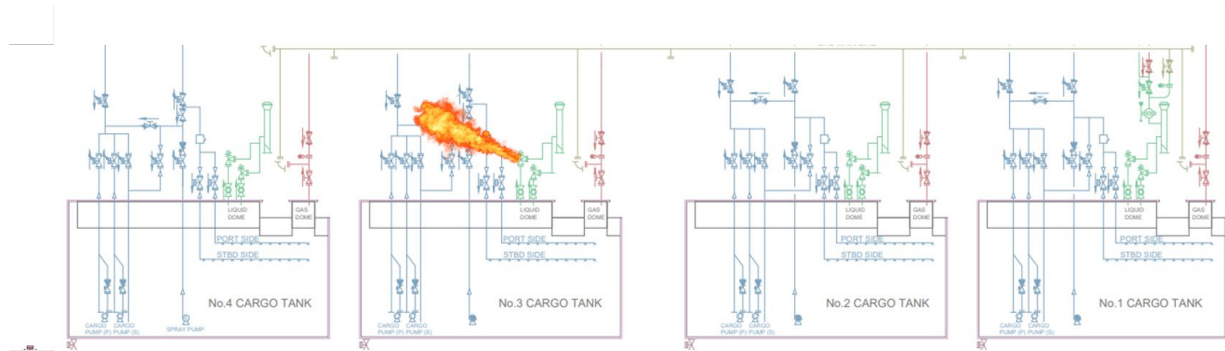


Figure 16. jet-fire event near cargo compressor room.

Scenario #3 addresses this critical failure mode by analysing the thermal consequences of a jet fire initiated near the No.3 PRV. The scenario assumes that hydrogen leakage originates either directly from the PRV or from the connected pipeline, and that ignition occurs shortly thereafter. The resulting jet fire poses a significant threat to nearby shipboard structures, particularly the cargo compressor room and the adjacent open deck area.

The primary objective of this study is to evaluate the thermal impact of the jet fire on these vulnerable regions. Through computational simulation, the temperature distribution and heat flux levels are assessed to determine the extent of thermal exposure and potential structural damage. This analysis provides essential insight into the fire resilience of LH₂ carrier designs and supports the development of effective risk mitigation strategies for cryogenic hydrogen transport systems.

4.2.MODEL VALIDATION

Model validation for Scenario #3 is omitted, as it follows the same methodology and setup as Scenario #1. For validation details, refer to the 'Model Validation' section of Scenario #1.

4.3.MODEL SCENARIOS AND CONDITIONS

Scenario #3 includes two simulation cases designed to assess the thermal impact of jet fire events resulting from hydrogen leakage near the pressure relief system. As summarised in Table 4, both cases assume a horizontal hydrogen release with a hole diameter of 2 mm. The key variable is the distance from the leak point to the cargo compressor room, referred to as the "distance to wall", which influences flame impingement and thermal exposure.

Table 4. Case description for scenario 3 including leak size, direction and distance to wall.

Case	Hole diameter [m]	Leak direction	Distance to wall [m]
S#3-H-1	0.002	Horizontal	0.5
S#3-H-2	0.002	Horizontal	2.0

Figure 17 illustrates the spatial configuration of the leak point relative to Vent Mast No. 3, Vent Mast No. 4, and the cargo compressor room. The leak is directed along the +X axis and originates 2.5 metres above the open deck. This geometric arrangement defines the boundary conditions for each case and supports the evaluation of thermal risk to nearby structures.

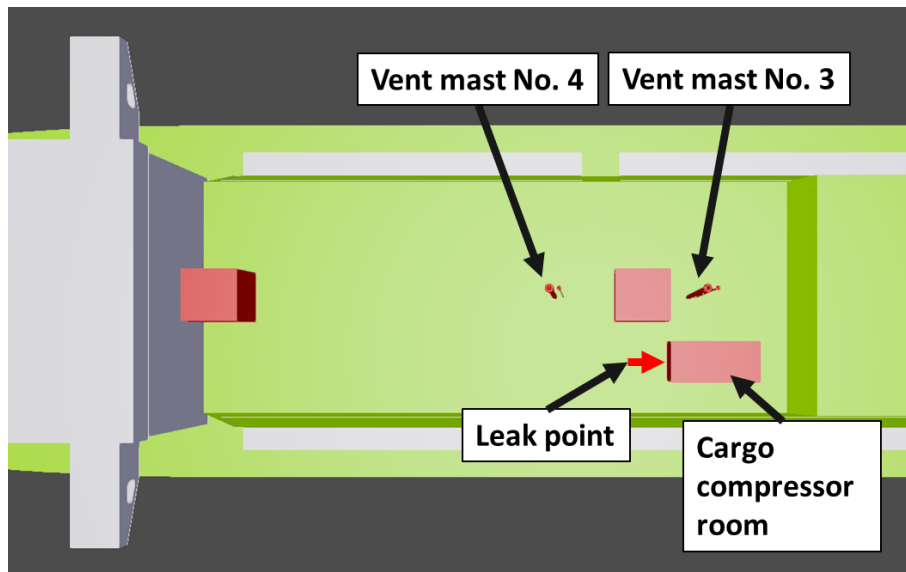


Figure 17. Cargo area schematic showing the leak point and its relation to Vent Masts and the cargo compressor room.

4.4.MODEL SETUP

The computational domain was constructed to represent a representative section of an LH₂ carrier, focusing on the vicinity of Vent Mast No. 3 and the adjacent cargo compressor room. In FLACS, the cargo compressor walls and structural boundaries are treated primarily as solid obstacles defined by their thickness, density, specific heat, and thermal conductivity. The simulation scenario assumes a hydrogen leak originating from a designated point located 2.5 metres vertically above the open deck, directed along the positive X-axis (see Figure 18). This location was selected to reflect a plausible failure point in the pressure relief system or associated piping.

The release characteristics were defined using a discharge coefficient of 0.85, and the pseudo-source behaviour was modelled using the Ewan–Moodie approach, which is suitable for high-pressure hydrogen releases. The release was treated as steady-state, assuming constant flow conditions throughout the simulation period. Both the internal pipe temperature and the ambient external temperature were set to 15°C, reflecting typical operating conditions in temperate marine environments.

This setup enables the assessment of thermal effects resulting from jet fire propagation and their potential impact on critical shipboard structures, particularly the cargo compressor room and surrounding open deck areas.

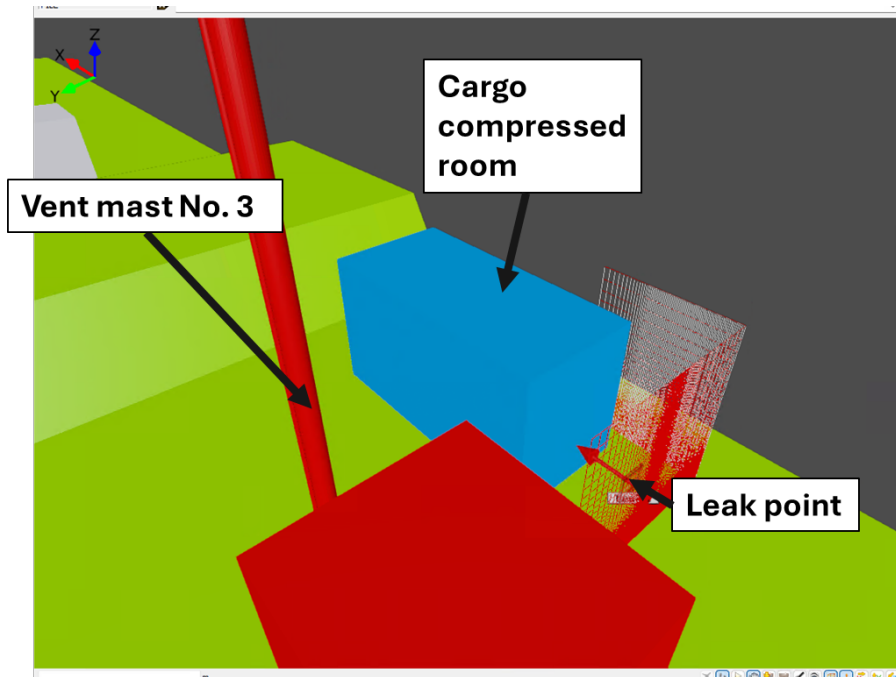


Figure 18. Simulation setup for hydrogen jet fire near cargo compressor room: grid configuration and leak point.

4.5.SIMULATION RESULTS

Figure 19 presents the total heat flux distribution on the wall of the cargo compressor room resulting from a jet fire initiated by a 2 mm hydrogen leak located 0.5 metres from the structure. The simulation snapshot corresponds to approximately 7 seconds after the onset of release ($t \approx 6.95$ s), representing the early-stage thermal impact of the jet flame.

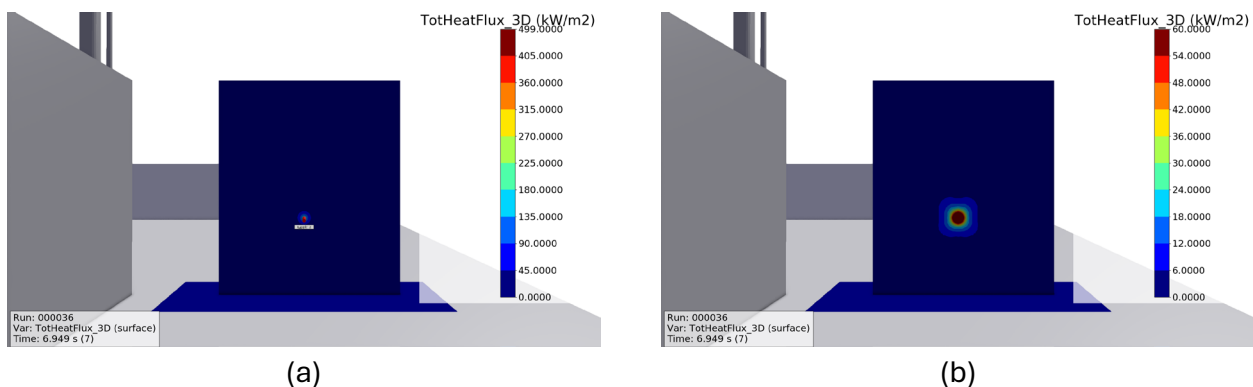


Figure 19. Total heat flux distribution on the cargo compressor room wall at $t \approx 7$ s for a 2 mm hydrogen leak located 0.5 m from the structure: (a) full-scale legend, (b) capped at 60 kW/m².

Figure 19(a) displays the full range of heat flux values, including the maximum observed intensity. The distribution indicates widespread thermal exposure, with peak values approaching 500 kW/m², suggesting severe localised heating. Figure 19(b) applies a capped legend with a maximum value of 60 kW/m², allowing clearer visualisation of the extent of high-risk zones. It is evident that a substantial portion of the wall surface exceeds this threshold, highlighting the potential for structural damage or failure under such conditions.

These results demonstrate that within a short duration—less than 7 seconds—the jet fire can generate intense thermal loads on adjacent structures. The findings underscore the importance of evaluating flame impingement scenarios and implementing appropriate thermal protection measures in LH₂ carrier designs.

Figure 20 presents the total heat flux distribution on the cargo compressor room wall for the case in which the hydrogen leak occurs at a distance of 2 metres from the structure. The simulation snapshot corresponds to approximately 6.08 seconds after release initiation. Compared to the previous case (Figure 19), the thermal impact is significantly reduced, with the maximum heat flux values remaining below 3 kW/m².

The colour map indicates that only a small central region experiences elevated thermal exposure, while the majority of the wall surface remains within low heat flux levels. This result confirms that increased separation between the leak point and the structure leads to a substantial reduction in flame impingement and thermal load.

These findings reinforce the importance of spatial configuration in mitigating fire hazards. Even with identical leak characteristics, the proximity of the release source to critical structures plays a decisive role in determining the severity of thermal effects.

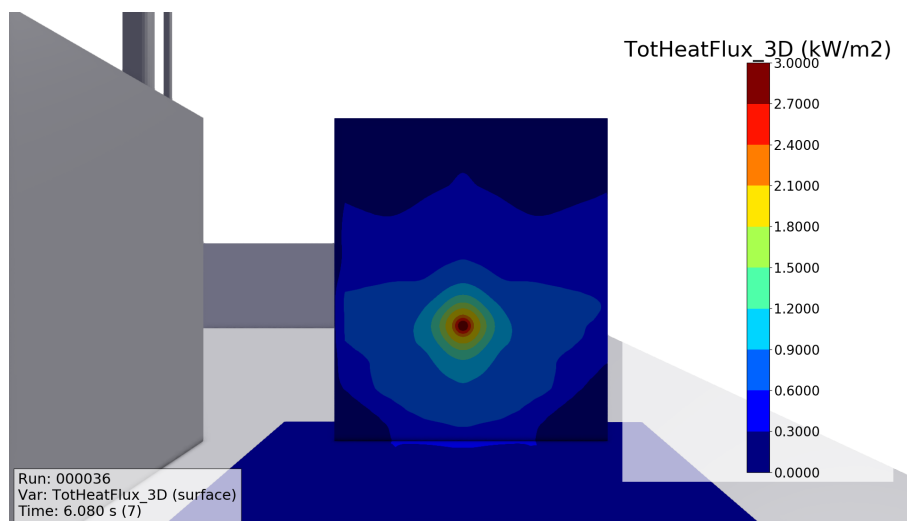


Figure 20. Total heat flux distribution on the cargo compressor room wall at $t \approx 6$ s for a 2 mm hydrogen leak located 2 m from the structure.

5. SCENARIO #4 – LH2 SPILL ANALYSIS

5.1.BACKGROUND

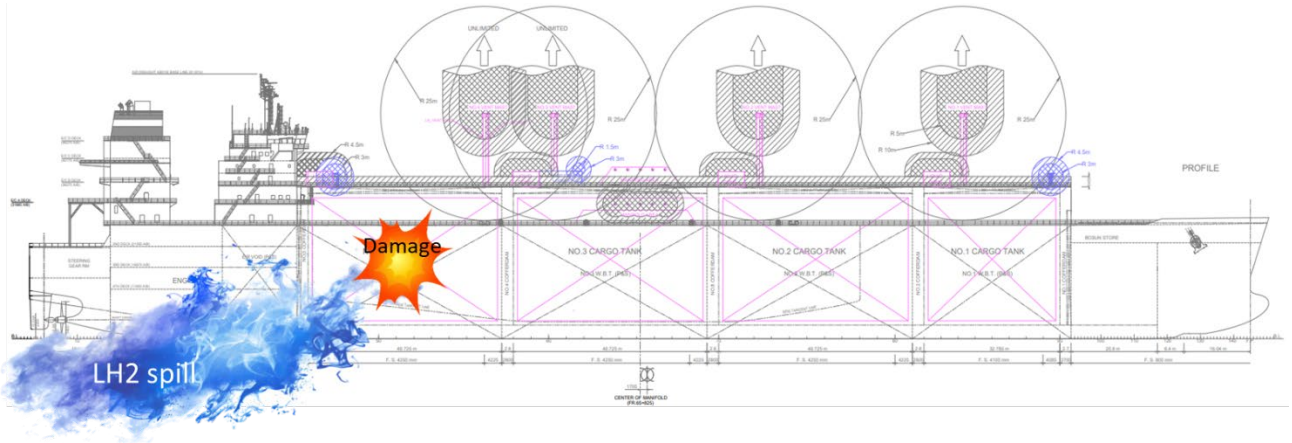


Figure 21. Liquid hydrogen (LH₂) spill occurs due to tank damage.

This study of Scenario #4 investigates the risk of liquid hydrogen (LH₂) spilling onto the seawater surface due to a collision or tank damage, as shown in Figure 21. The dispersion analysis of LH₂ is based on CFD simulations, considering the spill behaviour and the formation of a vapour gas cloud caused by the large temperature difference between the liquid hydrogen and seawater. This study aims to predict the behaviour of leaked LH₂ from a damaged tank spilling onto the sea. The main focuses are the size, shape, and location of the flammable gas cloud, as well as the shape, location, and size of the resulting pool.

The phenomenon of pool spreading and vaporisation of LH₂ has been analysed theoretically by comparison with other liquid fuels, such as liquefied natural gas (LNG) [4][5]. Cryogenic liquid spills behave differently depending on the ground type. On liquid surfaces, the pool area and vaporisation rate remain roughly constant, whereas on solid ground, the vaporisation rate decreases. Therefore, in the scenario of an LH₂ spill on seawater, the vaporisation rate is expected to stay nearly constant due to the natural convection caused by the large and steady temperature difference, along with stable film boiling. Additionally, LH₂ has a density of 70.8 kg/m³, which is significantly lower than LNG's density of 438 kg/m³. As a result, only about 7% of spilled LH₂ penetrates the water, compared to around 40% for LNG. The mixing of LH₂ with water is minimal. Moreover, the phenomenon of Rapid Phase Transition (RPT) is unlikely to occur with LH₂ spills, as its Leidenfrost temperature (−245 °C) is far lower than the temperature of water, which is close to its freezing point (0 °C). The predicted peak pressure generated by LH₂ during RPT events is only 17% of that produced by LNG, and the estimated explosive energy per unit volume is just 5% of LNG [4]. Hence, RPT resulting from an LH₂ spill on water is considered a minor safety concern.

An experiment conducted by Verfondern and Dienhart [6] provides a brief understanding of the phenomenon of LH₂ spill on water surface. Parameters of the LH₂ spill can be found in Table 5. The findings from the experiment are:

- A closed pool was formed, clearly visible and hardly covered by the white cloud of condensed water.
- The equilibrium pool radius moved forward and backward within the range of 0.4 to 0.6 m.

- c. Single small floes of ice escaped the pool front and moved outwards.
- d. After cutting off the source, a massive ice layer was identified where the pool was boiling.

Based on the literature review, a general understanding can be formed of what may occur when LH₂ spills onto the sea surface. This study focuses on investigating the formation of the vaporised gas cloud and its behaviour under varying conditions, such as different damage hole sizes and wind speeds.

5.2.MODEL VALIDATION

In this scenario study, FLACS is used to simulate the LH₂ pool spill process. To verify the reliability of the numerical method applied in this model, a validation study is carried out. The results are compared with an experimental test conducted by NASA on an LH₂ spill over white sand [7], as well as with a reference study by Holborn et al. [8], which used the same software. In the NASA experiment, liquid hydrogen was released through a downward-facing diffuser into a spill pond with a diameter of 9.1 m. Nine instrumented towers were placed downwind of the pond's centre to measure temperature, hydrogen concentration, and turbulence levels. Data collected from Tower 2 are used in this validation study.

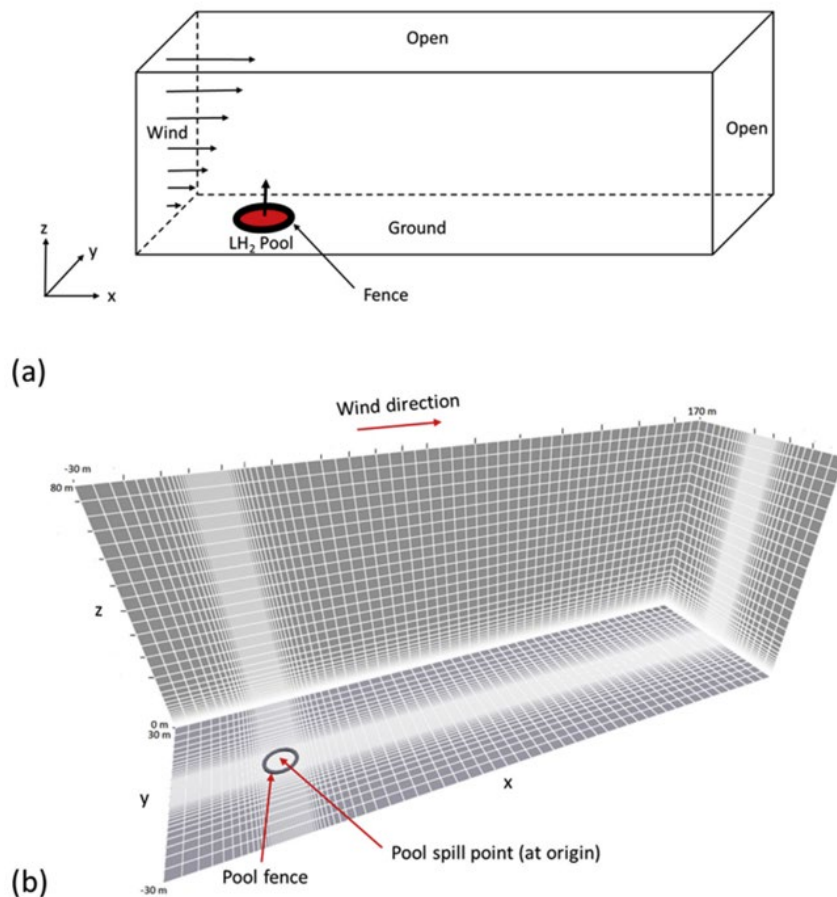


Figure 22. (a) Illustration of the boundary used in the based-line simulations; (b) the grid domain used in the base-lined simulations [8].

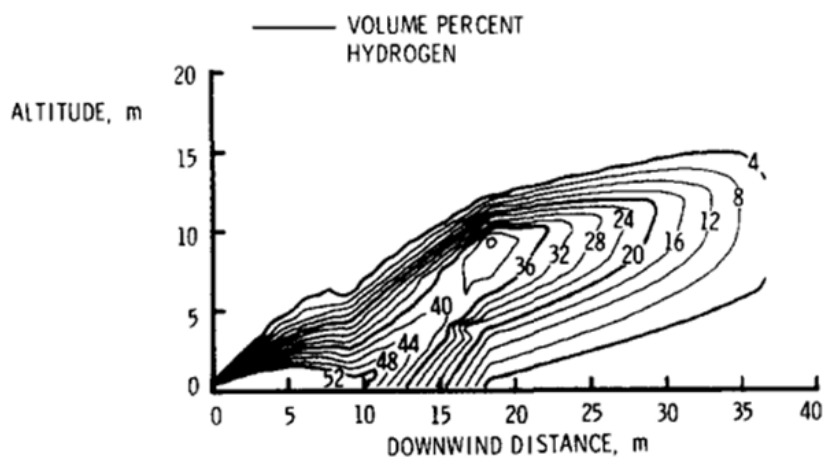
Considering the model setting in the reference as shown in Figure 22, the parameters for LH₂ spill simulation are listed in Table 5. In the pool region a grid cell size of 0.5 m was used in the X and Y directions

and 0.12 m in the Z direction. Outside the core region an expansion factor of 1.2 is used to for the increased grid size to a maximum grid size of 4 m. The wind speed in this validation study is 1.75 m/s at a height of 10 m. The direction of the wind is set to along the positive X-axis at 270°. A pool fence, 0.6 m high, was defined at radius of 4.25 m from the spill point. The thermal conductivity and thermal diffusivity of the ground used in the pool model were set to the properties of white sand. The pool release is set to start after 10s of simulation time to allow the wind field to become established.

Table 5. Parameters for LH₂ spill simulation.

Parameter	Value
Release rate	9.5 kg·s ⁻¹
Spill duration	38 s
Wind speed	1.75 m·s ⁻¹
Height of wind	10 m
Density of LH ₂	70.8 kg·m ⁻³
Ambient temperature	15 °C
Humidity	15 %
Inlet boundary type	wind
Ground roughness	3 mm
Ground thermal conductivity	3.72 W/m/K
Ground thermal diffusivity	1.45 × 10 ⁻⁶ m ² ·s ⁻¹

Figure 23 shows the hydrogen gas cloud concentration contours observed in NASA experiment, predicted in the referenced study and result from this validation study. Comparing those three results, the cloud shapes are basically consistent, with the plume being 1.5 times longer and higher. These three results exhibit the same dominant dispersion characteristics: a high-concentration region forms near the release point close to the ground, the plume extends downstream with progressive dilution, and the cloud gradually rises with distance due to hydrogen buoyancy. The differences between the NASA experiment and the CFD studies may be attributed to atmospheric turbulence fluctuations, sparse tower measurements, transient wind variations, and uncertainties associated with reconstructing hydrogen concentration from temperature measurements in the actual experiment. In contrast, many of these localized fluctuations are suppressed in the CFD simulations.



(a)

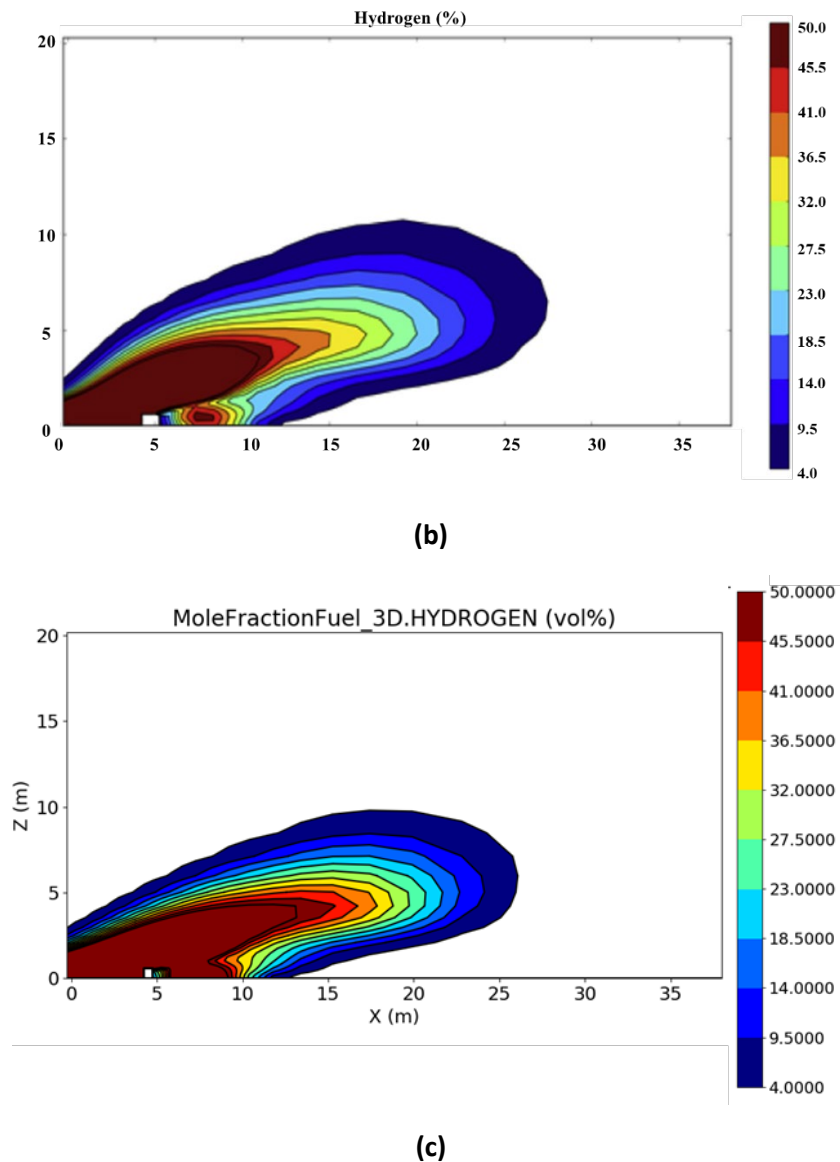


Figure 23. (a) Contours of constant hydrogen concentration deduced from temperature data from towers 2, 5 and 8; Test 6, time = 21.33 s [7] (b) The hydrogen gas cloud concentration contours after 20 s predicted in the referenced study [8] (c) Hydrogen concentration distribution at t=20 s from this validation simulation.

Figure 24 shows a comparison of hydrogen concentration (vol%) over time (s) for NASA Test 6 conducted at Tower 2, including Simulation results from the current FLACS pool model validation study (solid green line), Reference simulations by Holborn using Standard grid (black dash-dot line) and High-resolution grid (blue dashed line), and experimental data from NASA (red squares). All simulation curves (green, black, blue) show a sharp increase in hydrogen concentration in the pool spill around 10 seconds, reaching a peak (around 90 vol%) between 25 and 35 seconds. The FLACS validation model (green) closely follows the trend of the high-resolution and standard grid simulations, suggesting good agreement across methods.

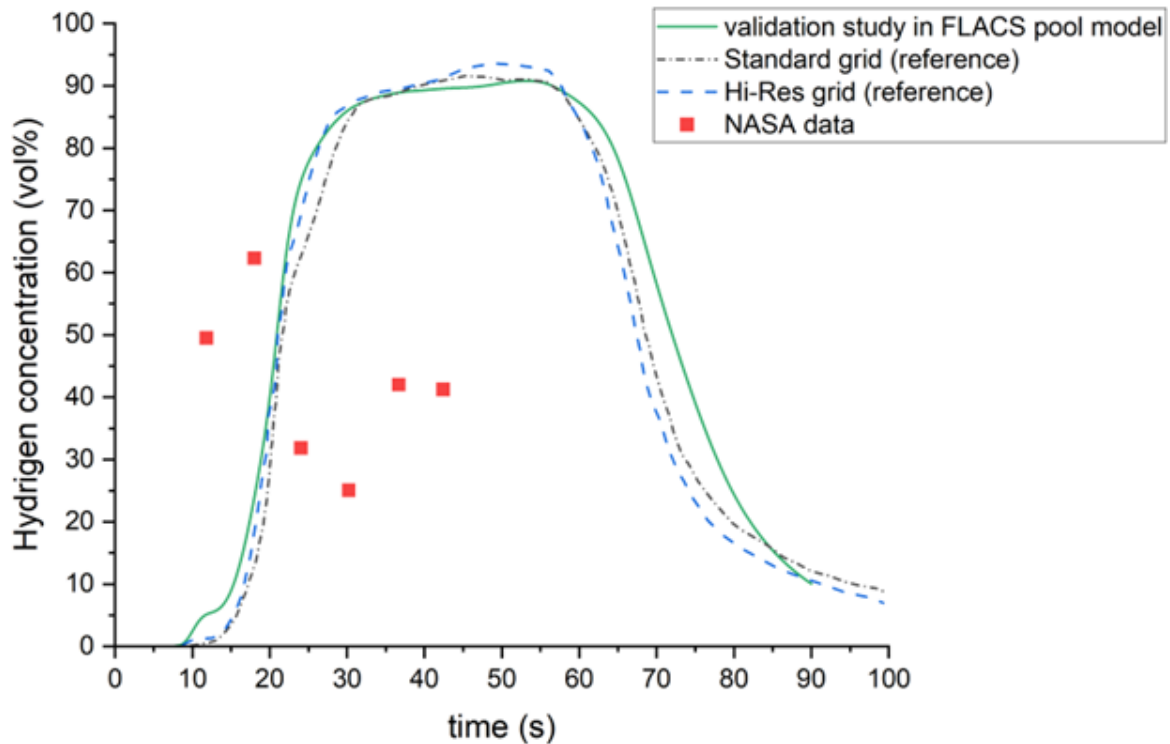


Figure 24. Comparison of hydrogen concentration results for NASA test 6 with referenced work and our simulation results for tower 2.

A grid sensitive test of this validation study is conducted under three different grid levels. Table 6 shows information of these three grid levels. Figure 25 shows the hydrogen concentration over time at monitor point T2, simulated using three different grid resolutions. All grids show a similar trend. The small grid predicts a slightly earlier rise and sharper peak compared to the others, indicating finer resolution captures faster diffusion or stronger gradients. The large grid shows a smoother transition and a slightly delayed peak, possibly due to numerical diffusion at lower resolution. The standard grid lies between the small and large grids in both rise time and peak sharpness, suggesting a balanced compromise between accuracy and computational cost.

Table 6. Information of grid levels.

	Small grid			Standard grid			Large grid		
	x	y	z	x	y	z	x	y	z
Minimum grid size	0.25	0.25	0.6	0.50	0.50	0.12	1.00	1.00	0.25
Control volumes	280, 896			159, 360			40, 656		

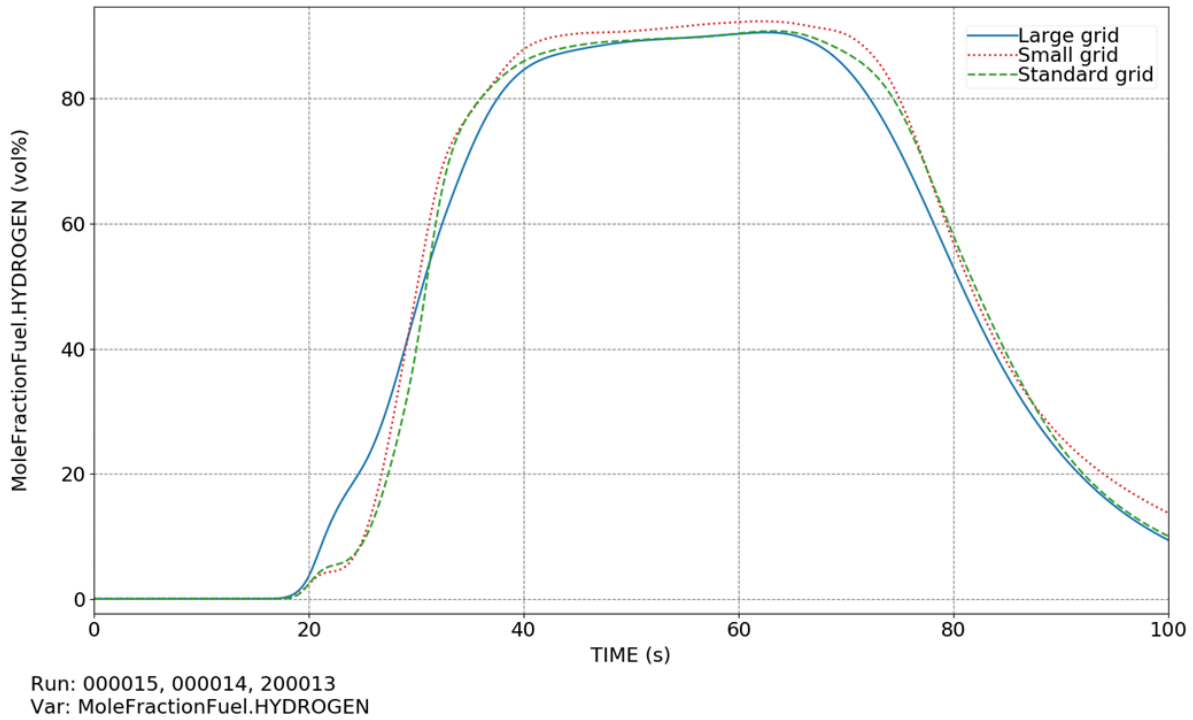


Figure 25. Hydrogen concentration of monitor point T2 for different grid levels.

5.3.MODEL SCENARIOS AND CONDITIONS

FLACS pool model allows the deposition and spread of a pool of liquid from a spill onto the ground simulated and the release source term to the gas dispersion model to be calculated. In FLACS pool model the liquid and gaseous states are modelled separately. Thus, the model does not consider relations expressing the thermodynamic state or slip velocity between two phase co-existing in the same control volume [9].

The behaviour of the spreading liquid pool is approximated by solving the shallow-water equations as a two-dimensional Cartesian grid along the ground and assuming that the pool properties are uniform in across the height of the pool. The pool model solves conservation equations for mass, momentum and enthalpy for the pool [9].

In this scenario, the mass flow rate of the pool is an essential parameter need to be calculated. The equation of release mass flow rate is as follows:

$$\dot{m} = C_d \cdot A \cdot \rho \cdot \sqrt{2\Delta P / \rho} \quad (1)$$

where C_d is discharge coefficient which value is 0.62, considering the coefficients that for side-damaged ships [10]. A is the area represents the leak hole size. ρ is liquid hydrogen density which value is 70 kg/m³. $\Delta P = P_{tank} - P_{atm}$ represents the pressure difference between the tank and the atmosphere; in this study, a value of 0.1 bar is considered.

To define the representative leak scenarios from ship collisions or tank damage, real-world accident cases were considered. For example, in 2007, a barge carrying a crane collided with the oil tanker Hebei Spirit off the west coast of South Korea. The crane punctured tanks 1, 3, and 5 on the port side of the tanker, causing

a significant spill of crude oil (see Figure 26). Based on this incident, a series of leak sizes was chosen for pool spill analysis, with leak radii varying from 0.2 m to 0.8 m.

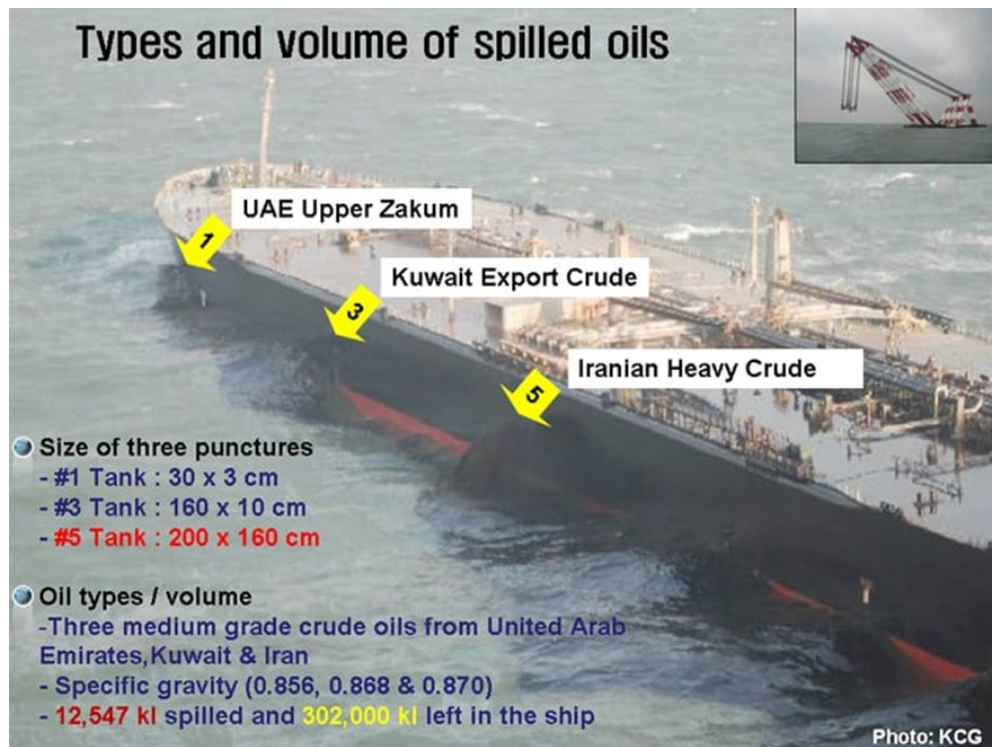


Figure 26. A real spill accident caused by collision and tank damage [11]

Table 7 presents the input parameters employed in the pool spill simulation, focusing on the pool release radius, mass flow rate, and environmental conditions. The released LH₂ pool is 100% LH₂ and simulations are performed under steady-state conditions. Ambient air condition is set to an atmospheric temperature of 20 °C and a pressure of 1 bar.

Table 7. Parameters for LH₂ spill simulation.

Parameter	Value
Leak radius	0.2 m
Release rate	92.14 kg·s ⁻¹
Spill duration	80 s
Simulation duration	160 s
Wind speed	10 m·s ⁻¹
Height of wind	10 m
Wind direction	+ X
Ground type	Sea water
Velocity of water	0.2 m/s

Water surface temperature	298K
Inlet boundary type	wind
Ground roughness	2 mm

5.4.MODEL SETUP

5.4.1. COMPUTATIONAL DOMAIN

The computational domain consists of a high-resolution core grid region and a stretched grid region to ensure both accuracy near the leak source and computational efficiency in the far field. As shown in Figure 27, the core grid region (highlighted in red) is densely meshed around the pool area enabling detailed resolution of hydrogen spill characteristics.

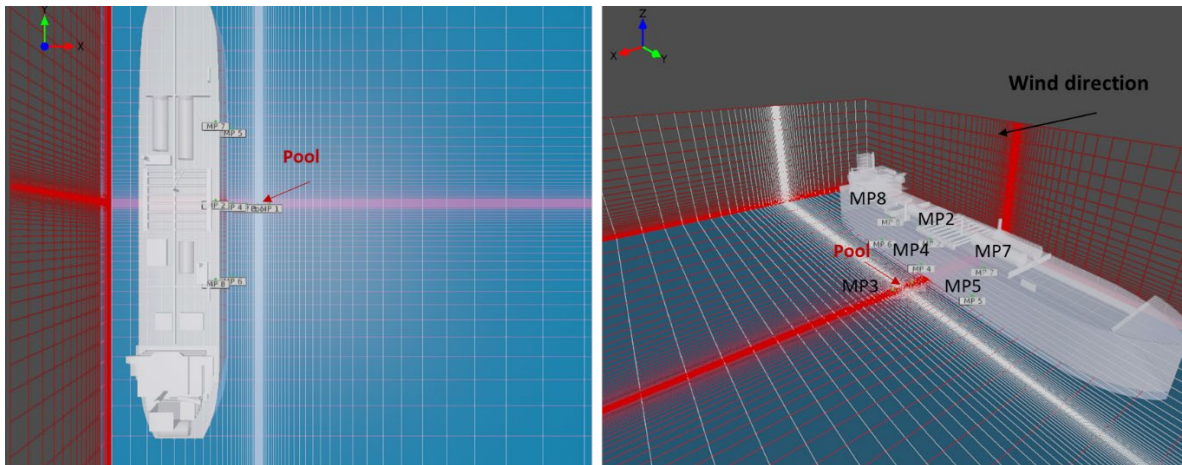


Figure 27. Computational domain for pool spill simulation.

The initial simulations were performed within a computational domain measuring 315 m × 260 m × 60 m in the X, Y, and Z directions. The grid employed has a total of 186,184 cells. Within the pool region, a refined core domain of 1.08 m × 1.08 m × 0.9 m was applied, using a uniform grid resolution of 0.18 in all three spatial directions. Outside this refined zone, the mesh was progressively coarsened using a stretched grid approach.

The simulations were performed using CFLC = 20 and CFLV = 1. The atmospheric stability was defined as Pasquill class F (stable conditions). The pool spill source was located at the origin (0, 0, 1), and the release was initiated after 10 seconds to allow the wind field to fully develop.

5.4.2. BOUNDARY CONDITIONS

Appropriate boundary conditions were applied for the LH2 pool model to ensure accurate representation of both pool spreading and gas cloud dispersion dynamics. The computational domain was bounded by six outer surfaces: XLO, XHI, YLO, YHI, ZLO, and ZHI. Each boundary was specified according to the prevailing

flow direction and its physical interaction with the surrounding environment. The detailed boundary conditions used in the simulation are presented in Table 8 (for wind direction, please see Figure 27).

Table 8. Boundary conditions for LH₂ pool spill simulation.

Name	Type
XHI	Nozzle
XLO	Wind
YHI	Wind
YLO	Wind
ZHI	Wind
ZLO	Nozzle + solid wall

5.5.SIMULATION RESULTS

Initial simulations were carried out for a pool spill with a leak radius of 0.2 m and a mass flow rate of 92.14 kg/s, over a spill duration of 80 s. To capture the complete process from gas cloud formation through to dispersion below the safety threshold concentration (4%), the total simulation time was extended to 160 seconds. Figure 28 presents a sequence of 3D visualisations showing the evolution of hydrogen gas cloud concentration contours over time following a LH₂ pool spill onto a water surface. The images display simulation snapshots at 40 s, 80 s, 110 s, and 140 s.

The location of the LH₂ pool significantly influences the spatial distribution of the resulting vapour cloud around the vessel. When the wind flows across the ship, and the pool is located in the downstream region, the flow over the ship geometry induces separation and recirculation zones. This results in a highly unsteady vapour cloud behaviour, characterised by pronounced fluctuations and an increased dispersion surrounding the vessel, as illustrated in Figure 28.

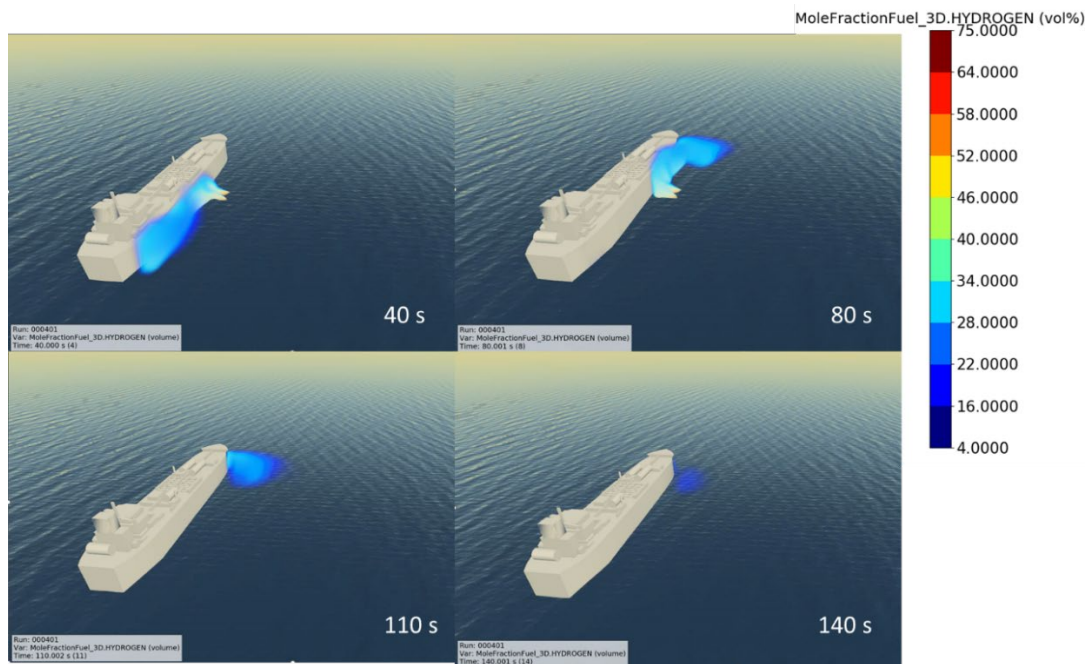


Figure 28. 3D plot for hydrogen gas cloud concentration contour development after LH₂ pool spill.

Figure 29 presents a sequence of 2D horizontal cross-section contour plots close to the ground, showing the concentration of hydrogen following an LH₂ pool spill at 40 s, 80 s, 110 s, and 140 s. The x-axis represents the downwind distance from the spill source. The y-axis represents lateral spread from the plume centreline. Colour contours show hydrogen concentration from 4% (blue) to 75% (dark red). Each plot illustrates the evolution of the hydrogen cloud as it develops and disperses over the sea surface. The cloud builds up during the active spill phase (0–80 s), after which it begins to spread and dilute downstream following the termination of the release. Ship-induced wake effects play a significant role in shaping the plume behaviour in the near-field region, particularly close to the source, where strong flow disturbances influence the initial dispersion pattern.

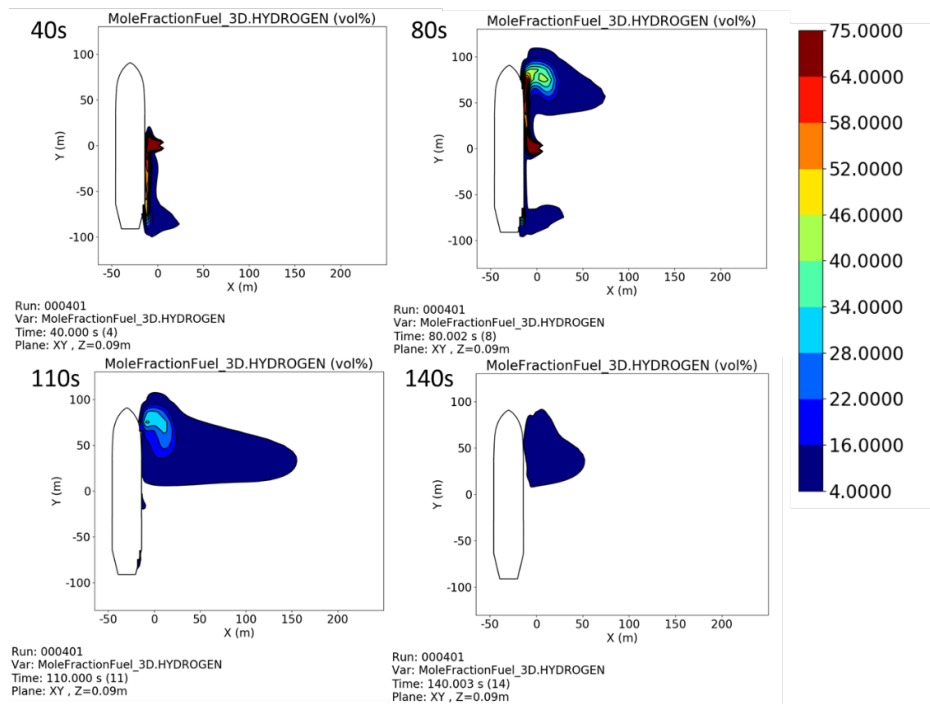


Figure 29. Horizontal cross-section contour plots after LH₂ pool spill at times 40 s, 80 s, 110 s, and 140 s.

Figure 30 shows a sequence of vertical cross-sectional contour plots illustrating the evolution of hydrogen gas concentration in the X-Z plane. These plots provide insight into the vertical height and downwind dispersion of the gas cloud. The results clearly show a buoyant hydrogen plume interacting with both the ambient wind field and the ship geometry. The gas initially rises rapidly from the spill location, forming a curved, elevated plume due to buoyant effects. It then transitions into a diluting, downstream-advected cloud with minimal ground-level concentration.

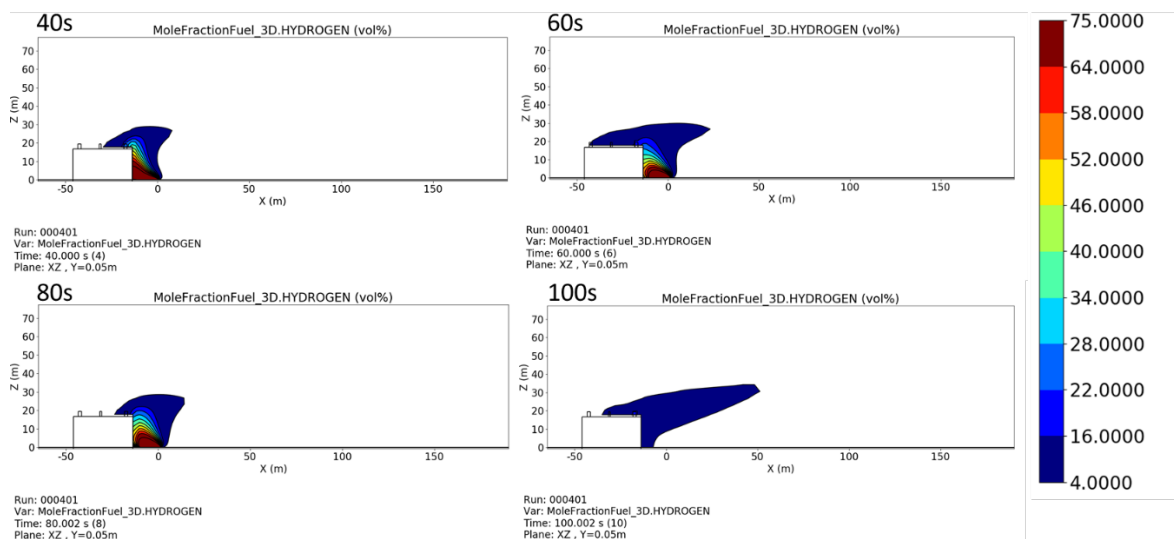


Figure 30. Vertical cross-section contour plots after LH₂ pool spill at times 40 s, 80 s, 110 s, and 140 s.

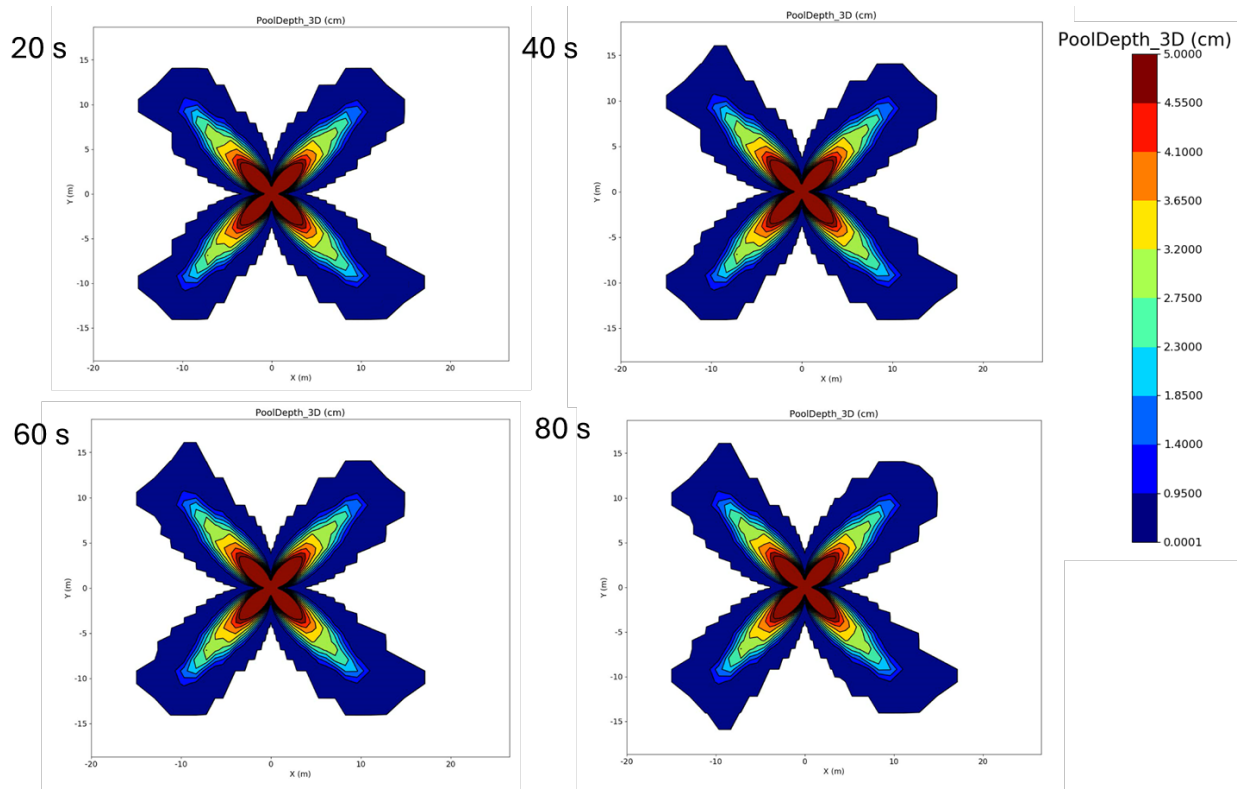


Figure 31. Top-down view of liquid hydrogen (LH₂) pool depth.

The behaviour of pool phase can also be obtained from this simulation study. Figure 31 shows a top-down view of LH₂ pool depth over time, following a spill on sea surface. The contours display the pool thickness in centimetres (cm) at four time steps: 20 s, 40 s, 60 s, and 80 s. The LH₂ pool initially spreads rapidly within the first 10 s, maintaining an approximately symmetric radial expansion pattern. The observed four-lobed structure may be attributed to the numerical grid effects, boundary condition influences, or the specific characteristics of the release geometry. Figure 32 presents a side view (X-Z plane) of LH₂ pool depth profile over time. The plots show how the vertical cross-section of the pool evolves. The LH₂ pool stabilises quickly after the initial spill, with only slow changes between 20–80 s. Pool depth and shape suggest quasi-steady state conditions during this period. These results for the pool phase are consistent with the theoretical conclusions and experimental findings from the literature review.

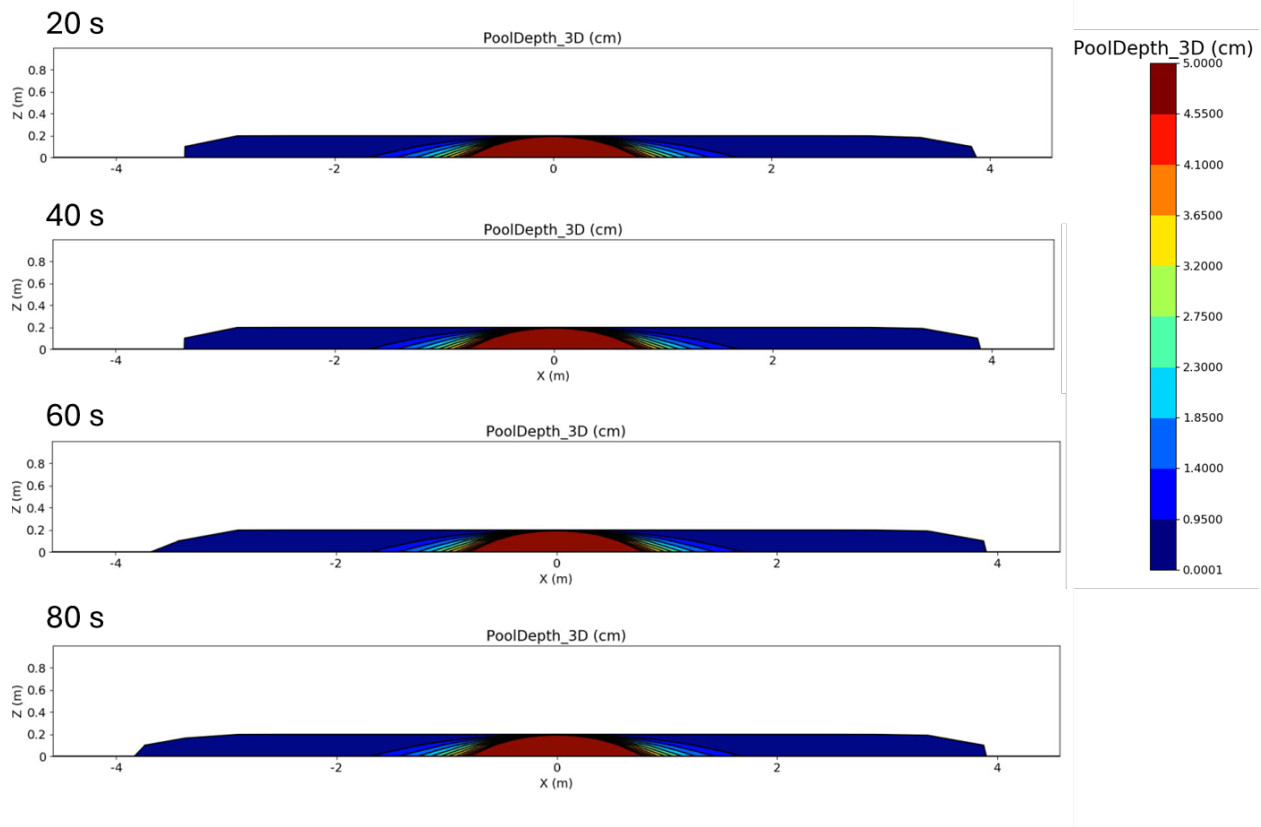


Figure 32. Side view (X-Z plane) of the LH₂ pool depth.

This simulation study helps clarify the phenomenon of LH₂ pool spill on the sea surface, which shows a quite different conditions from other liquid fuels like LNG and LN₂. Besides, these results help define hazard zones, safety standoff distances, and critical timescales for emergency response.

The effect of leak radius and wind conditions are also studied in this scenario. Figure 33 shows how hydrogen concentration changes over time at a single monitoring point for wind speeds of 1, 5, 10, and 15 m/s. At low wind speeds, the gravitational spreading and weak mixing cause the accumulation of a hydrogen gas cloud. At higher wind speeds, turbulent entrainment increases, resulting in faster dilution and transport away. Figure 34 presents contour plots of hydrogen concentration in the X-Z plane for different wind speeds at roughly the same time. With increasing wind speed, vertical mixing accelerates, reducing near-ground accumulation. Under low wind conditions, buoyancy governs the dispersion behaviour of the hydrogen cloud, promoting vertical rise. In contrast, at higher wind speeds, momentum-driven advection enhances horizontal dispersion.

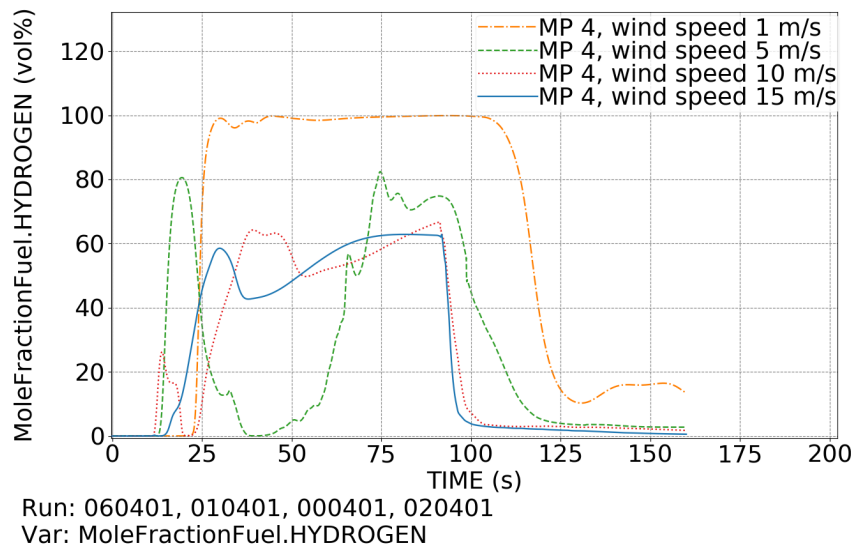


Figure 33. Temporal evolution of hydrogen mole fraction at MP4 under varying wind speeds

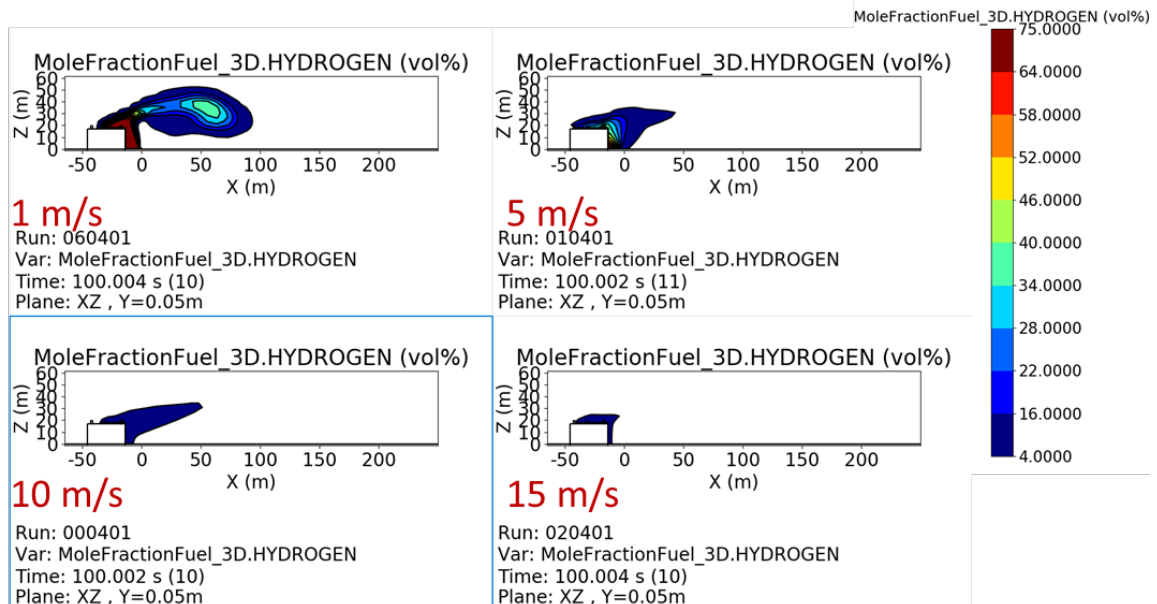


Figure 34. Distribution of hydrogen vapour cloud in the X-Z plane at 100 s for different wind speeds

Figure 35 shows how different pool radii ($r = 0.2, 0.4, 0.6, 0.8$ m), representing increasing mass flow rates (92.14 kg/s, 368.56 kg/s, 829.26 kg/s, 1474.23 kg/s), influence hydrogen concentration over time at MP5. A larger mass flow rate leads to stronger plume momentum, slower dilution relative to the supply, which results in higher concentration and longer persistence. Figure 36 presents contour plots of hydrogen concentration in the X-Z plane for different pool radii at roughly the same time. The 2D results provide a more intuitive demonstration that an increase in pool radius corresponds to a higher LH2 release rate, which leads to a larger hydrogen cloud with greater spatial extent and longer persistence in the environment.

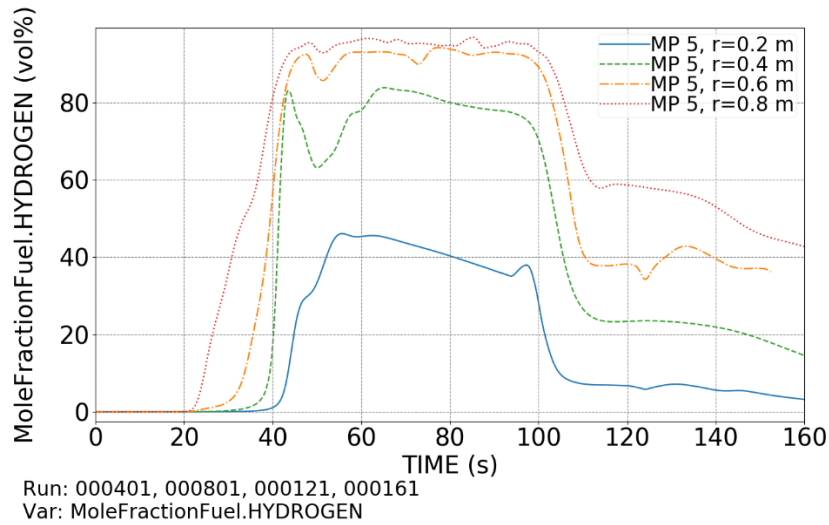


Figure 35. Temporal evolution of hydrogen mole fraction at MP5 under varying pool radius: 0.2 m, 0.4 m, 0.6 m, 0.8 m (mass flow rate: 92.14 kg/s, 368.56 kg/s, 829.26 kg/s, 1474.23 kg/s)

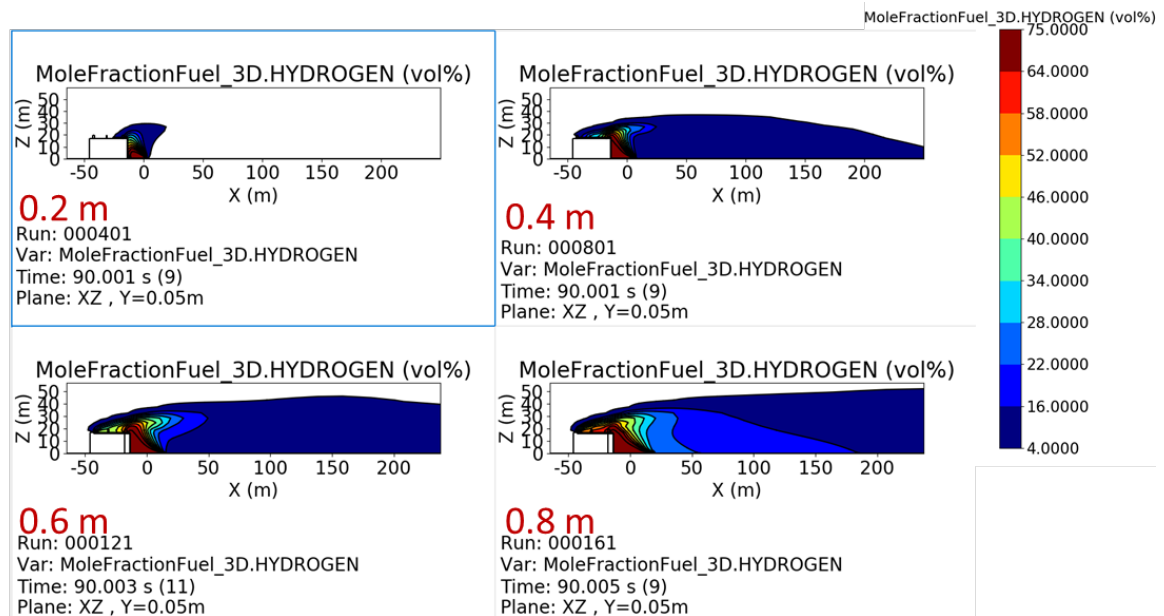


Figure 36. Distribution of hydrogen vapour cloud in the X-Z plane at 90 s for different pool radius: 0.2 m, 0.4 m, 0.6 m, 0.8 m (mass flow rate: 92.14 kg/s, 368.56 kg/s, 829.26 kg/s, 1474.23 kg/s)

6. SCENARIO #5 – DISPERSION ANALYSIS AT VENT MAST NO.3 & 4

6.1.BACKGROUND

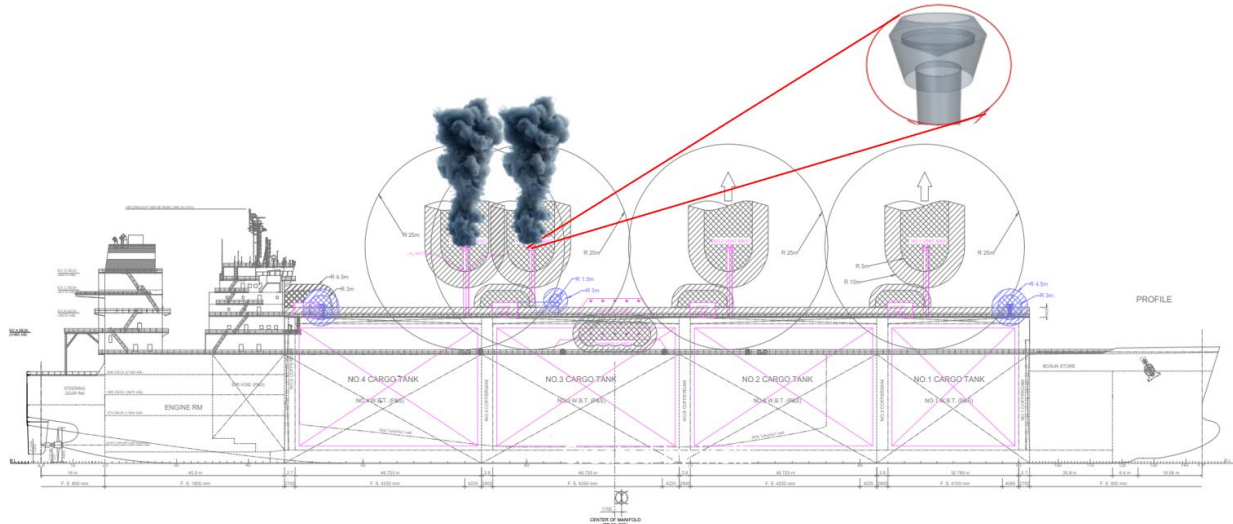


Figure 37. Dispersion event at vent mast No.3 & 4.

This study addresses a scenario involving the simultaneous release of hydrogen gas from No.3 & No.4 vent masts on an LH₂ carrier. The primary objective is to evaluate whether the combined release could lead to the formation of a flammable or asphyxiant atmosphere in the vicinity of the accommodation area.

A key concern is that elevated hydrogen concentrations near crew living quarters may compromise designated escape routes or hinder safe evacuation during an emergency. Using CFD simulation, the dispersion behaviour of hydrogen is analysed under this dual-release condition to:

- Identify hazardous zones around the accommodation block.
- Determine the horizontal and vertical extent of the hydrogen cloud.
- Support the safe placement of vent masts to minimise risk to personnel.

The findings will contribute directly to design validation and inform potential mitigation strategies during the vessel's development phase.

6.2.MODEL VALIDATION

To verify the reliability and accuracy of the CFD model used for the LH₂ vent mast hydrogen dispersion scenario, a detailed validation study was performed using experimental data from a well-established hydrogen release experiment. The aim was to identify a numerical setup that reliably reproduces the physical behaviour of hydrogen jets and to establish confidence in the model before applying it to shipboard scenarios.

The experimental data used for validation was obtained from the study on jet-formed hydrogen-air mixture [1], which systematically investigated the behaviour of vertical hydrogen gas jets under varying release conditions. The tests were conducted in a low-confinement environment to emulate free jet development and were specifically designed to support safety studies for hydrogen applications.

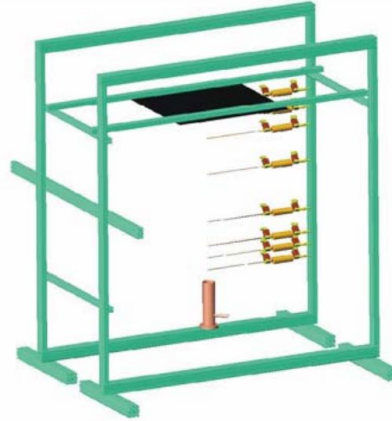


Figure 38. Sketch of the experimental setup for the distribution experiments. [12]

As illustrated in Figure 38 of the reference study, the experiment consisted of:

- A vertical hydrogen release nozzle fixed to the ground.
- A horizontal plate located 1.5 m above the nozzle to partially confine the rising jet and simulate ceiling-like obstruction effects.
- A series of eight gas sampling cylinders mounted on a vertical rod to measure hydrogen concentration at discrete heights between the nozzle and the plate.

Table 9. Release scenarios investigated in the experiments. [12]

Experimental series	Nozzle-diameter [mm]	Exit velocity [m/s]	Mass flow [g(H ₂)/s]	Release duration for 10g H ₂ inventory [s]
PIA	100	0.2	0.14	71.3
PIB	100	1	0.7	14.3
PIC	100	5	3.5	2.85
PID	21	5	0.15	64.7
PIE	21	100	3	3.23
PIF	21	200	6	1.62
PIG	4	100	0.14	71
PIH	4	200	0.29	35
PII	4	400	0.57	17.5

Table 9 summarises nine experimental hydrogen release scenarios, each combining one of three nozzle diameters (100 mm, 21 mm, 4 mm) with three different exit velocities. For each case, the corresponding mass flow rate and the release duration for a 10 g hydrogen inventory are provided. Among these, the PIA, PIB, and PIC series, which use a 100 mm nozzle, are the most relevant to this study. Given that the nozzle diameter in the current simulation is 0.6 m, the 100 mm experimental series offers the closest geometric

similarity and was therefore selected for CFD model validation. These cases span a range of exit velocities from 0.2 m/s to 5 m/s, resulting in mass flow rates from 0.14 g/s to 3.5 g/s, and provide a useful reference for plume development under large-diameter, low-to-moderate velocity conditions.

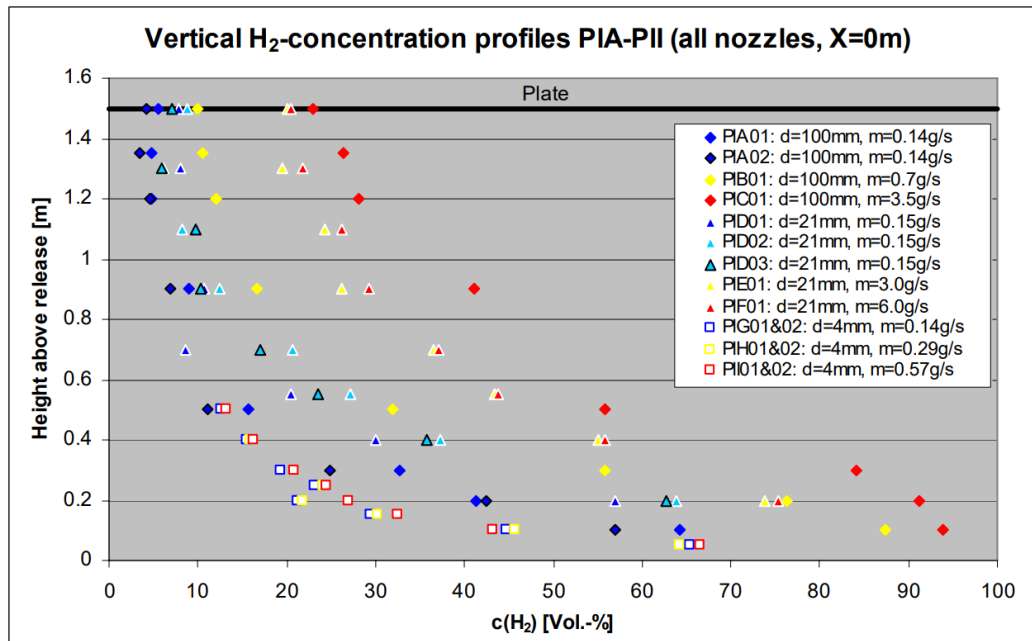


Figure 39. Vertical hydrogen concentration profiles along the axis of the release for experiments with all nozzles at different release rates. [12]

Figure 39 shows the hydrogen concentration results across different experimental series. For the 100 mm nozzle (PIA–PIC), concentration levels increased with higher mass flow rates and decreased with height. The highest concentration (>90 vol%) was observed near the nozzle at a mass flow rate of 3.5 g/s (PIC).

The difference between PIA01 and PIA02 is the positioning of the gas sampling points relative to the hydrogen release axis.

- PIA01 refers to measurements taken directly along the axis of the hydrogen jet. This is the central line extending vertically from the nozzle, where the highest hydrogen concentrations were generally observed because the released jet follows this path.
- PIA02 refers to measurements taken 5 cm offset from the axis, meaning the sampling points were laterally shifted 5 cm to the side of the jet's central axis. These off-axis points, especially those close to the nozzle, showed lower hydrogen concentrations, indicating the steep concentration gradient near the core of the hydrogen jet.

As a validation methodology, the CFD model was set up to replicate the experimental geometry, release conditions, and sampling locations. Simulations were performed using a range of numerical configurations varying in grid resolution, and initial condition profile.

To quantitatively evaluate the model's predictive performance, two statistical metrics were used based on the approach outlined in [13].

- Geometric Mean Bias (MG):**

$$MG = \exp(\overline{\ln(Co/Cp)}) \quad (1)$$

- Geometric Variance (VG):

$$VG = \exp(\overline{\ln(Co/Cp)^2}) \quad (2)$$

Where C_o and C_p represent the observed and predicted centreline hydrogen concentrations, respectively. An ideal model yields $MG=1$ and $VG=1$. Acceptable predictions lie within a range of $0.5 \leq MG \leq 2.0$, and VG values above the theoretical minimum curve $\ln VG = (\ln MG)^2$.

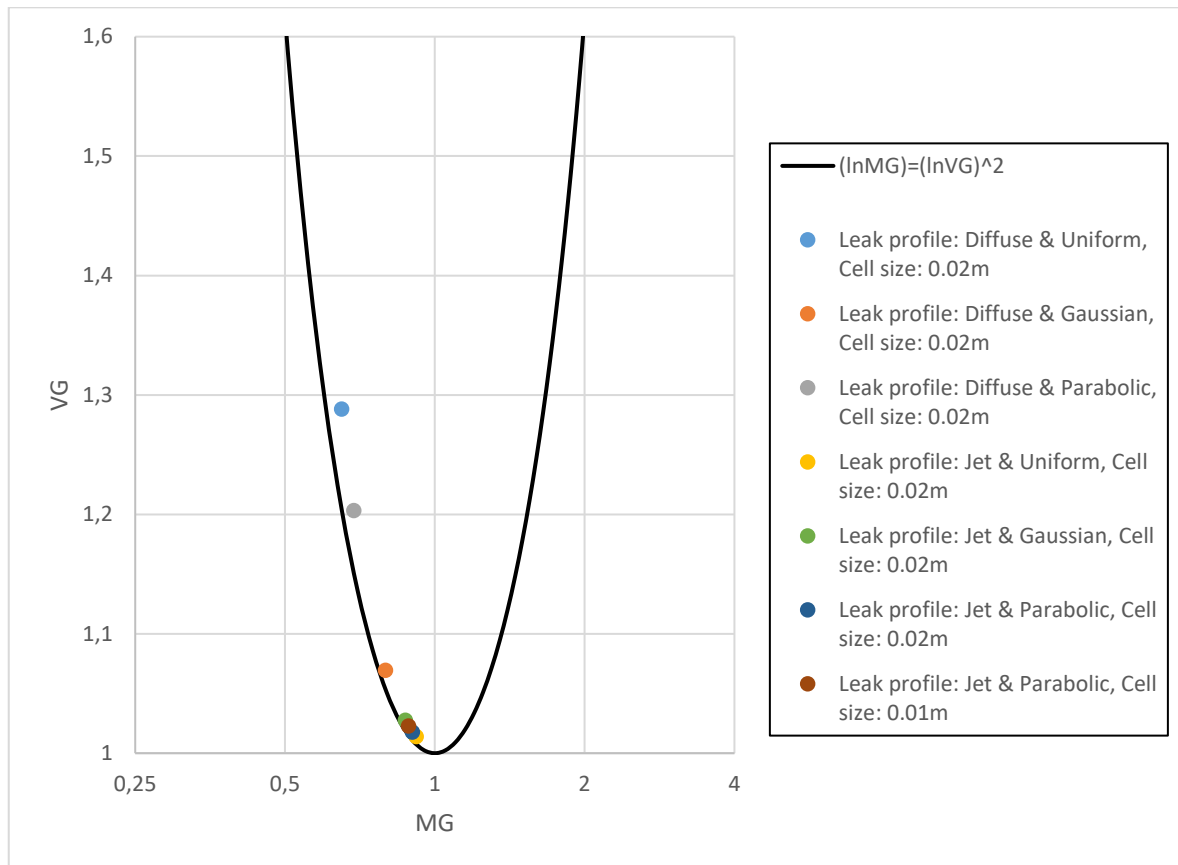


Figure 40. CFD model performance measures, geometric mean bias, MG, and geometric variance, VG.

Figure 40 presents the MG–VG plot summarising the validation results. Each point corresponds to a specific simulation configuration. The plot shows that:

- All tested configurations produced MG values within the 0.5–2.0 acceptable range, indicating that model-predicted mean concentrations were within a factor of two of observed values.
- VG values remained mostly below 1.4, signifying low variability and stable predictions across sensor locations.
- The best-performing case was "Leak profile: Jet & Uniform, cell size: 0.02m" or "Leak profile: Jet & Parabolic, cell size: 0.02m", which had MG and VG values very close to (1,1), indicating minimal bias and low scatter.

This case effectively reproduced the measured vertical hydrogen concentration distribution under jet-dominated release conditions and demonstrated good agreement across all sensor heights.

Selected Setup for Scenario Simulations

Based on this validation, the following configuration was selected for all subsequent scenario-based simulations of simultaneous hydrogen release from No.3 & No.4 vent masts aboard the LH₂ carrier:

- Jet-type release with uniform or parabolic initial concentration profile
- Grid resolution: minimum cell size 0.02 m

This setup provided a reliable balance between numerical stability, physical fidelity, and agreement with experimental observations, and is therefore considered an appropriate basis for dispersion risk assessment in shipboard applications.

6.3.MODEL SCENARIOS AND CONDITIONS

Table 10 presents the input parameters employed in the vent mast simulation, focusing on the LH₂ storage tank, environmental boundary conditions, and release characteristics. The LH₂ storage tank has a volume of 43,485.67 m³ and operates at a low temperature of –253 °C.

Table 10. Input Parameters for Vent Mast Simulation.

Parameter	Value
Hydrogen storage tank volume	43,485.67 m ³
Hydrogen storage tank temperature	-253 °C
Wind speed	1 m/s (Pasquill stability class F) 6 m/s (Pasquill stability class D)
Wind direction	180 degree (Bow to Stern)
Ship speed	0 m/s
Release rate	2.72 kg/s
Solver mode	Steady state
Vent mast exit diameter	0.6 m
Vent mast length	16.3 m
Vent mast orientation	Upward
Released gas	100% H ₂
Atmospheric temperature	20°C
Atmospheric pressure	1.0 bara

The environmental conditions account for two wind scenarios: 1 m/s (Pasquill stability class F) and 6 m/s (Pasquill stability class D), representing calm and moderately turbulent atmospheric conditions,

respectively. Wind direction is fixed at 180 degrees (from bow to stern), and the vessel is assumed to be stationary with a ship speed of 0 m/s during the release event.

Hydrogen gas is vented vertically upward through a mast with an exit diameter of 0.6 m, under representative mass flow rates: 2.72 kg/s (refer to Chapter 2.3, Scenario #1). The released gas is 100% H₂, and simulations are performed under steady-state conditions.

Ambient air conditions are set to an atmospheric temperature of 20 °C and a pressure of 1.0 bara. These parameters are critical for modelling the gas dispersion behaviour and evaluating the effectiveness of the vent mast system under representative emergency scenarios.

6.4.MODEL SETUP

6.4.1. COMPUTATIONAL DOMAIN

FLACS-CFD simulation primarily utilises a three-dimensional Cartesian grid composed of cubic or rectangular cells. The computational domain is structured into a high-resolution core grid region and a stretched grid region to ensure both accuracy near the leak source and computational efficiency in the far field. As shown in Figure 41, the core grid region (highlighted in red) is densely meshed around the vent mast outlet and the ship superstructure, enabling detailed resolution of hydrogen dispersion characteristics. Three views—side, top, and rear—illustrate how the mesh is concentrated near the potential release zone while gradually transitioning to a coarser grid away from the ship.

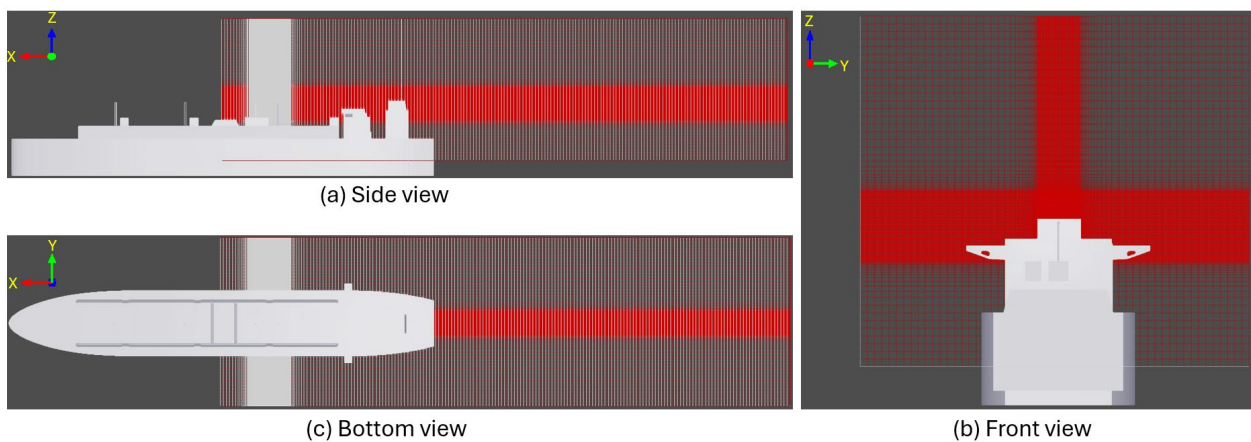


Figure 41. Computational domain for vent mast simulation.

As summarised in Table 11, the core grid region spans from 103.1 m to 129.26 m in the X-direction, –6.18 m to 6.18 m in the Y-direction, and 30.0 m to 50.0 m in the Z-direction for all scenarios (i.e., leak rate of

2.72 kg/s under wind speeds of 1 m/s and 6 m/s). To accurately capture localised dispersion phenomena near the vent mast, the minimum grid size in this region was set to 0.12 m.

The surrounding stretched grid region varies by scenario to optimise domain coverage while minimizing computational cost. In most cases, it extends from –250 m to 150 m in the X-direction, –60 m to 60 m in the Y-direction, and 0 m to 100 m in the Z-direction.

To ensure a smooth transition between grid zones, the maximum stretching factor was set to 1.2, and the largest cell size in the stretched region was limited to 2.2 m, balancing computational efficiency with spatial accuracy in far-field plume evolution.

Table 11. Computational domain for core and stretched grid regions.

		Case 1		Case 2	
Leak rate [kg/s]		2.72 kg/s		2.72 kg/s	
Wind speed [m/s]		1 m/s		6 m/s	
	Grid Coordinates	Minimum [m]	Maximum [m]	Minimum [m]	Maximum [m]
Core grid region	X	103.1	129.26	103.1	129.26
	Y	-6.18	6.18	-6.18	6.18
	Z	30	50	30	50
Stretched grid region	X	-250	150	-250	150
	Y	-60	60	-60	60
	Z	0	100	0	100

6.4.2. BOUNDARY CONDITIONS

Boundary conditions were selected for hydrogen dispersion modelling to ensure accurate representation of fluid dynamics around the vent mast during gas release. The simulation domain was defined with six outer boundaries: XLO, XHI, YLO, YHI, ZLO, and ZHI. Each boundary condition (shown in Table 12) was carefully configured based on the direction of flow and physical interaction with the surrounding environment.

Given a scenario where the wind blows from the bow to the stern (i.e., in the positive X-direction), the XLO boundary—the lower boundary in the X-direction—was modelled using a NOZZLE boundary condition. This setup treats the boundary as a semi-porous discharge interface suitable for representing gas outflow through vents or grilles. Unlike the EULER condition, the NOZZLE formulation uses a discharge coefficient derived from area porosity and drag parameters, enabling more robust handling of subsonic outflows. This choice helps minimise artificial pressure reflections and allows smooth outflow of hydrogen gas from the vent.

The XHI boundary, located at the stern-facing side of the domain, was configured as a WIND boundary with an incoming airflow velocity of either 1 m/s or 6 m/s, simulating real-world wind conditions under variable weather scenarios. The WIND boundary condition explicitly accounts for atmospheric turbulence and velocity profiles, making it well-suited for dispersion modelling in outdoor environments. This boundary

condition enables the controlled introduction of external wind fields into the domain, ensuring accurate development of the hydrogen plume along the wind direction.

The YHI and YLO boundaries (port and starboard sides) and the ZHI boundary (top of the domain) were assigned WIND boundary conditions as well, but with a wind speed of 0 m/s. This setup mimics neutral boundaries, allowing neither additional inflow nor outflow of ambient air. These settings help maintain lateral and vertical stability in the simulation by minimising disturbances caused by secondary flows or pressure imbalances at the outer edges.

The ZLO boundary (the lowest vertical boundary, representing sea surface) was treated as a hybrid condition: part NOZZLE and part solid surface. This configuration simulates a smooth, non-reflective wall that allows for boundary layer development near the surface, while also accommodating potential minor leakage or near-surface flow behaviour through the NOZZLE formulation. The NOZZLE portion ensures subsonic flow consistency, and the solid surface acts as a shear interface critical for accurate near-wall dispersion and plume lift-off behaviour.

This comprehensive configuration balances computational robustness and physical realism, while maintaining fidelity in modelling hydrogen dispersion under various atmospheric conditions.

Table 12. Boundary conditions for hydrogen dispersion simulation.

Name	Type	Remark
XHI	Nozzle	Pressure outlet
XLO	Wind	Wind speed: 1 m/s or 6 m/s
YHI	Wind	Wind speed: 0 m/s
YLO	Wind	Wind speed: 0 m/s
ZHI	Wind	Wind speed: 0 m/s
ZLO	Nozzle + solid wall	Smooth wall

6.5.SIMULATION RESULTS

Figure 42 and Table 13 illustrate the hydrogen dispersion characteristics from vent masts No.3 and No.4 on a LH₂ carrier under two environmental conditions, both with a constant leak rate of 2.72 kg/s but differing wind speeds of 1 m/s and 6 m/s. The primary aim of this analysis is to assess whether the released hydrogen cloud poses any risk to the accommodation block, particularly in terms of blocking escape routes or creating flammable atmosphere near crew-occupied areas.

These simulations were conducted to examine the extent of the hydrogen plume using three spatial indicators: A, representing the horizontal dispersion distance; B, the distance between the cloud and the accommodation structure; and C, the distance between the cloud and the funnel.

As the wind speed increases from 1 m/s (Case 1) to 6 m/s (Case 2), the hydrogen cloud shows a significant reduction in dispersion range across all parameters. The horizontal dispersion length (A) decreases from 371 meters to 146 meters, while the distance to the accommodation block (B) is reduced from 13.5 meters to 9 meters. Similarly, the vertical proximity to the funnel (C) decreases from 8 meters to 5 meters. This

trend indicates that higher wind speeds enhance advection and promote quicker lateral dispersion, while also suppressing the upward buoyant lift of the hydrogen cloud.

In evaluating overall safety, parameter B is an important consideration, as it indicates the closest distance between the hydrogen plume and the accommodation block. This distance influences the potential exposure of crew members and the accessibility of evacuation routes. Although B decreases slightly under stronger wind conditions, it remains at a safe margin in both cases. This demonstrates that the current vent arrangement prevents hazardous proximity to the accommodation area, even under low wind conditions.

Parameter C, which measures the distance between the hydrogen cloud and the funnel, is also of high relevance in safety assessments. The funnel is a potential ignition source due to its high-temperature exhaust gases. In low wind conditions, the hydrogen cloud rises higher due to buoyant forces and reaches up to 8 meters from the funnel. In higher wind conditions, vertical lift is constrained and the distance is reduced to 5 meters.

In conclusion, the results confirm that the current exhaust layout provides a sufficient safety buffer between the hydrogen plume and both the accommodation block and the funnel structure under the studied conditions. Nonetheless, these findings reinforce the importance of incorporating wind direction variability, exhaust elevation, and potential ignition sources into future design optimizations. A clear understanding of environmental dispersion mechanisms is essential to ensure safe hydrogen handling and protect onboard personnel.

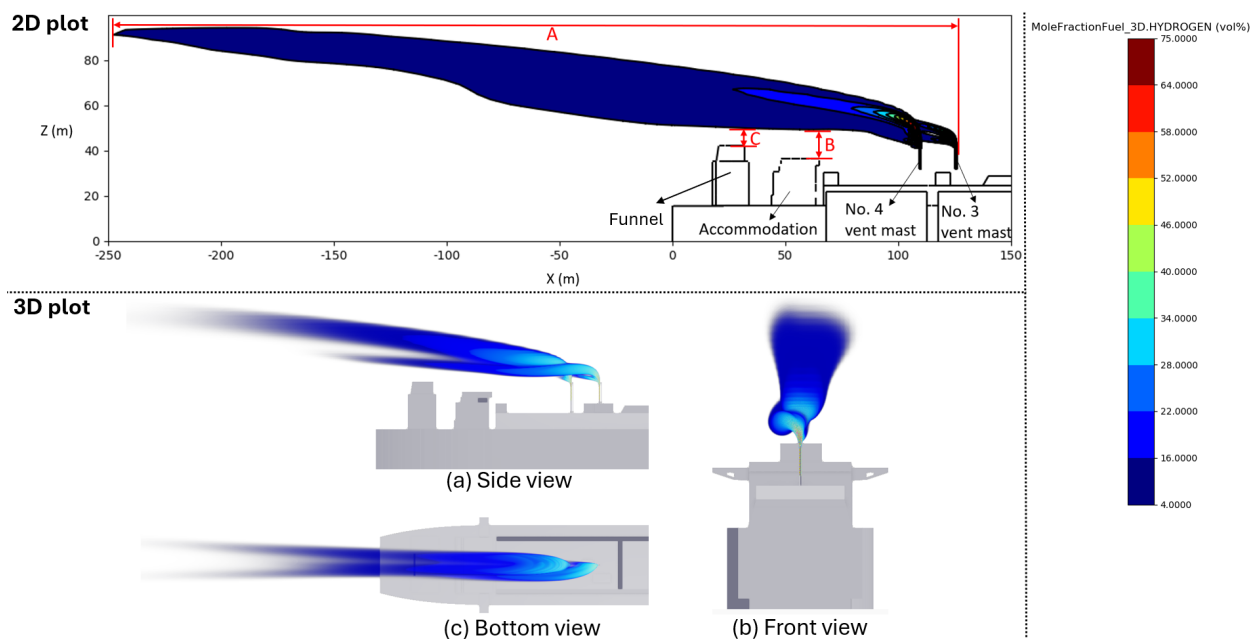


Figure 42. 2D and 3D Dispersion Profiles of Hydrogen Gas Released from No.3 and No.4 Vent Masts: (A) Horizontal dispersion length, (B) Distance between the cloud and the accommodation structure, and (C) Distance between the cloud and the funnel.

Table 13. Summary of Hydrogen Dispersion Distances under Different Leak and Wind Conditions.

Case no.	Leak rate [kg/s]	Wind speed [m/s]	A [m]	B [m]	C [m]
Case 1	2.72	1	371	13.5	8
Case 2	2.72	6	146	9	5

7. SCENARIO #6 – DISPERSION ANALYSIS AT VENT MAST NO.4

7.1.BACKGROUND

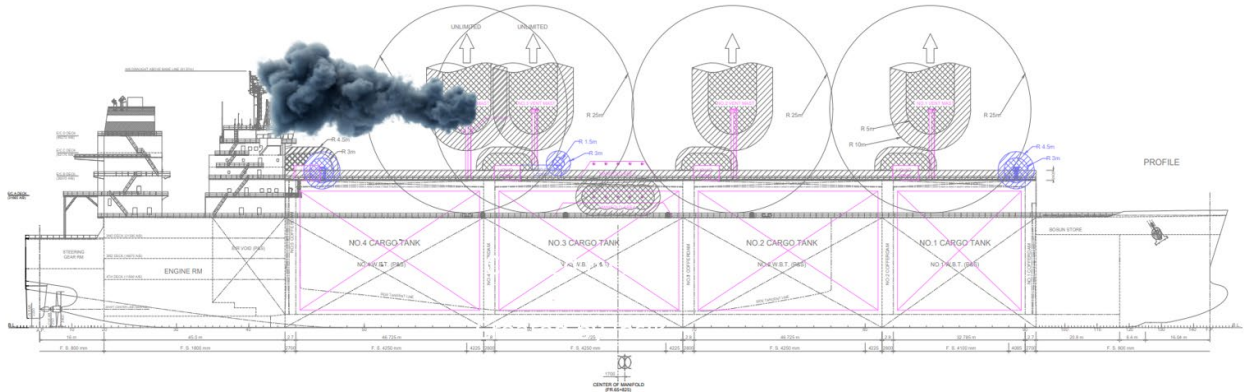


Figure 43. Dispersion event at vent mast No.4.

The analysis focuses on whether hydrogen gas released from either No. 3 or No. 4 vent mast can propagate toward and enter the ventilation system inlets that serve the accommodation, machinery, and service spaces. The potential ingress of hydrogen into these systems poses a severe risk, including the formation of flammable mixtures in confined spaces and the possibility of internal ignition.

This scenario highlights the importance of:

- Verifying the placement of ventilation inlets during detailed risk assessment phases.
- Ensuring that released hydrogen does not lead to hazardous conditions inside the vessel.
- Supporting future mitigation strategies or layout modifications to ensure internal safety.

Through these two scenarios, the CFD-based hydrogen dispersion analysis contributes directly to the risk-informed design of the LH₂ carrier, ensuring compliance with safety standards and enhancing crew protection in line with the vessel's operational profile.

7.2.MODEL VALIDATION

This simulation also focuses on the H₂ dispersion study. Refer to Chapter 6.2, *Model Validation*.

7.3.MODEL SCENARIOS AND CONDITIONS

This simulation also focuses on the H₂ dispersion study. Refer to Chapter 6.3, *Model Scenarios and Conditions*.

7.4.MODEL SETUP

7.4.1. COMPUTATIONAL DOMAIN

FLACS-CFD simulation primarily utilizes a three-dimensional Cartesian grid composed of cubic or rectangular cells. The computational domain is divided into a high-resolution core grid region and a stretched grid region to ensure accurate resolution of hydrogen dispersion near the leak source and computational efficiency in the far field. As illustrated in Figure 44, the core grid region (highlighted in red) is densely meshed around the vent mast outlets (No. 4) and the ship's superstructure. The mesh structure is visualised from three perspectives—side view (a), front view (b), and bottom view (c)—demonstrating the localised refinement near the leak zone and the gradual transition to coarser grids farther from the ship.

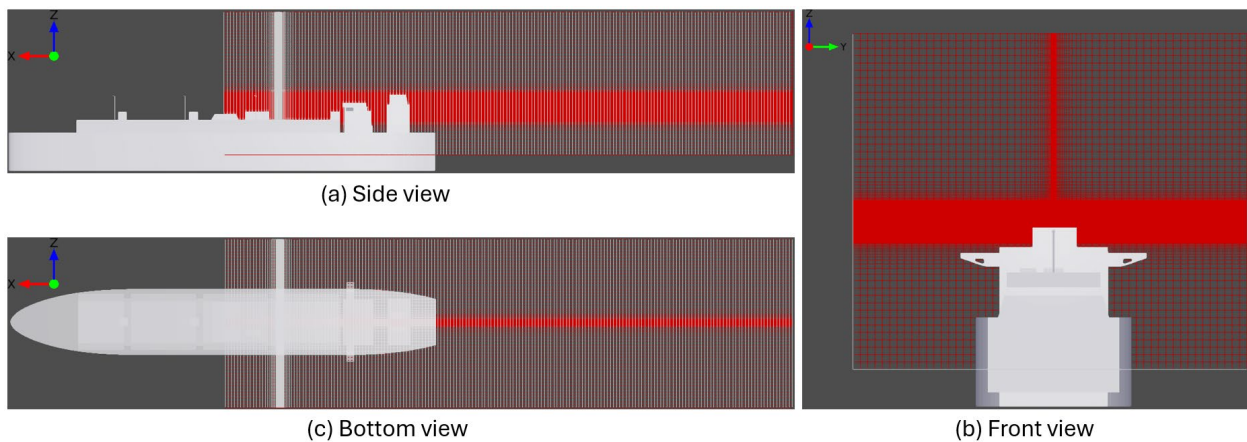


Figure 44. Computational domain for vent mast simulation.

According to Table 14, the computational domain for both Case 1 and Case 2 is configured with a constant leak rate of 2.72 kg/s, but under different wind speeds (1 m/s and 6 m/s, respectively). The core grid region is identical for both scenarios, defined in the X-direction from 109.1 m to 110.16 m, in the Y-direction from -0.54 m to 0.54 m, and in the Z-direction from 30.0 m to 46.0 m. This narrow, high-resolution domain is designed to accurately capture near-field hydrogen plume behaviour around the vent mast. The minimum cell size within the core grid is 0.12 m, providing fine spatial resolution critical for resolving sharp concentration gradients and buoyant flow near the source.

Surrounding the core, the stretched grid region extends from -250 m to 150 m in the X-direction, -60 m to 60 m in the Y-direction, and 0 m to 100 m in the Z-direction for both cases. This results in a total computational domain size of 400 m (X) × 120 m (Y) × 100 m (Z). The stretched region allows for gradual mesh expansion to capture far-field dispersion while reducing the overall computational cost. To maintain numerical stability and ensure a smooth transition between the fine and coarse mesh regions, a maximum stretching ratio of 1.2 is applied, and the largest cell size in the outer domain is limited to 2.2 m. This domain design provides an effective balance between computational efficiency and spatial accuracy in modelling hydrogen dispersion under varying wind conditions.

Table 14. Computational domain for core and stretched grid regions.

Case 1				Case 2	
Leak rate [kg/s]		2.72 kg/s		2.72 kg/s	
Wind speed [m/s]		1 m/s		6 m/s	
	Grid Coordinates	Minimum [m]	Maximum [m]	Minimum [m]	Maximum [m]
Core grid region	X	109.1	110.16	109.1	110.16
	Y	-0.54	0.54	-0.54	0.54
	Z	30	46	30	46
Stretched grid region	X	-250	150	-250	150
	Y	-60	60	-60	60
	Z	0	100	0	100

7.4.2. BOUNDARY CONDITIONS

This simulation also focuses on the H₂ dispersion study. Refer to Chapter 6.4.2, *Boundary conditions*.

7.5.SIMULATION RESULTS

Figure 45 and Table 15 summarise the hydrogen dispersion characteristics resulting from a simultaneous release at the No. 4 vent mast on an LH₂ carrier, under two distinct wind conditions. The primary objective is to assess whether the hydrogen cloud poses a risk to the accommodation block, particularly regarding potential obstruction of escape routes or the formation of flammable or asphyxiant atmospheres near crew-occupied zones.

These simulations were conducted at a fixed leak rate of 2.72 kg/s, with wind speeds of 1 m/s (Case 1) and 6 m/s (Case 2). The dispersion results were evaluated using three key spatial indicators: A (horizontal dispersion distance), B (distance to the accommodation block), and C (distance to the funnel).

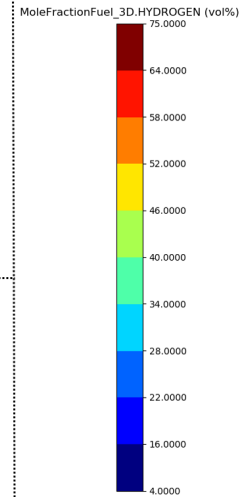
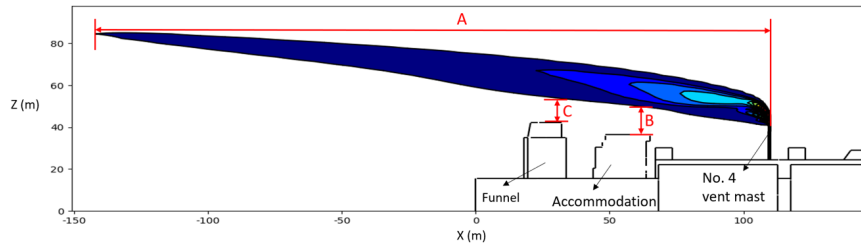
Under low wind conditions (Case 1), the hydrogen cloud spreads horizontally up to 252 meters, with 12 meters of clearance from both the accommodation block and the funnel. When wind speed increases to 6 m/s (Case 2), the extent of the cloud decreases significantly—A reduces to 76 meters, while both B and C shrink to 8 meters. This pattern demonstrates that stronger wind speeds limit both the horizontal reach and vertical rise of the hydrogen plume due to enhanced advection and reduced buoyancy-driven uplift.

Among the spatial indicators, parameter B is a key safety consideration, as it represents the minimum distance between the cloud and the accommodation area. While both cases remain within acceptable margins, the reduced clearance in Case 2 underlines the need for continuous monitoring and risk-informed layout design.

Similarly, the shorter C distance under high wind conditions suggests the plume's proximity to the funnel increases, underscoring the need to assess potential ignition risks during design validation.

In conclusion, the current vent configuration maintains a conservative buffer zone between the hydrogen cloud and sensitive structures, effectively minimising hazard even under varying environmental conditions. These results provide critical design input and highlight the need to consider wind speed, direction, and vent height in optimising hydrogen vent safety on LH₂ carriers.

2D plot



3D plot

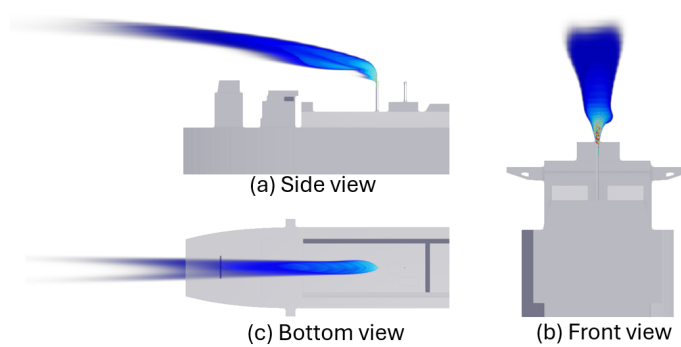


Figure 45. 2D and 3D dispersion profiles of hydrogen gas released from no.4 vent masts: (a) horizontal dispersion length, (b) distance between the cloud and the accommodation structure, and (c) distance between the cloud and the funnel.

Table 15. Summary of hydrogen dispersion distances under different leak and wind conditions.

Case no.	Leak rate [kg/s]	Wind speed [m/s]	A [m]	B [m]	C [m]
Case 1	2.72	1	252	12	12
Case 2	2.72	6	76	8	8

8. SCENARIO #7 – DISPERSION ANALYSIS FROM CARGO COMPRESSOR ROOM

8.1.BACKGROUND

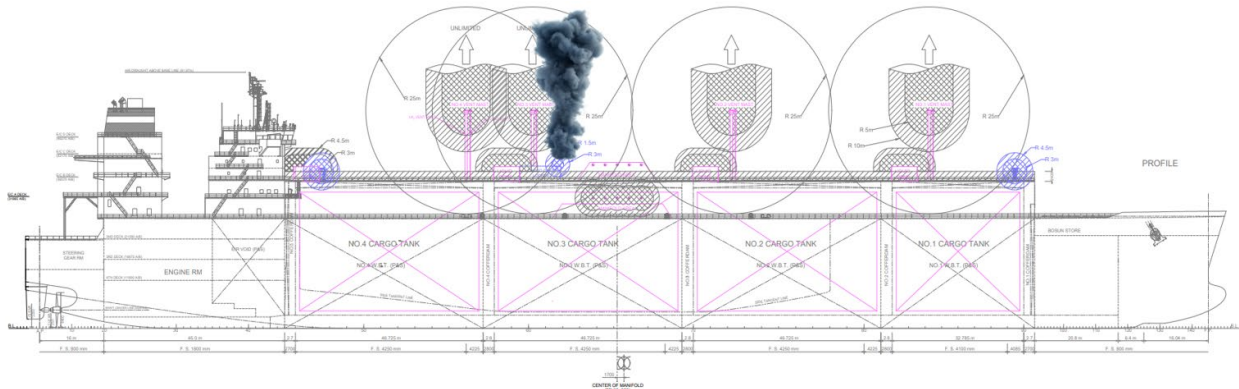


Figure 46. Dispersion event at cargo compressor room.

This study addresses a scenario involving the unintended release of hydrogen gas on the weather deck, potentially originating from the cargo handling system. Specifically, concern is raised about leakage within the cargo compressor room, which may result in hydrogen being emitted through the ventilation fan. The primary objective is to evaluate whether such a release could pose a flammability hazard or result in unsafe gas concentrations in operational areas.

A key concern is that hydrogen dispersion in this region may create ignition risks on exposed upper decks or impair localised ventilation efficiency. Using CFD simulation, the release behaviour is analysed to:

- Identify the hydrogen concentration profile at the ventilation fan outlet.
- Quantify the gas cloud's spread from the cargo compressor room to nearby zones.
- Support risk-informed decision-making regarding ventilation design and leak detection measures.

The findings will serve as a basis for scenario inclusion in the QRA and help validate the integrity of hydrogen management systems on the vessel.

8.2.MODEL VALIDATION

This simulation also focuses on the H₂ dispersion study. Refer to Chapter 6.2, *Model Validation*.

8.3.MODEL SCENARIOS AND CONDITIONS

As Figure 47 depicted, this scenario considers the potential release of hydrogen from the cargo compressor room on the weather deck, particularly via the room's mechanical ventilation system. The purpose is to simulate and quantify the hydrogen concentration emitted from the ventilation fan, which serves the cargo compressor room, under a full-room leakage event.

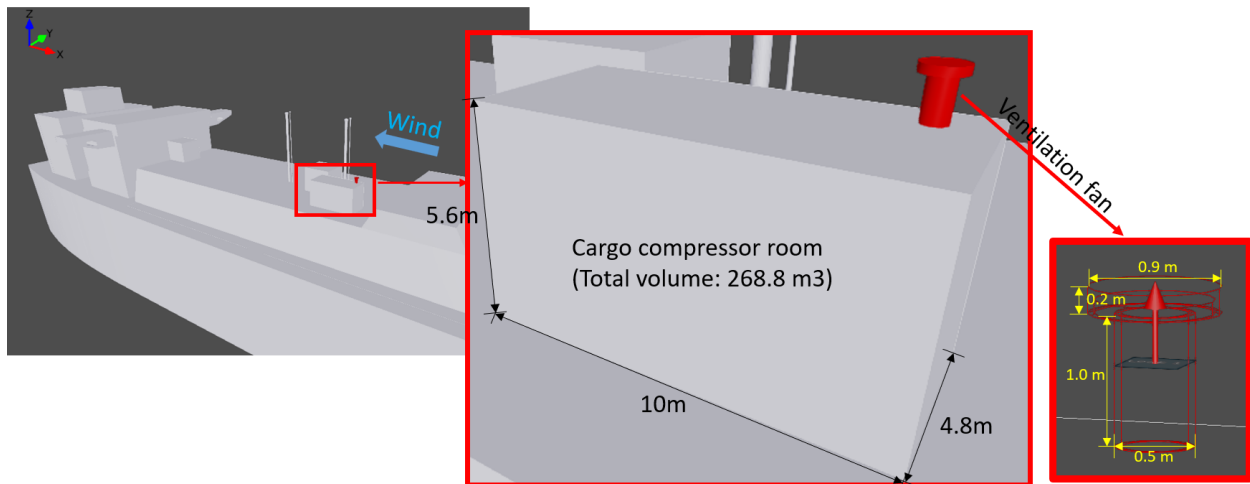


Figure 47. Configuration of the cargo compressor room and ventilation fan arrangement on board.

The cargo compressor room has a total internal volume of 268.8 m³. Under the assumption of a full hydrogen release within the room (i.e., 100% hydrogen fill by volume), the ventilation fan is modelled to operate at a 30 air changes per hour rate. This corresponds to a volumetric extraction rate of:

$$\frac{268.8 \text{ m}^3 \times 30 \text{ air changes}}{3600 \text{ s}} = 2.24 \text{ m}^3/\text{s} \quad (1)$$

Assuming normal temperature and pressure (NTP: 20°C, 1 atm), the average density of hydrogen is taken as 0.084 kg/m³. Thus, the resulting hydrogen mass flow rate (release rate) from the ventilation fan is calculated as:

$$\dot{m} = \rho Q = 0.084 \text{ kg/m}^3 \times 2.24 \text{ m}^3/\text{s} = 0.19 \text{ kg/s} \quad (2)$$

This release rate serves as the boundary condition for hydrogen outflow through the ventilation duct, representing a plausible worst-case scenario in which the room air is completely displaced by hydrogen due to internal leakage, and subsequently ejected by the ventilation system.

The dispersion modelling under this scenario aims to assess the spatial distribution of hydrogen concentration in the vicinity of the weather deck, particularly near exhaust outlets and adjacent equipment, in order to evaluate flammable zones, ignition potential, and sensor placement strategies.

Table 16 presents the input parameters addressing an unintended hydrogen release scenario from the cargo compressor room ventilation system. The cargo compressor room volume is assumed to be 268.8 m³, and the release rate is set at 0.19 kg/s, reflecting a low-intensity leak condition. The released gas is 100% hydrogen. Environmental conditions are defined by two wind speeds: 1 m/s (Pasquill stability class F) and 6 m/s (Pasquill stability class D), with the wind direction fixed at 180 degrees (from bow to stern). The ship is assumed to remain stationary during the event. Ambient atmospheric conditions are set at a temperature of 20 °C and a pressure of 1.0 bara. These parameters provide the boundary conditions for assessing hydrogen dispersion behaviour, particularly focusing on gas accumulation near the ventilation fan outlet and evaluating potential flammability risks on the weather deck.

Table 16. Input Parameters for hydrogen dispersion simulation.

Parameter	Value
Cargo compressor room volume	268.8 m ³
Wind speed	1 m/s (Pasquill stability class F) 6 m/s (Pasquill stability class D)
Wind direction	180 degree (Bow to Stern)
Ship speed	0 m/s
Release rate	0.19 kg/s
Released gas	100% H ₂
Atmospheric temperature	20°C
Atmospheric pressure	1.0 bara

8.4.MODEL SETUP

8.4.1. COMPUTATIONAL DOMAIN

FLACS-CFD simulation employs a three-dimensional Cartesian grid consisting of cubic or rectangular cells to simulate hydrogen dispersion from the cargo compressor room ventilation system. The computational domain is divided into a high-resolution core grid region and a stretched grid region, designed to achieve a balance between localized accuracy near the hydrogen release point and computational efficiency in the extended domain.

As shown in Figure 48, the core region (highlighted in red) is densely meshed around the ventilation fan outlet of the cargo compressor room, enabling high-resolution modelling of gas behaviour at the source. Three perspectives—side, front, and bottom views—clearly show how the mesh transitions outward from this high-resolution region into coarser grids as distance from the source increases.

According to Table 17, for both Case 1 (1 m/s wind speed) and Case 2 (6 m/s wind speed), the core grid region is consistently defined, extending from 130.05 m to 133.94 m in the X-direction, –7.84 m to –5.20 m in the Y-direction, and 30.27 m to 34.00 m in the Z-direction. This narrow and focused mesh window ensures detailed resolution of the hydrogen release flow field within the ventilation duct vicinity.

Beyond the core, the stretched grid region is applied to cover a broader area of potential dispersion, ranging from 100.00 m to 140.00 m in X, –17.88 m to 4.80 m in Y, and 0.00 m to 60.00 m in Z. This domain setup captures the full extent of the hydrogen plume while minimising computational load. The maximum stretching factor used in this region is 1.2, and the largest cell size is limited to 2.2 m, maintaining appropriate resolution to detect flammable gas concentrations in the far field.

This domain configuration provides a localised and efficient modelling environment tailored to the specific hydrogen release scenario from the cargo compressor room ventilation outlet, where the steady-state hydrogen release rate is approximately 0.19 kg/s. The defined computational space is sufficient to capture plume behaviours under both low and high wind conditions without unnecessary extension beyond the region of interest.

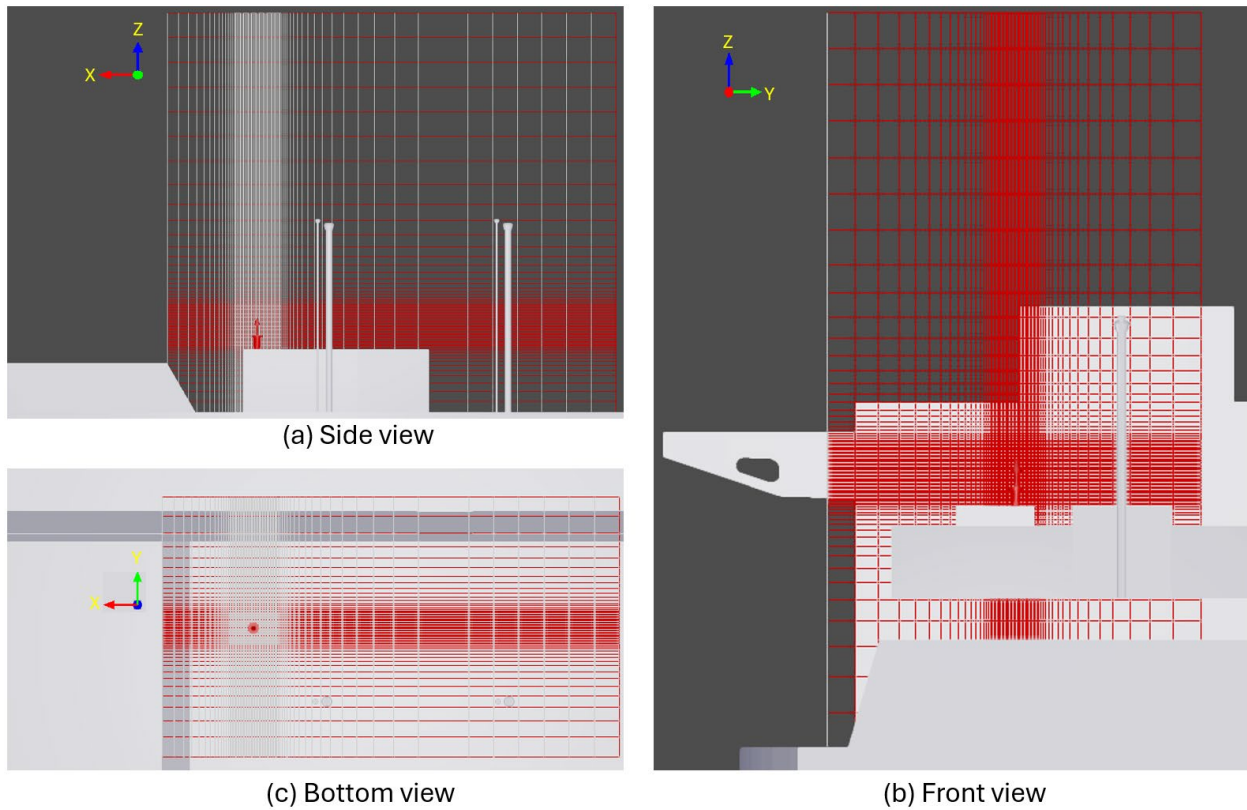


Figure 48. Computational domain for hydrogen release simulation.

Table 17. Computational domain for core and stretched grid regions.

		Case 1		Case 2	
Leak rate [kg/s]		0.19 kg/s		0.19 kg/s	
Wind speed [m/s]		1 m/s		6 m/s	
	Grid Coordinates	Minimum [m]	Maximum [m]	Minimum [m]	Maximum [m]
Core grid region	X	130.05	133.94	130.05	133.94
	Y	-7.84	-5.20	-7.84	-5.20
	Z	30.27	34	30.27	34
Stretched grid region	X	100	140	100	140
	Y	-17.88	4.80	-17.88	4.80
	Z	0	60	0	60

8.4.2. BOUNDARY CONDITIONS

This simulation also focuses on the H₂ dispersion study. Refer to Chapter 6.4.2, *Boundary conditions*.

8.5.SIMULATION RESULTS

As depicted in Figure 49, Case 1 models hydrogen dispersion under low-wind conditions with a wind speed of 1 m/s. Under this scenario, the released hydrogen plume remains relatively concentrated and rises upward due to buoyancy, with minimal horizontal deflection. The maximum horizontal dispersion from the vent outlet was approximately 6.4 metres. The plume geometry indicates localized accumulation near the weather deck exhaust area, posing potential risks to nearby elevated structures or confined spaces. The dominant vertical trajectory reflects the weak wind influence and underscores the importance of implementing vertical gas detection strategies under calm conditions.

Case 2 assesses hydrogen dispersion under high-wind conditions with a wind speed of 6 m/s, assuming the same hydrogen release rate. Unlike the low-wind case, the plume exhibits a markedly different behaviour, becoming horizontally elongated due to strong wind advection. Hydrogen disperses approximately 16 meters downwind from the vent outlet, with reduced vertical rise. This indicates that stronger winds enhance horizontal transport, enlarging the affected area and shifting the flammable zone farther away from the release point. These results highlight the critical need to account for wind conditions in safety assessments and gas detection system designs, particularly in exposed weather deck environments.

Both cases clearly demonstrate how ambient wind conditions influence hydrogen behaviour during a release event. Low wind conditions favour vertical accumulation near the source, while higher wind speeds lead to wider horizontal dispersion and broader hazard zones. These effects must be considered in the design of ventilation systems and risk mitigation strategies. As shown in the simulations, the size and shape of the hydrogen plume vary significantly with wind speed. Specifically, in Case 2, where the plume extends up to 16 metres, it is essential to ensure that within a 16 metre radius around the ventilation fan of the cargo compressor room, explosion-proof equipment is installed and no potential ignition sources are present.

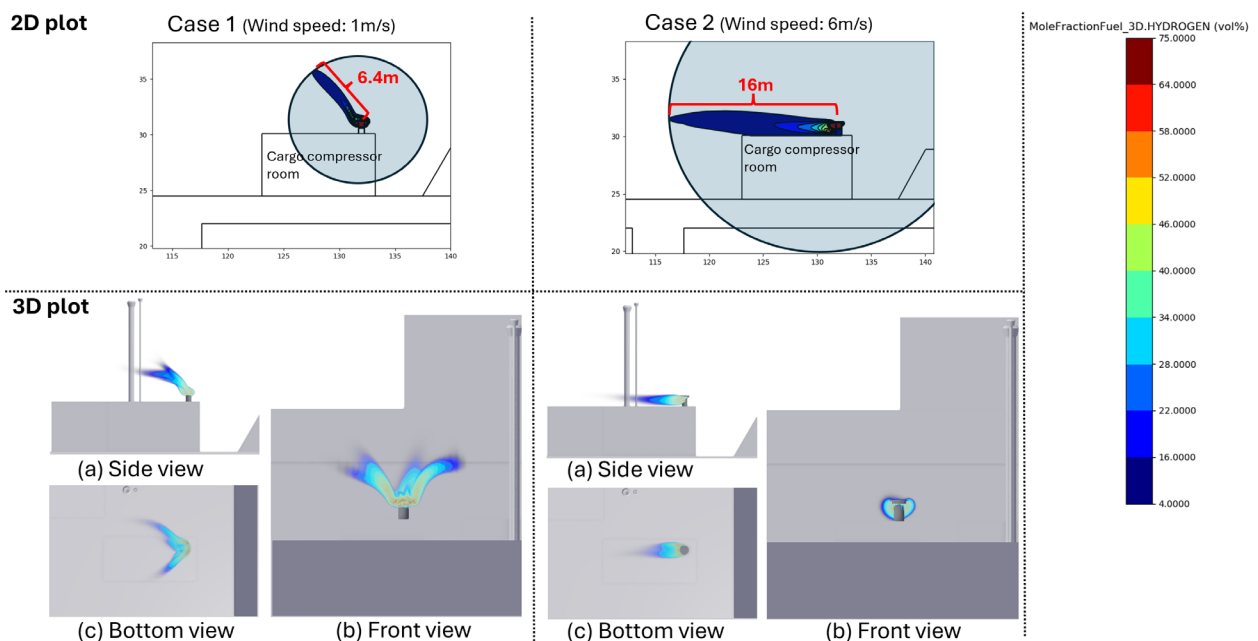


Figure 49. 2D and 3D Dispersion Profiles of Hydrogen Gas Released from ventilation fan.

9. SCENARIO #8 - SAFETY ZONES FOR CARGO OPERATION

9.1.BACKGROUND

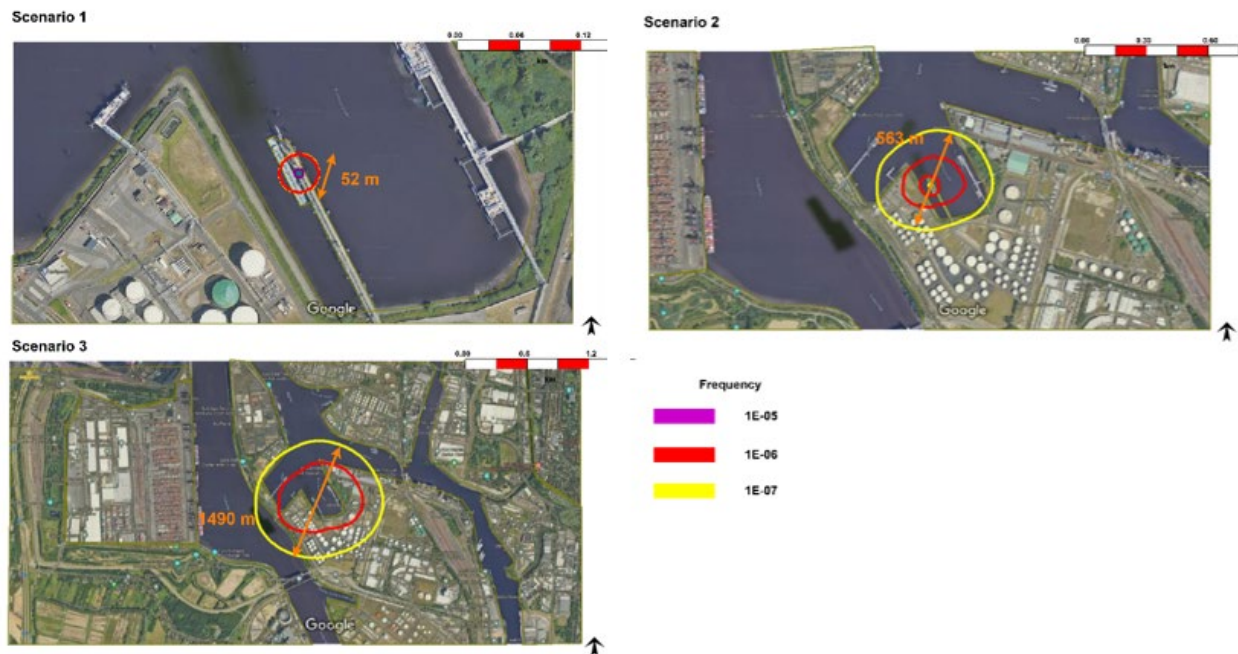


Figure 50. Overview of scenario #8.

The objective of this scenario is to perform a liquified hydrogen dispersion analysis, including fire and explosion events, during the cargo operation and the resulting impact on the port area. The case studies identified from the qualitative risk assessment (HAZID), are as follows:

- ‘Evaluate the impact of liquid hydrogen cargo leaked and dispersed to the port area and define the cargo operation safety zone’
- ‘Evaluate the fatalities within the defined cargo operation safety zone and provide mitigations according to the results.’

In other words, this study aims to define the boundaries that may lead to fatalities caused by leaked liquid hydrogen during cargo operations.

9.2.MODEL VALIDATION

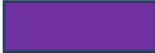
9.2.1. STORAGE OF LIQUID HYDROGEN

In this study, hydrogen is to be supplied in liquid form between shore-based facilities and the demonstration vessel. During cargo operations, the vessel is berthed at the port, and five transfer hoses are connected—four for liquid hydrogen supply and one for vapour return. To prevent the direct release of hydrogen vapour into the atmosphere and to avoid overpressure in the cargo tanks during both loading and unloading, vaporised hydrogen from the ship is returned to the shore facility.

The liquid hydrogen is maintained at a pressure of 0.10 bar and a temperature of -253°C, while the storage and supply conditions are assumed to be 0.15 bar at the same temperature. The returned hydrogen vapour is considered to be at -100°C and 0.5 bar. Given hydrogen's highly flammable and explosive nature, this study includes a risk assessment focusing on the potential fatalities among crew members and third parties due to fire and explosion resulting from hydrogen leaks.

Based on the defined cargo operation scenario, risk analysis was conducted using DNV's SAFETI software. The quantitative risk assessment (QRA) applied risk acceptance criteria for third-party individual risk in accordance with the standards used in DNV risk assessment reports, as presented in Table 18.

Table 18. Summary of DSB risk tolerance criteria [14]

Consideration zones	Individual risk up to (per year)	Descriptions
Inner zone 	1E-05	This is basically the business' own area. In addition, for example, LNF area (Landbruks-, natur- og friluftsområder) can be included in the inner zone. Only short-term passage for third parties.
Middle zone 	1E-06	Public road, rail, dock and similar. Permanent industry and office can also be found here. In this zone, there should not be accommodation or housing. Scattered housing can be accepted in some cases.
Outer zone 	1E-07	Areas regulated for residential purposes and other uses of the general population can be included in the outer zone, including shops and smaller accommodations.
Outside outer zone	Not Defined	Schools, kindergartens, nursing homes, hospitals and similar institutions, shopping centres, hotels or large public arenas must normally be placed outside the outer zone.

9.2.2. POTENTIAL CARGO OPERATION LOCATION

This study presents potential hydrogen cargo operations available at ports regardless of the continents. Europe - Africa route is chosen for target ports, which have high possibility of hydrogen cargo operation ports. The most prominent example is the case of the potential trade from North Africa to Europe. Especially, the port of Rotterdam, the Port Authority is working with various partners towards the introduction of a large-scale hydrogen network across the port complex, making Rotterdam an international hub for hydrogen production, import, application and transport to other countries in Northwest Europe. The Port of Rotterdam Authority, Invest International, Gasunie and Hyphen and the Namibian Implementation Authority Office, Namport, Nampower signed a MoU (Memorandum of Understanding) where the organisations confirmed their intent to work together to develop a cooperation agreement for green hydrogen related infrastructure that will lead to new hydrogen supply chains from Luderiz to Rotterdam and its hinterland. As such, the port of Rotterdam and the port of Luderiz would be case locations in this study to define the impacts caused by hydrogen leaks during cargo operation locations.

The differences in weather conditions, such as wind speed and direction, population density, and distance from the city centre area, will show the various cases of impacts of hydrogen leakage accidents.



Figure 51. Potential cargo operation location and wind rose (Port of Rotterdam)

Rotterdam covers an area of 217.6 square kilometres and has a population of approximately 664,071, resulting in an average population density of about 3,052 people per square kilometre. As shown in Figure 51, the proposed hydrogen cargo operation site in Rotterdam is located at a relatively greater distance from the city centre. This allows for increased flexibility in selecting a suitable site for hydrogen cargo operations. Due to the defined boundaries and potential accident impact zones in this area, the consequences of an incident may be less severe compared to other locations.

Regarding meteorological conditions, Rotterdam experiences prevailing annual winds predominantly from the southwest, with frequent wind directions including SW, WSW, and SSW. These directions account for nearly 30% of the 16 compass points in the wind rose. Most wind speeds observed in the area range between 4.25 mph and 6.48 mph, as illustrated in Figure 51.

Table 19. Population density with weighting factor (Port of Rotterdam).

Area	Weighting factor	Population density (per sq km)
City centre area	120%	3,662.4
Residential area	100%	3,052
Working area	100%	3,052
Harbour area	50%	1,526
Car parking area	10%	305.2
Bulk storage area	10%	305.2

The weighting factor applied to the Port of Rotterdam is presented in Table 19. This factor plays a crucial role in estimating potential fatalities resulting from accidents, highlighting the importance of selecting a cargo operation site that is situated away from densely populated areas.

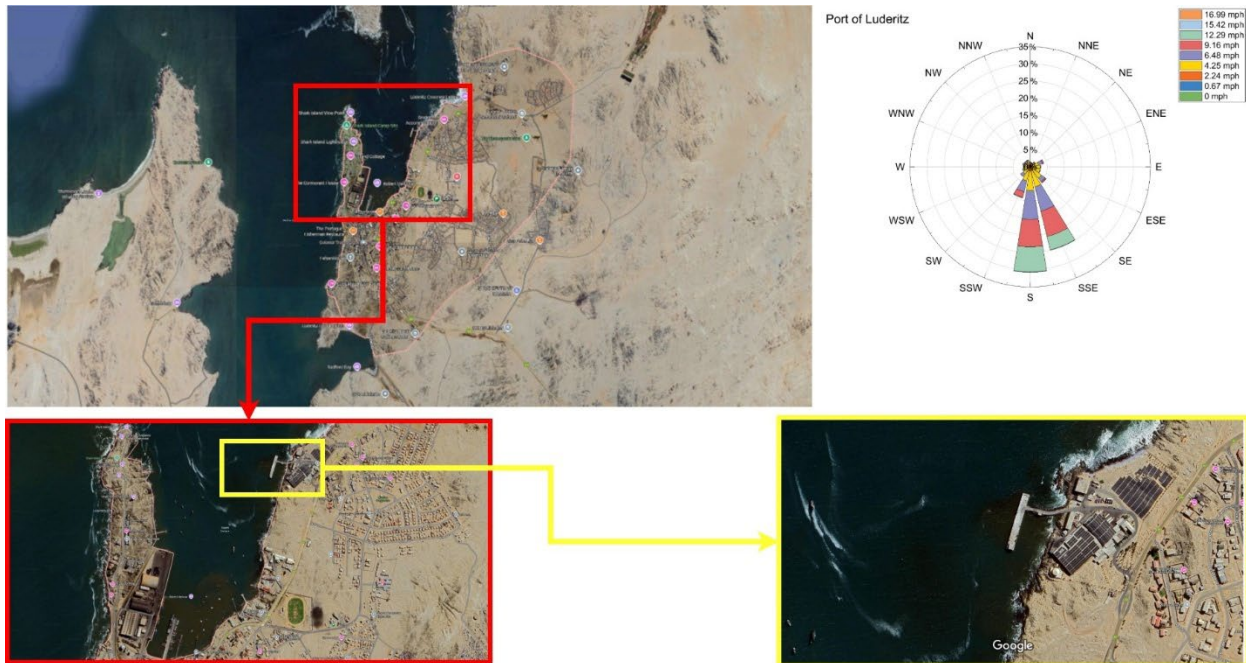


Figure 52. Potential cargo operation location and wind rose (Port of Luderitz).

Lüderitz has a population of approximately 16,156 spread across an area of 14.2 square kilometres, resulting in an average population density of about 1,140 people per square kilometre. As illustrated in Figure 52, the proposed hydrogen cargo operation site is situated relatively close to the city centre. This proximity necessitates careful consideration when selecting an appropriate location for hydrogen handling activities. Given the short distance and the potential extent of accident impact zones, any incident in this area could result in more severe consequences compared to other sites.

In terms of meteorological conditions, Lüderitz experiences predominantly southerly winds throughout the year, with the most frequent directions being south (S) and south-southeast (SSE). These two directions alone account for approximately 55% of the wind rose's 16 compass points. Typical wind speeds in the region range between 6.48 mph, 9.16 mph, and 12.29 mph, as depicted in Figure 52.

Table 20. Population density with weighting factor (Port of Luderitz).

Area	Weighting factor	Population density (per sq km)
City centre area	120%	1,368
Residential area	100%	1,140
Working area	100%	1,140
Harbour area	50%	570
Car parking area	10%	114
Bulk storage area	10%	114

The same weighting factor ratio used for the Port of Rotterdam is applied in Table 20. While Lüderitz shows comparatively lower values across all categories—primarily due to its smaller overall population—the close proximity between the cargo operation site and residential areas may lead to more severe consequences in the event of an incident.

9.3.MODEL SCENARIOS AND CONDITIONS

9.3.1. CARGO OPERATION SCENARIO

The voyage duration between the Port of Rotterdam in the Netherlands and the Port of Lüderitz in Namibia is estimated to be 17 days. Given a boil-off gas (BOG) generation rate of 0.2% per day and a full loading capacity of 152,200 m³, approximately 5,174.8 m³ of BOG is expected during transit, resulting in a net unloading volume of 147,025.2 m³.

Cargo loading is performed by the shore facility at a feed rate of 784,000 kg/h, requiring approximately 13.7 hours to complete the loading of liquid hydrogen. In contrast, cargo unloading is conducted using onboard cargo pumps, which operate at a lower feed rate of 131,000 kg/h, leading to a discharge duration of approximately 79.5 hours.

An additional two hours is allocated for preparatory and completion processes—such as inerting, gassing up, and cooldown—for both loading and unloading operations. Consequently, the total estimated operation time is 330.6 hours for loading and 1,710.7 hours for unloading per annum.

9.3.2. RISK FREQUENCY

The risk frequency calculation is based on the operational schedule and the P&ID provided by the qualitative risk assessment study, which details the cargo transfer volume, duration, and frequency. The cargo handling systems and pipelines are not in continuous operation; they are activated only during loading and unloading activities.

P&ID of the demonstrator ship's cargo piping system is provided in Annex. While both the liquid hydrogen supply line and the gaseous hydrogen return line are depicted. This study specifically focuses on the hydrogen supply line, which consists of various components including pipes, valves, and flanges.

Based on the cargo P&ID, the specific components and their corresponding leak frequencies are summarised in Table 21. The leak frequency for each component is calculated using standard schematic references. To simulate potential leakage scenarios, three representative hole sizes—5 mm, 50 mm, and 150 mm—are considered, reflecting various crack sizes that could lead to hydrogen release.

As previously mentioned, the durations for loading and unloading operations differ. Based on these varying operation times, the leak frequencies for each operation are calculated and presented in Table 22.

Table 21. Frequency of leakage

	Components	Size	EA	Leak frequency (per year)		
				5 mm	50 mm	150 mm
Liquid line	Strainer (cone type)	6 inch	8	9.31E-04	4.82E-04	1.26E-04
	Butterfly valve	20 inch	27	2.06E-04	9.85E-05	2.36E-05
	Globe valve	20 inch	68	2.06E-04	9.85E-05	2.36E-05
	Swing check valve	20 inch	4	2.06E-04	9.85E-05	2.36E-05
	Pressure relief valve	20 inch	19	1.99E-04	6.04E-05	8.82E-06
	Flow meter	20 inch	3	1.99E-04	6.04E-05	8.82E-06
	Strainer (y type)	6 inch	4	9.31E-04	4.82E-04	1.26E-04
	Pipeline	20 inch	400	1.42E-05	5.27E-06	9.59E-07
	Flange	20 inch	200	5.05E-05	1.83E-05	3.23E-06
	Total frequency			2.94E-03	1.40E-03	3.44E-04
Gas line	Strainer (cone type)	6 inch	2	9.31E-04	4.82E-04	1.26E-04
	Butterfly valve	20 inch	20	2.06E-04	9.85E-05	2.36E-05
	Globe valve	20 inch	12	2.06E-04	9.85E-05	2.36E-05
	Swing check valve	20 inch	4	2.06E-04	9.85E-05	2.36E-05
	Pressure relief valve	20 inch	12	1.99E-04	6.04E-05	8.82E-06
	Flow meter	20 inch	3	1.99E-04	6.04E-05	8.82E-06
	Ball valve	20 inch	8	2.06E-04	9.85E-05	2.36E-05
	HD Compressor (Reciprocating)	6 inch	2	1.58E-02	6.97E-03	1.53E-03
	LH2 Vaporizer (Tube side)	6 inch	1	7.73E-04	3.55E-04	8.14E-05
	HD heater (Tube side)	6 inch	1	7.73E-04	3.55E-04	8.14E-05
	Pipeline	20 inch	150	1.42E-05	5.27E-06	9.59E-07
	Flange	20 inch	80	5.05E-05	1.83E-05	3.23E-06
	Total frequency			1.96E-02	8.70E-03	1.94E-03

Table 22. Leak frequency per operation

Leak frequency	Loading operation			Unloading operation		
	5 mm	50 mm	150 mm	5 mm	50 mm	150 mm
Liquid	1.95E-03	8.54E-04	1.91E-04	1.01E-02	4.42E-03	9.90E-04
Gas	2.01E-03	8.72E-04	1.91E-04	1.04E-02	4.51E-03	9.86E-04

9.3.3. LEAKAGE QUANTITY

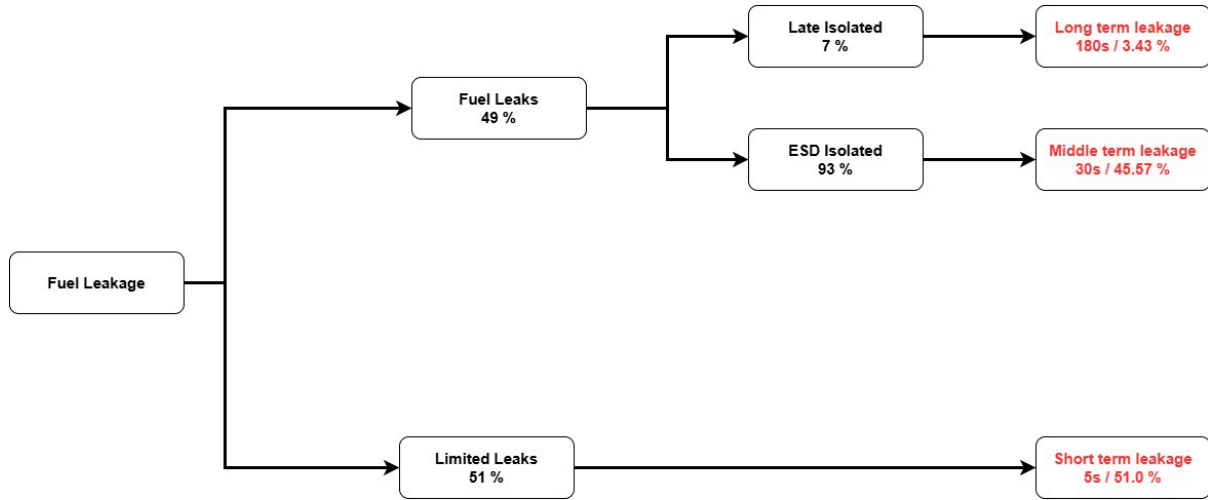


Figure 53. Fuel leakage event tree

The event tree for fuel leakage scenarios, illustrated in Figure 53, is developed by the researcher based on leakage scenarios outlined by DNV [15]. In the event of a leak during cargo operation, sensors are assumed to detect the leak and initiate an automatic shutdown within 5 seconds (short-term leakage). If this system fails, the crew or watchkeeper must manually activate the emergency shutdown, which takes approximately 30 seconds (medium-term leakage). Should the emergency shutdown also fail, a full manual shutdown process is required, taking up to 180 seconds (long-term leakage). By accounting for the three possible leakage durations and their associated probabilities, the average leakage duration is calculated to be 22.40 seconds.

Three leak hole sizes—5 mm, 50 mm, and 150 mm—are considered, representing small cracks, standard failures, and catastrophic ruptures, respectively. The leak frequencies associated with each hole size are treated as independent events, meaning leakage incidents may involve one or more hole sizes simultaneously. This approach allows for a broader assessment of possible accident scenarios and consequences.

The fuel leakage rate varies depending on the fuel's density (kg/m^3), the size of the leak, and the initial pressure. A simplified leakage rate equation referenced from DNV is applied. This study uses separate equations for liquid and gaseous fuels to accurately reflect the physical behaviour of each state.

$$Q_L = 2.1 \times 10^{-4} d^2 \sqrt{\rho_L P_L} \quad (1) \quad [15]$$

where Q_L represents the liquid fuel leakage rate, d is the hole diameter (mm), ρ_L is the liquid density (kg/m^3), and P_L is the initial pressure of the liquid (bar gauge).

For liquid hydrogen, a density of 70.8 kg/m^3 is assumed under conditions of -253°C and 0.15 bar [16].

$$Q_g = 1.4 \times 10^{-4} d^2 \sqrt{\rho_g P_g} \quad (2) \quad [15]$$

where Q_g represents the liquid fuel leakage rate, d is the hole diameter (mm), ρ_g is the initial density of gas (kg/m^3), and P_g is the initial pressure of gas (bar gauge).

For gaseous hydrogen, a density of 39.75 kg/m^3 is assumed at -100°C and 0.5 bar [17]. From calculated leakage rate for loading and unloading operation, the total leakage quantity per hole size with general leakage time is calculated in Table 23.

Table 23. Fuel leaked quantity per operation

	Leakage quantity per hole size (d) (kg)		
	5 mm	50 mm	150 mm
Loading operation	0.38	38.32	344.84
Unloading operation	0.35	34.94	314.50

9.4.SIMULATION RESULTS

DNV's Safeti software was utilised to model and calculate the impact range, producing detailed results overlaid on a geographical map that integrates both frequency and consequence analyses. The consequence analysis defines accident boundaries on the map, indicating areas where fatalities are likely to occur due to fires or explosions resulting from hydrogen leaks. Each accident scenario is represented by a distinct boundary line, with different colours indicating varying cases. The accident frequencies are categorised by occurrence rates during specific time periods. In this study, the analysis focuses on three frequency thresholds— $1\text{E-}05$, $1\text{E-}06$, and $1\text{E-}07$ —as outlined in Table 18.

9.4.1. LOADING OPERATION

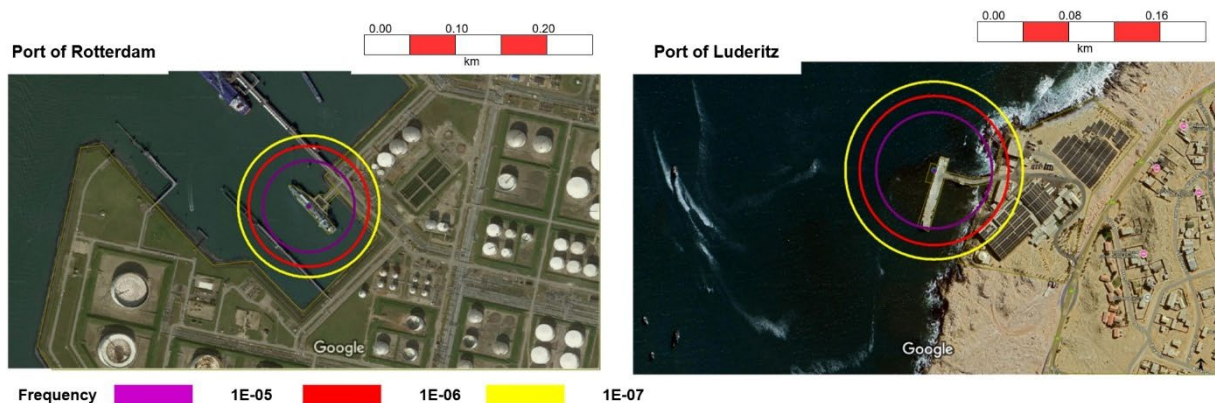


Figure 54. Simulation results for loading operation.

Table 24 summarises the zonal distribution derived from the results presented in Figure 54. Overall, the two sets of results show similar patterns both in the Port of Rotterdam and the Port of Luderitz. A large-scale hydrogen leak poses a significant risk of fire and explosion when released into the atmosphere. The inner zone extends to approximately 170 metres from the source, the middle zone reaches up to 220 metres, and the outer zone extends beyond 260 metres at both potential locations.

Table 24. Detailed range for simulation results (loading operation)

Location	Zone	Range (m)	Location	Zone	Range (m)
Port of Rotterdam	Inner zone	171.6	Port of Luderitz	Inner zone	170
	Middle zone	224.82		Middle zone	218.76
	Outer zone	263.77		Outer zone	262.73

Additionally, an FN curve for each port is presented in Figure 55. An FN diagram, also known as a Farmer diagram, is a graphical tool used to represent societal risk associated with a specific technology, activity, or project. The vertical axis (F) indicates the frequency of events, while the horizontal axis (N) shows the number of fatalities. Each point (F, N) on the curve represents the cumulative frequency of incidents resulting in N or more fatalities.

In the diagram, the green line represents the calculated number of fatalities, while the yellow line indicates the minimum acceptable risk threshold. The red line marks the maximum tolerable risk level, beyond which additional safety measures or barriers are required. The FN curve provides a more practical evaluation of risk acceptability than safety zone boundaries alone.

For both the Port of Rotterdam and the Port of Luderitz, the FN curves for loading operations exceed the maximum criterion line, indicating an unacceptable level of risk. Although Rotterdam has a population density roughly three times higher than Luderitz, the proximity of the cargo operation site to residential areas in Luderitz results in a comparable severity level.

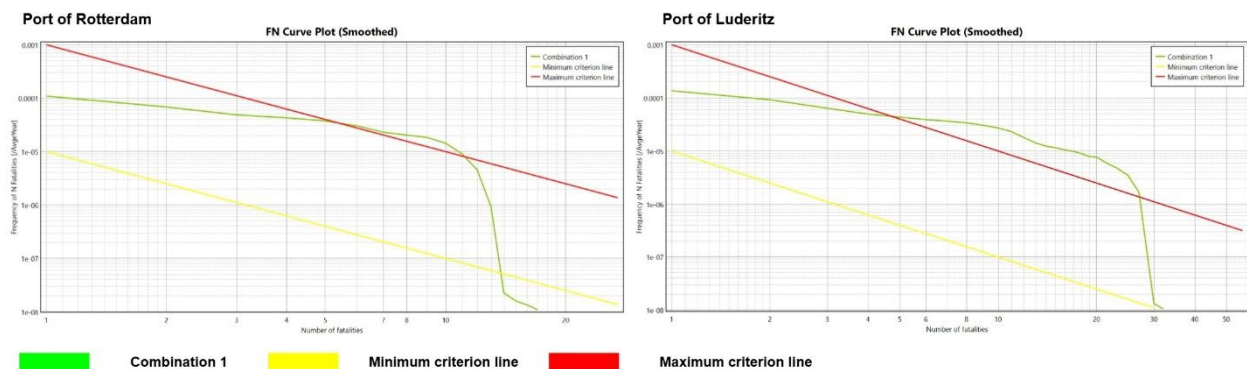


Figure 55. FN Curve result for loading operation.

9.4.2. UNLOADING OPERATION

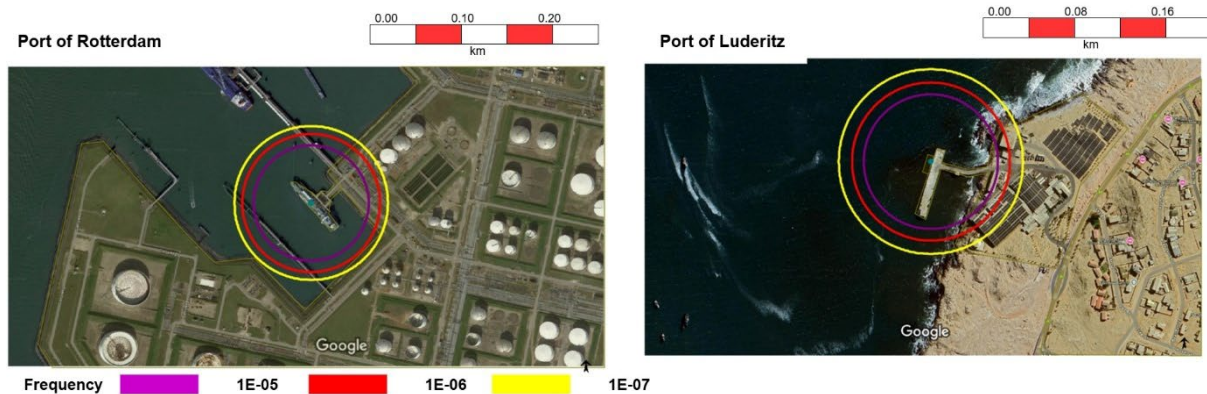


Figure 56. Simulation results for unloading operation.

Table 25 provides a summary of the zonal distribution based on the results shown in Figure 56. In both the Port of Rotterdam and the Port of Luderitz, the results for unloading operations are notably more extensive than those for loading operations. This is primarily due to the unloading operation time being approximately five times longer, which leads to more severe consequences. The inner zone extends up to 210 metres—an increase of 40 metres compared to the loading scenario. For the middle zone, the expansion is around 30 metres, reaching up to 253.05 metres in the case of the Port of Rotterdam. In contrast, the outer zone shows a relatively smaller increase of 20 metres, exceeding 280 metres in total.

Table 25. Detailed range for simulation results (unloading operation)

Location	Zone	Range (m)	Location	Zone	Range (m)
Port of Rotterdam	Inner zone	210.13	Port of Luderitz	Inner zone	208.75
	Middle zone	253.05		Middle zone	243.21
	Outer zone	284.48		Outer zone	287.31

At both the Port of Rotterdam and the Port of Luderitz, as shown in Figure 57, the FN curves for loading operations surpass the maximum criterion line, signifying a critical and unacceptable level of societal risk. Moreover, unloading operations clearly present broader accident impact zones than loading operations. These findings highlight the necessity for enhanced safety measures, particularly during extended unloading operations, to ensure risk remains within acceptable thresholds.

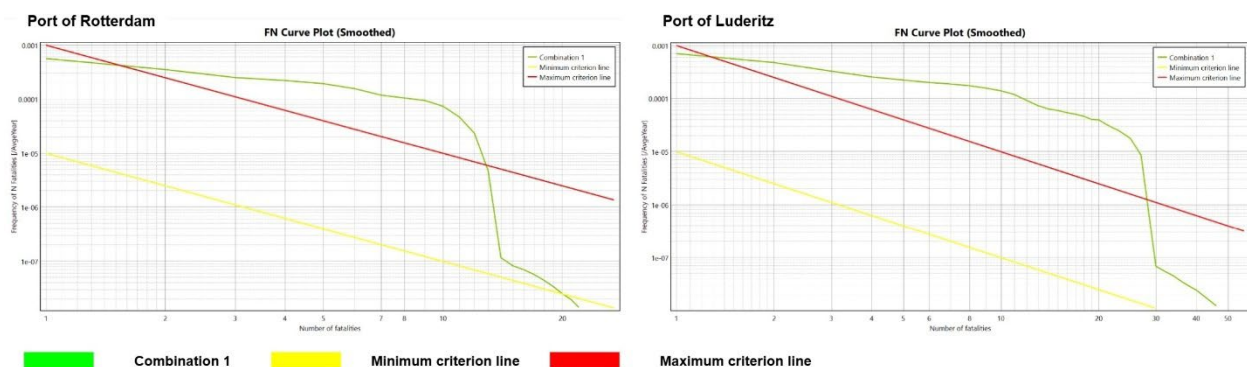


Figure 57. FN Curve result for unloading operation

10. CONCLUSION

This study systematically addressed the dispersion, flammability, and explosion risks associated with hydrogen venting and accidental release scenarios by employing validated CFD methodologies. Full-scale simulations using FLACS and DNV-Phast were conducted to assess how hydrogen's physicochemical properties—such as high diffusivity, and broad flammable and detonable concentration ranges—can rapidly lead to the formation of hazardous atmospheres. The integrated application of these modelling tools provided critical insights into the spatial and temporal evolution of flammable gas clouds, ignition probabilities, and explosion overpressures, thereby enabling data-driven design and safety decisions tailored to LH₂ carrier operations.

The main findings for each scenario are summarised below.

Scenario #1

The analysis of jet fire behaviour from Vent Mast No.3 and No.4 on an LH₂ carrier demonstrates that the current vent mast layout maintains a safe buffer from critical structures, even under conservative fire conditions.

- Under low wind speed (1 m/s), flames rise vertically with limited horizontal spread, concentrating thermal loads near the vent mast.
- As wind speed increases to 6 and 10 m/s, flames are deflected downstream, extending horizontally but remaining clear of the accommodation block.
- Dual venting cases show delayed ignition at Vent Mast No.4 due to thermal interaction, however, flame tips do not reach habitable zones.

These results confirm that vent mast spacing, and orientation effectively prevent flame impingement on critical areas, supporting the adequacy of current fire safety design.

Scenario #2

The analysis of hydrogen explosion within the cargo compressor room reveals that internal gas accumulation poses significant risks depending on concentration levels.

- At 5% hydrogen, combustion is mild, with negligible overpressure and thermal effects.
- At 10%, peak pressure reaches ~0.5 bar and temperature exceeds 1100 K, indicating moderate deflagration.
- At 18%, the explosion produces ~4.5 bar peak pressure and temperatures over 2200 K within one second, suggesting severe structural and thermal impact.

These findings highlight the importance of detection, ventilation and concentration control in enclosed spaces to prevent escalation of explosion hazards.

Scenario #3

The analysis of jet fire resulting from PRV leakage near the cargo compressed room demonstrates that spatial separation is critical in mitigating thermal impact.

- A 2 mm hydrogen leak at 0.5 m from the wall generates heat flux exceeding 500 kW/m² within 7 seconds, indicating high risk of structural damage.
- When the leak point is moved to 2.0 m, the maximum heat flux drops below 3 kW/m², with minimal thermal exposure across the wall surface.
- The contrast between the two cases underscores the sensitivity of flame impingement to leak proximity and supports the establishment of minimum clearance zones.

These results provide a technical basis for refining PRV placement and protective measures around sensitive shipboard structures.

Scenario #4

The analysis of LH₂ spilling onto the seawater surface, resulting from collision or tank damage, focuses on the size, shape, and location of the flammable gas cloud, as well as the characteristics of the resulting liquid pool.

- The pool depth and shape exhibit quasi-steady-state behaviour, aligning with theoretical predictions and experimental findings.
- The location of the LH₂ pool significantly affects the spatial distribution of the gas cloud around the vessel. When the ship moves parallel to the wind direction, this effect is considerably reduced.
- Gas cloud distribution is sensitive to wind speed and LH₂ release rate.
- Low wind speeds promote vertical formation of the hydrogen cloud, resulting in higher concentrations and longer persistence. High wind speeds lead to a more horizontal, widespread cloud with lower concentrations and faster dispersion.
- Larger LH₂ pool sizes correspond to higher release rates, producing a larger affected area and extending the presence of the hydrogen cloud.

This simulation study clarifies the distinct behaviour of LH₂ spills on the sea surface, which differs notably from that of other cryogenic liquids such as LNG and LN₂. The findings support the definition of hazard zones, safety standoff distances, and critical response times in emergency planning.

Scenario #5

The analysis of simultaneous hydrogen release from No.3 and No.4 vent masts on an LH₂ carrier demonstrates that the current vent mast layout maintains a safe buffer from critical structures, even under conservative dispersion conditions.

- These simulations reveal that under low wind speed (1 m/s), the hydrogen cloud disperses horizontally up to 371 meters, with a vertical rise of 8 metres, maintaining a 13.5 metre distance from the accommodation block.
- In higher wind conditions (6 m/s), dispersion is significantly reduced, with the cloud extending 146 metres horizontally and reaching only 5 metres vertically, while the closest distance to the accommodation block narrows to 9 metres.
- Despite the reduced proximity under high wind, the cloud remains safely separated from both the accommodation and funnel structures, avoiding critical ignition risk.

These results support the adequacy of the current vent mast arrangement and provide a technical basis for validating design safety while informing potential enhancements for future vessel configurations.

Scenario #6

The analysis of hydrogen dispersion from the No.4 vent mast on an LH₂ carrier shows that wind speed significantly influences the spread and proximity of the gas cloud to critical shipboard structures.

- Under low wind conditions (1 m/s), the hydrogen cloud disperses up to 252 metres horizontally, maintaining a 12-metre distance from both the accommodation block and the funnel.
- With stronger wind (6 m/s), the dispersion is reduced to 76 metres, and the minimum distance to the accommodation and funnel drops to 8 metres, indicating higher local concentration despite a smaller dispersion footprint.

These results confirm that the current vent mast configuration maintains a safe buffer. However, the reduced clearance under high wind speeds highlights the importance of incorporating wind variability, exhaust height, and vent placement into the design phase to ensure robust protection of crew-occupied areas and critical structures.

Scenario #7

The analysis of hydrogen dispersion under varying wind conditions reveals that wind speed has a decisive impact on plume behaviour and hazard distribution near weather deck vent outlets.

- In low-wind conditions (1 m/s), the hydrogen plume remains concentrated and rises vertically, with a horizontal dispersion of only 6.4 metres, creating localised accumulation near the exhaust area and increasing risks to nearby elevated or enclosed structures.
- In high-wind conditions (6 m/s), the plume is stretched horizontally due to strong wind advection, reaching 16 metres downwind, with reduced vertical lift. This widens the hazardous zone and relocates it farther from the release point.

These findings highlight the necessity of wind-aware risk assessments and gas detection strategies. In particular, a 16-metre radius around the cargo compressor room vent fan should be designated as a hazard zone, requiring explosion-proof equipment and exclusion of ignition sources to ensure safe operations.

Scenario #8

The analysis of hydrogen leakage scenarios during cargo operations reveals that unloading operations pose a greater safety risk compared to loading operations.

- The hazardous zone for loading extends up to approximately 260 metres, while that of unloading exceeds 280 metres due to the significantly longer operation time.

- FN curve assessments indicate that both loading and unloading scenarios surpass the maximum acceptable risk threshold, demonstrating an unacceptable level of societal risk based on the expected number of fatalities within the affected zones.

These findings underscore the need for additional safety measures, such as secondary barriers or mitigation systems, to reduce risk to acceptable levels and ensure safe hydrogen cargo operations at ports.

11. BIBLIOGRAPHY

- [1] Mogi, T., & Horiguchi, S. (2009). Experimental study on the hazards of high-pressure hydrogen jet diffusion flames. *Journal of Loss Prevention in the Process Industries*, 22(1), 45–51.
- [2] Bauwens, C. R., Chaffee, J., & Dorofeev, S. B. (2011). Vented explosion overpressures from combustion of hydrogen and hydrocarbon mixtures. *International Journal of Hydrogen Energy*, 36(3), 2329–2336.
- [3] Tolia, I. C., Venetsanos, A. G., Gyenes, Z., Tchouvelev, A. V., & Markatos, N. C. (2018). Numerical simulations of vented hydrogen deflagration in a medium-scale enclosure. *Journal of Loss Prevention in the Process Industries*, 52, 125–139.
- [4] Verfondern, K., & Dienhart, B. (2007). Pool spreading and vaporization of liquid hydrogen. *International Journal of Hydrogen Energy*, 32(13), 2106–2117.
- [5] Odsæter, L. H., Skarsvåg, H. L., Aursand, E., Ustolin, F., Reigstad, G. A., & Paltrinieri, N. (2021). Liquid hydrogen spills on water—risk and consequences of rapid phase transition. *Energies*, 14(16).
- [6] Verfondern, K., & Dienhart, B. (1997). Experimental and theoretical investigation of liquid hydrogen pool spreading and vaporization. *International Journal of Hydrogen Energy*, 22(7), 649–660.
- [7] Witcofski, R. D., & Chirivella, J. E. (1984). Experimental and analytical analyses of the mechanisms governing the dispersion of flammable clouds formed by liquid hydrogen spills. *International Journal of Hydrogen Energy* (Vol. 9).
- [8] Holborn, P. G., Benson, C. M., & Ingram, J. M. (2020). Modelling hazardous distances for large-scale liquid hydrogen pool releases. *International Journal of Hydrogen Energy*, 45(43), 23851–23871.
- [9] *FLACS-CFD v24.1 User's Manual*. (2025).
- [10] Tian, S., Peng, F., Wang, Z., & Li, J. (2024). Numerical Study of Discharge Coefficients for Side-Damaged Compartments. *Journal of Marine Science and Engineering*, 12(9), 1502.
- [11] Image M/V Hebei Spirit Oil in the Ocean. (2014, July 28). <https://www.whoi.edu/oilinocean/page.do?pid=51892&tid=441&cid=141094&ct=61&article=98109>.
- [12] Friedrich A, Grüne J, Kotchourko N, Kotchourko A, Stern G, Sempert K and Kuznetsov M (2007) Experimental study of jet-formed hydrogen-air mixtures and pressure loads from their deflagrations in low confined surroundings. *International Journal of Hydrogen Energy* 32, 2129–2144.
- [13] Hanna SR, Chang JC and Strimaitis DG (1993) Hazardous gas model evaluation with field observations. *Atmospheric Environment. Part A. General Topics* 27(15), 2265–2285.
- [14] DNV (2021) Ammonia Bunkering of Passenger Vessel – Concept Quantitative Risk Assessment. Det Norske Veritas, Oslo.
- [15] DNV (2013) Failure Frequency Guidance. Det Norske Veritas, Oslo.
- [16] Dagdougui H, Sacile R, Bersani C and Ouammi A (2018) Hydrogen storage and distribution: implementation scenarios. In: *Hydrogen Infrastructure for Energy Applications*, Chapter 4, pp. 37–52.

[17] Kolthoum M, Cantu Medrano VH, Mahmood S and Jean-Marie A (2025) Techno-economic study of green liquid hydrogen production for aircraft refuelling: a case study of Toulouse-Blagnac Airport. SSRN Electronic Journal.

12. ANNEX: DEMONSTRATOR SHIP'S CARGO P&ID