九州大学学術情報リポジトリ
Kyushu University Institutional Repository

Preliminary Risk Assessment of Hydrogen Fuel Release on Storage Facility for Zero Emission Shipping

Nubli, Haris
Department of Marine Convergence Design Engineering, Pukyong National University

Jung Min Sohn
Department of Marine Convergence Design Engineering, Pukyong National University

Muttaqie, Teguh
Research Center for Testing Technology and Standard, National Research and Innovation Agency

Tjolleng, Amir
Department of Industrial Engineering, Faculty of Engineering, Bina Nusantara University

他

https://doi.org/10.5109/7236906

出版情報：Evergreen. 11 (3), pp.2668-2677, 2024-09. Interdisciplinary Graduate School of Engineering Sciences, Kyushu University, Japan
バージョン：
権利関係：Creative Commons Attribution 4.0 International

KYUSHU UNIVERSITY

Preliminary Risk Assessment of Hydrogen Fuel Release on Storage Facility for Zero Emission Shipping

Haris Nubli[1], Jung Min Sohn[1], Teguh Muttaqie[2,*], Amir Tjolleng[3], Yudiawan Fajar Kusuma[4], Abid Paripurna Fuadi[4], Park Sang Hyun[5], Do Quang Thang[6], Aditya Rio Prabowo[7]

[1]Department of Marine Convergence Design Engineering, Pukyong National University, Republic of Korea
[2]Research Center for Testing Technology and Standard, National Research and Innovation Agency, Indonesia
[3]Department of Industrial Engineering, Faculty of Engineering, Bina Nusantara University, Indonesia
[4]Research Center for Hydrodynamics Technology, National Research and Innovation Agency, Indonesia
[5]UlsanLab Inc., Republic of Korea
[6]Department of Naval Architecture and Ocean Engineering, Nha Trang University, Vietnam
[7]Department of Mechanical Engineering, Universitas Sebelas Maret, Indonesia,

*Author to whom correspondence should be addressed:
E-mail: tegu019@brin.go.id

(Received May 13, 2024: Revised June 17, 2024: Accepted August 2, 2024).

Abstract: In response to the emissions reduction plan proposed by the International Maritime Organization (IMO), alternative fuels such as liquefied natural gas (LNG), hydrogen, and ammonia is gaining prominence. Hydrogen is one of alternative fuel due to its applicability in both direct combustion and fuel cells, which are more widely adopted. The potential risk associated with utilizing hydrogen as a fuel source lies in the possibility of equipment failure, leading to an uncontrolled release of flammable gas. In the occurrence of an ignition source, the consequences can be significantly more devastating as hydrogen combustion occurs in a supersonic flow, releasing a higher amount of energy. This paper aims to investigate the preliminary aspects of hydrogen fuel safety. An in-house Computational Fluid Dynamics (CFD) solver, OpenFOAM was employed, to simulate gas release scenarios: vapor cloud explosion (VCE). A gas cloud of 2.0 m^3 was modeled in the numerical platform to predict the overpressure distribution in the surrounding. Finally, this paper presents and discusses the findings from the scenario, which was used as a justification for the safety of hydrogen facilities in the risk of an accidental gas release.

Keywords: gas leakage; hydrogen fuel; escape route; net zero emission; computational fluid dynamics

1.Introduction

Between 2007 and 2014, the maritime sector significantly contributed to global carbon dioxide (CO_2) emissions, accounting for roughly 2.4% of the total annual CO_2 emissions, which equates to 816 million tons of CO_2[1]. During this timeframe, reported data indicated that the annual emissions of nitrogen oxide (NO_X) and sulfur oxide (SO_X) averaged at 18.6 million and 10.6 million tons, respectively, constituting 15% and 13% of worldwide emissions. Responding to these concerning pollution statistics, the IMO introduced the Initial IMO Greenhouse Gas (GHG) Strategy in 2018. This strategy aimed to substantially reduce annual GHG emissions from ships, targeting a minimum 50% reduction by 2050[2]. In 2023, the IMO revised its GHG strategy, setting more ambitious targets and regulations. These new objectives encompass a 20% emissions reduction by 2030, a 70% reduction by 2040, and ultimately achieving net-zero emissions by 2050[3]. It is expected that these GHG regulations will become effective in 2027.

One such alternative energy concept is wind-assisted propulsion systems (WAPS), which utilize aerodynamic forces alongside internal combustion engine power. WAPS have already exhibited fuel savings ranging from 5% to 9% for certain vessels, with the potential for savings up to 25%. However, the effectiveness of WAPS depends on prevailing wind conditions, and concerns persist regarding the efficient utilization of cargo space due to the substantial size of WAPS equipment.

Another alternative involves using ammonia as a fuel source, offering the potential for near-zero CO_2 emissions.

Nevertheless, it is crucial to acknowledge that ammonia usage does not entirely eliminate NO_X emissions. Moreover, ammonia is a synthetic product requiring the utilization of either fossil fuel in its production process. Furthermore, hydrogen (H_2) emerges as a promising alternative fuel for the maritime industry.

During the design phase of hydrogen bunkering, it becomes essential to anticipate the potential outcomes of an accidental gas release. This task can be accomplished through one of two methods: the first involves an experimental approach, which can be notably costly, while the second entails the application of CFD numerical analysis. CFD has emerged as a valuable tool for analyzing gas dispersion patterns, fire, and explosion models[4,5]. Furthermore, it is noteworthy that CFD simulations have undergone rigorous validation and verification processes through experimental validation[6–11].

The adoption of hydrogen as a fuel source in the maritime sector warrants specific attention due to the inherent challenges posed by cryogenic temperatures. Hydrogen, maintained at temperatures as low as -253°C, necessitates specialized storage and transportation in cryogenic tanks. These tanks introduce a particular risk related to brittle fracture due to the ductile-brittle transition temperature characteristics of the steel employed in their construction[12]. This risk is particularly pronounced for ships involved in hydrogen transportation or bunkering, as they face the potential for unintended gas leaks that could not only compromise the structural integrity of the ship but also result in substantial financial losses[13,14]. Addressing this risk requires the development of high-strength or high-manganese steels capable of withstanding prolonged exposure to hydrogen and maintaining their ductile properties at these extremely low temperatures[15,16].

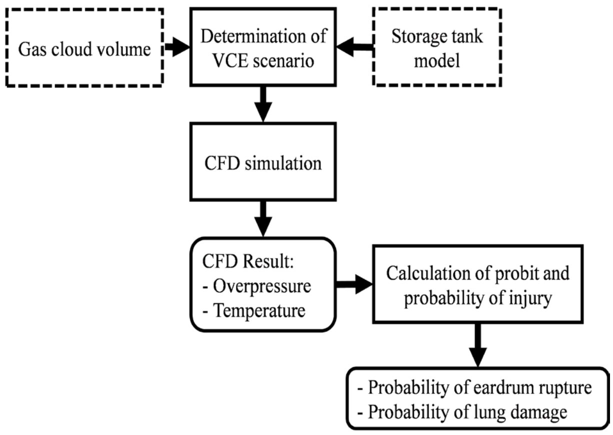


Fig. 1: Framework of the study.

This study aims to examine the initial stage of safety related to hydrogen fuel. To achieve this, we utilized an in-house CFD solver, specifically OpenFOAM, to simulate gas release scenario, particularly focusing on VCE. Within the numerical framework, we constructed a gas cloud with a volume of 2.0 m^3 to predict the distribution of overpressure in the surrounding environment.

Recent research on hydrogen leaks and risk mitigation has encompassed various safety aspects and related technologies. The safety aspects of fuel cell systems, particularly jet fires and vapor cloud explosions[17]. Risk mitigation for hydrogen releases in hydrogen vehicles such as passenger cars and fuel cell vehicle systems also been a focus[18,19]. Flame propagation with various hydrogen mixtures has also been analyzed through experiments[20]. Several instances of leakage at refueling stations have been studied using probabilistic assessments[21,22], dynamic Bayesian network analysis[23,24], and confined spaces in storage compartments[25]. In another model, hydrogen leaks and diffusion from tube fittings have been modeled using the Flame Acceleration Simulator (FLACS) software, with the numerical models validated by experimental data[26]. Collectively, a deeper understanding of the risks and safety measures associated with hydrogen leaks has been offered by these studies, providing valuable insights to enhance the safety of hydrogen technologies.

The distinctive contribution of current study lies in its specific attention to the safety considerations associated with hydrogen fuel, particularly concerning accidental gas releases on storage. The research employs CFD numerical analysis, specifically utilizing OpenFOAM, to model gas release scenarios, placing particular emphasis on vapor VCE. The study's framework, as illustrated in Fig. 1, provides a visual representation of the research methodology.

In summary, this research not only presents outcomes derived from simulated scenarios but also underscores the safety aspects of hydrogen facilities in the event of accidental gas releases. The paper contributes significantly by providing comprehensive insights into the methodology, scenario selection, and CFD analysis, thereby enhancing the understanding of the safety implications of utilizing hydrogen as a fuel source in the maritime industry.

2.Review of Vapor Cloud Explosion

Recently, the application of CFD simulations has emerged as a crucial tool for enhancing safety and mitigating losses in the design of LNG handling facilities[27].

The widely accepted k-ε turbulence model plays a pivotal role in predicting gas dispersion, accounting for factors like atmospheric stability, which significantly impacts gas dispersion and extinction[28–30].

Furthermore, the combustion process can be represented using the eddy dissipation concept, as introduced by Hjertager and Magnussen[31]. This combustion model integrates chemical reactions within turbulent flows. To simulate VCE scenarios, both deterministic and probabilistic approaches with varying parameters can be employed. In contemporary CFD simulations, diverse combinations of flammable gases, including hydrogen and hydrocarbon mixtures, can be accurately model.

Prominent CFD software packages like FLACS and KFX yield a wide array of outcomes, encompassing blast overpressures, air temperatures, drag forces, and more. With regards to blast overpressure, it is noteworthy that the presence of obstructions and congestion around objects can significantly influence the magnitude of overpressure. This phenomenon has been corroborated through sensitivity analyses involving the manipulation of stacked pipe configurations[32]. To estimate the void volume within stacked pipes, a porosity ratio is employed. Lower porosity values correspond to reduced void volumes in stacked pipes, resulting in amplified blast overpressure. The presence of gas-filled void spaces within the explosion zone contributes to the rebound overpressure phenomenon.

3.Risk Assessment Framework and Methodologies Review

The summary of the review methodologies employing CFD as a tool for risk assessment is presented in Table 1. Tauseef et al[33] employed CFD, specifically the realizable *k-ε* turbulence model, to successfully simulate VCEs in a realistic industrial setting. Their study captured the influence of obstacles on flame acceleration and documented an increase in overpressure during flame propagation between tanks.

Table 1. Summary of many references using CFD for VCE

Author (year)	**Tools**	**Solver**	**Object**
Tauseef et. al. (2011)[33]	FLUENT 6.3	*k-ε* turbulence model	Real gas industry environment
Sajid, et. al. (2021)[34]	FLACS	*k-ε* turbulence model	The Amuay refinery and Jaipur oil
Momferatos, et. al. (2022)[35]	SOLVEX & ADREA-HF CFD	*k-ε* turbulence model	Ventilated buildings, highway tunnels with several cars, and methane gas storage cylinders.
Jang, et. al. (2018)[36]	FLACS	*k-ε* turbulence model	A chemical plant's wastewater storage pond in Korea
Duong, et. al. (2023)[37]	FLACS ver. 21.3	*k-ε* turbulence model	Simulate the NH3/LNG leak characteristics during the ship-to-ship bunkering process
Langree, et. al. (2021)[38]	OpenFOAM	Solver XiFoam	The explosion "at rest" case of the EXJET (INERIS GDF-SUEZ)
Mishra, et. al. (2021)[39]	FLACS	*k-ε* turbulence model	The receiving terminal facility
Kim, et. al (2015)[40]	FLACS ver. 10 and Flownex 2014	*k-ε* turbulence model	Gas Fuelled Ship

According to Sajid et al[34], the FLACS CFD simulation tool was employed to gain valuable insights into cloud explosion phenomena. The study conducts in-depth analyses of parameters influencing explosions, providing valuable information to enhance facility design for leak mitigation and understanding worst-case scenario events that could lead to catastrophic accidents.

Momferatos et al[35] leveraged the ADREA-HF CFD code was used to simulate three distinct kinds of vapor cloud explosion studies using various modelled geometry such as, SOLVEX, tunnel experiment and sydGEX experiment. The Reynold Average Navier-Stoke RANS approach is applied using the Kato and Launder modification of the *k- ε* model. The suggested model accurately captures both the physics governing the explosions and the experiment results. Jang, et. al[36] present a detailed analysis of a South Korean wastewater storage pond explosion that tragically claimed six lives and left one person injured, highlighting the research's focus on both the technical aspects and the human impact of the incident. This study seeks to quantify the structural integrity of concrete structures subjected to VCEs, with a twofold objective: analyzing the potential for fracture and identifying the ignition source. This analysis utilizes a coupled CFD-FEA methodology, where CFD accurately models the explosive phenomenon and FEA rigorously evaluates the consequent structural damage. The findings shed light on the accident's mechanism, and recommendations for preventive measures against such incidents are provided.

Duong et. al[37] conducted a CFD simulation of NH3/LNG leakage during ship-to-ship bunkering using FLACS-CFD 21.3 and a *k-ε* turbulence model. Their investigation focused on establishing optimal safety zone and inter-ship spacing under diverse weather conditions. The study's findings reveal a strong correlation between weather and safety requirements, emphasizing the need for dynamic safety protocols that account for changes in leak characteristics and meteorological conditions. Langree et. al[38] in their study involves simulating a methane-air explosion in a confined space, specifically using the "at rest" case of the EXJET (Ineris GDF-Suez) with OpenFoam R. The simulation shows good agreement with the experiment in several aspects such as it successfully replicates the accelerating effect of obstacles, and both flame speed and over-pressure peaks align in magnitude.

Current efforts are focused on refining the transport equation for X to address this aspect, and the results will be updated accordingly. Mishra et. al[39] in the study employed the FLACS-CFD software to simulate the explosion characteristics of LNG-air clouds within a model receiving terminal facility. Kim et. al[40] using FLACS and flownex for prediction VCE in gas fuelled ship. In this research, an investigation into the explosion dynamics of a gas supply machinery room on an LNG-fueled container ship is outlined. The study focuses on the gas fuel system utilized in the high-pressure methane ethane gas injection, specifically analyzing scenarios involving leakage in the dual natural gas supply pipes to the engines. CFD tools are employed to simulate the outcomes of a gas leak, predicting various leak scenarios, sizes of gas clouds, and potential explosion pressures.

4.Probability of Injury

In the field of gas safety, when there's a potential ignition source inside a gas cloud, it can lead to two dangerous outcomes: a fast flash fire or a vapor explosion. Even a short delay of 5 to 10 minutes in ignition can increase the risk of an explosion, as highlighted by Assael in 2010[41]. A gas explosion can release a lot of energy, creating a powerful shockwave that can harm people near the explosion.

Hydrogen gas is of particular concern in this context because it can cause a rapid shockwave known as a "detonation". This shockwave travels at incredibly high speeds, typically around 2,200 meters per second, and generates strong pressure, around 20 bar. Thus, when Vapor Cloud Explosions happen in gas facilities, it need to carefully assess the impact of this strong pressure on persons safety.

The consequences of an explosion on people can be divided into two categories: direct and indirect effects. Direct effects include injuries to sensitive organs or, in severe cases, loss of life. For instance, this could lead to lung damage or ruptured eardrums. On the other hand, indirect effects involve injuries or fatalities caused by flying fragments or the displacement of the entire body. Furthermore, the probability of injury can be expressed as,

$$POI = \left(\frac{1}{2}\right)\left[1 + erf\left(\frac{pr-5}{\sqrt{2}}\right)\right] \quad (1)$$

where *pr* is the probit of lung damage or eardrum rupture under direct effects and *erf* is an error function. The probit of both injuries[21] can be determined as,

$$Pr_{lung} = 5.0 - 5.74 \ln S_{lung} \quad (2)$$

$$S_{lung} = \frac{4.2}{\bar{P}} + \frac{1.3}{\bar{\iota}} \quad (3)$$

$$Pr_{ear} = -12.6 + 1.524 \ln P_s \quad (4)$$

where *Pr* means probit, *Ps* is the peak overpressure, *S* is a variable depending on the type of injury, P_i means the function of scaled overpressure, and *i* denotes the impulse of the explosion. Impulse can be obtained by calculating the area under the curve of the overpressure depending on the time.

5.CFD-based Simulation of VCE

5.1.Technical Reference of BlastFOAM

Turbulence model is widely used to calculate the fluid flow, particularly in the context of CFD method[30]. This model can be applied in the model of RANS which expense a lower resource in the context of the fast simulation time. The standard *k-ε* turbulence model was applied to the BlastFOAM simulation. It consists of two terms such as turbulent kinetic energy (*k*) and turbulent dissipation (*ε*). These can be expressed as follows[42) :

$$k = \frac{1}{2}\overline{u'_\iota}\,\overline{u'_\iota} \quad (5)$$

$$\varepsilon = v\left(\frac{\partial u'_\iota}{\partial x_j}\right)^2 \quad (6)$$

where *u* is the turbulent velocity fluctuation (m/s), *v* is kinematic viscosity (m^2/s), and *x* is Cartesian coordinate. Then, these two terms can be utilized to describe the combustion rate which is called the Eddy Dissipation Concept (EDC)[31]. Here is the basic form of combustion rate[43].

$$R_f = A\bar{\rho}\frac{\varepsilon}{k} min\left(\tilde{Y}_f, \frac{\tilde{Y}_o}{r}\right) \quad (7)$$

where *A* is a constant which is 4.0. *Y* is the specie of both fuel (*f*) and oxygen (*o*). Furthermore, the governing Navier-Stoke equation for combustion rate can be expressed as follows[19] :

$$\frac{\partial(\rho\tilde{Y}_n)}{\partial t} + \frac{\partial(\rho\tilde{Y})}{\partial x_i} = \frac{\partial(-J_{n,i})}{\partial x_i} + R_n \quad (8)$$

$$-J_{n,i} = -\rho D_n \frac{\partial \tilde{Y}_n}{\partial x_i} \quad (9)$$

where subscript *n* is the specie (e.g. CH_4 or H_2), *ρ* is the specie density (kg/m^3) and *D* is mass diffusion.

5.2. Numerical Solver Validation

An experiment was conducted by Bauwens et al.[43], within a test chamber to validate the OpenFOAM software, which is the origin of the BlastFOAM solver by simulating a gas explosion. The chamber contained 63.7 m^3 of methane gas fuel and measured 4.6×4.6×3.0 meters in length, width, and height, respectively. Ventilation was provided through the front wall of the chamber, with a cross-sectional area of 2.7 m^2, equivalent to dimensions of 1.64 meters in length and width. Additionally, an alternative ventilation option was available, with dimensions of 2.32 meters in length and width, totaling 5.4 m2 in cross-sectional area. The ignition points varied, with one positioned at the center (P1) and the other at the back (P2) of the chamber. The center ignition point was situated at coordinates of 2.3×2.3×1.5 meters along the *X*, *Y*, and *Z*-axes, respectively. The back ignition point was placed at coordinates of 0.0×2.3×1.5 meters. Prior to filling the chamber with gas, the ventilation was covered by a thin sheet measuring 0.02 mm in thickness, which would rupture upon explosion. The validation aims to compare the peak overpressure between the experiment and the OpenFOAM results. The peak overpressure is presented in Tables 2 and 3 which gives the conditions and the result of peak overpressures. Overall, both overpressure is exhibiting a good agreement regarding the obtained errors. Thus, it is relatively reliable to use numerical analysis to model a gas explosion.

Table 2. Conditions of several cases

No. Case	Ventilation size (m^2)	Ignition point	Fuel Composition (%)
1	5.4	Center/I1	9.0
2	5.4	Back /I2	9.5
3	2.7	Center/I1	9.2
4	2.7	Back /I2	9.2

Table 3. Result of peak overpressure between the experiment and OpenFOAM simulation

No. Case	Peak Overpressure		Error (%)
	OpenFOAM Pressure (bar)	Experiment Pressure (bar)	
1	0.023	0.022	5.86
2	0.047	0.047	0.32
3	0.027	0.026	5.10
4	0.072	0.079	10.33

5.3. Model and Boundary Condition

A CFD simulation was constructed to depict the scenario of the small storage facility in a port for handling the hydrogen fuel bunkering. The potential hazard in this scenario was pinpointed as a malfunction in the valve, leading to an accidental release of gas. This released gas accumulated in proximity to the storage tanks, resulting in the formation of a gas cloud with a volume of 2.00 cubic meters. The specific parameters of the gas leakage are detailed in Table 4.

Table 4. Specific of gas leakage.

Parameters (unit)	Value
Flow pressure (bar)	5.00
Leak diameter (mm)	101.60
Mass flow rate (kg/s)	15.44
Flow velocity (m/s)	4.45
Leak duration (s)	50.00
Gas cloud volume (m^3)	2.00

Based on the volume of the accumulated gas, we created an equivalent gas cloud with dimensions matching the standard parameters typically used for VCE simulations, including its length, width, and height[44].

Additionally, the chemical reaction was calculated using the irreversible Arrhenius reaction model. This reaction involves the interaction between hydrogen and oxygen, resulting in the formation of water vapor. For this specific case, the reaction constants A, β, and Ta were set at 1.80E+13, 0.0, and 73,745, respectively[45]. All the necessary parameters related to geometry, leakage, thermophysical properties, and the reaction were integrated into our in-house CFD software, OpenFOAM[46].

To present the storage tank geometry within OpenFOAM, a uniform structured Cartesian grid configuration was employed, utilizing cube-shaped elements standardized at 0.5 meter in size. Additionally, the lower surface of the model was configured to simulate concrete ground and tank structure, classified as "wall" type patches within the OpenFOAM framework. The remaining areas, including internal elements, were designated as "empty" type. Furthermore, these "empty" elements were parameterized to simulate the presence of hydrogen gas and air by incorporating gas concentration values. Specifically, the elements allocated for hydrogen gas were set with a gas concentration of 10.0%. A visual representation of this CFD model configuration is exhibited in Fig. 2.

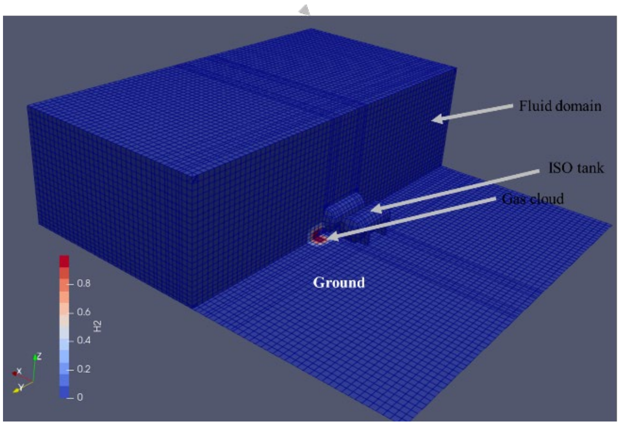


Fig. 2: The CFD model of hydrogen storage tanks and the concentration contour of hydrogen gas (%).

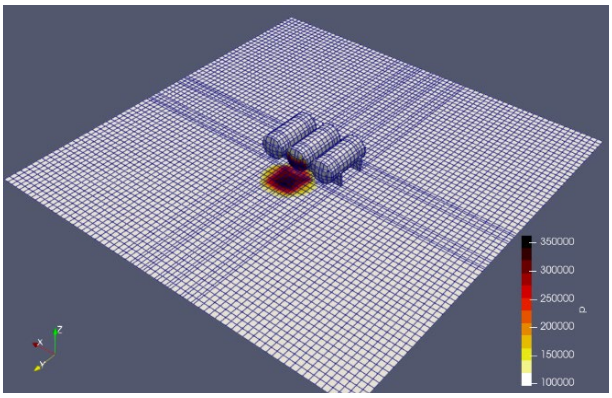
Given the occurrence of VCE at exceptionally high speeds, the time step for the simulation was set at 1.0E-06s for each computational increment. This decision was informed by considerations of element size and flow velocity, ensuring that the Courant number remained well below 1.0, specifically at 3.2E-04. Such important attention to the time step helps maintain the stability and consistency of the CFD simulation results. Furthermore, it is worth noting that the maximum time step was established at 0.01s. This decision was made to account for the rapid dynamics of the explosion event and to capture relevant data with sufficient temporal resolution.

6.Results and Discussion

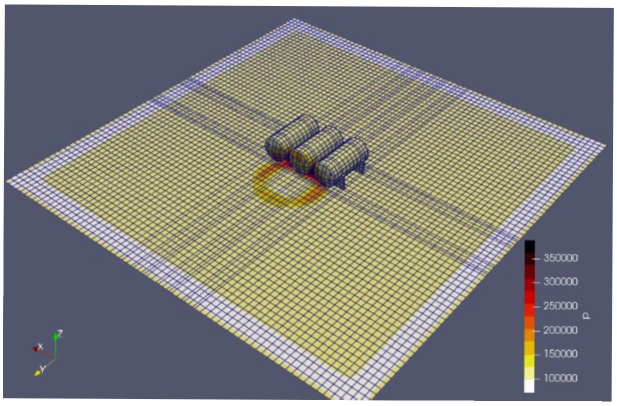
6.1.Overpressure and Temperature

As mentioned previously, the overpressure propagates at very high speeds. Thus, it would expose the storage tank at 0.003s. To check the overpressure propagation, a 3D contour plot was established.

The overpressure plot around the storage tank structure at 0.001s, 0.003s, and 0.01s is shown in Fig 3.



(a)



(b)

Fig. 3: The overpressure plot in Pascal unit (a) at 0.001s, (b) 0.003s, and 0.01s.

At the center of the explosion, the overpressure can reach up to 350,000Pa or 3.5bar at 0.001s. Moreover, at 0.01s, when the outer side of the overpressure propagated to the tanks, the overpressure decreased to around 0.8bar. At this moment, the center of the explosion has a lower pressure than the atmosphere which reaches 0.5bar.

This phenomenon is usually called a negative phase of overpressure, where this low-pressure area then equalizes with the surrounding pressure, eventually. To observe this overpressure value, Fig. 4 shows a time series plot that consists of overpressure in proximity with the middle tank head.

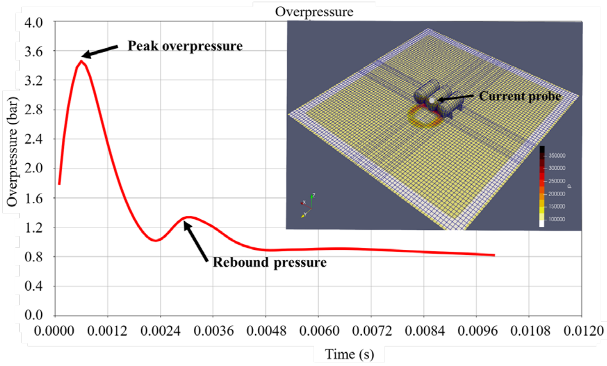
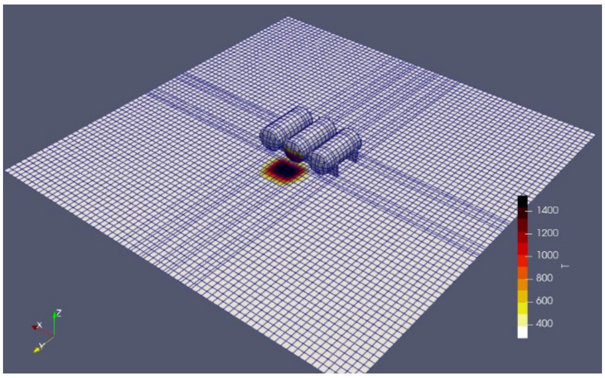
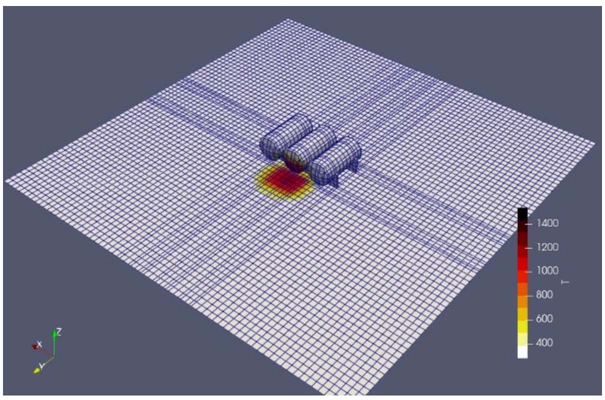


Fig. 4: Time series plots of maximum overpressure near the explosion source



(a)



(b)

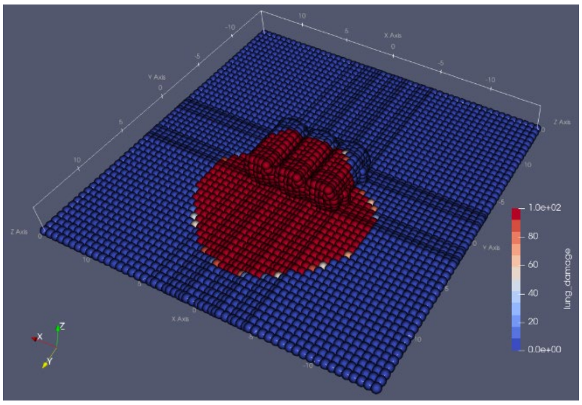
Fig. 5: Plots of temperature contour (a) at 0.001s and (b) 0.003s.

Furthermore, the temperature propagation plots were also observed. It is a different characteristic from the jet fire, the VCE insignificantly propagates the gas temperature. This happened because the explosion occurred at very high-speed. The energy is mostly converted to pressure rather than heat. However, the temperature at the center of the explosion exhibits a high temperature that can reach 1,500K. Here, the plots of temperature contour are exhibited in Fig. 5 at 0.001s and 0.003s.

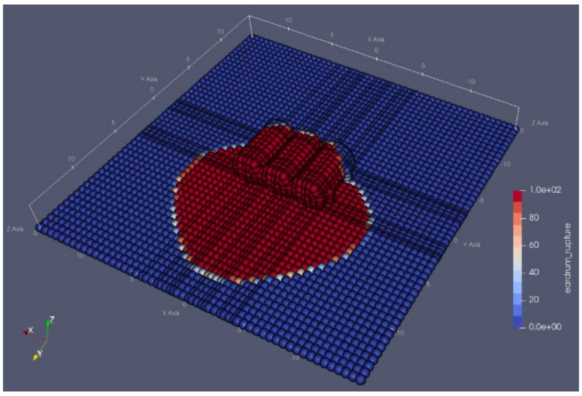
6.2.Estimation of Injury

In this research endeavor, the assessment of injury probability primarily revolves around the impact of overpressure on human safety. Specifically, it encompasses the likelihood of two key injuries: lung damage and eardrum rupture. The outcomes of these probability calculations hold significant implications, influencing decisions related to port design, personnel limitations within exclusion zones, mitigation strategies, evacuation routes, and, most importantly, enhancing safety protocols normal and bunkering operation conditions.

For both categories of injury probability, a percentage unit scale ranging from 0.0% to 100.0% was adopted. Within this framework, the probabilities were visualized, spanning from the central point of the explosion to encompass partial regions of the storage tanks.



(a)



(b)

Fig. 6: Probability of injury for both (a) lung damage and (b) eardrum rupture.

It is noteworthy that another side of the tank displayed a 0.0% probability of injury, signifying the negligible impact of overpressure in that specific zone. A graphical representation of the injury probabilities for both lung damage and eardrum rupture is presented in Fig. 6.

The geometric shape of the exposed area closely resembles that of a sphere, mirroring the propagation characteristics of the overpressure. Furthermore, the periphery of this sphere exhibits a sharp decline in injury probability, signifying the rapid dissipation of overpressure energy following the explosion event. In addition, the 100% probability of injury exhibited by 15.14 meter and 17.45 meter of radius for lung damage and eardrum rupture, respectively, from the ground zero of the explosion. This observation underscores the exceedingly swift nature of the explosion phenomena. Moreover, the probability value can be used as a decision in designing the layout of the hydrogen storage tank.

7.Conclusions

In the field of gas safety, understanding the potential consequences of ignition within a gas cloud is crucial. This study highlighted the vapor explosions and emphasized the risk amplification with even a delayed ignition. Hydrogen gas, known for rapid shockwave generation, demands careful consideration in VCE assessments. Direct effects of explosions encompass organ injuries and, in severe cases, fatalities such as lung damage or eardrum ruptures. Indirect effects involve injuries from fragments or body displacement. Injury probabilities were quantified, aiding safety decisions during hydrogen bunkering. Several points that reflect this study are summarized as follows.

A CFD-based VCE simulation depicted hydrogen bunkering from port storage to a vessel, simulating a gas release due to a valve of storage tank malfunction. An equivalent gas cloud was modeled following VCE standards, employing the irreversible Arrhenius reaction model. Parameters were integrated into OpenFOAM.

The storage tank geometry in OpenFOAM utilized a uniform structured cartesian grid with “wall” surfaces for concrete ground and tank structure. “Empty” elements simulated hydrogen gas and air with 10.0% concentration. A precise time step ensured simulation stability, accommodating rapid VCE dynamics. Results highlighted rapid overpressure propagation and minimal gas temperature impact. Injury probability assessment visualized risk distribution.

The simulation featured a structured Cartesian grid for the storage tank geometry and precise time steps to stabilize the simulation, revealing rapid overpressure propagation with minimal gas temperature impact. Injury probability assessment aided in visualizing risk distribution, thereby informing facility design, safety protocols, and personnel protection strategies crucial for enhancing hydrogen fuel bunkering safety. Future advancements in multi-scale simulations, enhanced data integration, improved computational tools, and broader risk assessment frameworks, are anticipated to further refine safety practices in hydrogen handling and storage.

In summary, this study informs gas safety, aiding facility design, safety measures, and personnel protection during hydrogen fuel bunkering.

Nomenclature

POI	Probability of injury (–)
erf	Error function (-)
Pr	Probit (-)
$\dot{E}$	exergy rate (W)
h	specific enthalpy (J kg^{-1})
s_0	specific enthalpy of the dead state (J kg^{-1} K^{-1})
k	Turbulent kinetic energy (m^2/s^2)
u	Turbulent velocity fluctuation (m/s)
x	Cartesian coordinate point (m)
$\tilde{Y}$	Specie
R	Combustion rate (kg.s/m^3)
A	Constant for combustion rate equation (–)
r	Stoichiometric amount to burn 1kg of fuel
t	Time (s)
D	Mass diffusion (–)
RANS	Reynold Averaged Navier-Stoke
IMO	International Maritime Organization
LNG	Liquefied natural gas
CFD	Computational Fluid Dynamics
VCE	Vapor cloud explosion
CO_2	Carbon dioxide
NO_X	Nitrogen oxide
SO_X	Sulfur oxide
FLACS	Flame Acceleration Simulator
GHG	Greenhouse Gas
WAPS	Wind-assisted propulsion systems
H_2	Hydrogen

Greek symbols

δ	exergy defect (–)
η	efficiency (–)
ε	Turbulent dissipation rate (m^2/s^3)
v	Turbulent kinematic viscosity (m^2/s)
ρ	Fluid density (kg/m^3)

Subscripts

lung	Lung damage
ear	Eardrum rupture

References

1) International Maritime Organization (IMO), "International maritime organization third imo ghg study 2014," (2014). https://www.imo.org/en/ourwork/environment/pages/greenhouse-gas-studies-2014.aspx.
2) International Maritime Organization (IMO), "IMO's work to cut ghg emissions from ships," (2014). https://www.imo.org/en/MediaCentre/HotTopics/Pages/Cutting-GHG-emissions.aspx.
3) DNV, "Maritime forecast to 2050.pdf," *Accessed Sept. 19, 2023*, (n.d.). https://www.dnv.com/maritime/publications/maritime-forecast-2023/index.html.
4) M. Dadashzadeh, A. Ahmad, and F. Khan, "Dispersion modelling and analysis of hydrogen fuel gas released in an enclosed area: a cfd-based approach," *Fuel*, **184** 192–201 (2016). doi:10.1016/j.fuel.2016.07.008.
5) H. Nubli, J.M. Sohn, and A.R. Prabowo, "Layout optimization for safety evaluation on lng-fueled ship under an accidental fuel release using mixed-integer nonlinear programming," *Int. J. Nav. Archit. Ocean Eng.*, **14** 100443 (2022). doi:10.1016/j.ijnaoe.2022.100443.
6) H. Xing, S. Spence, and H. Chen, "A comprehensive review on countermeasures for co2 emissions from ships," *Renew. Sustain. Energy Rev.*, **134** (*June*) 110222 (2020). doi:10.1016/j.rser.2020.110222.
7) T. Baalisampang, F. Khan, R. Abbassi, and V. Garaniya, "Methodology to analyse lng spill on steel structure in congested marine offshore facility," *J. Loss Prev. Process Ind.*, **62** 103936 (2019). doi:10.1016/j.jlp.2019.103936.
8) X.J. Li, R.P. Zhou, and D. Konovessis, "CFD analysis of natural gas dispersion in engine room space based on multi-factor coupling," *Ocean Eng.*, **111** 524–532 (2016). doi:10.1016/j.oceaneng.2015.11.018.
9) H. Nubli, and J.M. Sohn, "CFD-based simulation of accidental fuel release from lng-fuelled ships," *Ships Offshore Struct.*, **17** (*2*) 339–358 (2022). doi:10.1080/17445302.2020.1827806.
10) S. Park, B. Jeong, B.S. Lee, S. Oterkus, and P. Zhou, "Potential risk of vapour cloud explosion in flng liquefaction modules," *Ocean Eng.*, **149** (*February*) 423–437 (2018). doi:10.1016/j.oceaneng.2017.08.032.
11) S. Suryanto, A.R. Prabowo, T. Muttaqie, I. Istanto, R. Adiputra, N. Muhayat, A. Fajri, M. Braun, and S. Ehlers, "Evaluation of high-tensile steel using nonlinear analysis: experiment-fe materials benchmarking of lng carrier structures under low-temperature conditions," *Energy Reports*, **9** (*February*) 149–161 (2023). doi:10.1016/j.egyr.2023.05.252.
12) H. Nubli, J.M. Sohn, and D. Jung, "Temperature reduction on lng bunkering ship structure under accidental cryogenic gas release using cfd simulation," *Proc. Int. Conf. Offshore Mech. Arct. Eng. - OMAE*, **2** (*2017*) 1–12 (2022). doi:10.1115/OMAE2022-78729.
13) B.H. Lim, and E.Y.K. Ng, "Model for cryogenic flashing lng leak," *Appl. Sci.*, **11** (*19*) 1–11 (2021).

doi:10.3390/app11199312.

14) J. Pujol, R.N. Kleiveland, N.I. Lileheie, T. Holmas, and J. Amdahl, "Advanced cryogenic structural collapse analysis csca - part ii: cryogenic flow and structural cooling," *Offshore Technol. Conf. Asia 2016, OTCA 2016*, **2016** 1–8 (2016).
15) J.K. Paik, D.H. Lee, S.H. Noh, D.K. Park, and J.W. Ringsberg, "Full-scale collapse testing of a steel stiffened plate structure under axial-compressive loading triggered by brittle fracture at cryogenic condition," *Ships Offshore Struct.*, 1–17 (2020). doi:10.1080/17445302.2020.1787930.
16) T. Muttaqie, L.S. Geon, S.-R. Cho, and J.M. Sohn, "Structural assessment review of type-c independent tank in lng bunkering ship," *Proc. 6th Int. Conf. Exhib. Sustain. Energy Adv. Mater. Lect. Notes Mech. Eng.*, 97–108 (2020). doi:10.1007/978-981-15-4481-1_10.
17) Z. Deng, M. Bai, M. Qi, Y. Tian, H. Qu, and Y. Liu, "Inherent safety assessment of on-board methanol reforming hydrogen production fuel cell system considering jet fire and vapor cloud explosion," *Int. J. Hydrogen Energy*, **50** 1047–1061 (2024). doi:10.1016/j.ijhydene.2023.08.257.
18) H. Han, S. Kim, J. Park, G. Kim, and S. Jung, "Risk mitigation study for hydrogen releases from hydrogen fuel cell vehicles," *Int. J. Hydrogen Energy*, **54** (*October 2022*) 1593–1602 (2024). doi:10.1016/j.ijhydene.2023.11.208.
19) Q. Ma, Y. He, J. You, J. Chen, and Z. Zhang, "Probabilistic risk assessment of fire and explosion of onboard high-pressure hydrogen system," *Int. J. Hydrogen Energy*, **50** (*August 2023*) 1261–1273 (2024). doi:10.1016/j.ijhydene.2023.10.157.
20) S. Li, Z. Xu, F. Wang, Z. Xiu, Z. Liu, P. Li, and M. Li, "Flame propagation characteristics and thermal radiation hazards of methane-hydrogen-mixed cloud explosion in unconfined area: experiment research and theoretical modeling," *Int. J. Hydrogen Energy*, **54** (*October 2023*) 1563–1574 (2024). doi:10.1016/j.ijhydene.2023.12.137.
21) K. Liu, C. He, Y. Yu, C. Guo, S. Lin, and J. Jiang, "A study of hydrogen leak and explosion in different regions of a hydrogen refueling station," *Int. J. Hydrogen Energy*, **48** (*37*) 14112–14126 (2023). doi:10.1016/j.ijhydene.2022.12.267.
22) X. Wang, and W. Gao, "Hydrogen leakage risk assessment for hydrogen refueling stations," *Int. J. Hydrogen Energy*, **48** (*91*) 35795–35808 (2023). doi:10.1016/j.ijhydene.2023.05.340.
23) X. Wang, C. Zhang, and W. Gao, "Risk assessment of hydrogen leakage in diesel hydrogenation process," *Int. J. Hydrogen Energy*, **47** (*10*) 6955–6964 (2022). doi:10.1016/j.ijhydene.2021.12.027.
24) X. Wang, X. Zou, and W. Gao, "Flammable gas leakage risk assessment for methanol to hydrogen refueling stations and liquid hydrogen refueling stations," *Int. J. Hydrogen Energy*, **54** (*November 2023*) 1286–1294 (2024). doi:10.1016/j.ijhydene.2023.11.255.
25) N. Yang, J. Deng, C. Wang, Z. Bai, and J. Qu, "High pressure hydrogen leakage diffusion: research progress," *Int. J. Hydrogen Energy*, **50** 1029–1046 (2024). doi:10.1016/j.ijhydene.2023.08.221.
26) Y. Shen, H. Lv, T. Zheng, Y. Liu, W. Zhou, and C. Zhang, "Temporal and spatial evolution of hydrogen leakage and diffusion from tube fittings on fuel cell vehicles under the effect of ambient wind," *Renew. Sustain. Energy Rev.*, **185** (*August*) 113596 (2023). doi:10.1016/j.rser.2023.113596.
27) H. Nubli, and J.M. Sohn, "CFD Gas Explosion Simulation on LNG-Fueled Ship," in: Proc. ASME 2021 40th Int. Conf. Ocean. Offshore Arct. Eng. OMAE2021, 2021: pp. 1–12.
28) A. Huser, P.J. Nilsen, and H. Skåtun, "Application of k-ε model to the stable abl: pollution in complex terrain," *J. Wind Eng. Ind. Aerodyn.*, **67**–**68** 425–436 (1997). doi:10.1016/S0167-6105(97)00091-3.
29) B.E. Launder, and D.B. Spalding, "The Numerical Computation of Turbulent Flows," Pergamon Press, Ltd, 1983. doi:10.1016/b978-0-08-030937-8.50016-7.
30) A.P. Van Ulden, and A.A.M. Holtslag, "Estimation of atmospheric boundary layer parameters for diffusion applications," *J. Clim. Appl. Meteorol.*, **24** 1196–1207 (1985). papers3://publication/uuid/2075185C-E7DA-4731-AFE2-A072F440B8E5%0Apapers3://publication/uuid/2C1B0D0E-9949-42CE-9BAF-46F0056D6793.
31) B.F. Magnussen, and B.H. Hjertager, "On mathematical modeling of turbulent combustion with special emphasis on soot formation and combustion," *Symp. Combust.*, **16** (*1*) 719–729 (1977). doi:10.1016/S0082-0784(77)80366-4.
32) M.H. Bae, and J.K. Paik, "Effects of structural congestion and surrounding obstacles on the overpressure loads in explosions: experiment and cfd simulations," *Ships Offshore Struct.*, **13** (*2*) 165–180 (2018). doi:10.1080/17445302.2017.1347978.
33) S.M. Tauseef, D. Rashtchian, T. Abbasi, and S.A. Abbasi, "A method for simulation of vapour cloud explosions based on computational fluid dynamics (cfd)," *J. Loss Prev. Process Ind.*, **24** (*5*) 638–647 (2011). doi:10.1016/j.jlp.2011.05.007.
34) Z. Sajid, M.K. Khan, A. Rahnama, F.S. Moghaddam, K. Vardhan, and R. Kalani, "Computational fluid dynamics (cfd) modeling and analysis of hydrocarbon vapor cloud explosions (vces) in amuay refinery and jaipur plant using flacs," *Processes*, **9** (*6*) 1–18 (2021). doi:10.3390/pr9060960.
35) G. Momferatos, S.G. Giannissi, I.C. Tolias, A.G. Venetsanos, A. Vlyssides, and N. Markatos, "Vapor cloud explosions in various types of confined environments: cfd analysis and model validation," *J.*

Loss Prev. Process Ind., **75** (*June 2021*) 104681 (2022). doi:10.1016/j.jlp.2021.104681.

36) C.B. Jang, J.Y. Han, G.S. Jeong, J.H. Baek, K.Y. Choi, and S. Jung, "CFD/fea connected modeling and analysis of vapor cloud explosion at a wastewater storage pond of a chemical plant in korea," *Energy Sci. Eng.*, **7** (*1*) 272–287 (2019). doi:10.1002/ese3.276.
37) P.A. Duong, B.R. Ryu, J. Jung, and H. Kang, "Comparative analysis on vapor cloud dispersion between lng/liquid nh3 leakage on the ship to ship bunkering for establishing safety zones," *J. Loss Prev. Process Ind.*, **85** (*August 2022*) 105167 (2023). doi:10.1016/j.jlp.2023.105167.
38) C. Langree, E. Franquet, J. Reveillon, G. Lecocq, and F.X. Demoulin, "CFD simulation of an unconfined vapor cloud explosion through obstacles using openfoam," 262–269 (2020). https://hal-ineris.archives-ouvertes.fr/ineris-03319942%0Ahttps://hal-ineris.archives-ouvertes.fr/ineris-03319942/document.
39) S. Mishra, and K.B. Mishra, "Numerical study of large-scale lng vapour cloud explosion in an unconfined space," *Process Saf. Environ. Prot.*, **149** 967–976 (2021). doi:10.1016/j.psep.2021.03.034.
40) K.-P. Kim, Y.-T. Kim, and H.-K. Kang, "CFD approach on gas explosion for sil in gas fuelled ship," *J. Korean Soc. Mar. Eng.*, **39** (*2*) 195–200 (2015). doi:10.5916/jkosme.2015.39.2.195.
41) M.J. Assael, and Konstantinos E. kakosimos, "Fires, Explosions, and Toxic Gas Dispersions," 2010.
42) Abel Tenu Mekonnen, "Modelling and numerical simulation of gas explosions for industrial safety analyses," 2019.
43) C.R. Bauwens, J. Chaffee, and S. Dorofeev, "Experimental and Numerical Study of Methane-air Deflagrations in a Vented Enclosure," in: Fire Saf. Sci. Ninth Int. Symp., 2004: pp. 1043–1054.
44) H. Nubli, and J.M. Sohn, "Procedure for determining design accidental loads in liquified-natural-gas-fuelled ships under explosion using a computational-fluid-dynamics-based simulation approach," *Ships Offshore Struct.*, **17** (*10*) 2254–2271 (2022). doi:10.1080/17445302.2021.1983249.
45) HySafe, "Hydrogen thermochemistry," *Accessed December 8, 2023*, (2007). http://www.hysafe.org/img/fhslect03.pdf.
46) Christopher J. Greenshields, "OpenFOAM User's Guide version 11," 2023.