

Modeling Development for Evaluating Effective Mitigation Measures for LNG Safety and Spill Response

Byung Kyu Kim, Roberto Ruiz, Subramanya Nayak, Ray Mentzer,
 Mahboobul Sam Mannan*

Mary Kay O'Connor Process Safety Center, Artie McFerrin Department of Chemical Engineering, Texas A&M University System, College Station, Texas 77843-3122, USA
mannan@tamu.edu

Liquefied natural gas (LNG) is expected to play a significant role in providing flexibility to the energy portfolio worldwide. Concerns over the safety and security of LNG facilities have prompted the need for continuous improvement of safety measures to mitigate and prevent escalation of the consequence of a LNG spill. Currently, the effectiveness of various mitigation measures used in LNG facilities and terminals is not well understood. Numerical simulation using computational fluid dynamics (CFD) has evolved significantly to provide in-depth analysis on the complex flow problems. CFD modeling has been utilized to investigate the various effects of passive/active mitigation measures on the behavior of LNG. Discussed are the theoretical approaches in understanding various effects of the design parameters for establishing effective passive/active mitigation systems using the CFD modeling for LNG facilities and terminals. The results from this work will provide an engineering analysis, which can serve as fundamentals to support formulating guidelines for the vapor cloud suppression and mitigation system applicable for the LNG industry.

1. Introduction

Liquefied natural gas (LNG) provides an alternative means in adding flexibility to alleviate the discrepancy of the supplies and demands of natural gas worldwide (Foss, 2007). Natural gas is liquefied by cooling to its boiling temperature at atmospheric pressure, where the volume is reduced by approximately 600 times less than its gas phase. With the liquefaction capacity expected to double by the year 2035 globally (IEA, 2011), the safety and security for the LNG facilities have raised concerns over the potential for LNG releases. This work discusses the recent modeling progresses at the Mary Kay O'Connor Process Safety Center (MKOPSC) at Texas A&M University (TAMU), as an extension of the LNG safety and spill response research program. The research aims to provide a better understanding of the complex LNG behavior and its underlying physical phenomenon, which could provide conclusive guidance in designing a site-specific mitigation system for the current LNG facilities.

2. LNG Safety Research at Mary Kay O'Connor Process Safety Center

The current US federal regulation requires an impoundment around LNG storage areas, designed to serve the volume equal to 100 or 110 % of the tanks' to provide confinement and to mitigate the initial impact from an accidental release (NFPA, 2008). The water curtain system, which had been widely used in the chemical and petrochemical industry, can be designed to reduce the LNG vapor concentration below the flammability limits to prevent a LNG fire or explosion in case of a spill (Martinsen et al., 1977).

The MKOPSC has been involved in a LNG safety and emergency spill response research program with BP Global Gas SPU since 2005. A series of outdoor LNG spill experiments were conducted at the Brayton Fire Training Field (BFTF) in College Station, Texas. The experimental works were designed to provide better understandings of the underlying physical phenomena involved during various LNG spill scenarios, including underwater LNG releases (Qi et al., 2011). Various mitigation measures, such as a vapor barrier

(Qi et al., 2010), and water spray (Rana et al., 2010a), were tested to evaluate the effectiveness in reducing the consequence of a LNG release.

In parallel to the outdoor experimental works, theoretical study of the complex LNG vapor behavior had been conducted using the computational fluid dynamics (CFD) modeling (Cormier et al., 2009). CFD modeling had been recommended for evaluating the high level of consequences related to an accidental LNG spill (Hightower et al., 2004). The CFD codes can provide numerical solutions to the complex flow problems of LNG vapor dispersions by directly solving the constitutive equations, whereas the traditional integral-type models have raised some concerns due to its limitations in describing the LNG vapors adequately (Gavelli et al., 2008). The theoretical study using CFD modeling can provide in-depth understanding of complex behavior of LNG vapors and help close existing gaps in understanding the complex interaction of LNG vapors with various safety measures. The commercial type CFD codes, such as FLACS, CFX, and Fluent, had been utilized to set up a validated model to study the complex behavior of LNG dispersion. FLACS has been used to study the effects of various physical barriers around the storage area (Ruiz, 2012). The forced dispersion of LNG vapors using a water curtain application has been modeled using the ANSYS Fluent (Kim et al., 2012).

3. Passive Mitigation System

The potential hazards of the worst credible scenario of a LNG release, which may result in flammable vapors engulfing a facility, must be assessed when designing a LNG terminal or facility (NFPA, 2008). Evaluating the safety distance is part of the requirement to develop mitigation measures to prevent the escalation of further damage outside the facility. Federal regulations, such as 49 CFR 193 and standard NFPA 59A, require implementation of accurate assessment tools to evaluate the potential impact areas for the accidental release scenarios (Hightower et al., 2004).

Several studies in vapor dispersion have been performed, however, there are only limited studies on the effects of passive barriers in reducing the safety distance by using the CFD approach. A study of the key parameters for vapor dispersion modeling was performed using CFX code to determine their effect on LNG vapor dispersion (Cormier et al., 2009). A source term study on LNG underwater releases (Qi et al., 2011) and validation with the MKOPSC experimental data had been conducted using CFX code (Qi et al., 2010), along with a sensitivity analysis on the mesh size and the source term turbulence. Parametric study on the key elements involved in vapor interaction with the obstacles demonstrated that the height and width of an obstacle have a significant effect in reducing the vapor concentration (Ruiz, 2012).

Passive mitigation systems are designed to provide inherent safety measures within a facility to function without any human interventions or additional mechanical or energy inputs (49CFR193, 2000). A dike is placed around the LNG storage area to serve as a confinement to hold any potential LNG release. This section discusses the effects of various designs of an LNG dike on reducing the safety distance, which is defined as the distance where the vapor concentration becomes one half of the lower flammability limit (2.5 [v/v] %). NFPA 59A indicates that an impoundment system (dike) must be designed to contain a possible LNG spill equal to the 110 % of the largest tank associated within the system (NFPA, 2008).

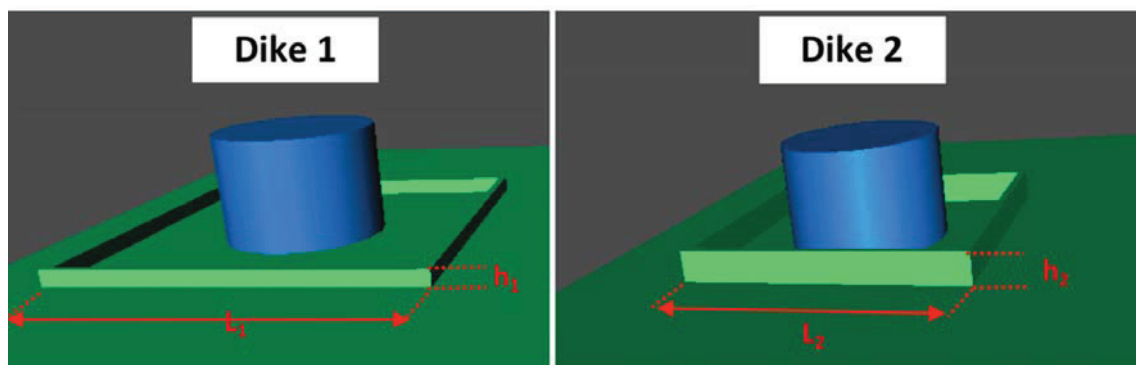


Figure 1: Comparison between two different dike designs; Dike 1: $L_1 = 110$ m and $h_1 = 5$ m, Dike 2: $L_2 = 85$ m and $h_2 = 8$ m

Figure 1 shows the two different designs that were used in this comparison study. The height and width of the dike were varied, while maintaining the same volume to evaluate the effects of different designs on the vapor behavior. A commercial CFD tool, FLACS was used for this analysis. FLACS has been approved by

the US Department of Transportation as an alternative LNG vapor gas model (Quarterman, 2011), as indicated in 49 CFR 193.2059(a) requirements (49CFR193, 2000). A typical LNG tank with a capacity of 50,000 m³ was considered in this work, and the dimensions of Dike 1 were set as 110 m x 110 m x 5 m as shown in Figure 1, which was set according to the NFPA 59A requirements in terms of volume capacity. The height was increased from 5 m to 8 m, and the width of the dike was set at 85 m for Dike 2 design.

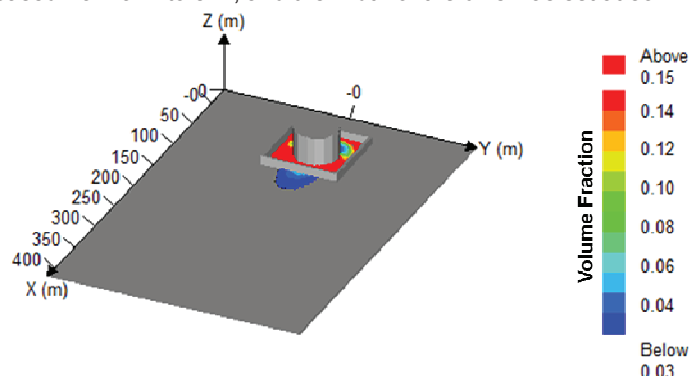


Figure 2: Pool formation of Dike 2 at time 600 s

A worst case credible scenario of a rupture at the pump-out transfer piping located within the dike area was considered, with the maximum flow rate of 2,000 m³/h. A LNG pool formation model was set up based on this leakage scenario within the dike area where a leak point was set at the highest x-coordinate of the LNG tank at the ground level. The governing equations for a spreading pool in FLACS take into account friction, heat and mass transfer (GEXCON, 2011). The FLACS pool model has been widely validated for a spreading pool with and without obstacles under different conditions: adiabatic, on water, and on soil. Figure 2 shows the simulation results of the pool generated within Dike 2 using FLACS at $t = 600$ s for the vapor concentration up to the LNG upper flammability limit, which is at 15 [v/v] %. The LNG pool fills the storage volume inside the dike initially, and vapor travels from the top of the dike wall, propagating in the downwind direction, as illustrated by the vapor contour outside the confinement in Figure 2.

NFPA 59A requires that the impoundment system must withstand a spill for 10 min. An atmospheric stability of 'F' with a wind speed of 1.5 m/s was considered to simulate the worst-case scenario, with a roughness of $z_0 = 0.03$ m. The mesh settings were refined around the LNG spill, tank, and dike to solve for the complex vapor behavior more in detail. The mesh sizes were 0.8 m in the x and y directions, and 0.5 m in the z direction. The measurement points for gas concentration were placed along the centerline, downwind of the dike borderline at distances of 50 m and 200 m.

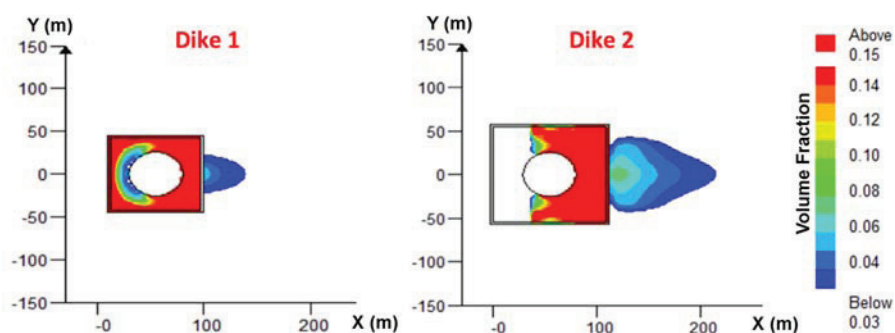


Figure 3: Comparison between safety distances with the two different types of dikes

Figure 3 shows the comparison of the safety distances of the two different dike designs, Dike 1 and Dike 2 from Figure 1. The LNG leakage is set at the highest x-coordinate of the LNG storage tank at the ground level, which is on the right side of Dike 1 and Dike 2 in the downwind direction. The results indicate that the safety distance was reduced for the design with higher dike design, Dike 2. The vapor concentration was reduced as the dike with higher confinement design holds vapor within the dike longer, promoting

turbulence effects, which allow better mixing with air. The safety distance for Dike 1, which has 5 m high confinement, was 220 m, while the Dike 2, which has 8 m high confinement, showed 140 m. The overall reduction of the safety distance was 35 % from increasing the confinement height from 5 m to 8 m. This study demonstrated that the height of a dike plays an important role in reducing downwind concentration values and requiring shorter safety distances. The area behind the tank for Dike 1 indicates that there is no pool formation, which is due to the vaporization rate being greater than the pool spread in this area.

This study could provide further evaluation of new dike designs as well as a practical application of the storage area in the LNG facilities. Future work includes investigating the turbulence effects generated from the different LNG dike configurations. The turbulence induced from the dike configurations may increase the overpressure in case of an explosion inside the storage area. Further analysis is required to assess the potential hazards that the new configuration may add to the current consequence assessment conducted for LNG facilities and terminals.

4. Active Mitigation System

Water curtains have proven to be effective in mitigating LNG vapors below the flammability limit (Brown et al., 1983). Previous studies of the water curtain application on LNG vapors were only limited by the experimental works conducted by the US Coast Guard (Martinsen et al., 1977) and by the Gas Research Institute (Heskestad et al., 1983). The MKOPSC conducted consecutive outdoor water curtain experiments on LNG dispersion at BFTF (Rana et al., 2010a). The water curtain has shown effective means to dissipate and dilute the LNG vapor concentration below the flammability limits in the downwind region.

The forced dispersion of LNG vapors was modeled using CFD modeling to address the current existing gaps in understanding the interaction of the droplet-vapor system. The discrete particle model (DPM) in ANSYS Fluent was used to simulate the upward-oriented conical-type water spray, which was found most effective in enhancing LNG dispersion (Rana and Mannan, 2010b). DPM solves the complex multiphase flow problems by tracking the droplets in discrete frame while the air-vapor mixture flow is solved in continuous phase to evaluate the interaction of the droplet-vapor system (Ranade, 2002). The experimental data from the March 2009 MKOPSC outdoor LNG spill experiments were used to calibrate the essential physical parameters in describing the LNG forced dispersion (Kim et al., 2012).

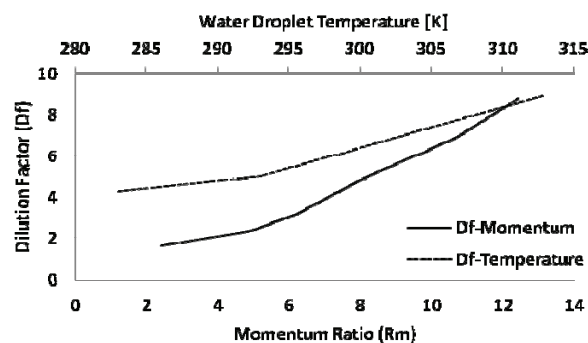


Figure 4: Dilution factor evaluated for various momentum and thermal effects

Figure 4 shows the effectiveness of forced mitigation from various momentum and thermal settings simulated from different water spray conditions. The momentum and thermal effects of the forced dispersion of LNG vapors were evaluated by varying the momentum ratio and droplet temperatures. The momentum imparted from the water curtain was quantified using the momentum ratio (R_M), which is the ratio of the momentum imparted from the water droplets to the air-vapor mixture movement (Hald et al., 2005). The dilution factor, which is the ratio of the vapor concentration before and after the water curtain application, was used to evaluate the effectiveness of the LNG forced mitigation. The dilution factor increased as the momentum imparted from the water droplet increased, while setting the water droplet temperature constant at 303 K. The LNG forced dispersion also became more effective as more thermal effects were involved from higher droplet temperatures with the momentum ratio set at approximately 9.

Figure 5 shows the vapor contour for two different spray settings, momentum ratios of 6.21 and 11.56, and two different droplet temperatures at 283 K and 313 K. As shown in Figure 5(a), most of the vapors travel towards ground level after they pass through the water spray region, mainly due to the insufficient momentum from the droplets for the momentum ratio of 6.21 case. The vapors are more effectively

dispersed to higher elevation with the momentum ratio of 11.56 as illustrated by Figure 5(b). At a droplet temperature of 283 K (Figure 5(c)), the vapors travel back to the lower level after passing through the water spray region, while with higher droplet temperature (Figure 5(d)), the vapor cloud disperses into the higher elevation more effectively.

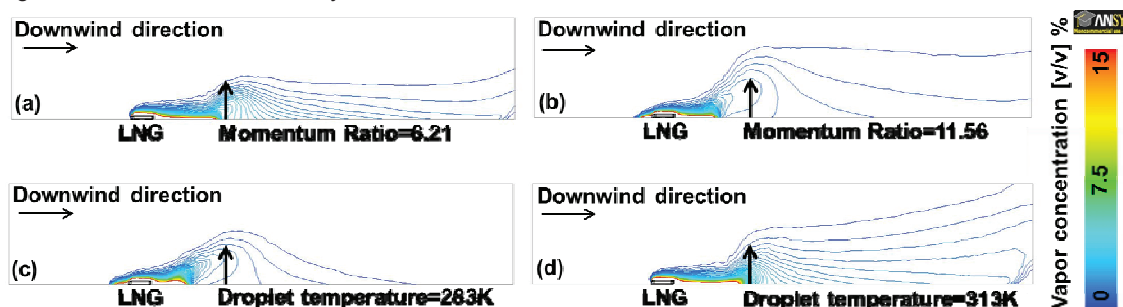


Figure 5: Vapor concentration contour of different momentum and thermal effects from water curtain: (a) $RM = 6.21$, (b) $RM = 11.56$, (c) droplet temperature = 283 K, and (d) droplet temperature = 313 K

The forced dispersion using the CFD modeling can describe the vapor behavior more in detail, which could serve as a better consequence assessment tool, compared to the traditional integral-type model. The different effects of momentum transfer on vapor behavior and minimum thermal transfer for effective forced mitigation can be investigated in detail from the forced dispersion modeling.

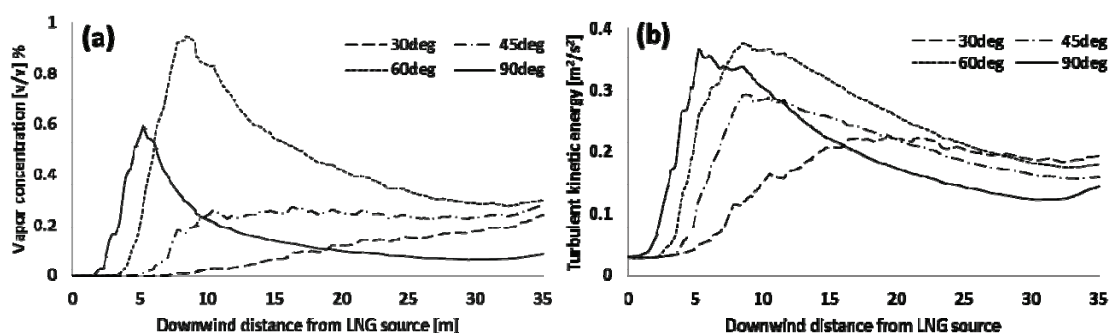


Figure 6: Various application of forced mitigation model: (a) vapor concentration and (b) turbulent kinetic energy recorded for various nozzle configurations

Using forced dispersion modeling, various design configurations can be investigated for an effective forced mitigation. LNG forced dispersion modeling was set up so that the nozzles were tilted towards the downwind direction at different angles to evaluate the effects of various configurations on diluting the LNG vapors. Figure 6(a) shows the vapor concentration at centerline ($z = 5$ m) in the downwind direction with different nozzle configurations. The nozzle angle with 60° is most effective in dispersing the LNG vapors towards higher elevations. The turbulent kinetic energy (TKE), which is the root mean square value of the velocity components, represents the local turbulence induced from the droplet interaction. Figure 6(b) shows the TKE values from the forced dispersion modeling, and the 60° configuration is most effective in enhancing the turbulence overall at the centerline of the downwind region ($z = 5$ m). The higher vapor concentration for the nozzles configured at 60° and 90° as shown in Figure 6(a) may be due to the high turbulence induced in the downwind region from these configurations. The LNG forced mitigation study using the CFD modeling can provide detailed analysis on the physical mechanisms of the forced dispersion of LNG vapors using water curtains; the air entrainments, and momentum and heat transfer.

5. Conclusions

The LNG safety program at the MKOPSC aims to provide site-specific recommendations for safer operation for the LNG industry. The theoretical modeling work using the CFD tool can provide a way to fill in the existing gaps of experimental work by providing a solution to the complex flow problems of LNG

vapors based on fundamental flow physics. The modeling work on passive barriers demonstrated the effects of various designs of the dike in reducing the downwind vapor concentration. This approach can be applied in facility layout of an LNG terminal and design of an LNG impoundment around the storage area. The forced dispersion modeling work provides a simple way to evaluate various design parameters to find design criteria for water spray applications.

The LNG safety research at MKOPSC combines the theoretical approach with the outdoor experimental work to provide a comprehensive understanding of the complex behavior of LNG vapors. This research aims to provide engineering guidelines on the design criteria applicable for LNG facilities in suppressing the LNG vapor dispersion and effectively mitigating the consequence of the LNG spill, which can be directly applied to the current LNG industry.

Reference

- 49CFR193, 2000, Title 49 Code of Federal Regulations Part 193 (49-CFR-193): Liquefied Natural Gas Facilities: Federal Safety Standards. US Government Printing Office, Washington DC, USA.
- Brown L.E., Martinsen W.E., Cornwell J.B., 1983, Spill protection for liquefied gas facilities. Spring Meeting of the American Petroleum Institute Committee on Safety and Fire Protection, Apr 5-8, 1983, Ft. Lauderdale, Florida, USA.
- Cormier B.R., Qi R., Yun G., Zhang Y., Mannan M.S., 2009, Application of computational fluid dynamics for LNG vapor dispersion modeling: A study of key parameters. *Journal of Loss Prevention in the Process Industries*, 22, 332-352.
- Foss M.M., 2007, Introduction to LNG: An overview on liquefied natural gas (LNG), its properties, organization of the LNG industry and safety considerations. Center for Energy Economics, Houston, Texas, USA.
- Gavelli F., Bullister E., Kytomaa H., 2008, Application of CFD (Fluent) to LNG spills into geometrically complex environments. *Journal of Hazardous Materials*, 159, 158-168.
- GEXCON, 2011, FLACS v9.1 User's Manual.
- Hald K., Buchlin J.M., Dandrieux A., Dusserre G., 2005, Heavy gas dispersion by water spray curtains: A research methodology. *Journal of Loss Prevention in the Process Industries*, 18, 506-511.
- Heskestad G., Meroney R.N., Kothari K.M., Neff D.E., 1983, Effectiveness of water spray curtains in dispersing LNG vapor clouds. American Gas Association Transmission Conference, May 2-4, 1983, Seattle, Washington, American Gas Association, Paper No 83-T-69, 169-183.
- Hightower M., Gritz L., Luketa-Hanlin A., Covan J., Tieszen S., Wellman G., Irwin M., Kaneshige M., Melof B., Morrow C., Ragland D., 2004, Guidance on Risk Analysis and Safety Implications of a Large Liquefied Natural Gas (LNG) Spill Over Water, SAND2004-6258. Sandia National Laboratories, Albuquerque, New Mexico, USA.
- IEA, 2011, World Energy Outlook 2011: Are we entering a golden age of gas?. International Energy Agency, Paris, France.
- Kim B.K., Ng D., Mentzer R.A., Mannan M.S., 2012, Modeling of water-spray application in the forced dispersion of LNG vapor cloud using a combined Eulerian-Lagrangian approach. *Industrial & Engineering Chemistry Research*, 51, 13803-13814.
- Martinsen W.E., Muhlenkamp S.P., Olson J., 1977, Disperse LNG vapors with water. *Hydrocarbon Processing*, 260-267.
- NFPA, 2008, NFPA 59A: Standard for the Production, Storage, and Handling of Liquefied Natural Gas (LNG). National Fire Protection Association, Quincy, Massachusetts, USA.
- Qi R., Ng D., Cormier B.R., Mannan M.S., 2010, Numerical simulations of LNG vapor dispersion in Brayton Fire Training Field tests with ANSYS CFX. *Journal of Hazardous Materials*, 183, 51-61.
- Qi R., Raj P.K., Mannan M.S., 2011, Underwater LNG release test findings: Experimental data and model results. *Journal of Loss Prevention in the Process Industries*, 24, 440-448.
- Quarterman C.L., 2011, Approval for GexCon's Petition to use FLACS as an alternative vapor gas model under 49 CFR 193.2059(a). US Department of Transportation, Washington DC, USA.
- Rana M.A., Guo Y., Mannan M.S., 2010a, Use of water spray curtain to disperse LNG vapor clouds. *Journal of Loss Prevention in the Process Industries*, 23, 77-78.
- Rana M.A., Mannan M.S., 2010b, Forced dispersion of LNG vapor with water curtain. *Journal of Loss Prevention in the Process Industries*, 23, 768-772.
- Ranade V., 2002, Computational flow modeling for chemical reactor engineering. Elsevier Academic Press, San Diego, California, USA.
- Ruiz R., 2012, Study of the effects of obstacles in Liquefied Natural Gas (LNG) Vapor Dispersion using CFD modeling, Master Thesis, Texas A&M University, College Station, Texas, USA.