

LNG Safety Research: FEM3A Model Development

Quarterly Report
01-01-06 to 03-31-06

Prepared by:

Iraj A. Salehi
Gas Technology Institute
and
Jerry Havens *and* Tom Spicer
University of Arkansas

May 1, 2006

Award No. DE-FG26-04NT42030

Submitted by:

Iraj A. Salehi
Gas Technology Institute
1700 S. Mount Prospect Rd.
Des Plaines, IL 60018

and

Jerry Havens *and* Tom Spicer
Chemical Hazards Research Center
University of Arkansas
Fayetteville, Arkansas 72701

DISCLAIMER: This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.

ABSTRACT

This quarterly report for DE-FG26-04NT42030 covers a period from January 1, 2006 to March 31, 2006. GTI's activities during the report quarter were limited to administrative work. The work at the University of Arkansas continued in line with the initial scope of work and the identified questions regarding surface to cloud heat transfer as being largely responsible for the instability problems previously encountered. A brief summary of results is discussed in this section and the complete report from University of Arkansas is attached.

TABLE OF CONTENTS

| | |
|------------------------------|---|
| ABSTRACT | 3 |
| EXECUTIVE SUMMARY | 3 |
| EXPERIMENTAL | 3 |
| RESULTS AND DISCUSSION | 4 |
| CONCLUSION | 4 |
| Appendix A | |

EXECUTIVE SUMMARY

Work continued to address numerical problems experienced with simulation of low-wind-speed, stable, atmospheric conditions with FEM3A. Steps 1 through 8 in the plan outlined in the first Quarterly report have been completed successfully for the FEM3A model utilizing the Planetary Boundary Layer (PBL) turbulence closure model. Researchers at the University of Arkansas have solved the problems related to stability of the simulations at regulatory conditions of low wind speed and stable atmospheric conditions with FEM3A using the PBL model, and are continuing our program to verify the operation of the model using an updated, verified, version of the k-epsilon turbulence closure model which has been modified to handle dense gas dispersion effects.

EXPERIMENTAL

University of Arkansas experienced delays in their experimental efforts to determine the spectral analysis of the turbulence in the wind tunnel as requested by representatives from DOE-NETL, but they believe that problem is solved now and expect to commence with the experimental program in May. The experimental program will focus on measurement of velocities and gas concentrations in the wake region of an LNG tank for use in our CFD model validation efforts, and it will provide the turbulence spectrum for the wind tunnel approach flow. Given the no-cost time extension granted to September 30, 2006, we are on track to finish the contract requirements on schedule.

The primary purpose of this task was to repeat and extend former experiments using uniform roughness elements covering the wind tunnel floor to create turbulence properties similar to field scale wind conditions. The roughness used had already been characterized in a related research program; consequently, only the gas concentration

measurements will have to be repeated. The resulting data set(s) will be a valuable addition to the data archives demonstrating the FEM3A model for application to LNG vapor dispersion prediction.

There are strong indications that the experimental data from the wind tunnel would be more applicable to field conditions, and therefore more useful for model validation, when the floor is artificially roughened.

RESULTS AND DISCUSSION

Data from experimental work will be used to verify the FEM3A model for application involving dispersion over rough surfaces (for example, suburban housing) with and without the presence of obstacles such as tank and/or dike structures and industrial buildings. The end product will be an advanced turbulence closure model (for describing the turbulent mixing involved in the dispersion process) that will allow for more realistic description of dispersion problems with obstacles and terrain features of greater complexity (the real world).

CONCLUSION

Researchers at the University of Arkansas have initiated an experimental program to measure velocities in the wake region of an LNG tank. They are also continuing to address the need for changes in the turbulence closure methods to bring the model predictions into closer alignment with the overall wind tunnel results.

Vapor Dispersion and Thermal Hazard Modeling

Eighth Quarterly Report
(January - March, 2006)

Prepared by

Jerry Havens
Tom Spicer

Chemical Hazards Research Center
University of Arkansas
Fayetteville, Arkansas 72701

For

GAS TECHNOLOGY INSTITUTE
Contract No. 4GTI-DE-FG26-04NT42030 (Prime)
K100029184 (Subcontract)

GTI Project Manager
Iraj Salehi

April 30, 2006

GTI DISCLAIMER

LEGAL NOTICE. This report was prepared by the University of Arkansas as an account of work sponsored by the Gas Technology Institute (GTI). Neither GTI, members of GTI, nor any person acting on behalf of either:

- a. Makes any warranty or representation, express or implied, with respect to the completeness, or usefulness of the information contained in this report, or that the use of any apparatus, method, or process disclosed in this report may not infringe privately owned rights; or
- b. Assumes any liability with respect to the use of, or for damages resulting from the use of, any information, apparatus, method, or process disclosed in this report.

CONTENTS

| <u>PART</u> | <u>PAGE</u> |
|---|-------------|
| 1.0.....Research Summary..... | 4 |
| 2.0.....Program Objective..... | 8 |
| 3.0.....Program Time Schedule..... | 10 |
| 4.0.....Work Performed January - March, 2006..... | 10 |
| 5.0.....New Developments Relative to 49 CFR 193 AND NFPA 59A..... | 10 |
| 6.0.....Plans for Future Work..... | 12 |

APPENDIX A - LNG VAPOR CLOUD EXCLUSION ZONES FOR SPILLS INTO IMPOUNDMENTS

APPENDIX B - UNITED STATES REGULATIONS FOR SITING LNG TERMINALS: PROBLEMS AND POTENTIAL

1.0 RESEARCH SUMMARY

| | |
|-------------------------|---|
| Title | Vapor Dispersion and Thermal Hazard Modeling |
| Contractor | University of Arkansas GTI Contract Number: K100029184 |
| Principal Investigators | Jerry Havens Tom Spicer |
| Contract Period | April 2004 – March 2006 No Cost Extension granted to September 30, 2006 |
| Objectives | <p>1. To develop the FEM3A model for application to general scenarios involving dispersion problems with obstacles and terrain features of realistic complexity, and for very low wind speed, stable weather conditions as required for LNG vapor dispersion application specified in 49 CFR 193 and NFPA 59A.</p> <p>2. To provide additional wind-tunnel dense gas dispersion data that can be used for verification of computational fluid dynamics (CFD) computer models for predicting LNG vapor dispersion influenced by terrain features and or obstacles, and to provide assistance to DOE in its consideration of FLUENT as an recommended alternative (to FEM3A) CFD model for approval by the DOT Administrator under 49 CFR 193 and NFPA 59A.</p> |
| Technical Perspective | The dispersion model DEGADIS specified in 49 CFR 193 is limited to application for dispersion over smooth, level terrain free of obstacles such as buildings, tanks, or dikes. There is a critical need for a dispersion model that allows consideration of the effects of terrain features and obstacles on the dispersion of LNG vapor clouds. This program will contribute to the further development of any CFD model by providing wind tunnel data for model verification. |
| Project Milestones | A. Simulation with FEM3A of low-wind-speed, stable atmospheric conditions |

- B. Verification for dispersion over rough surfaces, with and without obstacles.
- C. Adapting the FEM3A model for more general application.
- D. Provide assistance and wind tunnel data to DOE for their consideration of FLUENT as an alternative (to FEM3A) model.

Results
In Quarter 8

Work continued to address numerical problems experienced with simulation of low-wind-speed, stable, atmospheric conditions with FEM3A. Steps 1 through 8 in the plan outlined in the first Quarterly report have been completed successfully for the FEM3A model utilizing the Planetary Boundary Layer (PBL) turbulence closure model. We have solved the problems related to stability of the simulations at regulatory conditions of low wind speed and stable atmospheric conditions with FEM3A using the PBL model, and are continuing our program to verify the operation of the model using an updated, verified, version of the k-epsilon turbulence closure model which has been modified to handle dense gas dispersion effects. We anticipate that our experience with the turbulence closure models used in FEM3A, particularly the advanced k-epsilon model which considers extreme density stratification effects upon dispersion, will be directly relevant to DOE's efforts to verify the FLUENT model, or any other CFD model, for possible application to LNG vapor dispersion under 49 CFR 193 and NFPA 59A.

We have experienced delays in our experimental efforts to determine the spectral analysis of the turbulence in the wind tunnel as requested by representatives from DOE-NETL, but we believe that problem is solved now, and we expect to commence with the experimental program in May. The experimental program will focus on measurement of velocities and gas concentrations in the wake region of an LNG tank for use in our CFD model validation efforts, and it will provide the turbulence spectrum for the wind tunnel approach flow. Given the no-cost time extension granted to September 30, 2006, we are on track to finish the contract requirements on schedule.

During the course of this contract there have been critically important developments regarding the use of models in general, and CFD models in particular, for determining vapor cloud exclusion zones for onshore LNG facilities as required by 49 CFR 193 and NFPA 59A.

First, it is generally accepted that vapor cloud exclusion zones determined using the SOURCE5 model to determine the input LNG vapor rate to DEGADIS are in error, as SOURCE5 does not provide for mixing of air with LNG vapor evolved inside the impoundment or the dike/vapor fence system. This is extremely important as there are a number of proposed import facilities which have already been approved using this erroneous procedure, and it is now understood that the method (which ignores LNG vapor mixing with air in the impounded area) is expected to systematically underpredict (predict too short) vapor cloud dispersion zones, resulting in failure to protect the public as intended by 49 CFR 193 and NFPA 59A. We think it particularly important to highlight this issue now, as we expect this finding to be applicable to any model that is considered for use by 49 CFR 193 and NFPA 59A for predicting such vapor cloud dispersion scenarios. Consequently, we are requesting that this information be provided the Department of Transportation and the National Fire Protection Association (the agencies which promulgated 49 CFR 193 and NFPA 59A, respectively) immediately so that they are advised of this critical situation.

Second, based on our experience gained with FEM3A during this two-year contract, we are now confident that simulations of time-limited LNG releases dispersing downwind of impoundment and dike systems cannot be assumed to give the maximum distance (as a function of wind speed) for the low wind speed, stable atmospheric conditions allowed (optionally) by 49 CFR 193 and NFPA 59A. It is

now clear, and it has been demonstrated in wind tunnel and field test programs, that “scooping” action of the wind in entraining LNG vapor/air mixtures from impoundment/dike systems is expected to increase with higher wind speeds, thus tending to lengthen the exclusion zone by increasing the amount and rate at which LNG vapor moves downwind. But since the dispersion downwind of the dike is also expected to be enhanced by the wind speed, thus tending to decrease the length of the exclusion zone, it is anticipated that these competing effects will result in the maximum distances to the limiting, safe gas concentration occurring at an intermediate wind speed. We are continuing to investigate these effects quantitatively, but we believe it is critically important to highlight this issue now, as we expect this finding to apply to any CFD code that is approved for use by 49 CFR 193 and NFPA 59A - and because reliance on simulations at low wind speed, stable conditions allowed currently by 49 CFR 193 are likely to underestimate the requirements for vapor cloud exclusion zones, thus having the potential to endanger the public to greater distances. Consequently, as we believe this finding suggests that the current regulations, which assume that the worst case vapor cloud travel occurs at low wind speeds (the regulation allows the project applicant to opt for the default use of 2 m/s wind speed along with F stability as worst case), do not adequately provide for public safety, we are also requesting here that this information be provided the Department of Transportation immediately so that they are advised of this critical situation. We note that the thermal radiation exclusion zones are already required to be determined at the wind speed that would give the maximum exclusion zone impact – and we recommend immediate consideration of a similar requirement for the determination of vapor cloud exclusion zones.

2.0 PROGRAM OBJECTIVE

The primary objective of this research was to further develop the FEM3A dispersion model for application to general scenarios involving dispersion problems with obstacle and terrain features of realistic complexity, and for very low wind speed, stable weather conditions as required for LNG vapor dispersion application in 49 CFR 193. Near the mid-point of the contract period, DOE redirected the primary effort to provide CHRC's assistance to DOE-NETL in their consideration of the FLUENT CFD model as an alternative (to FEM3A) model for use under 49 CFR 193. The original program involved three principal tasks, and a fourth task was added regarding the assistance to DOE in their evaluation of FLUENT.

Task A – Simulation of Low-Wind-Speed Stable Atmospheric Conditions

It has been necessary to validate the FEM3A model from neutral (stability) wind tunnel boundary layer experiments, since suitable experimental facilities for simulating a stable boundary layer (at the required scale) in a wind tunnel do not exist. However, 49 CFR 193 sometimes requires the model prediction to be made for very low wind speed, stable weather conditions. The FEM3A code had not been applied previously for such conditions, and calculations at the University of Arkansas had shown that FEM3A simulations of stably stratified conditions were subject to numerical stability problems. We were confident that such problems could be eliminated, and research has been underway to modify the turbulence closure model as well as certain boundary condition inconsistencies that had been identified and to verify the model changes by conducting experiments in the University of Arkansas' Ultra-Low-Speed wind tunnel. This was a high priority requirement since the normal application of the code for compliance with the regulation, as well as for application to counter-terrorism issues, frequently require the simulations to be made for such conditions, which are often considered worst-case. The FEM3A code has been modified so that, using the presently recommended planetary boundary layer (PBL) turbulence closure model, it will operate correctly when the regulatory conditions of 2 m/s wind (@ 10 m elevation) and F category atmospheric stability are specified. Present efforts are underway to extend this operability to the model using the k-epsilon turbulence closure model, as it is anticipated that eventually the k-epsilon model, corrected to account for extreme density stratification, will be recommended to succeed the PBL model when the code is used to predict dispersion influenced by terrain features and or obstacles.

Task B – Verification for Dispersion over Rough Surfaces With and Without Obstacles

Previous (to this contract) experiments in the CHRC wind tunnel to validate the FEM3A model prediction of the effect of the presence of tank and dike structures utilized a smooth wind tunnel floor. But further research in the CHRC wind tunnel indicated that the presence of the smooth floor combined with the low wind speeds required to simulate the dense gas effects involved in LNG vapor dispersion can result in the tendency for the boundary layer near the floor to laminarize. Under such conditions the wind tunnel flow is not similar to the atmospheric wind flow because field conditions are normally fully

turbulent (i.e., laminarization does not normally occur at field scale). There are strong indications that the experimental data from the wind tunnel would be more applicable to field conditions, and therefore more useful for model validation, if the floor were artificially roughened.

The primary purpose of this task was to repeat and extend former experiments using uniform roughness elements covering the wind tunnel floor to create turbulence properties similar to field scale wind conditions. The roughness used in this program had already been characterized in a related research program conducted in cooperation with the Environmental Protection Agency; consequently, only the gas concentration measurements had to be repeated. The resulting data set(s) will be a valuable addition to the data archives demonstrating the FEM3A model for application to LNG vapor dispersion prediction.

Data from this Task will be used to verify the FEM3A model for application involving dispersion over rough surfaces (for example, suburban housing) with and without the presence of obstacles such as tank and/or dike structures and industrial buildings. The product of this task will be an advanced, k-epsilon, turbulence closure model (for describing the turbulent mixing involved in the dispersion process) that will allow for more realistic description of dispersion problems with obstacles and terrain features of greater complexity (the real world).

Task C – Adapting the FEM3A Model for More General Application

As more complex applications of the FEM3A model are proposed, it was anticipated that there will be additional questions that can best be addressed by experimentation in the CHRC wind tunnel. The major advantage of this approach is that the specific questions regarding the application of the model to different scenarios can be addressed experimentally without having to recreate all of the experimental conditions in the real scenario, and without the high cost and insufficient controllability that is inherent in larger scale field tests. FEM3A simulations of more complex scenarios will inevitably require experimental verification efforts, and the continued availability of the CHRC wind tunnel for such verification is a necessary adjunct for the successful standardization of the FEM3A model, or of any alternative model, for more general application. Examples of complex scenarios that will be considered are evaluations of vapor fences for containment of flammable gases and aerosols, scenarios containing multiple obstacles, and major terrain features.

Task D - Provide assistance and wind tunnel data to DOE for FLUENT development

CHRC was requested to provide additional wind-tunnel dense gas dispersion data that could be used generally for verification of computational fluid dynamics (CFD) computer models for predicting LNG vapor dispersion influenced by terrain features and or obstacles, and to provide specific assistance to DOE in its consideration of FLUENT as a recommended alternative (to FEM3A) CFD model for approval by the DOT Administrator under 49 CFR 193 and NFPA 59A.

3.0 PROGRAM TIME SCHEDULE

Tasks A and B were pursued concurrently because they are interrelated. However, preparations for Task A, which are computational in nature, were initiated first, with concurrent experimental validation efforts immediately following. Task A was completed in quarter (Q6) for the FEM3A model using the PBL turbulence closure model. We are still working on the stability problem for the FEM3A model using the k-epsilon turbulence closure model. Tasks B, C, and D constituted the primary efforts during the last two Quarters, as all three require continuing experimental work.

| | <u>Q1</u> | <u>Q2</u> | <u>Q3</u> | <u>Q4</u> | <u>Q5</u> | <u>Q6</u> | <u>Q7</u> | <u>Q8</u> |
|---------------------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|
| <u>Task A</u> | | | | | | | | |
| Numerical Stability | X | X | X | X | X | X | X | X |
| <u>Task B</u> | | | | | | | | |
| Model Verification | | X | X | X | X | X | X | X |
| <u>Task C</u> | | | | | | | | |
| General Application | | | | | | X | X | X |
| <u>Task D</u> | | | | | | | | |
| FLUENT Development | | | | | X | X | X | X |

4.0 WORK PERFORMED DURING JANUARY-MARCH 2006 (QUARTER 8)

We continued to work on the verification of the FEM3A model using the k-epsilon turbulence closure scheme as well as to work to solve the stability problems experienced with the model (also using the k-epsilon closure model) applied to regulatory conditions of low wind speed and stable atmospheric conditions. We commenced work to determine the spectral analysis of the turbulence in the wind tunnel approach flow as requested by representatives from DOE-NETL during their visit during the sixth quarter as well as to measure near field gas concentrations and velocities (in the wake of the tank and near the point at which gas overflows the dike), but that work was delayed because of equipment problems with our hot wire anemometry system. We anticipate restarting those experiment tasks in May, and having received notice that the contract will be extended at no cost to September 30, we are on schedule to complete the program by the new contract end date.

5.0 NEW DEVELOPMENTS RELATIVE TO 49 CFR 193 AND NFPA 59A

During the course of this contract there have been new and critically important developments regarding the use of models in general, and CFD models in particular, for determining vapor cloud exclusion zones for onshore LNG facilities as required by 49 CFR 193 and NFPA 59A.

It is now generally accepted that vapor cloud exclusion zones determined using the SOURCE5 model to determine the input LNG vapor rate to DEGADIS are in error, as SOURCE5 does not provide for mixing of air with LNG vapor evolved inside the impoundment or the dike/vapor fence system. This is extremely important as there are a number of proposed import facilities which have already been approved using this erroneous procedure, and it is now understood that the method (which ignores LNG vapor mixing with air in the impounded area) is expected to systematically underpredict (predict too short) vapor cloud dispersion zones, resulting in failure to protect the public as intended by 49 CFR 193 and NFPA 59A. We think it particularly important to highlight this issue now, as we expect this finding to be applicable to any model that is considered for use by 49 CFR 193 and NFPA 59A for predicting such vapor cloud dispersion scenarios. **Consequently, we are requesting that this information be provided the Department of Transportation and the National Fire Protection Association (the agencies which promulgated 49 CFR 193 and NFPA 59A, respectively) immediately so that they are advised of this critical situation.** A paper entitled “LNG Vapor Cloud Exclusion Zones for Spills Into Impoundments”, published in the AIChE’s Process Safety Progress, which details the arguments described summarily herein, is attached as Appendix A.

Based on our experience gained with FEM3A during this two-year contract, we are now confident that simulations of time-limited LNG releases dispersing downwind of impoundment and dike systems cannot be assumed to give the maximum distance (as a function of wind speed) for the low wind speed, stable atmospheric conditions allowed (optionally) by 49 CFR 193 and NFPA 59A. It is now clear, and it has been demonstrated in wind tunnel and field test programs, that the “scooping” action of the wind in entraining LNG vapor/air mixtures from impoundment/dike systems is expected to increase with higher wind speeds, thus tending to lengthen the exclusion zone by increasing the amount and rate at which LNG vapor moves downwind. But since the dispersion downwind of the dike is also expected to be enhanced by greater wind speeds, thus tending to decrease the length of the exclusion zone, it is anticipated that these competing effects will result in the maximum distances to the limiting, safe gas concentration occurring at an intermediate wind speed. We are continuing to investigate these effects quantitatively, but we believe it is critically important to highlight this issue now, as we expect this finding to apply to any CFD code that is approved for use by 49 CFR 193 and NFPA 59A - and because reliance on simulations at low wind speed, stable conditions allowed currently by 49 CFR 193 are likely to underestimate the requirements for vapor cloud exclusion zones, thus having the potential to endanger the public to greater distances. **Consequently, as we believe this finding suggests that the current regulations, which assume that the worst case vapor cloud travel occurs at low wind speeds (the regulation allows the project applicant to opt for the default use of 2 m/s wind speed along with F stability as worst case), do not adequately provide for public safety, we are also requesting here that this information be provided the Department of Transportation and the National Fire Protection Association immediately so that they are advised of this critical situation.** We note that the thermal radiation exclusion zones are already required to be determined at the wind speed that would give the maximum exclusion zone impact – and we recommend

immediate consideration of a similar requirement for the determination of vapor cloud exclusion zones. A paper entitled "United States Regulations for Siting LNG Terminals: Problems and Potential", submitted for inclusion in the upcoming special issue on LNG of the Journal of Hazardous Materials, and further detailing the issues which are more summarily described herein, is attached as Appendix B.

6.0 PLANS FOR FUTURE WORK

Contract tasks are expected to be completed by the new contract end date – September 30, 2006.

APPENDIX A

LNG VAPOR CLOUD EXCLUSION ZONES FOR SPILLS INTO IMPOUNDMENTS

LNG VAPOR CLOUD EXCLUSION ZONES FOR SPILLS INTO IMPOUNDMENTS

Jerry Havens and Tom Spicer
Chemical Hazards Research Center
Ralph E. Martin Department of Chemical Engineering
University of Arkansas
Primary contact: jhavens@uark.edu

ABSTRACT

LNG spills on land are most likely to happen in dike/impoundment areas, introducing the need to consider effects of dike/impoundment walls on vapor holdup and dispersion. The DEGADIS and FEM3A models are approved for determination of vapor cloud exclusion zones in 49 CFR 193, but DEGADIS is limited to prediction of dispersion from area sources over flat, obstacle-free terrain and is therefore not directly applicable to spills in dike/impoundment areas. However, FEM3A, which is a computational fluid dynamics (CFD) model, is directly applicable. Before FEM3A was approved, *ad hoc* methods were widely used to determine input conditions for DEGADIS to estimate dispersion downwind of a dike system for such releases. Vapor holdup and mixing with air in a dike system has been studied in wind tunnel and field experiments, and the fallacy of using an overly simplistic estimation of the time and rate of vapor cloud overflow has been demonstrated. In some cases, these *ad hoc* methods have resulted in exclusion zone determinations that do not provide for public safety as intended by 49 CFR 193. This paper summarizes the current state of knowledge about this important vapor dispersion scenario, including key experimental data that address the question. Applications of FEM3A to determine exclusion zones for a hypothetical spill within a typical dike/tank configuration, as well as shortcut-use of FEM3A to describe vapor overflow from the same impoundment for use as input information for DEGADIS simulation of the dispersion downwind from the dike, are illustrated.

1. INTRODUCTION AND BACKGROUND

Vapor cloud dispersion exclusion zones for credible LNG releases from primary containment and transfer systems are required by 49 CFR 193 and NFPA 59A. Since such spills are inevitably into dike/impoundment areas, estimation of such vapor cloud exclusion zones requires consideration of the effects on gas holdup and dispersion of the dike/impoundment walls. The DEGADIS and FEM3A models are approved for determination of such vapor cloud exclusion zones, but DEGADIS is limited to prediction of dispersion from area sources over flat, obstacle-free terrain and is therefore not directly applicable because of the atmospheric flow diversion that is caused by the dike/tank system [1,2]. Various *ad hoc* methods continue to be used to determine the source conditions (rate and area of vapor release) for input to

DEGADIS to predict dispersion downwind of a dike system. Such methods are commonly based on determination of the rate of evaporation from the spill, coupled with application of a liquid-to-vapor expansion factor, to estimate the time and rate that a vapor cloud overflows the impoundment wall. The overflow gas is typically assumed to be pure LNG vapor and the overflow rate, which can be time varying, is used as input to DEGADIS. For time-limited spills (provided for in 49 CFR 193), such methods often lead to the conclusion that the gas will not overflow the dike, or that if the gas does overflow the dike, the gas overflow will be substantially delayed and will consequently occur at a diminished rate.

Figure 1 shows several dike system configurations in common use; it is adapted from the documentation of the SOURCE5 model. The SOURCE5 model appears to have evolved from the original SOURCE model developed by the Gas Research Institute [3] to determine input information for DEGADIS prediction of vapor cloud exclusion zones extending downwind of a dike system. In SOURCE5, the gas within the dike system and the overflow gas are assumed to be pure. As will be discussed below, air mixes with the gas in the dike system causing the dike to fill and overflow more rapidly than predicted by SOURCE5. For an LNG spill, boil off (evaporation) rates decrease with time for all of these systems because the cryogenic liquid pool cools the dike wall/floor surface as the liquid boils. Because the overflow gas is assumed to be pure, the overflow gas rate from the dike system is just equal to the boil off rate once the impoundment is filled with gas. Consequently, the gas overflow is delayed, and the SOURCE5 predicted overflow rate will be reduced; such effects can be important in reducing the required vapor cloud exclusion zone. Indeed, the choice of dike system offers the potential for minimizing the vapor exclusion zone by incorporation of insulating materials to reduce heat transfer from the dike system surfaces as well as judicious design of liquid surface contact areas. However, there is no basis for the assumption that the overflow gas is pure, and consideration of a wind flow across the facility strongly suggests that the evolving LNG vapors will mix with air.

A brief review of the history of this issue will clarify how we got to the present state. 49 CFR 193 was developed at least partly under the guidance of Mr. Walter Dennis, now deceased, of the United States Department of Transportation. DEGADIS was yet to be developed, and a vapor dispersion model, referred to as the "MTB" (for Materials Transportation Bureau) model, was specified initially in the regulation. The MTB model was not designed to provide for dense gas effects and was subsequently displaced by DEGADIS, which had been developed, as its name implies, for DENSE GAS DISPERSION. However, the DEGADIS model was, and remains, directly applicable only to dispersion over flat, obstacle-free terrain. Mr. Dennis, with whom the principal author enjoyed a direct association, recognized that the "holdup" of vapor in a dike system could have important effects on the ultimate downwind dispersion of the vapor cloud. Consequently, at his recommendation, the Department of Transportation (DOT) and the Gas Research Institute (GRI) initiated in 1983 a research program to

study this question [4-7]. At the outset of the research program, DOT (Mr. Dennis) suggested interim modeling methods for use until such time as the research program results were available. Those interim measures have continued to be used, particularly the assumption that no LNG vapor/air mixing occurs within the impoundment before the vapor cloud overflows. It is a primary conclusion of the research program begun in 1983 that the interim measures were not justified, i.e. the research program provides clear, undisputable evidence that the LNG vapor mixes with air before overflowing the impoundment system and that such mixing with air is an important determinant of the vapor cloud hazard extent.

2. EXPERIMENTAL DATA AND FEM3A SIMULATION

Vapor holdup and vapor/air mixing in an impoundment volume have been thoroughly studied in field and wind tunnel experiments, and the fallacy of using an overly simplistic estimation of the rate and concentration of the vapor cloud that overflows the dike has been demonstrated. All test data to date indicates that vapor/air mixing inside the impoundment is an important determinant of the downwind vapor cloud travel.

Falcon Field Tests

In 1987, DOT and GRI conducted the Falcon test series at the Department of Energy's Liquefied Gaseous Fuels Spill Test Facility at the Nevada Test Site [8,9]. The Falcon tests were large scale field releases of LNG to determine the effectiveness of vapor fences to decrease downwind flammable cloud travel distances following spills into a fenced (containment) area intended to simulate a dike structure. The tests were also intended to provide data for the evaluation of the FEM3A model. Figure 1 shows a photograph of a typical test (thought to be Falcon 1).

The Falcon test series involved five moderate-scale (20 to 66 m³) releases of LNG into a fenced (vapor-containment) area with dimensions 44 m x 88 m x 10 m (tall). The total (vapor containment) volume within the fence was 38,720 cubic meters. Table 1 shows the LNG volumes spilled in each test, along with the volume of LNG vapor (at approximately -260F) that would be evolved, assuming a liquid to gas volume expansion factor of 270. The last column shows the ratio of the volume of LNG vapor formed to the containment volume (Vapor Fill Ratio), clearly indicating that the dike would have contained all of the LNG vapor if it was undiluted with air.

Table 1. Comparison of Spill and Containment Volume for Falcon LNG Tests [8,9]

| Test | LNG volume spilled (m ³) | LNG vapor volume formed (m ³) | Ratio of LNG vapor volume to containment volume |
|------|--------------------------------------|---|---|
| 1 | 66.4 | 17,928 | 0.46 |
| 2 | 20.6 | 5,562 | 0.14 |
| 3 | 50.7 | 13,689 | 0.35 |
| 4 | 44.9 | 12,123 | 0.31 |

However, flammable gas/air mixtures overflowed the fence (vapor containment), and flammable concentrations extended downwind of the fenced area in all four tests. The 2.5% gas concentration extended to approximately 240 m (787 ft) in Falcon 1. Hence, the Falcon tests clearly indicate that neglect of LNG vapor/air mixing within the impoundment cannot be justified, since if the vapor evolved were pure and did not mix with air, the vapor clouds in the Falcon tests would never have exceeded the volume of the impoundment, and consequently would have remained inside the dike.

Chemical Hazards Research Center (CHRC) Wind Tunnel Tests

DOT and GRI jointly sponsored a wind tunnel test program at the University of Arkansas CHRC intended to develop and verify methodologies for regulatory use. Guidelines were to be developed for making wind tunnel simulations, independently or conjunctively with a selected CFD model, to predict dispersion distance where diffusion is influenced by: (a) eddy entrainment from excess capacity LNG vapor detention systems, (b) wake turbulence from on site structures and natural obstacles, and (c) topographically induced diversion or meander. The first goal, which is the subject of this paper, relates to the prediction of the mixing with air of the LNG vapor which evolves from a (liquid) spill and the entrainment by the wind of that vapor/air mixture out of the impoundment volume.

The following discussion focuses on a specific wind tunnel experiment that has been carefully studied at the CHRC: the atmospheric dispersion of vapors from an LNG spill into the annular space between an LNG tank and its surrounding dike. These experiments were performed independently of, but parallel with, the Falcon field tests to study the effects of vapor holdup on dispersion downwind of LNG spills into impounded areas. The primary goal of the wind tunnel experiments and the Falcon tests was to determine the effect of a typical dike and tank structure on the flammable gas extent resulting from an LNG release into the impoundment area, and to provide data which could be used to verify the FEM3A model for such application.

The tank and dike configuration used in the experiment is illustrated in Figure 3 which is a photograph taken upwind of the tank/dike system and from above showing a flow visualization experiment. The tank and dike configuration is a 1:150 scale model intended to represent a typical peakshaving facility with a nominal 1 billion (standard) cubic feet (BCF) storage tank. The tank was located in the center of the dike, on a platform topped with a mesh screen (flush with the tunnel floor surface), that covered a gas source box mounted below the floor. A steady flow of 33.4 slpm carbon dioxide traced with 0.5 slpm (~1.5%) propane was introduced uniformly through the source box into the impoundment area. 150:1 scaling dictates that the equivalent field scale LNG release would be the steady evaporation, at the spill rate, of a continuous release of 0.6 m³/s LNG, considered here to be pure methane. The experiment design information is summarized below.

| | <u>FIELD</u> | <u>TUNNEL</u> |
|----------------------------|--------------------------|-----------------------------|
| Release Rate | 0.6 m ³ LNG/s | 33.9 L CO ₂ /min |
| Dike side length (inside) | 95.4 m | 0.636 m |
| Dike side length (outside) | 169.5 m | 1.13 m |
| Tank Wall Height | 32.5 m | 21.6 cm |
| Tank Top Height | 42.5 m | 28.3 cm |
| Tank Diameter | 46.5 m | 31 cm |
| Dike height | 5.6 m | 3.7 cm |
| Wind Speed | 4.9 m/s at 10 m | 40 cm/s at 6.7 cm |
| Atmospheric stability | neutral | neutral |

The model experiments were releases of carbon dioxide at ambient temperature (typically 23 °C) and atmospheric pressure (typically 14.2 psia) for the following reasons:

- The density of carbon dioxide (at ambient temperature and pressure) is approximately equal to the density of methane vapor at its normal boiling point.
- It is not possible to simultaneously (scale) model the turbulent mixing processes and the surface-to-cloud heat transfer effects in the wind tunnel.
- The isothermal releases, uncomplicated by heat transfer effects, allow for more direct study of the validity of the Reynolds Number independence assumption upon which direct scaling of the lab scale experiments to field conditions would be based.

The scale model experimental conditions, as well as scaled results cited in this paper, assume Reynolds Number independence and consequently are based on Froude Number scaling, as follows:

$$L_m = L_f/150 \quad U_m = U_f/150^{1/2} \quad Q_m = Q_f/150^{5/2}$$

where subscripts m and f denote the model and field scales, respectively, and the symbols L, U, and Q denote the characteristic length, velocity, and volumetric gas flow rate scales, respectively. Since the time scale T is a ratio of the length scale to the velocity scale ($T=L/U$), it follows that the model time scale (T_m) and the field time scale (T_f) are related as $T_m = T_f/150^{1/2}$.

Steady-State Experiment Results

A complete description of the experimental results and their use in the ongoing verification effort which supports the FEM3A model (approved for use by 49 CFR 193 in 2000) has been published [10-12]. Key results are summarized here to consider the central issue of this paper: the vapor/air mixing which occurs following a release of LNG into an impoundment system, and the rate and characteristics of the vapor/air mixture that overflows the impoundment.

Figure 4 shows the FEM3A predicted steady-state extents of the UFL, LFL, and LFL/2 gas concentrations for this vapor dispersion scenario. Wind tunnel scale predictions and predictions scaled to field scale are shown. The typically observed bifurcation of the cloud by the tank/dike structure is clearly evident, as are the presence of highly mixed, but spatially nonuniform vapor air regions within the impoundment (Figure 4b). The concentration of the gas overflowing the dike walls for this particular configuration is slightly above the UFL concentration, suggesting a factor of five or more dilution of the vapor overflowing the dike. It is noted that this FEM3A simulation used a provisional (k-epsilon) turbulence closure scheme currently being evaluated at CHRC under a sub-contract with the Department of Energy [13,14].

The downwind distance to the LFL/2 gas concentration is estimated to be 2.6 m (at wind tunnel scale), and the approximate downwind distance observed in the wind tunnel for this experiment was 3.4 meters. This is an approximately 30% underprediction of the distance indicated by the wind tunnel results. Current efforts are directed to bringing this agreement closer by consideration of more appropriate description of the turbulence closure methods used as well as boundary condition specification issues that have been shown to be important.

We have estimated, using this simulation, the concentration and mass rate of methane vapor overflowing the (field-scale) impoundment and have input that information to the DEGADIS model to determine the maximum downwind extent of the LFL/2 gas contour. The result is approximately 480 meters beyond the dike, which is in good agreement with the scaled wind tunnel data.

Consideration of Impoundment Overflow Conditions

For purposes of comparison, the results described above focused on steady state experimental data and FEM3A simulations. We consider here additional information that is provided by the FEM3A simulation regarding the transient development of the vapor cloud within the impoundment, and the resulting overflow of the dike, all of which are consistent with experimental observations.

The purpose here is to illustrate the use of the FEM3A model for determination of the overflow that would (or would not) occur for a time-limited release of LNG within the impoundment. The case under consideration assumes that the methane gas source begins at time zero (in the simulation) and is represented as a step change from zero rate to an evaporation rate equal to the spill rate ($0.6 \text{ m}^3/\text{s}$ LNG). Consequently, the simulation is for a LNG vapor input rate of approximately $162 \text{ m}^3/\text{s}$ (assuming a liquid-to-vapor expansion ratio at the boiling point temperature of 270).

The volume (field-scale) within the impoundment is approximately $60,000 \text{ m}^3$. If the gas remained pure with a volume expansion factor of 270, the dike would overflow in approximately 370 s (6.2 min). However, scaling the FEM3A simulation results (150:1) using the time scale discussed above indicates that the vapor cloud would overflow the dike in about 21 s (at field scale). Furthermore, the FEM3A simulation results indicate that the maximum downwind extent for the steady-state release is reached at approximately 6 min (at field scale). Consequently, the FEM3A prediction indicates that if this spill were limited to 6 min duration (by proper functioning of automatic shut down features, for example), the downwind extent of the vapor cloud would already be approximately 325 m downwind of the dike edge when the (liquid) LNG spill was stopped.

3. CONCLUSIONS

Extensive data from wind tunnel and field experiments indicate that the assumption that LNG vapor evolving from a spill within an impoundment area would not mix with air, thus lengthening the time to vapor cloud overflow, is untenable.

Since the lower flammability limit of LNG vapor/air mixtures is approximately 5% (reflecting a twenty-fold dilution with air) it follows that vapor/air mixtures overflowing the impoundment can pose hazards to the public and should be provided for in the determination of vapor cloud exclusion zones.

It is unlikely that a “one-size-fits-all” estimate of the amount of air dilution that would occur within an impoundment volume can be specified.

There appear to be only two demonstrated approaches to the determination of the effect upon vapor dispersion of the presence of dike/impoundment walls – physical or mathematical modeling.

Since lumped parameter models, such as DEGADIS, cannot be directly applied to this problem, the most cost effective alternative appears to be computational fluid dynamic modeling.

The FEM3A model, which is approved for such use in CFR 193 and NFPA 59A and is available by contacting the Gas Technology Institute (www.gastechnology.org).

REFERENCES

1. Havens, J. A., and T. O. Spicer, "Development of an Atmospheric Dispersion Model for Heavier-than-Air Gas Mixtures," Final Report to U.S. Coast Guard, Contract DT-CG-23-80-C-20029, May 1985, Volumes I, II, and III.
2. Havens, J. A., and T. O. Spicer, "LNG Vapor Dispersion Prediction with the DEGADIS Dense Gas Dispersion Model," Gas Research Institute Report No. 89-0242, 1990.
3. Atallah, S., M. Sirdesai, and D.M. Jennings, "Source1: A Model for Creating a DEGADIS Input File," Gas Research Institute Report GRI-92/0534, January 1993. (Author's note: The SOURCE5 model was cited by the US Federal Energy Regulatory Commission as having been used to determine vapor cloud exclusion zones in the Final Environmental Impact Statement for the proposed Weaver's Cove LNG Project in Fall River, MA, issued May 20, 2005.)
4. Havens, J. A., and T. O. Spicer, "Evaluation of 3-D Hydrodynamic Computer Models for Prediction of LNG Vapor Dispersion in the Atmosphere," Series of reports (including final report) to the Gas Research Institute, GRI Contract No. 5083-252-0788, 1983-88.
5. Havens, J. A., and T. O. Spicer, "Modification and Extension of LNG Vapor Dispersion Simulation Methods for Regulatory Application," series of reports to Gas Research Institute, GRI Contract No. 5086-252-1287, 1988-present.
6. Havens, J. A., and T. O. Spicer, "Feasibility Assessment of a Conjunctive Modeling Approach for LNG Vapor Dispersion Prediction," Gas Research Institute Report, 1991.
7. Havens, J. A., T. O. Spicer, and H. L. Walker, "Regulatory Application of Wind Tunnel Models and Complex Mathematical Models for Simulating Atmospheric Dispersion of LNG Vapor," Gas Research Institute Report No. 92-0257, August 1994.
8. Brown, T. C., R. T. Cederwall, S. T. Chan, D. L. Ermak, R. P. Koopman, K. C. Lamson, J. W. McClure, and L. K. Morris, "Falcon Series Data Report: 1987 LNG Vapor Barrier Verification Field Trials," GRI-89/0138, June 1990.
9. Chan, S.T., Journal of Hazardous Materials, 30 (1992) 195-224.

10. Havens, J. A., T.O. Spicer, and H. L. Walker, "Evaluation of Mitigation Methods for Accidental LNG Releases: Volume 1/5--Wind Tunnel Experiments and Mathematical Model Simulations to Study Dispersion of a Vapor Cloud Formed following LNG Spillage into a Diked Area Surrounding a Storage Tank," Topical Report for Gas Research Institute, November 1996.
11. Havens, J. A., T. O. Spicer, and H. L. Walker, "Evaluation of Mitigation Methods for Accidental LNG Releases: Volume 2/5--Wind Tunnel Experiments and Mathematical Model Simulations to Study Heat Transfer from a Flat Surface to a Cold Nitrogen Cloud in a Simulated Atmospheric Boundary Layer," Topical Report for Gas Research Institute, November 1996.
12. Spicer, T. O., J. A. Havens, and H. L. Walker, "Evaluation of Mitigation Methods for Accidental LNG Releases: Volume 5/5--Using FEM3A for LNG Accident Consequence Analysis," Topical Report for Gas Research Institute, April 1997.
13. Havens, J. A., Tom Spicer and Kent Perry, "New Models Predict Consequences of LNG Releases," Gas Technology Institute and US Department of Energy Gas TIPS, Vol. 8, No. 4, Fall 2002.
14. Havens, J. A., Tom Spicer, Wendy Sheppard, and Kent Perry, LNG Terminal Siting: FEM3A Vapor Dispersion Model Development, Proceedings AIChE Annual Spring Meeting, Houston, Texas, 2004.

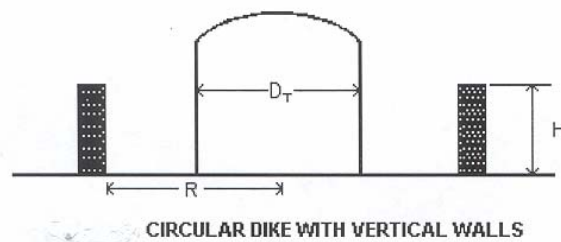
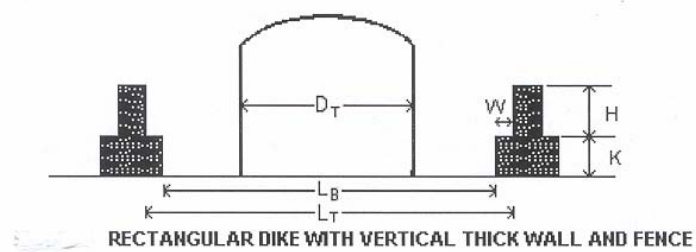
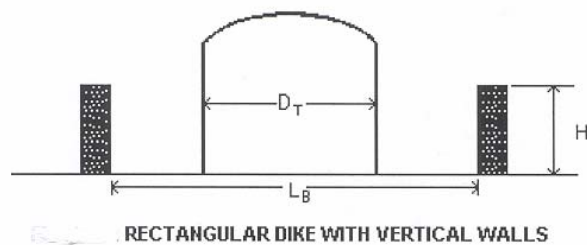
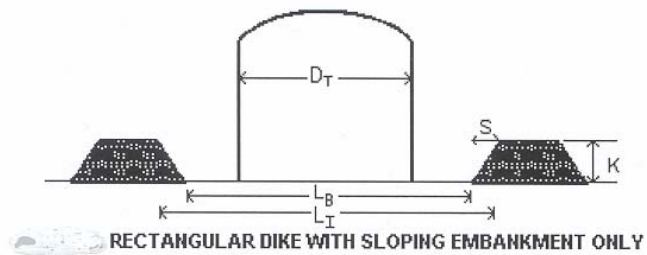
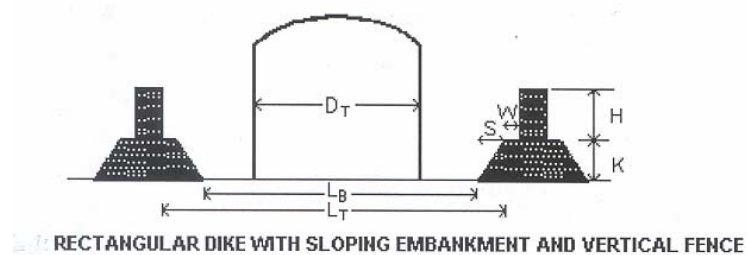


Figure 1. Dike and tank configurations considered in the SOURCE5 model.



Figure 2. Typical Falcon test showing overflow of gas from fence intended to simulate a dike enclosure.

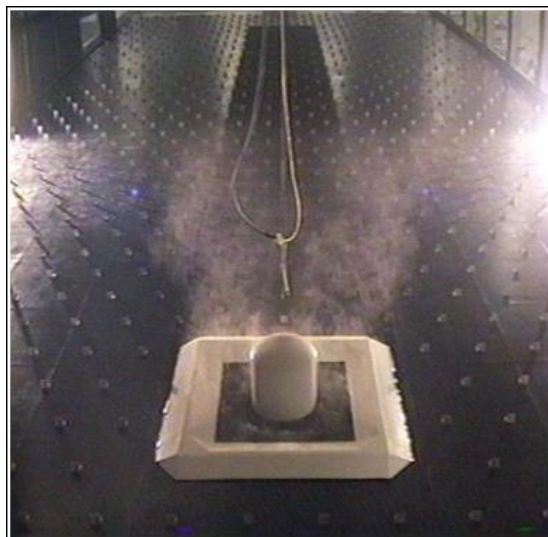


Figure 3. Tank and dike configuration used in wind tunnel experiments.

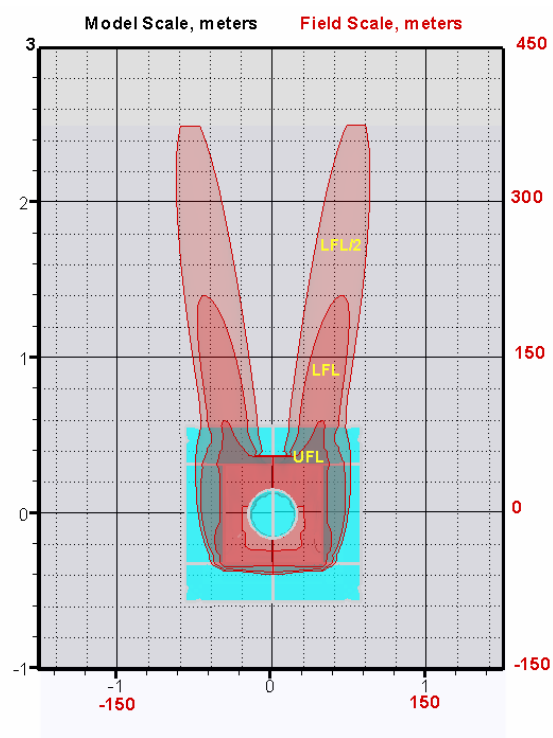


Figure 4a. FEM3A predicted steady-state extent of the UFL, LFL, and LFL/2 gas concentration levels for the example scenario; top view.

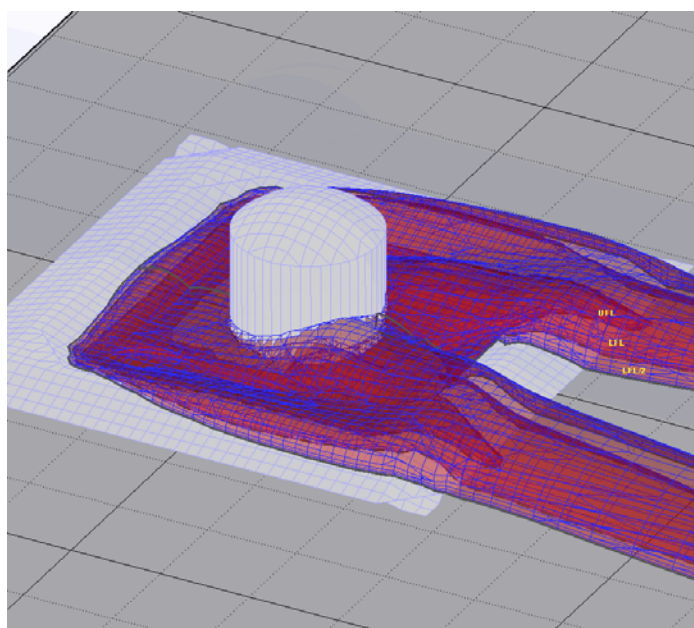


Figure 4b. FEM3A predicted steady-state extent of the UFL, LFL, and LFL/2 gas concentration levels for the example scenario; side view.

APPENDIX B

UNITED STATES REGULATIONS FOR SITING LNG TERMINALS: PROBLEMS AND POTENTIAL

United States Regulations for Siting LNG Terminals: Problems and Potential

Jerry Havens and Tom Spicer
Chemical Hazards Research Center
University of Arkansas

Abstract

The regulations being applied to LNG import terminal siting in the United States are reviewed. There are no requirements for exclusion zones to protect the public from LNG spills onto water. Serious problems with current practices used to determine exclusion zones on the land-based part of the facility are identified. Many of the questions that are considered relate to the use of computational fluid dynamic (CFD) models, which appear to offer the best potential for realistic modeling to determine vapor cloud exclusion zones that result from LNG spills into impounded areas. Failure to use CFD models, which are already approved by the regulation, and continued use of practices which have been demonstrated to be in error, raises important questions of credibility as well as denies the applicant full use of scientific tools that are available to optimize the design of such facilities so as to best provide for safety of the public.

Key words: LNG, regulation, public safety, plant siting, dispersion exclusion zone, thermal radiation exclusion zone

1. Introduction

The United States is considering greatly increased importation of liquefied natural gas (LNG), and there is a rush to identify terminal sites that appear economically viable and that meet local and national governments' requirements to provide public safety.

It is not generally recognized, as three decades have passed, that the current Federal siting regulations, 49 CFR 193, grew out of concerns about the hazards to the public of LNG import terminals proposed in the early Seventies in California at Oxnard, Point Conception, and Los Angeles. At that time four LNG import terminals were already planned or in operation in the United States - at Everett, Massachusetts; Elba Island, Georgia; Lake Charles, Louisiana; and Cove Point, Maryland. As 49 CFR 193 had not been promulgated, the first four import terminals were sited, with Federal approval, under the provisions of National Fire Protection (NFPA) 59A, entitled "Standard for the Production, Storage, and Handling of Liquefied Natural Gas (LNG)". But as a result of heightened public concern, there followed an extensive research program to determine more confidently what consequences could result from credible releases of LNG that could result in fires or explosions [1].

As a result of the research program described by Koopman and Ermak in this issue, the principal hazards from episodic LNG releases were determined to be two, both fire (thermal radiation) hazards – from pool fires and/or vapor cloud fires. A third hazard, the

hazard of unconfined vapor cloud explosion (UVCE), has been relegated to secondary importance based on the determination that LNG vapor, typically containing more than 90% methane, is highly unlikely to explode (i.e., burn fast enough to cause damaging overpressure) if it is not confined (so as to increase the potential for turbulence induced run-up) and if it does not contain abnormally large amounts, say greater than about 15%, “hot” gases such as propane. A fourth hazard, Rapid Phase Transition (RPT), was also relegated to secondary importance, as scaling considerations appeared to limit the amount of danger (overpressure) that could occur. Consequently, the regulations do not presently address either UVCE or RPT hazards. Although these hazards will not be discussed further here, it is noted that neither should be entirely dismissed. The potential for UVCE can be important if the “hot gas” content exceeds normal limits, and it is certainly possible that there will be LNG received at terminal sites in the U.S that exceeds those limiting concentrations. For RPT’s, the remaining concern is for the potential of RPT’s to cause secondary structural damage, which might lead to cascading containment failures.

But there followed a three-decade lull in interest in imported LNG in the United States, while LNG transoceanic shipping grew rapidly in other parts of the world, most spectacularly in Japan. Then, in 2000, following announcements by the United States Government of renewed interest, a rapidly growing list of proposals resulted, finally surpassing fifty. Although the rush began with proposals for onshore terminals, there quickly followed numerous applications for offshore sites, and the first import facility built in the United States since the Seventies commenced operation in 2005 in the Gulf of Mexico.

The need is obvious for carefully conceived regulations that address the question of public safety - What separation distances should be required to ensure that the public is out of harm’s way from credible LNG releases at an LNG import facility? The public wants, understandably, to be confident of its safety, and since 9/11 the terrorist threat has immensely complicated the issue, forcing consideration of the consequences of LNG releases that could result from malicious intent.

Offshore, the Coast Guard and Maritime Administration (MARAD) are the responsible agencies, and the process seems to be proceeding largely without contentious debate about the public safety issue, if not about all of the environmental issues. It is clear that the offshore option can, under the right circumstances, obviate the (onshore) public safety concern. The authors of this paper believe that updating the regulations to consider post 9/11 hazard separation distances will result in a finding that people will be out of harm’s way from offshore LNG terminals of the size presently being considered that are sited ten or more miles offshore.

Regarding the regulations for onshore siting, then, it is admittedly late in the game to be considering changes, with numerous LNG terminal proposals already approved and others in process. But this paper will argue that changes and clarifications are urgently needed. Furthermore, the authors believe that this call for action has much broader implications, as the changes recommended here are necessitated by the failure of the

Government to follow good science and engineering advice to enforce the requirements for public safety that were intended when 49 CFR 193 became law. The authors have observed numerous complaints of disturbingly similar developments regarding the Federal Government's reticence to accept scientific advice in other areas dealing with important issues in which the public is a primary stakeholder – with concerns ranging from environmental to homeland security issues. If these problems are really indicative of a general change in Government policy, we suggest that a much more serious problem – loss of public confidence – looms [2,3,4].

2. The Problems

2.1 Overview

The Federal Energy Regulatory Commission (FERC), newly empowered by the Energy Act of 2005, is the lead agency that determines the acceptability of land-based LNG import terminal sites. But the DOT regulation, 49 CFR 193, which FERC relies on for guidance and determination of those siting requirements, was developed during a period when few people were interested – and the regulation remained largely unused and untested – until 2000. More importantly, 49 CFR 193 has morphed, accompanied and aided by the incorporation of NFPA 59A, into a regulation that does not fulfill the intent of its writers to incorporate the experience and knowledge that resulted from the \$40,000,000 research program concluded in the Eighties. Zinn [5] has described the evolution of 49 CFR 193, beginning with its shortening and incorporation of 59A in 2000. We address here, in summary fashion, only the principal changes that impact questions relating to public safety.

49 CFR 193, and NFPA 59A as well, adopt as their means for ensuring public safety a requirement for *exclusion zones*, defined as areas which are controlled by the terminal operator or the government – effectively prohibiting the public's presence. Although some have disagreed with this approach, preferring a method based on quantitative risk analysis (QRA) which would allow consideration of the probability (likelihood) of events as well as their consequences, 49 CFR is the law, and the authors believe that it should either be followed or changed.

The regulation(s) prescribe the events (spills) which must be considered and then require that specified methods be used to determine the hazard zones that could result. The terminal can receive approval (without a waiver) only if the exclusion zones so determined do not extend beyond the plant boundary onto areas not controlled by the applicant or the Government. Two types of exclusion zones are required to be determined: thermal radiation exclusion zones and vapor cloud dispersion exclusion zones. The problems, both of which can and have lead to downplaying the hazards that are to be quantified (by prediction of too-small exclusion zones), are of two types:

- Misleading or erroneous specification of the input parameters (such as spill amount) or of end-point criteria upon which the exclusion zone extent is based.

- Misleading or erroneous determination (modeling) of exclusion zone extents.

It is important to recognize that 49 CFR 193 and NFPA 59A consider only the land-based part of the facility, indeed FERC's "jurisdiction" effectively ends at the shoreline. Consequently, the requirements for exclusion zones do not apply to spills that could occur from the ship, either when it is in transit to or located at the unloading pier. As spills from the ship might be in larger amounts (because of the vulnerability of the ship containment systems) and as spreading of LNG spills on water could not be controlled, the present requirement leaves a gaping hole in the regulatory provisions. Although the Coast Guard does consider these risks attending spills on water in their evaluation (with FERC) of a terminal application, the fact remains that there are no exclusion zones required presently at an LNG import terminal to protect the public from spills that might occur onto the water. The following discussion relates solely to spills on land.

2.2 Thermal Radiation Exclusion Zones

Thermal radiation exclusion zones are required to be determined using the LNGFIRE III computer model. LNGFIRE III can be obtained from the Gas Technology Institute and is available to any interested party. We believe the LNGFIRE III model represents reasonably the best available technology. We do not object to current requirements for data input to the model – the fire dimensions are (properly) input by describing the dimensions of the impoundments which must be provided for the spills specified. However, we note that the criterion used to delimit the exclusion zone extent (to protect the public) is a thermal flux exposure of 5 kW/m^2 . This thermal exposure to unprotected skin would cause second degree burns in about 30 seconds to persons who could not take shelter. A lower criterion of approximately 1.5 kW/m^2 is used in some regulations [6], being the accepted exposure that would not cause serious injury for meaningfully longer exposure. Consequently, there have been calls to consider lowering the thermal flux criteria to a level that would ensure public safety. We note that the regulation has been reinterpreted recently so as to require that the LNGFIRE III model be used to determine the wind speed at which the thermal exclusion zone is greatest, and that this requirement typically results in a higher wind speed, and greater exclusion zone extent, because of the wind-bending effect (which can place people "under" the fire).

2.3 Vapor Cloud Exclusion Zones

Vapor cloud dispersion exclusion zones are required to be determined using the DEGADIS [7] or FEM3A [8] model(s), both of which can be obtained by interested parties from the Gas Technology Institute. FEM3A, a complex computational fluid dynamics (CFD) model, was approved by the DOT Administrator in 2000 for optional use by applicants desiring to determine exclusion zones so as to include the effects of dispersion that can result from variation in the wind flow (and cloud movement) caused by obstacles (tanks, dikes, vapor fences ...) or terrain features. DEGADIS, approved in the early Nineties, does not allow for consideration of any such complex effects, and is consequently considered to be more conservative (predict longer distance) than FEM3A when both models are applied for the same amounts of LNG released into the wind field.

As shown below, current practice to determine a too-small overflow rate of LNG vapor from an impoundment for input to DEGADIS results in DEGADIS underpredicting the exclusion zone – an example of garbage in, garbage out.

2.3.1 Specification of Design Spills for Determining Exclusion Zones

The LNG spills for which vapor cloud dispersion zones are required to be determined, called *design spills*, key on the regulation's requirements to provide impoundment basins to ensure that the spills are fully contained, i.e., no *liquid* overflows the impoundment. Following historical precedent in 49 CFR 193, the spills for which impoundments are required are ten minute, full-rate spills from the largest transfer line in the plant area served by the impoundment. The largest such transfer line in an import terminal is normally the ship unloading line, which for presently proposed plants would give a ten minute full bore spill volume of around 600,000 gallons.

However, when NFPA 59A was incorporated in 49 CFR 193, and although the requirements for impoundment basins to collect maximum ten minute duration full-line spills remained, the definitions for design spills (for vapor cloud exclusion zones) were changed – to require only what are called “accidental leakage sources”. The spills so determined vary depending on the specific plant area that is being considered, but as they are rarely larger than 3 inches in diameter, the spill amounts are typically smaller by a factor of ten (or more) than the ship unloading line spills. So, while the regulation continues to require provision of impoundment basins that will hold (typically) 600,000 gallons of LNG, the vapor dispersion exclusion zone is allowed to be determined for a spill perhaps one tenth the size. As the requirement for the spill basin clearly suggests the credibility of the larger spill, it follows that the vapor dispersion exclusion distance should be determined for the same spill, not an arbitrarily designated smaller one.

2.3.2 Errors in Application of Models to Determine Exclusion Zones

Spills into impoundments or diked areas (which normally contain storage tank(s) and other service equipment) cannot be directly modeled with DEGADIS (a flat surface, no obstacles model). A program began in the late Eighties, funded by the Gas Research Institute (now the Gas Technology Institute) and DOT, to select a candidate CFD model for consideration as a tool to provide more realistic vapor cloud travel distances for spills in LNG facilities. After more than a decade of continuous work on that issue, the FEM3A model was approved by DOT in 2000 for use in 49 CFR 193. The primary basis for the quick approval of FEM3A was the extensive verification effort using wind tunnel data developed in the purpose-built ultra-low-wind-speed boundary layer tunnel at the Chemical Hazards Research Center of the University of Arkansas. The FEM3A model is intended to provide for consideration of vapor dispersion effects that can not be treated by DEGADIS:

- FEM3A can account for obstacles to the wind and cloud flow, as well as terrain.

- FEM3A can predict the scooping (entrainment) by the wind of vapor clouds forming in an impoundment into which LNG is spilled and subsequently evaporates.

But both DEGADIS and FEM3A must be provided, via input data, with the amount and rates at which LNG vapor enters the wind field, and this is where DEGADIS is being used incorrectly to determine the vapor cloud exclusion zones for spills into impoundments.

The present practice is to use the SOURCE5 model, which calculates the rate of LNG vapor formation following LNG spills into impoundments, given the dimensions and thermal properties of the impoundment. Although there appear to remain questions about the general validity of SOURCE5, there is no disagreement with its defining feature: SOURCE5 determines the volumetric rate of formation of pure LNG vapor (at its boiling point) and integrates that volume production rate to determine when (or if) that pure LNG vapor exceeds the volume of the impoundment, and if so, the time at which overflow occurs and the rate of overflow. That overflow rate is then used as input to DEGADIS to predict the exclusion zones. The principal error in this process is the assumption that no air mixes with the LNG vapor in the impoundment – it is assumed to fill the impoundment as a plug flow! The assumption that no air would mix with the vapor forming in the impoundment if there is a wind blowing over the impoundment has no validity – it can be dismissed on physical grounds alone. Indeed, there have been extensive field tests and wind tunnel tests that disprove the premise [9]. Nevertheless, the faulty process continues, with the result that the vapor cloud exclusion zones so determined are decreased in extent (downplayed), resulting in insufficient protection of the public.

Furthermore, it is now clear as a result of both wind tunnel research and CFD modeling that the current regulation is likely to be in error in allowing low wind speed, stable atmospheric conditions (2 m/s at 10 m, F atmospheric stability) to be used to calculate the worst case exclusion zone distance. Since the scooping action of the wind (the mechanism for removal of the vapor-air mixture from the impoundment) increases with wind speed, while downwind dispersion also is expected to increase with wind speed, there is a clear indication that the worst case wind speed for determining vapor cloud exclusion distance will be an intermediate wind speed - very likely greater than 2 m/s. The LNGFIRE III model has already been required by DOT and NFPA to be used for the wind speed that will give the maximum thermal radiation exclusion zone. The vapor cloud dispersion zone determination should also be required to be made at the wind speed that would give the maximum exclusion zone extent.

2. The Potential

There has been little or no interest by any of the applicants for import terminals in using FEM3A to determine the exclusion zones for spills into impoundments, although it was designed and verified for that purpose. We have observed that the continued use of SOURCE5 with subsequent input to DEGADIS, is expected to underpredict the vapor

cloud exclusion distance for spills into impoundments. In some applications with which one of us (Havens [10]) is involved, there have been claims from the applicant that they were given provisional permission by DOT to perform the calculations using SOURCE5 described above. That “provisional” permission appears to have been given (by DOT) in the mid-Eighties (with the implied caveat that permission was granted only until the research program designed to solve this problem was complete), but it continues to be used today, even though the completed and widely reported research clearly indicates that the assumption of no air mixing is in error.

Although it appears that the industry is being permitted by FERC to use practices that have been demonstrated to be incorrect, the motivation may be understandable, if disconcerting – as they appear to be using this failed methodology in many cases to simply calculate their problem away, however incorrect the methodology. It appears that the industry is not much interested in increased utilization of computational fluid dynamics (CFD) models for this application, which has been proven by extensive wind tunnel and field test research. It is worth noting that CFD models have been used in the consideration of safety issues for offshore facilities. Whether simulating onshore or offshore LNG release scenarios, a CFD model should undergo verification of its theoretical and numerical foundation as well as validation by comparison with relevant wind tunnel and field test research.

But the irony is profound for siting of land based facilities, because it is widely accepted in the scientific community that CFD models probably hold the key to the determination of optimal designs of tank/dike systems that minimize the resulting hazard extent should a spill occur, and also are the best means available for considering mitigation measures that could be applied to reduce those hazard extents. It appears that the applicants are taking a myopic and short term view of the situation. This is neither good science nor good business practice.

References

1. R. P. Koopman and D. L. Ermak, “Lessons Learned from LNG Safety Research”, This issue of JHM.
2. Havens, J., “Terrorism: Ready to Blow?”, Bulletin of the Atomic Scientists, 2004.
3. Havens, J., “LNG: Safety in Science”, Bulletin of the Atomic Scientists, 2005.
4. Havens, J., “Public Hazards of LNG Import Terminal Operations”, Touch Business Briefing: LNG Review 2005, London, UK.
5. Zinn, C.D., “LNG Codes and Process Safety”, Process Safety Progress, Vol. 24, No. 3, September 2005
6. M. Sam Mannan, Jane Y Wang and Harry H West, “LNG Safety – An Update on Recent Issues”, Touch Business Briefing: LNG Review 2005, London, UK.
7. Havens, J. A. and T. O. Spicer, "LNG Vapor Dispersion Prediction with the DEGADIS Dense Gas Dispersion Model," Gas Research Institute Report No. 89-0242, 1990.

8. Spicer, T. O., J. A. Havens, and H. L. Walker, "Evaluation of Mitigation Methods for Accidental LNG Releases: Volume 5/5--Using FEM3A for LNG Accident Consequence Analysis," Topical Report for Gas Research Institute, April 1997.
9. Havens, J. and T. Spicer, "LNG Vapor Dispersion in Impoundments", Plant Safety Progress, Vol. 24, No.3, September 2005.
10. Havens, J., Testimony on Behalf of Fall River, MA, FERC Docket CP04-36.