MITIGATION OF LNG DISPERSION USING TWO DIMENSIONAL (2D) WATER CURTAINS

Tomasz Olewski[1], Omar Basha[1], Simon Waldram[1], Sam Mannan[2] and Luc Vechot[1]
[1]Texas A&M University at Qatar
[2]Mary Kay O'Connor Process Safety Centre, Texas A&M University, Texas

Water curtains are used as a mitigation method to limit dispersion of vapour created after an accidental release of Liquefied Natural Gas (LNG). Experiments were made at Ras Laffan Industrial City (RLIC) using full scale two-dimensional water curtains and coloured smoke as a tracer in air. Air velocities, both upstream and downstream of the water curtain were measured using ultrasonic anemometers under a variety of conditions. Computational Fluid Dynamics (CFD) modelling of these experiments provides a direct comparison between the experimental observations and counterpart theoretical predictions.

INTRODUCTION

Water curtains are used widely as a mitigation measure to limit dispersion of accidental gas releases: although primarily intended for use with gases that are highly water-soluble, (such as ammonia or chlorine), they are also now used in Liquefied Natural Gas (LNG) installations. In such applications their effectiveness relies on transferring upward momentum to the dispersing natural gas and thereby promoting mixing (and hence warming) with air. Water curtains can be used in combination with vapour fences including those that incorporate novel designs that are intended to promote turbulence and mixing.

The rapid growth of the LNG market all over the world has emphasized the need for internationally accepted design standards and risk assessment procedures for LNG facilities (GAO, 2007; Havens and Spicer, 2007) and it is widely accepted that the risk assessment, among others, is one of the most important methods to assure safety. As the first result of this, the US standard NFPA 59A requires the calculation of the consequences of LNG spills to be validated "with a model that is acceptable for use by the authority having jurisdiction that has been evaluated by an independent body" (NFPA 59A, 2009). Computational Fluid Dynamics (CFD) models have become more popular in consequence modelling and risk assessment with the advent of high speed computers and CFD codes. However, to perform a good quality validation excellent experimental data from large scale tests are required.

As part of a research project funded by the Fire Protection Research Foundation, the Health and Safety Laboratory (HSL) have summarized LNG field trial data. These data are recommended for validation of both mathematical models and software (Coldrick et al., 2009). However, this database is not comprehensive and thus not perfect. It summarizes all the good quality data sets that are available, but unfortunately many of these are rather old (from the 1980s) and mostly, which is perhaps more important, they cover only cases for which the atmosphere is neutrally stable. Also, there is no trial regarding an effect of mitigation methods on the vapour dispersion.

The Mary Kay O'Connor Process Safety Center (MKOPSC) at the Texas A&M University has been carrying out continuous, long term, theoretical and experimental research on LNG releases and the associated vapour dispersion as well as methods to mitigate their consequences. BP Global Gas SPU (BP) has sponsored a series of experiments that since 2004 have been performed at the Brayton Field Fire School in Texas. From 2008 associated but independent research has been initiated in Qatar as part of Texas A&M University at Qatar's (TAMUQ's) safety research activities. This is made possible with continued sponsorship from BP and additional support from Qatar Petroleum (QP). As a result of this collaboration, experiments on different scales are being performed in Ras Laffan Industrial City (RLIC). For the purpose of this research the TAMUQ-LNG team has been created: this consists of full-time postdoctoral researchers as well as undergraduate and graduate students from both TAMU in Texas and TAMUQ in Doha, Qatar.

TAMUQ has performed several experiments at the small and medium scales. These were to test the operation and performance of the data acquisition system that has been built at the university, to train team staff and to help establish close collaborative relationships with staff at RLIC. These experiments serve as a precursor to the larger scale, and much more detailed, experiments to be conducted at Ras Laffan Emergency and Safety College in RLIC starting from 2011.

The medium scale experiments presented in this paper, which involve two-dimensional (2D) water curtains as mitigation devices, were performed at an emergency response training facility in RLIC. They were planned so they generated data sets that can be used to validate CFD simulations of 2D water curtains. For safety reasons these experiments cannot involve LNG since they are performed at facilities that would not be appropriate for the associated hazards. High intensity coloured smoke (produced from marine emergency flares) was used as a "tracer" in air and an analogue of the vapour cloud instead.

WATER CURTAIN EXPERIMENT SETUP

The processes involved when a water curtain is used to reduce the concentration of a methane, or natural gas, vapour cloud include the mechanical effect of upward momentum imparted by the water curtain to the gas, mixing and dilution with air, and thus also heat transfer between the vapour, the water curtain and air. The open air experiments presented in this paper were mainly designed to investigate the effectiveness of water curtains through the momentum transfer mechanism.

Prior to mixing, the air itself may have a significant relative humidity. Natural gas clouds are of course colourless. But when they are derived from LNG they normally have the appearance of a white cloud due to the condensation of moisture in the air. It is not always easy to photograph, or video, these and to abstract images of the flow patterns and turbulence, especially when the water curtain is used, since it creates cloud of water droplets which is also white. In this respect the coloured smoke ensures much better visibility in still and video recordings. The experiments described were conducted in autumn 2009 and spring 2010. The flow pattern of the smoke through the curtain was monitored at different parameters related to the water curtain (i.e. water pressure and type of nozzle). The pictures and videos were used in order to assess behaviour qualitatively. The experimental setup, together with dimensions, is shown in Figure 1.

A large (12 m × 2.4 m × 2.4 m) improvised wind tunnel was constructed in the open air and was placed at 90° to the prevailing natural wind direction to help maintain an essentially undisturbed environment. In addition, the entire experimental location was surrounded by the trees, which also reduce the natural wind influence. An artificially induced wind was then created and controlled by using a 1.2 m diameter variable speed fan located in the middle of the tunnel: this supplied an air flow rate of up to 640 $m^3 min^{-1}$. Only data at this maximum air flow are discussed in this paper. The coloured smoke source, a maritime emergency flare, was setup downwind of the fan, at the exit of tunnel. The water curtain nozzle was located a further 1.3 m downwind.

The ultrasonic anemometers monitored the wind velocities at three locations close to the water curtain: one was placed upwind from the curtain nozzle at 72 cm above the ground and the other two downwind at 72 cm and 1.5m above the ground. Three tripods with the thermocouples (symbols tr01, tr02 and tr03 in Figure 1) were used for air temperature measurements. Four thermocouples were located on each tripod at 20, 70, 110 and 190 cm above the ground level. They were intended to measure the air temperature; however the recorded trends were not clear and recorded temperatures were randomly different by a few degrees Celsius. Water was provided from a fire truck and experiments were performed with several different water flow rates. Water temperature (TI-000), pressure (PI-02) and flow rate (FI-01) were measured at the curtain nozzle. Water flow rate (FI-03) and pressure (PI-03) were also monitored at the fire engine control panel and noted manually, since these indicators were not connected to the data acquisition system. Two sizes of Hydroshield Portable Water Curtain Devices – two-dimensional (2D) water curtain nozzles (small and large head), supplied by Angus Fire Armour Limited, UK, were tested. The small head has a 3.5 inch id inlet pipe whereas the larger nozzle has a 4 inch id water delivery nozzle. The water curtain dimensions

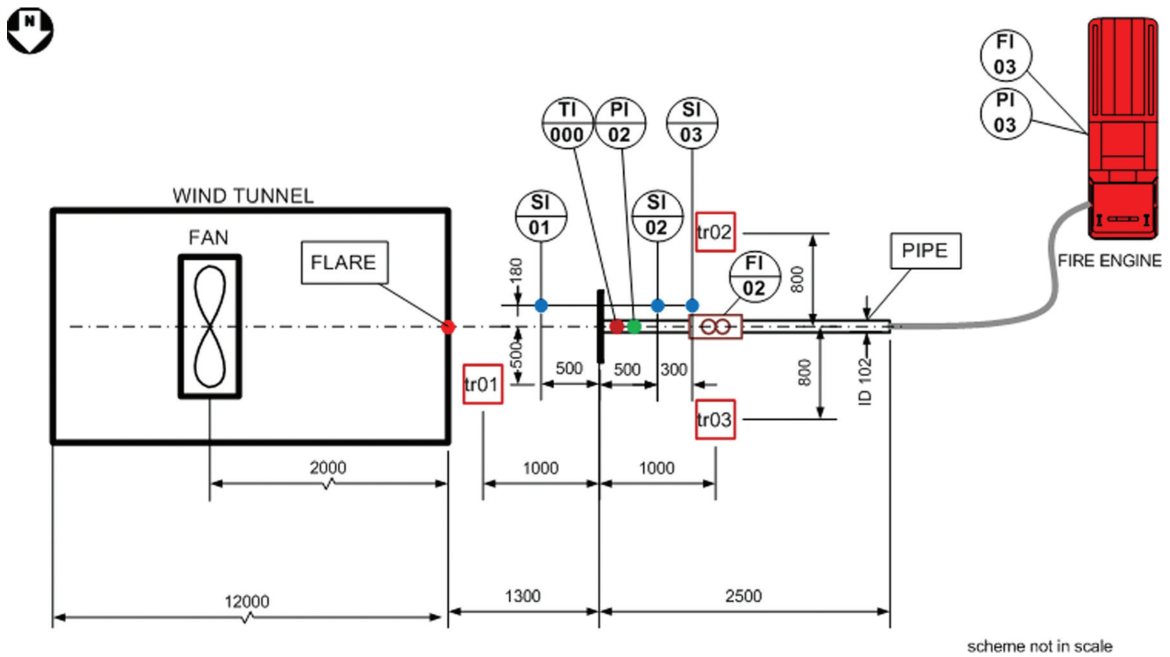


Figure 1. Experimental setup for the water curtain tests at RLIC. Symbols tr01, tr02 and tr03 indicate tripods with four thermocouples located on each at 20, 70, 110 and 190 cm above the ground level. Symbol SI is for an ultrasonic anemometer, TI is for a thermocouple, PI is a pressure transducer and FI a flow meter. All dimensions are in mm

were measured with a resolution of 0.5 metre by two persons to try to assure more objective results. Dimensions were measured to the point where the water fan starts losing continuity. Measurements were recorded using the data acquisition system. Some replicate experiments were also conducted to assess reproducibility. A summary of the test conditions as well as the observed dimensions of the water curtain are provided in Table 1. It should be noted that during some of the tests the data recorded, especially for the water flow rate at the nozzle, were unreliable and thus they are not shown: such data are indicated with "nd" in a table. The poor quality of the water flow rate data, particularly at low flow rates, was associated with a turbine meter. All data were collected by a data acquisition system built from IOtech's DaqScan/2005 data logger and its extension modules.

RESULT AND DATA ANALYSES

The purpose of these experiments was to observe and analyze the air entrainment into and around the water curtain in terms of the flow patterns of the high intensity coloured smoke and also to report the dimensions of the water curtain. Figure 2a shows the two dimensional water curtain sizes as a function of the supplied water flow rate. The "optimum" water flow rate can be found for a particular nozzle. Above this optimum point the dimensions of the 2D water barrier do not change significantly: similarly, below it, the water curtain becomes ineffective. The water curtain dimension data were fitted with Equation (1): the confidence statistics for this are provided in Table 2. Parameter estimation was done in Matlab® R2009a using nonlinear least squares regression ("nlinfit") with the "Robust" option chosen.

$$Dim = \boldsymbol{b}_1 \cdot (1 - e^{-\boldsymbol{b}_2 \cdot q}) \qquad (1)$$

Dim is a particular dimension of the water curtain in meters, Q is water flow rate in $m^3 \cdot min^{-1}$, and $\boldsymbol{b}$ is a two-element vector of parameters. The analysis of dimensions of the water curtains shows almost the same height and width (radius at ground level) regardless of the size of nozzle, and the flow rate of water seems to be the only important factor. The plot of dimensions versus water pressure at the nozzle displays this even better (see Figure 2b).

Figure 3a shows the air/wind velocities at three locations close to the nozzle when the water is switched off. The fan generates an air flow from the right hand side of the figure towards the location of the water spray heads. The axis of the water curtain nozzle is aligned with the vertical bearing line between 90 and 270°. Air velocities range between $0\ m \cdot s^{-1}$ at the centre of the Figure 1 and $2\ m \cdot s^{-1}$ at the outermost periphery. Anemometer SI-01 is located upstream of the water barrier, whereas SI-02 and SI-03 are downstream at two different levels above a ground. It can be seen the wind typically blows from the direction of about 330 to 45° and at a speed between 0.5 to $1.5\ m \cdot s^{-1}$.

Once the water supply is turned on, the 2D curtain forms and as a result of air entrainment, the air flow rate is directed towards the water curtain from **both** upstream

Table 1. Conditions and data for the experiments looking at the interaction of coloured smoke with 2D water curtains

Test #	Type of nozzle	PI-03 at fire engine, barg	FI-03 at fire engine, dm^3min^{-1}	PI-02[1] at nozzle, barg	FI-01[1] at nozzle, dm^3min^{-1}	Water curtain dimensions	
						Radius, m	Height, m
12	small 2D	5	930	2.86	955	14.75	8
14	small 2D	1.2	545	0.761	nd	9	4.5
16	small 2D	1.2	535	0.769	nd	8.75	4.75
17	big 2D	1.2	550	0.533	nd	7	4
18	big 2D	3	830	1.13	nd	9.75	5.5
19	big 2D	3.9	940	1.55	nd	11	6.5
20	small 2D	12	1400	6.77	1440	15.5	9.5
21	small 2D	9	1155	5	1240	14.5	8
23	small 2D	9	1200	5.03	1240	13.5	7
24	small 2D	5.05	950	2.81	920	11.5	6.5
25	small 2D	9	1480	3.59	1440	14	7
26	small 2D	5	1015	2.06	1080	11.5	6.5
27	large 2D	12	1700	4.58	1630	15	8.5
28	large 2D	9	1330	3.53	1420	14.5	7.5
29	large 2D	9	1335	3.55	nd	14	7
30	large 2D	5	780	2.02	1070	10.5	6.5
31	large 2D	3	740	1.27	nd	10	6

[1]Pressure and flow rate at the nozzle were measured by appropriate sensors and were recorded with the data acquisition system.
nd – no reliable data on account of noise or other error.

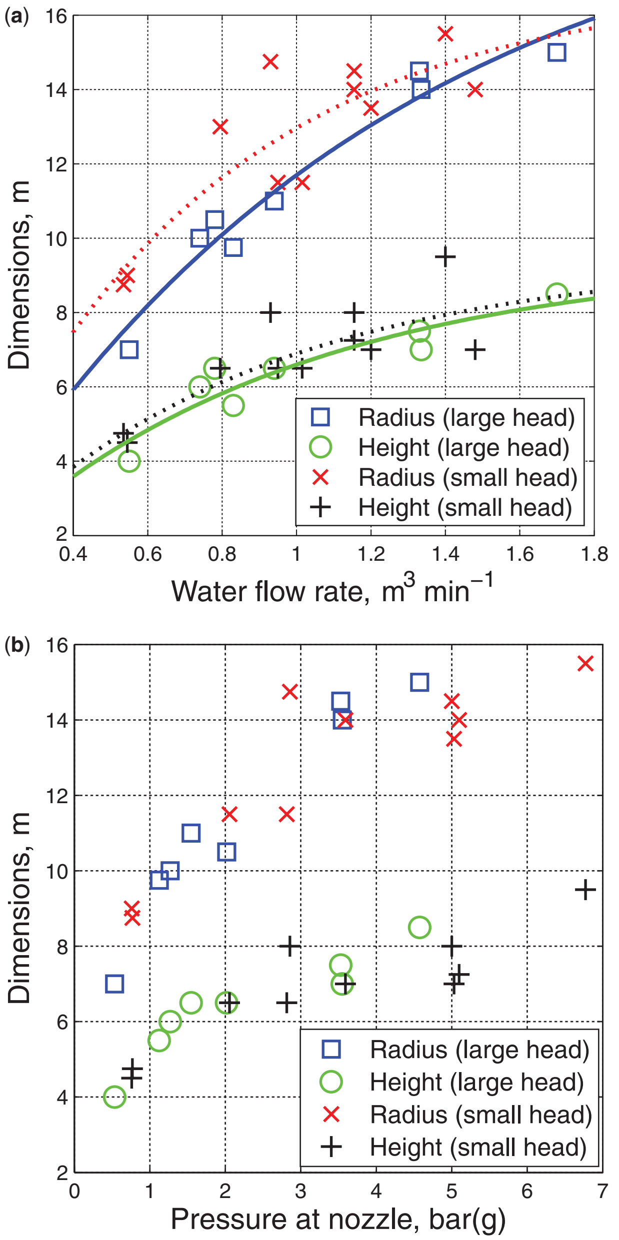


Figure 2. Water curtain dimensions as a function of water flow rate (a) and water pressure at the water curtain nozzle (b). Points are experimental measurements and lines are best fits with Equation (1). Results are shown for both small and large heads of the water curtain nozzle

and downstream sides, i.e. the direction of the horizontal velocity component downstream of the curtain is reversed, (see the data from SI-02 and SI-03 as presented in Figure 3b). This shows the horizontal components of the air velocity vector when the water barrier is present: the flow reversal is from a bearing of about 180°. This effect is very strong, and in practice, there is no air flow in the "natural or prevailing" direction immediately behind the

Table 2. Fitting parameters and statistics for approximation of water curtain dimensions

Dim, m	b_1	Δb_1^*	b_2	Δb_2	MSE**
Radius (large head)	20.1	±4.4	0.87	±0.32	0.37
Height (large head)	9.5	±2.4	1.19	±0.59	0.28
Radius (small head)	16.9	±3.6	1.46	±0.71	1.37
Height (small head)	9.5	±3.1	1.30	±0.89	0.66

*Half of the parameter 95% confidence interval.
**Mean square error of fitting.

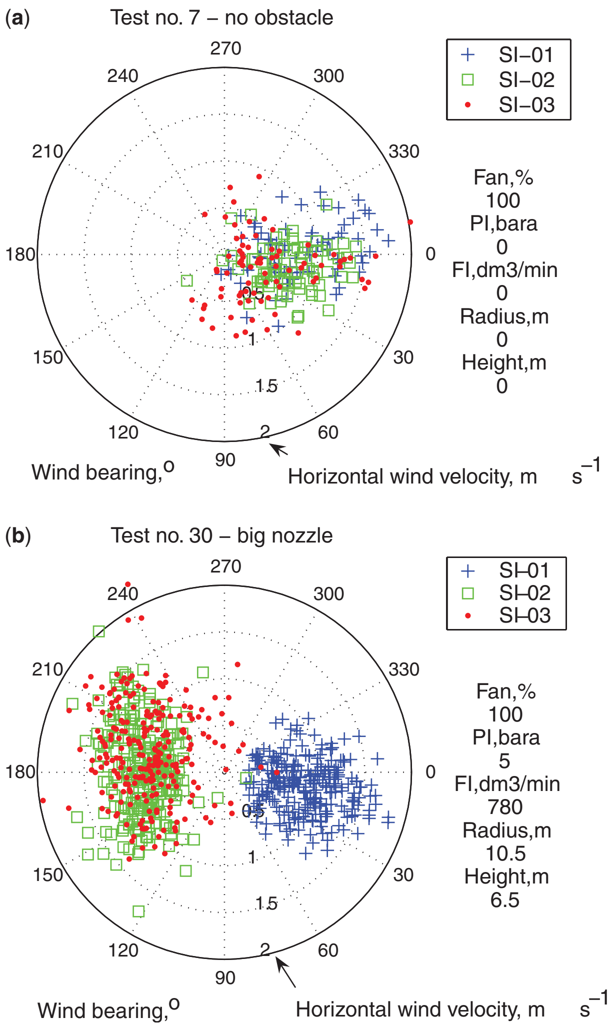


Figure 3. Air velocities and bearings in a horizontal plane with no water (a), and with the water curtain present (b). Blue crosses represent velocities measured by an ultrasonic anemometer located in front of the water curtain (upwind) at 72 cm above the ground. Green squares and red dots represent wind velocities behind the water barrier (downwind) at heights 72 cm (green) and 150 cm (red) above ground level

2D water curtain. Also a bearing variation of wind velocity upstream (blue crosses) stabilizes (at 330 to 30°), whereas behind the water curtain turbulence is stronger and lies between 120 and 240° (green squares and red dots). The result for the large water curtain nozzle with a water flow rate of 780 $dm^3 \cdot min^{-1}$ is shown in Figure 3b but an analogous effect was observed in all cases. Note that the "natural" prevailing wind speed (created artificially by the fan) was rather low and generally did not exceed 2 $m \cdot s^{-1}$. Future experiments with higher air velocities are planned.

Figures 4a and b show the horizontal and vertical components of the air velocities in a vertical plane through the locations where the ultrasonic anemometers were placed (SI-01, 02 and 03). The first figure shows velocities without the water curtain and the second when it is present. Again, the effect of the air entrainment can be seen as SI-02 and SI-03, which are behind the water barrier, show negative values for the horizontal component of the wind velocity (Figure 4b, green squares and red dots). This means that at these low air approach velocities to the water curtain, the air behind the water curtain flows against the natural, prevailing direction. In addition, no effect on the vertical component of the air velocity has been observed, i.e. the distribution of the vertical component with respect to the abscissa has not changed (the points distribution around the abscissa is the same in both Figure 4a and b). This probably means that the ultrasonic anemometers were located too far from the water curtain to observe the air uplift seen in the videos of the coloured smoke flow patterns.

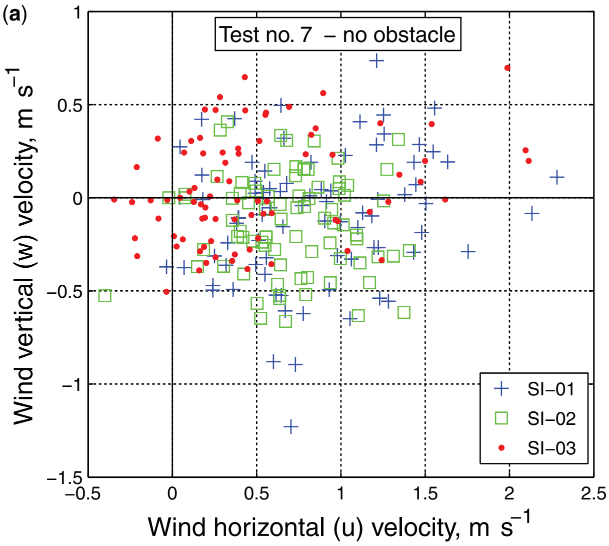
Qualitative analysis of still pictures and videos of these experiments, taken simultaneously with the recorded data, show that the coloured smoke initially stays relatively close to the ground, which is consistent with LNG vapour behaviour when cold. As with the recorded data, the photographic evidence confirms that the water curtain transfers vertical momentum to the coloured smoke, with consequential uplift when the water is turned on. The videos also confirm the induced back-flow behind the curtain. Some pictures are shown in Figures 5a and b. All such observations clearly show the ability of a two-dimensional water curtain to hold back and lift up (and thus also dilute) the vapour cloud due to the mechanical, momentum transfer effect.

CFD MODELLING OF THE WATER CURTAIN

The experiment was simulated using the FLACS® 9.0 code developed by GexCon. The code contains a source model for predicting the evaporation rate of an LNG pool on land, developed by Kim and Salvesen (2002). The water curtain model was created by combining two inbuilt models, those of the "porous wall" and the "air jet".

The porous wall was chosen to simulate the barrier nature of the water curtain and was defined as a half-oval of 8 meters height, 24 m width at ground level and 2 cm thickness: it was located just behind a series of upward air jets, which were intended to simulate the momentum

(a)



(b)

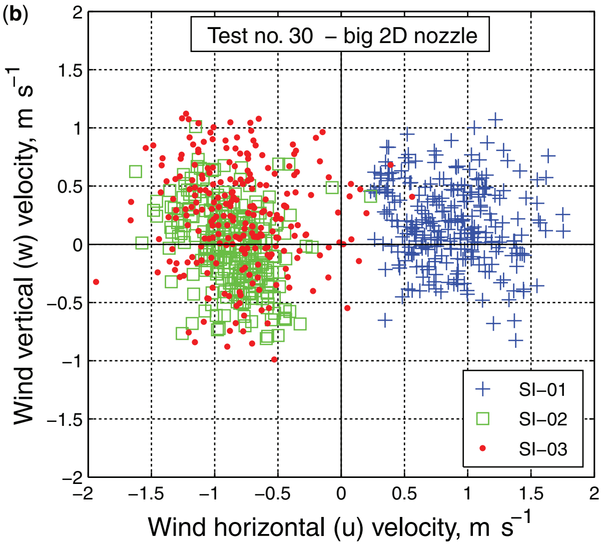


Figure 4. Vertical and horizontal air velocities in a vertical plane through the sonic anemometers with no water curtain (a), and with a water curtain present (b). Blue crosses represent velocities measured by the ultrasonic anemometer located in front of the water curtain (upwind) at 72 cm above ground level. Green squares and red dots represent wind velocities behind the water barrier (downwind) at heights 72 cm (green) and 150 cm (red) above ground level

transfer effects of the water curtain. The porosity of the porous wall varied with height from zero at the ground (no porosity) to 100% at the periphery of the barrier (no water). The velocity of the air jets was varied. The sensitivity of this model to the velocity, as well as temperature and droplet size was performed prior to comparison with experimental data (Basha et al., 2010). The results of these simulations were compared qualitatively with the pictures and videos from the experiments. One typical simulation

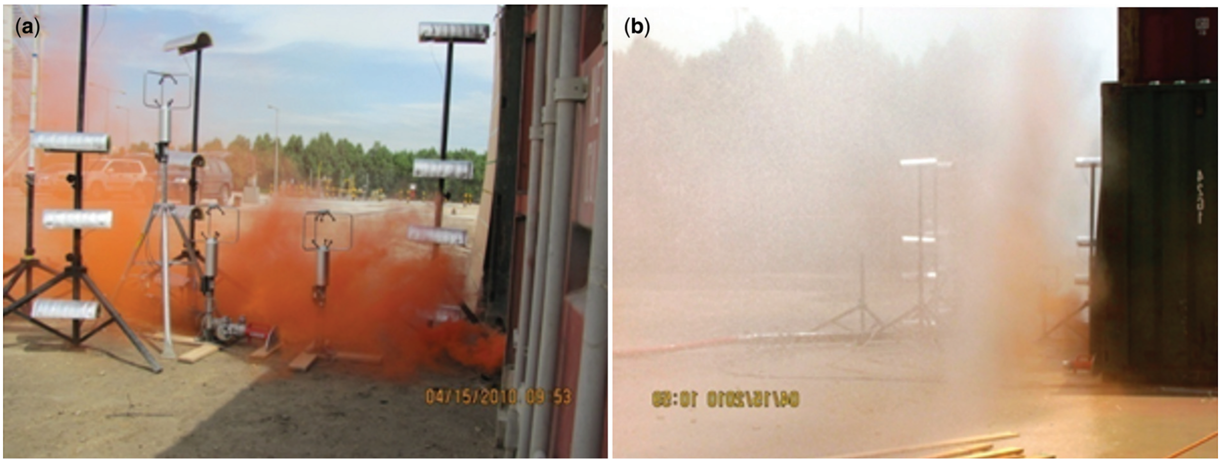


Figure 5. Pictures of experiments: the orange coloured smoke stays relatively close to the ground without the water curtain present and extends across the field of view being carried on the air flow from right to left (a); the smoke is held back and lifted by the presence of the water curtain (b)

result is shown in Figure 6. This shows calculations from FLACS but visualized using the Walkinside® software from VRcontext International s.a. The latter is a software application for 3D real-time visualization and simulation. The water curtain effects of both "lift up" and "hold back" can be seen. Despite being able qualitatively to successfully reproduce experimental results, our FLACS CFD model for the two dimensional water curtain is not yet fully developed and verified quantitatively. Other results which are being used to help validate this CFD model are the data for the local wind velocities and directions from the ultrasonic anemometers on each side of the water curtain. One of the problems, which can be already seen with using an air jet model, is that it gives zero momentum zones close to the air injection points: these are not present in the experimental data. In addition, the air jets in simulations shown in this work are pointed only vertically upwards and there is no momentum transfer in the horizontal direction: this is also not consistent with the experimental data. Also, although the porous wall holds back the vapour, the simulated momentum effect close to a water curtain nozzle is not exactly the same as in an experiment. Thus, there is still much work to be done in order to simulate 2D water curtains adequately. Additional and more extensive experimental data are also needed so as to be able to better validate the model predictions under a wide variety of circumstances. This will eventually lead to confidence in consequence analysis, which involves the mitigation effects of water curtains.

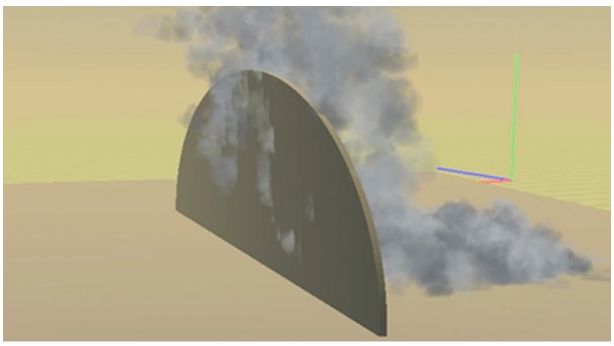


Figure 6. CFD simulation in FLACS of a methane vapour cloud meeting a 2D water curtain. The action of the water curtain is simulated in terms of a "porous wall" and upward "air jets." The gray colour in the figure represents the methane vapour cloud: a small fraction passes through the curtain. Most is lifted up at the frontal face of the water curtain

ACKNOWLEDGEMENT

We would like to acknowledge the long term financial research support provided by BP Global Gas SPU for the LNG safety research being conducted at Texas A&M at Qatar (TAMUQ). We also acknowledge the support of Qatar Petroleum in the form of the facilities used for experiments at RLESC and the provision of staff to work with the TAMUQ-LNG research team.

REFERENCES

Basha, O., Olewski, T. and Waldram, S.P., 2010, A parameter sensitivity analysis for the optimal design of water spray curtains, Mary Kay O'Connor Process Safety Center International Symposium, Texas A&M University, College Station, Texas, October 26–28, 2010.

Coldrick, S., Lea, C.J. and Ivings, M.J., 2009, Validation database for evaluating vapour dispersion models for safety analysis of LNG facilities, Guide to the LNG model validation database, Health and Safety Laboratory report for the Fire protection Research Foundation, February 2009, last rev. May 2010.

GAO, 2007, United States Government Accountability Office (GAO), Maritime security: "Public safety consequences of a liquefied natural gas spill need clarification", report 07-633T, March 21st, 2007 and "Public safety consequences of a terrorist attack on a tanker carrying liquefied natural gas need clarification", report 07-316, February 2007.

Kim, W.K. and Salvesen, H.C., 2002, A study for prevention of unconfined vapour cloud explosion from spilled LNG confined in dike, CCPS Conference Proceedings, Jacksonville, Florida, 2002.

Havens, J. and Spicer, T., 2007, United States regulations for siting LNG terminals: problems and potential, Journal of Hazardous Materials, 140: 439–443.

NFPA 59A, 2009, Standard for the production, storage, and handling of natural gas (LNG), National Fire Protection Association.