Norman Khumalo
Town Manager
Town of Hopkinton
18 Main Street
Hopkinton, MA 01748

April 1, 2016

Response to Sanborn Head Technical Review Report

Following our meetings and discussions regarding Sanborn Head's technical review of the Smith & Burgess report, we wish to issue a response to each of the items posted in the final technical review report. For simplicity, each item will be discussed in the order that it appears in Sanborn Head's report, in tabular format.

#	Page	Sanborn Head Remark	Smith & Burgess Response
1	2	There is a lack of a proper test case for comparison.	While there are several test cases publicly available for LNG spills (e.g. Maplin Sands 1980, Burro 1980, Coyote 1981, Falcon 1987 which are all spills into water), the technical data available for LNG releases over soil, let alone jet releases of LNG, is lacking. Evaluation of the accuracy of the models was done on a qualitative basis, such as by viewing videos of other cryogenic releases (a proprietary video of a high pressure jet release of cryogenic ammonia appeared similar) and also looking at slice planes of pressure, temperature, velocity, and other variables to ensure vaporization and buoyancy were behaving as expected. Our forensics expert also looked at the models and they appeared similar to what he has observed in industry with other cryogenic fluid releases.
2	2-3	Based on our process understanding, the pipe in question that runs under Wilson Street does not operate at the assumed 800 psig pressure. It operates instead at a pressure of about 110 psig... A guillotine failure at the high pressure pump house would by design trigger an automatic shutdown of LNG pumping from the storage tanks…	We did not have access to PFDs, P&IDs, material and energy balances, or other standard information to definitively identify the normal and maximum operating pressure of each pipe in the process. We received drawings of the LNG tanks which included pipe sizes and typical service, as well as plot plans of the main facility and LNG storage area. We also received previous studies that were done by ioMosaic and CH-IV that had pipe size and some pressure information along with the Jim Davis testimony. Without knowing the location of the high pressure pumps or anything further with the process, we went with the 800 psig cited in both the Jim Davis testimony and the CH-IV study. We also did not have information on safety systems such as automatic shutdown. However, it should be

#	Page	Sanborn Head Remark	Smith & Burgess Response
			noted that no safety system in industry is 100% fail-safe and there is a possibility of safety systems not working (many industry incidents in recent history have involved failures of at least 2 different protective safeguards). For the purposes of a worst case scenario model, this pressure could be sustained for a long period of time, particularly if there is no intervention and the safety systems fail to activate for any reason. That being said, we do not know what the actual piping size is at the vaporizers since we now know it is not the piping near Wilson street. If the piping at the vaporizers is smaller, then that will also substantially decrease the maximum flow rate during a guillotine pipe break. As noted in the technical review, we agree that additional simulations should rely on facility specific information to definitively identify appropriate piping pressures, piping locations, and safeguards in place to shut off flow, as well as if to consider safeguards and include their response time in the simulations.
3	3	Similarly, there are also various mechanisms in place that are designed to contain and limit the release of LNG from storage tank failures.	Again, we did not have information on safety systems such as the internal shut off valves mentioned by Sanborn Head. We agree with the recommendation that experts familiar with the Hopkinton facility safety systems be consulted to determine worst-case scenarios.
4	4	…In these cases, Luketa-Hanlin et al. (2007) suggest that the liquid phase is difficult to model in CFD simulations… The Smith & Burgess report does not provide details on the procedure used to model two-phase flow, but if the liquid phase is assumed to be distributed throughout the ground-based grid cell layer, the behavior of the liquid pool may be incorrectly modeled.	There are CFD models that are actively used in industry today for cryogenic applications. GexCon's FLACS CFD software, for example, is approved by FERC for LNG spill and dispersion modeling and has gone through experimental validation. Likewise, CD-Adapco presented us with an example Star-CCM+ model of a cryogenic helium tank with vaporization due to heat intrusion that used a Volume of Fluid (VOF) model. Our model of the cryogenic LNG spill likewise used a VOF model to capture the liquid to gas phase transition appropriately. CD-Adapco spends extensive time and resources validating their software through experimental comparison. If Luketa-Hanlin had a difficult time modeling liquid and gas phases in CFD simulations, they probably were not using the right type of CFD model.
5	4	The potential resistance to vapor transport caused by forested areas	We agree that inclusion of forests as semi-porous elements would have been desirable in the CFD

#	Page	Sanborn Head Remark	Smith & Burgess Response
		in the near vicinity of the Eversource terminal is not explicitly considered. Smith & Burgess addresses this resistance through specification of surface roughness lengths that will affect the way in which wind speed increases with height in the atmosphere, but because cold natural gas vapor stays near the ground (until it mixes with enough air), the flow around individual trees can potentially slow the progress of vapor plumes and introduce additional turbulence/dispersion. Other CFD modelers (e.g., Zeleti et al., 2014) have employed semi-porous elements in near-ground locations to simulate transport directly through forested areas. We recommend investigation of this method should further CFD simulations be undertaken.	model. While we were successful in using these elements and specifying viscous and inertial resistances in the Star-CCM+ steady state solver, the software unfortunately had an incompatibility with these elements between the selected mesh type (hexahedral trimmer mesh – per region meshing) and the unsteady state simulator with the small time steps that are required to develop the LNG jet plume. After making every attempt at trying to include these elements in the model and not having any success, we reported the problem to CD-Adapco and they also could not provide an answer for a workaround. They acknowledged an issue with their software in regards to the mesh. We have been informed that they are planning on fixing this issue at some point in the future, but this likely will not be fixed within the next year. Semi-porous elements work well in the FLACS CFD software, and also work in the ANSYS CFX CFD software. It is unknown how well ANSYS CFX can handle topography in combination with semi-porous elements. In contacting one of their engineers to evaluate the suitability of their software to handle topography, they unfortunately could not provide us with a working example. We would therefore be hesitant to recommend this software without having confirmation that their software can use topography, semi-porous elements, hexahedral elements, and an unsteady state solver simultaneously. As for FLACS, their software currently is unable to handle topography, so unfortunately there may not be an ideal software available to simulate both topography and semi-porous elements simultaneously. The 2014 Zeleti CFD reference, while it describes what would be expected in modeling semi-porous elements in CFD models, unfortunately does not state what software was used as well as if it was a steady or unsteady simulation. The pictures also show flat terrain, which could have been easily done in FLACS. In conclusion, we are left with representing the forests in terms of surface roughness or using a model with the semi-permeable elements but without topography until a better working solution can be found.

#	Page	Sanborn Head Remark	Smith & Burgess Response
6	4	Meteorological conditions are inconsistently applied. Specifically, stable atmospheric conditions are assumed in conjunction with solar heating and an elevated ground temperature. Stable atmospheres typically occur during nighttime periods, and solar heating and elevated ground temperatures occur only during the daytime. Resolving this inconsistency in either direction will serve to decrease the extent of vapor impacts. If nighttime conditions are assumed and the solar flux and elevated ground temperature are removed, heat transfer to the LNG is reduced and cold vapor emissions are generated at a lower rate. If daytime conditions are assumed and the solar flux and elevated ground temperature assumptions are maintained, unstable atmospheric conditions should be assumed, which will increase the rate of mixing between the air and cold vapor.	It is true that the textbook reference on Pasquill class F states that it occurs at night under stable conditions. However, it is important to keep in mind that Class F is an approximation of turbulence conditions that are normally observed during the combined solar and weather conditions. As such, it is possible to have higher or lower turbulence values than what is listed for the class. NFPA 59A and 49 CFR 193 only state that Class F should be used for the 2 m/s wind speed, but these codes never state whether to include or not to include solar radiation in the models. Since we modeled a single worst case scenario for 2 wind speeds (and the lower one only says exclude the worst 10%), we chose a solar radiation that would be considered average for the day to correspond with the average temperature of the region. If you look at actual weather turbulence data (such as that from the NASA paper cited in the report), you will find the turbulence values have a lot of scatter around what it is predicted to be under the particular weather condition. These conditions are therefore theoretically possible and would be consistent with being a worst case scenario.
7	4-5	The Smith & Burgess report does not indicate whether the heat transfer characteristics of the ground were considered. Subsequent information from Smith & Burgess indicates potentially significant errors in the way in which heat transfer to the LNG was modeled. The rate at which cold vapor is generated is largely controlled by heat conduction from the ground to the cold (-260°F) LNG. If the ground surface is assumed to be kept at a constant temperature, the rate of heat transfer, and hence LNG boiling, will be overestimated, probably by a considerable degree as the simulations proceed in time… As an example, Luketa-Hanlin et al. (2007) suggest an independent model outside of the CFD environment be used to predict the	We disagree with several of the assertions being made here. First, Sanborn Head assumed we kept the ground surface at a constant temperature, but this was never the case in the model. In the model, all grounds were modeled as adiabatic (meaning no heat transfer to or from the ground), with the exception of heating from the sun (which was varied on different surfaces depending on a rough estimate of shading from vegetation). This means that only the sun is heating the LNG on the ground surface in the model. Second, they suggested using an independent model (again from Luketa-Hanlin) outside of CFD to predict pool formation and vapor generation from LNG releases. While this may be acceptable in some cases, such a simplification is dangerous and can lead to highly erroneous results when applied to dissimilar scenarios. The weakness of such models outside of CFD is that the physics are generally not well defined and they depend entirely upon

#	Page	Sanborn Head Remark	Smith & Burgess Response
		pool formation and cold vapor generation from the LNG release, and the predicted mass emission rates of cold vapor be used as boundary condition inputs to the bottom (ground surface) layer of the CFD simulations.	empirical data. Ideally, the scenario modeled should be comparable to the application range of empirical data, but it will not be comparable when dealing with different physics regimes (e.g. turbulent jets versus steady spills of LNG). Reading the abstract to Luketa-Hanlin's publication, they mentioned Burro, Coyote, and Falcon LNG tests. These tests were for spills of LNG at low to moderate release rates ranging between 1-30 m^3/minute and were released directly onto water. Without having full access to the article, I would venture to guess that the model they are suggesting is likewise within the same ballpark. The Hopkinton scenarios were modeled with a much higher release rate. The release rate is so high that it comes out as a highly turbulent jet that mixes very rapidly with the ambient air. This phenomenon is not modeled well outside of CFD since CFD is required to model turbulence eddies. It is highly important to model the turbulence eddies the lead to LNG and air mixing heat transfer with jet releases. As for the source of the heat transfer to vaporize the LNG, this actually predominantly comes from the rapid mixing of the LNG with the air. Sanborn Head asserted that running a DEGADIS SOURCE5 model of the LNG release resulted in a large pool forming and filling in the containment area. However, this again comes back to the applicability of the model being used. First, DEGADIS is not a CFD model and does not model turbulence. In fact, it doesn't even model heat transfer from the air. PHMSA has stated that DEGADIS does not account for jetting and flashing of LNG releases from failure of pressurized piping and equipment, and further states that the SOURCE5 model can no longer be used to calculate vapor gas dispersion zones for LNG facilities.[1] Next, Exponent also states that this model is not physically accurate, mainly because it does not account for air entrainment in the evaporating gas and also because it does not adequately model heating of the vapor cloud within impoundments.[2] Exponent likewise suggests using a CFD model for simulating LNG spills. As stated in item #1, a qualitative assessment was used by comparing CFD modeling results with available videos of similar cryogenic fluid releases

#	Page	Sanborn Head Remark	Smith & Burgess Response
			to determine that the release and vaporization nature was realistic. Our assertion that vaporization of the LNG jet predominantly occurs due to mixing in the air before reaching the ground is further confirmed from the LNG Source Term report issued by Health Safety Executive, particularly under section 2.3 Jets. Statements that side with our position include "For the case of an unobstructed jet a large fraction of the LNG may vaporize in the air before the liquid rains out and forms a pool, as shown in tests undertaken by Advantica and Shell" (Dr. Webber et al, HSE RR789 Report). In our case, the jet is mechanically fragmented due to the very high velocities, and the sudden depressurization from moving from very high pressures to low pressures sends shock waves through the liquid, resulting in rapid vaporization of the liquid and the development of an aerosol spray. This spray in turn has a high surface area contact with the air, and in combination with the large amount of turbulence and mixing that occurs with the air, a large amount of heat transfer occurs, thus leading to the large amount of LNG vaporization seen in the CFD models.[3]

Sincerely,

Jesse Brumbaugh, P.E.
Process Consultant IV
Smith & Burgess.

References

1. Gale, John. Director, Office of Standards and Rulemaking. PHMSA Interpretation #PI-10-0017. PHMSA.dot.gov website, accessed April 1st, 2016. http://phmsa.dot.gov/portal/site/PHMSA/menuitem.6f23687cf7b00b0f22e4c6962d9c8789/?vgnextoid=749a7428d17ae210VgnVCM1000001ecb7898RCRD&vgnextchannel=2b9b34d513f95410VgnVCM100000d2c97898RCRD&vgnextfmt=print
2. Exponent Engineering and Scientific Consulting. LNG – Spills & Vapor Dispersion. m.exponent.com website, accessed April 1st, 2016. http://m.exponent.com/lng_spill_out_dispersion/
3. Dr. DM Webber, Dr SE Gant, Dr MJ Ivings & SF Jagger, Health and Safety Laboratory. LNG source term models for hazard analysis. RR789 Research Report, 2010.