MARY KAY O'CONNOR PROCESS SAFETY CENTER
TEXAS A&M ENGINEERING EXPERIMENT STATION

19th Annual International Symposium
October 25-27, 2016 • College Station, Texas

Evaluation of Computational Fluid Dynamics (CFD) vs. Target Gas Cloud for Indoor Gas Detection Design

James McNay BSc MIFireE CFSP MIET, Chair of ISA Committee for Fire and Gas
Ryan Hilditch PhD BSc BA MIFireE SFPE, Technical Safety Consultant

Background

After Piper Alpha in 1988, the oil and gas industry was given an abrupt wake up call to its shortcomings in terms of safety. With particular attention to the behaviour of Hydrocarbon gas and the potential for explosions, it was highlighted that while there was a great deal of academic knowledge behind this issue, very little was available to those designing the fire and gas systems for the industry, and even less of this information had been validated through full scale testing.

As a result of this, the UK HSE carried out a piece of research and analysis of validation works which resulted in the OTO 93 002 [1] document. Even to this day, the document details the only method for the design of gas detection where the intent, specifically, is to prevent damaging over pressures as a result of gas accumulation in partially enclosed areas, which has a sufficient theoretical basis with a review of the empirical evidence to back this up.

The result of this research shows that if a 6m diameter stoichiometrically mixed gas cloud can occur, in an area classified as partially enclosed (blockage ratio around 0.3 - 0.4), then explosion overpressures (overpressures in excess of 150mB typically associated with substantial damage) would be credible in the event of an ignition.

As this research was carried out by experienced practitioners who understood the practical implications of the issue as well as the theory behind the principals, a factor of safety was introduced to allow for construction constraints during installation/ commissioning of the systems, among other factors, and this is the origin of the 5m target gas cloud.

Through the years this has frequently been misinterpreted and simplified to the 5m rule of thumb, which is often incorrectly specified and applied rigidly as a result of practitioners not being aware of the origins and theories behind its application, and the subsequent best practice for gas detection design of using the principals presented in the OTO report, along with practical appreciation of the environment as well as the hardware itself to generate an adequate design (which may not mean having a detector every 5m, but still satisfying the principals of design). The method is often criticised for not having sufficient 'mathematics' around its application however this is generally by those unfamiliar with its origins and the subsequent dangers of deviating from such a design. In fact, unless gas leak detection is being applied (which is generally not recommended), the principals of the OTO simply have to be applied, even if the designer is not aware of applying them at the time. This is because some form of layout of detectors within the area, where gas accumulation is credible, has to be applied, therefore the designer must determine what appropriate 'spacing' or 'cloud size' would be suitable within that area.

Hazard Representation & Analysis

The original (and still widely used) gas detection assessments of a three dimensional volume presents coverage data in elevation 'slices'. The hazard as derived in the OTO suggests that our target cloud is a 5m hard edged sphere of perfectly mixed gas and air (stoichiometrically mixed). In reality the gas cloud will not have a perfectly solid edge at the 5m spherical boundary, however this simplification of the gas behaviour and concentration following a release is necessary in the absence of adequate/ reliable dispersion data for any given scenario; differing diameters of hard edge sphere are often specified to assess gas detection arrangement adequacy in relation to the level of congestion within the area of interest. There also emerged the issue that while we know the sphere will not have a solid edge, it is impossible to predict what it will look like on any given site at any given time. While CFD can be applied to gain a reasonable understanding of what *may* occur on a given day, the differences in interpretation and varying levels of user competence in applying the tools could credibly result in wildly different detection designs for similar areas.

This provides the first common misconception that the coverage of the device is represented in the mapping. This is simply untrue. The target gas clouds we are looking for are not to detect the leaks and presence of gas, but to detect a cloud of suitable volume which, if ignited, will likely generate a damaging overpressure. This is a fundamental principle of the target gas cloud methodology which is often overlooked, with critics believing the 'grid based' layout of detectors is simply to increase the probability of a gas release touching a detector. This is a fundamental misunderstanding of why gas detectors are installed in petrochemical applications.

Further to this point, we often see comparative analysis of the target gas cloud method vs scenario based/ CFD analysis which grossly misrepresents a performance based geographical approach. An example of this is in 'Performance Based Gas Detection: Geographic Vs Scenario Based Approaches using CFD' [2] whereby an area is specified a target 5m diameter cloud size, with only point gas detectors applied. This results in a detection layout that no experienced engineer utilising a performance based geographical approach would recommend. This layout can be optimised by applying widely available gas detection technologies not addressed in the paper, and a performance based approach to the target cloud can also be applied (i.e. not simply applying 5m as the cloud size, but determining what cloud presents the explosion overpressure within the area).

Other misrepresentation of this methodology include 'Performance-Based Gas Detection System Design Using Computational Fluid Dynamics (CFD) Modeling Of Gas Dispersion' [3], and 'A Quantitative Assessment on the Placement Practices of Gas Detectors' [4]. These papers both fall under the issue of misunderstanding the basis of the OTO objective. Within Reference 3, the conclusion based on geographic coverage results in a significant number of point gas detectors. The scenario based approach results in slightly fewer detectors. The issue is that a performance based geographical approach would apply a maximum of 4 OPGDs to this area (which is around 10% of the detector count proposed in the study), potentially only 3. This is before even optimising the target gas cloud size. This therefore shows that in this case, a geographical approach applied correctly with an understanding of the fundamentals can actually reduce the detector count.

It is generally accepted that the industry is moving towards a scenario based approach with the intention of reducing the overall detector counts, however this example shows that in fact an even further optimised design can be achieved using geographic based coverage, while also providing an auditable system that will not have a significantly different detection layout depending upon who has carried out the analysis. This can be seen in The Benefits of Using CFD for Designing Gas Detection Systems [5], when the different detection layouts are analysed based on the scenario based approach.

While some may apply scenario based design under the guise of improving the detection performance over the geographic design, there is simply no baseline to measure this against. There is an inherent assumption in this argument that the recommendations of the OTO were actually applied in industry.

Taking a walk across the vast majority of offshore installations or congested onshore petrochemical sites will quickly highlight that sites barely took note of this recommendation, and as such gas detectors are still located at locations where gas will 'likely' migrate to. Therefore the arguments that there are still a significant number of undetected releases despite the OTO recommendations, is not an adequate critique of the suitability of the methodology as for the most part it is simply not followed. An interesting area of future research would be the analysis of significant undetected gas releases on sites which follow the target gas cloud principle vs. those where the detection configuration was based on likely gas migration.

This also begins to show the issue related to competency in carrying out the analysis. It is important to note that 'competent' is a difficult quality to demonstrate. Naturally with performance based design, we can see significantly varying degrees of competency, and also cross field competency issues. An example of this is an expert in computational fluid dynamics (CFD) designing a F&G system under TR84.00.07 [6]. The individual will no doubt be suitably competent within the field of CFD, however if that individual has no direct F&G experience, the design may be seriously lacking in several crucial areas. The same can also be said vice versa.

This is also an issue experienced in commercial fire engineering and is expertly evaluated by Michael Woodrow, Luke Bisby and Jose L. Torero of Edinburgh University [7] and certainly applies to safety design in the process sector.

Limitations of the OTO Report

While the OTO report provides a robust method in which to design gas detection in congested regions, three important areas remain un-addressed. These areas of neglect are: the effects of gas clouds in uncongested regions; the effect of gas migration; and the effect of gas ingress to non-hazardous spaces. As a result of this gap in knowledge, specific operators introduced their own rules of thumb which have been applied since. These are the areas in which our more modern methods can begin to be applied, in the absence of robust empirical evidence. For this specific example however, indoor applications are to be analysed with respect to HVAC driven environments and the impact this may have on how our detection systems can be designed.

Despite the fact the review of research was carried out in 1993, the recommendations from the study were validated by the IChemE work entitled 'Sensitivity Studies of Offshore Gas Detector Networks Based on Experimental Simulations of High Pressure Gas Releases' in 2006 [8], which provided a more recent justification for application of the findings. It is important to note that no such independent body has validated the use of predictive tools such as CFD when being used in designing F&G detection. This does not preclude us from applying these tools, however caution must be applied. This therefore puts weight behind the application of these tools being carried out in tandem with competent personnel in the field of F&G detection who can apply an appropriate methodology which can be justified, with evidence, as to why such a method is applied/ relevant.

The issue we currently have is that the full scale testing the industry witnessed in the wake of Piper Alpha and the investment made in improving safety methodologies has reduced greatly in recent times. Similar detailed design and justification, based on scientific principles, of the more recent attempts at methodologies for designing gas detection systems are very rare or have been carried out by private firms meaning the data is not easily accessed in the public domain, at least not without a commercial spin - a similar scenario to that in the lead up to Piper Alpha.

With relevance to this particular review of internal gas processing sites, the OTO also did not address the application of gas detection strategies specifically for internal HVAC driven applications. It is therefore the role of the engineer to analyse the difference between the congestion of an open based process site vs the confinement of a solid boundary enclosure. This will have a significant impact on the overall explosion overpressures credible within the enclosure and as such must be addressed when reviewing the geographical approach.

It is an important point to note that the difference between confinement and congestion should not be overlooked and has a significant impact on the overall performance of the gas detection system and the escalation potential involved in the ignition of a gas cloud. This is however out with the scope of this review.

Joint Industry Project (JIP) on Gas Dispersion

In 2000, due to the increase in scrutiny of the target gas cloud methodology and an analysis of the sharp edge of the gas clouds being a topic for discussion, a Joint Industry Project was initiated to validate the true behaviour of a flammable gas cloud.

Included within this research was testing of a 3 dimensional layout of gas detectors (fuel cell oxygen based type) to analyse gas dispersion characteristics.

This data was then reviewed by Micropack (Engineering) Ltd., BP and others, which highlighted (as expected) that the central sphere of the cloud at high flammability reading was subject to a gradient of decreasing concentration, proportional to the rate of dispersion, allowing designers to input an additional, theoretical sphere in which the concentration of gas had fallen, which could then be tied into the specific areas gas detection set points and provide gas detection results specifically for voted systems using the correct detector setting. The problem faced with this methodology was that it is difficult to apply a uniform decreasing concentration for even the most predictable environments (which will then be the case on the day of any given release scenario), and can result in the ability to place gas detection 20m apart (and further in certain circumstances) and achieve sufficient control action coverage in the model, when in reality a damaging gas cloud could credibly accumulate undetected and result in potential explosion overpressures in some credible worst case scenarios. This is a similar drawback to the main criticism of scenario based gas detector placement.

This data was used in the industry to ‘optimise’ gas detection systems, however suffered from similar drawbacks of using CFD modelling to determine gas detection design - we cannot predict with any certainty the environment or leak conditions on any given day. Subsequently the operators, rightly or wrongly, reverted back to using the initial target gas cloud methodology.

Linear Spacing

Recently, operators have opted to design based on a prescriptive 5 meter linear spacing design. This is often incorrectly assumed as the same as the target gas cloud methodology. While it is based on the principals of the target gas cloud methodology, it does not account for the practical application of the findings of the OTO and therefore limits the application of an optimised engineered design using performance based principles which the target gas cloud methodology allows.

Computational Fluid Dynamics (CFD) Modelling

On the market today there are a number of available numerical modelling tools which are now being transposed across to design F&G systems. While there are certain applications where this will be a relevant and acceptable design tool, the increase in desire for the 'pretty pictures' of these tools may in fact be outweighing the requirement of engineering principals. That is not to say they do not have their place in 21st Century gas detection design, but they must be used with caution and limitations in design must be fully understood.

Typically these tools allow the user to analyse gas dispersion and/or the results of ignition of various gas accumulations based on the surrounding environment. There are significant differences however between these tools, and the inherent capabilities of the model and how the Navier-Stokes equations are solved/ converged. Certain models, for example, are better suited for momentum driven releases than others, and certain models cannot account for buoyancy as well as momentum driven fluid flow in transient assessments. Ensuring that we use an appropriate model for the problems we are trying to solve is crucial.

Anyone who has experience with CFD products will be aware of the limitations and it is important to note that many assumptions are included with any CFD modelling project. As a result of this, engineering judgement is still vital in achieving an appropriate model and subsequent design and therefore these assumptions must be fully justified and, where appropriate, provide a credible worst case scenario to ensure the resulting design is fit for purpose, and all associated risk is reduced to ALARP. This is all to be carried out before we even review the implications of the model on gas detection design. If the design process is to be optimised, CFD should be reserved for special/problematic areas of interest.

With reference to the topics discussed in this analysis, again, the physics which underpins most commercial packages (and some free ones) is verified to a good extent so we can analyse external cloud behaviour influenced by wind – the question is does the CFD-user have the knowledge to depict a credible worst-cast scenario in terms of gas hazard for the specific application? The question of what is the worst case scenario with respect to gas detection is not a straight forward one, when the principles of why gas detectors are used are fully understood. An example of this would be a calm day where a low pressure gas release can accumulate and create an explosion overpressure if ignited. This is a worst case scenario for explosion potential, but a good scenario with respect to achieving detection and making the area safe. Another scenario would include high winds which act to disperse the gas quickly and effectively, or a highly pressurised leak which migrates to a different area or out of the site boundary. This is a good scenario for eliminating the explosion potential as the gas is out of the area of concern. This is, however, an undesirable scenario for detection as it is unlikely the detectors will detect this release, and if the leak is of sufficient pressure it may by pass the perimeter detection (if applied). So what is the worst case scenario when placing gas detection? In reality we must model enough scenarios to cover all of these permutations, but in doing so one may arrive at the conclusion that the gas can actually accumulate at any point within that area, and therefore a performance based geographic approach would be carried out. This is why CFD and scenario based mapping should typically be focused upon special applications, for example an internal, HVAC driven environment.

When we review internal environments, the conditions become more predictable and provide an example of an area where additional weight can be added to the use of CFD in detection design.

For internal locations, good engineering principals will allow for a design which will be safe both when the HVAC is in operation and when it is not. Therefore for occasions where the HVAC is running we can utilise CFD tools to review the probable behaviour of the cloud and also analyse whether our target gas cloud is credible with the HVAC running. This therefore allows insight into how to design our gas detection.

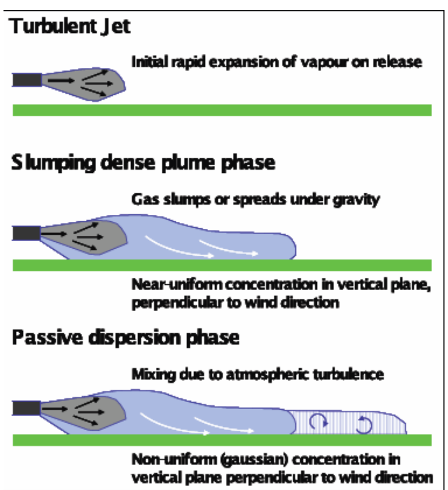
Another important point to note is that the CFD model applied must be trusted and validated on an appropriate scale for the specific application. If an onshore refinery is being modelled for example, it is crucial not to use an unverified CFD tool as it will provide completely different results from the typical industry standard tools, which have undergone significant full scale validation and testing carried out by independent 3rd party testing facilities. CFD models which are in their infancy can provide misleading results and potentially result in a dangerous design.

Potential Hurdles when Incorporating CFD

While CFD modelling is practical within certain applications, the problem, which has been alluded to, is the misinterpretation that these models can predict the outcomes of an event. This is simply not the case. In order to achieve this, a cumbersome number of scenarios (based on the packages available at the time of writing) would be required, which can remove practicality. An additional point would be: who decides how many scenarios are enough? These tools must only be used to provide a greater understanding of the probable outcome (i.e. defining credible-worst case for design basis is the job of the experienced and competent designer), with good engineering principals then applied for the design.

Even retrospective CFD modelling can struggle to replicate the events of a release or fire so how can we realistically use this to predict the conditions on a given day in order to design safety systems? A fine example of this is the fact the retrospective modelling of the Buncefield explosion took some time before CFD experts, after numerous simulations which simply did not present what occurred that day, decided to include the surrounding trees in the retrospective design. This then provided the solution to what caused the achieved flame front velocities and overpressures. This highlights that even if the CFD experts had been involved in design, second thought would probably not have been given to the trees, and the disaster could possibly still have occurred.

Specifically within the context of indoor applications, the orientation and specific leak source identified will provide a significant bearing on the behaviour of the release; therefore it can be challenging to select the credible worst case from this situation. This is highlighted in the following, from the 'OGP Report 434-7 March 2010, Consequence Modelling' [9]:



If this release was pointing up, a sufficient HVAC system would more than likely pick it up and drive it out. Therefore in order to achieve a high enough level of confidence, one has to model a vast number of scenarios in order to be able to validate and justify the removal of gas detectors - for scenarios where the HVAC is both running and out of action, and modelling a leak from every credible leak source, of which there can literally be thousands.

If we take one such incident investigation which Micropack were involved in, a significant explosion occurred from a gas release which exited through ‘cat flaps’ at each end of a building. Conditions were conducive for achieving a well-mixed gas cloud and the direction of the wind was ideal for drift back to the building air intake. This resulted in the explosion within the building, despite having adequate ventilation. This explosion was caused by the overpressures of the internal environment. It is fair to say this was such a unique event, it is credible to suggest this form of scenario may not have been modelled as part of an initial design of the safety systems. Designing based on good principals and F&G knowledge however, may have prevented the disaster and the CFD could have been a useful tool in validating the design against the potential scenarios which may occur.

Example Simplistic CFD Modelling of Internal HVAC Driven Environment

Appendix A presents simulations of a simplified fictitious internal environment to represent how CFD modelling can be used to analyse cloud propagation and migration in an enclosed HVAC driven environment. Each simulation is explained within appendix A, however what these analysis show is that CFD can be a useful tool in reviewing the credible behaviour of the gas clouds in such an environment where there are predictable circumstances and the conditions of release can be credibly defined to determine the credible worst case scenario leak with respect to detection. The analysis also shows that where the gas cloud which could create potentially explosive overpressures (as determined by a consequence or explosion analysis), the target gas cloud methodology could also be applied in tandem with the CFD simulation. This finding is consistent with work carried out by the UK HSE in response to the conservative guidance document PM84 with respect to gas turbine enclosures [10].

A basic recommendation from this would suggest a robust methodology would be to analyse the environment using CFD analysis (for specialised areas such as an internal volume) with respect to the expected dilution/ flushing of the gas cloud, where an assessment of whether to use the tried and trusted target gas cloud methodology, or place the detectors on the expected path of the gas because the target gas cloud has not been produced during normal operation of the HVAC system. It must however be noted that many operators will require the detection system to be fully operational when the HVAC is and is not in operation. The most stringent requirement should always be followed in circumstances like this.

Conclusions

With respect to internal applications where airflow is dictated by the air change rate provided by the HVAC system, good gas detection design should be such that it will operate effectively when the HVAC is working and when it is not in operation, therefore good practice would allow for CFD modelling to give an understanding of the nature of the airflow in an indoor environment to provide insight as to whether the target gas cloud which would provide an explosion overpressure could credibly exist. This would then dictate whether a volumetric detection design would be applied or one which focused more on the placement within the vicinity of HVAC ducting - another area which has had significant research carried out by the UK HSE.

It is often believed that the aesthetically pleasing outputs of most CFD tools provides a prediction of what will happen in the event of a release, this is simply not the case. Generally the results of CFD are such that concise recommendations for a safe system are not presented but rather options for the user to select from. Having the software is not qualification to design F&G which can become apparent when CFD is used as the tool to optimise/ design detection.

It can be gleaned from this that CFD is a useful tool when used correctly for internal and specialised external environments; ambient and turbulent environments; in addition to open and congested regions. It is highly recommended that after the environment is populated within the model, that on site testing is carried out to validate the analyses, for example through smoke testing on site, which is then replicated within the model to ensure consistency in the analysis. This will add credible weight to any findings of the CFD review.

CFD is generally analysed in light of how many scenarios can be run. However this is not the end of the story. Out of the number of simulations run, it is possible, a credible worst case scenario may have been omitted, which if realised on site, and the detection layout has been based on such an analysis, the probability increases that the alarm and control action capability shown in the design document will not be achieved. From a practical point of view, by the time a representative number of simulations have been analysed (from literally thousands of potential leak scenarios - which is even on the conservative end). Depending on the local pressures, it may be likely that gas will be deposited throughout a large volume of an area in question to the point where it would be most desirable simply to apply the target gas cloud methodology throughout. This is a common case in typical process application modules. This approach can be time-consuming and computationally resource-heavy and the cost-benefit is likely to vary considerably in each specific case, where the environment is relatively unpredictable, with so many potential outcomes; it becomes impractical to use this analysis to design the system in many instances. Quantifying a finite number of scenarios as risk-based justification for removal of detectors is dangerous practice. As to what may represent an appropriate minimum number of scenarios is still undefined and no guidance has been produced. This process shows potential, but has a number of gaps which must be addressed through industry collaboration to ensure an appropriate method is to be adopted moving forward.

To this day many ‘practioners’ who are new to gas detection design and claim that F&G Mapping is in its infancy will claim the OTO report is outdated and has little mathematical basis, however it remains the only review with enough practical data we can adequately rely on to safely design gas detection systems. While the introduction of the more modern (but still a well-established field with well-established working practices) CFD modelling tools have their place, there is simply not enough validation in place to rely on these results to solely base our detection designs from.

The main points to be taken with respect to CFD modelling is that the CFD-user must have education and working knowledge of underlying principles of F&G detection design practices in order to properly interpret results in terms of designing the F&G detection system. If this is not the case then the CFD-user must not make specifications (especially optimisation) of detectors, or the only thing we can safely assume is that we are traveling towards a disaster on the scale of Piper Alpha once again.

References

1. Offshore Technology Report OTO 93 002 UK HSE April 1993
2. ‘Performance Based Gas Detection: Geographic Vs Scenario Based Approaches using CFD’, G. Ferrara1, R. Zhvansky1, T. Bengherbia1, S. H. Ledin1, G. Rocha2, I. Ahmed1, D. M. Johnson3, V. Tam, DNV GL
3. ‘Performance-Based Gas Detection System Design Using Computational Fluid Dynamics (CFD) Modeling Of Gas Dispersion’, Kevin Mitchell, Kenexis
4. ‘A Quantitative Assessment on the Placement Practices of Gas Detectors’, A.J. Benavides-Serrano, M.S. Mannan, C.D. Laird
5. The Benefits of Using CFD for Designing Gas Detection Systems, Pablo Giacopinelli & Rohan Samaraweera, Gexcon, UK
6. ISA TR84.00.07 Guidance on the Evaluation of Fire and Gas System Effectiveness
7. ‘A nascent educational framework for fire safety engineering’, Michael Woodrow, Luke Bisby, Jose L. Torero, Fire Safety Journal 58 (2013) 180-194
8. IChemE paper Trans ICHemE, Part B, Process Safety and Environmental Protection, 83(B3):262-26, 2005. Sensitivity Studies of Offshore Gas Detector Networks Based on Experimental Simulations of High Pressure Gas Releases.
9. ‘OGP Report 434-7 March 2010, Consequence Modelling’, March 2010, International Association of Oil and Gas Producers
10. PM84 Guidance Note 84: Control of Safety Risks at Gas Turbines used for Power Generation, UK HSE, 2000

Appendix A: CFD Modelling of a Basic Internal, HVAC Driven Environment

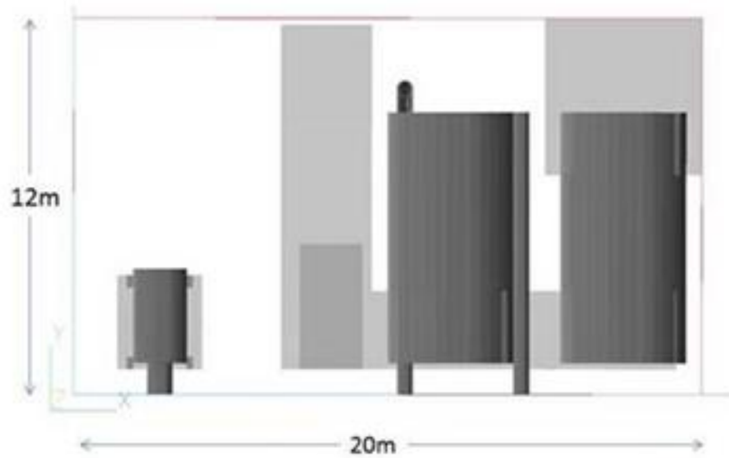


Figure 1a) Compartment plan view

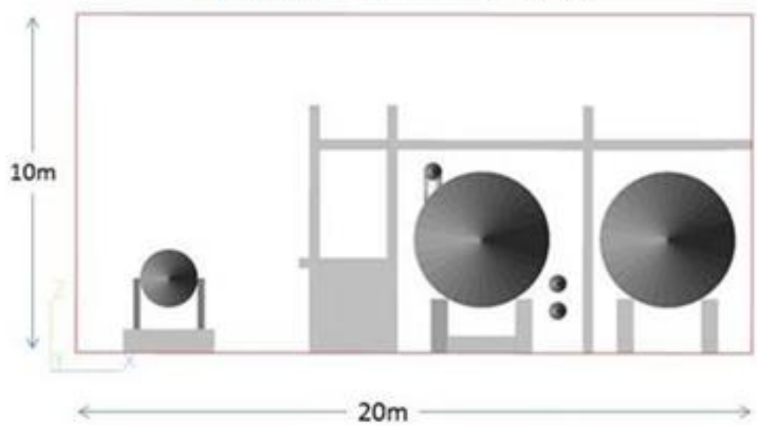


Figure 1b) Compartment lateral view

The compartment (Figures 1a-b) is 20x12x10m tall and contains a small vessel to the left-hand side (LHS), two large vessels to the centre and right-hand side (RHS), a central walkway (at 2m El.) and a small compartment (representing an equipment room) just south of the centre of the compartment. The space was designed to represent a typical, but not overly-congested process area on an offshore platform. Pipework has been omitted from the model in order to improve computation convergence and reduce computation time. In reality, further detailed analysis would record the results as more and more pipework and blockages are added to the space. The goal of the current analysis is to provide a high level overview of the potential impact HVAC design can have on fluid flow including gas movement and concentration distribution. The gas leak considered (100% methane) is assumed to originate from pipework connected to the RHS of the small vessel and comes from a 2cm^2 orifice. The leak propagates to the right of the model and strikes the equipment room wall, effectively killing the initial jet momentum and allowing the gas to disperse outward from this point.

Figures 2a-c show the three HVAC orientations considered here. Scenario 1 (Ambient case) includes two openings (on LHS and RHS walls) at 8m El. which have no forced ventilation and thus allow fluid to flow into or out of the compartment naturally. Figure 2b depicts HVAC design 1 (HVAC 1) where the high elevation extracts (LHS and RHS walls) draw fluid out of the compartment by mechanical ventilation in order to achieve a 50 air-change-rate per hour (ACH) ventilation flow while lower level openings at 3m EL (LHS and RHS walls) allow fluid to be drawn into the space naturally. Figure 2c shows HVAC design 2 (HVAC 2) where-by extracts at 8m EL on the RHS wall draw bulk fluid out of the compartment to achieve the same 50ACH and air is drawn naturally into the compartment from the two inlet openings at 3m EL on the LHS wall. Figures 2a-c include inflow/outflow arrows for clarification. All inlet/outlets are 1m x 1m and are located at 2m and 9m in the y plane.

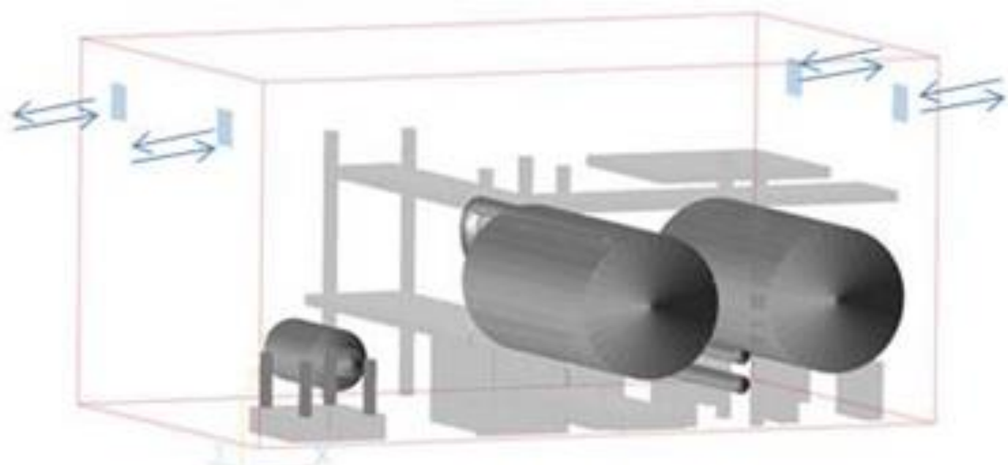


Figure 2a) Ambient (natural ventilation)

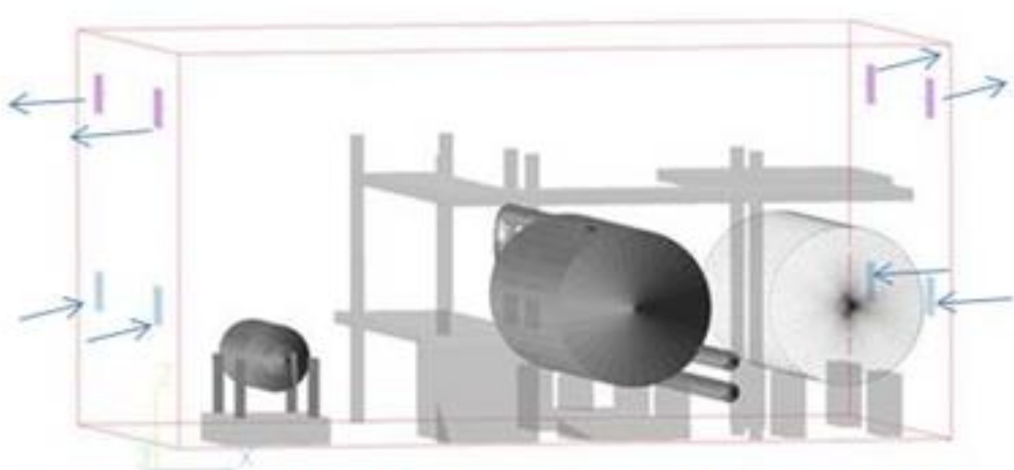


Figure 2b) HVAC design 1 (Mechanical extract LHS and RHS)

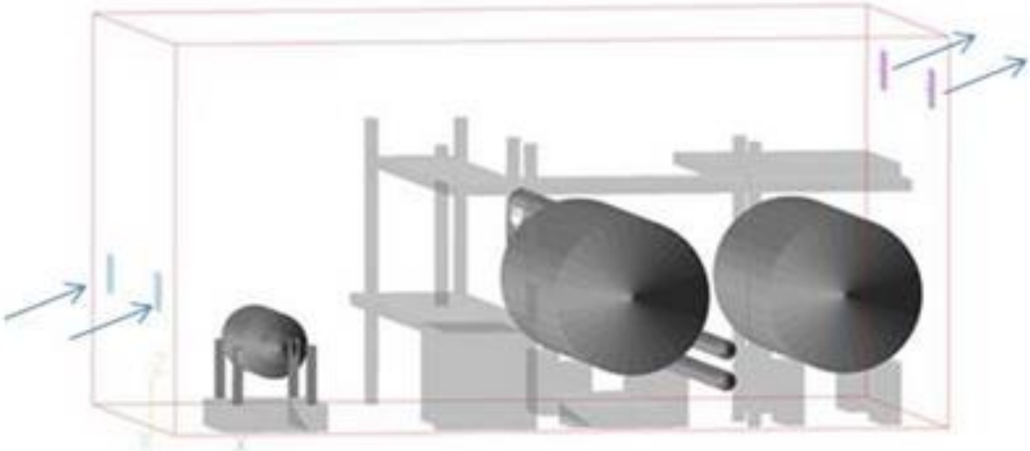


Figure 2c) HVAC design 2 (Mechanical extract RHS only)

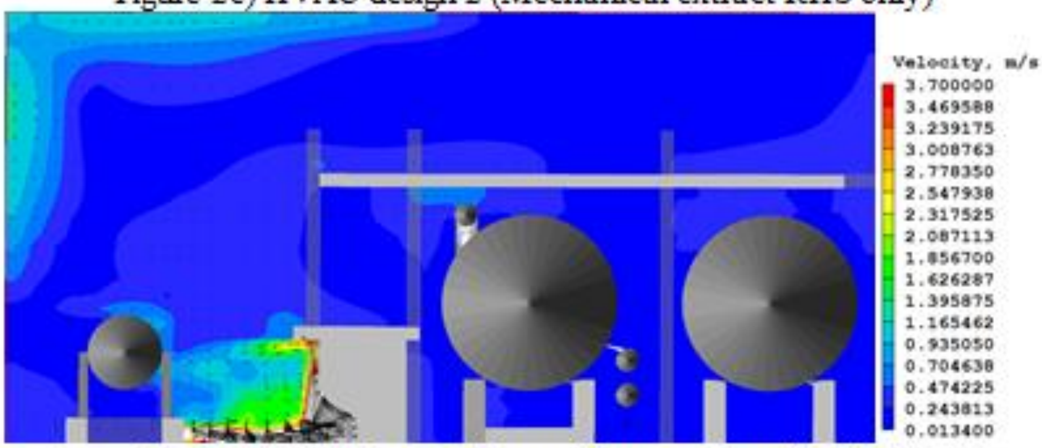


Figure 3a) Velocity profile, ambient, y=2.2m (Section)

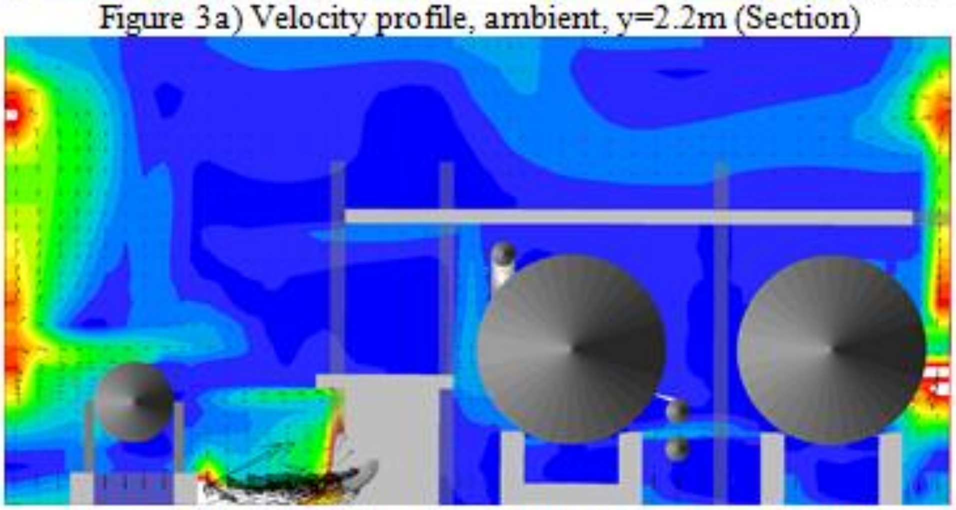


Figure 3b) Velocity profile, HVAC(1), y=2.2m (Section)

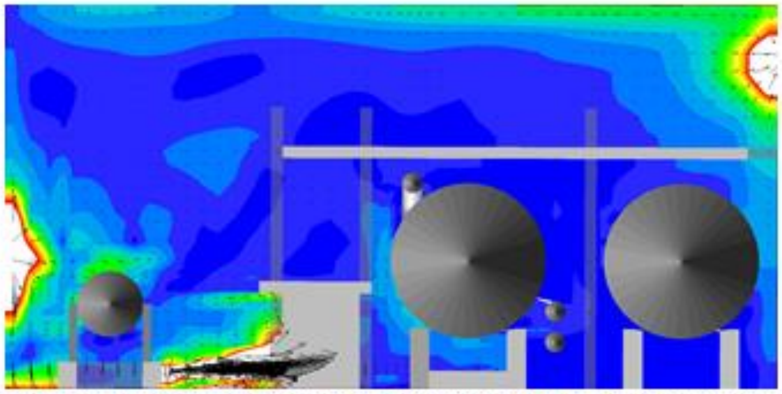


Figure 3c) Velocity profile, HVAC(2), y=2.2m (Section) [White areas represent >scale max.]

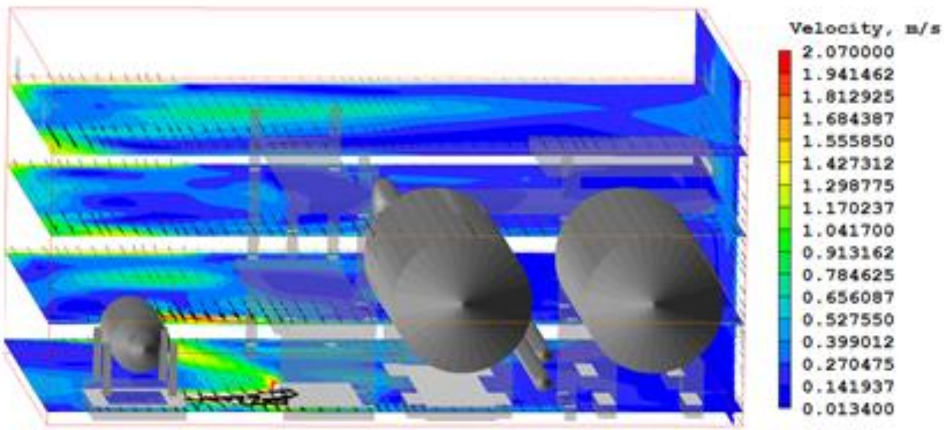


Figure 4a) Velocity profile, Ambient, (y=8, 5.5, 3, 0.4, x=20)

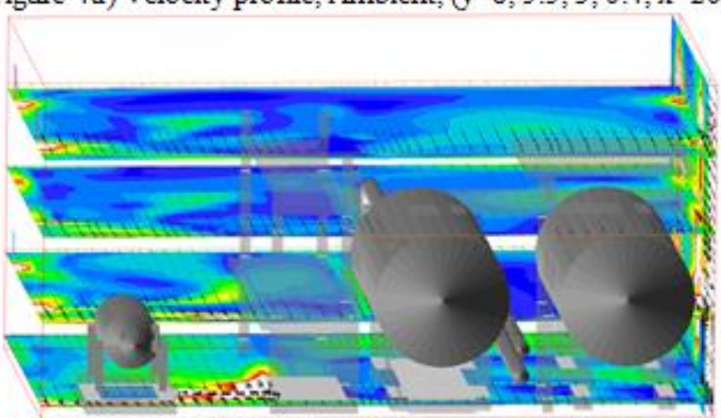


Figure 4b) Velocity profile, HVAC(1), (y=8, 5.5, 3, 0.4, x=20)

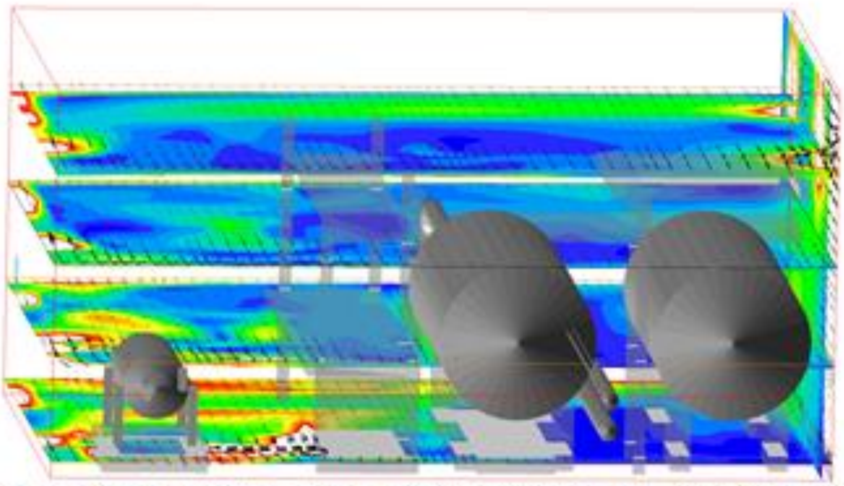


Figure 4c) Velocity profile, HVAC(2), (y=8, 5.5, 3, 0.4, x=20)

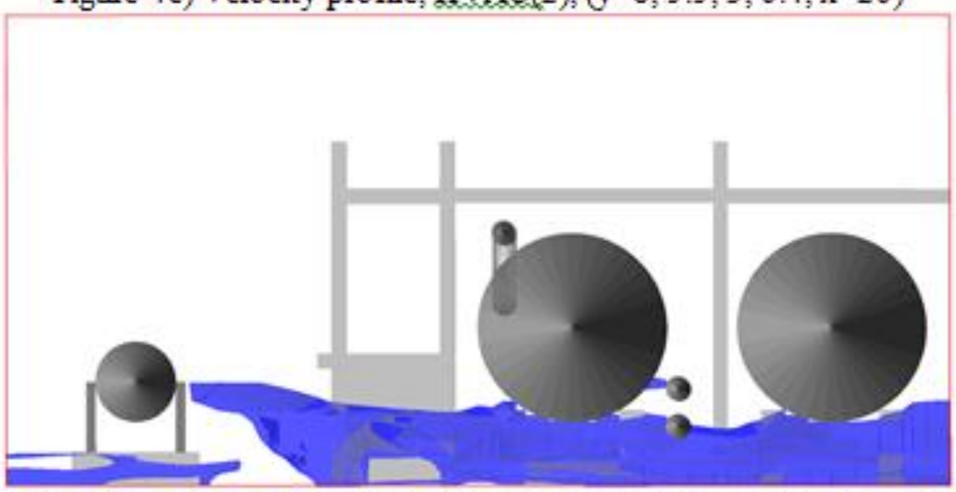


Figure 5a) Ambient, Gas concentration Iso-surface at 20%LEL (Section)

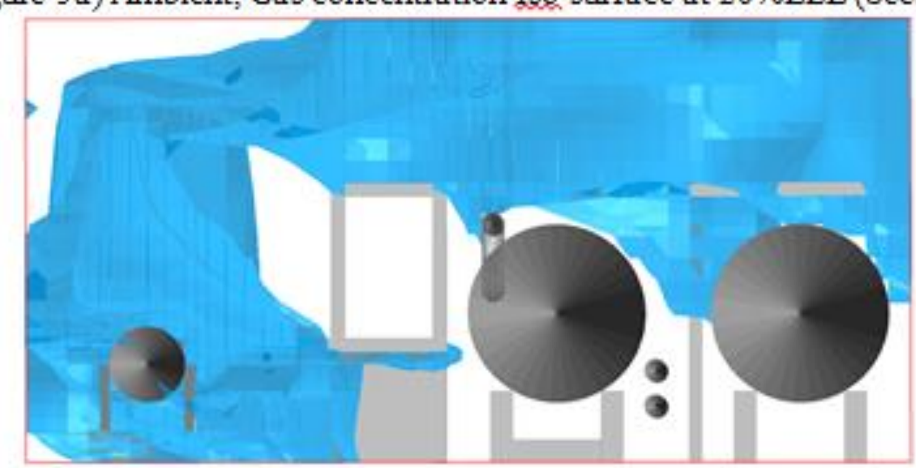


Figure 5b) Ambient, Gas concentration Iso-surface at 60%LEL (Section)

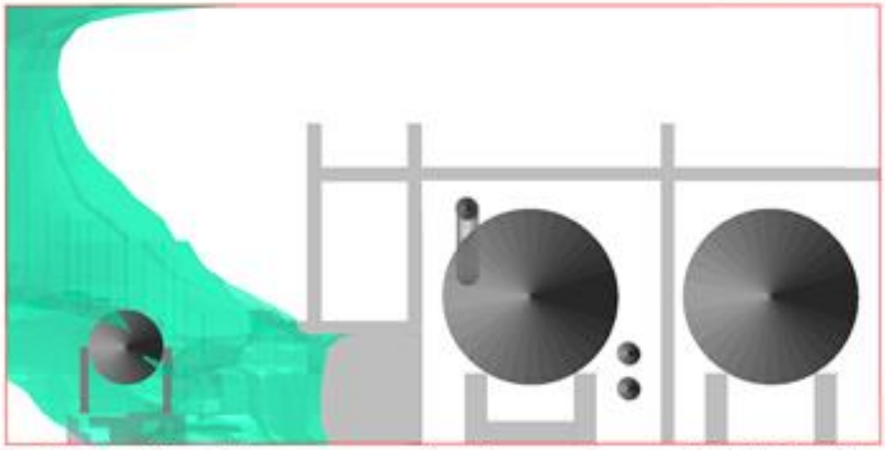


Figure 5c) Ambient, Gas concentration Iso-surface at 100%LEL (Section)

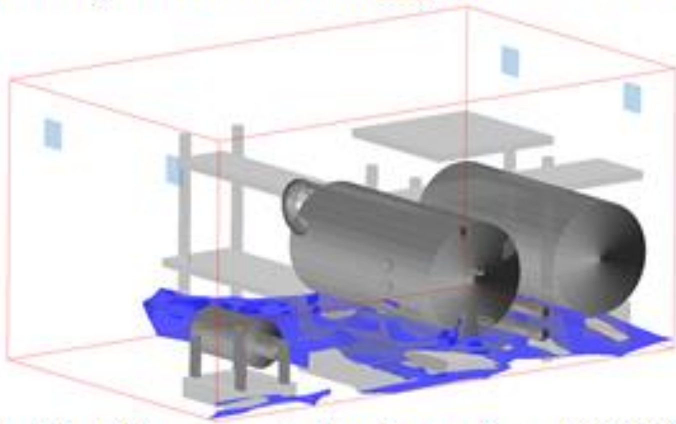


Figure 5d) Ambient, Gas concentration Iso-surface at 20%LEL (3D view)

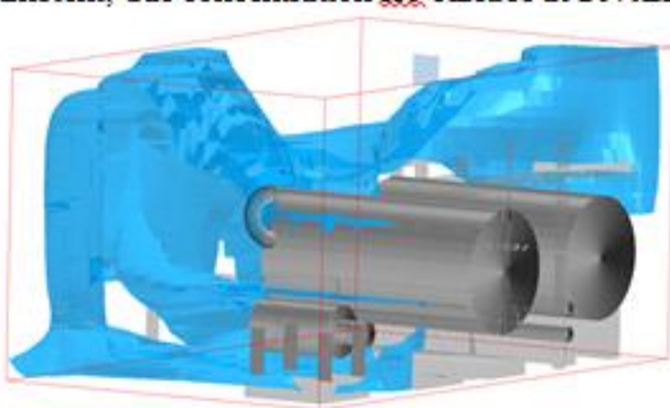


Figure 5e) Ambient, Gas concentration Iso-surface at 60%LEL (3D view)

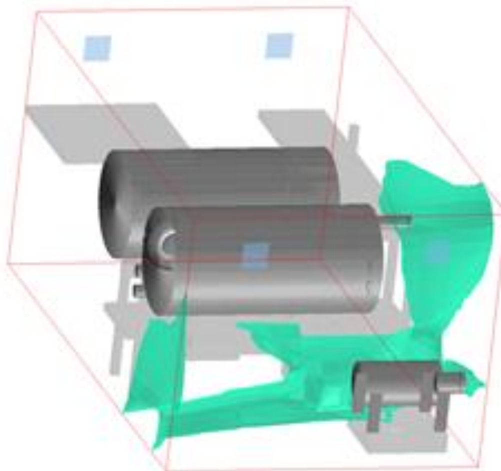


Figure 5f) Ambient, Gas concentration Iso-surface at 100%LEL (3D view)

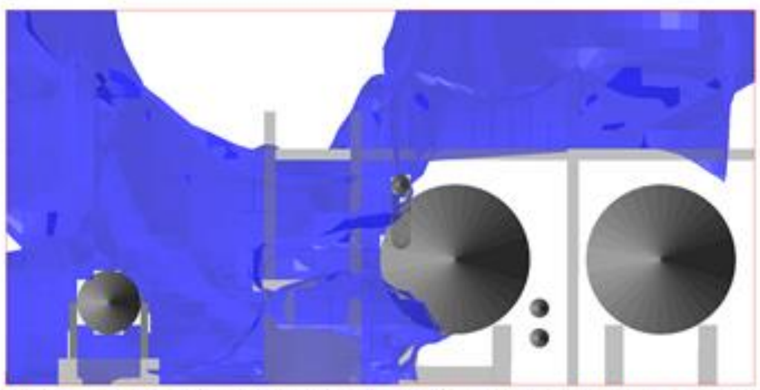


Figure 6a) HVAC(1), Gas concentration Iso-surface at 20%LEL (Section)

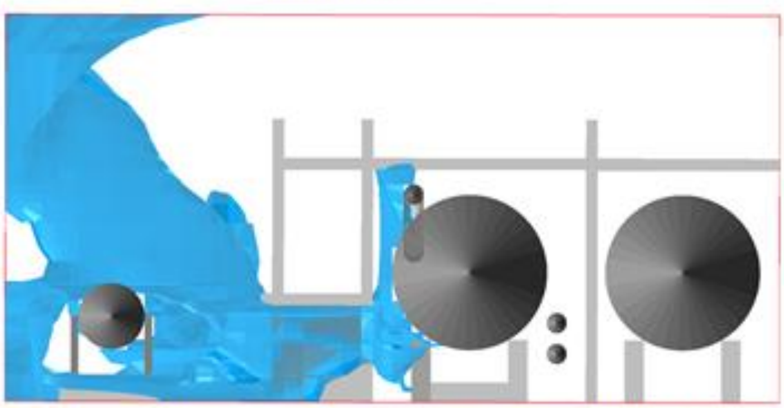


Figure 6b) HVAC(1), Gas concentration Iso-surface at 60%LEL (Section)

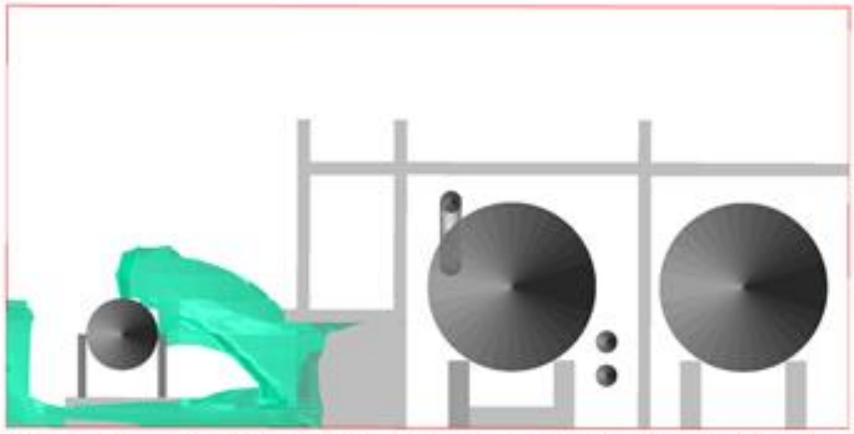


Figure 6c) HVAC(1), Gas concentration Iso-surface at 100%LEL (Section)

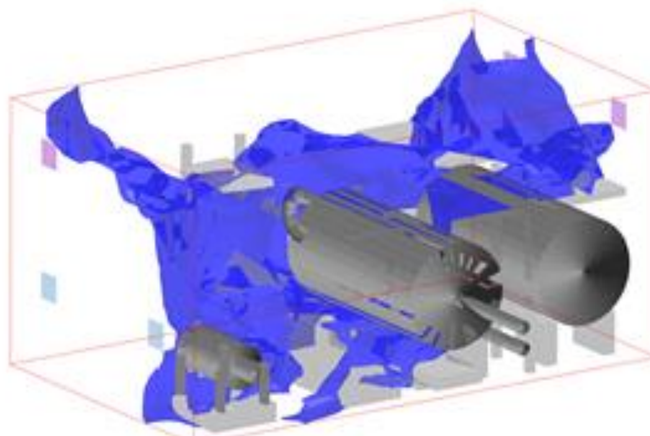


Figure 6d) HVAC(1), Gas concentration Iso-surface at 20%LEL (3D view)

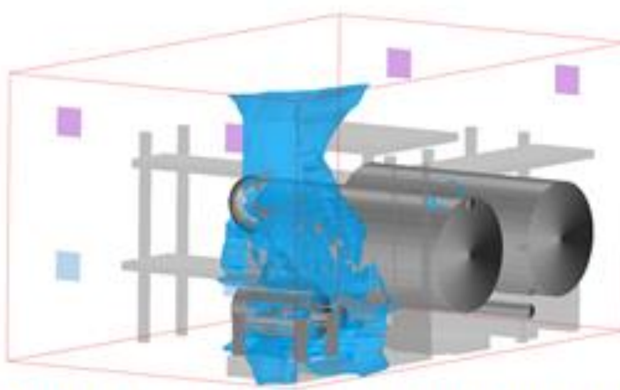


Figure 6e) HVAC(1), Gas concentration Iso-surface at 60%LEL (3D view)

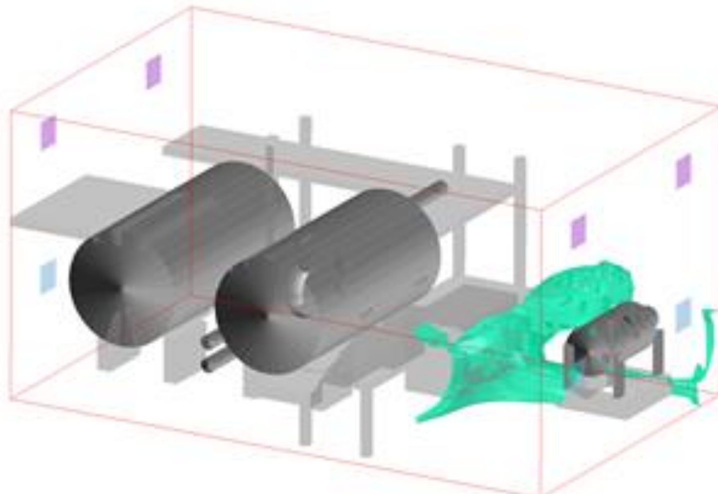


Figure 6f) HVAC(1), Gas concentration Iso-surface at 100%LEL (3D view)

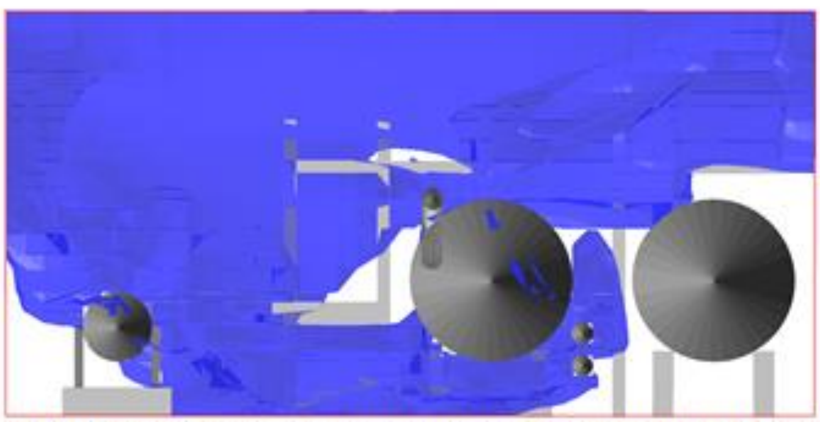


Figure 7a) HVAC(2), Gas concentration Iso-surface at 20%LEL (Section)

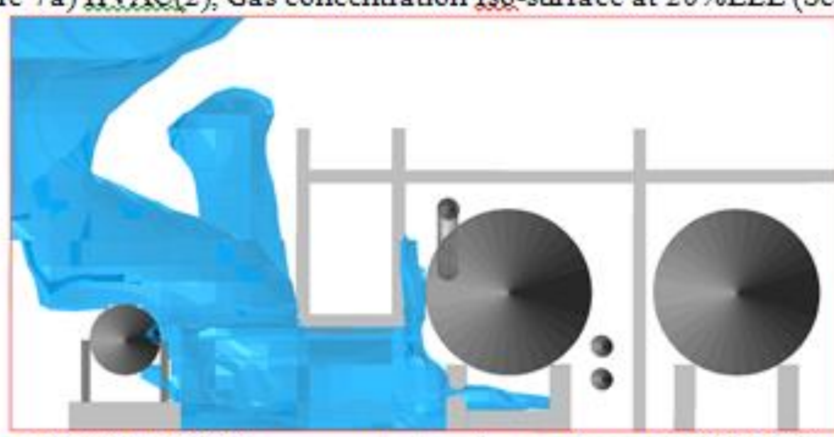


Figure 7b) HVAC(2), Gas concentration Iso-surface at 60%LEL (Section)

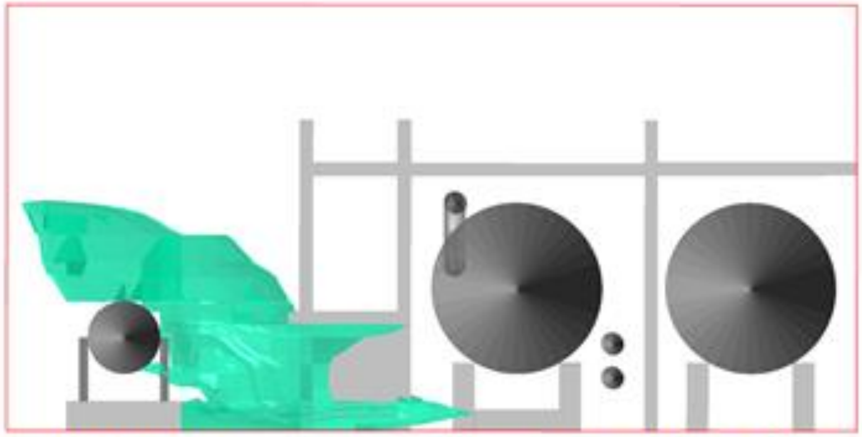


Figure 7c) HVAC(2), Gas concentration Iso-surface at 100%LEL (Section)

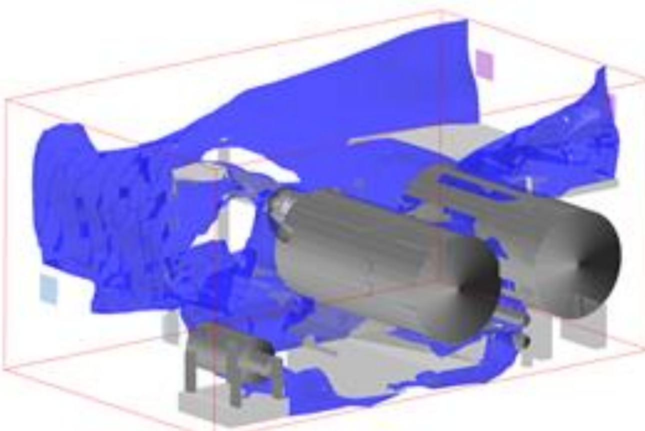


Figure 7d) HVAC(2), Gas concentration Iso-surface at 20%LEL (3D view)

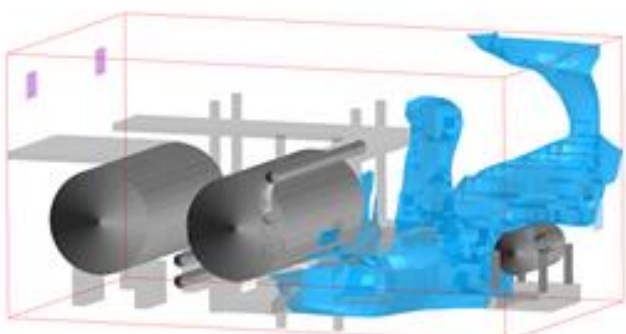


Figure 7e) HVAC(2), Gas concentration Iso-surface at 60%LEL (3D view)

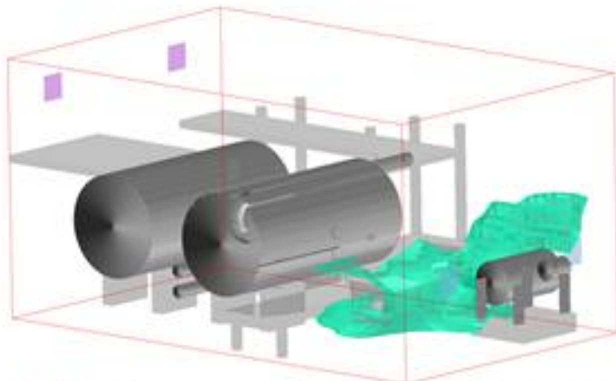


Figure 7f) HVAC(2), Gas concentration Iso-surface at 100%LEL (3D view)

Figure 3 highlights the general bulk-fluid flow pattern induced by each flow regime at a slice taken at the same depth as the gas leak jet (y=2.2m). In 3a the openings at x-min and x-max (LHS and RHS respectively) have no mechanical forced flow and bulk fluid will be forced out of the openings due to the slight pressure increase following the fluid injection (gas leak). This equates to an approximately equivalent mass flowrate of 0.107kgs-1 across each of the 4 openings. Figure 3a shows an elevated flow velocity along the upper half of the LHS wall due to the formation of a buoyant (gas) plume on this side of the compartment. Figure 3b shows HVAC design 1, where air is introduced via low level inlets and exhausted via high level outlets at both LHS and RHS walls. Fans are located in the outlets and the inlets are passive thus creating a negative pressure system. Figure 3b depicts the higher velocities generated as the inlets and outlets at both sides of the compartment. Finally 3c shows velocity vectors invoked by HVAC design 2 where air is drawn in at the low level inlet located on the LHS wall and is exhausted through the high level exhaust located on the RHS wall. Again this is a negative pressure system and an increase in velocity can be seen around each inlet/outlet. In both HVAC cases the fans are programed to exhaust a volumetric flow rate equal to an air change rate per hour of 50 for the compartment volume.

Figure 4a highlights that a buoyant plume is generally formed on the LHS of the compartment (due to the comparative density of the methane gas) after the jet impinges upon the equipment room wall and loses momentum. There is very slow air movement throughout the compartment, especially to the RHS. Figure 4c (HVAC 2) by contrast, shows general air flow from left to right across most elevations, particularly at low and high levels, again due to the HVAC design. Figure 4b shows in intermediate regime where-by bulk fluid on the LHS of the compartment generally moves toward the exhaust on the LHS wall, and bulk fluid on the RHS of the compartment generally moves toward the exhaust on the RHS of wall. Due to the nature of the leak scenario (direction and loss of momentum) much of the gas from the leak is generally entrained into the exhaust flow on the LHS.

Figure 5-7 give some context as to the dispersion and mixing levels of the gas in each ventilation regime. Figures 5a-f show iso-surfaces for 20%, 60% and 100% LEL gas concentration in the ambient case. It can be seen that a large volume of the compartment contains 60%+LEL. A layer of 20%LEL gas covers around two thirds of the floor surface and feeds the larger 60%LEL cloud above over time. 100%LEL gas can be seen in figure 5c to form a cloud around the initial jet area and is greater than 5m in diameter in different directions across multiple elevations.

Figures 6a-f shows the same gas concentration iso-surface outlines for HVAC design 1. As discussed previously the majority of the gas leak seems to be entrained into the exhaust flow generated between the LHS inlet/exhaust as the 60%LEL cloud has been significantly reduced in volume and area from the ambient case. Subsequently however, the concentration of the gas cloud in the centre of the compartment has now been reduced to around 20%LEL and the volume of 100%LEL cloud in the area surrounding the leak point has been reduced further.

Figures 7a-f shows the gas concentration iso-surface for HVAC design 2. What is immediately apparent is that even though a greater bulk fluid movement is achieved across the width of the compartment, because this leak occurred on the LHS (and lost momentum quickly) this left-to-right HVAC design appears to distribute the gas throughout the compartment to a greater extent than HVAC design 1. This is certainly true in the case of 20% and 60%LEL. The 20%LEL iso-surface in particular can be seen in figure 7d to become entrained into the flow field(s) as it is drawn across the compartment and exhausted through the RHS outlets.

This outcome is of course biased based on the details of this particular gas leak. A similar gas leak occurring on the RHS of the compartment would likely be exhausted much more effectively by HVAC design 2 than the current leak. Consider further however that if the leak did occur on the RHS of the compartment but was in the LHS direction, and did not impinge upon a solid surface the jet itself could distribute the gas across the compartment from right-to-left, and HVAC design 2 would redistribute that gas again from left-to-right in a similar fashion as demonstrated here, but with potentially less-desirable concentration distribution.

What we can conclude from a brief overview of these results is that the physical layout of the space, the attributes and location of the leak and of course, the design of the HVAC system (and whether it is operational or not) can each have a profound impact upon the evolution and consequence of gas cloud formation following a leak in a process area. We can understand how congestion might affect species migration and cloud formation and how air currents induced by HVAC systems can affect concentration distribution. We could further study the possibility of dilution-ventilation whereby the HVAC system is designed with gas cloud dilution in mind and we could gain insight into ‘dead-zones’ within the space where dilution of bulk fluid is not sufficiently achieved.

In practical terms, this model build and brief analysis took approximately 2 days. The modeller has a competent working knowledge of the CFD program. Having an understanding of the inherent limitations of the CFD model results (both inherent assumptions and user-input variability) as well as an intrinsic appreciation for the underpinning science behind gas detection methodology allows the user to interpret the results as an additional piece of information contributing to the best holistic detection arrangement.

What is not advisable, or arguably even practical from the point of view of a safety practitioner, is to use a percentage scoring system from a small number of leak scenarios as a risk-based justification for leaving large volumes of the compartment with no gas detection. The question of accounting for an almost infinite number of potential leak outcomes with a finite number of (inherently uncertain) models is an extremely difficult one to argue and to validate. To demonstrate that all credible leak scenarios have been accounted for with a limited number of CFD models would be difficult. One would have to categorise leak scenarios based on a range of attributes such as orifice size, pressure, direction, location, impinging upon congestion or unimpeded jet, atmospheric conditions and inventory details. Subsequently, an appropriate range of leak models which represent a sufficient cross-section of all credible leaks within each category must be analysed. Qualifying the definition of what constitutes a “sufficient cross-section” of potential credible cases is a daunting prospect alone and building, analysing (sensitivity analysis) and running the range of realistic models scenarios will in all likelihood be a very time-consuming endeavour. Consider further that for even a relatively small asset review there may be 20 areas like the one considered here and the costs and time requirement become disproportionately large for the expected yield or benefit of the study. This, however, should be a decision made by the specific operator, of the specific asset.

CFD certainly has its place in F&G detection system design and there is no doubt that it can offer great insight into fluid flow questions for particular offshore/onshore areas of concern but application as a one-stop-shop for removing detectors from a methodologically designed arrangement should not be seen as its primary purpose, and indeed, doing so is disingenuous to the importance of the educated and experienced fire/gas safety professional and exaggerates beyond realistic limits the role of the CFD operator with little or no background in fire/gas safety engineering and in particular offshore/onshore detection approach methodology.