Abstract

Multi-point ground flares (MPGF) are normally selected for processing large quantities of hydrocarbon gases generated in chemical processing and petrochemical refining units. These flares normally consist of hundreds of individual flare burners. These burners, oriented in a staggered configuration along feed lines, operate in a staged fashion. Staged operation allows safe and efficient processing of large flows of saturated and unsaturated hydrocarbon flare gases ranging from purge to full operating conditions. These flares include a wind fence designed to support efficient mixing of air with flare gas to reduce smoke formation and lower radiation levels associated with soot radiation. Safe operation is a significant concern due to expected radiation flux and associated high temperatures on surrounding equipment and in nearby work zones. Large MPGF’s must operate safely and cleanly during all ambient conditions including high winds. The potential for non-standard ignition of large quantities of hydrocarbon fuels is also a significant safety concern. Previous testing by Zeeco has quantified the expected thermal radiation levels from multiple flare burners for various hydrocarbon fuels (i.e., ethylene, propane, natural gas, etc.). This paper compares predicted and measured radiation levels from testing conducted at Zeeco’s flare test facility in Broken Arrow, Oklahoma. This paper presents results of a CFD analysis of a large MPGF to assess radiation flux and associated surface temperatures on the wind fence and surrounding locations. The impact of burner-burner spacing on cross lighting for wind blowing along and perpendicular to burner rows has been assessed. Results assess expected radiation flux and temperatures of nearby equipment and expected levels in designated work zones. Potential over-pressure conditions caused by partial ignition of a MPGF have been examined. This work helps establish guidelines for safe and efficient operation of large MPGF under various wind scenarios. Special attention was paid to safety issues associated with radiation flux and hot gas exposure of nearby operating process equipment and associated work locations.
Introduction

Low profile Multi-Point Ground flares (MPGF) represent a special class of flares capable of safely processing significant quantities of flare gas in an environmentally responsible fashion. A detailed computational fluid dynamics (CFD) model of three low-profile multi-point ground flares located in close proximity (see Figure 1) has been developed using the flare modeling tool called C3d. This CFD tool has also been used to simulate many other flare systems including enclosed flares, elevated steam and air-assisted flares, pressure-assisted flares, and other MPGFs. The present MPGF systems is part of a chemical production plant located on the gulf coast region of the United States. The present flare system includes three MPGFs which can all operate individually and under certain conditions all may be operated simultaneously. The present study analyzed each MPGF individually under various wind conditions. Additional analyses were conducted with all three MPGFs operating simultaneously under various wind conditions. This paper focuses on safe operation of MPGF’s using results from the largest of the three MPGFs, the Ethylene flare, as well as the full 3-flare system. The Ethylene MPGF is approximately 430’ long by 280’ wide, has 19 stages (756 tips) and processes approximately 4.1 million pounds per hour (lb/hr) of flare gas composed of light hydrocarbons, hydrogen and inerts (average flare gas MW=22-25). Two other MPGF’s were also included in the overall system; the LDPE flare (115 tips; ~500,000 lb/hr; average flare gas MW = 28) and the LLDPE flare (80 tips; ~280,000 lb/hr; average flare gas MW =32). An important safety issue for MPGF’s is their impact on surrounding structures and personnel. Computational Fluid Dynamics simulations were used to analyze various safety issues including radiation levels, potential over-pressure caused by ignition delay and plume dispersion around the flare considering several wind directions/speeds and relief rates.

Previous work reported earlier has been done to validate the flare CFD model in general [1] and the combustion model used in this analysis [2]. The main focus of this paper is the use of CFD analysis to assess the impact of the flare and flare plume on surrounding structures. The information presented below describes this work and presents results and conclusions for large ground flare operation.

The Flare Model

The CFD tool used in this work simulates turbulent reaction chemistry coupled with radiative transport between buoyancy driven fires (i.e., pool fires, gas flares, etc.) and surrounding objects (i.e., wind fence, process equipment, etc.). The code provides “reasonably” accurate estimates of various risk scenarios including wind, % flame coverage, and thermal fatigue for a given geometry. Typical simulations generally require CPU times on the order of hours to a few days on a “standard” windows (or LINUX) desktop workstation. Large Eddy Simulation (LES) is used to approximate turbulent mixing. The code used in this work is based on an earlier CFD tool called ISIS-3D [3-5]. ISIS-3D was previously validated for simulating pool fires to predict the thermal performance of nuclear transport packages [6-9]. ISIS-3D, originally developed at Sandia

Page 2 of 17
National Laboratory, has been commercialized into a new CFD tool called \textit{C3d} which is specifically tailored to analyze large gas flare performance. \textit{C3d} has previously been applied to large multipoint ground flares, air-assisted flares, and utility flares [1, 2] with new combustion models developed, implemented, and tested for various flare gas compositions including methane, ethane, ethylene, propane, propylene and xylene. \textit{C3d} has been used to predict flare flame size and shape, estimate the smoking potential for a given flare design firing typical flare gas, and to estimate the radiation flux from the flare flame to surrounding objects. \textit{C3d} simulations of flame height and flame-to-ground radiation have been validated by direct comparison to measured flame size, shape, and radiation measurements taken during single-burner and multi-burner tests conducted under no-wind and low-wind ambient conditions [10].

![Figure 1 - Plan view of MPGF and structures included in the CFD model](image)

For the flares shown above, \textit{C3d} predictions were made for maximum flow at a variety of wind directions (five or more wind directions for each flare). The potential for over-pressure due to delayed ignition of some stages was also analyzed. Wind fences are designed to minimize wind effects on the flare flames but also create unexpected flow profiles inside the flare field which can affect combustion performance of the MPGF and dispersion of the resulting plumes, which in turn
can dramatically affect radiation levels to the wind fence and surrounding structures outside the fence. These CFD analyses were done to investigate these safety concerns and identify potential problem areas.

**Technical Approach**

Due to the size of the flare field, modeling the exact fence geometry for the full simulation was not practical since it would require excessive computational cells and the associated CPU time to perform the analysis. The same approach of using porous plates to simulate the actual fence geometry described previously [11] was also used in this study.

During these analyses, several structured grids were developed for both the overall multi-MPGF flare system (see Figure 2 and Figure 3) and each individual MPGF flare (see Figure 4 for the Ethylene MPGF). Grids were refined several times to improve calculation robustness and speed and to assure grid independent results for each simulation. The final grid for the overall multi-MPGF system consisted of 11.5 million cells which included refinement near the burners in each MPGF. The final grid for the ethylene MPGF used was a structured grid with of 4.5 million cells. As shown in Figure 4 the cells were clustered around areas of high flow gradients which existed near the flare tips.

![Figure 2 - Perspective view of three flare geometry overlaid by mesh in z-direction](image-url)
Figure 3 - Plan view of mesh for three flare simulation

Figure 4 – Mesh for Ethylene flare (note fine grid in near burner region to resolve ignition/combustion phenomena)
Combustion Model

The combustion model described initially by Said et.al. [13] and used previously by Smith et.al. [1,2] considers Fuel (H2, C2H4) from the flare tip, oxygen (O2) from surrounding air, products of combustion (PC) produced in complete combustion (includes H2O and CO2), radiating carbon soot (Soot), and other non-radiating intermediate species (H2, CO, C2H2). The specific reactions considered for these analyses included:

Combustion Reaction 1:
\[ \text{H}_2 + 8\text{O}_2 \rightarrow 9\text{H}_2\text{O} + 141 \text{ MJ/kg} \quad (A=10^{15}, T_A=10,500\text{K}) \quad \text{Eq. 1} \]

Combustion Reaction 2:
\[ \text{CO} + 0.57[\text{O}_2]^{1/2} + 0.64[\text{H}_2\text{O}]^{1/2} \rightarrow 1.57\text{CO}_2 + 0.64\text{H}_2\text{O} + 10.1 \text{ MJ/kg} \quad (A=10^{13}, T_A=15,151\text{K}) \quad \text{Eq. 2} \]

Combustion Reaction 3:
\[ [\text{C}_2\text{H}_4]^{1/2} + 0.769\text{O}_2 \rightarrow 0.769\text{H}_2\text{O} + 0.801\text{C}_2\text{H}_2 + 11.5 \text{ MJ/kg} \quad (A=10^{15}, T_A=10,500\text{K}) \quad \text{Eq. 3} \]

Combustion Reaction 4:
\[ [\text{C}_2\text{H}_2]^{1/2} + 2.46\text{O}_2 \rightarrow + 2.62\text{CO}_2 + 0.588\text{H}_2\text{O} + 0.3 \text{ Soot} + 29.2 \text{ MJ/kg} \quad (A=10^{15}, T_A=15,500\text{K}) \quad \text{Eq. 4} \]

Combustion Reaction 5:
\[ \text{Soot} + 1.33\text{O}_2 \rightarrow 2.33\text{CO} + 13.6 \text{ MJ/kg} \quad (A=10^{15}, T_A=13,590\text{K}) \quad \text{Eq. 5} \]

where the coefficients are selected so that complete combustion of soot and the intermediate species produce the same species and thermal energy as direct combustion of the fuel. The coefficients in this reaction mechanism are mass based (kilograms of reactant) instead of moles.

The advantage of using this approach is that the initial reaction for burning the fuel has a low activation energy, which allows partial burning and heat release of the flare gas. This maintains combustion since the partial heat released allows subsequent reactions, that produce most of the heat and all of the soot, to occur. As in the previous combustion models developed for MPGF analyses [1, 2], the flare gas Arrhenius combustion time scale is combined with the turbulence eddy breakup time scale to yield an overall time scale for the reaction rate. The characteristic time from the kinetics equation was combined with the characteristic turbulence time scale as:

\[ t_{\text{total}} = t_{\text{arrhenius}} + t_{\text{turb}} = \frac{1}{C_i} = \frac{1}{A_kT_b \exp \left( \frac{-T_{A}}{T_b} \right)} + \frac{C_{eb}\Delta x^2}{\varepsilon_{\text{diff}}} \quad \text{Eq. 6} \]

where \( A_k \) is the pre exponential coefficient, \( T_A \) is an activation temperature, \( T \) is the local gas temperature, and \( b \) is a global exponent, \( \Delta x \) is the characteristic cell size, \( C_{eb} \) is a user input constant (~0.2E-04) that is cell size dependent, \( \varepsilon_{\text{diff}} \) is the eddy diffusivity from the turbulence model, and \( t_{\text{turb}} \) is the turbulence time scale, i.e. characteristic time required to mix the contents of a computational cell. The reaction rates are combined by simple addition of the time scales. Note that depending upon the scale of Arrhenius vs. eddy breakup time scales the characteristic time for
each individual reaction can be different. Thus, the simplified combustion model approximates
turbulent combustion using the eddy dissipation concept and local equivalence ratio effects. The
Arrhenius kinetics and turbulent mixing approach are similar to the commonly used Eddy-Breakup
(EBU) type combustion model.

In addition to these dynamic models, sequences of irreversible chemical reactions that describe
the combustion chemistry are required. To minimize computational load, a minimum number of
chemical reactions are used that fulfill the requirements of total energy yield and species
consumption and production. From the basis of heat transfer, flame size, and air demand the details
of the chemical reactions are not critical as long as the oxygen consumption is correctly balanced
for a given fuel type and the amount of soot produced is calibrated to match experimental data. To
this end, a multi-step chemical reaction model for ethylene gas was used to approximate the global
reaction mechanism as shown in the equation below:

\[
\frac{dX_i}{dt} = C_i X_1^c X_2^d
\]

where \( X_i \) is the mole fraction of the rate equation for the \( i^{th} \) reaction, \( C_i \) is the global reaction rate
(Eqn. 6), and \( c \) and \( d \) are global exponents. All of the rate equations are solved simultaneously for
each reaction and the stoichiometric coefficients (Eqns. 1-5) are used as constraints that couple the
equations and insure conservation of energy and chemical species.

Global reaction kinetics are often used to model combustion as a single step in CFD combustion
simulations. The coefficients and powers are fit to existing experimental data. Although it is
possible to use a global reaction mechanism with the same coefficients as those which have been
published elsewhere, this could be misleading because the coefficients were originally fit to
experimental data chosen by other authors for a specific combustion experiment being modeled
and it is well known that simulation results are very sensitive to both the computational grid (cell
size, aspect ratio, and density) and the experimental data chosen by the original authors. A different
computational grid or experiment would likely require a different set of reaction coefficients.

In the present work, the global reaction mechanism described by Smith, et. al., 2010 [2] was
used. This work relied on work by Duterque et. al. [15] and Kim [16] as starting points. However,
since these authors adjusted their global reaction coefficients to match “laminar” flame speed data
and since the combustion occurring in gas flares is governed by turbulent mixing, the original
coefficients had limited applicability. The coefficients associated with the activation temperature
and the exponents for mole fractions were based on the physics of the reaction mechanism thus
were not expected to be affected by local grid structure. However, this is not the case for the pre-
exponential coefficient. To match reaction rates to measured combustion rates, the pre-exponential
coefficients for all of the reactions was varied to develop the validated combustion model. Also,
since the combustion model depended upon turbulent mixing of flare gas, the combustion was also
governed by turbulent mixing with air. The C3d code uses an LES formulation to approximate
turbulent mixing, which depends upon two additional factors, a proportionality coefficient and cell size. The recommended LES proportionality coefficient of 0.15 was used. To capture the cell size dependency correctly, the same computational mesh that was used in the full flare field simulations was calibrated to triple ethylene flare experimental radiation and flame size measurements (see Figure 5. Using this approach, the required parameters shown in Eq. 6 and 7 were determined to establish the validated combustion model for the present work.

Figure 5 – Comparison of Predicted and Measured Flame Shape for the 3-Flare Test (sequential predicted images overlaid to test flame)

The combustion model described by Eqns. 1-7 was used when calculating flame size, shape, and radiation effects. However the model is not suitable for ignition/overpressure effects which can occur when a group of flares remain unignited and the vapor cloud builds up to where it is ignited remotely by another ignited row or pilot. A large vapor cloud has the potential to create sizable overpressures which can injure personnel and damage equipment. In order to calculate the potential overpressures a new ignition model was added to C3d that simulates the propagation of a combustion/deflagration wave. The new ignition model is based upon a user input propagation speed which is the minimum velocity that a deflagration wave would have. C3d will propagate the deflagration wave through the vapor cloud at this speed or greater. After a computational cell is burning (due to passage of deflagration wave) the code resorts to the kinetics model described earlier in Eqns. 1-7. The speed of the deflagration wave at a minimum is the user input speed. However when the vapor cloud is large enough the hot gas expansion can cause adjacent computational cells to ignite at a time scale shorter than the propagation velocity. In this way flame acceleration occurs, and was found to occur in a number of test cases where the input propagation velocity was large ~100m/s. The deflagration velocity was set by values found in the literature for ethylene (20 - 40m/s) for unobstructed large clouds [17]. This velocity was observed for large lenticular shaped balloons (~10m length) of ethylene gas. Higher propagation velocities such as
those for hydrogen (100m/s) would often result in large overpressures (a fraction of an atmosphere) at the MPFG wind fence boundary due to flame acceleration.

### Radiation Validation

An new set of radiation validation simulations have been performed with propylene fuel using the nozzle tips associated with this project. This radiation test utilized a Zeeco 2-in² area tip designed for the Dry Ethylene flare field. In the test a single tip was used, with 5465 kg/sec propylene injected at 22.5 PSIG. There was a crosswind with a steady component of 3-7 mph gusting to 9-13 mph. Radiation was measured at 3 distances (75 ft, 100 ft, and 150 ft) and two elevations (5 ft and 20 ft). The radiometers were placed due east of the flare and the wind came from the SSE 169 degrees from true north.

Two simulations were performed each with a different wind velocity. As noted by Smith [1], flare radiation is very sensitivity to wind speed. Higher radiation fluxes are measured under no (low) wind since the flare flame is much taller compared to the flame with high winds where the flame bends over and becomes smaller and bushier. The wind speeds chosen for these simulation were 5 and 10 mph which covers the steady and gusting components of the wind encountered in the flare test. The simulations included atmospheric absorption due to CO₂ and H₂O utilizing the Hamins and Fuss correlation [9].

The numerical simulation utilized a computational domain of 12m square and 20m high (40ft x 40ft x 65ft) with a variable grid composed of 328,000 cells. This simulation required approximately 4 CPU hours from ignition to steady state operation on a standard desktop computer.

<table>
<thead>
<tr>
<th>Elevation (wind speed)</th>
<th>Radiometer distance from flare</th>
<th>Measured Flux (BTU/hr-ft²)</th>
<th>Predicted Flux (BTU/hr-ft²)</th>
<th>Predicted Flux (BTU/hr-ft²)</th>
<th>Predicted Flux (BTU/hr-ft²)</th>
<th>Predicted Flux (BTU/hr-ft²)</th>
<th>Predicted Flux (BTU/hr-ft²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>75 feet</td>
<td>5 ft high (3-7mph measured wind)</td>
<td>171</td>
<td>190</td>
<td>168</td>
<td>205</td>
<td>221</td>
<td>183</td>
</tr>
<tr>
<td>100 feet</td>
<td>5 ft high (5mph predicted wind)</td>
<td>102</td>
<td>117</td>
<td>95</td>
<td>102</td>
<td>120</td>
<td>104</td>
</tr>
<tr>
<td>150 feet</td>
<td>5 ft high (10mph predicted wind)</td>
<td>34</td>
<td>53</td>
<td>38</td>
<td>34</td>
<td>53</td>
<td>38</td>
</tr>
</tbody>
</table>

As shown in the table all of the measured results fall within the predicted band limited by the two wind speeds. The table also confirms how sensitive flare radiation is to wind speed.

### Modeling Assumptions

The following assumptions were utilized in modeling the MPGF flare:
1. Combustion of the flare gas was approximated by the appropriate irreversible chemical reaction mechanism with specified kinetics (see above).
2. Thermal radiation was calculated using standard radiation models.
3. Ambient wind condition, flare gas inlet temperature and pressure, were set to match as closely as possible the conditions provided.

**Boundary Conditions**

The Boundary conditions used were a combination of imposed wind and hydrostatic pressure on all boundaries except the ground boundary (z-axis minimum) which was set as a zero mass-flux wall. A 20 mph (8.94 m/s) wind was set with appropriate velocity vectors imposed upon all flux boundaries. The thermal and species boundary conditions were set for each case with typical air composition and ambient temperature set at 73°F (23°C).

**Physical and Numerical Sub-Model Selection**

To simulate fluid flow, the momentum solver used in *C3d* was the LES turbulence model described earlier. The energy equation was utilized to capture the temperature changes due to combustion and mixing. The energy equation also included radiation effects.

The species equations were solved to keep track of the distribution and concentration of fuel, oxygen, intermediate species, soot, and products of combustion (CO₂ and H₂O). The combustion model was used to provide the species equations source and sink terms as a function of species concentrations, local gas temperature, and turbulent diffusivity.

*C3d* includes a series of sub-models that predict flame emissivity as a function of molecular gas composition, soot volume fraction, flame size, shape and temperature distribution which in turn depend on solutions to the mass, momentum, energy and species transport equations. The radiation transport model is used not only to predict radiation flux on external (and internal) surfaces, but it also provides source and sink terms to the energy transport equation so that the flame temperature distribution can be accurately predicted.

Thermal radiation effects in *C3d* are calculated in two ways. Within the flame zone, radiation is assumed to be diffusive and outside the flame zone radiative transport is calculated using view-factor methods. The flame surface used in the view-factor calculation is set by finding the dynamic surface wherein a product of combustion, typically water vapor, has a mass fraction above and below a user specified value, typically 0.04. This dynamic surface, its temperature, and a correction factor (dependent upon flame optical thickness) are all used to calculate view-factor radiation from all of the flare surfaces to surrounding objects including nearby process instruments and equipment and is used to establish safe work zones. The view-factor radiation calculation includes shadowing due to intervening objects and radiation absorption due to participating media including water vapor, carbon dioxide, and soot along the ray path.
The view-factor radiation has also been extended to the C3d Multi-Block formulation. The Multi-Block formulation allows the user to split a large flare problem into separate zones which are coupled together by boundary conditions. Each separate zone is solved on a different CPU and are time synchronized so that large problems requiring millions of cells can be solved on several CPU’s simultaneously to reduce the required calculation time. The view-factor thermal radiation from one block (zone) can be calculated to any geometric position either within or outside that zone. This allows the user to add radiation contributions from adjacent zones to get the total incident radiation value for the entire problem. The only restriction upon the zone-to-zone radiation transport is that shadowing or media absorption in adjacent zones is not taken into account because any zone only knows about the geometric and compositional details within its own boundaries.

**Transient Calculation and Post-Processing Results**

To set up the steady wind profile, the transient simulation was run for approximately 10 to 15 seconds before turning on the flare gas flow. Once the wind profile was established and the flare gas was turned on and ignited, it was allowed to burn for about 17 seconds to capture the fluctuations caused by interactions with the wind. The 17 second burn time provided essentially a “steady-state” burning condition of about 10 to 12 seconds.

Since a transient solver was used, all field variables fluctuate in time due to turbulence and the other non-linearity’s in the equation system. However, when examining any field variable, no gradual slope was observed - just short term fluctuations as expected in turbulent flows.

The convergence criteria chosen for the simulations were that the equation of state was always satisfied to within 0.01% or less at any location in the computational domain. Typically, the convergence criterion was better than the maximum allowable since the time step constraint was limited by Courant conditions, which allows the flow field to be solved to a higher degree of accuracy.

**Results**

All three flares shown in Figure 1 were analyzed individually for multiple wind conditions. Analyses were also performed with all three flares operating at the same time, again under multiple wind conditions. This paper focuses on safe operation of MPGF’s using results from the largest of the three MPGFs, the Ethylene flare, operating by itself, with a 20 MPH (8.9 m/s) cross wind as shown in Figure 6. The flare gas flowrate was approximately 4,100,000 lb/hr with the composition being mostly ethylene and 11-20% hydrogen, depending on the specific stage. This analyses considered a total of 30 receptor located around the flare in areas where staff might be required to work or where key plant equipment will be located. Each of these receptors monitors local radiation levels. Of these, five receptors (# 21, 22, 25, 26, 27) shown in Figure 6 were considered key monitoring locations due to the wind pushing the plume in their direction.
The inner fence surface temperatures, for this wind condition, were highest near the downwind corner as shown in Figure 7, and the maximum fence temperatures were near the bottom of the fence. The CFD model included conduction of heat through the layers of fence slats and re-radiation from the fence to surroundings. As shown in Figure 8, the outer fence temperatures in these same locations was considerably lower than the inner fence surface. Associated radiation levels at each of the five critical receptors from these analyses are summarized in Table 2.

The results shown below consider radiation flux from the flame and hot plume and include convective heat gain/loss from the receptors. As shown, the total radiation flux to each of these receptors is less than the API recommended level for safe operation of 500 Btu/hr/ft². The respective temperatures shown are also within safe operating levels though they may be uncomfortable for prolonged working conditions.

![Figure 6 – Example of key receptors (shown as red dots with numbers identifying each receptor) for the Ethylene flare field with wind direction indicated](image)

Table 2 - Receptor radiation levels and metal temperatures

<table>
<thead>
<tr>
<th>Receptor No.</th>
<th>Receptor Elevation</th>
<th>Metal Type</th>
<th>Total Radiation (BTU/hr/ft²)</th>
<th>Receptor Metal Surface Temperature (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>45°-6”</td>
<td>CS</td>
<td>4</td>
<td>32</td>
</tr>
<tr>
<td>22</td>
<td>69’</td>
<td>CS</td>
<td>4</td>
<td>32</td>
</tr>
<tr>
<td>25</td>
<td>63°-2”</td>
<td>CS</td>
<td>88</td>
<td>35</td>
</tr>
<tr>
<td>26</td>
<td>54°-3”</td>
<td>SS</td>
<td>69</td>
<td>42</td>
</tr>
<tr>
<td>27</td>
<td>80°-3”</td>
<td>SS</td>
<td>105</td>
<td>48</td>
</tr>
</tbody>
</table>
The results shown below consider radiation flux from the flame and hot plume and include convective heat gain/loss from the receptors. As shown, the total radiation flux to each of these receptors is less than the API recommended level for safe operation of 500 Btu/hr/ft². The respective temperatures shown are also within safe operating levels though they may be uncomfortable for prolonged working conditions.
As noted above, the potential of having an over-pressure wave being generated by delayed ignition of a MPGF is a significant safety concern that was also investigated. Potential scenarios provided by the flare manufacturer were considered to have a very low probability of occurring but were considered as a way to reduce the insurance risk profile associated with this flare system. Previous work showed that very short ignition delays (<150 ms) at maximum flare gas flow rate created significant over pressure conditions (>7 psig) with the associated potential to damage nearby equipment and structures as well as plant personnel working in the vicinity. Of the several scenarios’ considered, one is presented here to illustrate the ability of CFD to investigate this risk. The scenario presented below considered Stages 1-4 firing (i.e., burning) but due to an inoperable pilot, Stage 5 was venting flare gas to the atmosphere but was not ignited thus the plume of unignited flare gas dispersed subject to various wind conditions (i.e., wind blowing parallel to the burner row and wind blowing perpendicular to the burner row). The flare gas dispersion (i.e. conditions just prior to ignition of Stage 5 flammable plume) is shown in Figure 9. The burning stages (1-4) are shown by red/yellow colored iso-surfaces with a transparent plume shown above the flames indicating the combustion by-products plume coming from the flames. The grey iso-surfaces adjacent to the burning stages represent unignited ethylene (2.6 mol%). The simulation was run under these conditions to predict how large the unignited flammable plume would get and how long it would take before ignition occurred. After ignition of the flammable plume, the predicted pressure was analyzed to evaluate the potential hazard.

Figure 9 – Flammable plume dispersion and ignition with subsequent over-pressure analysis (selected scenario included no Stage 5 pilot, wind blowing perpendicular to ignited stage upwind of unignited stage)

1 Using CFD to examine potential risk as a way to reduce the insurance risk profile was recommended at the 2015 AFRC meeting held in Salt Lake City which resulted in the formation of the API Academic Liaison Sub-committee.
This scenario, coupled with the indicated cross-wind (wind blowing perpendicular to stages) resulted in a peak over-pressure of about 1400 Pa which was the highest predicted over-pressure conditions for all over-pressure analyses considered (8 total analyses with various operating scenarios performed). The pressures at various locations on the fence are shown in Figure 10.

![Gas Pressure Graph](Image)

- Ignition time after start of venting = 0.6 seconds
- Max sustained pressure at fence = 1400 Pa (0.2 PSI)

**Figure 10 – Over-Pressures predictions at various locations as a function of time on the wind fence for the worst-case scenario (perpendicular wind blowing upstream ignited stage to unignited stage).**

**Conclusions and Recommendations**

The LES code, *C3d*, was used to assess safety issues related to the performance of multiple large Multi-Point Ground Flares (MPGF) located in close proximity to each other and the radiation fluxes (and temperatures) at several nearby receptors as well as the resulting plumes from each flare at maximum-flow conditions with a wind speed of 20 mph (8.9 m/s) coming from multiple directions. Results from the largest of the three MPGFs at one wind condition were shown to illustrate how CFD can be used to address safety concerns related to fence temperatures, radiation levels at key equipment locations, plume dispersion and potential over-pressure levels related to ignition delay of MPGFs for selected scenarios. Simulations predicted the average temperature of the inner fence surface was on average nearly 275°C (maximum of 400°C). The MPGF fence was shown to shield nearby structures from radiation, reducing heat flux levels by up to a factor of 3.
or more. Several wind conditions were examined with simulation results illustrating how the wind forces the plume toward surrounding structures increasing the heat flux level as expected – but levels on key structures remained below 110 BTU/hr-ft². Of the unignited stage over-pressure scenarios analyzed, the highest predicted over-pressure was approximately 1400 Pa (0.2 psig). Results from this MPGF CFD study were used to evaluate the final MPGF system design and help answer key safety concerns which may be used to reduce the insurance risk profile thus impacting the financial performance of the new flare system.

Based on this work, it is recommended that CFD be used to investigate safety issues surrounding MPGF design and operation. Flare vendors such as Zeeco have routinely applied CFD to investigate potential operating scenarios for their design before the flares are built. It is also recommended that companies who own and operate MPGF should conduct similar analyses of their systems considering the worse-case scenarios to investigate potential safety hazards as a way to reduce their insurance risk portfolio which may reduce insurance costs to the operators.

References


