Prediction of Plume Formation and Dispersion from Gas Flares

Joseph D. Smith, Ph.D. Ahti Suo-Antilla, Ph.D., and Robert Jackson
Systems Analyses & Solutions, Inc. Tulsa, OK
Scot K. Smith, P.E.
Zeeco, Inc., Broken Arrow, OK

ABSTRACT
Results from three different applications are presented to illustrate how plumes from various types of flares have been analyzed with different levels of detail. The first application considers an enclosed flare plume. Results from a thermodynamic estimate of combustion temperature and species are used to calculate plume characteristics using buoyancy flux and Gaussian dispersion. The second application considers an air assisted flare and employees a steady state CFD analysis to investigate the impact flare tip geometry and a cross wind has on plume formation and dispersion. The third application considers a single tip ground flare and uses a transient LES based CFD analysis investigate the impact that a flare plume has on nearby equipment. Results from these applications provide examples of how flare plumes may be analyzed and the various types of information available from each approach.

INTRODUCTION AND BACKGROUND
Plume formation from large ground flares, elevated flares, off shore flares, and other types of flares is a subject of interest due to potential safety hazards for workers and potential damage to surrounding buildings and equipment. Also, air pollutants are dispersed into the atmosphere downwind of flare stacks from the flare plume. Understanding how flare plumes form and are impacted by cross winds, how these plumes affect personnel and equipment, and how they disperse air pollutants are the main topics of this paper. An introduction to plume formation and how plumes are analyzed will first be discussed. Next, several examples of various flares and the associated plume analysis will be discussed. Lastly, a summary of plume analysis with conclusions on the appropriate analysis techniques will be given.

Plume Formation and Dispersion Modeling
Plumes from an elevated flare forms at the stack exit while plumes from a multi-tip ground flare form as plumes from individual burner tips merge into a large composite plume (Figure 1). Although plumes can be considered as starting as a point source subject to atmospheric conditions (i.e., stability class, wind conditions), detailed flare plume analysis must also consider the overall flare geometry. Questions of plume shape, rise height, and downwind concentration profiles must be answered. In general, taller flare stacks disperse pollutants better than shorter ones (i.e., ground flares verses elevated flares) because the resulting plume must disperse further before it reaches ground level as it spreads and disperses.

For an elevated flare, flare gases mix with surrounding air and combusted above the flare tip. The plume evolves from the visible flame and travels downwind of the flare stack. Due to the temperature difference between the flare plume and the surrounding air, buoyance causes the plume to rise and expand as it mixes with cooler surrounding air. As the plume rises and expands, pollutants (and heat) disperse into the surrounding atmosphere so pollutants have further to travel to reach ground level. The final “effective plume height ($H_p$)” is the sum of the physical stack height ($h_s$), the flame height ($h_f$) and the average plume rise ($\Delta h$) as shown in Figure 1. Two factors mainly control plume rise: momentum and buoyancy. Plume momentum is determined by flare gas volumetric flow rate through the flare tip geometry. Plume buoyancy is determined by flare gas composition and resulting combustion products and effluent gas temperature.
Plume momentum carries the flare effluent downwind from the stack to a point where atmospheric conditions control its trajectory. Once emitted, the initial stack exit velocity is quickly reduced by ambient air entrainment and wind effects which cause the plume to bend over as it acquires horizontal momentum. Since wind speed usually increases with distance above ground level, as plume elevation increases its horizontal momentum increases and it bends over until it appears level with the ground. Depending on ambient conditions, the point where the plume becomes horizontal may be a considerable distance from its origin.

Buoyance mainly depends on the temperature difference between combustion products temperature ($T_g$) and the surrounding ambient air temperature ($T_a$). In an unstable atmosphere, the plume buoyance decreases as it rises while in neutral conditions the plume buoyancy remains constant with increasing elevation. Wind also reduces plume buoyancy by increasing the entrainment rate of cooler surrounding air into the plume. The greater the wind speed the faster the plume mixes with the atmosphere and cools thus losing its buoyancy. Thus, in “windy” regions a flare plume will not rise as high as in regions with little wind.

**FLARE PLUME DISPERSION ANALYSIS**

Flare plume dispersion models range in complexity from standard Gaussian dispersion models to transient reacting flow multi-physics CFD models. Each approach provides different amounts of dispersion information but also requires more input data and more effort to complete. The following illustrative examples demonstrate various levels of flare dispersion analysis for different types of gas flares.

**1-D Plume Dispersion Analysis**

Flare plume dispersion may be analyzed using standard formulas for plume rise and distribution. The first example illustrates application of this approach. To set the stage for this analysis, the model is briefly discussed with application illustrated with an analysis of plume dispersion from an enclosed flare.

The simplest form of plume dispersion model consists of a set of mathematical equations that predict pollutant concentrations downwind of the plume origin. These models use empirical equations to determine effective plume height and shape and estimate pollutant dispersion based on various meteorological conditions including temperatures, wind speeds, stabilities, and topography. A common model based on the
Gaussian dispersion equation estimates dispersion of nonreactive pollutants as a function of complexity of source and topography surrounding the flare.

**Plume Rise Model**

To calculate plume rise to estimate effective plume height, Brigg’s standard plume rise formulas [1, 2, 3] relating “buoyancy flux” and wind speed to plume rise can be used:

\[
\Delta h = \frac{1.6F_b^{1/3}(x)^{2/3}}{u} \tag{1}
\]

Here, \( \overline{u} \) is the average wind speed, \( x \) is the downwind distance from the plume origin (i.e., flare) and \( F_b \) is the “buoyancy flux” factor defined as:

\[
F_b = g v_g \frac{d_s^2}{4} \left( \frac{T_g - T_a}{T_a} \right) \tag{2}
\]

where \( g \) is the acceleration due to gravity, \( v_g \) is the gas velocity at the flare exit, \( T_g \) is the plume gas temperature and \( T_a \) is the ambient air temperature. Eq. 1 estimates plume rise for buoyancy dominated plumes. Some plumes are momentum dominated (i.e., high tip velocity from elevated flares). In this case, plume rise is more appropriately calculated from:

\[
\Delta h = \left( \frac{3F_m x}{\beta_j^2 \overline{u}^2} \right)^{1/2} \tag{3}
\]

where \( \beta_j \) is the jet entrainment coefficient defined as:

\[
\beta_j = \frac{1}{3} + \frac{\overline{u}}{v_g} \tag{4}
\]

For Eq. 4, the “momentum flux” parameter is defined as:

\[
F_m = \frac{v_g^2 d_s^2}{4} \left( \frac{T_a}{T_g} \right) \tag{5}
\]

with \( v_g \) as the gas velocity at the flare exit and \( d_s \) as the flare diameter (\( T_a \) and \( T_g \) defined as before).

To determine whether a plume is buoyancy or momentum dominated, the critical temperature difference (\( \Delta T_c \)) must be determined. For unstable or neutral atmospheric conditions, the following equations are recommended:

\[
(\Delta T)_c = \begin{cases} 
0.0297T_g^{3/5} & \text{for } F_h < 55 \\
0.00575T_g^{3/5} & \text{for } F_h \geq 55
\end{cases} \tag{6a}
\]

When \( \Delta T \geq \Delta T_c \), the plume is buoyancy dominated and when \( \Delta T < \Delta T_c \) the plume is momentum dominated.
Plume Dispersion Model

Source complexity (i.e., flare plume formed from a wake stabilized flame) is an important issue in determining the plume shape so the 1-D model must account for this issue. In addition, local topography and buildings play a key role in plume dispersion. Elevated plumes may impact airplane flight paths\(^1\) while elevated terrains (buildings) may affect plume formation and lead to higher surface concentrations since the predicted plume centerline is closer to the surface.

The general Gaussian distribution equation is:

\[
\chi = \frac{Q}{2\pi\sigma_y\sigma_z\mu} \left\{ e^{-\frac{1}{2}\left(\frac{y^2}{\sigma_y^2} + \frac{z^2}{\sigma_z^2}\right)} + e^{-\frac{1}{2}\left(\frac{z+H}{\sigma_z}\right)^2} \right\}
\]

(7)

where:

- \(\chi\) = ground level pollutant concentration (g/m\(^3\))
- \(Q\) = mass emitted per unit time
- \(\sigma_y\) = standard deviation of pollutant concentration in y (horizontal) direction
- \(\sigma_z\) = standard deviation of pollutant concentration in z (vertical) direction
- \(u\) = wind speed
- \(y\) = distance in horizontal direction
- \(z\) = distance in vertical direction
- \(H\) = effective stack height

As discussed elsewhere [4], the Gaussian distribution equation uses relatively simple calculations using two dispersion parameters (i.e. \(\sigma_y\) for horizontal dispersion and \(\sigma_z\) for vertical dispersion) to identify the variation of plume concentration away from plume center. Plume distribution predicted by Eq. 7 determines ground level plume concentration using time-averaged atmospheric temperature and wind speed (and direction). This approach limits predicted plume concentration downwind of the stack in terms of normally distributed “average” concentration as illustrated in Figure 2:

\(\chi\) - Mean (maximum concentration)
\(\sigma\) - Standard deviation symbol

---

\(^1\) For example, in Tulsa, OK the flight path for the Tulsa International airport passes directly over the John Zink flare test facility. When flares are tested, John Zink must inform the airport of their testing to avoid potential hazards to low flying aircraft on their final approach.
solar radiation. This approach assumes: 1) relatively flat surface along plume path, 2) plume reflected from surface, 3) constant plume emission rate and 4) uniform wind speed and direction. In this case, the horizontal and vertical dispersion coefficients are obtained graphically as a function of atmospheric stability\(^2\) with each increasing with decreasing atmospheric stability.

One-dimensional plume rise formulas discussed previously calculate the imaginary centerline of a flare plume trajectory where the maximum concentration exists and assumes a uniform distribution from the centerline. However, these simple one-dimensional formulas are not able to estimate how pollutant concentrations vary non-uniformly from the centerline path. Gaussian dispersion predicts pollutant concentrations at points of interest away from the source (i.e., downwind ground level, surface of an elevated building platform, etc.). Existing plume models developed by the US Environmental Protection Agency (USEPA) are available to perform this type of analysis. [5]

**Application: Enclosed Flare Plume Formation and Dispersion**

To illustrate the Gaussian approach for plume analysis, the formation and dispersion of a plume from the enclosed flare shown in Figure 3 was analyzed. The flare system included up to 3 units similar to the one shown in Figure 3 with the system processing up to 37,000 kg/hr flare gas with a molecular weight of approximately 25 containing saturated and unsaturated hydrocarbons plus halogens with heavy metal contaminants and a Lower Heating Value between 1200 and 6200 KJ/m\(^3\). The analysis estimated the initial plume conditions using the NASA-Glenn Chemical Equilibrium with Applications code (NASAcea2) to calculate expected plume composition at various stoichiometries and temperatures [6]. This information was then used to determine the flare tip gas exit velocity.

For this plume analysis, the objective was to estimate ground level concentration of certain emissions at a predetermined location to support the air permit application process. Using NASAcea2, the combustion effluent composition with solid particulates and the expected gas temperature as a function of combustion stoichiometry were calculated. Given operating conditions and flare diameter, the flare tip velocity was estimated (see Figure 4). With the tip velocity, effluent composition, and the atmospheric stability, plume dispersion was estimated for the downwind location.

![Figure 3 - Enclosed flare with burner system](image)

\(^2\) Atmospheric stability depends on temperature difference between plume and surrounding air. Stability levels can be stable, conditionally stable, neutral, conditionally unstable, or unstable which for plume modeling are classified by five surface wind speed categories, three types of daytime insolation, and two types of nighttime cloudiness and are referred to as Pasquill-Gifford stability classes. Stabilities A, B, and C refer to daytime hours with unstable conditions while stability D represents overcast days or nights with neutral conditions and stabilities E and F refer to nighttime, stable conditions based on amount of cloud cover.
Results shown in Table 1 include estimated levels of SO2, NO2, CO, Total Solid Particulate (TPS) and concentration of particles with mean diameters less than 2.5 microns (PM2.5). This information was then reported to the client and used as part of their air permit application for the enclosed flare system.

Table 1 – Estimated Flare Plume Dispersion

<table>
<thead>
<tr>
<th>Source Coordinates</th>
<th>Base Elevation (m)</th>
<th>Stack Height (m)</th>
<th>Stack Diameter (m)</th>
<th>Exit Temperature (K)</th>
<th>Exit Velocity (m/s)</th>
<th>Stack Capped or Not</th>
<th>Pollutants Emission Rates at Normal Operating Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>X (m)</td>
<td>Y (m)</td>
<td>(H)</td>
<td>(H)</td>
<td>(K)</td>
<td>(m/s)</td>
<td>(Yes/Not)</td>
<td>SO2 (g/s)</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>7.00</td>
<td>16.76</td>
<td>3.51</td>
<td>939.26</td>
<td>10.84</td>
<td>Not</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>7.00</td>
<td>16.76</td>
<td>3.51</td>
<td>939.26</td>
<td>10.84</td>
<td>Not</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>7.00</td>
<td>16.76</td>
<td>3.51</td>
<td>939.26</td>
<td>10.84</td>
<td>Not</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>7.00</td>
<td>16.76</td>
<td>3.51</td>
<td>939.26</td>
<td>10.84</td>
<td>Not</td>
</tr>
</tbody>
</table>

3-D Steady State Plume Dispersion Model

To more accurately predict plume formation, rise and dispersion, computational fluid dynamics (CFD) can be used. To analyze the steady state plume dispersion, the CFD code Fluent has been used. This code describes the reactive turbulent flow associated with the combustion of flare gas and the subsequent formation and dispersion of the flare plume. A detailed discussion of the CFD code is not given here but a summary is given for comparison of the complexity of the CFD approach compared to the Gaussian dispersion approach.

Steady State RANS Model

In general, the general Reynolds Averaged Navier-Stokes (RANS) steady state equations describing the conservation of mass, momentum, and energy written in Cartesian tensor form are [7, 8]:

\[
\frac{\partial (\rho \hat{u}_i)}{\partial x_j} = 0; \text{ mass conservation} \tag{8}
\]

\[
\frac{\partial (\rho \hat{u}_i \hat{u}_j)}{\partial x_j} = -\frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \tau_{ij} - \rho \hat{u}_i \hat{u}_j \right) + \rho \hat{f}_i; \text{ momentum conservation} \tag{9}
\]

\[
\frac{\partial (\rho \Phi)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \Gamma_{\Phi} \frac{\partial \Phi}{\partial x_j} - \rho \Phi \hat{u}_i \hat{u}_j \right) + \tau_{ij}; \text{ general conserved scaler } \Phi \text{ (i.e., energy)} \tag{10}
\]

where:

- \(\tau_{ij}\) = viscous stress tensor
- \(\Phi\) = transported variable (chemical species, enthalpy, etc.)
- \(\Gamma_{\Phi}\) = molecular diffusion coefficient for \(\Phi\)
\[ S_\phi = \text{source term for } \Phi \]
\[ \rho = \text{density of the fluid} \]
\[ P = \text{static pressure} \]
\[ g = \text{acceleration of gravity} \]

Individual species transport is described by:
\[ \frac{\partial}{\partial t} \rho Y_i + \nabla \cdot (\rho \vec{u} Y_i) = \nabla \cdot (\rho D_i \nabla Y_i) + \dot{Y}_i \]  \hspace{1cm} (11)

where the species source term, \( \dot{Y}_i \), accounts for the consumption and production of each specie described by:
\[ \frac{\partial h_i}{\partial t} = \dot{Y}_i = \sum_{k=1}^{K} V_{ik} \rho_k \]  \hspace{1cm} (12)

Together with the total heat released during combustion which is written as the sum of the product of reaction enthalpy \( "h" \) and concentration change for the species:
\[ q_k = \sum_{k=1}^{K} q_k = \sum_{k=1}^{K} \left( \frac{\Delta H_i}{\chi_{i,k}} \right) \left( \frac{dc_i}{dt} \right)_k = -\sum_{i=1}^{N} h_i \omega_i \]  \hspace{1cm} (13)

Turbulent pre-mixed combustion is simulated using either the Eddy-Breakup model (EBU) \([9]\) or the more general Eddy-Dissipation Concept (EDC). \([10]\)

Several turbulence models have been proposed \([11]\), but the \( k - \varepsilon \) turbulence model, originally proposed by Harlow and Nakayama \([12]\) remains the most widely used to describe practical flow systems \([13]\). Using the modified Boussinesq hypothesis \([14]\) the transport equations for the standard \( k-\varepsilon \) model are:

\[ \rho U_j \frac{\partial k}{\partial x_j} = C_{\mu} \varepsilon k^{\frac{2}{3}} \rho k \varepsilon \frac{1}{\chi_{i,j}} - C_{\varepsilon 2} \frac{\partial}{\partial x_j} \left( \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \] : Turbulence Kinetic Energy  \hspace{1cm} (14)

\[ \rho U_j \frac{\partial \varepsilon}{\partial x_j} = C_{\varepsilon 1} \frac{\varepsilon^2}{k} \rho k \frac{1}{\chi_{i,j}} \frac{\partial U_i}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] \] : Dissipation Rate  \hspace{1cm} (15)

\[ \mu_t = \frac{C_{\mu} \rho k^2}{\varepsilon} \] : Eddy Viscosity  \hspace{1cm} (16)

With the empirical closure coefficients included in Eqs. 14-16 taken as:
\[ C_{\varepsilon 1} = 1.44, \quad C_{\varepsilon 2} = 1.92, \quad C_{\mu} = 0.09, \quad \sigma_k = 1.0, \quad \sigma_\varepsilon = 1.3 \]  \hspace{1cm} (17)

Equations 8 through 16 include several steady-state, second-order, non-linear, elliptic partial differential equations (PDE). Solving these continuous PDE’s requires that they be transformed into discrete finite difference equations (FDE’s). The set of FDE’s are then solved using various numerical schemes (i.e., SIMPLE = Semi-Implicit Method for solving Pressure Linked Equations) to obtain a “converged” solution for the flow, temperature, and concentration profiles of an operating flare and its resulting plume.

**Application: Air-Flare Tip Design Impact on Plume Shape**

A recent analysis of an air-assisted gas flare illustrates the use of a RANS based CFD code to predict flare plume formation and dispersion. Typically, air-assisted flares employ complex geometries to enhance
mixing between the flare gas and surrounding air. The present air-assisted flare included alternating air and gas tubes (see Figure 5) through which approximately 170,000 lbs/hr flare gas and 29,000 CFM is flowing at 60°F. The plume formed with this condition was predicted using the Fluent CFD model for no-wind and a 3-20 mph wind (see Figure 6). As shown, no wind (or low wind) has little effect on this plume while at higher wind velocities (~20 mph) the plume clearly is deflected and begins to form the expect barrel vortices which are commonly observed in plumes as they rise from a stack into the wind above the stack. Recalling the

earlier discussion about plume dynamics and the relationship between stack exit velocity and cross wind velocity, this behavior is not unexpected. As shown at no or low wind conditions, plume momentum dominates as the plume rises well above the stack exit and maintains a cylindrical shape. As wind velocity increases, the outer regions of the plume are deformed and the shape changes into a horse shoe shape with vortices forming on the outer edges perpendicular to the wind. Plume rise is reduced as the wind bends the plume over and deforms its shape. This example shows the limitation of a Gaussian plume dispersion analysis which predicts an average plume path with a constant concentration along this path.

3-D Transient Plume Dispersion Model

An even more completed modeling approach utilizes an LES based turbulent reacting flow CFD model called C3D. This tool has previously been applied to analyze plumes from pool fires [15, 16, 17] and gas flares. [18, 19]

To simulate flare plumes, C3D uses an LES based turbulent mixing model. Turbulent mixing sets the combustion rate and controls plume mixing with surrounding air. The energy equation combines combustion with turbulent mixing and radiation heat transfer. Individual species transport equations predict the distribution and concentration of fuel, oxygen, intermediate species, soot, and combustion products. The combustion model (described below) provides source and sink terms as a function of species concentrations, local gas temperature, and turbulent diffusivity.
The radiation model estimates flame emissivity as a function of molecular gas composition, soot volume fraction, flame size, shape and temperature distribution. Radiation flux between the flame and external surfaces is predicted assuming the internal flame is optically thick so radiation transport is diffuse. Diffuse radiation is modeled employing an effective, temperature-dependent flame thermal conductivity. Outside the flame (defined as local soot volume fraction < specified minimum soot volume fraction) the gaseous medium does not participate in radiation heat transfer. The radiation model uses view factors between all points on the flame surface and all points on surrounding objects in the computational domain. Solid objects in the domain also radiate to the environment when they are not engulfed in flames.

The combustion scheme used is based on a proposed hybrid EBU method. Species included in this hybrid combustion model include fuel vapor (F) from the flare tip, oxygen (O2), products of combustion (PC) including water vapor and carbon dioxide, radiating carbon soot (C), and non-radiating intermediate species (IS). The general combustion reactions involving these species include:

\[
1\text{kg} \, F + (2.87 - 2.6S_1) \text{kg} \, O_2 \rightarrow S_1 \text{kg} \, C + (3.87 - 3.6S_1) \text{kg} \, PC + (50 - 32S_1) \text{MJ} \tag{18}
\]

which describes incomplete combustion of fuel vapor (F) and produces carbon soot (C) and products of combustion (PC) plus energy. The standard combustion soot stoichiometric parameter (S1) is set to 0.05 but can be adjusted based on fuel type. For natural gas a value of 0.005 is used.

The endothermic fuel pyrolysis or cracking reaction (soot producing) consumes fuel vapor (F) and energy and produces radiating carbon soot (C) plus the intermediate species (IS):

\[
1\text{kg} \, F + 0.3 \text{MJ} \rightarrow S_2 \text{kg} \, C + (1 - S_2) \text{kg} \, IS \tag{19}
\]

This step includes the cracking parameter (S2) which is set as 0.15 but can also be adjusted based on fuel type. Finally, soot combustion is described by:

\[
1\text{kg} \, C + 2.6 \text{kg} \, O_2 \rightarrow 3.6 \text{kg} \, CO_2 + 32 \text{MJ} \tag{20}
\]

which consumes carbon soot (C) and oxygen (O2) and produces carbon dioxide (CO2) plus some energy. Combustion of intermediate species (IS) is described by:

\[
1 \text{kg} \, IS + \frac{2.87 - 2.6S_2}{1 - S_2} \text{kg} \, O_2 \rightarrow \frac{3.87 - 3.6S_2}{1 - S_2} \text{kg} \, PC + \frac{50 - 32S_2}{1 - S_2} \text{MJ} \tag{21}
\]

where the coefficients are selected so that complete combustion of carbon soot (C) and intermediate species (IS) produce the same species and thermal energy as direct combustion of the fuel. The coefficients in the formula are in terms of mass weight and not moles.

The advantage of the three-step reaction is that the first reaction has a low activation energy, which allows the partial burning and heat release of the flare gas. This maintains combustion since the partial heat released allows the second reaction, which produces most of the heat and all of the soot, to proceed.

The flare gas Arrhenius combustion time scale is combined with the turbulence time scale to yield an overall reaction time scale for the reaction:

\[
\tau_{\text{turb}} = C \frac{\Delta x^2}{\varepsilon_{\text{diff}}} \tag{22}
\]

---

3 The view factor calculation does not include shadowing effects from irregular fire surfaces.
where $\Delta x$ is the characteristic cell size, $C$ is a user input constant ($0.2E-4$), $\varepsilon_{diff}$ is the eddy diffusivity from the turbulence model, and $\tau_{turb}$ is the turbulence time scale (i.e., characteristic time required to mix contents in computational cell). The reaction rates are combined by simple addition of the time scales. All reactions are based on Arrhenius kinetics:

$$\frac{df_{R_i}}{dt} = -C \left[ \prod_{i=1}^{N} f_{R_i} \right] e^{-\frac{T_A}{T}} \tag{22}$$

where the “$C$” coefficients and the Activation Temperatures, $T_A$, are supplied for each reaction. This hybrid combustion approach efficiently simulates turbulent hydrocarbon combustion to estimate the flare plume composition and then uses an LES based mixing model to simulate plume transport.

**Application: Single Tip Ground Flare Plume Impact on Surrounding Equipment**

Sometimes flare plumes impact surrounding equipment and may pose safety hazardous to operations staff. The next application considers a single tip ground flare tip located 2 meters above grade and 23 meters below an overhead cable. The problem considered was whether the flare plume temperature would exceed the safe operating temperature (250ºC) of the nearby equipment (see Figure 7). To address this question, a transient LES based combustion analysis of the flare was completed using the C3D code describe earlier.

The ground flare burned ethylene flowing at 200 Nm3/hr at 20ºC. The combustion model used for this analysis included three reaction steps:

$$C_2H_4 + 0.57 O_2 \rightarrow 0.93 C_2H_2 + 0.64 H_2O + 0.94 MJ/Kg \text{ Ethylene}$$

$$C_2H_2 + 2.58 O_2 \rightarrow 2.7 CO_2 + 0.7 H_2O + 0.25 Soot + 34.1 MJ/Kg \text{ Intermediate}$$

$$\text{Soot} + 2.66 O_2 \rightarrow 3.66 CO_2 + 32 MJ/Kg \text{ Soot}$$

The computational domain and respective mesh structure are shown in Figure 8:
As shown above, the mesh is refined near the cable and around the flare tip. This allows improved resolution of the flare plume and better prediction of temperature profile. Performing transient LES analysis of the flare system using the 3-step combustion model on the computational mesh provided estimates of plume temperature up to 50 meters above the flare tip (see Figure 9). The elevation shown in Figure 9 represents total elevation above ground level. Given the flare tip was located approximately 2 meters above grade, the actual cable elevation is 25 meters above grade.

Since C3D provides a transient analysis of the flare plume, a time averaged calculated temperature profile is possible (see Figure 10). This clearly shows that the plume temperature at the cable elevation is 221°C but actually ranges between about 100°C and 350°C. This type of plume analysis also provides the same level of detail for gas species which allows one to assess O2 and CO levels around nearby buildings.
SUMMARY AND CONCLUSIONS

Flare plume analysis is an important topic that must be addressed when designing a flare system located in an operating plant. This paper presented a summary of three different types of dispersion models used to analyze flare plumes. Depending on the amount and type of information required, the selected model may range from a 1-dimensional Gaussian dispersion model to a 3-dimensional steady state RANS based CFD model to a 3-dimensional LES based transient combustion model.

To estimate the concentration of flare emissions downwind of a flare stack required for environmental air permits, a simple Gaussian dispersion analysis may be appropriate. This may be appropriate when complex chemistry and solid particulates are involved. The Gaussian dispersion model predicts the average plume path and assumes a constant source location with uniformly distributed dispersion from the plume along its path.

As shown, a flare plume can be dramatically affected by windy conditions and flare tip geometry. In this case, a steady state CFD model was used to estimate plume shape. Results of this analysis showed that the flare plume may not have a uniformly distributed dispersion profile along its path. In this case, the plume shape changed as it transitioned from a momentum dominated plume to a buoyancy dominated plume. The plume momentum was also shown to be a strong function of flare tip geometry and assist media flow rate. This is important as the plume dynamics will change as a function of operating condition for a given tip geometry. Thus, for an accurate estimate of plume dispersion downwind of the flare a detailed CFD plume analysis should be conducted.

In some cases, plume dynamics are also critical when evaluating potential hazards to nearby equipment and operations personnel. For the case shown, the average predicted temperature was below the critical operating temperature but dynamic fluctuations in plume temperature exceeded the safe operating condition for the equipment. Thus, when considering safety issues related to flare plume dispersion, a full transient LES based CFD plume dispersion analysis will be required.
REFERENCES