Numerical Study of Smoke Emissions of an Air-Assisted Elevated Flare

Qi Guan1, *, Chao Guo1 and Shoukun Hou1
1Shanghai Marine Diesel Engine Research Institute, Shanghai, 201108, China

*mech_guanqi@163.com

Abstract. In the petroleum industry, flares are used to burn off unwanted waste gas generated during upstream processes. For elevated flares where a huge amount of waste gases is gathered and burned at a single flare header, visible smoke emission is a common problem. It is known that smoke emission is a complex thermochemical process and is affected by many factors. To suppress smoke production, air or steam is often injected into the header to supply additional oxygen and enhance mixing. In this paper, computational fluid dynamics (CFD) is used to study the smoking behavior of an air-assisted elevated flare of a domestic petroleum company. The Moss-Brookes model is used to simulate the soot generation. The impacts of fuel composition, fuel flow rate, and crosswind are investigated by varying the fuel type, waste gas mass flow rate and crosswind speed. Our results suggest that soot concentration is strongly affected by the fuel composition and flow rate. The crosswind speed also greatly affects the flame shape and size and the overall soot generation.

1. Introduction
In petroleum companies, waste hydrocarbon gases are centralized in the flare system and burned before releasing to the atmosphere. According to the estimation by the World Bank, there were nearly 150 billion cubic meters of waste gases burned through flaring annually [1]. Successful disposal of the waste gas requires complete combustion and maintains emissions of NOx, SOx, poisonous gases and/or smoke at permissible levels. For elevated flares, combustion normally takes place above 100 meters or higher. If not carefully controlled, toxic emissions can spread downstream to nearby cities and cause serious health problems to local residents.

Smoke emission usually happens when the waste gas has a large flow rate or contains unsaturated hydrocarbon molecule, e.g. toluene or butadiene, which leads to soot precursors during combustion. Visible smoke emissions cause air pollution which leads to respiratory problems and crowd panic and therefore is strictly prohibited by environmental law enforcement. In order to control smoke emission, one often injects assisted air or steam into the combustion zone to provide additional oxygen and improve the fuel-air mixing.

Numerous work has been devoted to the study of soot formation mechanisms and emission measurements. However, most of these works have been performed in the laboratory environment where combustion is well controlled. Due to the unpredictability of the ambient and operating conditions as well as the open-air combustion mode, a quantitative field measurement of the smoke emission is nearly impossible.
In recent years, with the development of computational fluid dynamics (CFD) and combustion mechanisms, numerical studies of flare operation have become a popular approach [2, 3], from which smoke emissions can be quantitatively studied by applying appropriate models.

2. Methodologies

2.1. Conservation Equations

In order to determine the distributions of temperature and molecular species in a fluid dynamics problem, one needs to solve the Navier-Stokes equations which includes the conservations of mass, momentum and energy. With the interactions of the fuel injection and the crosswind, the fluid dynamics is turbulent. In this study, the standard $k-\varepsilon$ model is chosen to solve the turbulence fluctuation. Neglecting the contribution of buoyancy and compressibility of fluid, the transportations of $k$ and $\varepsilon$ are expressed as:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i k) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - \rho \varepsilon \tag{1}$$

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i \varepsilon) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} G_k + C_{2\varepsilon} \rho \mu_t \varepsilon^2 \tag{2}$$

Where $G_k$ represents the turbulence generation due to the mean velocity gradients. $\sigma_k$ and $\sigma_\varepsilon$ are turbulent Prandtl numbers for $k$ and $\varepsilon$. $C_{1\varepsilon}$ and $C_{2\varepsilon}$ are constants.

2.2. Moss-Brookes Soot Model

Smoke generated in a diffusion flame is also called soot. Its formation follows four important stages: (a) nucleation; (b) molecular growth; (c) aggregation and (d) oxidation.

Several soot models are available in ANSYS Fluent, which includes the one-step Khan and Greeves model, two-step Tesner model, Moss-Brookes model and Moss-Brookes-Hall model. The kinetic mechanism of the Moss-Brookes model is based on data from lab experiments, so it is chosen to solve the soot generation and destruction.

The Moss-Brookes model solves transport equations for normalized radical nuclei concentration $b_{nuc}^*$ and soot mass fraction $Y_{soot}$:

$$\frac{\partial (\rho Y_{soot})}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i Y_{soot}) = \frac{\partial}{\partial x_i} \left( \frac{\mu_t}{\sigma_{soot}} \frac{\partial Y_{soot}}{\partial x_i} \right) + \frac{dM}{dt} \tag{3}$$

$$\frac{\partial (\rho b_{nuc}^*)}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i b_{nuc}^*) = \frac{\partial}{\partial x_i} \left( \frac{\mu_t}{\sigma_{nuc}} \frac{\partial b_{nuc}^*}{\partial x_i} \right) + \frac{1}{N_{norm}} \frac{dN}{dt} \tag{4}$$

Where $Y_{soot}$ is soot mass fraction, $M$ is soot mass concentration (kg/m$^3$), $b_{nuc}^*$ is the normalized radical nuclei concentration (particles/$10^{15}$/kg) and $N$ is the soot particle number density (particles/m$^3$). $N_{norm} = 10^{15}$ particles [4].

3. Numerical Modelling

3.1. Geometry and Mesh

A typical air assisted flare header is shown in Figure 1, where flare gas and assisted air are separately conveyed to the flare header through inner and outer cylinders. The outer diameter of the flare header is about 3.3 m and the diameter for the fuel tunnel is about 1.7 m. A flare with this size runs a maximum disposal flow rate of 250 kg/s. The assisted air is injected into the header through frequency conversion blowers and the maximum air supply is approximately 118 kg/s. On top of the header, vent tunnels of flare gas are evenly distributed among the air tunnels to enhance the fuel and air mixing. In our study, the flare structure, combustion domain and mesh are all created in the pre-processing software Gambit 2.6. Figure 2(a) shows the computational domain and the flare position while Figure
2(b) shows a simplified header geometry where the waste gas inlet is marked in blue and assisted air inlet in red.

![Figure 1](image1.png)

**Figure 1.** Air-assisted flare header

![Figure 2](image2.png)

**Figure 2.** Geometry model used in this study

Meshing is critical factor in CFD simulation. A good quality mesh can not only lead to more accurate results but also save iterations on re-model and re-mesh. In our work, the cell number for the entire computational domain is about 2.21 million. Denser mesh is built near the flare header where mixing and combustion take place, while coarser mesh is built near the domain boundary. Size functions are applied to adapt the cell size variation between dense mesh and coarse mesh. Multiple iterations of mesh improvements are performed along the simulation to reach converged results. The series of this study are performed on a DELL Precision T7600 workstation with a processor of 16 Intel® Xeon® CPU at 3.1 GHz and a RAM of 128 GB.
3.2. Numerical Settings and Boundary Conditions
All simulations are performed in the steady state pressure-based solver. The SIMPLE algorithm is chosen for velocity-pressure coupling. Standard k-ε model is used for turbulence and the P1 model is used for radiation. The combustion chemistry is solved using a non-premixed combustion species model (PDF model) [4]. The flamelets are generated using USC 2.0 mechanism and thermodynamic data [5]. The soot precursors are C2H2, C2H4, and C6H6, while C2H2 and C6H6 were used as the surface growth species. Fenimore-Jones was chosen as the soot oxidation model. Gravity is considered by applying a gravitational acceleration of -9.81 m/s². All transport equations are applied to a 2nd order upwind scheme except for the turbulent terms to a 1st order upwind scheme.

The inlets of waste gas and assisted air are applied with mass inlet boundary conditions. The surrounding and the upper faces of the domain have pressure outlet boundary conditions. The temperatures of the air and the waste gas are 276 K and 473 K, respectively.

4. Results and Discussion
In this section, results for all series of simulations are presented. We compare the contour plots of temperature, flame shape and soot distribution to analyse the impacts of the fuel composition, the flow rate of waste gas and the crosswind speed.

4.1. Impact of fuel composition

Table 1. Waste gas fuel composition (by mole fraction)

| Composition | Fuel 1 | Fuel 2 | Fuel 3 |
|-------------|--------|--------|--------|
| Methane     | CH₄    | 5%     | 50%    | 80%    |
| Ethylene    | C₂H₄   | 10%    | 20%    | 10%    |
| Propylene   | C₃H₆   | 85%    | 30%    | 10%    |

Figure 3. Contours of temperature for different fuel types
Figure 3 shows the temperature contours on the z=0 plane of the domain for the three cases. Since all three cases have the same mass flow rates of fuel and assisted air and the heating values of fuels are similar, comparable temperature distributions are obtained. One can also notice that the maximum temperature when using fuel 1 is about 100 K higher than when using fuel 2, which in turn is slightly higher than fuel 3. This is probably because fuels 2 and 3 have lighter molecular weights, and the ejecting speeds are higher, thus more air is entrained into the combustion zone to cool down the temperature.

On the other hand, smoke emissions of the three flares behave quite differently. Figure 4 shows the contours of soot mass fraction on the z=0 plane, where the range of soot yield of the three cases are similar to each other. Figure 5 is a plot of the soot mass fraction centreline distributions for the three fuel cases. As shown, the incipient and terminating smoking points of the three cases lie closely on the axis of distance, but the soot mass fraction of fuel 1 is about twice of fuel 3. In real observation [6-7], since most of the visible light is emitted from the hot soot particles, the flame of fuel 1 will look much brighter than the fuel 2 and fuel 3 cases, even though their temperature values are close.

4.2. Impact of fuel flow rate

Ideally, it would be nice to have the correlation between smoke visibility and air supply so that smoke can be accurately and timely captured and suppressed. This is hard to achieve because the waste gas
flow rate often varies significantly during operation. In this work, we study the impact of fuel flow rate by comparing the temperature and soot distributions under the fuel flow rates of 5 kg/s, 15 kg/s and 25 kg/s, i.e. 2%, 6%, and 10% of the full designed disposal capacity. Fuel 1 in Table 1 is used as the waste gas and the assisted air flow rate is maintained at 39.4 kg/s.

As is shown in Figure 6, the flame length increases with the flow rate of the waste gas in addition to the maximum temperature. As shown in Figure 7, the soot mass fraction follows a similar pattern of the temperature distribution. When the flow rate of the fuel increases, the soot yield area becomes significantly longer and wider. This agrees with laboratory observations: more fuel needs more air to combust and therefore the flame becomes longer and wider. One may also notice that the maximum soot mass fraction only slightly increases with the fuel flow rate, from about 8% to 10%. This means the brightness of the flame is only slightly affected by the fuel flow rate.
Figure 8. Centerline profile of soot mass fraction under different fuel flow rates

Figure 8 plots the soot mass fraction along the flame centerline. As shown, when flare runs with higher fuel flow rates, the range of soot emission becomes wider. The noticeable soot emission range increases from 20 m for the 5 kg/s case to about 35 m for the 25 kg/s. Therefore, the overall soot production also increases with the fuel flow rate. To suppress the smoke emission during operation, one often needs to increase the assisted air supply or enhance the fuel-air mixing such as partially premixed combustion.

4.3. Impact of crosswind

During operation, elevated flare combustion is also always affected by the ambient wind. Air entrainment and mixing can be very different depending on the wind speed. To classify the flare flame, one often uses the momentum flux ratio \( R \) [8]:

\[
R = \frac{\rho_j V_j^2}{\rho_\infty V_\infty^2}
\]  

(5)

Where \( \rho_j \) and \( \rho_\infty \) represent the density of the waste gas and the air, while \( V_j \) and \( V_\infty \) represent the velocity. When the wind momentum is smaller than the fuel momentum, the flame tends to be vertical, as previously shown in section 4.1 and 4.2 where wind speed is zero. Under quiescent or mild wind condition, the combustion is said in the “lifted jet flame” region. The surrounding air is entrained continuously into the combustion zone. However, when the wind momentum is greater than the fuel, the flame tends to shift into the wake. In this case, the combustion is in the “wake-stabilized flames” region where the fluid dynamics is dominated by the wind momentum.

In this study, we investigate the effect of crosswind by simulating the flare flame under five crosswind speeds, i.e. 1 m/s, 3 m/s, 5 m/s, 7 m/s and 10 m/s, while the result of quiescent condition is used as the base case. The flow rates of the fuel and the assisted air are maintained at 25 kg/s and 39.4 kg/s, respectively, and fuel 1 in Table 1 is used as the waste gas.

Figure 9 shows the temperature contours under different crosswind speeds. As the crosswind gets stronger, the flame shape tends to shift with the wind and the flame length becomes significantly shorter. When the wind speed is smaller than 5 m/s, flare jet momentum still dominates, as most of the products flow out through the upper boundary. The combustion can be classified in the “lifted jet flame” region. When the wind speed reaches 7 m/s, crosswind becomes a more dominant force, as most of the products flow to the right. At a wind speed of 10 m/s, the flame becomes even shorter and close to the ground. In this case it is considered in the “wake-stabilized flame” region.
Figure 9. Contours of temperature under different crosswind speeds
Figure 10. Contours of soot mass fraction under different crosswind speeds

Figure 10 shows the contours of the soot mass fraction. As indicated, soot distribution generally follows the flow direction of the flame. And although the maximum soot mass concentration for each case is about the same level, one cannot infer the same conclusion for the total smoke emission. By summing the dot product of the flow density $\rho$, soot mass fraction $x_{soot}$, normal velocity $v$ and area $A$ over the outlet boundary, one can get the overall soot production rate [3],

$$\dot{m}_{soot} = \sum_{boundary} \rho x_{soot} v A$$
Figure 11. Overall soot production rate under different crosswind speeds

As shown in Figure 11, the overall soot production rate generally increases with the wind. Soot production in the 10 m/s case is about 10 times that of the quiescent case. This is somewhat contradictory to the perceived impression that stronger ambient wind leads to less visible smoke. One explanation might be that smoke diffuses more quickly under high wind conditions but the fuel in the wake region has a low momentum which leads to less air entrainment and incomplete combustion. Our modeling study suggests that stronger ambient wind can cause more total smoke production and therefore one needs to increase the fuel jet momentum under high crosswind conditions.

Acknowledgments
This work was supported by Shanghai Pujiang Program (No.16PJ1432000).

5. Conclusions
The combustion and smoke emission of an air-assisted flare under various conditions are numerically simulated. The impacts of fuel composition, fuel flow rate and crosswind speed on smoke emission is analyzed using CFD software ANSYS Fluent. Based on the simulated results, it can be concluded that soot mass concentration is strongly affected by the fraction of unsaturated hydrocarbons in the fuel. This study also indicates that higher fuel flow rates leads to higher flame temperatures and wider soot production range. Strong crosswind can greatly vary the flame shape and length which can significantly increase the overall soot production.

References
[1] Global Gas Flaring Reduction Partnership (GGFR), World Bank, October 2011 Brochure p. 3
[2] Castiñeira D and Edgar T F 2006 CFD for simulation of steam-assisted and air-assisted flare combustion system. Energy & Fuels 20(3) 1044-56
[3] Castiñeira D and Edgar T F 2008 CFD for simulation of crosswind on the efficiency of high momentum jet turbulent combustion flames. J. Environ. Eng. 134(7) 561-71
[4] Fluent Inc 2011 FLUENT 13.0 User’s Guide
[5] Information on http://ignis.usc.edu/USC_Mech_II.htm
[6] Wagner H G 1981 Soot formation – an overview. Particulate Carbon. 8(4) 1-29
[7] U.S. EPA. 1995 AP 42 – compilation of air pollutant emission factors, Vol. 1, 5th ed. – section 13.5 industrial flares; AP-42; United States Environmental Protection Agency (U.S. EPA): Research Triangle Park, NC
[8] Huang R F and Wang S M 1999 Characteristic flow modes of wake-stabilized jet flames in a transverse air stream. Combust. Flame 117 59-77