Application of CFD Simulation to Safety Problems – Challenges and Experience Including a Comparative Analysis of Hot Plume Dispersion from a Ground Flare

Cristina Zuliani*[a], Claudio De Lorenzi[b], Sabatino Ditali[b]

[a]Saipem SpA, Via Toniolo 1, 61032 Fano, Italy
[b]Saipem SpA, Viale De Gasperi 16, 20097 San Donato Milanese, Italy
cristina.zuliani@saipem.com

Integral or phenomenological consequence models are commonly used for safety related studies at onshore and offshore facilities. These are usually 2D and therefore ignore the influence of the geometry on the ventilation patterns, the dispersion substances and the generation of turbulence that plays a fundamental role in case of explosions. More sophisticated Computational Fluid Dynamics (CFD) models can take into account all these and other relevant phenomena that greatly affect the results. Several design matters can benefit from the use of CFD models in alternative to the simplified consequence models; these include the dispersion of dense gases in low wind conditions, the assessment of explosion risk in congested areas, the dispersion of gases in complex terrains and countercurrent wind, the evaluation of any smoke related impact on plant safety and efficiency. In this paper, the phenomenon of hot plume dispersion from a ground flare is investigated with CFD simulations. The results of a comparative analysis between two CFD codes (Fluent and FLACS), applied to an LNG project, and a phenomenological approach are presented and discussed to identify some peculiar differences between these models. In addition, the study aims at evaluating the effect two adjacent flare pits in simultaneous operation.

1. Introduction

LNG facilities often include large ground flares for the safe disposal of large quantities of process gases in the rare event of emergency situations and/or plant depressurization. One of the major safety concerns related to this type of installations is the large amount of heat transported by the hot plume. The hot gases emitted from the flare, due to their high temperature, may pose an hazard to personnel present on elevated platforms and may lead to degraded performances of some equipment that rely on strict design temperature margins (i.e. gas turbine air intakes, fin-fan air coolers), especially in LNG plants (Pandya, 2014). For this reason, dispersion modelling is often performed to calculate the wind induced patterns of the hot plumes leaving the ground flare, to evaluate the temperature field at specific target points and any interaction with other plant equipment. Due to the complexity of the phenomenon (crosswind effects, influence of geometry, etc.) CFD models represent a more reliable mean to perform such analysis and support the engineering team in making proper design choices.

The purpose of this study is to compare and discuss the results obtained for a large LNG ground flare with two CFD models: a multipurpose code i.e. ANSYS FLUENT (v. 12) and a specialized tool i.e. FLACS v.10.2 (GexCon). FLACS is developed mainly for safety applications; it is the industry standard for CFD explosion modelling and is commonly used for modelling flammable and toxic releases. Even though dispersion of hot combustion gases is not a common application, it is indeed within its range of applicability. Results of a project study performed for a ground flare of an LNG plant with ANSYS FLUENT (v. 12) are used as a term of comparison to define the most appropriate set up in FLACS v.10.2 so as to model such phenomenon. The comparison is based on the iso-surface temperature profile, temperature slice profile and temperature evaluated at predefined target locations. In addition the behaviour of adjacent flare pits operating simultaneously is investigated with respect to the same parameters.

Moreover, in order to highlight the main differences and limitations, the same dispersion case form a single flare has been modelled with a phenomenological proprietary code, DISPGAS (HSE UK, 1996), usually used to simulate continuous releases from smaller vents / accidental ruptures.

2. Comparative study

2.1 Project case description
The choice of a ground flare system as a design alternative to the elevated flare system configuration for a large LNG onshore-plant has been evaluated by addressing the potential detrimental effects on personnel and equipment taking into consideration the temperature mapping at specific locations. A simplified sketch with the flare dimensions is presented in Figure 1.

The two targets monitored in the study are:
- The gas turbine / power generation area located 400m apart from the flare
- The process air coolers arrays located 670m east and 520m south apart from the flare.

Vertical sampling lines provide information on the temperature trend over the plant's height, in correspondence of these two targets. Furthermore detailed 2D plots in correspondence of selected cut-planes are used to obtain temperature patterns. Also 3D plots of temperature iso-surfaces provide effective views of the hot plume patterns.

The temperature thresholds considered for the analysis are:
- 70°C for personnel exposure at any elevation (HSE UK, 2013)
- +2°C temperature rise above ambient for efficient operation of air coolers and gas turbine air inlets (Pandya, 2014).
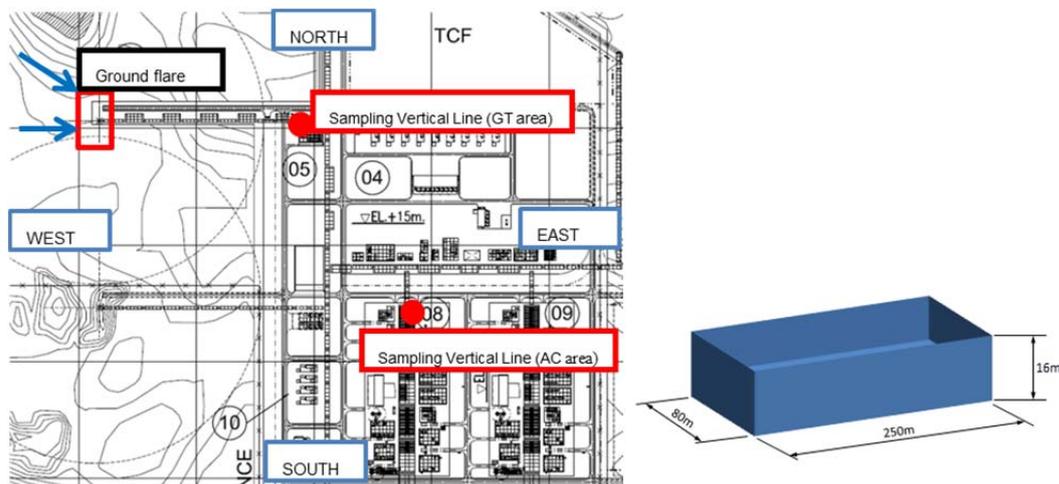
Figure 1: Ground flare geometry and sampling lines over the plant field

2.2 Scenarios analysed
Flow patterns of the hot plume are mainly affected by wind speed and direction. Based on site specific wind statistics and preliminary plant layout, the most frequent wind speeds have been selected for the investigation. The phenomenon is transient as the system operates in case of emergency (e.g. train depressurization) and for a limited period of time, however conservatively the analysis is carried out considering a constant design flare load that corresponds to a constant combustion gases discharge rate calculated considering 100% excess air. Flue gas composition and temperature have been calculated considering complete combustion of the design release case that consists mainly of methane. Four scenarios are investigated. Table 1 summarises the conditions that have been used in the numerical runs.

Table 1: Conditions used for the scenarios

Case ID	Wind direction	Wind velocity (m/s)	Ambient temperature (°C)	Discharge rate (t/h)	Flue gas temperature (°C)
1	W	7	33	74870	1195
2	W	15	33	74870	1195
3	N-W	7	33	74870	1195
4	N-W	15	33	74870	1195

2.3 Modelling approach
The approach used to model the system is based on the following simplifying assumptions:

- the flue gases are thoroughly mixed with entrained combustion air above the flames (at the top of the flare enclosure). Holding this assumption, the top surface of the flare box has been modeled as an emitting source of flue-gases with a uniform value of temperature, constant upward velocity and fixed composition. This has been taken as the initial point for the atmospheric dispersion calculation;
- the mass flow-rate and temperature have been calculated assuming 100% combustion excess air (respect to the stoichiometric value) and used as input to the CFD model; no combustion chemistry;
- flames are entirely shielded by the flare fences. The possible impairing effects of the plume are only related to the convective term of the combustion gases.

This same approach was used with both CFD models (FLUENT and FLACS), however some differences in the computational domain, grid and general set up are highlighted.

In the simulations with ANSYS FLUENT, two different domains were used, based on wind direction: a 1100x900(symmetrical)x500 m with a symmetry plane for the West wind direction; a 1100x1100x700 m for the North-West wind direction. The numerical grid is an unstructured hexahedral grid.

The boundary conditions applied at the domain faces are the ones suggested by Blocken (Blocken et al., 2007) so as to guarantee a proper modeling of the wind profile and turbulence parameters in the atmospheric boundary layer (ABL) which is of paramount importance in determining the plume behavior. A neutrally stratified ABL (corresponding to the stability class D) is implemented in the study following the subroutines based on Richards and Hoxey (Richards and Hoxey, 1993). RNG k-epsilon turbulence model is selected.

In the simulations performed with FLACS (GexCon), the domain is 1500x1500x600m. The numerical grid is a Cartesian grid with cubical 2x2x2m cells in proximity of the flare source. Cells are stretched in the remaining part of the domain. Wind boundary conditions are defined at the inlet faces: wind speed at a reference height and Pasquill stability class are the required inputs used to define the stability of the ABL and calculate its turbulence properties. In FLACS the theory of the characteristic length scale of Monin and Obukhov (Monin and Obukhov, 1954) is implemented to characterize the stability class. Standard k-epsilon model is used. An area leak with rectangular shape and dimensions equal to the ground flare section is located in correspondence of the top surface. The leak is defined as steady state and with a uniform profile across the section. A dispersion and ventilation simulation set up is used to model the cases. Sufficient time is allowed for the wind profile to stabilize throughout the domain before the leak is started.

3. Results

3.1 Temperature comparison in correspondence of gas turbine (GT) location – Case 1&2
In case1 and 2, the plume at one of the above-defined threshold temperature can affect only the gas turbine target located 23m of elevation on the ground level.

FLACS estimates slightly higher temperatures than FLUENT both at height and at ground level. Differences are in the range of max 5%.

Similar behaviour of the plume can be observed. The main difference is identified in the top view of the iso-surface temperature profile (Figure 2): FLUENT predicts a bifurcated plume while in FLACS this behaviour is not so evident in the temperature plot but it is clear from the velocity streamlines (Figure 3). Plume bifurcation is a well-known phenomenon (Ponchaut et al., 2012) that occurs in certain weather and terrain conditions when the plume enters into a cross flow (i.e. wind) giving rise to counteracting vortices. This difference is probably due to the different constant of the k-ε turbulence model in the equation of the dissipation rate and the description of the inlet wind turbulence.

In case 2 the wind has a higher bending effect on the hot plume: this remains close to the ground for longer downwind distances.

For both scenarios FLACS predicts higher temperatures at large downwind distances from the flare. +2°C temperature contour extends more than in FLUENT results. However conclusions are unvaried.

Table 2: Comparison of FLACS vs. FLUENT temperature readings for CASE 1 and CASE 2

	CASE 1 – W 7 m/s		CASE 2 – W 15 m/s	
	FLUENT	FLACS	FLUENT	FLACS
70°C temperature on GT at 400m from flare	Not reached	Not reached	Reached	Reached
Max downwind distance of 70°C contour from flare (reached at height)	600m at 220m	380m at 160m	800m at 180m	550m at 115m
Max T reached at height	325 K at 205m	334 K at 226m	354 K at 105m	354K at 87m
T reached at 23m height at GT	307K	316K (+3%)	329K	345K (+5%)

70°C
FLACS

70°C
FLUENT
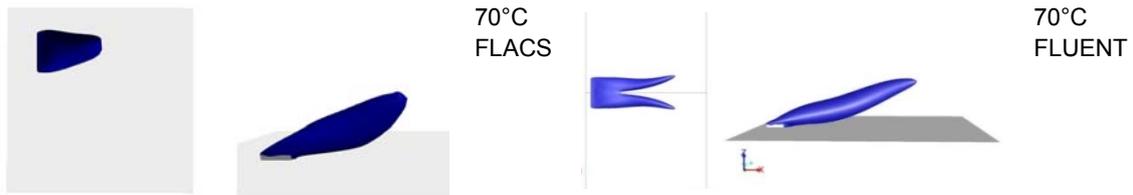
Figure 2: Case 1 - Wind from West; Wind Velocity=7m/s –70°C temperature iso-surface
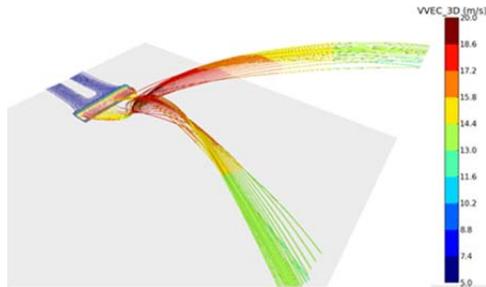
Figure 3: Case 1 - Wind from West; Wind Velocity=7m/s – Velocity streamline

FLACS

FLUENT
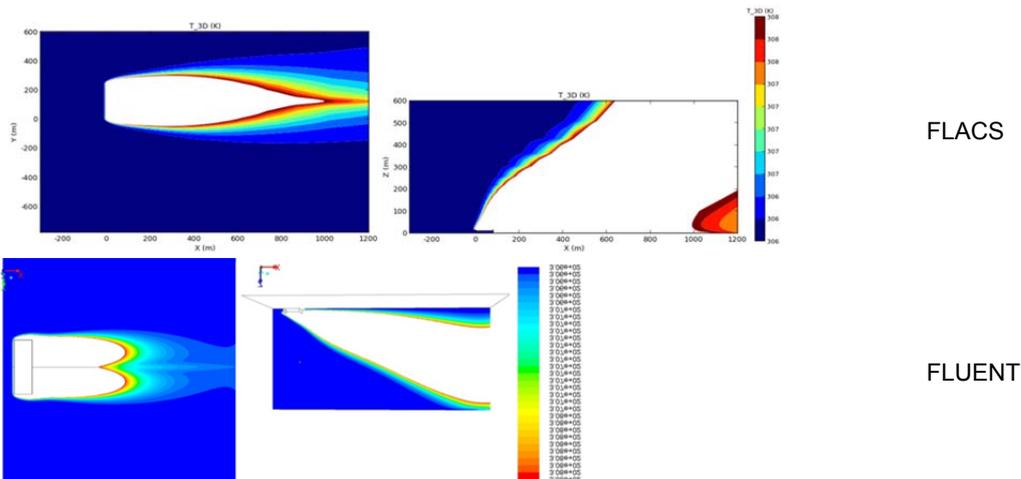
Figure 4: Case 1 - Wind from West; Wind Velocity=7m/s (+2°C temperature respect to the T_{amb} threshold Pic 1 at 23m height AC, Pic 2 lateral view)

3.2 Temperature comparison in correspondence of air coolers (AC) location – Case 3&4
In case 3 and 4, the plume at one of the above-defined threshold temperature can affect only the air coolers array located in elevation. FLACS estimates pretty much the same temperatures as FLUENT both at height and at ground level. In correspondence of the selected target differences are in the range of 1% and however are very close to ambient value (306K).
Slightly different plume behaviour is observed: FLUENT predicts iso-surface plume bifurcation while FLACS predicts a more uniform iso-surface temperature of limited extension with respect to FLUENT. In addition to the points previously highlighted, this may be related also to grid refinement that may affect vortex/turbulence prediction in the far field. FLACS predicts a slightly wider +2°C temperature profile at ground level but overall results are quite similar and same conclusions can be drawn.

Table 3: Comparison of FLACS vs. FLUENT temperature readings for CASE 3 and CASE 4

	CASE 3 – NW 7 m/s		CASE 4 – NW 15 m/s	
	FLUENT	FLACS	FLUENT	FLACS
70°C temperature on AC at 500m from flare	Not reached	Not reached	Not reached	Not reached
Max T reached at height (AC)	317K at 275m 325K at 400m	317K at 362m	322K at115m	317K at 192m
T reached at 23m height at AC	306K	308K (+0.5%)	306K	310K (+1%)
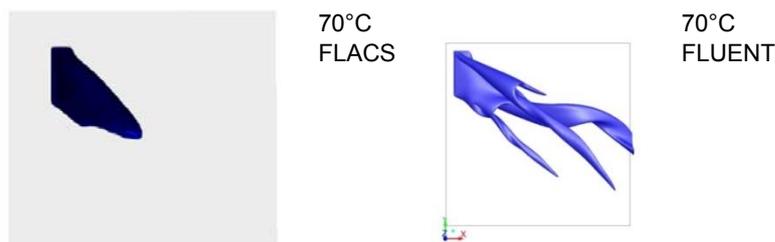
Figure 5: CASE 3 Wind from North West; Wind Velocity=7m/s –70°C temperature iso-surface

FLACS

FLUENT
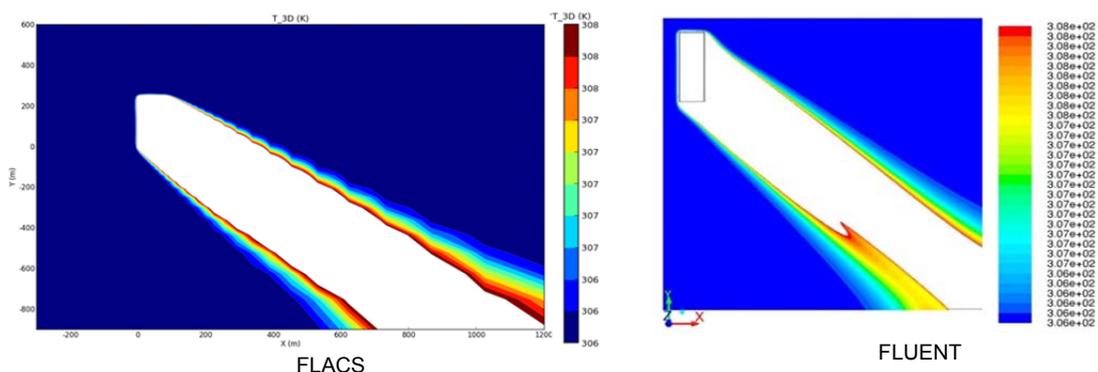
Figure 6: CASE 4 Wind from North West; Wind Velocity=15m/s – (+2°C temperature respect to the T_{amb} threshold at 23m height AC)

3.3 Analysis with a phenomenological model (DISPGAS)

The phenomenological code DIPSGAS has been applied to the single flare pit considering an equivalent circular source of 160m diameter at 16m above ground level. The same cases were screened with the characteristic that W and N-W wind directions make no difference due to the symmetry. The results given below show a much shorter distance and higher elevation reached by the 70°C contour respect to the distances evaluated with the CFD models. Target locations are not reached by the plume.

Table 4: Results of DISPGAS for CASE 1 and CASE 2

	CASE 1	CASE 2
Max downwind distance of 70°C contour from flare (reached at height)	163m at 301m	243m at 160m
T reached at 23m height at target distance	305K at 400m	306.3K at 400m

4. Effect of adjacent flare pits operations

In many plants more than one flare pit can operate simultaneously. The effect of two adjacent flare pits (40m separation distance) is investigated with FLACS for the same scenarios listed in Table 1. Results in correspondence of targets are reported in Table 5. Comparison of system behaviour for CASE 1 is shown in Figure 7. Overall the plume behaves as if it originates from a single larger flare; this is due to the limited separation distance which is responsible for the interaction of the vortices originating from the adjacent flares. Their interaction, especially in the low wind scenarios, generates a single longer and wider plume. In this specific case the presence of an adjacent flare does not alter the conclusions of the analysis.

Table 5: Temperatures results of FLACS for two adjacent flares

ADJACENT FLARES	CASE 1	CASE 2	CASE 3	CASE 4
70°C temperature at target (GT/AC)	Not reached	Reached	Not reached	Not reached
Max downwind dist. of 70°C contour from flare	540m at 128m	610m at 43m	400m at 128m	510m at 81m
Max T reached at height - GT/AC monitor points	331K at 102m	348K at 65m	324K at 226m	318K at 119m
T reached at 23m height	321K	345K	309K	312K
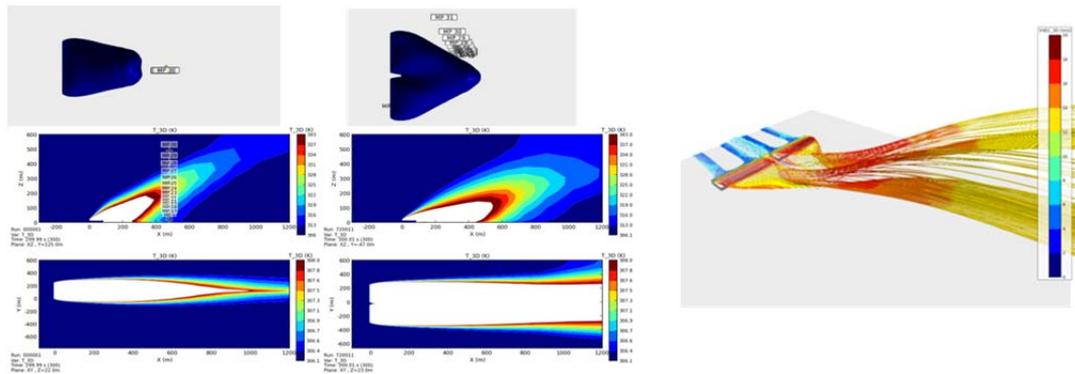
Figure 7: Comparison of CASE 1 results – Single vs Adjacent flares (Pic 1&2 70°C iso-surface; Pic 3&4 70°C temperature side view; Pic 5&6 +2°C contour respect to the T_{amb} at 23m height; Pic 7 velocity streamlines)

5. Conclusions

Hot plume dispersion from a ground flare has been investigated with two CFD models: ANSYS FLUENT and FLACS and compared also with the results of a simpler model DISPGAS. The study presents results of the iso-surface temperature profile, temperature slice profile and temperature evaluated at predefined targets.

Even though the two CFD models are based on slightly different approaches, results are in relatively good agreement and conclusions are the same. Temperatures at specific target locations differ no more than 5%. The profile of the +2°C temperature rise above ambient extends a bit more in FLACS. On the other hand the extension of the 70°C temperature profile is slightly smaller in FLACS.

Possible reasons for these differences in the results are:

- Turbulence model: strength of buoyancy production in the transport equation of the turbulence dissipation rate is different; in FLACS there is less production of turbulence dissipation (more turbulence), therefore more mixing, that in the lower layers, at temperature close to ambient leads to a larger extension of the +2°C temperature rise above ambient, but at the same time leads to a shorter 70°C iso-surface;
- Inlet wind turbulence conditions; this is responsible for the length of the plume in particular at large distances downwind;
- Grid resolution.

Overall the differences between the results of the two CFD models are minimal and main conclusions are not affected. On the other hand, DISPGAS shows a more buoyant plume probably due to fact that effect of geometry (flare fence) is not considered and the different way wind turbulence is modelled.

The simultaneous operation of adjacent flares shall always be investigated as the interaction of the hot gases may produce longer and wider plumes leading to larger areas affected by the temperature increase at ground level or at sensitive locations.

References

Blocken B., Stathopoulos T., Carmeliet J., 2007, Atmospheric Environment vol. 41, CFD Simulations of the Atmospheric Boundary Layer: Wall Functions Problems, 238-252

HSE UK, 1996, Evaluation study of models used in predicting smoke and gas ingress on offshore structures, OTH 95 498

HSE UK, 2013, Indicative human vulnerability to the hazardous agents present offshore for application in risk assessment of major accidents, SPC/Tech/OSD/30, <www.hse.gov.uk/foi/internalops/hid_circs/technical_osd/spc_tech_osd_30/> accessed 26.02.2016

Monin A. S., Obukhov, A. M., 1954, Basic laws of turbulent mixing in the surface layer of the atmosphere, Tr. Akad. Nauk SSSR Geofiz. Inst 24, 163–187

Ponchaut N. F., Kytoomaa H. et al., 2012, Modelling of ground flare pits, 12th Spring meeting & 8th global congress on process safety

Pandya H., 2014, Consider factors affecting onshore LNG plant design, Gas processing <www.gasprocessingnews.com/features/201404/consider-factors-affecting-onshore-lng-plant-design.aspx> accessed 26.02.2016

Richards P.J., Hoxey R.P., 1993, Journal of Wind Engineering and Industrial Aerodynamics 46-47, Appropriate Boundary Conditions for Computational Wind Engineering Models Using the k-ε Turbulence Model