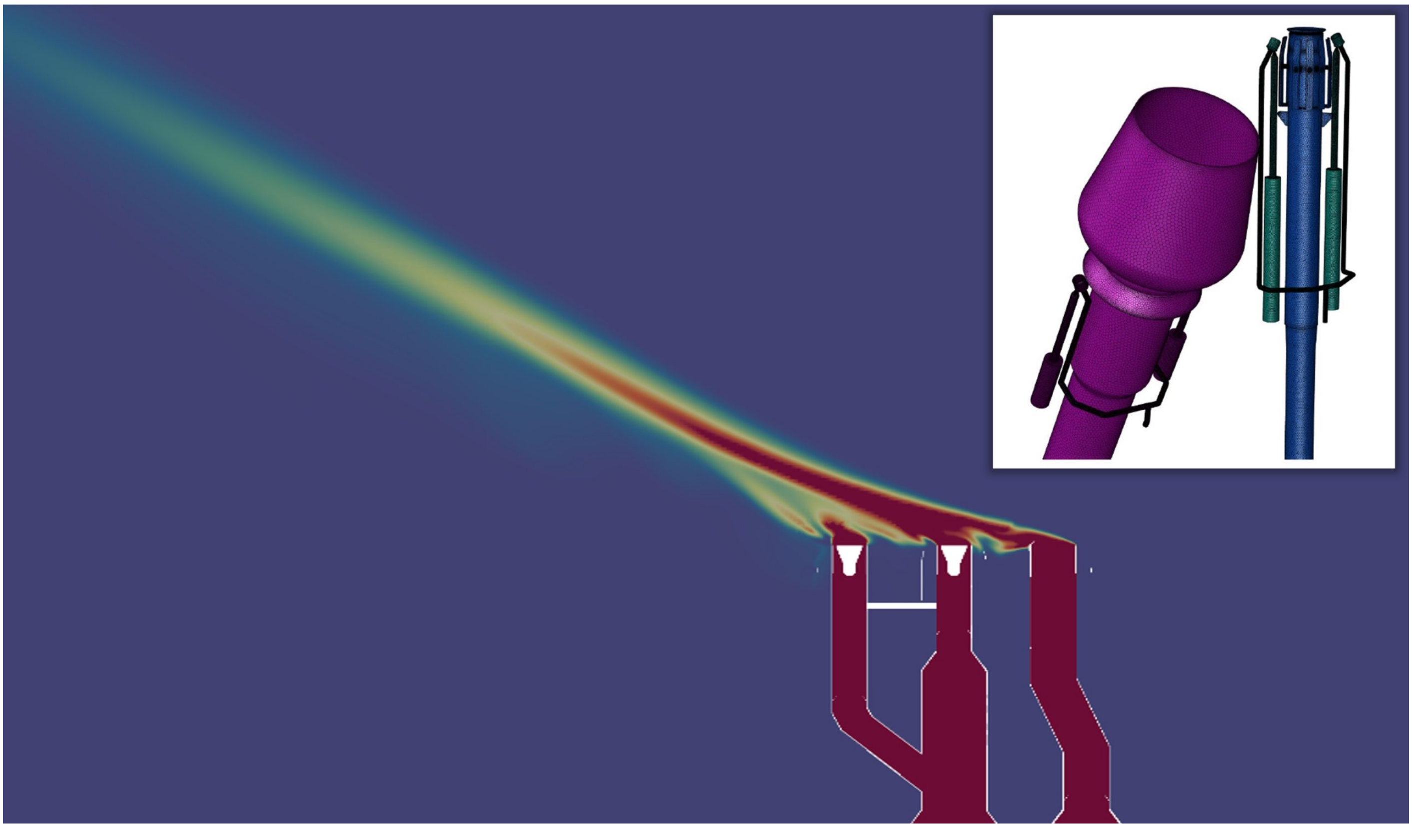
METHANE FROM FLARING TOOLKIT



Environmental Impacts: Computational Fluid Dynamics (CFD) modelling to determine the effect of crosswind on flares efficiency

Do I understand the impacts of the environment on my flare? > Environmental Impacts: Computational Fluid Dynamics (CFD) modelling to determine the effect of crosswind on flares efficiency

Summary

Computational Fluid Dynamics (CFD) modelling simulates combustion in the flare. It can be used to find the optimum flare operating envelope, including assessing performance from changes such as the composition of the gas and identify other flare performance issues such as the impact of varying wind speeds. CFD negates the need for complex empirical tests and can be used to simulate weather conditions in harsh environments (such as storms) that would be hard to replicate in other ways. CFD models require access to advanced computing facilities and expert users to ensure that model is an accurate reflection of real-world conditions.

How it Works

The partial differential equations that govern fluid flow and heat transfer are highly non-linear and must be solved numerically. Therefore, in order to analyze fluid flows, flow domains are split into smaller sub domains. The governing equations are then numerically discretized and solved inside each of these subdomains. The subdomains are often called finite volumes, elements or cells, and the collection of all elements is called a mesh.

Each cell reports values of pressure, velocity and other physical properties that allow heat maps and streamlines to be constructed to visualize the flow field.

By applying modelling and the knowledge of combustion chemistry, it is possible to simulate the combustion at the flare tip to estimate the combustion and destruction efficiencies and measure flare performance from other perspectives (flame stability, flashback, etc.)

Advantages

- Possibility to model any flare type, with/without assist features, any fluid composition
- Helps identify optimal flare at different wind conditions
- Can be used as a tool for design of new flares as well as operations troubleshooting for existing systems. It can support optimisation in terms of combustion, destruction efficiency and other flare performance parameters
- It can identify hazardous scenarios where flame flashback or formation of explosive mixture in the flare system is possible
- Can be used to train data driven digital twin models of flares to support more efficient operation

Limitations

- Steady-state simulation provides a time-averaged solution, however, may not always reflect the transient turbulent mixing. Transient simulations, capturing most turbulence effects could be performed, yet is much more resource intensive.
- Flare tip may sustain defects or deformation during operations, which can introduce discrepancies between the actual operating performance and CFD model if it is based on a new flare tip
- Would require additional modelling to factor in precipitation (rain, snow) effects on combustion
- Liquid entrainment in flared gas stream affects the combustion, yet it requires more complex modelling and is less well understood, hence, most models are based on single-phase gas flow.

- Access to advanced computing capability to reduce the time taken to run models

- Limited validation data available comparing model outputs to physical observations

Go Deeper

- TUV
- DNV
- Argo

Case study

Assessing the impacts of wind speed on an offshore flare system

CFD modelling was commissioned to assess the and the methane Destruction & Removal Efficiencies (DRE) at different wind conditions in an offshore production facility. The 3D model of the multi-tip sonic nozzle tip was developed and then the flare combustion was modelled at different wind speeds to determine the effect of cross winds on both CE and DRE. The work can determine if safety concerns could arise (such as internal combustion) as well as operational issues such as the flame stabilizing on the windshield which can result in damage to the tip.

HP flow rate	LP flow rate	Wind speed	Combustion Efficiency	Destruction Efficiency
Nm^3/h	Nm^3/h	m/s	%	%
590	140	2.6	99.1	99.6
		8.5	98.4	99.2
		17.1	97.7	98.2

Key findings:

- Combustion Efficiency and Methane Destruction & Removal Efficiency both as the windspeed increases
- Higher purge rates achieve higher CE and DRE values, and hence might be required to reduce methane emissions under higher wind speed
- At low purge rate, there is a potential for flashback to the knockout drum was identified (due to oxygen ingress in the flare tip creating an explosive mixture) which could compromise platform safety

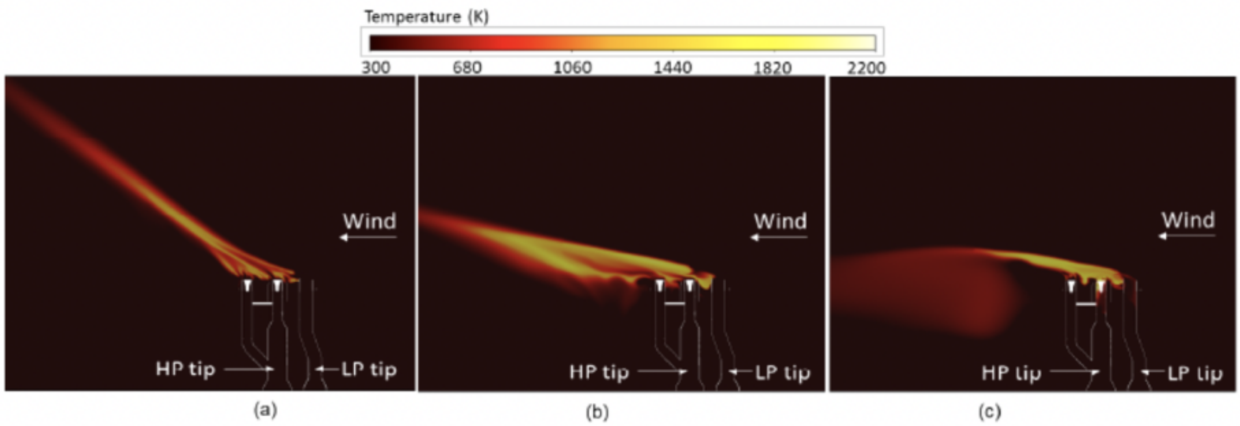


Figure 1. Flare combustion model at three different wind speeds: a) 2.6m/s b) 8.5m/s c) 17.1m/s

---

Do I understand the impacts of the environment on my flare?

Environmental Impacts: Crosswinds – Empirical data and observations

Environmental Impacts: Flare Tip Integrity Inspection # Manual Inspection

Environmental Impacts: Flare Tip Integrity Inspection # Drone deployed

METHANE FLARING TOOLKIT

© 2025