Hot Air Recirculation Simulation

By Martin Walters, Senior CFD Consultant, ANSYS UK Ltd.

Tel: +44 (0) 114 281 8888, Email: ukinfo@ansys.com

Background

LNG liquefaction processes rely on air coolers to provide the cooling duties. Plant performance is dependent on the temperature of the air supplied to both the air coolers and the gas turbines used to power the process being close to the ambient air temperature. With the congested nature of plant equipment around LNG facilities and the large volume of heat emitted on a typical site, there is a real threat of warm air recirculation on air coolers and gas turbine intakes. If this occurs then the performance of the plant can be significantly impacted.

Hot Air Recirculation studies done using Computational Fluid Dynamics simulation are an established tool used during the design and operation of LNG liquefaction facilities to assess this issue.

To achieve this, a 3D model of the entire site and surrounding terrain is built and airflow across the site under various wind conditions is simulated. The hot air plumes from all the equipment around the site can be monitored to understand whether this is pulled into the intakes of critical equipment.

ANSYS consultancy staff have over 10 years' experience of carrying out hot air recirculation studies on numerous LNG facilities around the world. In addition, they have experience of many other oil and gas simulation projects. As a consultancy that works on such projects, we work closely with our clients to advise them on how their design may respond to such issues and on ways to reduce hot air recirculation.

Such studies allow process engineers, designers and operators to understand the potential for warm air recirculation and to assess the effectiveness of mitigations and design changes that can be applied to reduce the likelihood of this occurring.

Design Factors

There are many factors that govern site design for LNG plant. It is frequently the case that consideration of hot air recirculation will come second to process requirements and practical aspects. There are several common problems that occur often in designs that can usually be avoided.

Placement of air coolers or intakes in the shadow of other equipment and blockages can cause recirculation to occur on these units under certain wind conditions. If an air cooler is in the wake of a building it is likely that recirculation will occur. It is therefore better to raise air coolers above surrounding equipment to ensure that ambient air is supplied to the units uninterrupted by upstream obstacles.

Grouping common types of sources of hot air together can help to reduce hot air recirculation by allowing plumes to combine, increasing their buoyancy, and allowing them to rise more quickly away from the site. Making sure that units with very different exhaust velocities are not in close proximity is also useful - for example, not mixing forced and induced draft air cooled heat exchangers in the same fan bank. Where these units are mixed and placed close together it is common to find the forced draft air cooler unit plumes are entrained by the intakes of the induced draft units. The relatively slower air exhausted from the forced draft air coolers is pulled into the induced units more easily. Where possible it is often best to replace forced draft units with induced draft units altogether.

More site wide considerations can include aligning the LNG trains and banks of coolers with the prevailing wind direction and ensuring adequate spacing between different sets of units. Gas turbine positioning and exhaust stack heights should also be given careful consideration.

Yamal LNG Case Study

On a recent project ANSYS looked at the proposed Yamal LNG facility. Yamgaz, a consortium between Technip and JGC, are performing the Engineering, Procurement and Construction (EPC) package for the design of the site to be located near Sabetta on the Yamal Peninsula in northern Siberia. The project will process gas from the South Tambey Field. The facility will consist of three LNG trains (5.5Mt/a each) with associated utilities, power generation and storage and export facilities.

Yamal LNG has significant challenges due to its remote location in one of the world's harshest environments. Yamgaz are keen to use the most advanced simulation technology to maximise their design and chose ANSYS consultancy services to carry out a hot air recirculation study.

This project posed particular challenges from a hot air recirculation perspective. The majority of the modules on the site have a high degree of cladding to protect them from the extreme weather conditions in this region. This dramatically increases the blockage represented by the modules. Heavily clad modules effectively act as bluff bodies that generate a large amount of turbulence in their wakes as the air flow separates around the modules. These turbulent wakes can be extremely problematic as they can allow any hot air plumes nearby to be pulled down to intake level as they mix in these regions. With more obstacles to the incoming airflow the paths taken for air to reach the intakes can be more convoluted. This can add to the problems seen across the site.

Many of the proposed mitigations from this work have now been incorporated into the design and from the simulation study we can show that this has been successful in significantly reducing the potential for hot air recirculation across the site. An example of this was the spacing between the LNG main piperacks and the modules. The original design would provide adequate distance between these units under normal circumstances, however, owing to the amount of cladding that was necessary it was found that if the spacing was increased then the airflow to the air coolers would be improved.

Analysing Data

Using more sophisticated methods of analysing the data from these models, it is possible to isolate individual hot air plumes from specific units of plant and calculate their impact on air intakes across the site. This ability allows process engineers to gain a detailed understanding of the exact causes of problems that would not be possible by any other means. We can apportion the sources of hot air recirculated on all units and map this data to build a detailed picture of how the entire site responds to changes in design and atmospheric conditions.

Based on the case study the image below shows an example where the hot air plumes rising from the simulated Yamal LNG plant are coloured by the units that the plumes originate from. From data such as this we can determine that the contribution to hot air entrained by a specific unit, for example a gas turbine intake is 20% from unit X and 80% from unit Y. This allows us to target design changes specifically to where they are needed to fix the root cause. The flow field around an LNG plant can be extremely complex and as the plumes from many units combine it can be difficult to define how some problems occur. These methods give us a way to understand these problems and then propose changes to fix them.

This data can also be used by plant operators to understand how the plant would respond to changes in the wind strength and direction and can allow them to modify operating conditions to optimise the output throughout a given time frame.

For example, most LNG sites operate in coastal regions where it is common to see shifts in wind patterns across a daily cycle. Having a method that allows the impact of hot air recirculation to be mapped against these changes can provide a tool that site operators can use to troubleshoot or guide production and improve efficiency.

Conclusions

It is important in reducing hot air recirculation to understand the impact of a range of wind directions and speeds on the performance of an LNG plant design. Different problems present themselves under different wind conditions and it is important to fully explore these issues. Computational Fluid Dynamics is an established tool in exploring how a design will perform in the real world and allows designers and engineers to examine different ways of improving designs.

Advances in simulation systems mean it is now common practice to run many models parametrically across large distributed computational resources to understand the full design envelope engineers are working within. By considering such issues at an early stage in plant design, problems can be identified and dealt with before major work is required to rectify issues. ANSYS develops, supports, markets and provides consultancy services for world leading engineering simulation software. Simulation Driven Product Development is at the core of the work that ANSYS does.

Yamal LNG Case Study Images

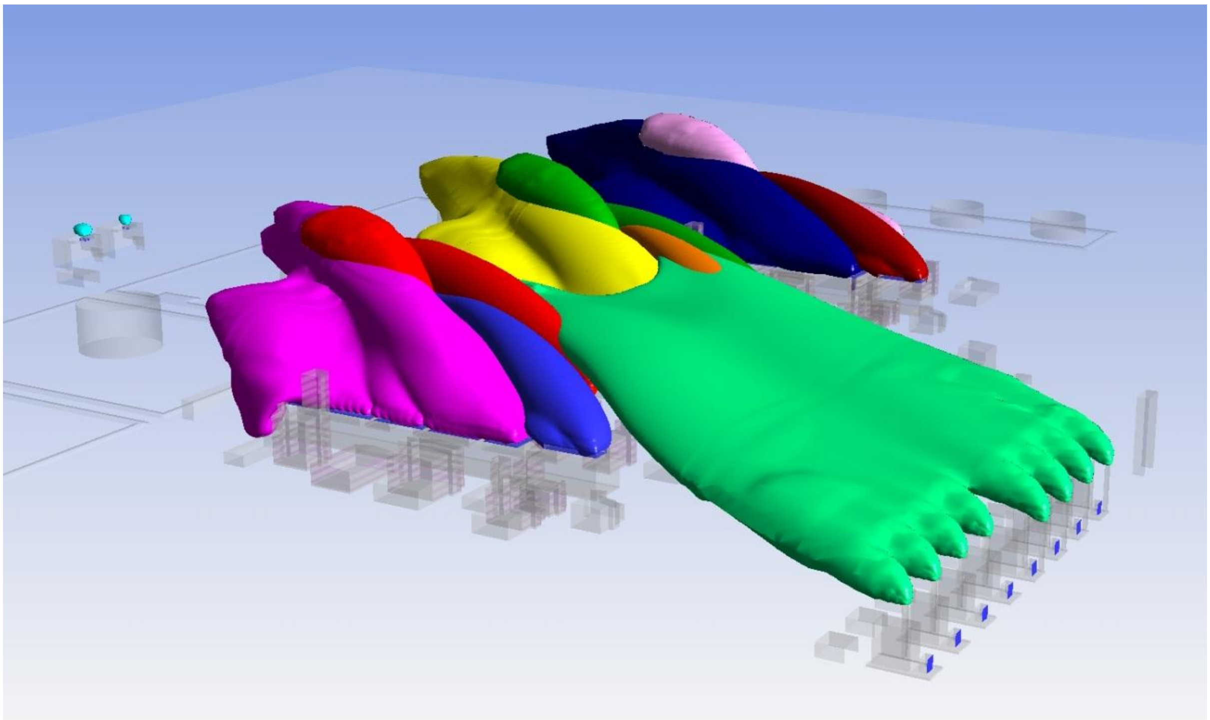

Image 1 - Hot air plume over the simulated Yamal LNG site coloured by the originating unit for the plumes.

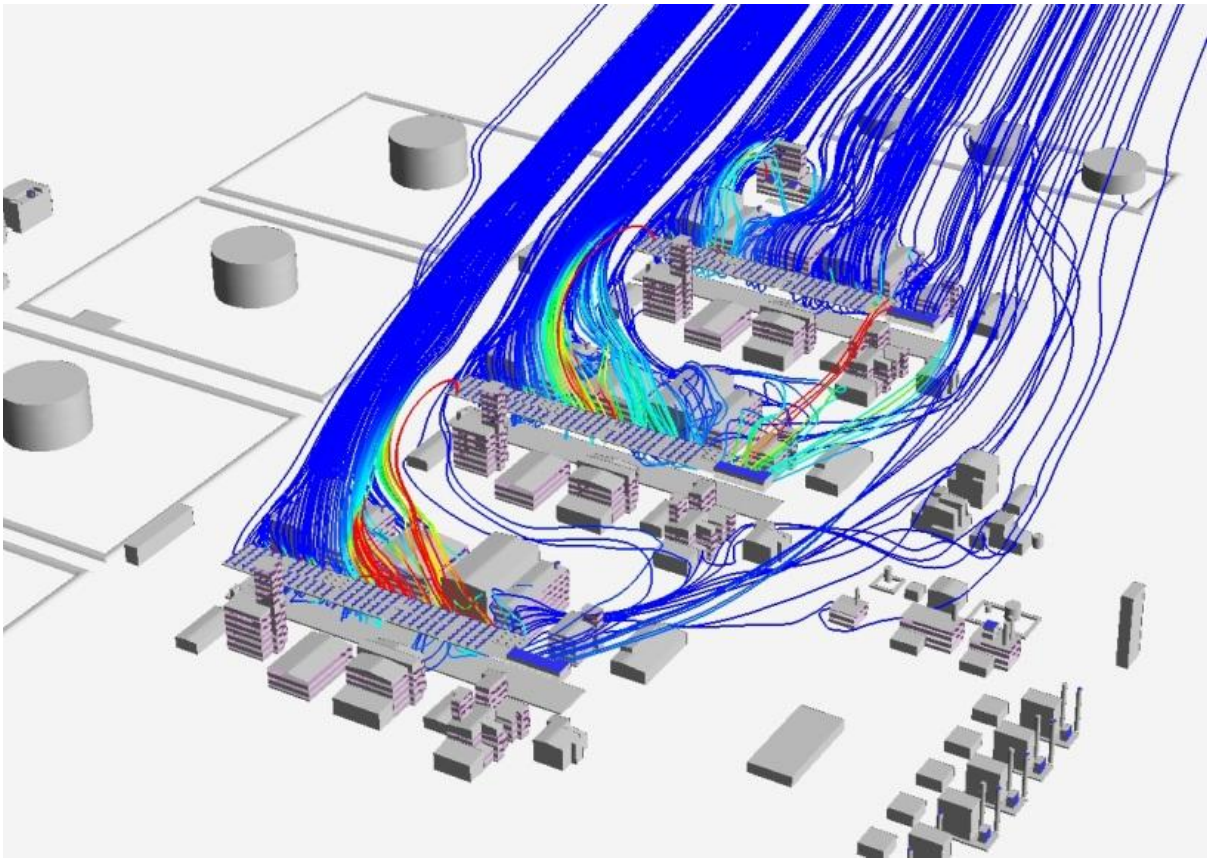

Image 2 - Pathlines tracked to the air cooler intakes coloured by temperature.