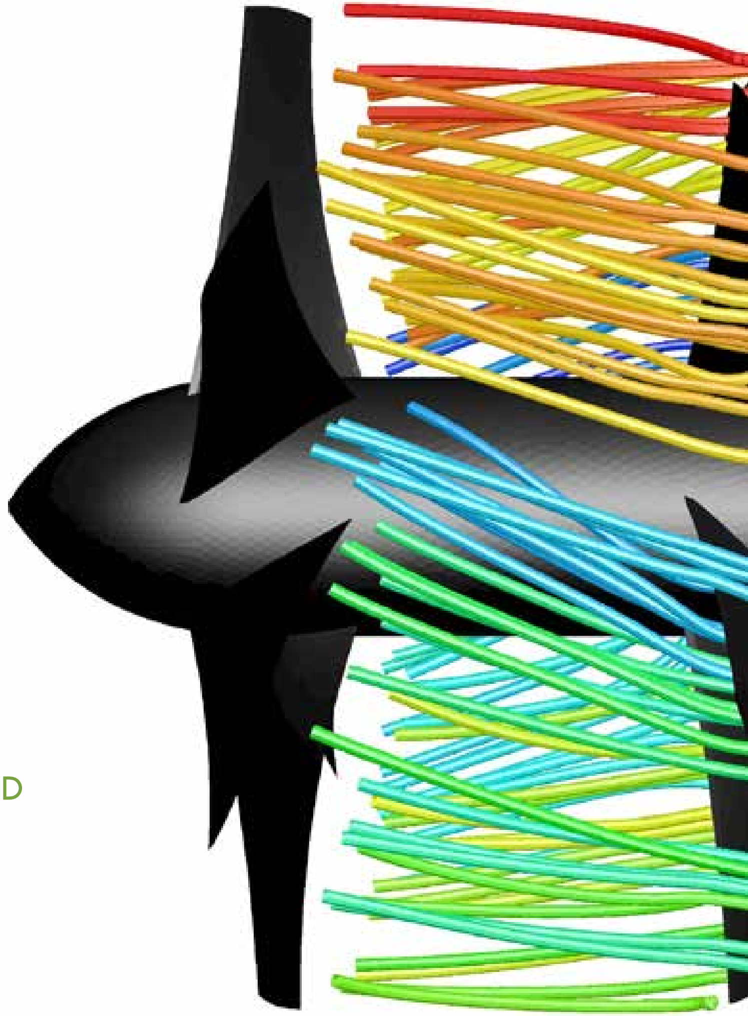


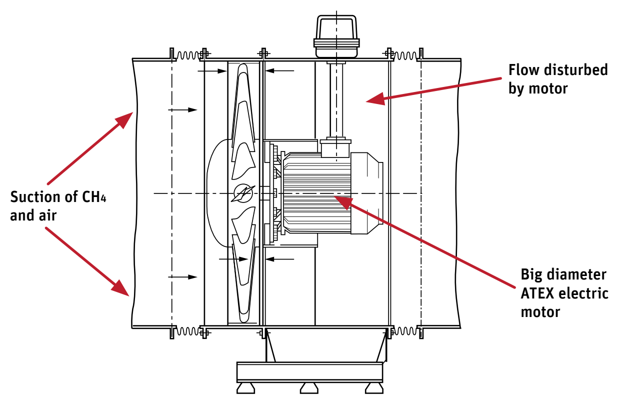
COLD CASH

A FAN DESIGNED WITH MULTIPHYSICS SIMULATION OFFERS A POTENTIAL OF 1 BILLION EUROS IN LIFETIME SAVINGS FOR ALL OF THE LNG PLANTS OPERATED BY A LARGE GLOBAL PRODUCER.

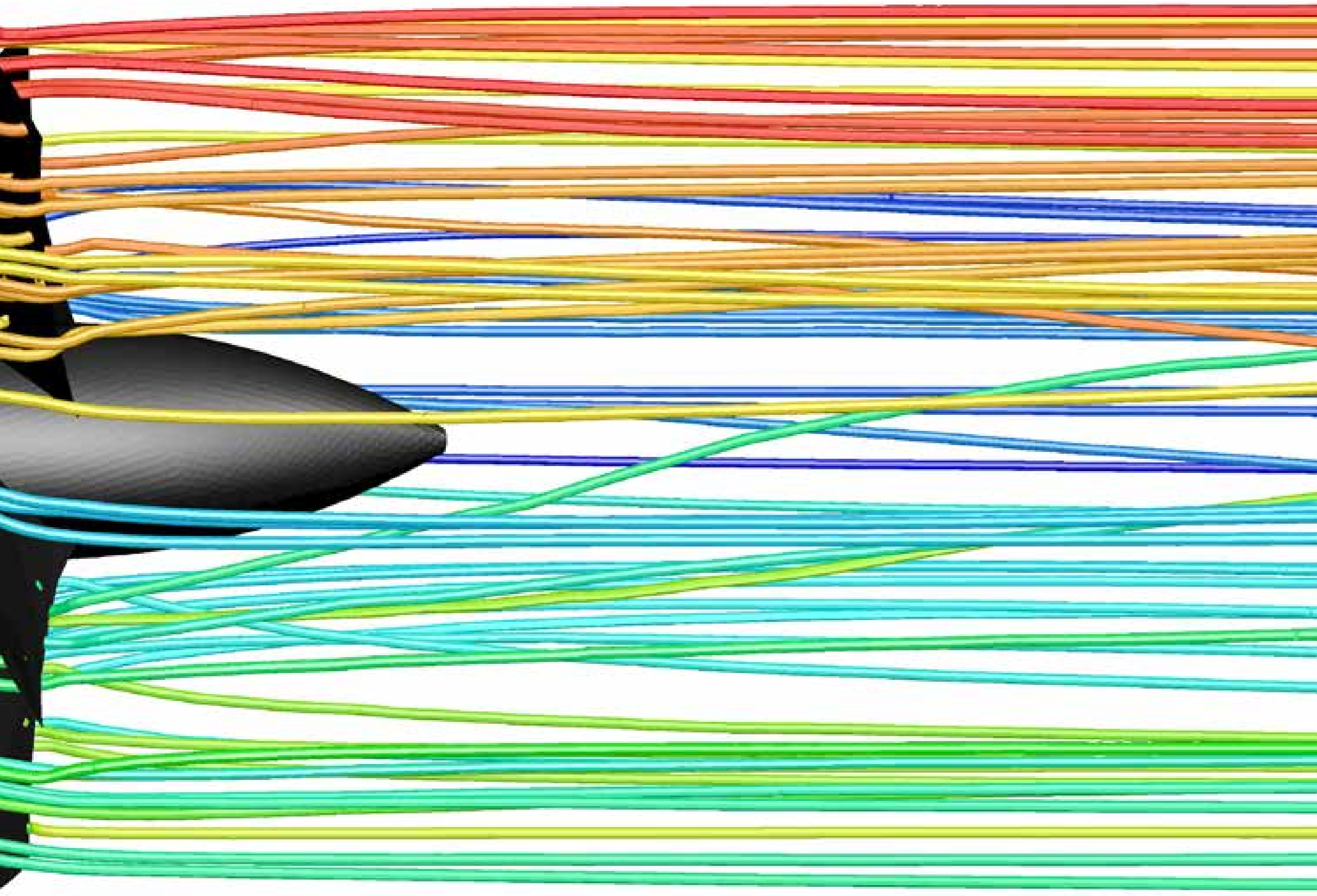
By **David Ohayon**,
Chief Technology Officer,
Nova Simulations,
Marseilles, France

The flammable nature of liquefied natural gas (LNG) creates the need for strict safety procedures while it is condensed by refrigeration from its gaseous state. LNG plants operating in extremely cold climates, such as in Russia and Norway, must use heated enclosures to provide comfortable working conditions for plant operators. Safety regulations require that these enclosures be equipped with ventilation capable of completely exchanging the air inside the enclosure with outside air (air change) 12 times per hour in normal operation and 18 times per hour in emergency conditions, when elevated levels of methane, the primary flammable component of natural gas, are detected.

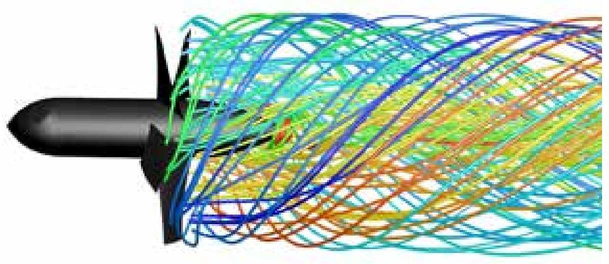
Existing gas plants typically address this ventilation challenge by using 20 off-the-shelf fans to ventilate a process room 50 meters long by 30 meters wide by 20 meters high. The large motors that are directly connected to each fan create an obstruction that reduces aerodynamic efficiency. Placing the motors inside ventilation ducts also puts them in contact with the airflow and creates an ignition risk. The fans consume about 311 kilowatts (kW) during normal operation, resulting in high electrical costs in the remote, off-the-power-grid areas where the plants are typically located. Furthermore, because each fan has its own motor, it is often necessary to shut down the entire condensation process when maintenance is required on just one of the fan motors because the remaining fans are not capable of meeting ventilation requirements.



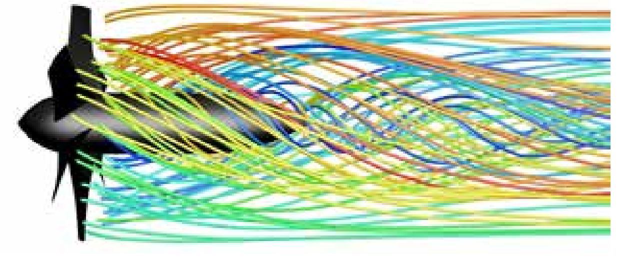
^ Current fan design



^ Flow streamlines generated by both rotors show how tangential flow is canceled out.



^ Flow streamlines generated by downstream rotor in isolation show high tangential flow.



^ Flow streamlines generated by upstream rotor in isolation show high tangential flow.

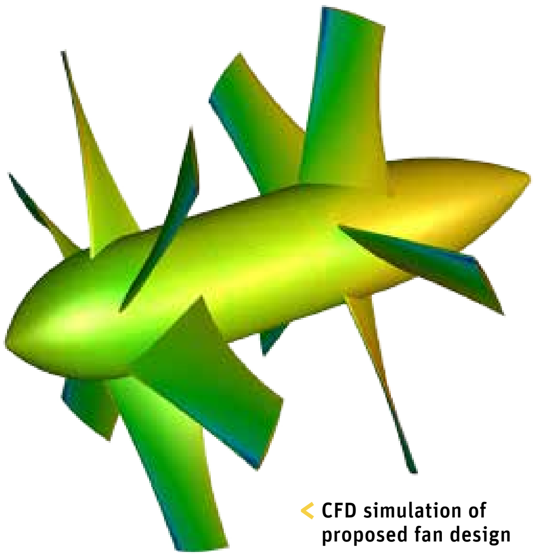
DEALING WITH TANGENTIAL FLOW

Nova Simulations proposed the design of a custom fan to one of the world's major oil and gas companies. The custom fan would drastically reduce operating costs by optimizing aerodynamic efficiency for this application, and by placing the motor outside the duct. This external motor placement would require the addition of a gearbox, which would create some power losses. But Nova Simulations engineers knew that a custom fan design optimized specifically for this application could more than make up for these losses. The key would be to maximize the amount of air flowing parallel to the axis of the fan (axial flow).

In a regular fan, the spinning motion of the blades generates two types of flow: the preferred axial flow and the less efficient tangential flow — tangential to the circumference of the fan. The energy that goes into producing tangential flow is not available for generating axial flow. Static guide vanes

Modeling Fan and Blower Systems
ansys.com/fan-and-blower

"The savings across all of the company's LNG plants are estimated at 1 billion euros once all of the plants are equipped with the new fan."



CFD simulation of proposed fan design

are often used to convert tangential flow into axial flow, but this conversion process also generates considerable losses.

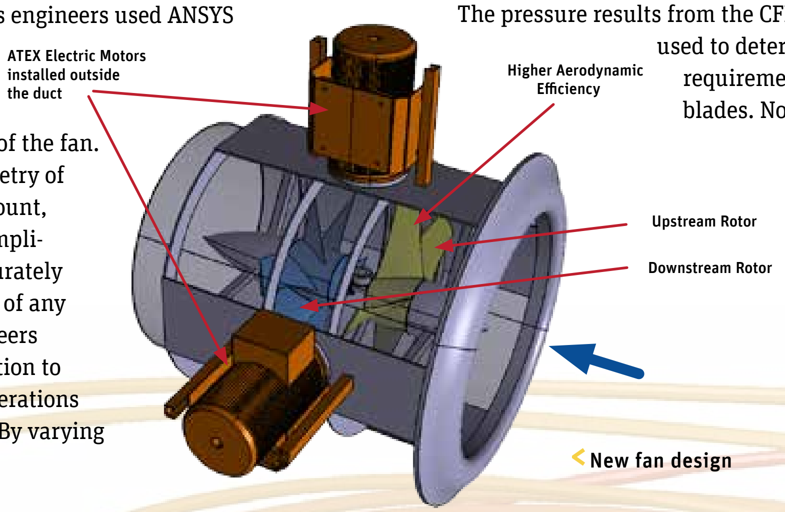
Rather than simply designing the fan to minimize tangential flow, Nova Simulations engineers had the idea of using tangential flow to their advantage by positioning two rotors operating in parallel but with opposite directions of rotation within each fan. They knew that each rotor by itself would generate substantial tangential flow, but the downstream rotor could be designed so its tangential flow would cancel out that produced by the upstream rotor, resulting in axial flow conditions. Working in tandem, the two rotors would maximize the desired axial flow through the ventilation duct — something that a regular fan with one set of rotors blades could not accomplish.

Next, they faced the challenge of validating this idea and optimizing the performance of the fan to generate the highest possible airflow for the lowest possible power consumption. In the past, engineers used airfoil design methods to estimate the performance of various design concepts. These methods required many simplifying assumptions, so building and testing a considerable number of prototypes was needed to produce a working design. But in this case the cost of building even a single prototype was so high that funding could be obtained to build only one prototype — and obtaining this funding required near certainty of success.

ITERATING TO AN OPTIMIZED DESIGN

To optimize their chances of getting the design right the first time, Nova Simulations engineers used ANSYS Fluent computational fluid dynamics (CFD) software to produce a complete virtual prototype of the fan. CFD takes the full 3-D geometry of the blade and duct into account, eliminating the need for simplifying assumptions and accurately predicting the performance of any proposed fan design. Engineers used ANSYS design exploration to perform the many design iterations required for the CFD study. By varying the number of blades, the blade profile, the pitch of each rotor, the distance between rotors, and the duct diameter, they were able to determine the most efficient design. First they optimized the performance of each individual rotor, and later, the combined performance of the two rotors. The result was that they optimized the aerodynamics of the two rotors synergistically so that each operates separately as efficiently as possible, while the two rotors combine to convert the tangential flow produced by both fans into straight axial streamlines. Simulation allowed them to virtually test many fan designs by automatically exploring many parameters and avoided the need for multiple prototypes.

Carbon fiber fan blades were used to deliver the high strength-to-weight ratio required for this application. The pressure results from the CFD simulation were used to determine the strength requirements for the rotor blades. Nova Simulations



New fan design

"Testing showed that the *prototype performed* nearly exactly as predicted by *simulation*."

engineers used ANSYS Composite PrepPost to model the complex composite structure, including the number of layers and the shape, thickness and orientation of each layer. ANSYS Composite PrepPost predicted the ultimate strength and progressive damage over time due to delamination, cracking, pull-out and other destructive mechanisms. Engineers optimized the design of the blade to meet the requirements of the application at a minimum weight.

TWO LAYERS OF REDUNDANCY

Engineers worked with a power transmission company to design a gearbox that provides 95 percent efficiency. The use of a gearbox makes it possible to move the motor out of the duct, which helps to improve the efficiency of the fans; it also increases safety by removing the motor from the airflow, which contains combustible gases. The two rotors operate at the same speed to maximize the efficiency of the gearbox. Each fan is coupled to two motors to provide intra-fan redundancy, enabling the plant to continue operating even when a motor is down for maintenance. In addition, the four fans needed for each process room are coupled to provide an additional layer of redundancy: Even if both motors coupled to a particular fan are down, the fan continues to be driven by other motors.

^ Prototype of new fan

The simulation predicted that the new fan design operating at 900 rpm would provide a flow rate of 24 cubic meters per second while consuming 12 kW. Increasing the flow rate to 40 cubic meters per second at 1,500 rpm while consuming 50 kW yields a maximum efficiency of 72 percent, nearly double the efficiency of the existing fans. Based on the simulation results, the oil and gas company funded the construction of a prototype and contracted with an independent ISO 5801 laboratory to test its performance. Testing showed that the prototype performed nearly exactly as predicted by simulation. It delivered a flow rate of 24 cubic meters per second at 900 rpm while consuming 10 kW, and 40 cubic meters per second at 1,500 rpm while consuming 46 kW, for a maximum efficiency of 74 percent. The fans will normally operate at 900 rpm to provide 12 air changes per hour, but in the event of an emergency they can be quickly increased to 1,500 rpm to deliver 20 air changes per hour — two air changes per hour higher than the current design. The increased number of air changes means that the process room can be more quickly cleared of methane in the event of a leak.

Nova Simulations engineers are now working on preparations to manufacture the new fan to equip the oil and gas company's LNG plants, and also for other possible applications where ventilation is critical, such as tunnels, underground parking garages and underground mines. Based on prototype testing, four of the new fans should be able to safely ventilate an LNG process room. The 20 fans used in current process rooms draw 311 kW, while four of the new fans draw only 184 kW, including the draw from the new gearbox. In a typical LNG plant, this reduction in power consumption alone will generate annual savings of 1.5 million euros per year. This approach also makes it possible for the process plant to continue operating under normal conditions as long as four of the eight total motors are available. Thus the plant should never need to be shut down again for fan maintenance, as frequently occurs using the current ventilation system. The reduction in operating costs and downtime provided by the new fan is expected to reduce the cost of operating a single LNG plant by 100 million euros over its lifetime. The lifetime savings across all of the company's LNG plants are estimated at 1 billion euros once all of the plants are equipped with the new fans.