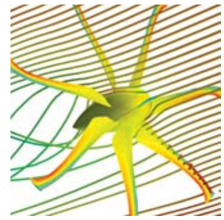
CD-adapco

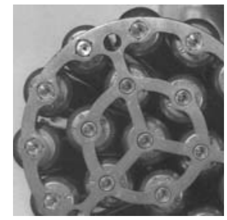
energy

SPECIAL REPORT

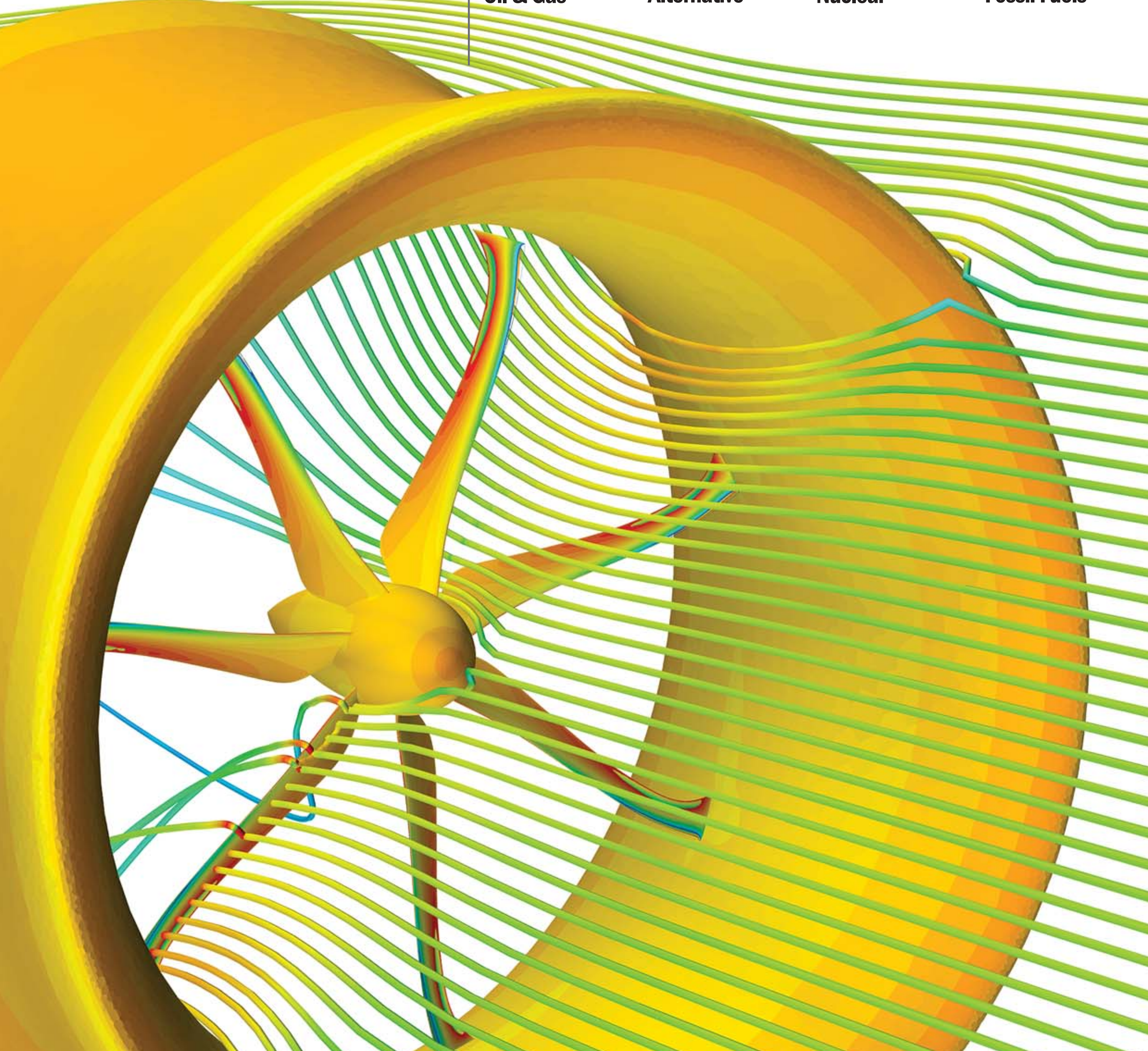
Energy Sectors


Oil & Gas


Alternative

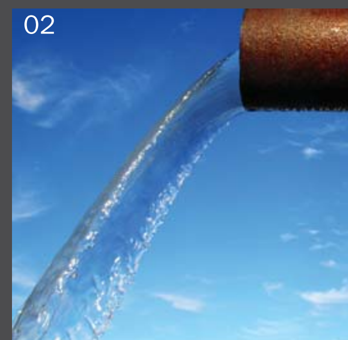

Nuclear

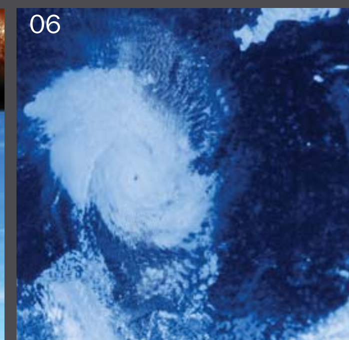

Fossil Fuels

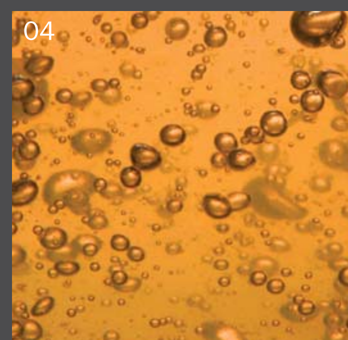


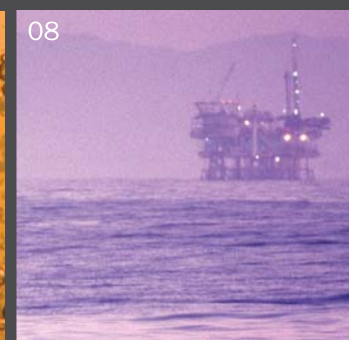
Contents

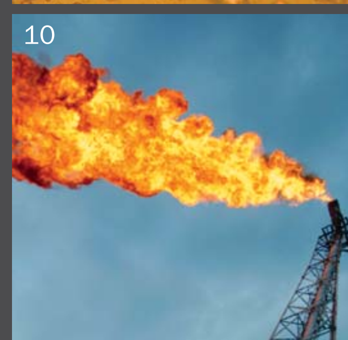
Cover Image Credit - Wingring

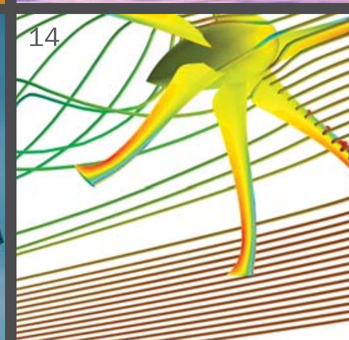


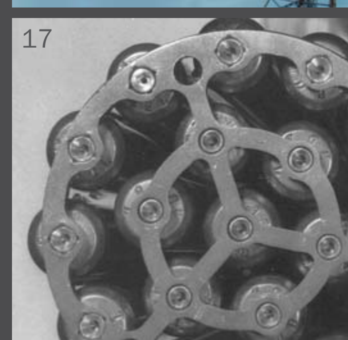


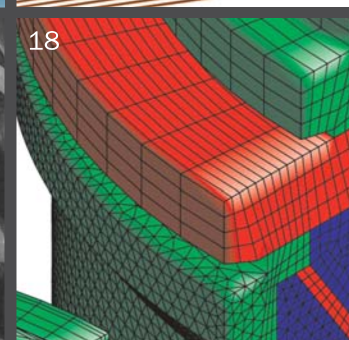


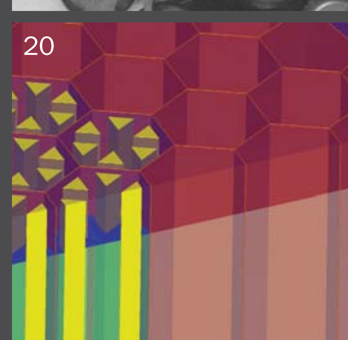


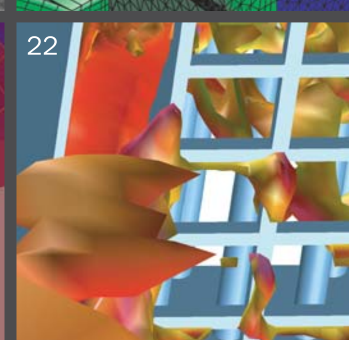




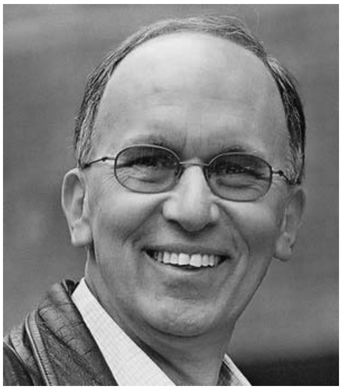








Printed on Recycled Paper
Reduce your Carbon Footprint today ***http://www.carbonfootprint.com/***



Energy Issues

Introduction by Dr Dennis Nagy

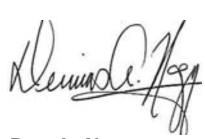
> "CAE technology has been called upon to demonstrate the viability, improve the safety and increase the efficiency of both renewable and non-renewable energy supplies."

Just a year or two ago, if you'd asked me to describe my "carbon-footprint" I'd have uncomfortably murmured something about not owning any Kevlar shoes.
Today, it is difficult to flick through the pages of any newspaper, or the news channels on cable, without encountering a story or two about "energy security" and "global warming".

These are exciting times to be an engineer. As our governments face the daunting dual challenge of maintaining access to secure and affordable energy supplies while mitigating the effects of climate change, they will increasingly rely on high-quality engineering, and particularly Computer Aided Engineering, to solve the difficult technological problems at the heart of the sustainability issue.

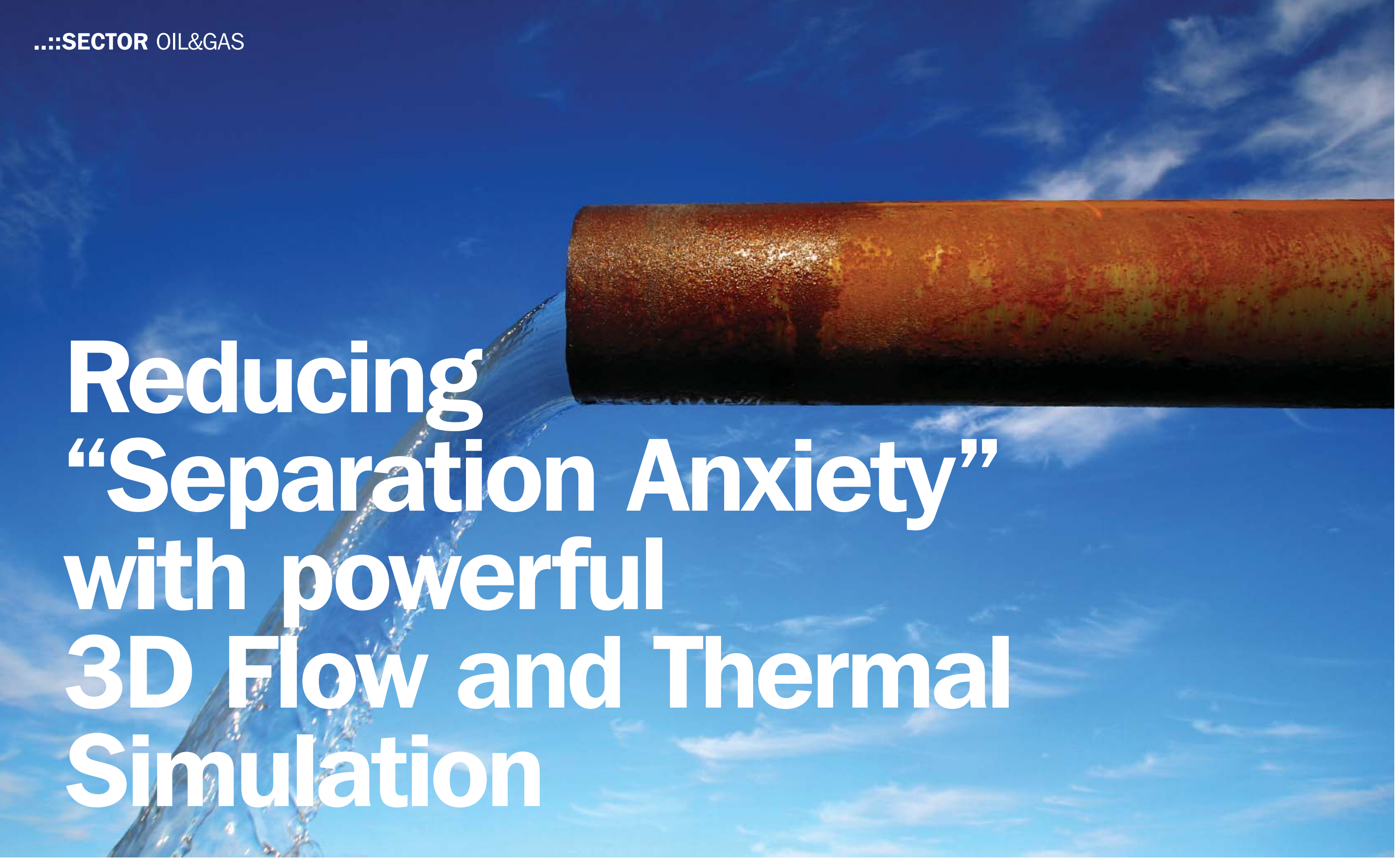
This magazine contains a small selection of the energy related projects that CD-adapco have been involved with over the last few years. There are many others, but we have tried to concentrate on diversity of application, and to illustrate projects in which CAE technology has been called upon to demonstrate the viability, improve the safety and increase the efficiency of both renewable and non-renewable energy supplies.

Enjoy your read!



Dennis Nagy
Director, Energy Sector
CD-adapco

EMAIL **oilandgas@uk.cd-adapco.com**



Reducing “Separation Anxiety” with powerful 3D Flow and Thermal Simulation

▲ Fig:01
The largest single product of the global oil and gas industry is neither oil nor gas, but water: produced at a rate of approximately 3 barrels to every barrel of oil, in 1999 the oil and gas industry was responsible for extracting 77 billion barrels of water.

Separating reservoir fluids into streams of oil, water and gas is a major concern to the global oil and gas industry, and has been almost since its inception. Historically, the major driver for effective separation was economics – extracting the maximum amount of usable hydrocarbon from the reservoir fluids. However, environmental concerns now mean that oil and gas producers are also increasingly bound by legislation that strictly controls the levels of pollution in discharged produced water – this combination is the growing ‘separation anxiety’.

Designing separators to meet these demands remains a significant engineering challenge. Critically, separators do not come in a ‘one-size-fits-all’ specification. They must be carefully chosen to not only account for the unique composition of fluids produced from a given reservoir, but also for the likely changes in composition that will occur over the lifetime of the well. Separator technology that is effective in early production might become less effective, or even fail, as the well matures or because of some temporary and unexpected change in the reservoir fluids. The increasing cost of platform real estate also means that there is also constant demand either to reduce the size of offshore separators or else to move them off the platform altogether, turning to newly developed subsea separation technologies.

▲ Fig:02
Environmental concerns mean that oil and gas producers are increasingly bound by legislation that strictly controls the level of pollution in discharged or reinjected water.

Whatever type of separation technology is employed, or retrofits and adjustments made, the cost of getting it wrong can be immense. The production capacity of any facility depends, to an extent, on the effectiveness of its separation process. Although most facilities employ at least two independent separation trains, diverting production while diagnostic analysis is performed on a poorly performing separator inevitably results in a reduction of throughput. With oil prices topping $60/b, even a 5% drop in production from a 50,000 b/d installation will cost in excess of $150,000/d. Worse, if significant problems occur in the separation process, the cause of which cannot easily be diagnosed, the only alternative is to stop production altogether, or ship the reservoir fluids for processing at another facility.

Separator flow simulation

CFD has been applied at every stage of the oil, gas and petrochemical production process and can provide insight into any problem involving fluid flow (whether liquid or gas or a mixture of both) or structural components that are influenced by flow, and thus is particularly suitable for separator analysis. CFD simulation can help both in the design of new separator technology and in determining the range of operating conditions under which existing technology might be successfully deployed.

Data from CFD calculations can also be used to assist other types of analysis, for example, the forces acting on the separator internals can be calculated, either directly within the CFD code or via an external stress-analysis software package. In extreme cases, where fluid forces cause large deflections of components, the CFD simulation can be coupled directly with the stress simulation tool and both stress and fluid simulations can be performed simultaneously, each simulation feeding new boundary conditions to the other.

Case study 1 – sloshing in a free water knockout drum

The free water knockout (FWKO) drum is perhaps the crudest form of separator. FWKO drums work on a gravitational principle, relying on the fact that oil has a lower specific gravity than water and, if allowed to settle, will float to the top, forming a layer than can easily be skimmed off and extracted. Water is extracted through a valve at the bottom of the tank, while in the example shown in Figure 4, the oil trickles over a weir plate at the left hand side of the drum into the oil stream outflow.

Under normal operating conditions, this system provides a very effective means of preliminary. However, when deployed aboard an FPSO (floating production, storage and offloading vessel), there is a risk of the tank being disturbed by the motion of a passing wave, causing sloshing within the tank and leading to significant amounts of water passing over the weir plate or oil-emulsion contaminating the water outtake and possibly damaging downstream separation equipment.

Figure 5 shows a large sloshing motion that has developed in the vessel due the disturbing motion of a passing wave (as predicted by the CFD calculation). The simulation predicts that, under these conditions, a significant amount of water will slosh over the weir plate into the oil outflow.

In order to prevent sloshing, separator manufacturers typically insert a series of permeable vertical baffles into the tank, which act to damp the motion of the fluid within the vessel, preventing large-scale sloshing motions from developing. CFD simulation allows separator designers to make informed decisions early in the design process, before even the first prototypes are available, allowing them to answer questions such as ‘How many baffles do I need?’, ‘How do the baffles influence separator performance?’, ‘What sort of forces are acting on the baffles and on the vessel walls?, and ‘Under what range of wave conditions can the separator safely operate?’

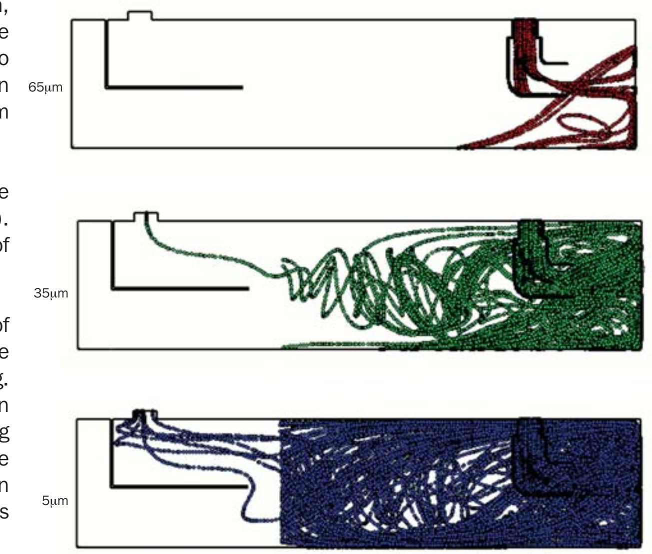
Case study 2 – redesign of a gas phase separator

The aim of a gas phase separator is to remove small particles of hydrocarbon condensate (and other well-fluids) from a stream of natural gas. To be effective, the separator needs to be able to remove the wide variety of droplet sizes transported in a typical gas stream, from large visible droplets of hydrocarbon, to individual mist particles measuring just a few microns in diameter.

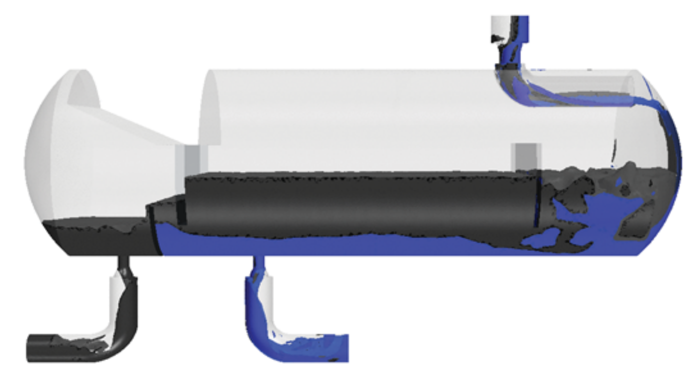
Exactly which fate each particle eventually meets depends largely on its size, but for the separator to work effectively, all but the smallest particles should be caught by one of the first three mechanisms. The vane pack demister acts as the final line of defence, removing a fine mist of droplets with diameters of around 10 mm or less. For effective operation, it is critical that the demister is not blocked by much larger oil particles, which, given enough time, should fall onto the surface of the liquid layer due to the influence of gravity. The separator therefore needs to be long enough, upstream of the demister, to ensure that the gas flow has sufficient residence time to allow these larger particles to fall into the liquid layer of hydrocarbon at the bottom of the tank.

The simulation results reported in Figure 3 show that the majority of 65 mm and 35 mm particles hit either the vessel wall or the liquid surface a short distance after entering the separator. By contrast, many of the small 5 mm particles are carried with the gas flow until it passes through the vane pack, at which stage they are removed. The simulation predicted an overall trapping efficiency of 90%, with almost a 100% of particles of diameter 40 mm or higher removed by the separator. The separator manufacturers were able to significantly reduce the length of the separator, after establishing with the aid of further simulation, that since larger particles were hitting the walls of the liquid surface soon after entering the separator, much of the length upstream of the demister was unnecessary.■

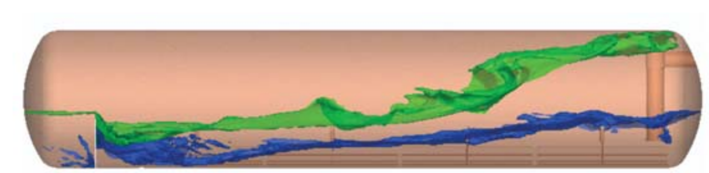
> **“CFD simulation allows separator designers to make informed decisions early in the design process, before even the first prototypes are available.”**



▲ Fig:03
CFD simulation of three-phase separator, showing path of various particle sizes.



▲ Fig:04
Three-phase separator - courtesy of Saudi Aramco.



▲ Fig:05
CFD simulation of FWKO drum disturbed by a passing wave while aboard FPSO.

ⓘ MORE INFORMATION http://www.cd-adapco.com/

Forever Blowing Bubbles

Revolutionary MBF™ Separator Design with CFD

Stephen Ferguson in conversation with GLR Solutions Ltd. CEO Douglas Lee

Faced with increased water-cuts from maturing wells and a combination of environmental legislation and operational demands, Oil and Gas operators are being forced to review their separation processes. Not only must separation trains now handle a larger throughput of well fluids (due to the increased water content), but the water that they deliver for disposal or re-injection must also be cleaner than ever before.

In this article, we explore how Canadian company GLR Solutions has applied advanced Computational Fluid Dynamic simulation in the design and implementation of their Micro Bubble Flotation technology which has become recognized in the industry as one of the highest performance methods for treatment of produced water.

At the heart of most Oil and Gas separation processes is the API Skim Tank. Slow and reliable, the skim tank depends on the difference in specific gravity between oil and water, lighter oil eventually floating to the top of the denser water content, where it can easily be skimmed off. Although ubiquitous, the standard skim tank suffers from the long retention times required to perform effective separation, which can be useful when buffering out the effects of upstream spikes in production, but ultimately is inefficient when separating small amounts of oil from large amounts of produced water. API gravity tanks are also relatively inefficient when dealing with heavy oils or emulsions.

To counter these shortcomings, many modifications to the traditional API vessel have been attempted, aimed at increasing the separation efficiency of the API vessels (wherever possible using existing separation equipment), often resulting in the introduction of internal structures or distribution nozzles, which are intended to encourage the coalescence of oil droplets within the tank. A more novel, and generally more effective approach, involves flooding the separation vessel with bubbles of gas, which adhere to similarly sized oil particles and float them to the surface of the tank. This approach, known as Induced Gas Flotation, or IGF, usually requires the partitioning of the vessel into various chambers so that the bubbles can act on successfully cleaner batches of produced water. The downside of IGF is the requirement for additional process vessels, limited efficiency due to bubble size limitations of conventional equipment, and that the relatively low retention times do not provide an adequate buffer against upstream disturbance (process upset).

GLR Solutions have devised, with the aid of extensive CFD simulation, an improved gas flotation technology that uses micro bubbles of gas (~10-50 microns in diameter) to assist separation. Douglas Lee, President and CEO of GLR Solutions, explains: *"A bubble of gas of a given size will attach itself to a similar sized oil droplet and encourage it to float to the surface where the oil coalesces, collects and is skimmed off."*

According to Lee, GLR Solution's Micro Bubble Flotation technology (MBF™) substantially enhances the separation process over conventional skim tanks and IGF and gives much improved separation results: *"The benefits of this are increased revenue from recovered oil, fewer problems in reinjection of the water, elimination or fewer chemicals where chemicals are used and the ability to separate oil even in an emulsified state with considerable savings in the reduction of chemicals."*

Fewer problems in reinjection include less plugging of the formation due to oil and solids contamination and a reduced need for expensive well workovers to remedy plugging. A reduced level of oil and solids in the outlet from the skim tank also offers much-reduced loading on final, oil removing filters when in use. One of the most unique applications of MBF™ is directly within API tanks which typically incorporate multiple stages to enhance removal of the oil, while eliminating of oil short-circuiting. The application of gas flotation within API tanks provides increased retention time, relative to an IGF separator, which would buffer any upsets in oil concentration or flow rates produced in upstream operations.

Case Study – MBF™ Separation for ENI Dacion

GLR Solutions have invested heavily in CFD simulation, both in the initial development of the MBF™ system, and while planning large-scale commercial installations, including the recent installation of an API tank based flotation system in Eastern Venezuela.

ENI Dacion BV, is a Venezuala based joint venture of ENI S.p.A. a worldwide oil producer and oilfield operator, owns and operates numerous facilities for the treatment of oil and water. GLR Solutions were engaged to provide the most appropriate separation technology for the GED-10 Station in the Dacion field. The GED-10 Station is one of ENI's smaller facilities; when it was originally designed and built, water cuts in the produced oil were typically very low, with anticipated flow rates of 5,000 – 10,000 bwpd. However, as is typical for reservoirs with active aquifers, water cuts have significantly increased in recent years (the station currently operates at 6,000 bopd, 15,000 bwpd), and it is expected that the facility will eventually need to handle flows of up to 25,000 bwpd. This has meant the majority of the equipment now has insufficient capacity to be used in the way it was originally designed.

During previous development in the Dacion field, expensive water treatment systems had been installed, based on de-sanding and de-oiling hydrocyclones and nutshell filter technology. Despite the high capital and operational costs of these (relatively complex) solutions, the level of treatment efficiency delivered by these systems was insufficient. ENI decided to evaluate other technologies for the selection of the most appropriate system for GED-10 in order to meet the treatment capacity and water quality specifications. ENI established a set of selection criterion in order to determine which separation technology was most appropriate, based on: ease of operation and maintenance; cost of operation and maintenance; capital cost; performance; and flexibility (the ability to handle a large range of inlet flow fluctuations, both in total flow, and oil and solid content).

After internal evaluations within ENI, the multi-chamber API tank configuration of MBF™ was selected, allowing GLR Solutions to set about designing the optimal configuration for the GED-10 field. Aware that there would be little opportunity for post-installation modification (without disrupting oil revenue from the station), GLR Solutions decided to undertake detailed design analysis using CD-adapco's STAR-CD CFD package: *"A major factor which aided in the determination of an optimal tank design was Computational Fluid Dynamics (CFD) modeling," says Doug Lee. "CFD simulation allows us to simulate fluids in a variety of compositions as they flow through the complex three-dimensional structure of the separator."*

An optimal design, as determined by GLR Solutions, had to include a number of attributes. As the design was to be incorporated into an existing tank at GED-10 Station, and it was known that flow rates would be increasing, it was important to make use of the available volume of the tank: *"For heavy oil applications, such as GED-10, it is essential to ensure that there is sufficient contact between oil-droplets and microbubbles throughout the tank,"* said Lee. *"Using CFD modeling we could not only visualize the flow patterns within the separator, but we could also track the progress of individual microbubbles. This allowed us to conclusively demonstrate that sufficient mixing occurred between the microbubble and produced water streams and that adhesion would readily occur in the designated region of the tank."*

As well as ensuring sufficient contact between bubbles and oil in the produced water, it was also necessary to ensure that, after contact with the oil, the bubbles were transported directly to the liquid surface, and not through the outflow of the separator where they might cause problems for downstream processes: *"CFD modeling allows us to track bubble particles through the tank. We can therefore predict whether bubbles travel to the lower portions of the tank and, if so, how we can prevent them doing so. Current chambered API tank designs have been modeled and the models field verified to confirme that bubbles will not exit the last chamber with the clean water."*

GLR Solutions Ltd. is a manufacturer and supplier of innovative technologies and services for the treatment of produced water. Notably we have pioneered new applications of Micro Bubble Flotation, Nutshell Filtration and Fluid Dynamic Modelling. GLR Solutions has distribution and service representatives Worldwide to assist you with projects of any size.

www.glrsolutions.com

"With the aid of CFD we were able to test a variety of internal configurations over a wide variety of operational conditions, that finally led to the optimal design that we proposed to ENI, and that was eventually implemented at the GED-10 facility. One of the benefits of CFD is that, using post-processing, we were able to effectively communicate to our client exactly how the separator would perform in action, using easy to understand graphics, rather than having to deal exclusively in abstract engineering descriptions."

Having determined the details of the optimal basic design (which includes two tanks – as shown), GLR Solutions then deployed further CFD simulation in order to refine other aspects of the separators performance, closely examining flow patterns within the tanks for additional benefits within the system: *"GED-10 had identified a major concern with solids in their water treatment system, to account for this we tried to incorporate internal modifications that would not only prevent solids from hindering performance, but might also aid in the removal of solids from outlet water,"* says Lee. Based on the CFD results, GLR incorporated a solids dropout area within the water weir of the first chamber. The CFD modeling predicted that upon entry of the produced water to the weir, the majority of the water would flow over the weir into the chamber, while a smaller volume of water containing the heavier solids would drop out and be directed to the bottom of the tank.

An additional benefit that was incorporated into the multi-chamber design was positioning and sizing of nozzles and weir shapes and sizes to allow for easy hydraulic skimming of oil into the oil trough. As seen by the CFD graphics (Figure 8), hydraulic patterns at the surface are such that oil collected on the surface readily flows into the trough, from all areas of the chamber.

Operational Results

Modifications of the existing tank 30-T-04 began in mid February of 2005. After cleaning and preparing the tank, nozzles were added and internals were welded as designed by GLR Solutions. The microbubble system was started up in May 2005, and favourable results for water quality were observed within days of start-up.

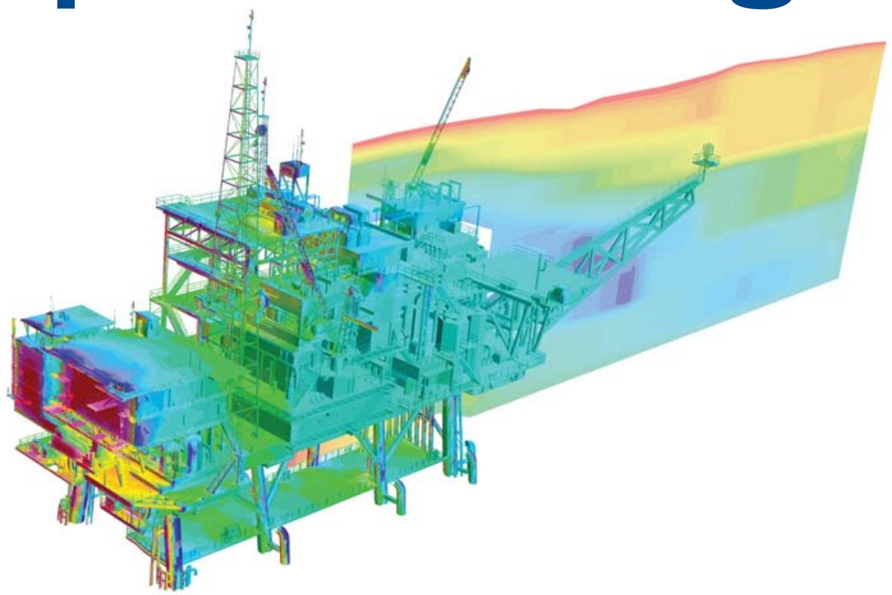
During normal operations over the spring/summer of 2005, the skim tank (30-T-04) receives approximately 15,000 bpd of produced water at oil concentrations ranging from 100-600 ppm. The quality of the clean water exiting the tank is consistently within 2-21 ppm during normal operations. Periodically high inlet oil concentrations would occur in which concentrations of oil would spike to 1,000-2,000 ppm and in rare cases much higher. During these upsets it was found that oil removal efficiencies would remain above 90% with outlet oil in water concentrations less than 40 ppm.

"CFD modeling has proven to be a continuously useful tool in the design of tanks and vessels and a critical tool to our R&D of new produced water treatment technologies ," says Douglas Lee. "Many on-going projects are being designed through the use of CFD modeling by GLR Solutions and in the future solids will be tracked through CFD models and a design will be determined in order to better handle produced water with high solids concentrations."

GLR Solutions is also exploring further is the modifications to the physical models of the CFD software itself, to allow them to include explicitly the effects of model bubble and oil coalescence, as this is an obvious occurrence within the system. ■

MORE INFORMATION **www.glrsolutions.com**

Hurricane resistant offshore platform design

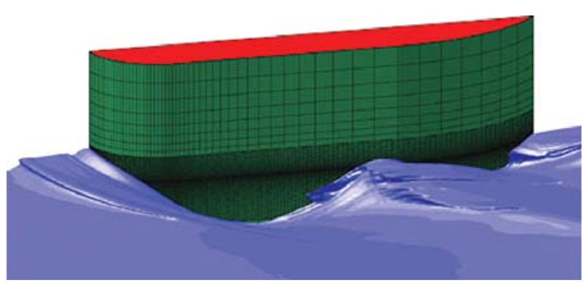


▶ **Fig:01**
CD-adapco software has been used to evaluate the wind and wave loading on platforms in storm conditions. These analyses allowed platform designers and operators to evaluate many different platform loading scenarios without the excessive cost of creating physical prototypes.

Current design standards require that platforms be built to survive so-called 100-year storms, which generate wave heights of up to about 70 feet. However, during Hurricane Ivan peak wave heights of over 90 ft were measured (including one that severely damaged the Chevron Petronius platform) consistent with a once in 2500-year storm. The problem is compounded by the fact that many of the 4000 platforms operating in the Gulf of Mexico were designed before 1988, when the current 100-year design standards came into operation (although some of the destroyed platforms were of recent design).

The scale of these losses combined with pressure from insurers, has led to a rapid re-evaluation to the techniques used to design offshore platforms. Many operators are turning towards Computational Fluid Dynamics in order to provide additional insight into how their platforms perform under the most extreme operating conditions.

Computational Fluid Dynamics (or CFD) is a technique that simulates fluid flow phenomena using super-computer technology. Although its origin is in the aerospace and automotive industries, CFD is increasingly finding application in many areas of the oil and gas industry. CFD can be used to simultaneously simulate the aerodynamic effect of strong winds on the platform with the hydrodynamic influence of waves impacting upon it.

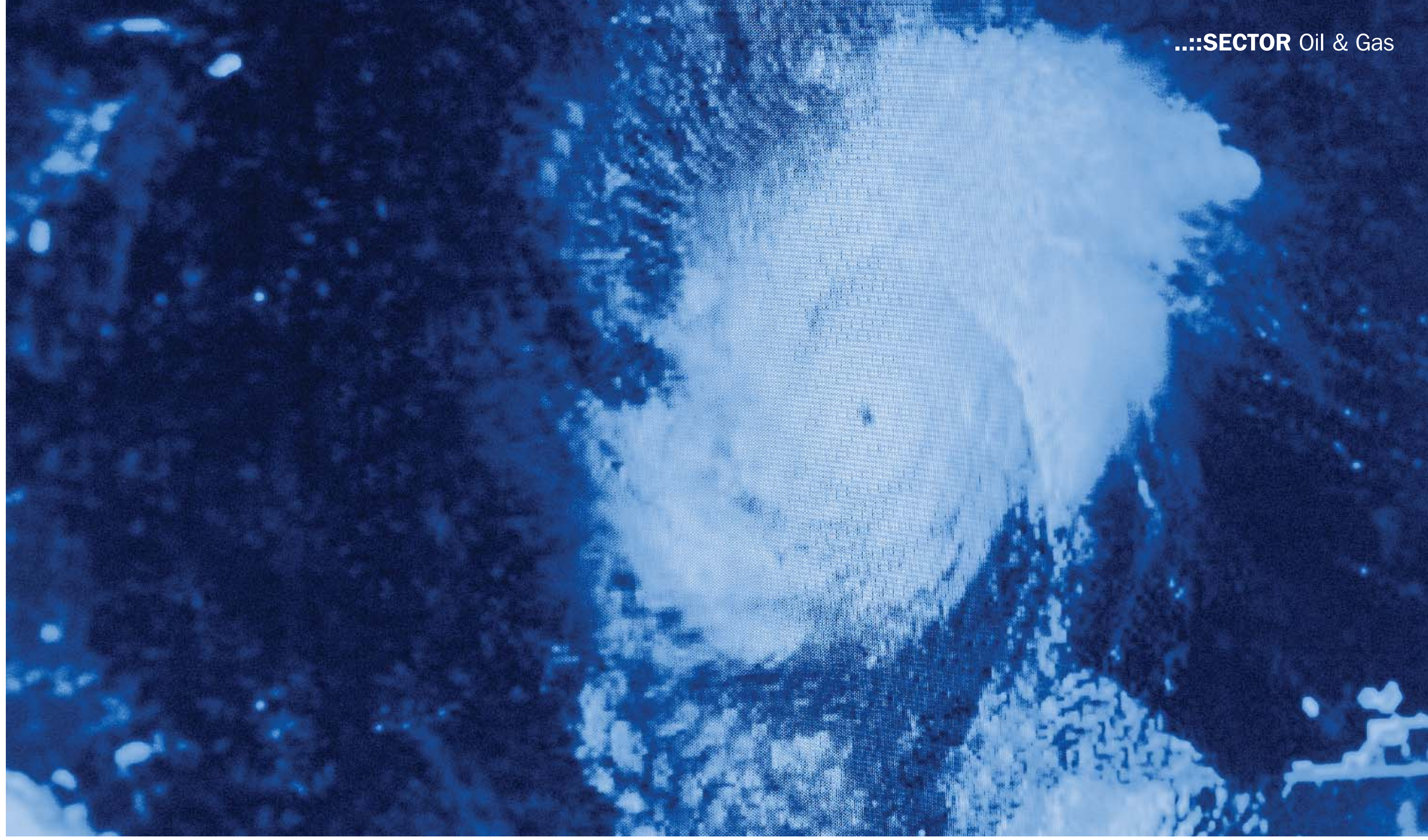


▶ **Fig:02**
CD-adapco's CFD solutions are routinely used in the design of marine applications to understand how a unit will react upon impact by a wave. This technology allows engineers to optimize the hydrodynamic performance of the ship, FPSO or platform and understand the range of conditions under which safe operation can be assured.

Although CFD technology has been routinely applied in many industries since the early eighties, it has only recently begun to be seriously used in offshore platform design. Most current offshore platforms were designed using extensive experi-mental model testing. Although experimental analysis provides considerable insight into the performance of a particular design, physical prototypes are expensive and time consuming to construct.

Dr Dennis Nagy, CD-adapco's Director for the Oil and Gas sector explains: *"It isn't that CFD technology wasn't available when the current generation of platforms was designed; CFD technology has been routinely applied in many industries since the early eighties. It is just that the cost of performing the analysis would, until very recently, have been too prohibitive."*

Nagy feels that the biggest advantage of CFD is that its rapid turn-around time helps to break the dependence of offshore design on pre-existing design codes. Although design wind



and wave conditions are a useful starting condition for offshore platform analysis, CFD simulation allows designers to more easily pursue multiple "what if?" scenarios. Once a CFD model for a platform is set up, it is relatively simple to repeat the calculation for multiple loading scenarios. *"Instead of becoming stuck by the fact that the design codes don't deal with wave heights above 70 feet, using CFD designers are free to consider the impact of wave heights of 80, 90, or even 100 feet,"* says Nagy. *"All they need to do is input the new condition and sit back while the computer does the number crunching. It is a very effective way of assessing the limit of your design."*

Unlike testing of physical prototypes, CFD simulations are typically carried out at full scale (the computer model has the same dimensions as the actual production platform rather than those of a smaller experimental model). This has the considerable advantage that results can be interpreted directly and do not have to undergo scaling, a process that can introduce a significant uncertainty, especially for transient phenomena such as the impact of a wave.

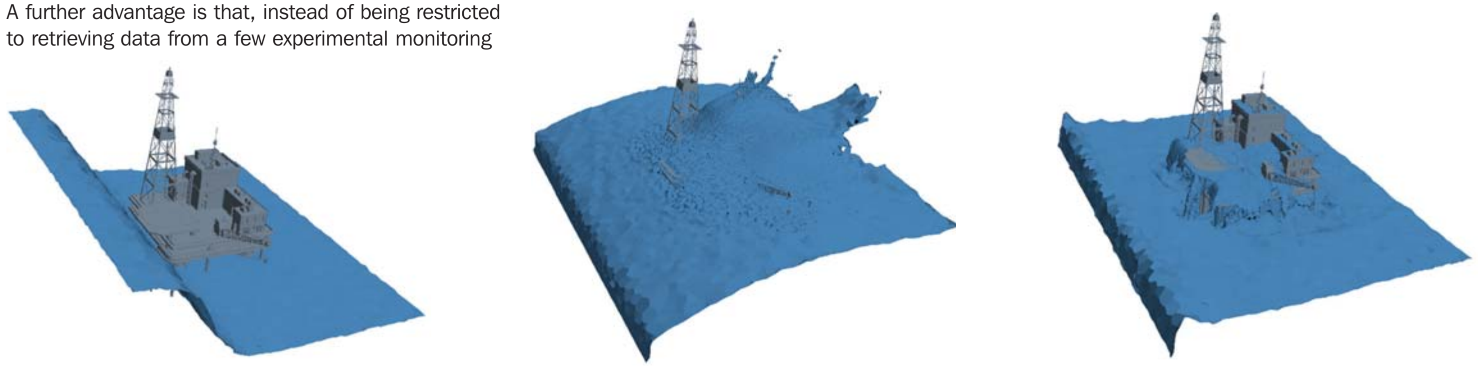
A further advantage is that, instead of being restricted to retrieving data from a few experimental monitoring probes, data is available at every point on the platform, at every discrete time interval for which the simulation is performed. The wave impact on a platform can be viewed from any angle, and the instantaneous forces acting on any part of the structure can be calculated.

Data from CFD calculations can also be used to assist other types of analysis, for example, the forces acting on a platform can be exported to a stress-analysis software package. In extreme cases, where fluid forces cause large deflections of components, the CFD simulation can be coupled directly with the stress analysis tool and both stress and fluid simulations can be performed simultaneously, each simulation feeding new boundary conditions to the other.

In Nagy's view, the adoption of CFD technology as a routine part of offshore design is inevitable. *"In the automotive industry almost every component is designed with the aid of CFD technology, to bring a new product to market without it would be unthinkable,"* he says. *"The financial and environ-mental impact of the recent hurricanes means that the oil and gas industry has no choice but to follow suit."* ■

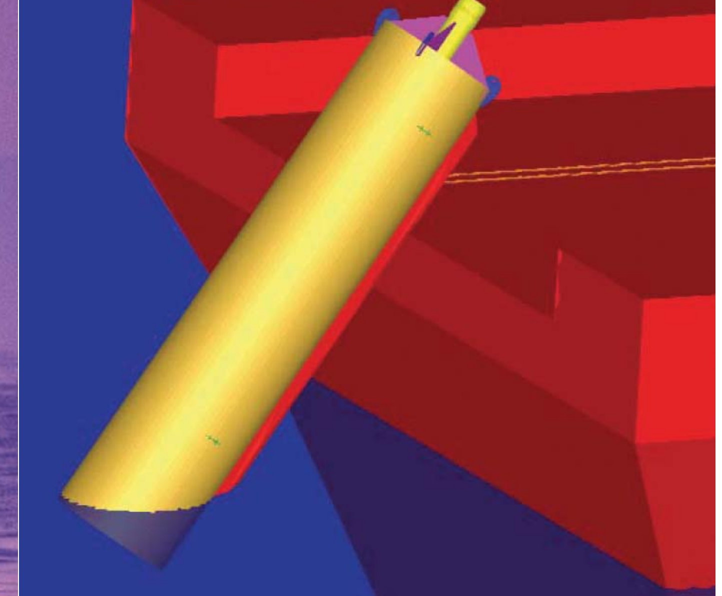
BENEFITS FACTS

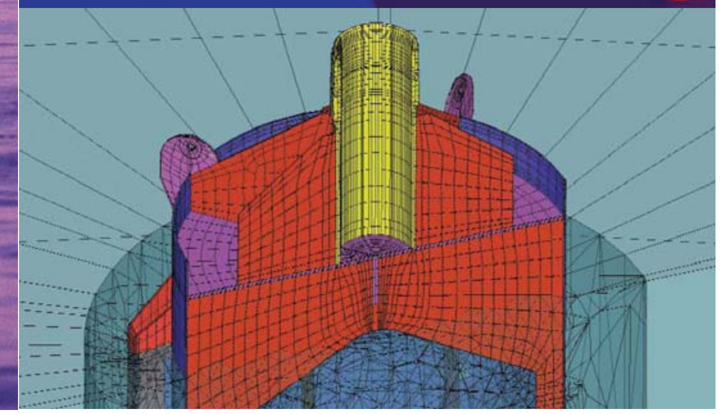
- **Rapid turnaround time.**
- **Easy investigation of 'what-if' scenarios.**
- **Simulations are carried out at full scale.**
- **Data available at every part on the platform at every interval**
- **Can be exported to a stress analysis package**



▼ **Fig:03**
Wave impact study of offshore platform

MORE INFORMATION **oil@uk.cd-adapco.com**





Offshore Mooring

Delmar Systems, Inc. is the world leader in offshore mooring, providing the safest, most efficient mooring solutions for the oil and gas industry. Every Delmar employee is empowered with knowledge to use the most sophisticated technology. Our success is measured only against uncompromising expectations.

www.delmarus.com

When Delmar Systems wanted to perform a design validation of the world's largest suction pile anchor system, they called upon the expertise of CD-adapco. These anchors are used to "park" and anchor large offshore structures such as mobile drilling units.

The anchoring of a drilling rig is somewhat more sophisticated than anchoring a boat. Installing these units is equivalent to towing an eight story building out into the middle of the Gulf of Mexico, tipping it over the side of a large vessel, and burying it two miles down into the sea floor.

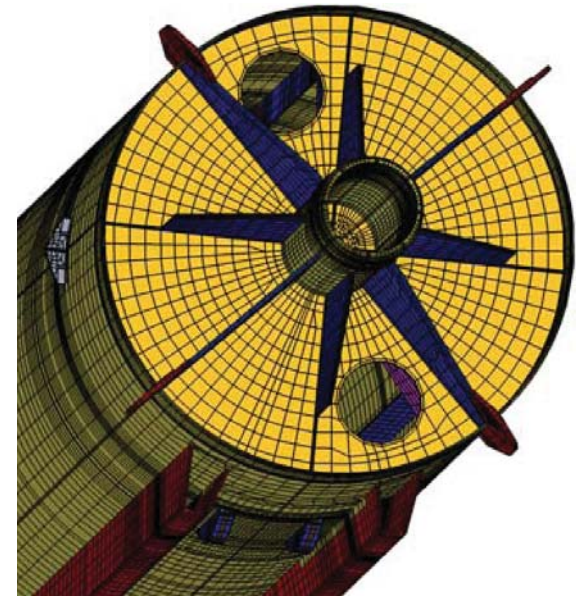
The structure is constructed from massive welded sections of plate steel. Internal ribbing and mounting flanges are designed to provide the necessary rigidity and strength. Not surprisingly, the shear weight of this structure makes its transport a major undertaking.

The most critical of these is the "overboarding event", when the pile is launched overboard using a sled and winch system. There is a potential for damage to the structure during deployment, when the massive weight is cantilevered off the ship.

Proper design of the sled structure and cable mounting system are essential; damage to the pile during this phase would be catastrophic.

Another "event" that requires investigation is the sinking of the structure. Suction piles drive themselves part way into the sea floor under their own weight. A remotely operated vehicle (ROV) is then used to pump the seawater from the tower, creating a vacuum that draws the tower further into the seabed. During this process, adequate internal ribbing is necessary to avoid buckling of the outer shell.

..::SECTOR Oil & Gas



All aspects of the installation were verified with FEA analysis. Analytical models were constructed in pro-fe, the FEA pre and postprocessor supported by the CD-adapco. High quality hex meshing was used for all components of the pile, to ensure good stress values at the mount points. pro-STAR's automatic trimmed cell meshing capability was used to easily fill the soil volume of the seabed surrounding the anchor.

The structural analyses included G-loadings during transport and over-boarding. Particular emphasis was placed on the skid support structure, as well as the cable mounting system. During the initial design phase, the analysis determined that the winch attachment needed to be redesigned for adequate margin.

Additional analysis addressed pressure loadings during installation, when the ROV creates a negative pump pressure on the inside of the pile. The seabed was modeled as a bi-phasic material, using soil properties developed for that particular location. Eigenvalue buckling calculations were also performed to check susceptibility during installation.

The anchor was successfully installed and remains at its installed location for approximately six months to allow the soil to settle. After this time the anchor becomes active as a parking anchor for mobile drilling units. ■

> **"pro-STAR's automatic trimmed cell meshing capability was used to easily fill the soil volume of the seabed surrounding the anchor."**

MORE INFORMATION VISIT **http://www.delmarus.com**

Safer flare design with CFD

Using CFD analysis to predict flare performance

The flaring of natural gas plays a critical role in the global oil and gas industry. According to the World Bank over 150 billion cubic meters (or 5.3 trillion cubic feet) of natural gas are flared and vented annually, mostly as part of the oil and gas production process.

Flaring systems were originally developed to dispose of the waste gas produced as a side effect of the oil production process, although continuous flaring has been effectively outlawed by strict legislation and economic and environmental concerns. Today, flaring systems are principally deployed as safety systems, protecting the production system from over-pressurisation during the extraction process. During surges in production, gas, and occasionally liquids, are routed by a pressure-relief valve (or by an emergency safety valve) up through the flare-header towards the flare tip.

An obvious consequence of combusting large amounts of natural gas is the considerable amount of radiation emitted by an operational flare. Modern flare systems are specifically designed to reduce the radiation, pollution and acoustic impact of a flare, by using the energy associated with the high-pressure gas to entrain large amounts of air (typically using Coanda effects, or sonic or super-sonic nozzles). These aerated flames are small and have a relatively low radiation signature – however the amount of radiation that they generate is still enough to cause significant damage to personnel and equipment on the installation.

Although the American Petroleum Institute API 521 effectively limits the amount of incident radiation on production surfaces to 1390 Wm^{-2} during normal operation -- a level at which "continuous exposure is allowed without causing permanent injury" –- the consequences of exceeding those levels can be serious. At 1580 Wm^{-2}, exposure of more than a minute will cause symptoms similar to mild sunburn. At 1890 Wm^{-2}, bare skin will begin to feel pain after 50 seconds of exposure; by 2840 Wm^{-2} this time is reduced to 30 seconds; at 4730 Wm^{-2} bare skin will begin to feel pain after 18 seconds, and personnel without protective clothing have just 23 seconds to escape to a safe area.

In practice, safety conscious operators try to limit radiation to well below the minimum API standard. This is partly due to economic necessity as, during unexpected surges in gas production, should the radiation levels rise to unacceptable levels, operators have no choice but to reduce production or to stop it all together, effectively limiting the overall profitability of the installation.

Traditionally, the radiation signature of a flare on a specific installation has been calculated using a combination of empirical calculations and ad-hoc post-installation modification. Increasingly manufacturers and operators are turning towards Computational Fluid Dynamics as a way of predicting how flares will perform under realistic operating conditions, before even the first prototype is built.

Computational Fluid Dynamics (or CFD) is a powerful technique that simulates fluid flow phenomena using computer technology. Although its origin is in the aerospace and automotive industries, CFD is increasingly finding application in many areas of the oil and gas industry.

Unlike testing of physical prototypes, CFD simulations are typically carried out at full scale (the computer model has the same dimensions as the actual production platform rather than those of a smaller experimental model). This has the considerable advantage that results can be interpreted directly and do not have to undergo scaling, a process that can introduce a significant uncertainty, especially for problems involving combustion and radiation.

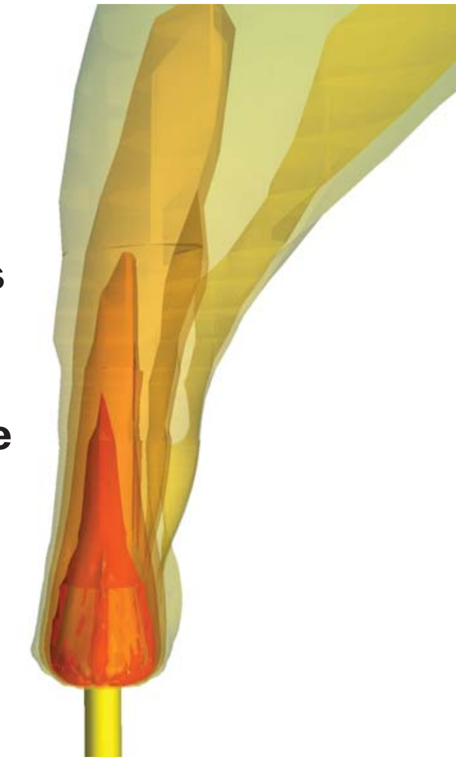
One of the biggest advantages of CFD is that its rapid turn-around time helps to break the dependence of design on pre-existing design codes. Although design conditions are a useful starting condition for offshore design analysis, CFD simulation allows designers to more easily pursue multiple "what if?" scenarios.

CD-adapco has recently performed a number of radiation studies deployed on both fixed and floating units, including a project undertaken on behalf of DPS, a leading engineering design company with a vast amount of experience in the design, supply and support of process equipment for the Oil and Gas industry, which involved the analysis of a flare deployed on a Floating Production Unit (FPU).

Unlike fixed installations, FPUs are limited by stability considerations in the length of boom that they can deploy in order to reduce the incident radiation on the deck. During the DPS study the impact of various mitigation scenarios was considered, including variations in boom angle, installation of physical shielding and the deployment of a protective water curtain. The simulation results allowed the design team to accurately assess which areas of the deck would be exposed to high levels of radiation and to adjust their protection strategy accordingly.

"Using CFD analysis we were able to predict the performance of the flare installation with confidence, which allowed us to carefully select a level of protection that would ensure the safety of personnel and equipment aboard the FPU" said Jasbir Landa, Project Manager for the FPU project at DPS.

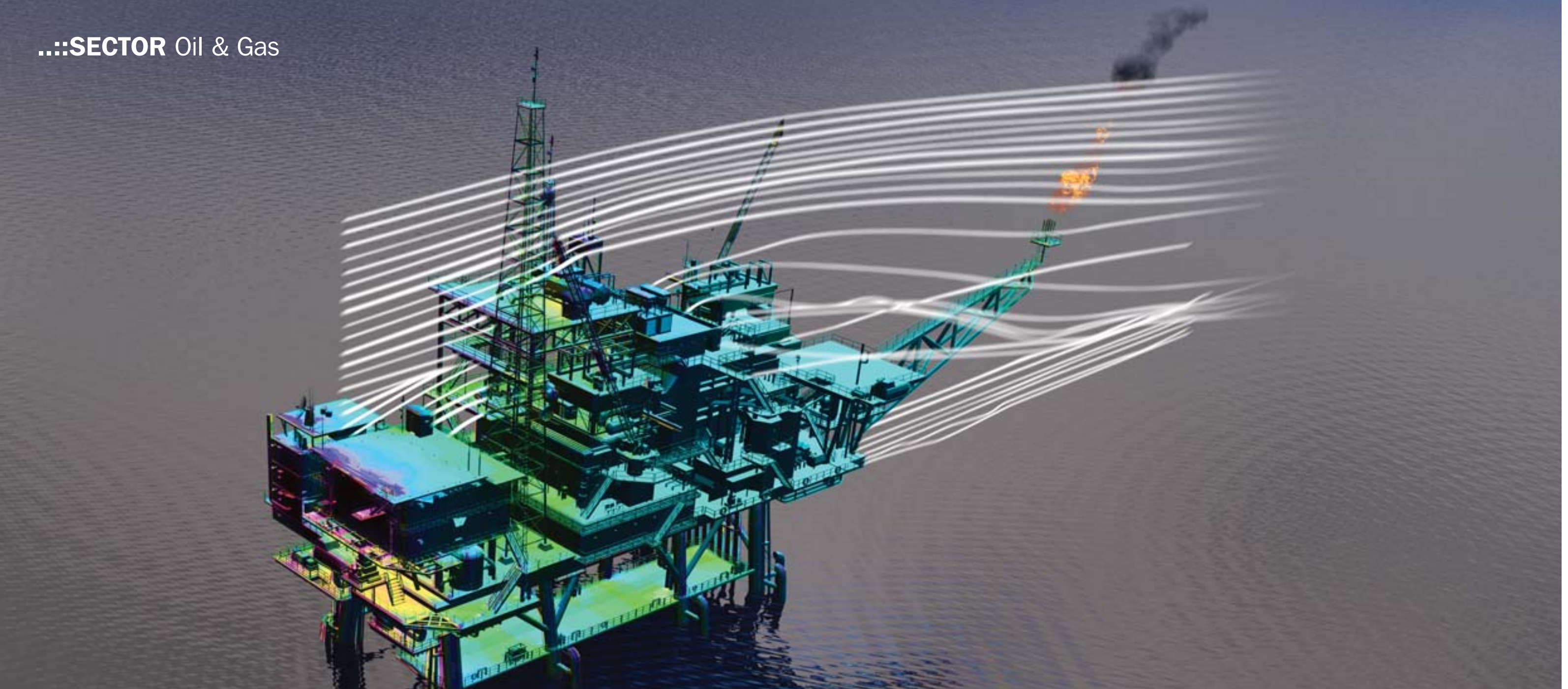
The CFD model was also used to examine the influence of wind speed and direction on the flame combustion and the shape of flame that it generates, something that is generally not possible with less sophisticated flare modeling packages. *"In this case flame shape had a significant impact on the radiation footprint of the flare, which was something we had to address carefully when choosing a mitigation strategy."* ■



▶ **Flare simulation in STAR-CCM+**

"Using CFD analysis we were able to predict the performance of the flare installation with confidence, which allowed us to carefully select a level of protection that would ensure the safety of personnel and equipment aboard the FPU."

MORE INFORMATION **info@dps-global.com**



Latest News

New consulting services from DNV with fluid dynamics software from CD-adapco

DNV (Det Norske Veritas) has selected a software solution for computational fluid dynamics (CFD) from CD-adapco to extend its leading-edge technical consulting service line. The new software will be a valuable addition to the DNV toolbox to provide accurate and reliable estimation of slamming and sloshing loads, which are critical for the design and operation of both ships and offshore structures.

Drawing on more than 140 years of experience, DNV deliver services that predict and assess the motions, loads and other dynamic responses of ships and offshore structures in waves or related fluid flow problems.

Says Dr Bo Cerup-Simonsen, head of DNV Maritime Technical Consulting and DNV Fellow in computational mechanics: "The shipping and energy industries are faced with a number of new challenges, driving the need for novel designs and technologies. For shipping this concerns, among others, container and LNG vessels as well as more specialised ships. The lack of experience for a novel design demands accurate prediction of loads, motions, resistance and propulsion efficiency. For example, slamming pressures on the bow and aft part of the ship and sloshing effects in LNG tanks are some of the areas that are critical and challenging. This new CFD solution combined with our world-class competence will extend our capabilities to better meet this demand."

CD-adapco has a long history of successful partnerships with leading companies in both the maritime and petrochemical industries. The company has invested heavily in providing capabilities within its software that meet the most challenging problems within these industries.

"We are delighted that DNV, with its global presence and as one of the 'big-three' classification societies, has justified this investment by choosing our software. By working closely with DNV, we intend to further refine our technology to meet the industry demands," says Dr Dennis Nagy, CD-adapco - Vice President of Marketing and Business Development and Director for the Energy Sector.

Simulating sloshing behavior

DNV's engineers will use CD-adapco's STAR-CCM+ to tackle problems involving sloshing resonance. Liquid motion resonance can lead to sloshing impacts at sharp corners and knuckles inside tanks, with a potential risk of damage. The software will allow DNV to simulate sloshing behavior driven by a wide range of sea-conditions, allowing engineers both to visualize the liquid motion and to identify critical events that may cause high sloshing induced impact forces. DNV will also use STAR-CCM+ for the analysis of vortex induced vibration and for general six-degree-of freedom free-surface calculations. ■

ⓘ MORE INFORMATION **oilandgas@cd-adapco.com** **Per.Wiggo.Richardsen@dnv.com**

FREE DVD

Flow, Thermal and Stress Simulation Technology from CD-adapco

CD-adapco invites you to explore the business advantage that can be gained through the application of flow, thermal, and stress simulation technology within the Oil and Gas industry with our new DVD.

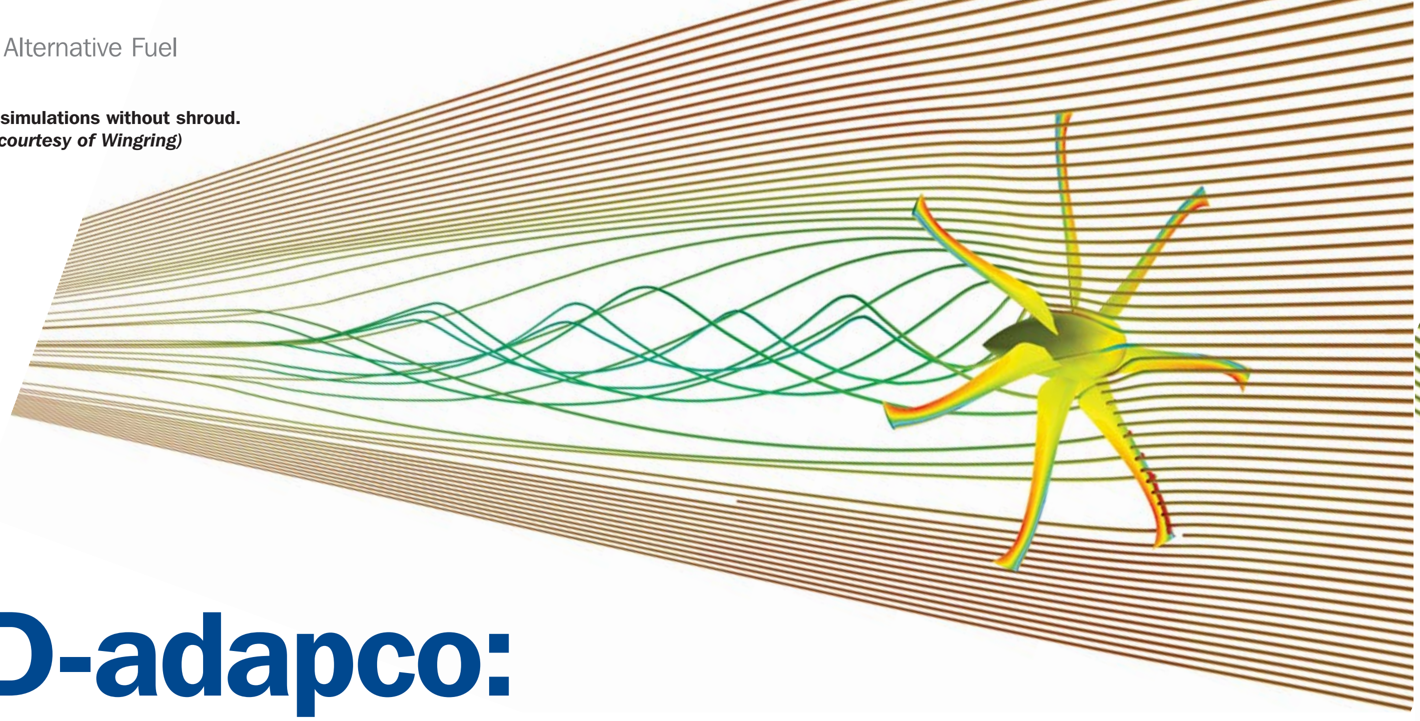
More than any other, the Oil and Gas industry is dominated by fluids. From the extraction, processing and delivery of the hydrocarbons, to designing installations that are more able to resist the most-extreme environmental conditions, understanding the behavior of fluids in and around your process is critical to success in an intensely competitive industry.

Computational Fluid Dynamics (or CFD) is a technique that simulates the fluid-dynamics using computer technology. CD-adapco's flow and thermal simulation technology can provide insight into any problem that involves fluid flow (liquid or gas or combinations of both) and has been applied at every stage of the oil and gas production process - from exploration to extraction, from transport to processing.

Our DVD includes material to explain how CD-adapco's advanced simulation technology can be used to increase safety, reduce costs and improve efficiency of a wide range of Oil and Gas processes. ■

ⓘ TO RECEIVE YOUR FREE COPY OF OUR DVD PLEASE EMAIL **oilandgas@cd-adapco.com** OR CONTACT YOUR LOCAL OFFICE.

▼ **Turbine simulations without shroud.**
(Image courtesy of Wingring)



CD-adapco: Providing Flow Simulation Power for Wind Energy Engineers

In order to meet the world's energy demands in a sustainable manner, engineers need to deliver robust innovative technology. For 25 years, CD-adapco has enabled energy engineers to do just that in the 'traditional' energy sectors.

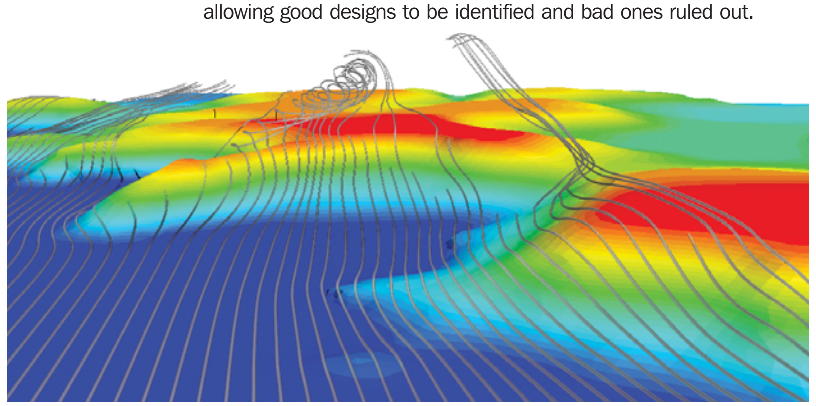
Now we're routinely applying our technology and expertise in the renewable energy sector as well.

CD-adapco's Proven Pedigree

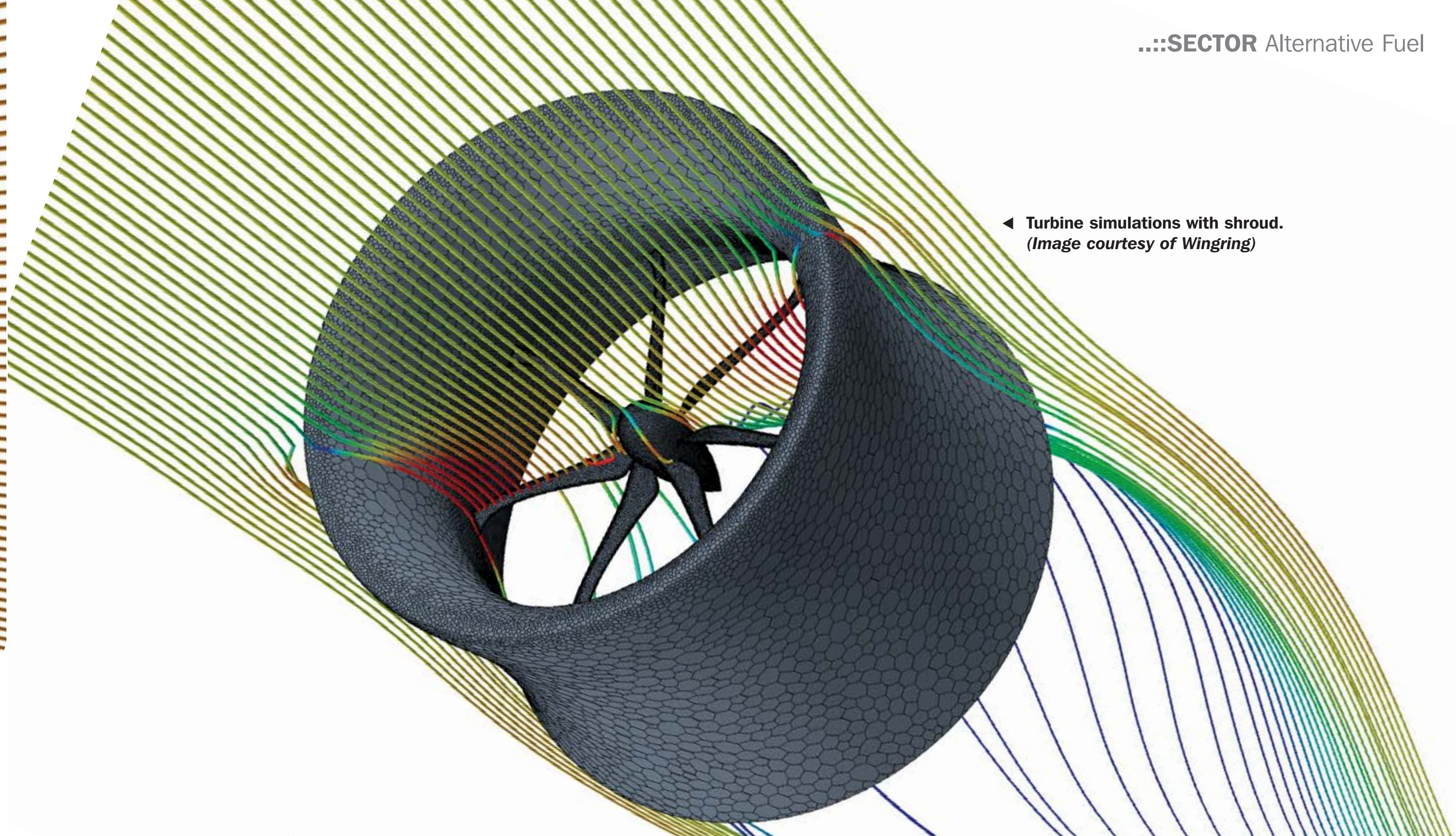
CD-adapco has a proud history of supplying 3D Computer Aided Engineering (CAE) simulation solutions to the Power Generation sector with the industry leading Computational Fluid Dynamics (CFD) codes STAR-CD and STAR-CCM+. Our software and services are increasingly being utilized in the alternative energy sector, as companies seek to improve the performance and durability of their designs in a cost effective way. CFD allows engineers to perform detailed design-analyses before even the first prototypes are built, allowing good designs to be identified and bad ones ruled out.

The Power of CFD in Wind Energy

CD-adapco's 3D flow and structural simulations are now a mainstay in the wind energy industry, routinely providing insight from design to installation. CFD is used in turbine design to understand rotor aerodynamics and the resulting forces, aeroacoustics and icing; to perform simulations of air paths over a site to determine its suitability and optimal turbine distribution; and, offshore it's used to model wave impacts on installations.



◄ **Fig: 01**
Flow paths over a potential wind farm site, indicating areas of high turbulence.



◄ **Turbine simulations with shroud.**
(Image courtesy of Wingring)

Aerodynamics and Structural Loading

During the design of wind turbines, designers aim to simultaneously increase a turbine's power output over the full range of operating conditions, while improving durability. In a recent project, wind turbine design company Wingring used STAR-CCM+ to improve the design of their innovative shrouded turbine. The simulations provided detailed insight into the aerodynamics of the turbine as well as the force loading on it, enabling engineers to quantify the effect of design changes.

Upfront design investigation can have a big impact on a project's profitability. In another wind turbine design study, this time by the University di Udine, Italy, the business benefit of early understanding of designs, prior to prototyping, was recognized. Having compared various designs using STAR, they gained a level of insight that had otherwise been unattainable. University di Udine's Hans Grassmann states CFD's impact on the bottom line.

"If the simulations had been carried out before the prototype stage, millions of dollars could have been saved with obvious benefits to the project profitability and overall success. But the success did not stop there. STAR was able to assist in finding a solution."

"If the simulations had been carried out before the prototype stage, millions of dollars could have been saved..."

Hans Grassmann - University di Udine

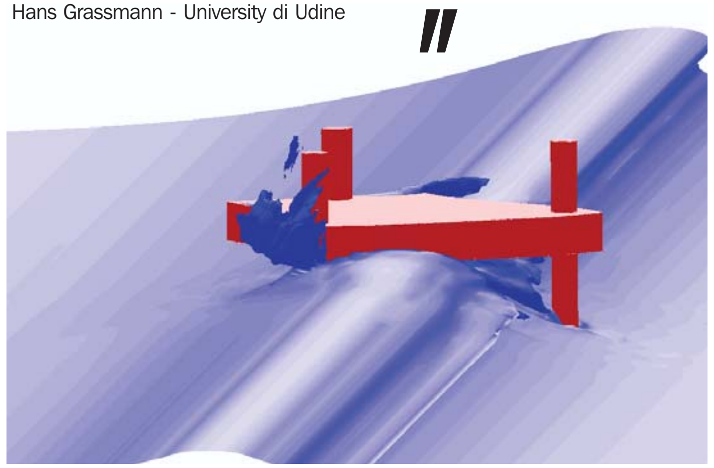
Onshore wind farms

Keeping turbines in a clean airflow significantly increases their power output and longevity. STAR has been used to simulate the airflow over a potential site and turbine configurations to enable wind farm developers to visualize complex wind patterns, identify areas of high wind speeds or turbulence and optimize turbine positioning.

A combination of atmospheric boundary-layer inflow conditions and coupling to meteorological codes ensures accurate representation of the local weather conditions.

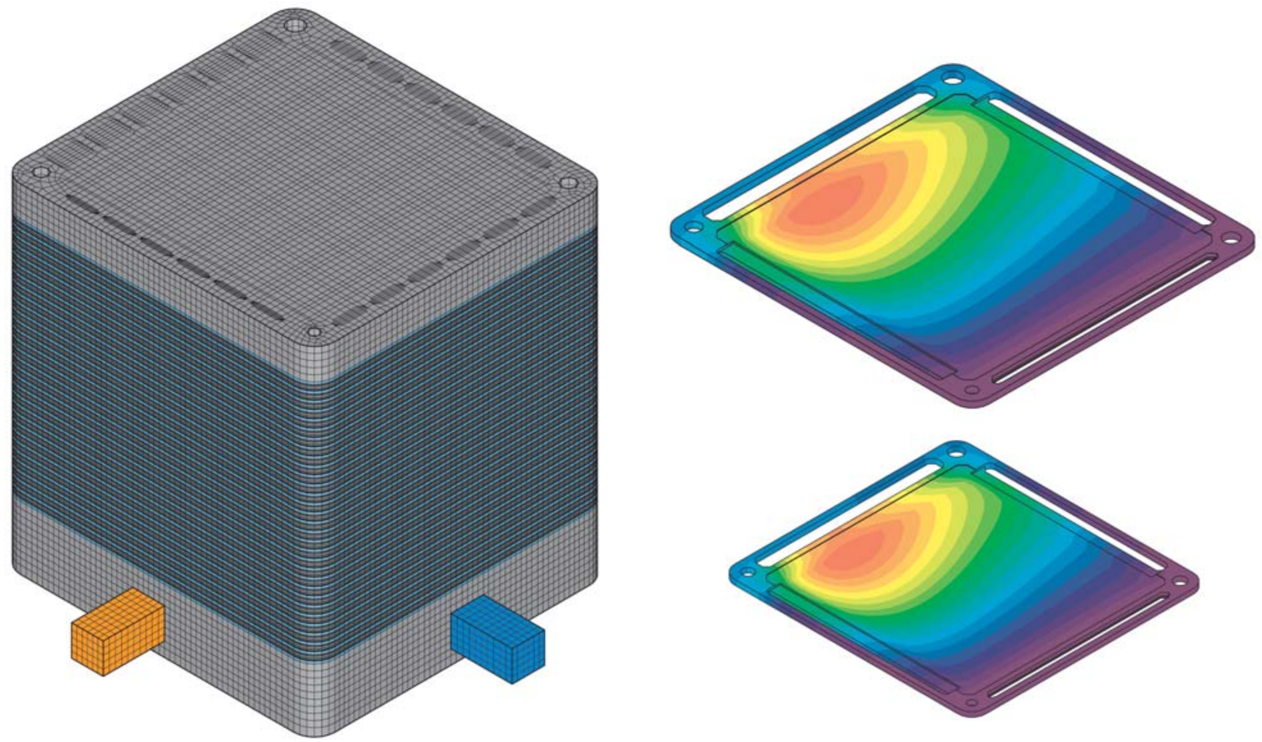
Offshore wind farms

Coping with higher wind-speeds, corrosion and wave impacts, the challenges of producing durable, cost-effective offshore wind farms are great, but far from insurmountable. One example where 3D flow simulation is having a significant impact is in modeling wave loading. The loading on the turbine-supporting structure can be determined for different wave heights, sea-states or wind speeds, at full-scale and without the expense of manufacturing prototypes or physical testing. Technology that's been routinely used in the marine industry for years is now being applied to reduce the impact of the harsh offshore environment on wind turbines. ■



▲ **Understanding the wave impact dynamics on a simple structure with CFD.**

ⓘ MORE INFORMATION **info@uk.cd-adapco.com**



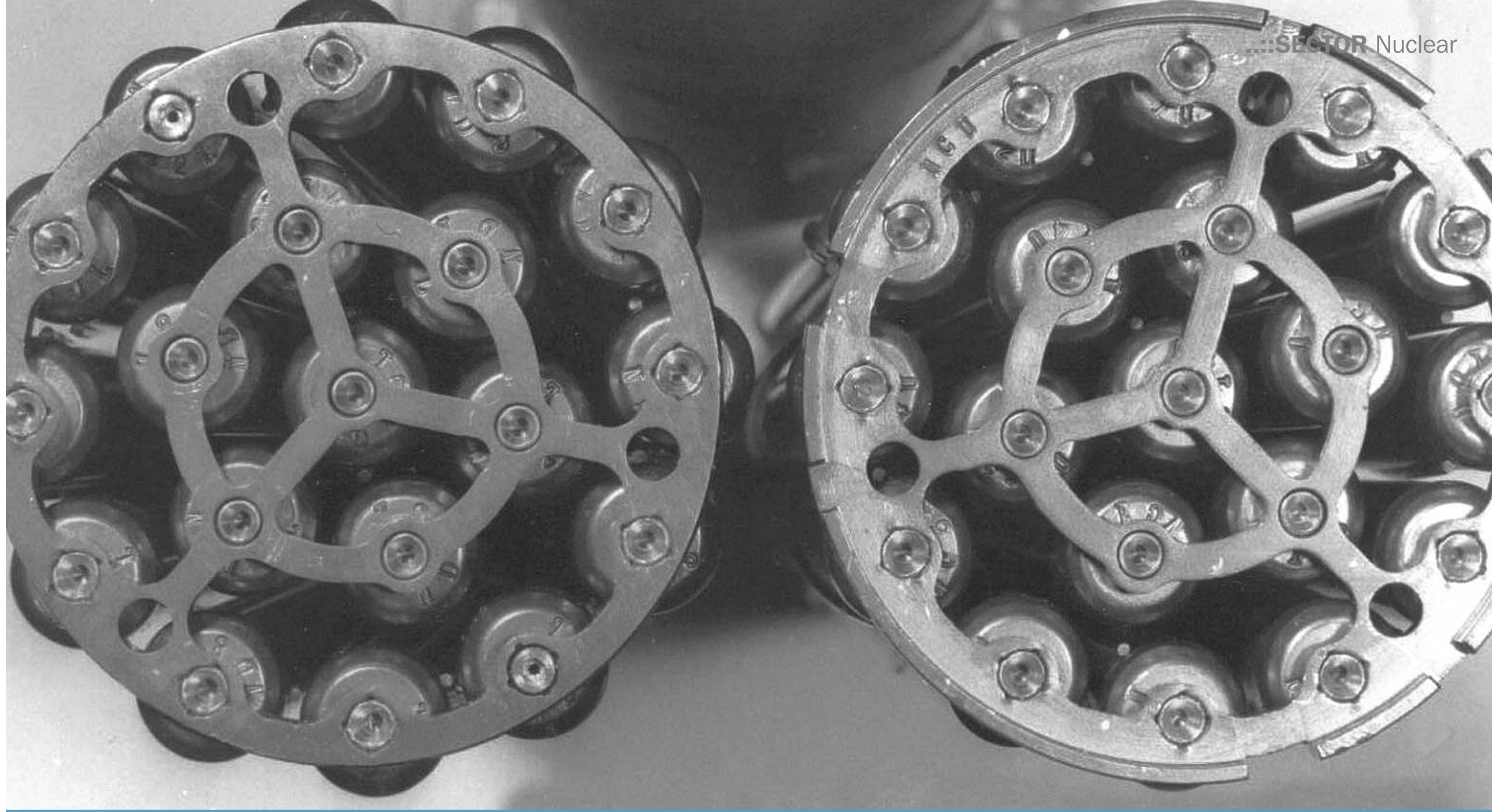
New simulation tool for solid oxide fuel cell design

With increasing environmental concerns and rising oil costs, fuel cells are attracting great interest for transportation and power generation applications as a cleaner and cheaper alternative. In recognition of this, CD-adapco in partnership with the Department of Energy's Pacific Northwest National Laboratory (PNNL), has developed a new Expert System, es-sofc, which is playing an important role in optimizing SOFC (Solid Oxide Fuel Cell) design.

Fuel cells convert the chemical energy of a fuel, such as hydrogen, into electrical energy without combustion, with little or no emission of pollutants and efficient electrical power generation. This is a significant improvement over internal combustion engines. The new es-sofc tool is a knowledge-based tool, carrying with it the essential electrochemistry, fluid flow, heat transfer, and geometrical modeling capabilities required for advanced SOFC design.

es-sofc works with STAR-CD, as a specialized virtual design, prototyping and testing environment. Typical issues that can be handled include: correcting the distributions of fuel and oxidant to the stack, mitigation of excessive thermal gradients along with temperature prediction for the calculation of thermally induced stresses, and manifold/flow passage optimization. These aspects together with gaining a better understanding of the electrochemistry and thermal properties involved, lead to optimized solid oxide fuel cell performance.

According to Gary McVay, who oversees fuel cell development at PNNL, *"As a national lab, our role in the development of solid-oxide fuel cell technology is to create the knowledge base and tools that assist industry in meeting production goals. For a company such as CD-adapco, commercialization of modeling tools is an important part of our overall mission."* ■



CD-adapco and ANL win benchmarking exercise for the

Prediction of boiling within Nuclear Reactor Fuel Bundles

CD-adapco's leading Computational Fluid Dynamics software, STAR-CD, produced very good void distribution results when compared to detailed void distribution experimental data in the BFBT benchmarking exercise at the 2007 BFBT-4 workshop.

In the past decade, a significant amount of effort has been invested toward the simulation of boiling in Boiling Water Reactor (BWR) fuel bundles. The detailed void distribution inside the fuel bundle is regarded as one of the important factors affecting the performance of BWR fuel assemblies.

The NUPEC Boiling Water Reactor Full-size Bundle Test (or BFBT) benchmarking exercise is jointly run by OECD Nuclear Energy Agency (NEA), US Nuclear Regulatory Commission (NRC), Penn State University (PSU), Japan Nuclear Safety Organization (JNES), NEA Nuclear Science Committee (NSC), and NEA Committee on Safety of Nuclear Installations (CSNI).

In the steady-state microscopic grade benchmark of the void distribution, the evaluation team concluded that the results submitted by Dr. David Pointer and Dr. Adrian Tentner from the U.S, Department of Energy's Argonne National Laboratory, obtained with the STAR-CD code, provide the best match with the measured detailed void distribution, when compared with two other major CFD codes.

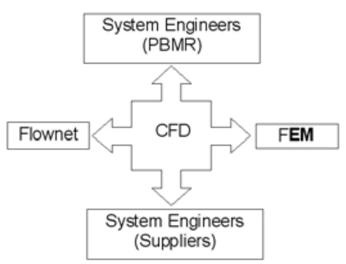
The results were obtained using STAR-CD and the Extended Boiling Framework developed by CD-adapco in collaboration with Argonne. The evaluation team concluded that STAR-CD showed very good agreement with both the overall bundle distribution and the detailed intra-channel distribution. Other CFD codes predicted the vapor to remain near the heated pin surface, unlike the experimental data that shows the vapor migrating towards the center of the channel.

"This is a significant success for the combined ANL and CD-adapco team and I want to thank CD-adapco for their close cooperation in the BFBT analysis and acknowledge their key role in the development of the Extended Boiling Framework, which was essential in these calculations", said Dr Adrian Tentner of Argonne National Laboratory. ■

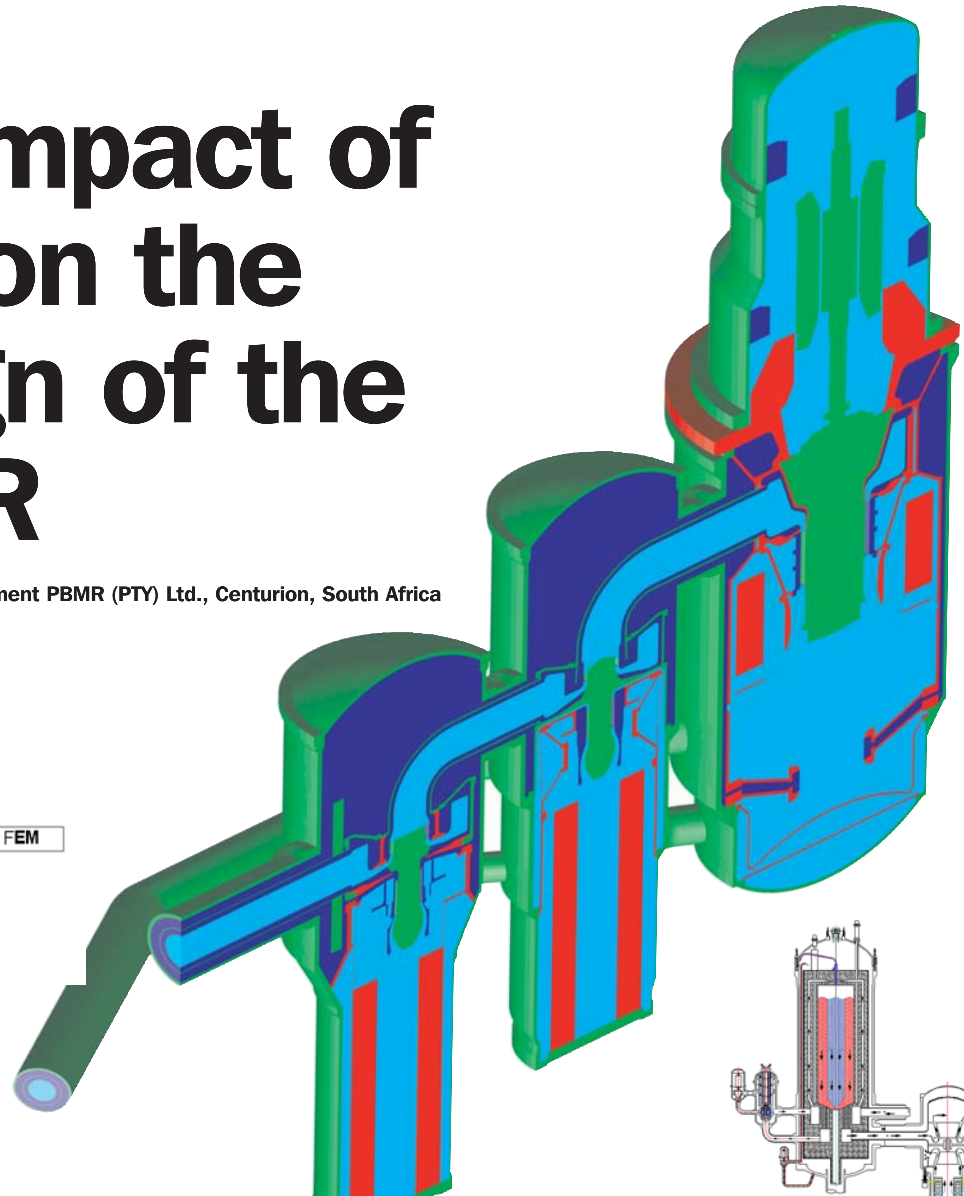
MORE INFORMATION **info@uk.cd-adapco.com**

The impact of CFD on the design of the PBMR

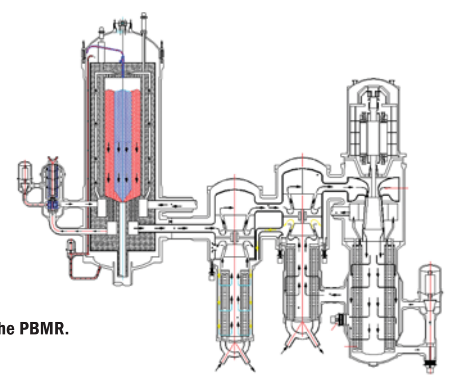
Sarel Coetzee, CFD Department PBMR (PTY) Ltd., Centurion, South Africa



▲ Fig:02
CFD interaction with internal and external groups.



▶ Fig:04
Meshed volumes of the PCU.



▶ Fig:01
Layout of the PBMR.

The Pebble Bed Modular Reactor (PBMR) is a next generation nuclear power plant with high thermal efficiency and inherent safety characteristics.
The extensive use of CFD in the design of the PBMR allows the engineers to tackle challenges during the design phases that would have otherwise only been encountered at high cost during the commissioning or operation of the plant.

The PBMR utilizes a direct cycle high temperature gas cooled Reactor Unit (RU) and Power Conversion Unit (PCU). The plant has a reactor of a pebble bed type and a three-shaft helium Brayton Cycle (Fig 1). The helium gas is heated by the reactor, passes through a high-pressure Turbine, low-pressure Turbine and Power Turbine, driving the generator. It passes through a Recuperator, Pre-Cooler, low-pressure Compressor, Inter-Cooler and high-pressure Compressor, back through the Recuperator to the Reactor. Helium is chosen as the working fluid due to the particular benefits that it brings to closed cycle high temperature reactors. Its advantages are that it is a chemically inert gas and thus not affected by radiation, high specific heat and its high sonic speed (three times higher than air), allows higher circumferential velocities on turbo machinery blades. The disadvantages are that some PCU components need to be either specifically developed for helium or adapted from existing components.

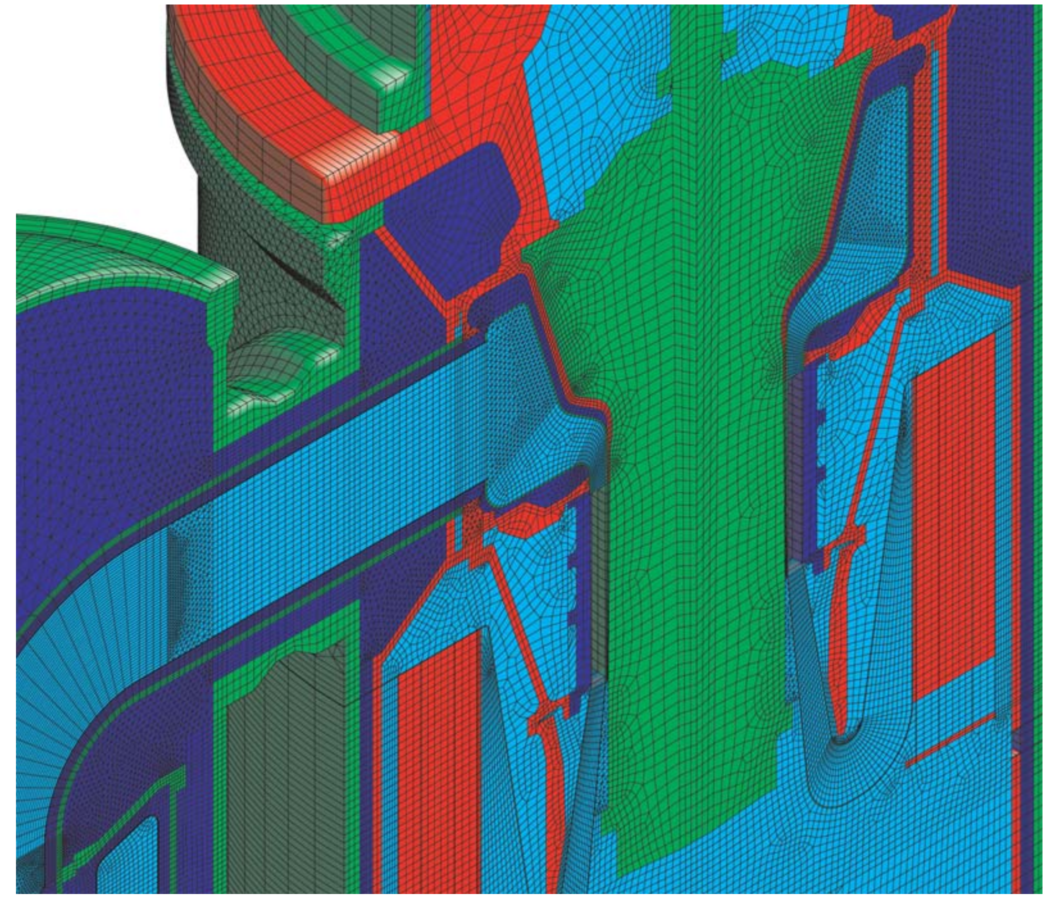
CFD in the design process

CFD provides detailed information to System Engineers from PBMR as well as external suppliers and serves as input to FEM analyses when integrated CFD-FEM results are required.

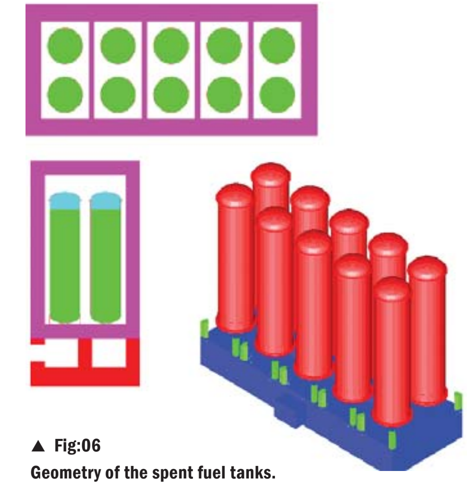
CFD provides component characteristics for complex geometries to Flownet, a one-dimensional thermo-hydraulic network solver. This interaction is graphically shown in Figure 2.

Reactor Unit, RU

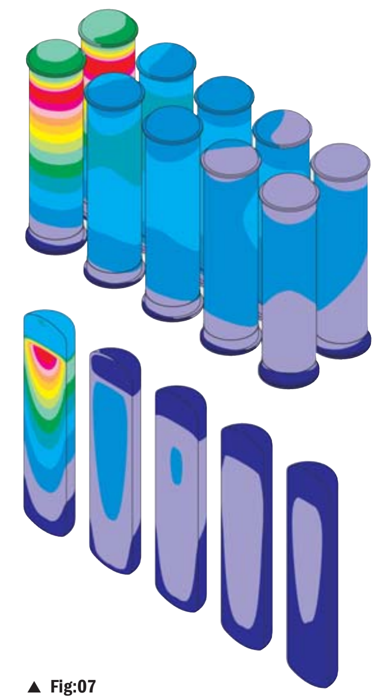
The RU of PBMR consists of a central column of graphite spheres surrounded by an annular fuel pebble bed, enclosed by graphite blocks on the inside of the core barrel. Between the core barrel and the reactor pressure vessel is a gap filled with helium. Between the reactor pressure vessel and the concrete is an array of water pipes, protecting the concrete against high temperatures (Fig 3). CFD is employed to investigate local as well as global thermal and fluidic effects.



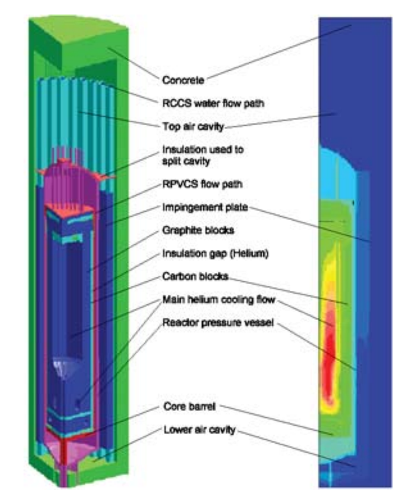
▲ Fig:05
Close-up of the PCU mesh.



▲ Fig:06
Geometry of the spent fuel tanks.



▲ Fig:07
Temperature contours of the spent fuel tank walls.



▲ Fig:03
Model of the PBMR reactor unit.

These CFD results have led to several design changes to satisfy the PBMR specifications, ranging from the design of the water pipes, support structure of the vessel to the design of the helium inlet and outlet slots. It is clear that the design of the RU has been greatly influenced by the detailed CFD results.

Power Conversion Unit, PCU

Cycle pressure losses and leakage flows have a major effect on cycle efficiency. The cycle pressure losses are primarily a function of the individual component designs and layout. Leakage and cooling flows are also a function of the component design and component cooling strategy. These pressure losses, leak flows and cooling flows must be determined across interfaces between components from different suppliers and through the components themselves. Some of these interfaces have a high temperature and/or pressure gradient. This calls for integrated CFD and FEM analyses. Therefore, a complete CFD model of the PCU was constructed, containing all the different components and interfaces (Fig 4 and Fig 5).

All fluids and solids were solved simultaneously to obtain temperature and pressure fields that were mapped onto a FEM mesh. The CFD results were also used to calculate pressure drops across the different components. The calculated loss coefficients are used by Flownet to improve the accuracy of the cycle calculations. Detailed information could also be supplied to the component designers regarding the thermal environment in which their components will operate.

Spent fuel storage tanks

The Spent Fuel Storage Tanks are used to store spent fuel from the power plant generated during its production lifetime (40 years). Thereafter, the tanks must store the spent fuel for another 40 years before being decommissioned. Detailed and accurate temperature distributions throughout the complete Spent Fuel Storage Area are needed, ensuring that the temperature limits for the fuel, tank, supports and concrete are not exceeded. CFD was used to simulate this complete Spent Fuel Storage Area. This model included the fuel, helium in the tank, the tank itself, the air surrounding the tank and the concrete walls of the area (Fig 6).

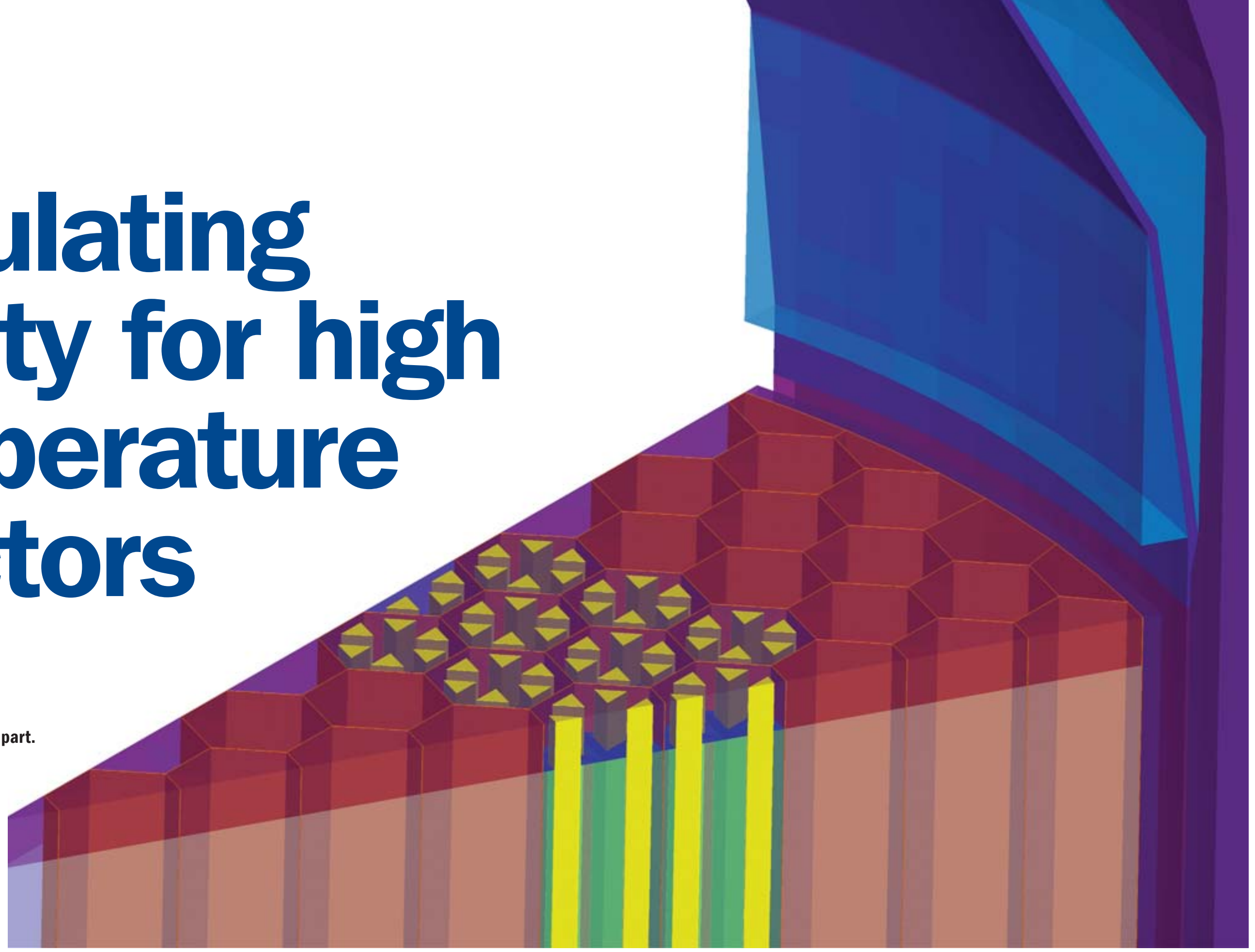
From the results, temperature distributions in all of the materials could be obtained. The temperature distribution for the tanks is shown in fig 7 and the temperature distribution of the fuel is shown in Figure 8. The high heat source from the "youngest" fuel can be clearly seen. Note also the effect this has on the tank temperatures. CFD supplied answers to the tank designers, the HVAC designers, the building designers and the nuclear physicists.

Conclusion

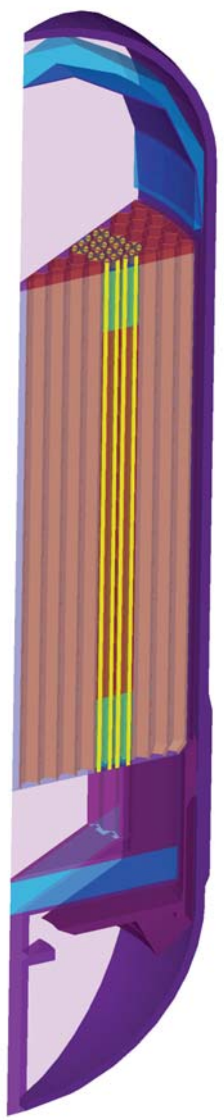
CFD has been a major contributor to improving PBMR design. Optimizing many design aspects before commissioning and operating the plant has saved time and cost. ■

ⓘ MORE INFORMATION **info@uk.cd-adapco.com**

Simulating safety for high temperature reactors



▶ Fig:01
Computational domain – solid part.



Jan-Patrice Simoneau, Julien Champigny, AREVA, France.
Brian Mays, Lewis Lommers, AREVA, USA.

When presented with a problem that combines 1000 ºC plus temperatures; convection, conduction, and radiation; time scales of over one hundred hours; and length scales that range from two millimeters to twenty meters, most CFD engineers and CFD codes would (quite understandably) admit defeat without so much as applying a boundary condition in anger.

But having been presented with just such a scenario, the engineers at AREVA didn't raise the white flag, they took up their STAR-CD manuals and got to work.

The result: they have successfully developed a methodology for using STAR-CD to simulate the cooldown of a High Temperature Reactor (HTR), thereby obtaining a level of understanding which would otherwise be unobtainable through experimental work alone.

Recent years have seen a resurgence in interest in HTRs. This is driven by the possibility of using nuclear energy for the production of process heat (to be used for hydrogen production, for example) as well as the inherent safety characteristics of HTRs.

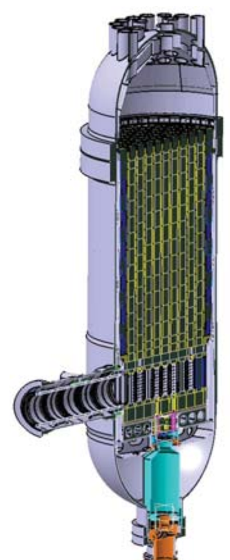
The most efficient means of hydrogen production, such as high-temperature electrolysis or thermo-chemical water splitting, require very high temperatures. As a result, HTRs have been adapted to increase their output temperatures from approximately 850ºC to temperatures approaching 1000ºC.

An important safety characteristic of the HTR design is that, if the normal heat sink becomes unavailable, the heat generated by the fuel is passively removed to cooling panels along the cavity walls by conduction and radiation. Consequently, the fuel particle temperatures remain at acceptable levels even when the normal heat removal method fails. Two variations of this "failure" scenario were investigated for an adapted HTR design. Figure 1, shows a schematic of the reactor.

The test case

Two hypothetical scenarios are under investigation here: in the first, the helium coolant is lost and the primary loop is in a depressurized state (called Depressurized Conduction Cooldown or DCC); in the other, the helium coolant is retained and the primary loop remains pressurized (called Pressurized Conduction Cooldown or PCC). The key difference for the analysis is that, during PCC, natural convection effects within the primary loop significantly alter the temperature profiles in the core.

Both STAR-CD simulations start at the point when the reactor is operating at full power (600 MW) and all heat sinks are lost except for the Reactor Cavity Cooling System (RCCS, a set of vertical panels along the concrete walls of the cavity containing the reactor vessel, which remove heat from the cavity to the outside). At this point the protection system automatically drops control rods into the core to shut down the reactor. The simulations end one hundred hours (or over four days) of simulated time later, when the energy removed from the reactor exceeds the heat produced by the decay of radionuclides in the core, thereby causing the total energy in the system to decrease.



▲ Fig:02
Section through reactor.

The main barrier to simulating such long time scales is that time steps in transient calculations are limited by the requirement that the Courant number-a measure of the number of cells that information travels across in one time step-remains below fifty. Here, this corresponds to a maximum time step size of 0.1 seconds. In order to maintain calculation times at feasible magnitudes, a novel method was used. The calculation begins by solving both the momentum and temperature fields for the initial one thousand seconds. Then it alternates between solving both momentum and temperature fields using small time steps and solving the temperature fields alone, with the momentum field frozen, using large time steps.

Running one calculation, in which both the momentum and temperature fields were solved at every time step for ten hours of simulated time, and then comparing the results of this calculation with the results of the method described above validated this approach. Very little difference was found.

The computational domain is a 30º section of the reactor vessel (shown in figure 2). The various materials in the reactor are either modeled explicitly in STAR-CD or homogeneously: i.e., the "mean" physical properties of a particular heterogeneous component are used (an example being the fuel elements, which are composed of fuel compacts and a graphite web containing holes for the compacts and the coolant).

DCC Results

Figure 3 shows the time history of temperatures in the reactor during a DCC transient. With coolant no longer flowing downward through the core, the hottest region moves upward from the bottom of the core to the center. Meanwhile, the upper and lower parts of the reactor vessel interior—which are at the coolant's inlet temperature during normal operation—become cooler. The core experiences its highest temperature of 1450 ºC at approximately eighty hours after shutdown. Initially during the transient, the temperature of the pressure vessel decreases as it is cooled by the RCCS. After ten hours, however, some of the heat from the core has passed through the outer graphite blocks and produces a rise in the peak vessel temperature, which reaches a maximum value (of nearly 480ºC) after one hundred hours of simulated time.

PCC Results

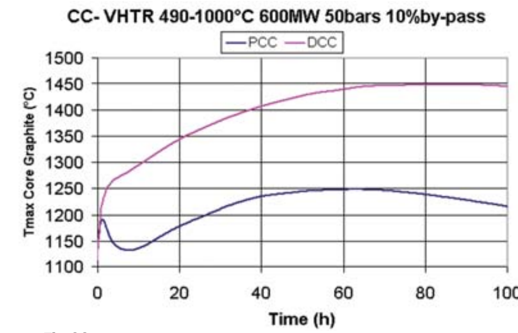
Figures 4 and 5 show the temperature profiles in the fluid and solid regions against time for a PCC transient. The natural convection currents develop quickly, with fluid circulating up through the core channels and reflector gaps. The STAR-CD simulation enabled detailed insight into the flow patterns and how they evolve over the cooldown period. A significant change in the flow patterns was observed after approximately ten hours, with the flow currents effectively reversing direction.

An important finding was that, although the peak temperatures are not much higher during PCC than under normal operation, the natural convection currents cause the hot spot to move up the reactor core, exposing the top of the reactor core to high temperatures. Although the DCC case exhibits higher peak fuel temperatures than the PCC case, the thermal stresses on the upper part of the reactor are much greater for the PCC.

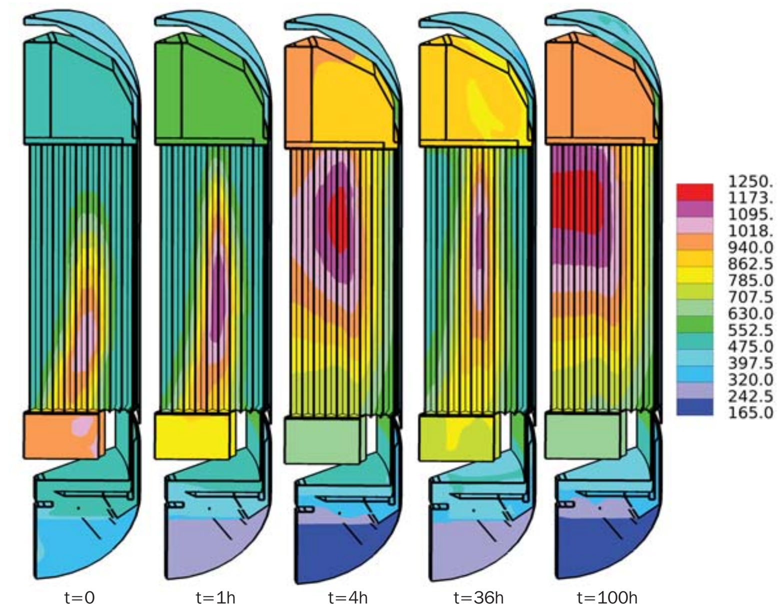
Finally, figure 3 shows that, once the initial ten hours of the transient have passed and an offset is established, the behavior in the DCC and PCC scenarios is very similar.

Conclusion

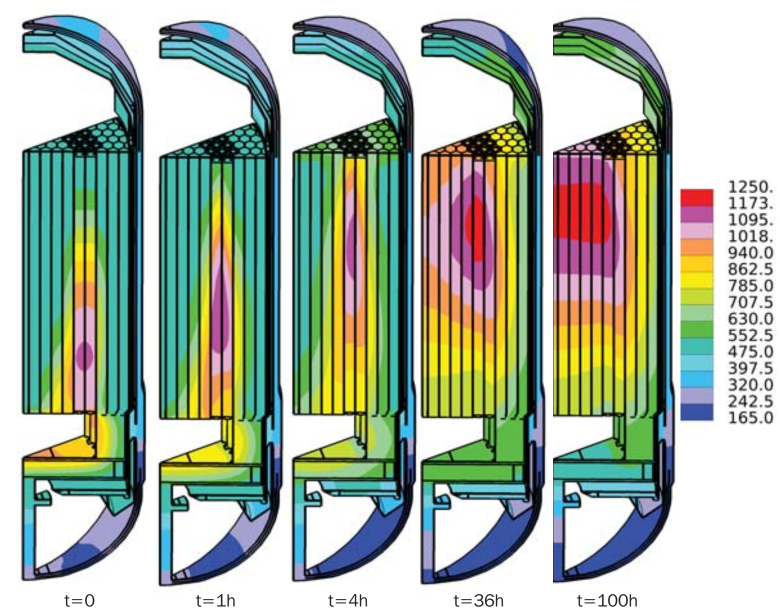
A key characteristic of HTRs is the inherent safety of their design, due to passive heat removal. A methodology has been developed and actuated to simulate the reactor cooldown over a duration of one hundred hours. Initial calculations have been carried out to compare two scenarios, DCC and PCC. The simulation yielded detailed information about the flow patterns in the vessel and the transient thermal behavior. ■



▲ Fig:03
Core temperature against time: PCC and DCC.

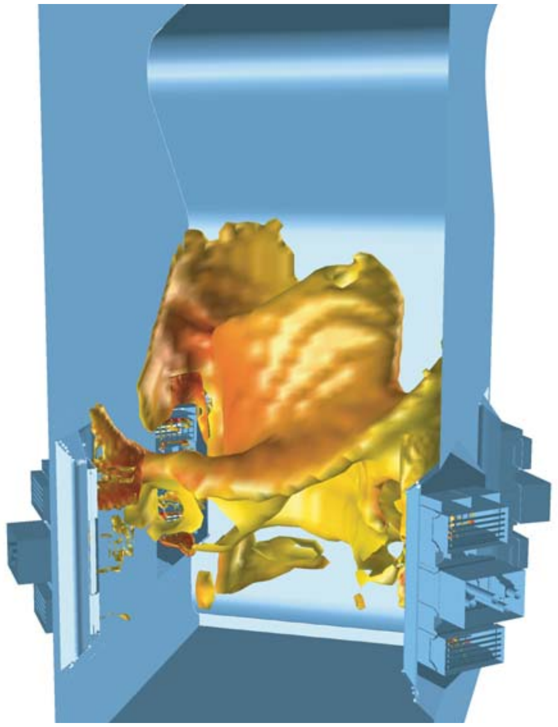


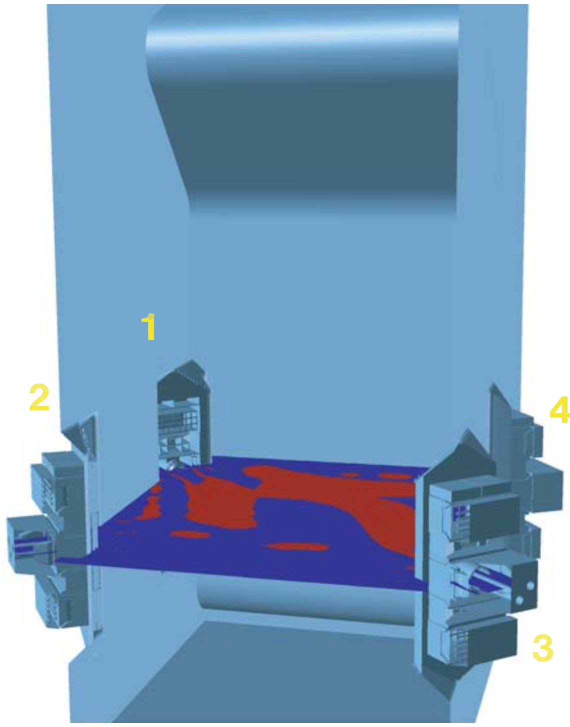
▲ Fig:04
Pressurized Conduction Cooldown – Fluid Part.



▲ Fig:05
Pressurized Conduction Cooldown – Solid Part.

MORE INFORMATION **info@uk.cd-adapco.com**





◄ Fig: 01 (far left)
Luminous flame contour.

◄ Fig: 02
Luminous flame contour;
Fuel-Rich port level.

Analysis of a low NOx burner re-fit of a tangentially fired boiler

Dr. F. McKenty, L. Gravel, M. Mifuji
BMA - Brais, Malouin and Associates Inc.

This study concerns an existing 260 million BTU/hr tangentially fired process boiler which was re-fitted with a low NOx natural gas firing system. The re-fit involved changing the burners, the windbox and the Air/FGR ductwork, as shown, with the boiler, in figure 1.

After a year of operation, some very unusual damage was witnessed in some of the burners' Fuel-Lean ports. This was rather puzzling to the manufacturer as this burner design had been tried and tested for almost 20 years without suffering such problems. The only difference between this particular unit and the other tangentially fired boilers where this technology was installed is in boiler capacity; all the other boilers had capacities ranging from 500 to 1000 million BTU/hr. Due to the impossibility of carrying out any kind of experimental measurements inside the furnace area of an operating boiler, Cerrey S.A. de C.V., an industrial boiler manufacturer, requested that a CFD analysis of this boiler's operation be carried out in order to determine the cause of these unusual problems.

In order to determine if the combustion system had any operational difficulties when fired according to design specifications, BMA first carried out a detailed furnace simulation (3.6 million cells). Details of the burner and windbox are shown in figure 2. The burner is divided into six zones: a lower Fuel-Lean port with pre-mixing chamber, a lower pure re-circulated flue gases (FGR) port, a central Fuel-Rich port, an upper FGR port, an upper Fuel-Lean port and an Over-Fire-Air (OFA) port. The pre-heated combustion air is diluted with FGR prior to entering the windbox with an average FGR mass fraction of 13%. The first simulation was carried out at 100% load using boundary conditions corresponding to a homogeneous Air/FGR mixture entering the windbox.

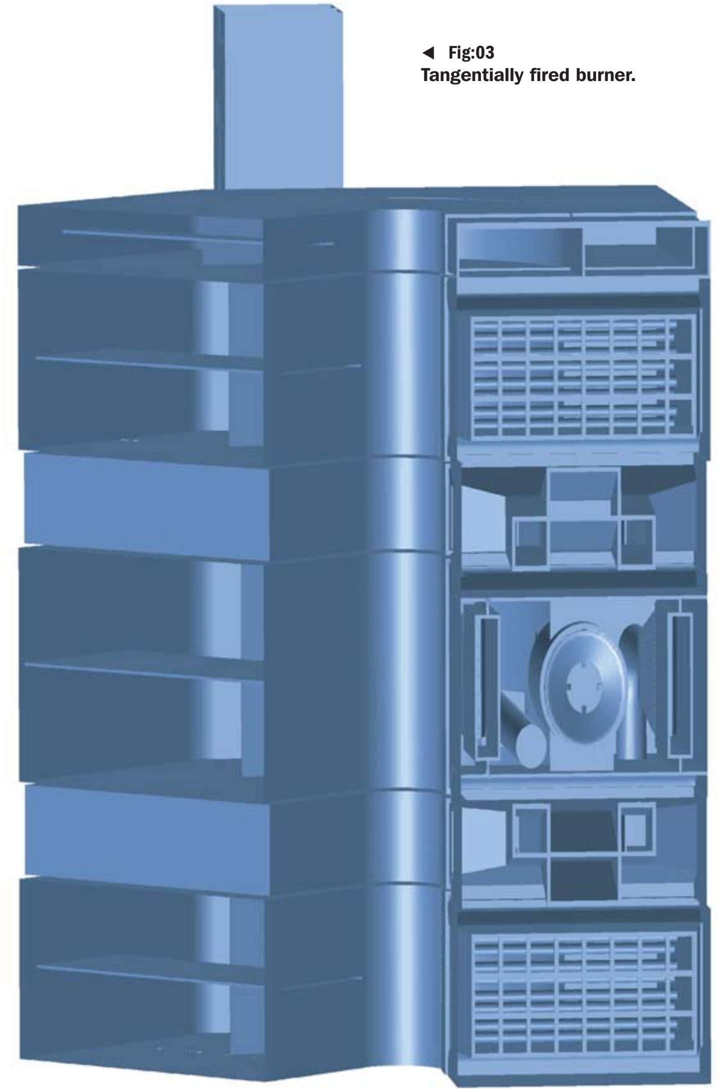
The results of the simulation showed that this firing system operates within specifications when subject to design conditions. Furthermore, no examples could be found of situations likely to cause the damage to the Fuel-Lean ports. Consequently, it was concluded that the problem with this unit probably lay upstream of the firing system. To explore this further, a second simulation (6.2 million cells) including all the Air and FGR ductwork was carried out at 60% load. The partial load operating conditions were chosen because, as this is a process boiler, it spends half of its time operating in turndown mode, which is when it is deemed that the damage to the Fuel-Lean ports is most likely to occur.

Figure 3 shows the FGR mass fraction distribution in the ductwork and windbox. The pipes shown in red contain 100% FGR and lead either directly to the FGR ports or into the pre-heated air stream. The figure clearly shows that the FGR distribution within the pre-heated air stream is uneven. Figure 4 shows the FGR mass fraction distribution at the burner face. Burners 1 and 2 have FGR concentration distributions that vary widely from port to port. Burners 3 and 4 have near-homogeneous FGR distributions; these are the burners that are located on the far side of the boiler and so there is more space for the FGR and air streams to mix. Nevertheless, the average FGR mass fraction at burners 3 and 4 is only 7%, which places the Fuel/Air/FGR mixture in the pre-mixing chamber within flammability limits. The FGR distribution imbalance was thus demonstrated to be the source of the damage to the Fuel-Lean ports caused by the mixture igniting within the port. The specified design concentration for FGR of 13% would have made this situation impossible.

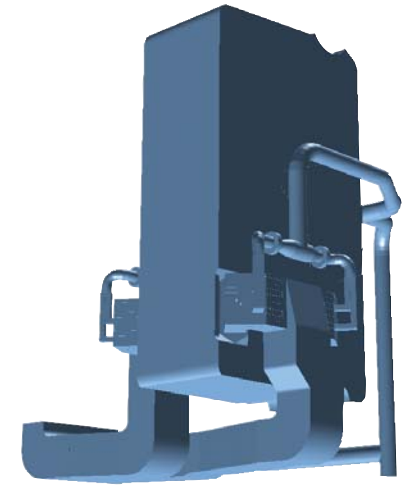
Figure 5 shows the resulting fireball. It is asymmetrical (Figure 6) as the flame from the Fuel-Rich port of burner 2 is severely lifted due to the very high FGR concentrations. This behavior was confirmed by visual inspection of the fireball through the boiler view-ports. Figure 7 shows flames developing inside the upper Fuel-Lean port of burner 3; this is the ultimate cause of the damage.

The STAR-CD simulations allowed BMA to pinpoint the cause of the damage as being the result of incomplete mixing of the pre-heated air and FGR streams. This mixing problem only presented itself on this small process boiler because the Air/FGR ductwork is considerably scaled down when compared to the larger installations where this technology had previously been installed. Consequently, the FGR stream has much less room to thoroughly mix with the pre-heated air stream before the ductwork splits the flow to each side of the boiler. This behavior would have been hard to predict without CFD as most aerodynamic mixing phenomenon do not scale linearly with geometrical size and flow rates. In this case maintaining geometrical similarity with larger units was insufficient to ensure identical flow patterns.

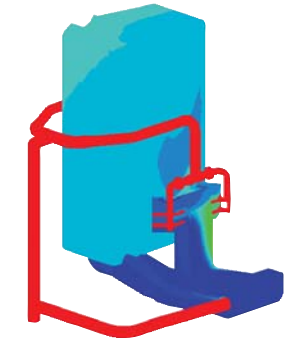
In summary, the STAR-CD simulations showed that a problem that was first thought to be the result a serious flaw in the burner design was instead linked to a simple imbalance in the FGR distribution. This was a very important conclusion as the problem was easily remedied once it was identified, but failure to balance the FGR distribution would have doomed any attempts to eliminate the problem by modifying the burner design. ■



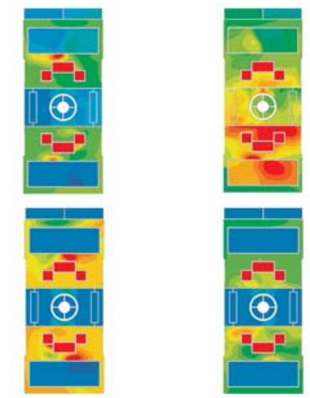
◄ Fig:03
Tangentially fired burner.



▲ Fig:04
Boiler, burners, windbox & ductwork.



▲ Fig:05
Ductwork Air/FGR distribution.

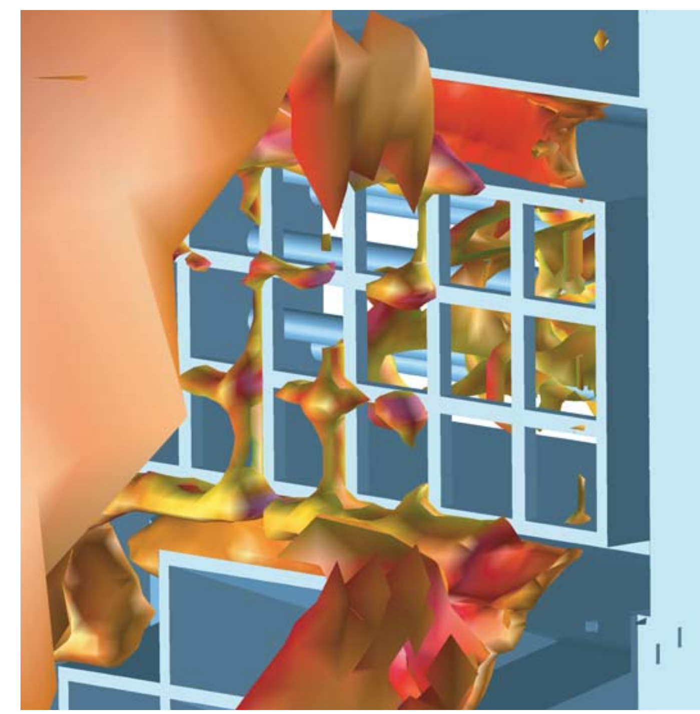


▲ Fig:06
FGR mass fraction at the face of burners 1– 4.

REFERENCES

*BMA - Brais, Malouin and Associates Inc 5450 Côte-des-Neiges, suite 600, Montréal, (Québec) Canada, H3T 1Y6 www.bma.ca

**Cerrey S.A. de C.V. Av. Republica Mexicana 300, San Nicolas de Los Garza, N.L. Mexico, C.P. 66450



▲ Fig:07
Luminous flame contour – upper Fuel-Lean port – Burner #3.

MORE INFORMATION **info@uk.cd-adapco.com**

Upcoming events

Trade Shows and Workshops

CD-adapco regularly participates in many global trade shows. To get the chance to talk in person with our experienced and friendly representatives, please make a note of the dates below. For more information please contact our events staff:

North America: Tara Firenze tara.firenze@us.cd-adapco.com

Europe: Maeve O'Brien maeve.obrien@de.cd-adapco.com

2007

Offshore Europe 2007
September 4-7, 2007
Aberdeen, ScotlandStand: 1155
http://www.offshore-europe.co.uk/

36th Turbomachinery Symposium
September 11-13, 2007
George R. Brown Convention Center, Houston, TX
Booth 819
http://turbolab.tamu.edu/turboshow/turbo.html

Deep Offshore Technology
October 10-12, 2007
Henry B. Gonzales Convention Center, San Antonio, TX - Booth 406
http://dot07.events.pennnet.com/fl/index.cfm

Fuel Cell Seminar
October 15-18, 2007
Henry B. Gonzales Convention Center
San Antonio, TX
Booth 406
www.fuelcellseminar.com

SPE
SPE (Society of Petroleum Engineers) Annual Technical Conference and Exhibition
November 11-14, 2007
Anaheim Convention Center
Anaheim, CA
www.spe.org/atce/2007

Nuclear Power International
(co-located w/ Power-Gen)
December 11-12, 2007
Booth 1334,
Donald E. Stephens Convention Center
Rosemont, IL
http://nip07.events.pennnet.com/fl/index.cfm

Power-Gen
December 11-13, 2007
Morial Convention Center
New Orleans, LA
Booth 5201
www.powergen.com

2008

All Energy
May 21-22, 2008
Aberdeen, Scotland
www.all-energy.co.uk

Power Gen Europe 2008
June 3-5, 2008
Milan, Italy
http://pge07.events.pen

ONS 2008
August 26-29, 2008
Stavanger, Norway
http://www.ons.no