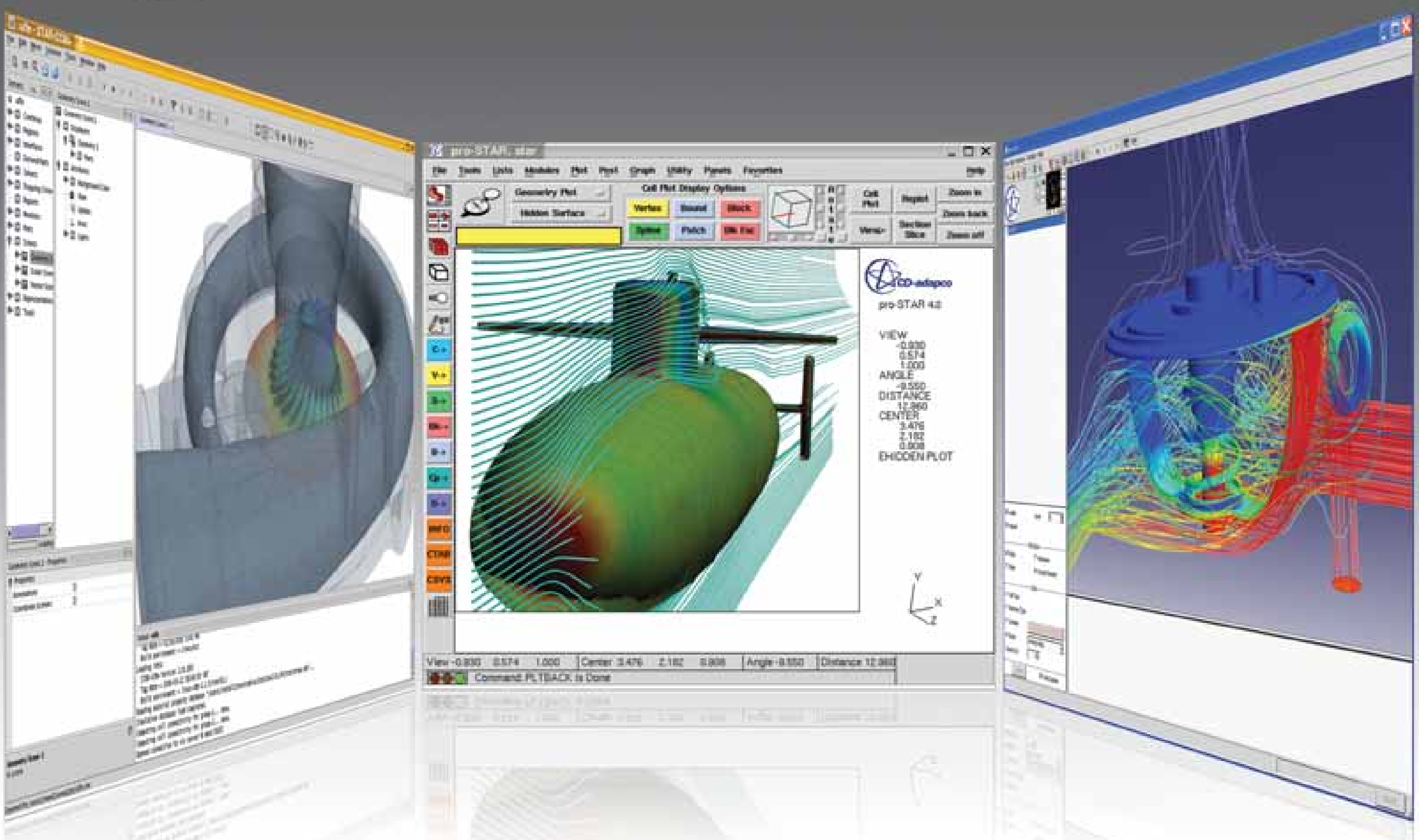
# dynamics #26




**STAR-CCM+ V2 • STAR-CD V4 • STAR-CAD Series**

# Three new releases
## One integrated solution

**Three more reasons to make CD-adapco your partner for success**

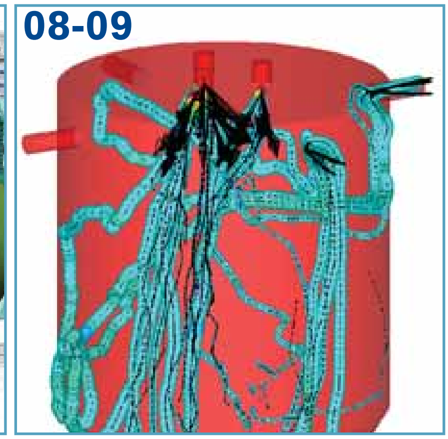
CD-adapco

**Snow, Water and Dirt »**
Simulation of Vehicle Soiling at Audi

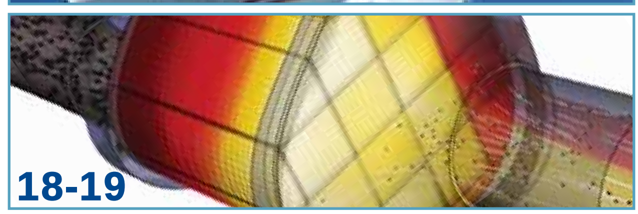
**Cool To The Core »**
Simulating Safety for High Temperature Reactors

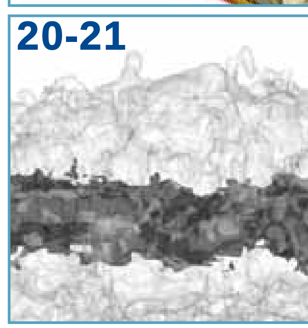
# Contents



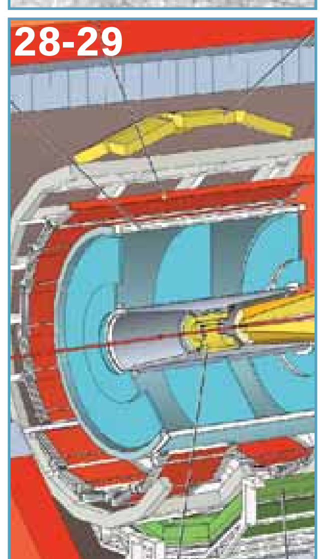



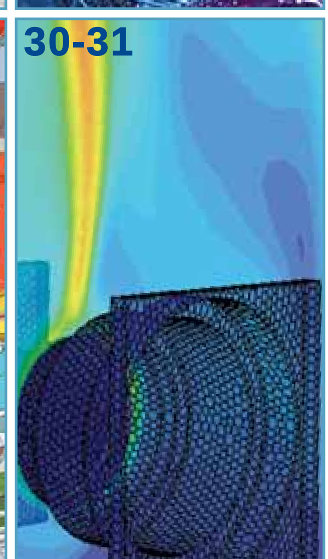








# CD-adapco continues to push forward

## delivering ever more effective tools to engineers and analysts across a wide range of applications

The first few months of 2006 have seen CD-adapco's most significant set of product releases since we first unleashed STAR-CD back in the late-eighties. We've therefore used the start of this issue to detail the benefits that STAR-CD V4, STAR-CCM+ V2 and STAR-CAD Series V4.02 bring to you as a partner of CD-adapco.

Highlights include: a new polyhedral solver for STAR-CD; our very latest surface preparation and meshing technology in STAR-CCM+; and the STAR-CAD Series; simulation for Design Engineers, equipped with the full power of parametric CAD.

One of the key objectives behind these releases has been to continue the convergence of our solutions: both in technology and inter-operability. This means that you have access to the best solutions – like a polyhedral solver – in STAR-CD, STAR-CCM+ and the STAR-CAD Series. Common technology also enables transparent exchange between products. For example, meshes produced in STAR-CCM+ V2 can be run just as easily in STAR-CD V4 as in STAR-CCM+ V2: the choice is yours!

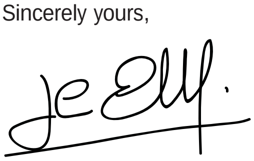
My own area of interest is utilizing these tools within the design cycle. Today, companies can obtain a clear competitive advantage by effectively introducing simulation into their product development cycle. Not just by adding CAE as an additional step in the design process, but by really assessing and adapting these processes to fully leverage virtual product development.

Our new releases help here too. For example, the STAR-CAD Series delivers tight integration with CAD and PLM systems: bringing CFD to the design department while significantly improving the CAD to simulation process for all. Again, our solutions fit around your process, so regardless of whether you use CATIA V5, Pro/ENGINEER, Unigraphics NX or SolidWorks as your CAD system, CD-adapco delivers the same added value.

The best demonstration of the power of CD-adapco's CAE solutions is of course, how our partners utilize them to add value in their engineering processes. As always, we're very grateful to all of them for sharing their applications of CAE and successes with CD-adapco. I'd like to thank them all: ABB, AREVA, ArvinMeritor, Audi AG, Dassault Aviation, FIAT Powertrain, QinetiQ, Webasto, Victoria Institute, Argonne National Laboratories and Sarov Laboratories.

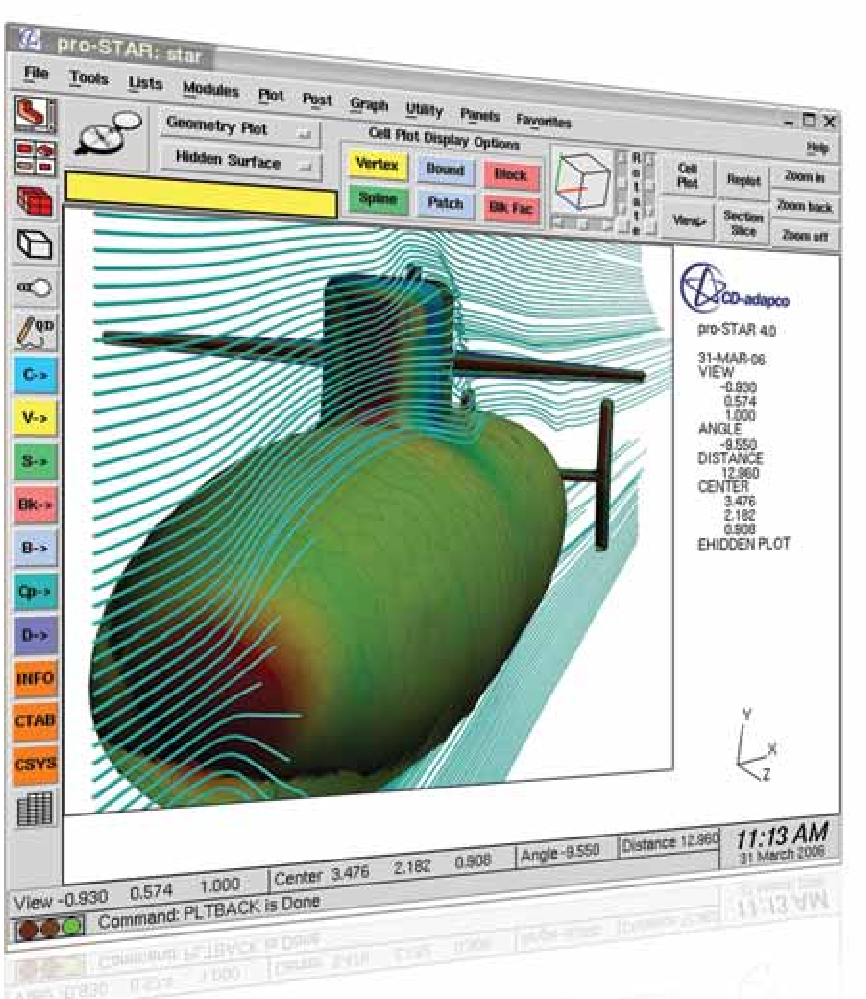
I hope you enjoy the issue.

Sincerely yours,



Jean-Claude Ercolanelli
Product Manager, STAR-CAD Series
CD-adapco

# STAR-CD V4:
## Available now!




Less memory, faster convergence, with fewer cells: STAR-CD V4, the latest version of CD-adapco's flagship CFD code, is available now.

STAR-CD V4 brings new state-of-the-art solver technology to STAR-CD users, without compromising the advanced modeling and complex physics capabilities for which STAR-CD is the undisputed market leader. By using the latest polyhedra-olsver technology, STAR-CD V4 delivers significant benefits in speed, robustness and usability.

### Benefits include:

- **Polyhedral cells:** STAR-CD V4 is fully polyhedra-enabled. Polyhedral meshes are as automatic to build as tetrahedral meshes, but have a number of other benefits: solving on a polyhedral mesh requires less memory, convergence is attained faster, and the same level of accuracy can be obtained using significantly fewer cells.

- **Improved Analysis of Free Surface Flows:** STAR-CD V4 introduces state-of-the-art free surface modeling, developing the COMET-based technology that is used extensively throughout the marine industry.

- **Extra Transient Options:** V4 extends STAR-CD's market leading capabilities for simulating transient flows with the addition of a new solution algorithm and a high-order discretisation scheme.

- **New Liquid Film model:** Reinforcing STAR-CD's leading spray modeling with the implementation of a state-of-the-art liquid film model.

- **Memory management:** STAR-CD V4 includes improved dynamic memory handling, further easing the solution of very large problems.

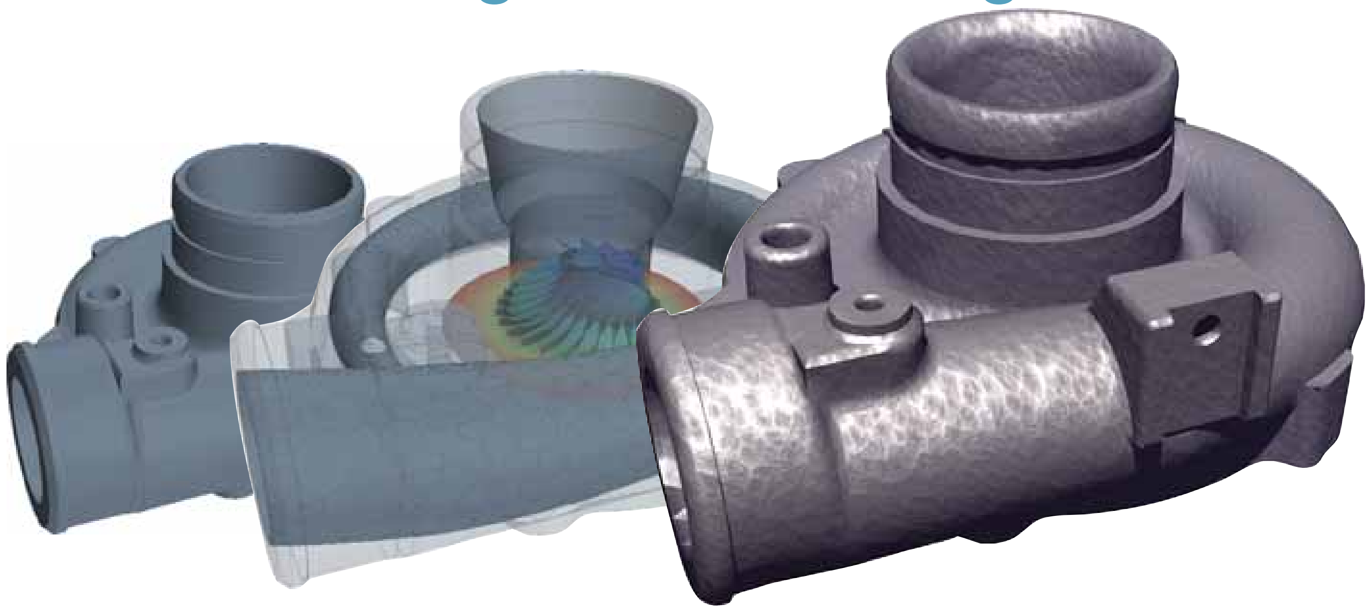
- **Inter-operability with STAR-CCM+:** STAR-CD and STAR-CCM+ now use a common file format, allowing transparent access to the full suite of CD-adapco's solutions.

Steve MacDonald endorses that this release represents a payoff for CD-adapco's decision to invest heavily in solver technology:

*"While other CFD vendors are content to simply build upon their existing aging software architectures, with STAR-CCM+ and now STAR-CD V4, we have achieved our aim of basing all of our solvers on the most advanced solver technology available."*

For more advice on the applicability of STAR-CD V4 to your cases please contact your local support office.

# STAR-CCM+ V2:
## Now with integrated meshing



STAR-CCM+ V2 is the second major release of CD-adapco's state-of-the-art CFD package and will now include full surface preparation and meshing technology, in addition to a host of new modeling capabilities.

Following the "from the ground up" philosophy that was the inspiration behind its original development, the mesh generation technology at the heart of the latest release has been rewritten from scratch. The resulting process, from surface import to post-processing, is more robust, more powerful and more automatic than ever.

Starting with a surface representation, a user can now perform a complete CFD analysis without ever leaving the STAR-CCM+ environment:

• **Surface Wrapping:** Automatic repair of imperfect or complex CAD using CD-adapco's world-leading surface wrapping technology. The surface wrapper is the most robust method currently available – guaranteeing a closed manifold surface at every wrap.

• **Surface Meshing:** High quality surface triangulations (suitable for polyhedral meshing) can be created automatically from any starting surface using STAR-CCM+ V2's surface remesher.

• **Multi-region meshing:** STAR-CCM+ V2 automatically creates conformal meshes across the boundaries between different regions, such as are required for simulations involving conjugate heat transfer, multiple reference frames or porous media.

• **Polyhedral Meshing:** Users of STAR-CCM+ have long been aware of the benefits of speed and accuracy offered by polyhedral meshes. Now, for the first time, high quality polyhedral meshes can be created from within the STAR-CCM+ environment.

• **Trimmed Cell Meshing:** Improved trimming technology enables trimmed cell meshes to be created from templates of tetrahedra and polyhedra as well as the traditional hexahedra.

The rapid development cycle of STAR-CCM+ has also ensured that V2 will be packed full of new modeling capabilities:

• **Vehicle Thermal Management:** STAR-CCM+ V2 includes integrated heat exchanger and fan models, which together with surface wrapping, make STAR-CCM+ an attractive tool for underhood analysis.

• **Free Surface Analysis:** STAR-CCM+ V2 introduces state-of-the-art free surface modeling, an improvement on the technology previously used extensively throughout the marine industry in COMET.

• **Cavitation:** STAR-CCM+ V2 also builds upon the cavitation technology that was extensively used and validated in COMET.

• **Radiation:** STAR-CCM+ V2 has improved thermal modelling capability with the inclusion of two radiation models: Surface to Surface and Discrete Ordinate.

• **Transition:** STAR-CCM+ V2 allows users the ability to specify location at which laminar to turbulent transition occurs.

• **Combustion:** STAR-CCM+ V2 offers a choice of EBU and PPDF combustion models.

For more advice on applicability of STAR-CCM+ V2 to your cases, please contact your local support office.

# STAR-CAD Series and STAR-CAD Gateways

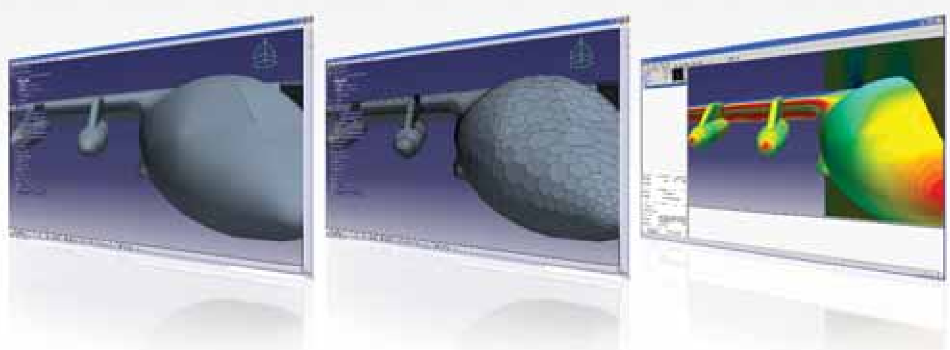
### Upfront flow and thermal solution integrated in your design process

## STAR-CAD Series

A recent CD-adapco survey of Engineering Managers, reiterated that the ability to “drive design from analysis” was a top priority in industry.  For many years CAE has added value through in-depth insight into isolated problems, but has struggled to keep pace with the fast moving design process.

With both designers and CFD specialists utilizing CAE, these enterprise-wide engineering tools enable teams to work together more efficiently. Engineers, CFD analysts, and product development management can collaborate on product designs; improving product performance, creating innovative designs and reducing development time from concept to final design.

STAR-CAT5



The integration of upfront engineering analysis doesn’t replace a company’s current design process, but rather facilitates and accelerates it with simulation capabilities and communication tools. This upfront approach to simulation provides design and professional engineers with quick but accurate feedback on product performance while products are being designed.

In conceptual stages, an engineer develops ideas and runs a “first-pass” analysis to evaluate alternative ideas and determine the part size and shape. This is accomplished within the environment of the CAD system. In the following more detailed design stages, an engineer can firm up dimensions and further refine the configuration, validating decisions with more detailed flow and thermal analysis. CFD analysts have immediate access to the project data and can further add their own input based on their experience and knowledge of the company’s products, as well as sign-off the preliminary analysis results.

STAR-CAD Series utilizes CD-adapco’s wealth of CAE know-how, by automatically using “best practice” settings and defaults. Companies can also utilize their existing problem specific knowledge, by having specialist CFD teams define simulation templates for designers.

CD-adapco has invested heavily in obviating these issues (e.g. CAD-embedding, surface wrapping and polyhedral meshing) to enable its partners to leverage CAE within the design cycle.  Key to this, are their CAD-embedded solutions: STAR-CAD Series.

Using STAR-CAD Series, an engineer developing parts in CAD can gain early insight into product behavior, and perform "what-if" simulations to achieve a superior product performance. When more detailed or advanced analysis is required, these models can be passed to the specialist CFD department: preventing duplication of work.

*“Engineers, CFD analysts, and product development management can collaborate on product designs; improving product performance, creating innovative designs and reducing development time from concept to final design”*

STAR-Pro/E

## STAR-CAD Gateways

STAR-CAD Gateways are an alternative solution environment for STAR-CD and STAR-CCM+. Similar to the STAR-CAD Series they enable STAR-CD and STAR-CCM+ users to set-up and run models entirely within the CAD environment. Users then benefit from having models associated with parametric CAD models. Two key benefits being: as the design changes in the CAD model, minimal additional work is required, all CFD settings are associated with the CAD parts and assemblies so new results are available at the click of a button; the CFD engineer can utilize the power of parametric CAD, allowing quick analysis of multiple geometries.

STAR-NX

*"The integration of upfront engineering analysis doesn't replace a company's current design process, but rather facilitates and accelerates it with simulation capabilities and communication tools"*

*"The user has access to the full-spectrum of capabilities of STAR-CD and STAR-CCM+, with the added benefit of associative CAD embedding"*

## Summary

At the core of the STAR-CAD Series and STAR-CAD Gateway products is a powerful and proven set of analysis and simulation, and database software running in the background and transparent to the user. Problem set-up, specific commands, and input/output requirements have been automated and integrated into the familiar CAD environments so users can take advantage of advanced technology without having to learn detailed operations and commands. Associativity with parametric CAD systems allows design engineers to freely modify parts without the need to redefine units, materials or boundary conditions, saving time and improving productivity.

CD-adapco upfront flow and thermal solutions can be used to solve a wide variety of engineering problems across industry. STAR-CAD Series and STAR-CAD Gateways handle internal and external fluid flows including conjugate heat transfer problems, rotating machines and porous media. Typical applications include (but are not limited to):

- flow equipment (pumps, nozzles, valves, and compressors, ...)
- fans and propellers,
- heat exchangers,
- household devices ( ovens, refrigerators, vacuum cleaners, ...)
- ventilation of air in rooms,
- HVAC ducts of automotive climate control systems,
- fans and propellers,
- vehicle aerodynamics and hydrodynamics

STAR-Works

For more information on the STAR-CAD Series and STAR-CAD Gateways please contact:
jc.ercolanelli@fr.cd-adapco.com

**The STAR-CAD Series includes:**
STAR-CAT5 for CATIA V5,
STAR-NX for UGS NX,
STAR-Works for SolidWorks
STAR-Pro/E for Pro/ENGINEER Wildfire.

**The STAR-CAD Gateway includes:**
STAR-CAT5 Gateway for CATIA V5,
STAR-NX Gateway for UGS NX,
STAR-Works Gateway for SolidWorks
STAR-Pro/E Gateway for Pro/ENGINEER.

# Tomorrow's engine simulation today

Alex Read, Gerald Schmidt and Erwan Gautier, CD-adapco

CD-adapco's position as a full-spectrum CAE solutions provider is built on a history of technology leadership: performing tomorrow's calculations today. STAR-CD was the first general purpose CFD code to handle unstructured meshes; it was the first to perform moving mesh calculations for complex industrial applications; it was the first to handle rotating and reciprocating machinery; and more recently, CD-adapco was the first of the mature CFD providers to deliver an arbitrary polyhedral mesh solution and to embed this into CAD and PLM systems.

This technology-leading position requires continuous innovation so that we are able to provide timely solutions for our customers. One such recent activity has been to run a complete V6 engine, shown in Figure 1, with intake and exhaust systems, moving pistons and valves and combustion.

fig:01

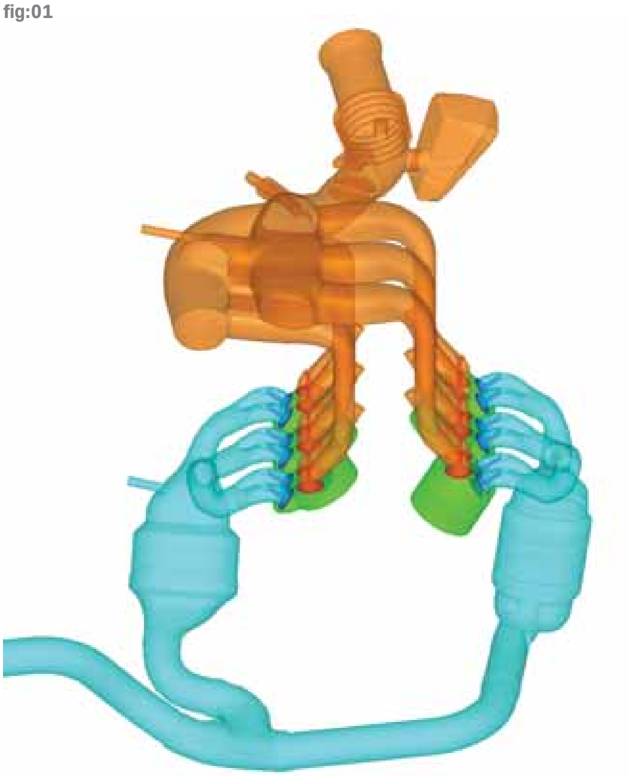
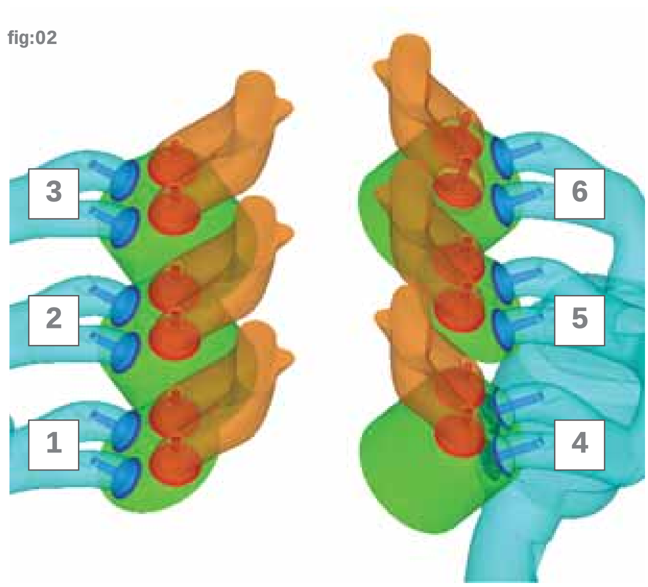


fig:02




Although such calculations can be considered embryonic today, as computer speeds continue to increase such calculations will become more common. Engineers will benefit from detailed insight that simply is not possible with current methods, and attain levels of accuracy that cannot be achieved with coupled 3D-1D simulations. There are a number of examples where current idealized methods are inadequate, including high-speed transient events, such as cylinder de-activation and re-activation and changes in variable geometry intake systems that may result in single cycle changes in air-fuel ratio and hydrocarbon breakthrough. In this calculation, the uniformity of charge mass from cylinder to cylinder was investigated, though follow-up studies for EGR distribution and other phenomenon are readily attainable.

However, even with the advanced capabilities of **es-ice**, such complex calculations - with thirty independent moving parts, combustion and multiple, simultaneous solution domains, are challenging to set-up and run. But, with the methodologies developed here going into **es-ice** and STAR-CD, CD-adapco is committed to delivering engineering solutions to even the most complex problems. There are enhancements to the specification of boundary conditions, initial flow field values, an idealized combustion model, and advanced post-processing.

The geometry is an engine in production, supplied by a major OEM. From the starting IGES data, pro-STAR's surface meshing module was used to generate the surface mesh. The volume mesh was then created in three parts. The inlet and exhaust systems were generated in pro-STAR's automatic meshing module. The mesh and movement of the pistons and valves was then dealt with in **es-ice**. This method has the significant advantage that **es-ice** will automatically deal with the set-up of the complex movement of the valves and pistons. Once this has been achieved for one cylinder, a simple "copy and paste" operation can be used to generate the multi-cylinder configuration. The final mesh contained 3.1 million cells, when all cylinders were at bottom dead centre.

During this first calculation, the effects of combustion were modeled with enthalpy sources, where the heat release was dependant on the mass of fluid in the cylinder. In future calculations this will be enhanced to use the ECFM-3Z combustion model. Figure 3 shows the temperature contours in the near-cylinder region, shortly after ignition has occurred in Cylinder 5.

fig:03

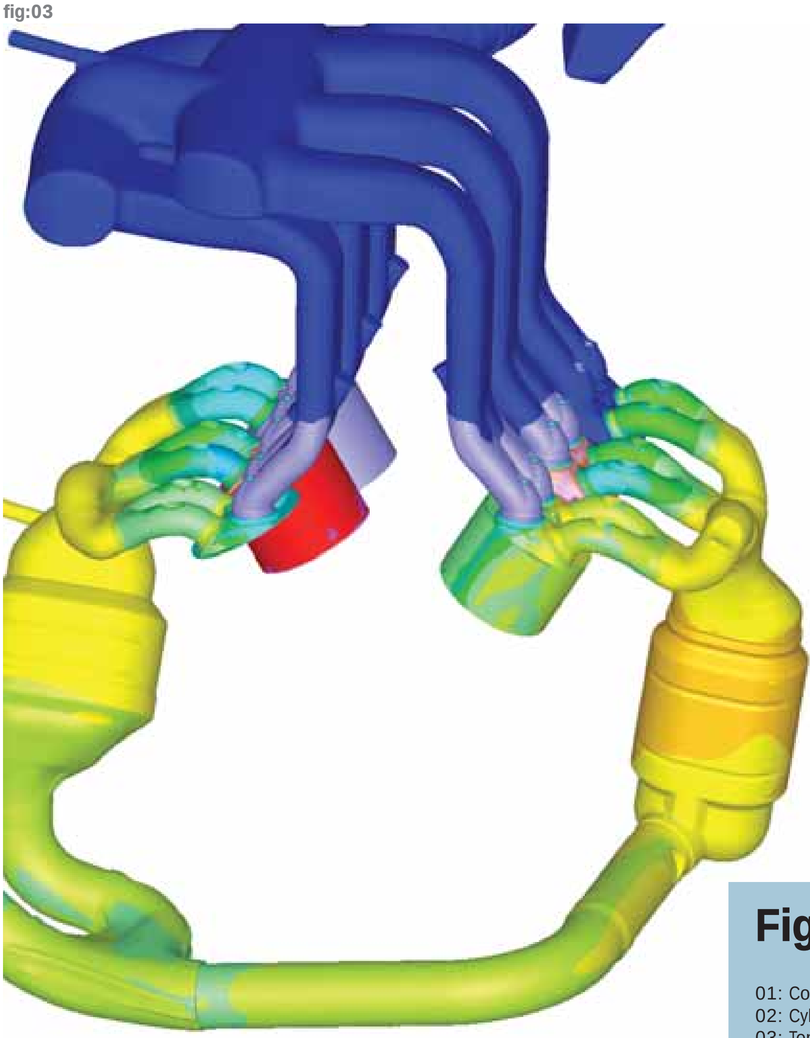
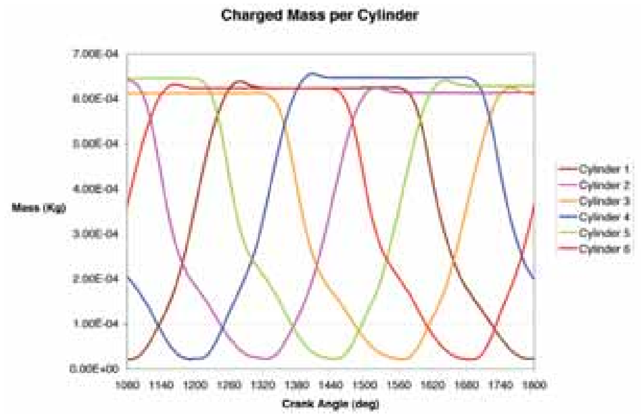


fig:04




One of the objectives of this analysis was to evaluate the charge mass in each cylinder, i.e. is there an even distribution of air going to each cylinder? An uneven distribution of air will affect the stoichiometry, combustion and ultimately engine performance. By performing such an analysis, a detailed understanding of the intake manifold system and its performance throughout the engine cycle can be obtained. Figure 4 shows the mass of air in each cylinder as a function of crank angle for one complete cycle. The charge mass distribution is near optimal, although Cylinders 4 and 5 get more air than the others.

Dr Richard Johns, CD-adapco's Director for the Automotive Sector, appreciates the importance of the engine community to CD-adapco and investigations such as these:

*"CD-adapco has a long track record of developing solutions "ahead of the curve" and the challenge for us now is to refine and package this technology so that it is available for routinely solving the most difficult of engine related flow and combustion problems. This is one of the most exciting developments to emerge for the engine CFD community for many years"*

At CD-adapco we pride ourselves on delivering full-spectrum CAE solutions. Our history tells us that by taking a lead in technology, as driven by our partners needs, we can become your CAE partner for success.

# Figures

01: Complete Engine Model.
02: Cylinder numbers.
03: Temperature contours, shortly after ignition in Cylinder 5.
04: Mass Charge per cylinder.

# Virtual Spray Drying

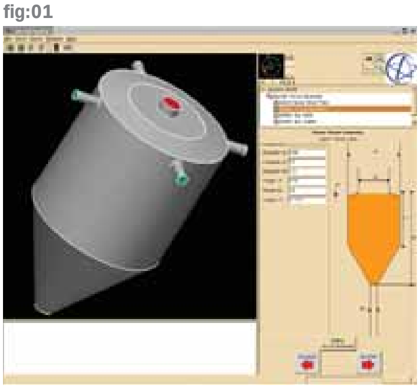
**Alex Read & Simon Lo, CD-adapco**

## Introduction

Throughout the Chemical, Pharmaceutical and Food industries, Computational Fluid Dynamics (CFD) is increasingly being embraced as part of the Virtual Product Development (VPD) process. This article details how CD-adapco has used its CFD expertise, in partnership with the application excellence of NIZO Food Research. The result is an easy-to-use, upfront CFD product for calculating the flow in spray driers: **es-spraydry**. **es-spraydry** enables Design Engineers - who are not CFD 'gurus' - to evaluate design critical parameters in a virtual environment, saving time and money.

Spray drying is used to turn a liquid into a powder: examples being detergents, herbal extracts, instant coffee and milk. These systems have design critical parameters which historically have been optimized through expensive and time consuming physical testing. The material residence time is paramount: it must be long enough to remove moisture but not so long as to cause over-heating. Optimizing the drying process is non-trivial, since it is dependant on a number of factors. These include: flow regime, temperature distribution in the system, humidity, initial particle temperature and moisture content, and particle drying characteristics.

fig:01



A key strength of CFD is the ability to carry out "what-if" and optimization analyses quickly. As an example, a dryer with a given set of feed conditions was considered. CFD simulations were carried out with the aim to find the optimum condition for the drying air. Key information of interest to the plant operator was extracted from the CFD results and presented in percentages of particles leaving the particle and air exits, and the particle conditions at these exits in terms of mean diameter, temperature and moisture content. From these results, the operator of the dryer can easily select the optimum operating condition, which allows him to achieve the desired product quality at minimum cost.

## Behind es-spraydry

Historically, companies have been reluctant to attempt such calculations since the initial investment – both in terms of personnel and hardware – was deemed prohibitively great. Complex, multiphase calculations were the exclusive domain of PhD educated specialist Engineers with large, expensive computers at their disposal. These requirements have recently become less and less stringent. Even relatively complex calculations can now be carried out on hardware obtainable on the high street for just a few hundred Dollars: such as the examples given here. But what about the know-how required performing such calculations?

CD-adapco has long identified industry's requirement for process specific tools: termed 'es' (Expert System) solutions. The model for these tools is that they encapsulate the CFD set-up, running and post-processing process, in an environment that is accessible to design engineers without specialist training. Thus, the requisite knowledge-transfer to get new starters up, running and adding value to the design process is greatly reduced.

The prerequisites to produce such a tool are twofold. First, you must be familiar with the design process in question. Second, you must be familiar with the modeling software and algorithms. This is where the partnership between CD-adapco and NIZO is so important.

CD-adapco is a CAE company – now in its 25th year – of unparalleled experience and expertise. NIZO has been researching spray drying since the 1950s and has extensive experimental, pilot plant and real plant operating experience. In this, experimental data used for validating CFD results is included Together, they have produced **es-spraydry**.

## The modeling process

In every CFD analysis it is necessary to go through several steps to build the computational model, carry out the computation and analyze the results as follow:

**1** Define the shape and dimensions of the dryer;

**2** Specify the inlet and outlet configurations;

**3** Define the atomizer and the spray characteristics;

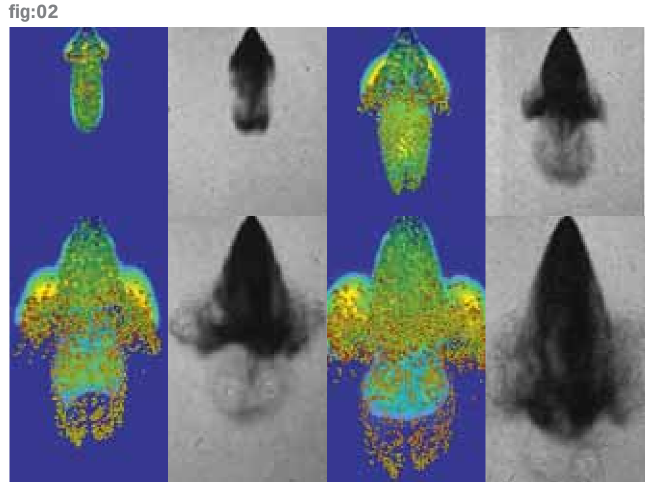
**4** Specify the flow rates of feed and drying air;

**5** Build a computational mesh;

**6** Run the solver to obtain a converged solution;

**7** Analyze the solution and produce a performance report of the dryer.

fig:02



This sequence of steps provides us with a well defined methodology for applying CFD in spray drying analyses. This methodology has been encapsulated, with the steps defined above automated, paying particular attention to ensure that the inputs and outputs are engineering values rather than CFD-speak. This was achieved through close collaboration with NIZO design engineers. Therefore, the end result from an **es-spraydry** simulation is not only detailed flow visualization images, but also an automatically generated report, providing engineers with the all important design metrics.

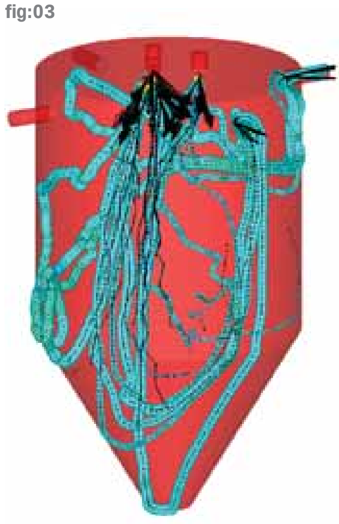
## Validation

Key to any modeling process is a validation stage. Here again, users benefit from CD-adapco and NIZO's years of experience and validation work. Just one example is the validation of spray models in **es-spraydry** and STAR-CD. In Figure 2, a direct comparison is made between STAR-CD results and Schlieren spray images at discrete time increments after injection in order to analyze the spray development and spreading. Due to the temporal delay of the swirl flow development after injection, the injection starts with a straight pre-jet and subsequently turns into a hollow cone spray. This effect is clearly seen in the visualization of analysis results.

Figure 2 also demonstrates the level of detailed understanding yielded by CFD analyses. As shown, such information can be obtained from experimental work, but only on highly idealized geometries and even then at great cost.

## Industrial example

fig:03



We have a simple dryer, shown in Figure 3, typical of that used in industry and evaluated by NIZO for producing dried milk powder. The outer diameter of the dryer is 9.5 m and an overall height of 14 m. There is one central air inlet and four circular air outlets. A rotary wheel atomizer spinning at 2600 rpm is used for the atomization of the feed.

The droplets produced are assumed to have a log-normal size distribution and a mean diameter of 100 micron with a geometric standard deviation of 0.6.

The dryer is required to process a feed at a rate of 4990 kg/h, a temperature of 25ºC and a solid content of 55% w/w (weight/weight). Drying air is to be supplied at 185ºC with a moisture content of 1%. We need to find the optimum air flow rate which will satisfy the following requirements:

**1** 90% of particles exit the bottom particle exit.
**2** Mean particle moisture content is less than 9%.
**3** Mean particle temperature is less than 100ºC.

fig:04

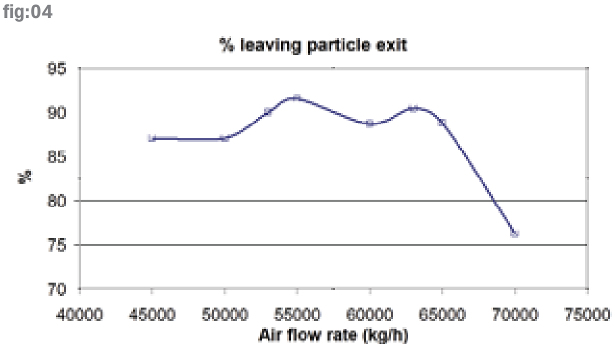
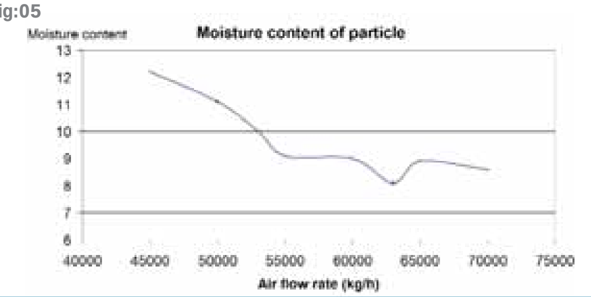



fig:05

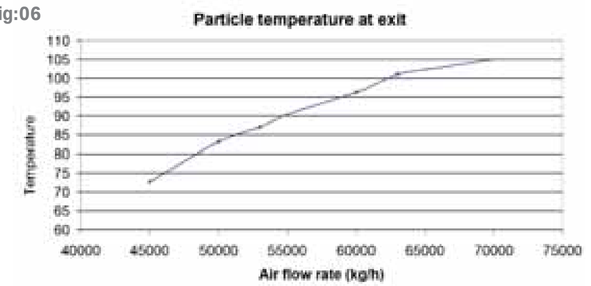



From the **es-spraydry** results we monitored the particle conditions exiting the dryer at the particle and air exits. Several **es-spraydry** calculations were performed with different air flow rates.

The results were further analyzed against the operation requirements, shown graphically in Figures 4 to 6. From the analysis we would select an air flow rate of 55,000 kg/h for minimum operating cost in terms of supplying the drying air.

The **es-spraydry** model used in the analysis has 20902 cells. 100 parcels of droplets were used to represent the spray. Converged solutions for all cases were obtained within 200 iterations. The CPU times for the cases range from 4000 to 7555 seconds on an Intel P3, 1.2GHz computer.

fig:06




## Conclusion

Historical barriers to running design-enhancing CFD simulations are quickly becoming a thing of the past. Through dedicated CFD software such as **es-spraydry**, the CFD analysis process has been simplified and automated. Close collaboration between CAE code vendors (CD-adapco) and experienced 'in the field' organizations (NIZO) has yielded an easy-to-use upfront CFD analysis tool, capable of adding value to any spray drying design process.

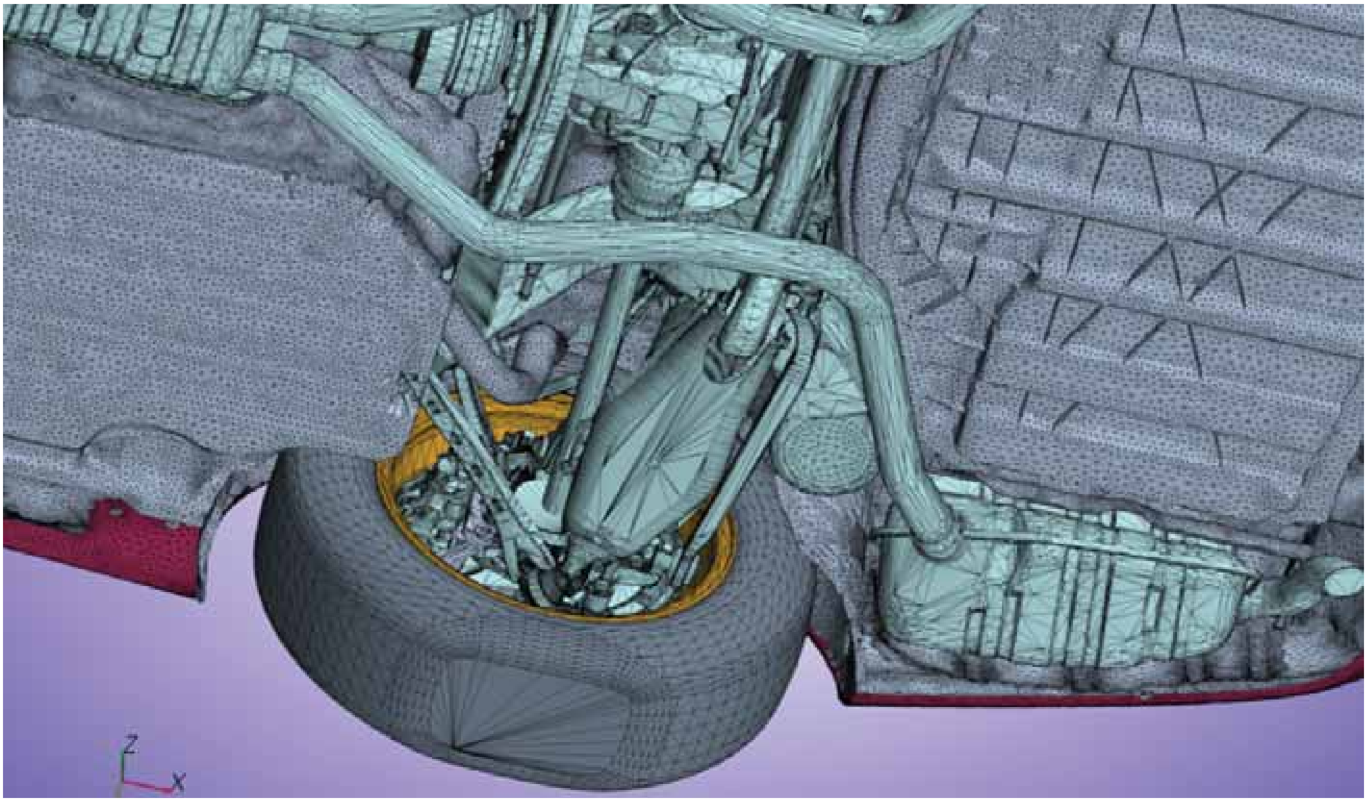
With this simulation tool it is now possible for the spray dryer operators to carry out systemic analyses of their dryers to ensure they operate at optimal conditions.

# Figures

01: **es-spraydry** screenshot.
02: Comparison of STAR-CD injection simulation vs. experiment Schlieren spray visualization.
03: Spray dryer with computed particle tracks
04: Percentage particles leaving dryer at particle exit
05: Average moisture content of particles leaving particle exit.
06: Average temperature of particles leaving particle exit.

# That's a wrap!

**Stephen Ferguson, CD-adapco**




Dealing with the consequences of imperfect CAD has long been the biggest bottleneck in CFD, and was, until recently, the single largest obstacle in implementing flow-simulation in the design process.

While CD-adapco can justifiably claim to lead the market in CAD-embedded CFD, there still exists a group of problems for which the CAD geometries are too big or too complex to be handled from within the CAD environment. These problems typically involve large assemblies of complex CAD parts, many of which contain more detail than is necessary for a CFD calculation.

Prior to the arrival of CD-adapco's automatic surface wrapping technology, the only viable alternative was manual repair; a process that is both time-consuming and expensive. For a typical case, producing a meshing quality triangulated surface from complex or dirty CAD could take days or even weeks, consuming up to 50% of the engineering budget of a CFD project, while tying up a highly qualified CFD engineer in monotonous manual labour.

A typical example is an underhood analysis (the application for which the surface wrapper was originally developed). A typical underhood geometry can consist of several hundred individual components. In the early stages of the design – the time at which CFD analysis is most critical – many of these components will touch or intersect. Before the advent of surface wrapping, extracting a single surface in this type of situation was extremely time consuming or sometimes even impossible.

CD-adapco's key motivation in developing surface wrapping technology was to eliminate manual repair from the CFD process altogether, guaranteeing a faster route from CAD to flow-simulation, and freeing CFD engineers to concentrate on engineering analysis.

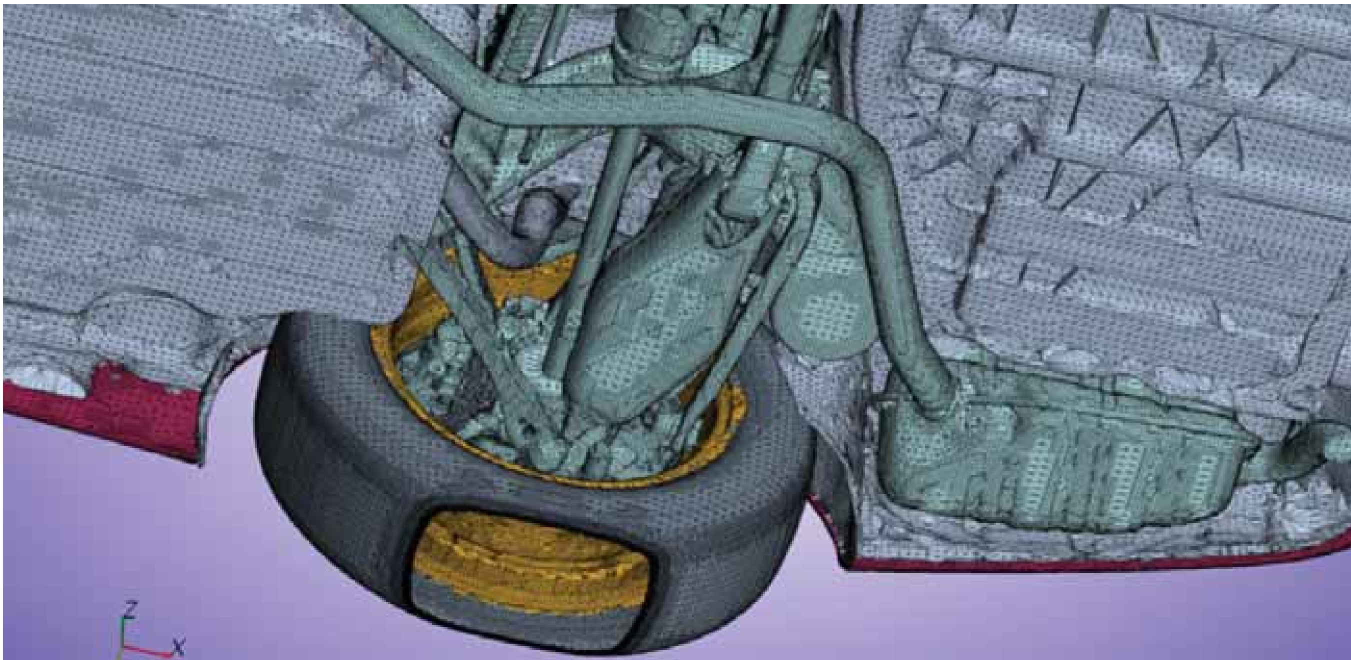
The surface wrapper works by 'shrink-wrapping' a high-quality triangulated surface mesh onto the geometry, closing holes in the geometry and joining disconnected and overlapping surfaces. The surface wrapper quickly calculates the wetted surface of the geometry, automatically discarding surfaces that are outside the flow-domain, instantly eliminating unnecessary detail.

Although the emphasis of the surface wrapper is on fully automatic repair, the tool is also fully customizable, allowing users to specify the level of resolution on a surface-by-surface basis if necessary, or using volume regions to specify larger areas of refinement. All size specifications are relative to a global base size, so that once set up, the wrapped surface can be fine tuned by just altering a single parameter.

The wrapper can also be used to quickly and automatically extract flow volumes from complex solid parts. Users have the option of wrapping from the outside of the geometry, or the inside. For complex geometries seed points can be defined within a geometry in order to prescribe which internal volume the wrapper will extract.

Importantly the surface wrapper respects the fidelity of the original surface. Unlike other surface-wrappers, CD-adapco's wrapper automatically respects the sharp edges and corners of the original model, as well as any other "feature curves" that the user chooses to prescribe.

Before wrapping, the user has the option of defining seed points inside and outside of the geometry, which allow the surface wrapper to automatically identify and display any leakage paths between the internal and external geometries. Having identified the size and the location of any holes, the user is able specify a closure tolerance which the wrapper uses to close any leakage paths and generate a single continuous wetted surface.



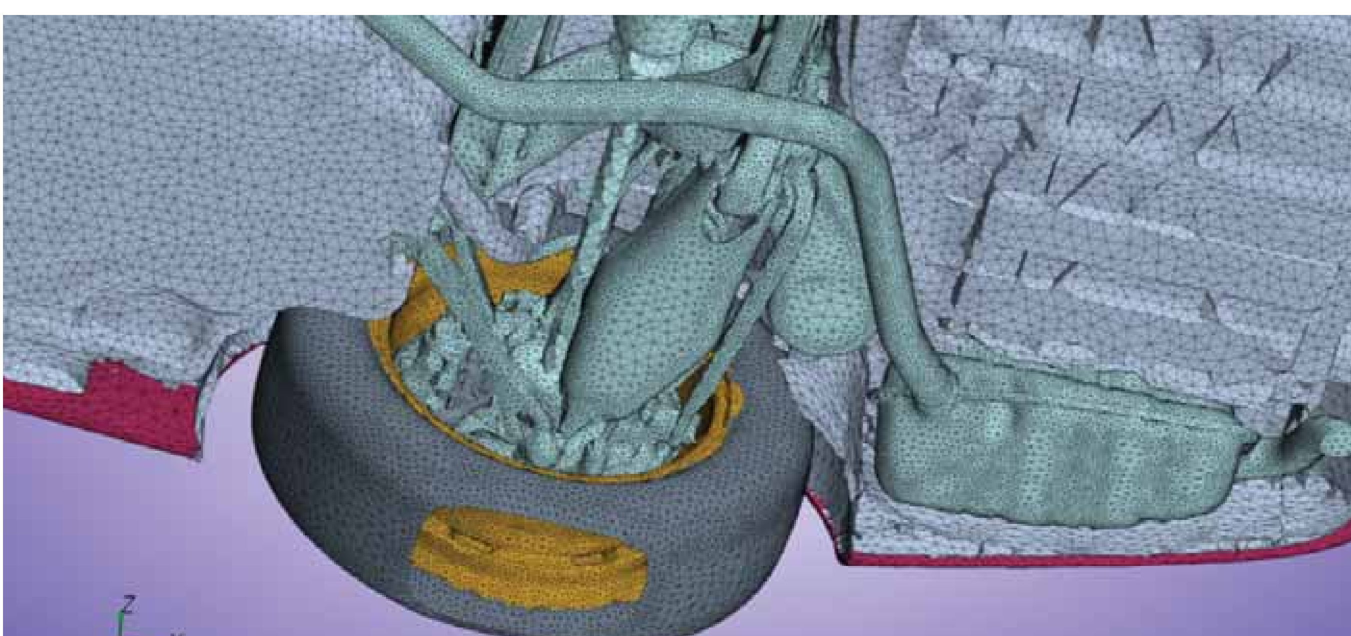

The idea of surface wrapping is not new (CD-adapco first implemented a surface wrapper in pro-STAR in 2004), however the surface wrapper implemented in STAR-CD V4 and STAR-CCM+ V2 is based on entirely new technology that is unique in one significant respect: CD-adapco's new surface wrapper guarantees a closed manifold surface at every wrap.

The surface wrapper is both speed and memory efficient, producing a high quality wrapped surface of 10 million triangles from a 5 million triangle input mesh typically takes less than 30 minutes and requires less than 3Gb of memory. The surface preparation time for an underhood analysis has been reduced from several weeks (using manual repair) to under a hour using CD-adapco's surface wrapping technology.

The path from "imperfect" CAD geometry to CFD mesher has never been easier or more automatic.

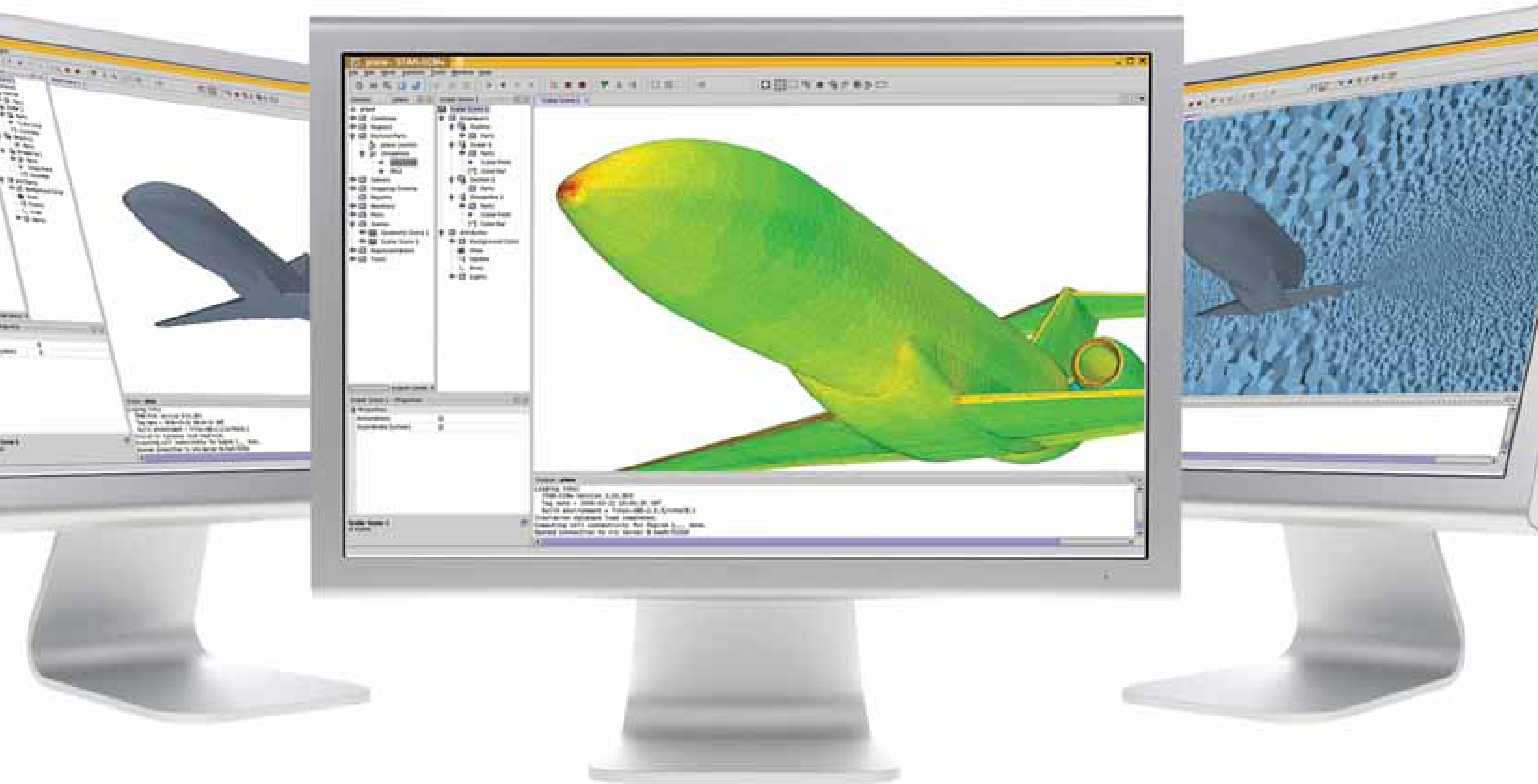
Major automotive companies such as Daimler Chrysler are already benefiting from huge time saving in the design of vehicle under-bonnet, external aerodynamics and passenger compartments. Daimler Chrysler's Walter Bauer described CD-adapco's surface wrapper as:

*"The most automatic approach that exists. It has saved us weeks of clean-up when carrying out our air flow studies and provides us with significant time and cost savings."*



# CAD embedded CAE simulation enables design for Six Sigma (DFSS)

**David L. Vaughn, CD-adapco**



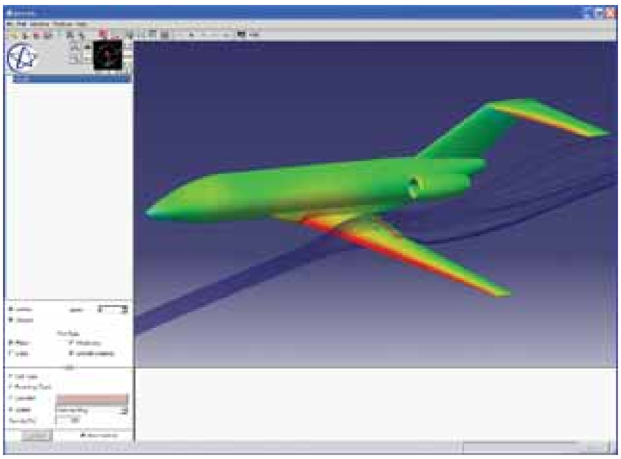
Whether it is lean manufacturing, six sigma, or some combination, quality improvement is now a primary focus of virtually every manufacturer in the aerospace industry. And now that the waste is squeezed out and quality is designed into manufacturing processes, visionary corporations are looking upstream toward the engineering of the product for further gains in quality. One of the keys to ensuring manufacturability and efficient integration of component systems is the implementation of high fidelity Computer Aided Engineering (CAE) software during the early design stages.

One of the more popular methods for addressing quality improvement during the design process is known as Design for Six Sigma (DFSS). Like most quality improvement acronyms this process has different meanings for different organizations but, from a "big picture" perspective, DFSS simply means designing products that meet the customers' requirements while maximizing the efficiency and robustness of the manufacturing and/or integration process. With modern production being centered on CAD/PLM data, it is imperative that engineering simulations, which predict the product performance, be tied directly to the same CAD/PLM data to ensure consistent quality. This also allows a more direct use of DFSS statistical tools for measuring quality.

CD-adapco realized the benefits of upfront computational simulation and were the first to bring to market a true CAD embedded computational fluid dynamics (CFD) solution. The STAR-CAD Series of products embed state-of-the-art CFD technology directly in all the major CAD systems which allows engineers to perform full CFD simulations (pre-processing, meshing, and analysis) from within the CAD system.

In the past the product definition was simply tossed over the wall for detailed design and manufacturing. If production constraints were considered in the early design stages, they were typically considered as "rule-of-thumb." This lead to major headaches and unexpected expenditures once production actually began. More recently CAD/PLM systems have helped, but there is often still a wall between engineering analysts and designers, which prevent the direct inclusion of manufacturing constraints in the product design cycle. Essentially analysts and designers must perform wasteful iterations to relate manufacturing constraints with performance goals.

Now with CAD embedded simulation, manufacturing constraints can be considered directly when engineering analysts perform design studies. More often these studies are being executed as



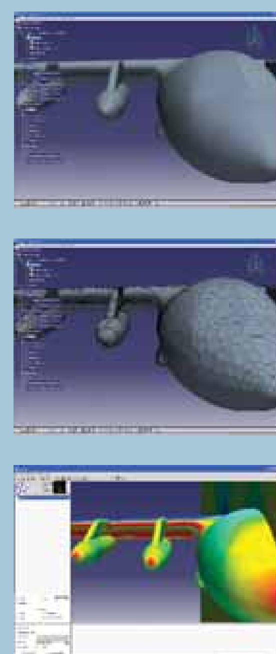
multidisciplinary design optimizations (MDO), which allow a six sigma approach to the product design. With direct access to parameterized CAD models, these MDO studies are much easier to setup, and manufacturing constraints are included in the formulation of the optimization problem. This results in the seamless inclusion of manufacturing requirements from the very inception of the product design.

Of course the benefits of DFSS via CAD embedded analyses extends beyond the initial manufacture of the product. Component reliability is a major contributor to the total lifecycle cost, and nowhere is this truer than in the aerospace industry. Let's look at the simple example of a flap actuation system. Using a CAD embedded CFD product such as STAR-CAT5 the static aerodynamic loads on the actuation system are easily obtained early in the design process. But perhaps of more value in terms of lifecycle costs is the early consideration of dynamic loads throughout the range of flap motion by integrating embedded CFD, FEA and multi-body dynamics simulations into a multidisciplinary design optimization. The entire actuation and flap track system can be optimized for lifecycle costs during early engineering stages.

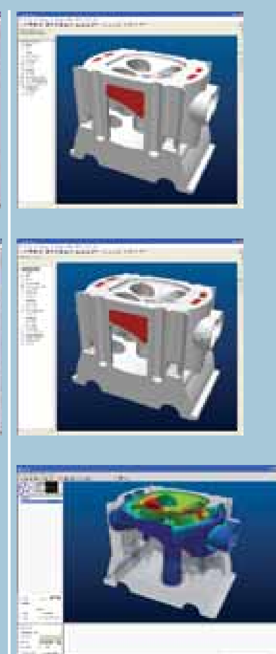
From the CAE perspective, there are many benefits associated with the STAR-CAD Series of CAD embedded CFD tools e.g., ease of use, superior accuracy, efficient numerical algorithms, etc. None of these, however, affect the bottom line more directly than the enabling of DFSS. CD-adapco refers to the STAR-CAD Series of products as "up front" solutions meaning that they bridge the gap between CAD and CAE analysis. Looking at the bottom line, however, perhaps the more meaningful metaphor is the gateway between quality improvement and upstream engineering design. It may be cliché to say that CD-adapco is breaking down barriers in the development and use of CAE technology, but the truth is - they've been doing it for over 25 years. ■

## The STAR-CAD Series

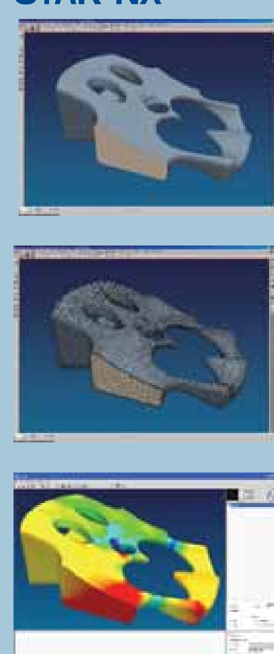
### STAR-CAT5



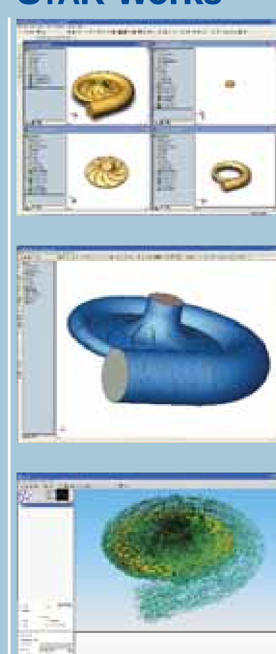
### STAR-Pro/E



### STAR-NX



### STAR-Works



# STAR-CAT5 helps to design "coolest business jet ever"

Stephen Ferguson, CD-adapco


Recently cited by Forbes magazine as "the coolest business jet ever", the Falcon 7X is the latest in a long line of high-class executive jets from Dassault Aviation. As well as being cool, the Falcon 7X is also revolutionary – being the first civil aircraft ever to be designed using virtual rather than physical prototypes. Designed entirely using Dassault Systèmes CATIA Product Lifecycle Management (PLM) tools, every component on the Falcon 7X was subjected to rigorous CAE analyses.

Dassault Aviation is committed to providing the very highest standards of comfort for Falcon 7X passengers. The aircraft features Dassault Aviation's breakthrough Environmental Control System, that maintains a constant cabin temperature and an adequate level of pressurization as the aircraft passes through extremes of external conditions. The Environmental Control System also manages safety critical aspects such as wing de-icing and avionics cooling. Central to the operation of the Environmental Control System is the mixing jet pump that mixes the air sent to the cabin from bleeds on different stages of the engine compressors. The pump's job is to maintain the pressure and temperature within the cabin air-conditioning system independent of the regime in which the engines are operating.

Dassault Aviation's current CAE process utilizes one-dimensional network analysis for sizing the mixing jet pumps. Although the process works well across a wide range of operating conditions, under certain extreme conditions – such as supersonic flow in the mixer – the accuracy of the network model was often found wanting due to the complex flow within the pump. In order to remedy this, Dassault Aviation adopted CD-adapco's STAR-CAT5 CFD software to better characterize the performance of the pump under extreme conditions. STAR-CAT5 is the first industrial-strength CFD software to be fully embedded within the CATIA V5 PLM system.

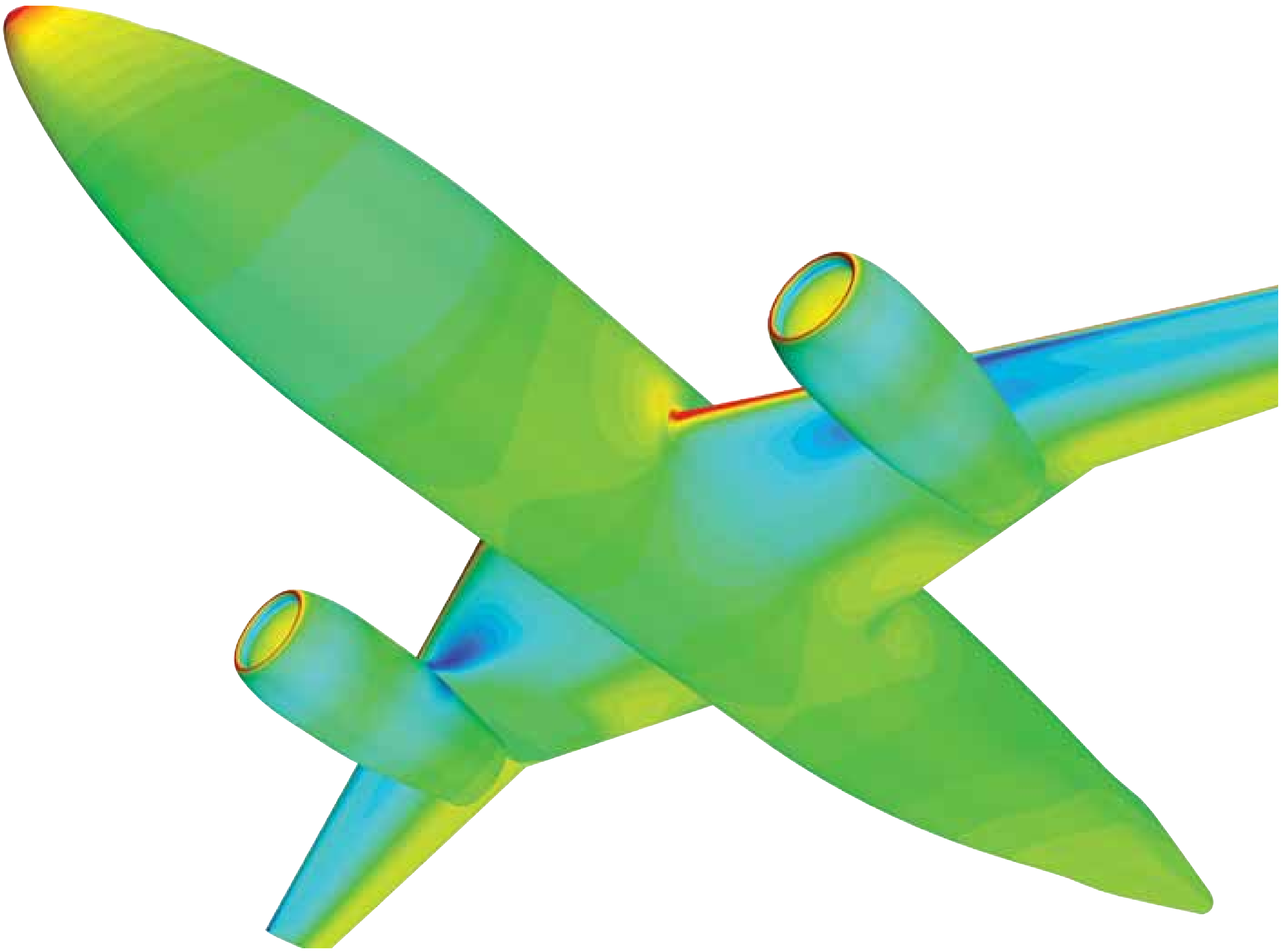
The aim of the CFD analyses was to numerically simulate the flow within the jet pump for different flight conditions and to compare the results with experimental data. The goal was to understand the phenomena that occur in the jet pump for each regime (subsonic, transonic or supersonic flow) in order to be able to refine the dimensions of the jet pump that would give an optimized mixing ratio for the downstream circuit.

The advanced capabilities of STAR-CAT5 enabled meshing of the real geometry of the jet pump without exiting CATIA V5. The mesh was exported to be optimized within STAR-CCM+, CD-adapco's next generation CFD solver. In order to obtain the adapted refinement level, STAR-CCM+'s advanced capabilities were used to cut, combine, or fuse different parts of the mesh. Several parts of the mesh were created with STAR-CAT5 and were cut at the desired place with STAR-CCM+, then all the parts were combined and the interfaces were joined. Finally the resulting mesh was adapted in terms of size and shape for all areas of the flow.

The results of the STAR-CCM+ calculations were compared with laboratory tests and have very good agreement for all studied cases of flow, even for the more complex ones. These very high quality results now allow Dassault Aviation to optimize the in-house 1D code for extreme conditions of flow and encourage Dassault Aviation to continue in this direction in order to study more complex geometries. ■

# QinetiQ and CD-adapco predict aircraft drag with STAR-CCM+

**Alex Read, CD-adapco**



QinetiQ and CD-adapco recently announced their joint participation in the 3rd American Institute of Aeronautics and Astronautics (AIAA) Drag Prediction Workshop (DPW), using STAR-CCM+.

The workshop is a meeting of minds in the aerospace industry, to assess the state-of-the-art in CFD methods for aircraft drag prediction. Participants are required to perform "blind" predictions on generic aircraft configurations, and ultimately compare against experimental data: making it an ideal proving ground for "next generation solutions" such as STAR-CCM+.

Matt Milne, from QinetiQ's Aerodynamics and Aeromechanical Systems group is confident of success, *"Over the past two years, we have used STAR-CCM+ to predict the aerodynamic performance of both commercial and military aircraft. In particular, we ran cases from the 2nd Drag Prediction Workshop (originally held in June 2003) and obtained excellent results compared with experiment."*

*"STAR-CCM+ is just one example of CD-adapco's commitment to delivering solutions that span the aerospace industry. Investments in new technology, like polyhedral meshing, and partnerships with key industry figures, such as QinetiQ, allow us to rapidly identify and meet the industry's needs."* says David Vaughn, CD-adapco's Director for the Aerospace and Defense Industry Sectors. *"The combination of QinetiQ's expertise in aerodynamic simulation, with CD-adapco's proven software solutions, forms a partnership that is win-win."*

QinetiQ participated in the 1st and 2nd Drag Prediction Workshops with great success. These Workshops provide an excellent opportunity for QinetiQ to showcase its expertise in aerodynamic performance prediction. This expertise has been developed and extensively validated over the last 20 years on projects funded by the Ministry of Defence and civil aerospace customers in the UK. In working closely with CD-adapco, and providing key input into the development of next generation CFD tools like STAR-CCM+, QinetiQ looks to keep its place at the leading edge of aerodynamic performance prediction for the foreseeable future.

For further details, watch this space! We'll be updating Dynamics readers on the project later on in the year. ■

# STAR-CD at Fiat Powertrain Technologies

**Angelo Rosetti, Fiat Powertrain Technologies, Arese, Italy.**

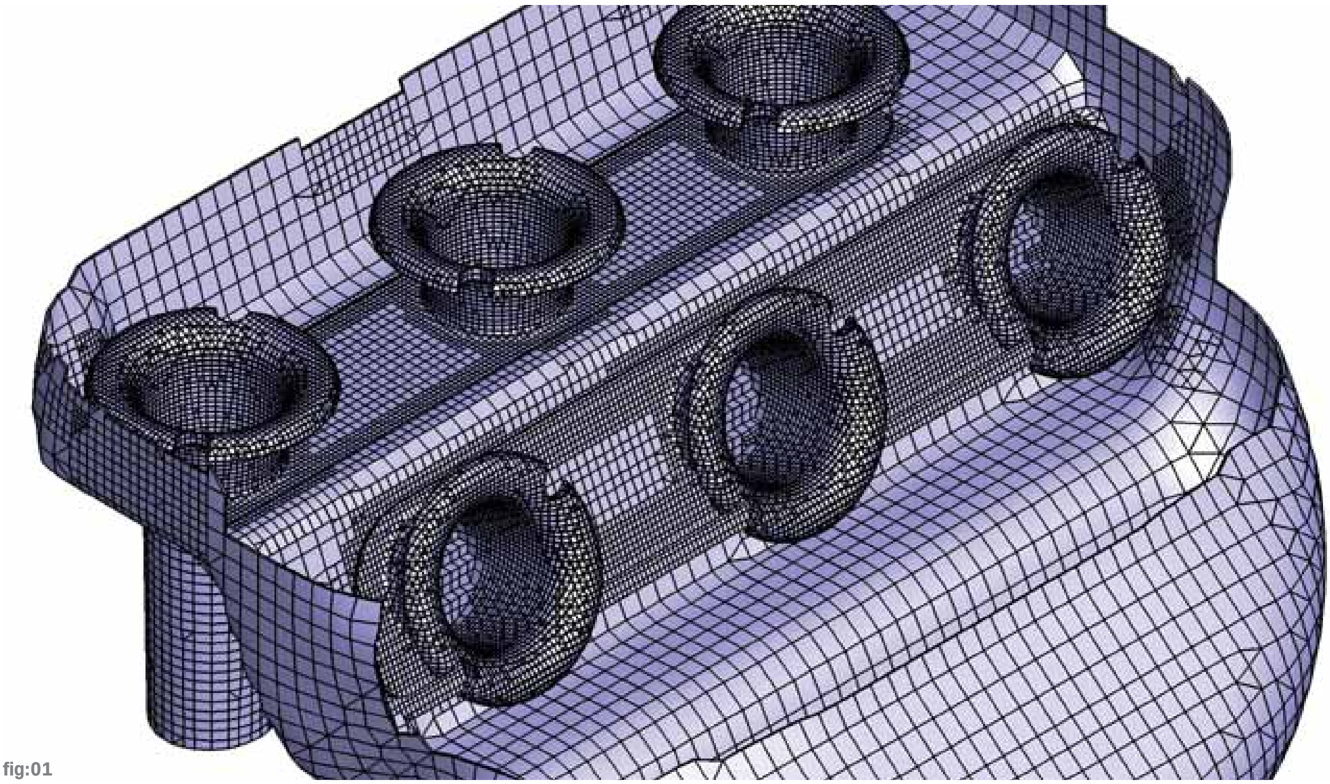


fig:01

STAR-CD is used to support the engine design process of Fiat Powertrain Technologies (FPT) in Arese. Advanced CFD calculations are usually performed, such as 1D-3D coupled analyses, cylinder head and block water jacket, manifold analyses and intake runner cold flow. Some spray-injection and combustion simulations have been performed with the latest multi-hole injector configurations. Here 1D-3D coupled and water cooling jacket analyses will be described.

## Manifold Design

1D-3D Coupled analyses are used to design manifolds. The one dimensional code GT-Power v6.1 and STAR-CD are used on Linux RH Enterprise3 machines.

Simulations are usually performed in full load condition, for the maximum power rpm and sometimes for the maximum torque rpm determination. Seven engine cycles are run, six of them to reach convergence and the last one to write results. A 0.5 degree crank-angle-step is usually set, to get the best compromise between analysis speed and accuracy.

In order to provide faster results to the design engineer, a particular cluster has recently been built, made of only one Linux machine and "n" number of WM-Vare Linux virtual machines mounted on Windows technical workstations. Each of these technical workstations have 2 processors; when you want to run a simulation with the cluster you simply have to "turn on" the virtual machine from Windows and you get a Linux machine. Then you have only to decide how many processors you want to add to the mini-cluster (only one or both of them). In that way you can work on a machine and use it with one processor with Windows and use the other processor for the Linux cluster: so you can decide how many machines to use among those available on your office! A Linux cluster has been preferred because of **es-ice** and ICE calculations can be run.

fig:02

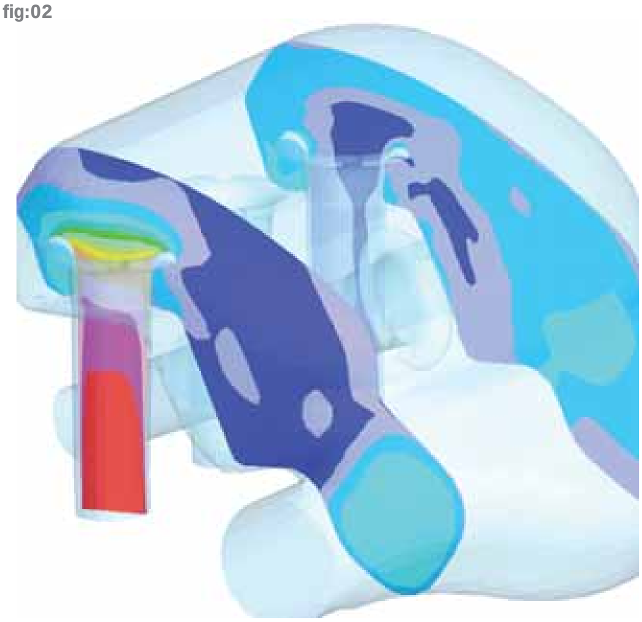


fig:03

The results obtained are analysed and discussed together with the designer. Velocity field, pressure, temperature, pressure waves, Mach number, flow separation and mass flow rate balance between cylinders are the quantities usually observed. A complete analysis for one manifold solution (meshing process, running, post process and report building) takes about 4 to 5 working days, depending on the mesh complexity. Figure 1 shows the mesh of a six cylinder engine intake manifold, the mesh contains about 400K cells, with one extrusion layer of about 1mm. The Standard K-ε turbulence model has been used. Figure 2 shows an instantaneous velocity field of some runner sections; Figure 3 shows the mach number of only one runner section.

## Water Cooling Jacket Analysis

The water jacket analyses performed at FPT are usually part of the complete thermo-structural analysis of the whole cylinder head and block. From the CFD code the wall heat transfer coefficients are passed to the FEM code in order to make the thermo-structural calculation. STAR-CD is then used to control all the water flow distribution inside the head and the block; furthermore the gasket hole position and dimension are optimized in order to keep the metal temperature regulated for all parts of the engine.

The standard calculation is usually represented by a steady run in maximum power condition. The mass flow is the inlet condition and the exact coolant properties are specified.

Both isothermal and non isothermal analyses can be performed and thanks to the flexibility of STAR-CD, boiling effect are taken into account, using an appropriate user subroutine. The Rohsenow nucleate boiling model has been used to correct the one-phase wall heat flux in case of critical wall temperature values.

fig:04

fig:05

fig:06

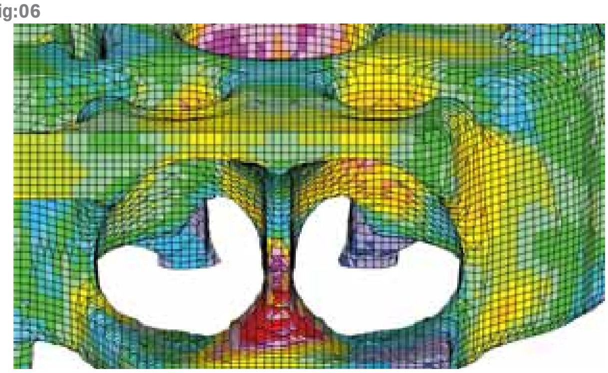
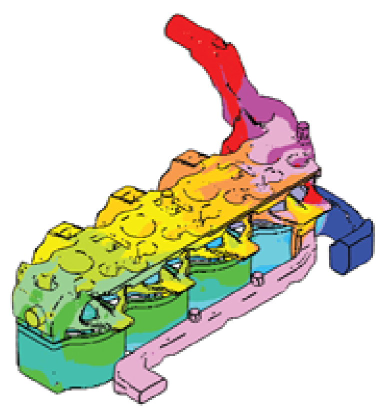


Figure 4 shows a complete mesh of the water cooling jacket of the cylinder block and head for a four cylinder engine. The mesh consists of about 800K cells. The coolant enters from the block and goes out of two outlets: one going to the radiator and the other to the interior compartment. The position of the gasket holes, that connect the cyl. block and head, are very important for the whole thermal balance of the engine: the role of the calculation is to best fit their position.

fig:07



As known, the exhaust valve bridge in the cylinder head is the critical point for monitoring for the thermal behaviour of the engine and an organized coolant motion without stagnation on the exhaust side is the starting point for water cooling jacket design.

Cylinder head pressure, wall heat transfer coefficients and temperature can be observed respectively in Figure 5, 6 and 7.

## Conclusions

STAR-CD is the CFD simulation software used in the engine design and analysis department of FPT, Arese Site. The article is an overview of how coupled 1D-3D and water cooling jacket CFD analyses are typically performed and clearly shows the fundamental role of the simulations in the design process of modern automotive engines. ■

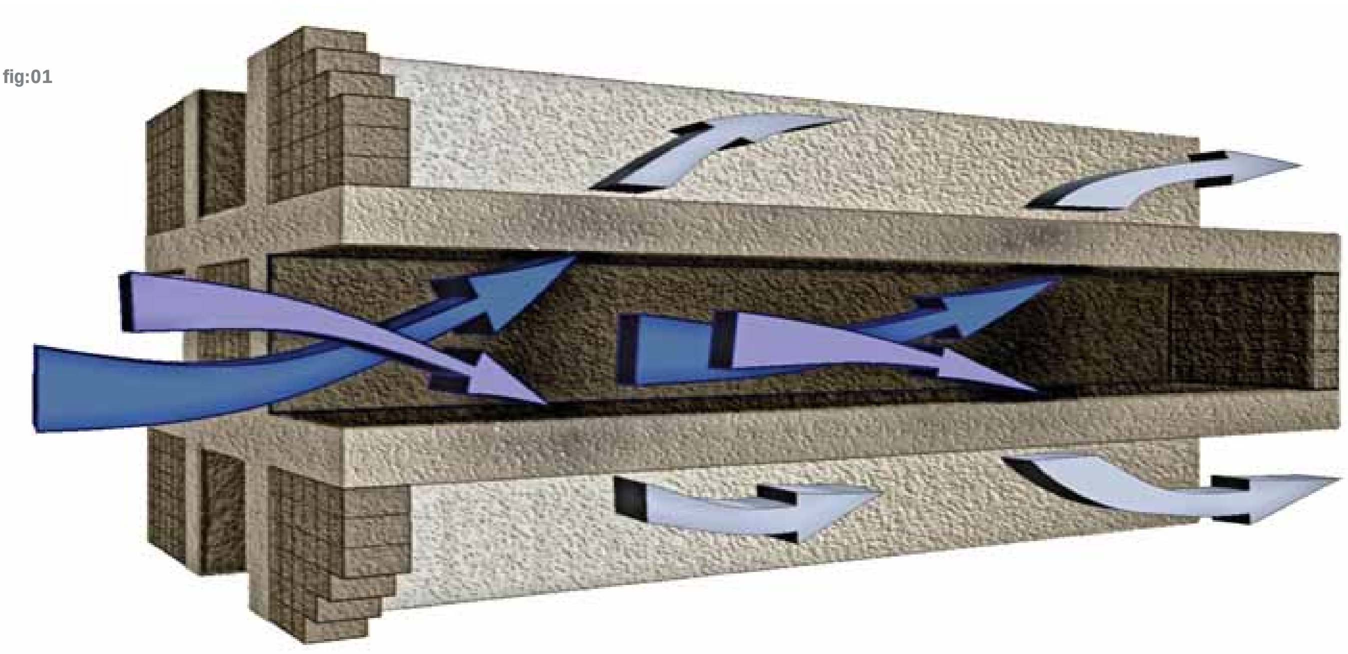
# Figures

01: Intake manifold of a six cylinder engine
02: Velocity field – Runner section
03: Mach number – Runner section
04: Water cooling jacket mesh of a 4 cylinder engine
05: Pressure filed- Water cooling jacket of a four cylinder engine
06: Wall heat transfer coefficients – Water jacket four clinder engine
07: Temperature field – Water jacket of a four cylinder engine

# Soot loading and regeneration of diesel particulate filters

**Dr. Christof Hinterberger & Dr. Mark Olesen, ArvinMeritor Emissions Technologies GmbH**

fig:01

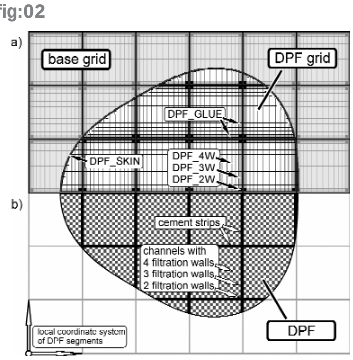


During the regeneration of Diesel Particulate Filters (DPF), high thermal loads can arise that are particularly detrimental to the lifespan of the filter ceramic. Extreme temperature peaks can lead to disintegration of the filter material. High temperature gradients cause thermal stresses that can result in micro-cracks in the brittle filter material that adversely affect the filter efficiency. To optimize the durability, function, cost and weight of DPF layouts, a novel simulation technique based on STAR-CD has been developed at ArvinMeritor Emissions Technologies that allows a highly realistic three-dimensional simulation of the thermofluiddynamic behaviour of diesel particulate filters during transient soot loading and regeneration.

## Modeling Background

The DPF filter has a structure of individual channels that are open on one end and plugged on the other. The exhaust gas enters a channel, passes through the porous sintered channel walls, and exits via adjacent channels. In the process, flow-borne soot particulates deposit in and on the walls (Fig. 1). The DPF thus exhibits three interconnected flow regions: the inlet channels, the wall flow and the outlet channels. Variations in the local wall filtration areas within the DPF segments, and the presence of the cement strips between DPF segments, influence the local flow distribution, and hence the soot deposition. Instead of four filtration walls, channels that border the segment corners or the cement strips have only two or three filtration walls, respectively (Fig. 2b).

fig:02

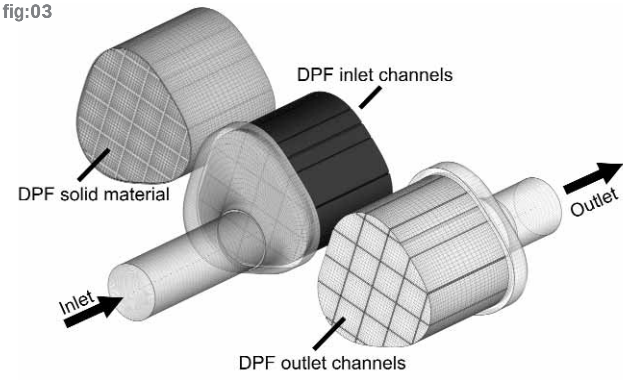



Since a detailed resolution of the flow within the DPF is impracticable (for calculation speed and meshing reasons), and since the wall flow is not a continuum, a macroscopic modeling approach is used. In the technique developed at ArvinMeritor, the DPF filter is partitioned into distinct inlet and outlet channel flow domains and a solid domain (Fig. 3), all of which are interlinked by special source terms. The inlet and outlet channel flow fields are modeled using anisotropic porosities, with the contraction and expansion losses at the entrance and exit of the DPF being modeled via baffles. The wall filtration mass flux is reflected through mass sources and sinks within the porosities and is proportional to the local pressure differential across the wall. The local flow resistances are affected by the soot deposition. The DPF material (including cement strips) is modeled as a separate solid that includes heat exchange to the surroundings. This approach allows the capabilities of STAR-CD, augmented with custom user subroutines, to solve the governing equations directly, without resorting to co-simulation. The resultant coupling of the flow fields inside and outside of the filter ensures an efficient numerical procedure.

## Meshing / Model Preparation

The meshing of the cement strips and the core, perimeter and corner regions of the DPF segments is specified via a parameter control file that includes mesh and geometry parameters. From an initial global mesh that spans the entire flow domain – without regard to the DPF internals – an immaculate surface geometry can be extracted for subsequent meshing of the DPF segmentation. Intersecting the DPF base segmentation mesh with the extracted surface yields the final DPF region (Fig. 2a). For this operation, the

fig:03

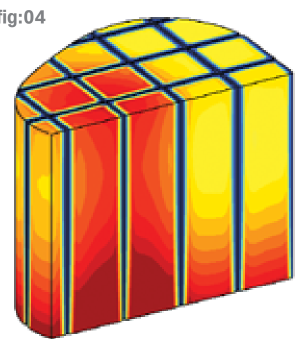



capabilities provided by CD-adapco's automatic meshing module are leveraged. Additional scripting is used to re-embed the DPF sub-domain in a suitably modified global mesh, to set the numerical parameters, and to prepare the final simulation model for job submission. In addition to the significant time-savings afforded by the automated scripting (typical times for meshing and embedding a new DPF layout are in the order of minutes), potential user errors are reduced and the reliability of the relatively complex simulation process is vastly improved.

## Simulation Results

A detailed three-dimensional soot distribution is shown in Figure 4. The local soot deposition is coupled to the mass flux through the wall, and hence to the local pressure difference across the filter wall.

fig:04



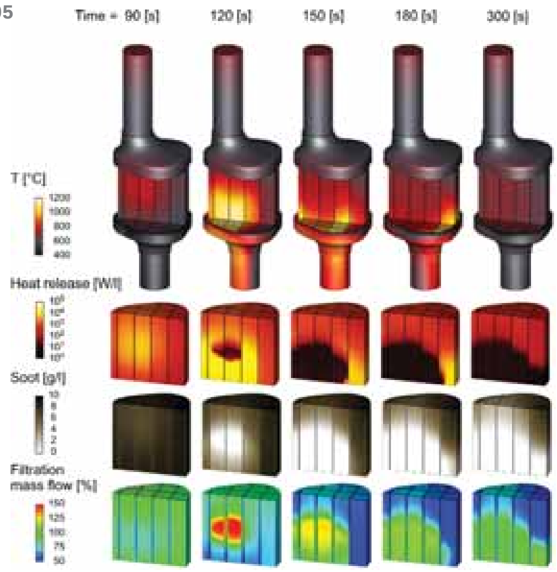
The effect of the higher flow resistances in the channels adjacent to the cement strips is evident in the reduction of the soot loading in these channels. The partial flow deceleration and the flow impingement on the face of the cement strips diverts the flow locally and promotes soot deposition in the neighbouring cells that have four filtration walls.

An example of a triggered filter regeneration process in the presence of excess oxygen is shown in Figure 5. After the initial warm-up phase (after 90s) with moderate regeneration, a significant increase in the reaction rates initiates an uncontrolled, accelerated regeneration (after 120s). The exothermic heat that is transported downstream by the flow causes a distinctive, accelerated, localized regeneration in the rear third of the DPF. The extreme local heat release spreads radially and uncontrolled until the rear of the DPF is burned free of soot. The resulting thermal loads are especially critical for the filter ceramic durability. During the final phase (after 180s), the temperature distribution is relatively homogeneous within the entire DPF and the remaining soot in the front of the filter continues to burn down slowly.

## Closure

Directly coupling the flow fields inside and outside the filter yields a very efficient simulation process. For example: a complete, transient soot loading calculation with 1/2 million cells requires approximately 10 CPU hours with a 3.4 GHz Xeon processor, or three hours with four processors. The finer temporal resolutions required for the regeneration physics (typically 1-2 sec time-steps) result in significantly more computationally intensive simulations – approximately 20 CPU hours per minute real time. However, with as few as eight CPUs, regeneration simulations can be conducted overnight.

fig:05




The newly developed technique provides fundamental insights into the soot loading and regeneration behaviour of diesel particulate filters – insights that are the prerequisite for founded decisions about optimizing DPF layouts. The detailed conclusions that can be drawn about soot loading and thermal characteristics during the regeneration help assure DPF function is maintained and DPF failures are avoided over the operational lifetime. By harnessing the capabilities of STAR-CD and pro-STAR, rapid turnaround times are possible – enabling CFD considerations to direct the product design, even in the earliest concept and design stages. ■

## Figures

01: Schematic of the flow through a DPF
02: Mesh structure of the DPF sub-domain
  a) Numerical grid showing cell types in the DPF sub-domain
  b) The underlying physical DPF segmentation
03: Layout of the computational model showing the partitioning of regions
04: Three-dimensional soot distribution
05: DPF Regeneration

# STAR-CD fueling injector primary breakup

**Jarrod Sinclair and Chris Seeling, Victorian Partnership for Advanced Computing, Centre for Computational Prototyping, Australia.**
**Peter Murdoch, GM Holden, Australia.**

Automotive engines of the future must meet increasingly stringent emission and fuel consumption demands. Efficiency improvements to the fuel injection system is one key activity to achieve this. However, a current lack of fundamental knowledge of the injection process is limiting the potential for future injector designs. In particular, a clearer understanding of the physical processes involved as liquid fuel exits a fuel injector and fragments into droplets is desirable. With this knowledge one could more effectively design the ideal combustible air/fuel cloud within the combustion system, and therefore significantly improve engine performance using currently available technologies.

fig:01

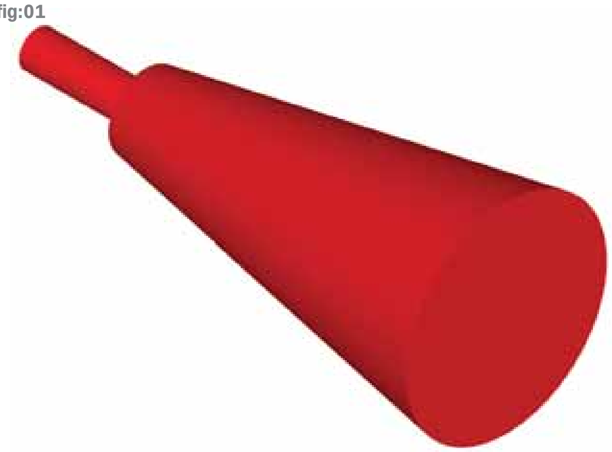
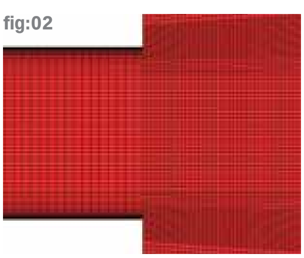


fig:02



Breakup or atomization are the terms used to describe the formation of droplets from a continuous stream of liquid or from larger droplets. Primary breakup is the initial phase of liquid fragmentation to form these larger droplets, ligaments or network structures. A liquid stream or sheet undergoes breakup by various modes that are classified by the Reynolds, Weber and Ohnesorge dimensionless ratios. The breakup of a high speed jet is more likely to fall within the turbulent atomization regime, where the exact physical processes leading to fragmentation are currently unclear. This regime was the subject of this work.

Experimental spray test rigs and measuring probes can be used to investigate primary breakup at the nozzle tip, however this is quite demanding. A resolution of the order of one micro-meter, and a time-scale of the order of one nano-second are required to capture the physics. In addition, it can be relatively costly and difficult to vary the system parameters in order to adequately map the injector response. Therefore, STAR-CD was used in this work to numerically investigate the primary breakup process.

A series of 3-dimensional multiphase analyses were conducted using STAR-CD on an angular segment of an annulus injector nozzle. The flow was defined by a Reynolds number of 3000, a Weber number of 3000, and an Ohnesorge number of 0.02. The physics of cavitation, important in many injector flows, was not accounted for as it does not play a major role for the type of injector under investigation.

The volume-of-fluid (VOF) two-phase approach was used to model the free surface between air and fuel. In STAR-CD, the CICSAM advection scheme was used to minimize artificial diffusion of the interface between the two immiscible fluids. To ensure solution stability and accuracy, a target Courant number of 0.3 was enforced throughout the entire simulation runtime. This corresponded to a time-step size of approximately 1 nano-second. Surface tension between air and fuel was accounted for using the continuum surface force (CSF) approach.

As breakup considered in this work is classified in the turbulent atomization regime, capturing turbulence accurately is critical. Therefore, a fine resolution Large-Eddy Simulation (LES) approach was adopted. This ensured that turbulent structures larger than a few cells were directly simulated by STAR-CD, while sub-grid structures were modeled using the one-equation k-l sub-grid model. Particular care was taken to ensure that the y+ values were less than one next to the internal channel walls. It is considered that this turbulence treatment is more appropriate than Reynolds averaged (RANS) models in predicting the separation of the boundary layer flow from the injector nozzle outlet and its transition into the shear layer formed further downstream.

fig:03

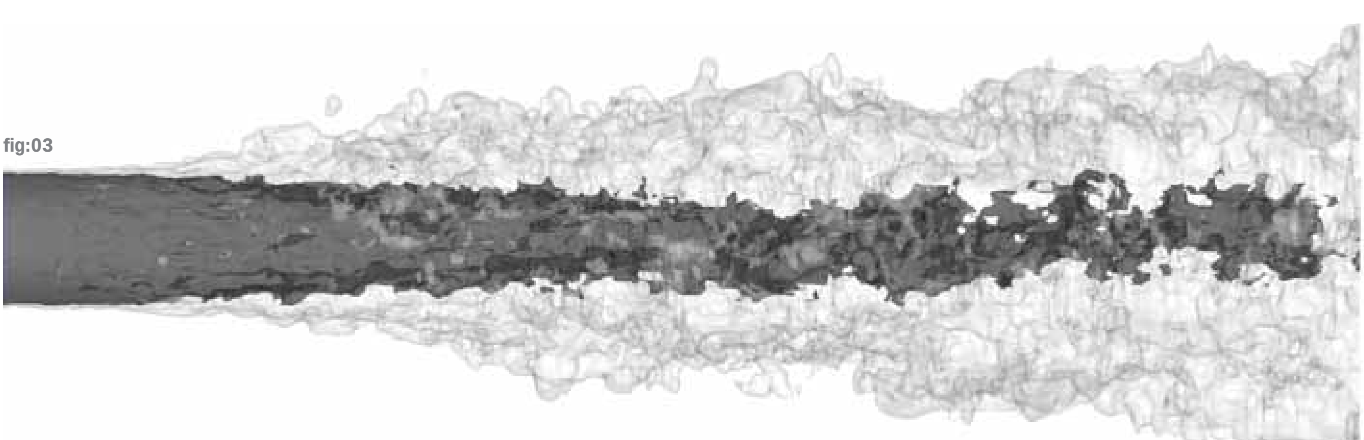


fig:04

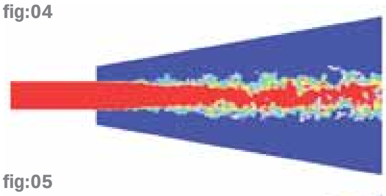


fig:05

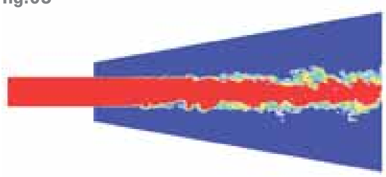
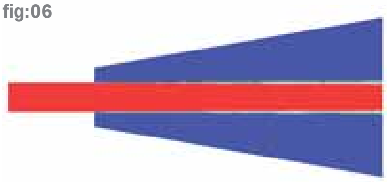


fig:06



The computational domain consisted of the nozzle outlet zone with an opening height of 20 micro-meters. The domain was extended about 1mm upstream on the inside of the nozzle channel, and 4mm downstream of the exit plane. From experimental and analytical results, this domain was considered to be large enough to contain the primary breakup region. A sub-micron mesh size was created at the nozzle outlet and core primary breakup region. Several layers of mesh refinement were used to reduce the cell count away from the region of interest without enforcing unrealistic boundary conditions. A mesh containing approximately 10 million cells was used for each simulation, highlighting the need for high-performance cluster computing.

Simulations were run on a 16-CPU Intel Itanium II cluster running HP-UX, taking on average five days to complete for each. To obtain time-averaged values, and to wash out the initial conditions, the total simulation time was chosen to be five times the fuel residence time in the domain. Waves form on the surface of the fuel after it has traveled some distance from the nozzle outlet plane. These waves grow in amplitude while maintaining a constant wavelength. A transition occurs as the waves form more of a localized 3-dimensional structure, most likely dominated by surface tension effects. Large-scale breakup of the sheet takes place quite rapidly at this transition point. It can be seen that a finer network structure comprising of transverse ligaments begins to form further downstream. This increase in fuel surface area causes the formation of pre-atomization droplets which would eventually compose a finely atomized spray cloud beyond the domain of the current simulations. The root cause of the initial instability, and a characterization of the turbulent structures on the amount of breakup are currently under investigation.

## Conclusion

From this work, STAR-CD has shown to be a valuable simulation tool in capturing the critical phases of fuel injector primary breakup, and has given key insights into the governing physical processes. The solver provided a near-linear parallel speed-up, giving accelerated throughput on such a large model. The techniques developed using STAR-CD in this work are showing promise as a design tool for future injectors.■

Contact details - **jarrod@vpac.org**

# Breaking up isn't hard to do…

**Marco Buonfiiglioli, CD-adapco UK**

Always in tune with our customers, CD-adapco has independently been devoting significant effort to the study of primary break up and atomization, using a similar VOF and LES based approach.

The CD-adapco study concentrated on analyzing the effect of inlet turbulence on spray formation, a phenomenon that is very difficult to measure empirically.

Since the flow at the inlet is not fully developed, a "synthetic turbulence" condition was applied (developed in collaboration with the University of Manchester). By varying the inflow velocities in both time and space, the vortex-like method generates synthetic but realistic turbulent structures at inflow. This approach allowed the use of a relatively small nozzle length (just three diameters), removing the need to use a computationally expensive long inlet duct.

The intensity of the synthetic turbulence at inflow was systematically reduced from an original highly turbulent level to almost zero.

Comparisons with the limited experimental data available indicated a good agreement in the prediction of both break-up length and spray angle. Figure 3 shows instantaneous iso-surfaces for VOF concentrations of both 1% and 99% heavy fluid, illustrating both the core and the external spray boundaries.

Figures 4, 5 and 6 show a section through the jet for high, medium and low inlet turbulence levels and clearly show the extent to which the break-up length and cone angle are influenced by the upstream condition.

An impressive aspect of this project is the surprisingly short run times required in order to produce impressive results, as both LES and VOF are often regarded to be highly computationally expensive. Each of the simulations was run until the spray reached a statistical steady state (from a quiescent initial condition). Using 4 2.8GHz processors of a Linux cluster was possible to compute 4ms of simulation in a single day.■

References:

[1] M.Buonfiglioli and F. Mendonça, "LES-VOF Simulation of Primary Diesel Spray Break-up with Synthetic Inlet Perturbations", ILASS Americas 2005.

[2] Benhamadouche at al, "Synthetic turbulence inflow conditions for LES", Turbulence, Heat and Mass Transfer 4 (2003)

## Figures

01: Computational design
02: Section through mesh
03: iso surface of 1% and 99% heavy fluid
Contours of VOF with
04: High inlet turbulence levels
05: Medium inlet turbulence levels
06: Low inlet turbulence levels

# Simulation of vehicle soiling at Audi

**Dr. Moni Islam, Audi AG**

fig:01

With CFD methods now firmly established as a development tool for the prediction of the aerodynamic forces acting on a vehicle, the use of CFD for predicting the soiling characteristics of a vehicle have been investigated at Audi for several years. Of interest is how particulate matter from the environment – water, snow or dirt – is deposited, accumulated and transported on the surface of the car and in its vicinity. Common examples of these phenomena are the flow of water from the wind screen onto the side windows, the deposition of rain drops or dirt on the surface of a wing mirror or the accumulation of dirt on the rear surface of the boot lid. Until recently, activities have focused primarily on assessing whether or not the existing two-phase-flow models in STAR-CD are suitable and adequate for this application. This article presents some of the results of this work.

The primary challenges of simulating soiling phenomena are the accurate description of the physics of the liquid and solid phases and the correct modeling of the interaction of those phases with the air. Of particular relevance are:

- size and velocity-distributions of rain drops;
- modeling of snow and its properties;
- impact behavior of rain drops on solid surfaces;
- deposition and accumulation of solid particulate (eg. dirt, snow) on solid surfaces;
- liquid-film formation and transport;
- primary break-up of liquid films due to aerodynamic forces;
- characterising the droplets generated at the road surface and at the wheels due to vehicle motion.

These issues, the last point in particular, present a major challenge to the successful modeling and simulation of vehicle soiling.

In the methodology applied at Audi, soiling is simulated using a one-way coupling between the gas and liquid/solid phases, namely that the liquid/solid phase is influenced by the gas phase, but not vice-versa. In this way, the computational overhead is kept relatively low, as calculations are performed on the liquid/solid phase only, with the gas phase “frozen”. The existing two-phase models in STAR-CD – both two-phase Lagrangian and liquid film – are applied to vehicle configurations known from experiments, and the results of the simulations are compared with the experimental results in a qualitative way. In what follows, a number of examples are presented.

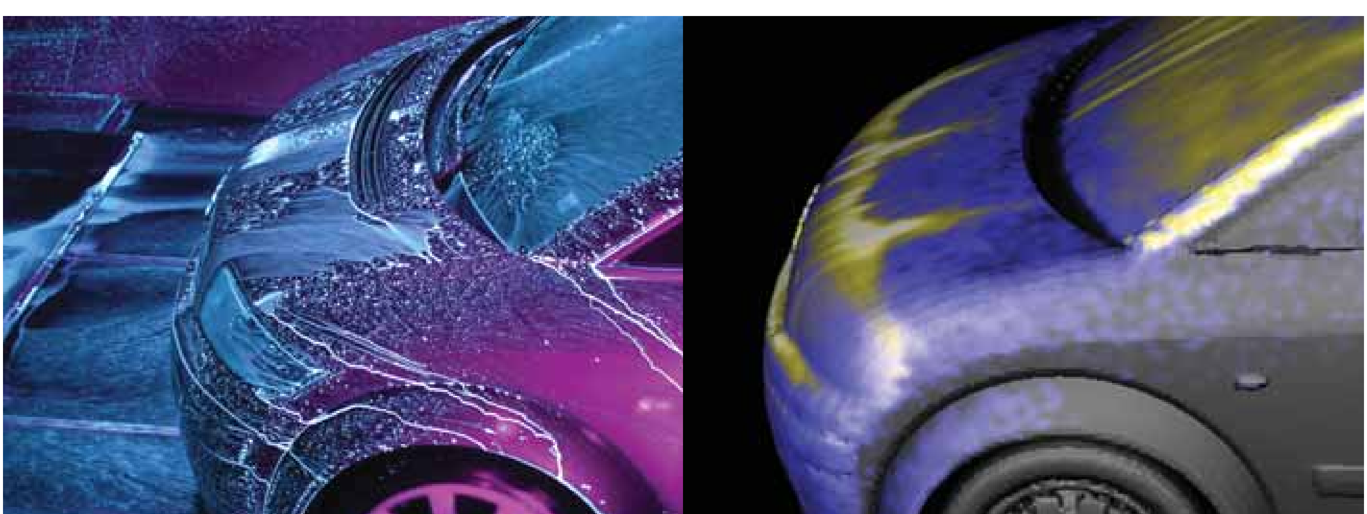


fig:02

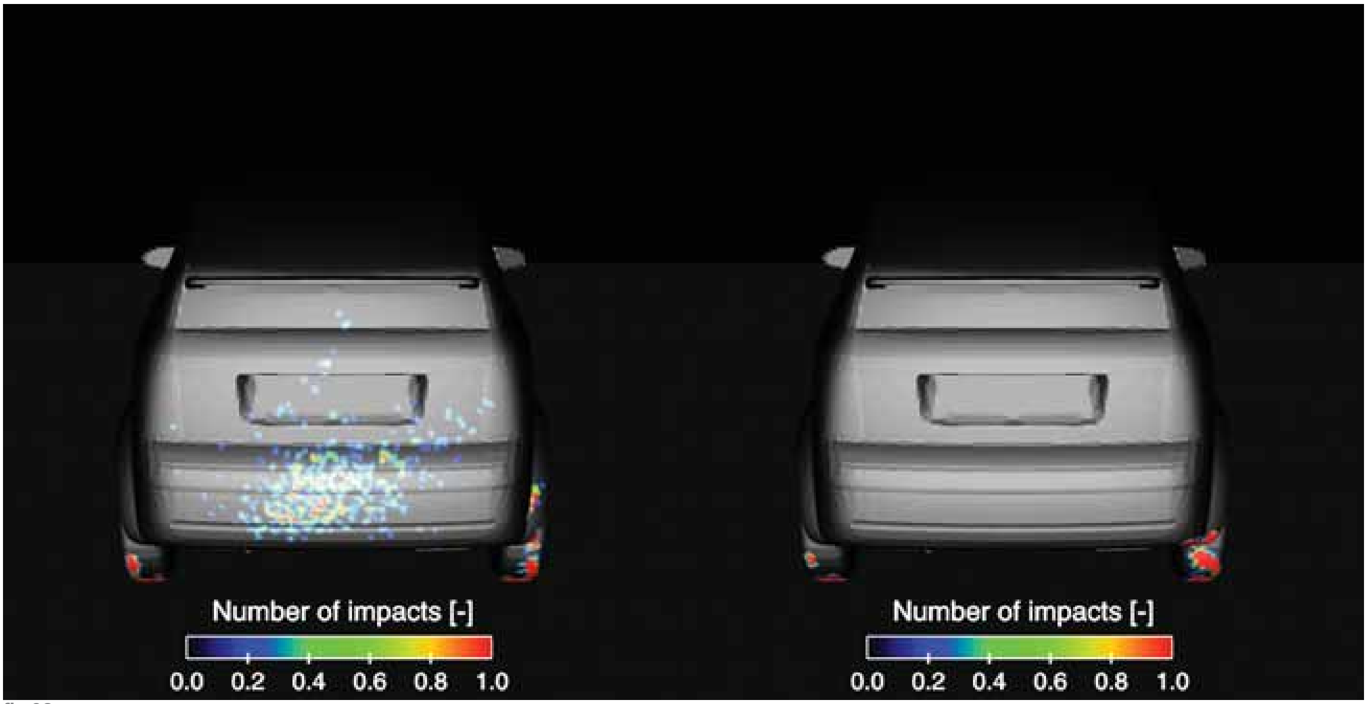



fig:03

Figure 1 shows an example taken from investigations to assess the prediction of the interaction of rain drops with the vehicle's surface. The wind-tunnel experiments, seen on the left, show a cloud of droplets standing off the front bumper surface, arising from the interaction of the oncoming flow and the droplets rebounding off the surface. In the simulation, the droplets are modeled according to the standard Lagrangian formulation, with the initial/boundary conditions for size and velocity distribution obtained from experiments. Their interaction with the surface – whether they rebound, break-up, stick or smear – is based on the models developed by Bai available in STAR-CD. Analysis of the simulation results shows that the experimentally observed phenomena are qualitatively predicted, and further investigations are underway to assess the predictions quantitatively and in more detail. Here, the formation of liquid films on the vehicle's surface was not considered.

Studies of this phenomenon have also been carried out, as illustrated in Figure 2. In the simulations, as in the experiments, a liquid film is allowed to accumulate on the vehicle surface through the impingement of liquid droplets originating upstream. The film then evolves under the influence of aerodynamic forces and is transported across the vehicle surface accordingly. A number of qualitative features of the film distribution are captured fairly well by the simulation, such as the local film-thickness maxima between the headlamps and the inlet grille, as well as the separation bubble and accumulated film just downstream of the Audi rings on the bonnet. The film model represents a statistically averaged film and therefore cannot capture the water streaks observed in the experiments, which are stochastic in nature. Furthermore, these streaks are highly dependent on small local geometric features of the trim and gaskets and microscopic surface properties, not contained in the simulation model.

Also of interest is the soiling of the aft portion of the vehicle arising from entrainment of particulate matter from the underbody and wheels into the vehicle's wake. Figure 3 shows the soiling patterns on the rear hatch of the Audi A2, represented by a normalised number of particle impacts, for particles of two different diameters. As can clearly be seen, the smaller particles are entrained by the wake of the vehicle and impact on the surface of the rear hatch. The larger particles have a sufficiently high momentum that they are not deflected sufficiently by the air flow to land on the rear hatch.

Currently, these investigations only consider the unsteady aspect of the liquid phase, i.e. the transport and accumulation of droplets or particulate matter in the vehicle's vicinity. The interaction of the unsteady air flow, with its large-scale turbulent motions, and the droplets, is clearly an aspect that must also be investigated in subsequent steps. Furthermore, essential for the accurate modeling of soiling phenomena is the correct description of two-phase boundary conditions for realistic road-like simulations, namely droplet atomisation from liquid films on road and tire surfaces and the two-phase wake of upstream vehicles. For these reasons, predicting vehicle soiling will remain a significant challenge for some time. ■

Acknowledgements: The assistance of Dr. L. Lührmann, Dr. M. Jaroch and Mr. H. Steinicke of Audi AG in carrying out the experimental investigations, and of Icon Computer Graphics Ltd. for the simulations, is gratefully acknowledged.

## Figures

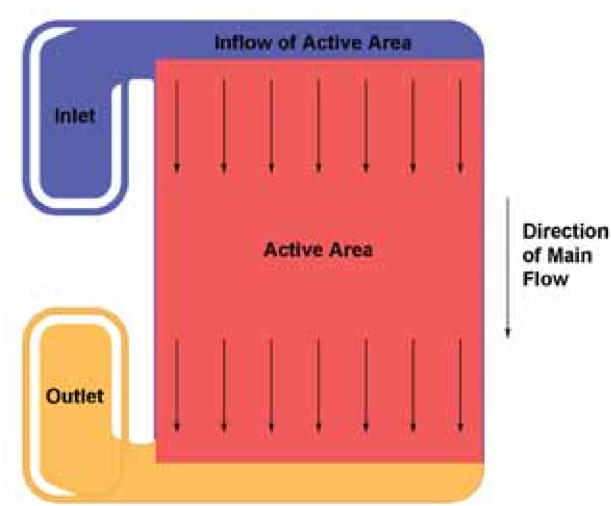
# More power with STAR-CD
## Optimizing flow in SOFC fuel cells

**Volker Schaika, Webasto AG, Germany**

Due to the increasing number of safety and comfort devices in modern vehicles, their consumption of electrical energy is on the rise. In order to fulfil this demand, Webasto AG is developing an Auxiliary Power Unit (APU) based on solid oxide fuel cells (SOFC). The APU generates electrical energy and heat using the vehicle's fuel, while operating independently of the engine. This obviates the limitations of traditional systems, such as the amount of energy available when the engine of the vehicle is not running. By using STAR-CD to optimise the geometry, a great improvement in the performance of the APU was achieved.

CFD has been extensively used in the development of the APU components, such as the reformer, after-burner and heat exchangers. This article describes the optimization of one of the APU's key components: the anode gas flow in the SOFC plate. The performance of a single fuel cell, and therefore the whole stack, strongly depends on the even distribution of the fuel gas in the anode plate, thus maximizing the reacting area.

fig:01

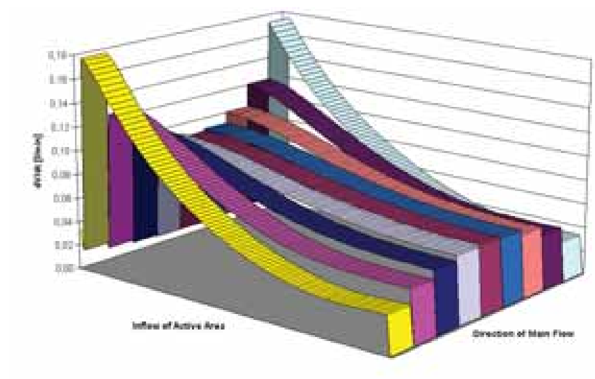



The anode plate consists of inlet and outlet manifolds and an active area connected to the solid oxide electrolyte. Approximately forty variations of the plate were examined in STAR-CD. The inlet and outlet manifolds were designed with various baffles. Different numbers of these baffles, as well as the arrangement and height of the inlets and outlets, were considered. The active area consists of a channel structure made of porous metal foam. Figure 1 shows the basic structure of the anode plate.

pro-STAR's trimmed cell automatic meshing was used to build the plate models. The manifolds and the active area were meshed separately. The different combinations were then connected using the coupling capabilities in STAR-CD. The resulting grids consisted of approximately 600k cells.

The CFD simulations were run using the AMG solver and the MARS advection scheme. As the flow in the plates is laminar, no turbulence model was necessary. The density and viscosity in the

fig:02



fuel gas were calculated for a given mixture at the temperature of 850°C and kept constant throughout the simulation. This set-up allowed short calculation times of approximately 60 minutes, running on a 6 CPU AMD Opteron Linux cluster.

fig:03

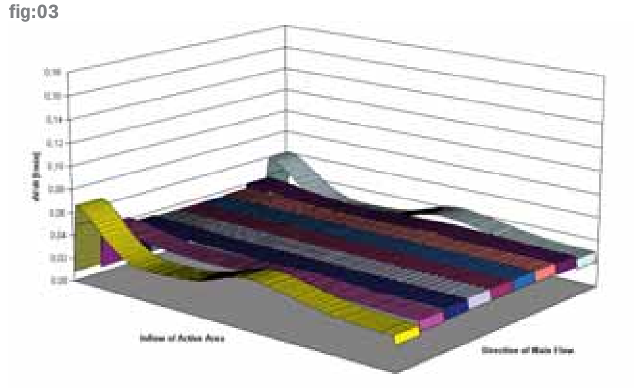


Figure 2 shows the flow distribution in the active area of one of the investigated geometries. The different velocities in the channel structure indicate the uneven distribution of the fuel gas and therefore a poor performance of the fuel cell. The flow distribution through the optimized model is given in figure 3. The uniform velocities show the improvement of the flow field.

This analysis demonstrates the use of STAR-CD to investigate a large number of design variations of a SOFC fuel cell in a short time. The resulting geometry shows a great improvement in a crucial component of the auxiliary power unit. ■

## Figures

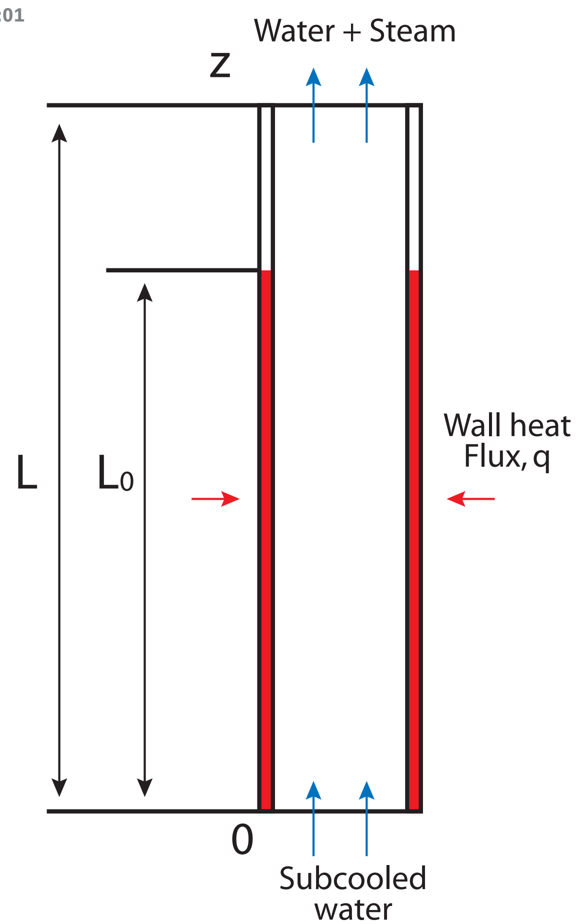
01: General Geometry of SOFC Anode Plate
02: Flow Distribution in Plate A
03: Flow Distribution in Plate B

# International collaboration on modeling of Boiling Water Reactors (BWR)

**Adrian Tentner, Argonne National Laboratory, USA**
**Andrey Ioilev, VNIIEF / Sarov Laboratories, Russia**
**Simon Lo, CD-adapco, UK**

In the extreme environment of the reactor core of a Nuclear Boiling Water Reactor (BWR), the ability to accurately predict the different boiling regimes is crucial. The boiling regime determines the rate of heat transfer from the fuel assemblies to the water, and subsequently the plant performance. Consequently, detailed understanding through simulation with STAR-CD is highly desirable. Modeling these phenomena in STAR-CD is the subject of a collaborative project between Argonne National Laboratory (USA), Sarov Laboratories (Russia), the Russian Federal Nuclear Centre (VNIIEF) and CD-adapco.

fig:01

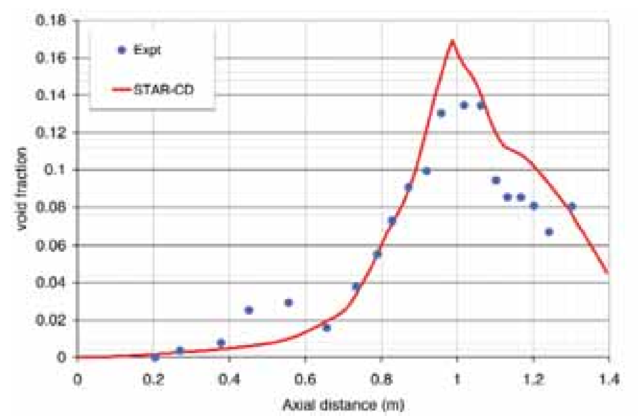



The project, sponsored by the US Department of Energy's Initiative for Proliferation Prevention Program, brings together experts in CFD and the thermal-hydraulic phenomena in BWRs. Its goal is to develop an analysis methodology for studying the two-phase boiling flows in BWR fuel assemblies. Ultimately, the models and experience from the project will be encapsulated into an "Expert System" (es) solution, to enable others to carry out such analyses.

The boiling flow phenomena in BWRs are very complex. In simple terms, the reactor core consists of many fuel assemblies; each containing an array of nuclear fuel pins. The coolant water flows between these fuel pins from bottom to the top. As the coolant water rises, heat from the fuel pins is transferred to the water. The water boils and turns into a mixture of water and steam in the upper section of the reactor: undergoing a phase change. In addition, the boiling goes to bubbly flow, slug flow, annular flow and mist flow.

In terms of heat transfer between the coolant and the fuel pins, the coolant goes from single-phase convective heating to sub-cooled boiling, and saturated boiling heat transfer. Each of these "flow" and "boiling" regimes require special models that describe the mass, momentum, and energy transfer between the liquid and vapor phases, as well as the heat transfer between each phase and the fuel pins.

fig:02




After the first year of the project, new models for bubbly flow and sub-cooled boiling have been developed, implemented and validated in STAR-CD. Several validation cases were carried out. Each case gives very satisfactory results and demonstrates that the models developed are capable of modeling the bubbly and sub-cooled boiling flow. Figures 1 and 2 show schematics and comparisons to STAR-CD for one of the validation cases involving boiling of sub-cooled water in the heated section of the pipe followed by condensation in the unheated section. Full details of the models and the validation cases can be found in Ref [1].

Further work for the project includes development of a flow regime prediction method, prediction of critical heat flux and extension of the models to all flow regimes found in the BWRs.

So far, the combined expertise of the partners has delivered validated models for a number of the complex phenomena in a BWR. This marks a significant milestone on our road to simulating the extreme environment of a BWR.■

Reference
1 A. Tentner, S. Lo, A. Ioilev, M. Samigulin and V. Ustinenko
"Computational fluid dynamics modeling of two-phase flow in a boiling water reactor fuel assembly" Mathematics & Computation, Supercomputing, Reactor Physics & Nuclear & Biological Applications, Palais des Papes, Avignon, France, September 12-15, 2005.

## Figures

01: Scheme of experimental section for study of boiling and condensation
02: Average void fraction as a function of distance along the pipe

# Simulating safety for high temperature reactors

**Jan-Patrice Simoneau, Julien Champigny, Framatome ANP, France.**
**Brian Mays, Lewis Lommers, Framatome ANP Inc., USA.**

## Introduction

fig:01

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

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.»

fig:03

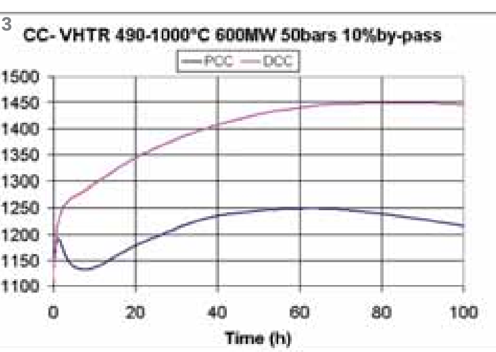


fig:04

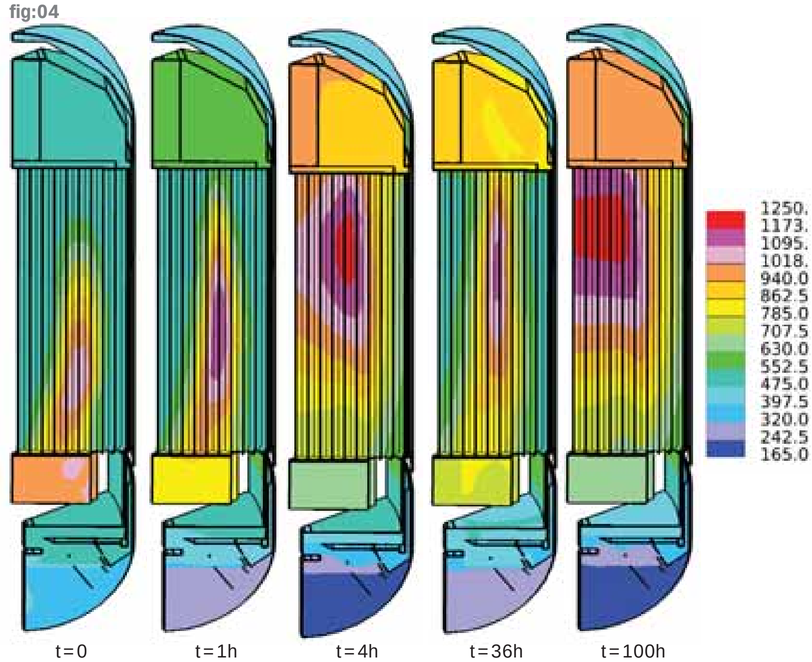
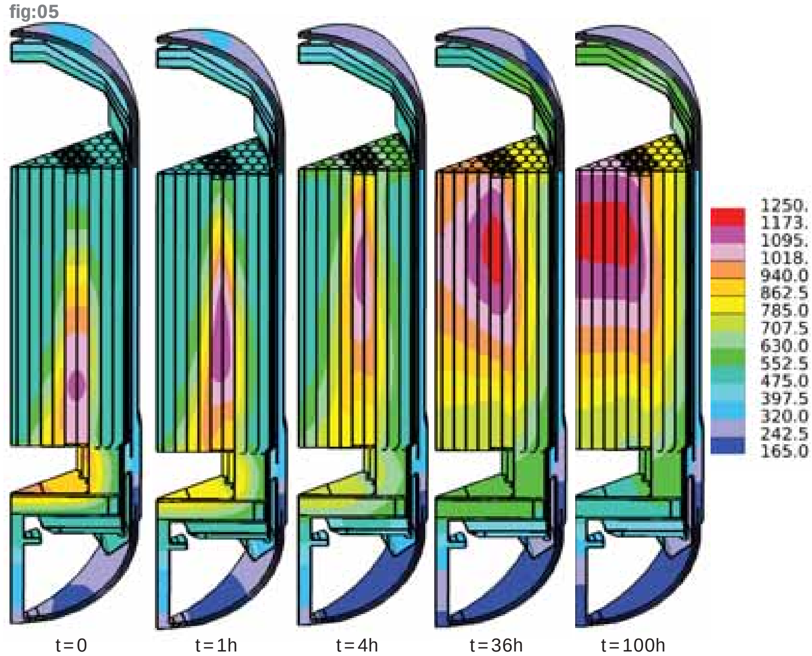



fig:05




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. ■

Further details can be found in "3D Simulation of the Coupled Convective, Conductive and Radiative Heat Transfer during Decay Heat Removal in a HTR" J. Simoneau, J. Champigny, B. Mays, and L. Lommers, Proceedings of the 11th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11), Avignon, France, October 2005.

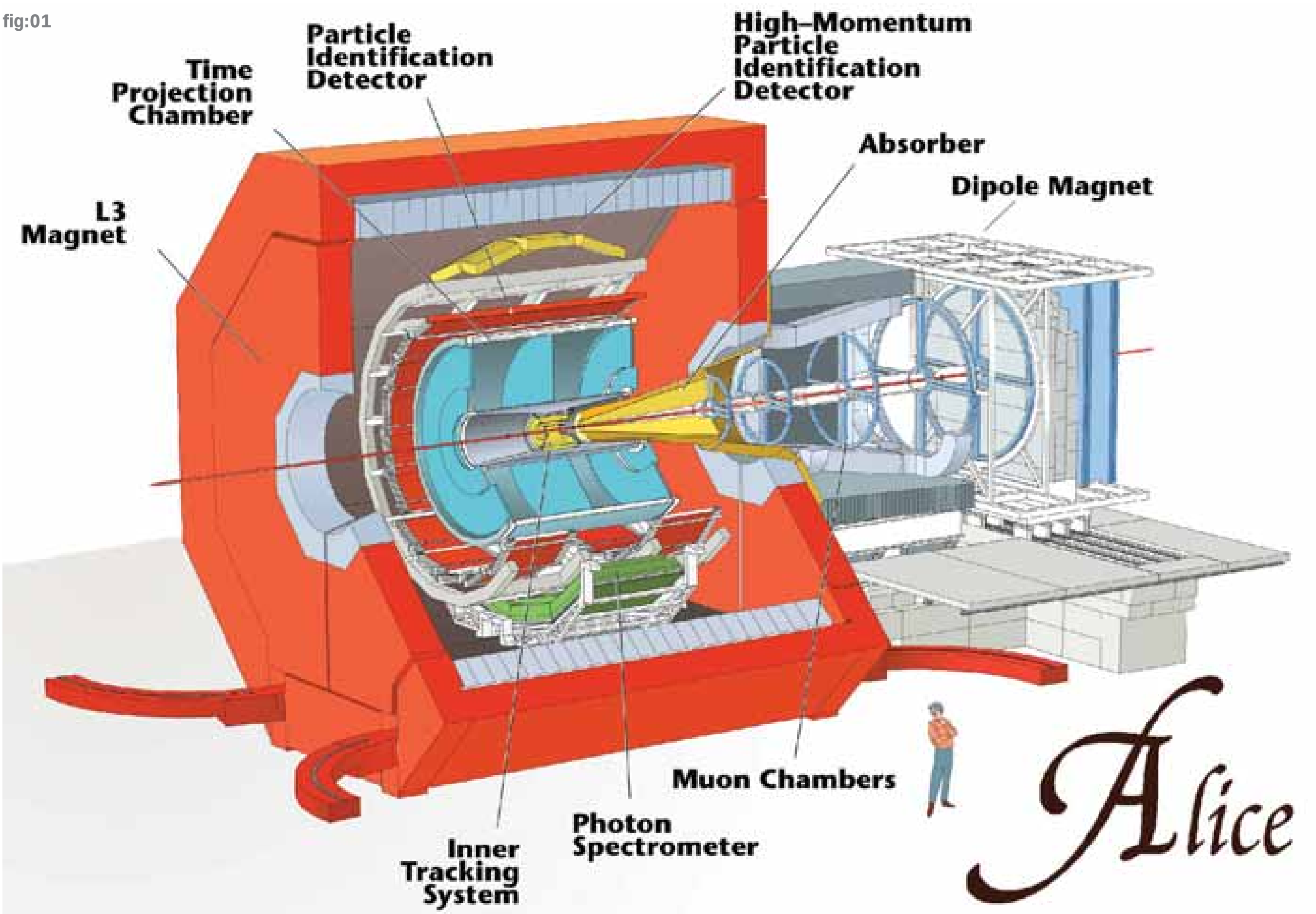
# Figures

01: Section through reactor
02: Computational domain – solid part
03: Core temperature against time: PCC and DCC
04: Pressurized Conduction Cooldown – Fluid Part
05: Pressurized Conduction Cooldown – Solid Part

# STAR-CD helps CERN to turn back time

TS/CV/DC CFD Team, CERN, CH

fig:01




## Introduction

CERN is the European Organization for Nuclear Research, the world's largest particle physics laboratory and the birthplace of the World Wide Web. Its primary objective is to provide the scientific community with facilities to study sub-nuclear particles and the forces of matter. Most of the activities at CERN are currently directed towards building a new particle accelerator and collider, the Large Hadron Collider (LHC) and the detector experiments for it. The final aim is to look back in time and recreate the environment present at the origin of the Universe to understand what matter is made of and what forces hold it together. Construction of these experiments requires an extraordinary engineering effort and STAR-CD has been used for numerical simulations of thermal-fluid related problems, particularly during the development, design and construction phases of the LHC experiments. This article presents, therefore, the study performed in one of the five experiments currently being built to run on the collider.

## The problem

ALICE (A Large Ion Collider Experiment) is a general-purpose heavy-ion experiment designed to study the physics of strongly interacting matter and the quark–gluon plasma in nucleus–nucleus collisions at the LHC. It consists of a variety of tracing devices, called sub-detectors, enclosed in a large solenoid magnet known as L3, see figure 1. Inside the L3 envelope there are heat sources, in the form of cables and electronic circuits, dissipating heat into the surrounding air. To remove this heat a dedicated ventilation system is in place and the internal surfaces of the magnet are lined with a thermal screen. However, to maintain the quality of the particle measurements, stringent temperature requirements must be satisfied inside the L3 volume. In this system, natural convection plays an important role in transferring this heat away and the resulting gradient of temperatures in the enclosed environment cannot be neglected. As a result, STAR-CD was employed to improve the ventilation system inside the L3 magnet.

## The CFD model

The geometry was built in STAR-Design solid modeler and meshed with the automatic mesh generation module, pro-STAR which uses trimmed cell technology, see figure 2. The mesh was locally refined in regions where larger gradients of temperature and velocity are expected to occur, such as close the walls of the internal sub-detectors.

Since we were only interested in the air region enclosed in the L3

fig:02

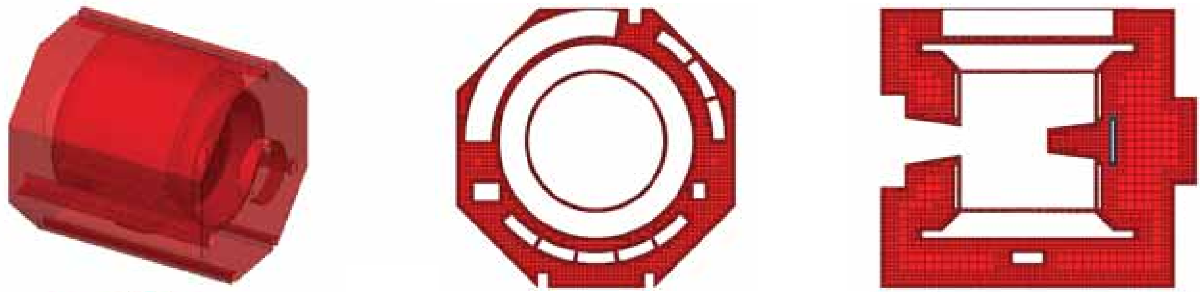


fig:03

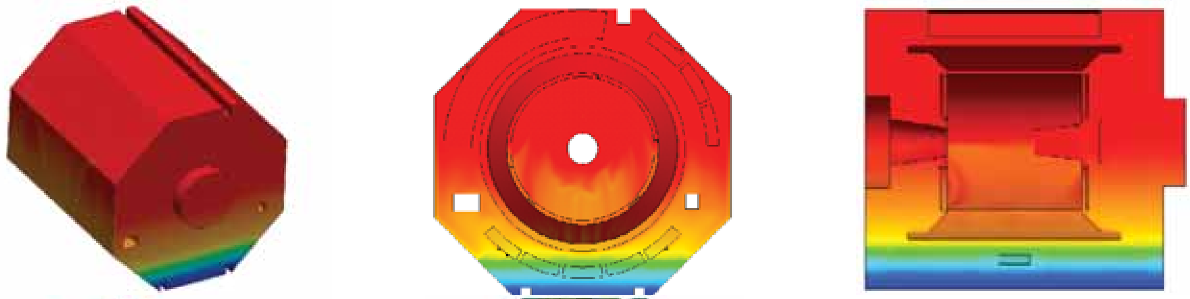
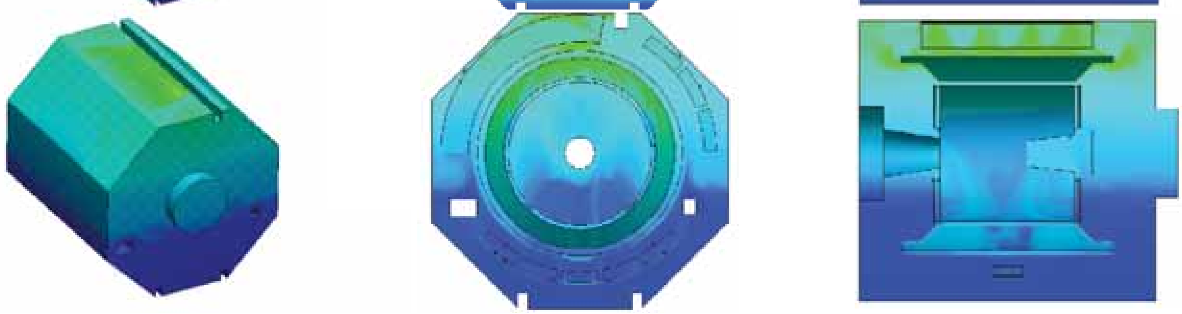


fig:04



magnet, no solid parts were included in the model. The final mesh arrangement contains approximately one million cells. Inlet and outlet boundaries were prescribed at inflow and outflow regions and constant heat flux conditions at the surfaces corresponding to the boundaries of the internal sub-detectors. Additionally, a constant temperature value was defined at the magnet thermal screen. Air was modeled as non-isothermal, incompressible (with density dependent on temperature) and turbulent gas. The widely applied high Reynolds k-ε turbulence model with default wall functions was believed to provide good representation of the enclosed flow. In order to accurately predict the effects of buoyancy, a transient solution was employed. The overall model predicts the air flow and temperature fields inside the L3 volume under different ventilation configurations. The CFD computations were performed on a parallel, double CPU Itanium Linux cluster, Openlab, available at CERN.

## Results and discussion

The results indicate that without forced ventilation the air temperature in the half upper region of the L3 volume considerably exceeds the maximum temperature limit that ensures adequate running conditions of the sub-detectors, see figure 3. As expected, the hot air accumulates in the volume in a stratified manner with a maximum temperature difference along the height surpassing the 20K.

On its own, the L3 thermal screen is insufficient for adequate removal of the heat generated by the internal sub-detectors so that a ventilation strategy is required to lower down and even out this temperature.

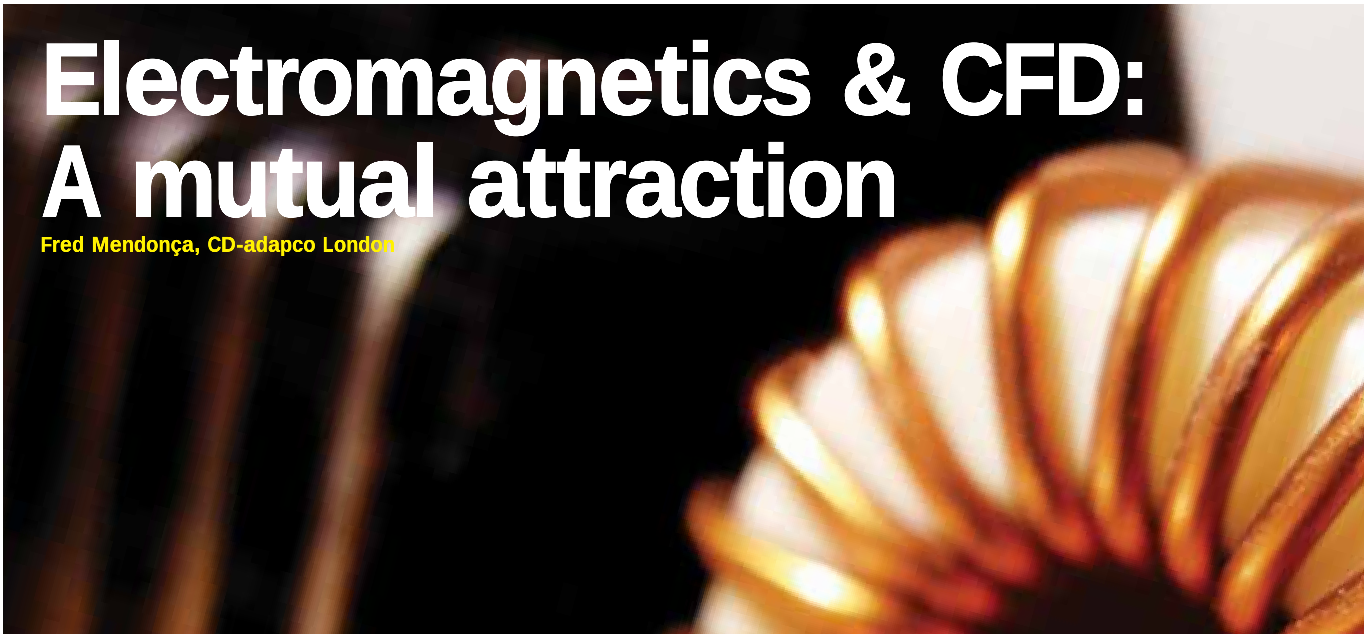
As a consequence, a number of ventilation configurations were considered and the effects of varying the flow rate, quantity, location and orientation of the inlet/outlet ducts investigated. It was found that a combination of 2 inlets placed on the floor and 2 others at the middle level with an outlet positioned at the top would provide good mixing and consequently adequate temperature uniformity of the air enclosed in the L3 volume. Temperature differences along the height were reduced to around 6K and the maximum temperatures registered at the top are now well within the acceptable working limits of the sub-detectors, see figure 4. Natural convection is still the main mode of heat transfer since the velocity magnitudes in the ventilation system and comparable to that of buoyancy driven flows.

## Conclusions

At CERN, STAR-CD has been proven to be a useful tool in assisting the development of cooling systems for high energy particle detectors. Because of the detectors large dimensions and tight project timescales, experimental work and prototype modeling is difficult and CFD models become the practical alternative. The CFD study presented allowed improvements to the ventilation system and consequently enhancement of the detector's performance. More details on this and other similar studies can be found at **http://cern.ch/cfd • http://aliceinfo.cern.ch • http://cern.ch/openlab** ■

# Figures

01: The ALICE Detector
02: Geometry and Mesh of the ALICE Detector
03: Temperature Field in the L3 volume without Forced Ventilation
04: Temperature Field in the L3 volume with Forced Ventilation



# Electromagnetics & CFD: A mutual attraction

Fred Mendonça, CD-adapco London

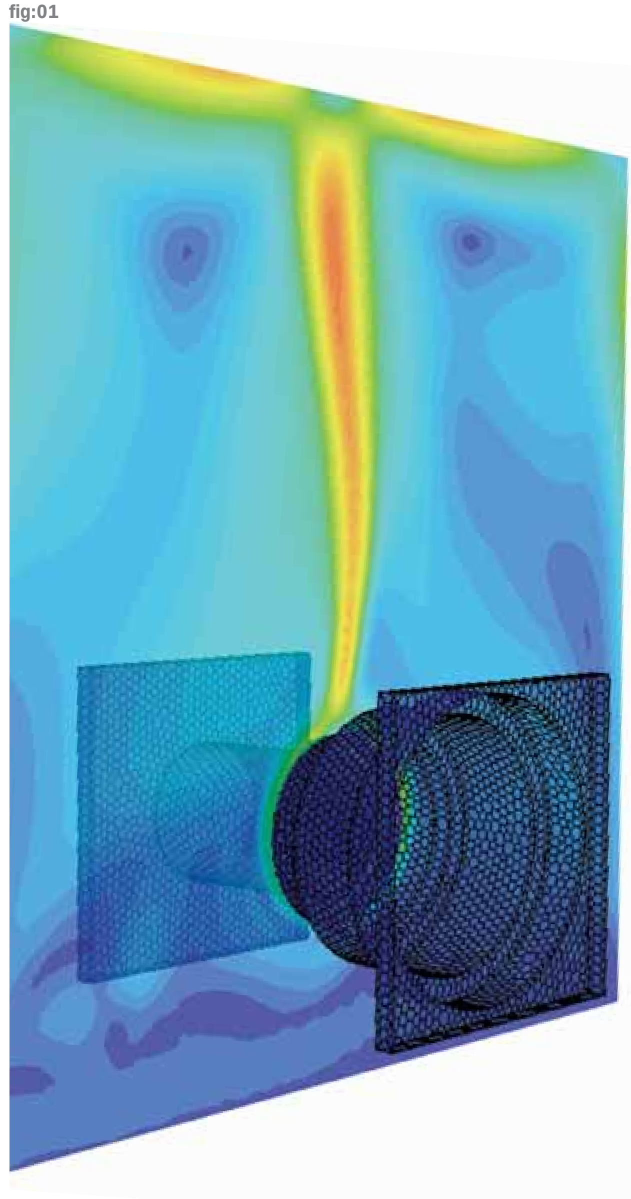
Design Engineers are all too often faced with an inexorable link between traditional engineering disciplines - fluid dynamics, structural mechanics, thermodynamics, electromagnetics. The overall performance of an engineering component, or system, requires knowledge of how these separate functions interact. Combining the analyses from separate physical disciplines, for a required repeatable product development workflow, through process integration is commonly referred to as 'Vertical Applications'.

Take the example of switch-gear in electrical power plants. A component in a high voltage (HV) circuit breaker assembly contains a copper rod and sleeve which, when connected, completes an electrical circuit. The electrical current, as it flows through the metal matrix, experiences a resistance which generates heat usually very close to the metal surface. For safety reasons, this heat needs to be dissipated, and it is most effectively done through conduction and convective air-cooling.

CD-adapco and ABB Corporate Research in Baden, Switzerland, have recently validated a modeling of the interaction between electromagnetic heat generation, heat transfer and air-cooling. According to Prof. Zoran Andjelic, Principal Senior Scientist at ABB, *"Safety regulations and component operational efficiencies require that the apparatus stays within reasonable temperature limits. While our electromagnetic codes can tell us how much heat is generated from the current flow in such components, CD-adapco has shown us how to combine this with fluidic heat transfer, and we are quite satisfied that the phenomena are well captured".*

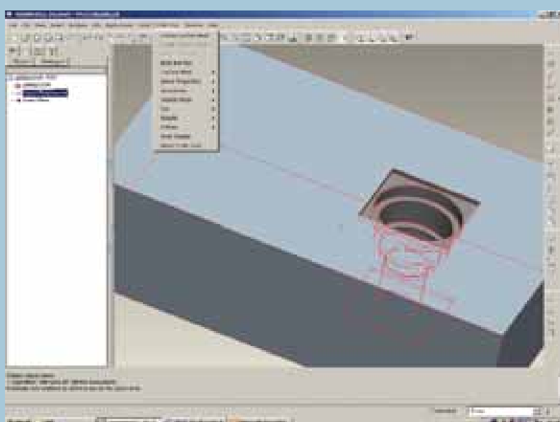
Over an operating time of twenty minutes, CD-adapco's world-leading CFD software STAR-CD takes information about the system heat-losses from the EMC code POLOPT, and computes the cooling air-flow and thermodynamics around the starting switch to predict the metal surface temperatures exposed to the atmosphere. Fred Mendonça, Manager of Vertical Applications at CD-adapco adds, *"There are important enablers necessary for this form of Process Integration to work successfully; analysts increasing believe in PLM, and realise the need for a strong link between their CAD definition and the design analysis processes. Our products, embedded in the most widely used CAD software, are linked to fast computational analysis by robust automatic meshing using polyhedral cells, a technology pioneered here at CD-adapco".*

fig:01

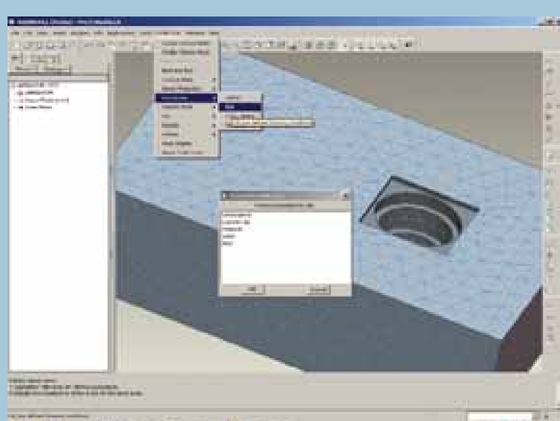


## Data management / knowledge capture

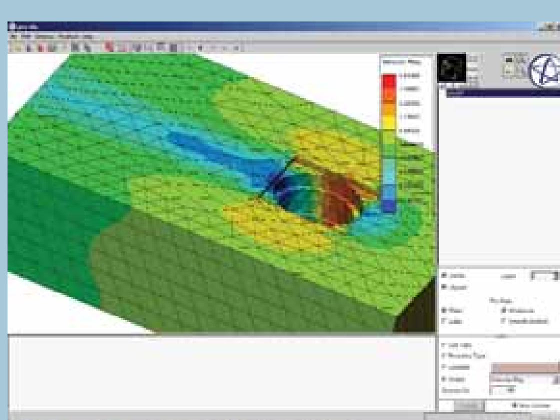
**Example - STAR-Pro/E**



**Step 1 - Your model**

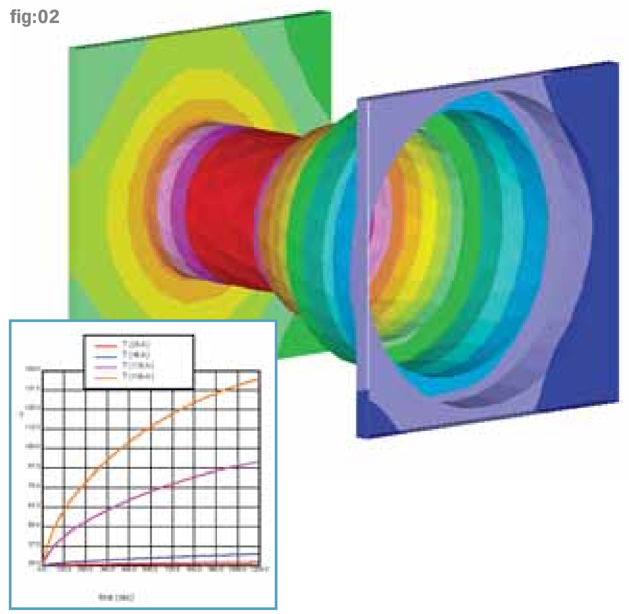


**Step 2 - Your mesh**



**Step 3 - Your results**

fig:02



STAR-CD is embedded into the native CAD systems thanks to the STAR-CAD Gateways (STAR-Pro/E Gateway is shown here in the Pro-ENGINEER environment) this means that the starting-point for this multi-disciplinary analysis is common. It's a key ingredient for design analysis. Surface and volume meshes for both the EMC and CFD can be generated automatically while still in the CAD environment. The CFD boundary conditions and solver settings can also be defaulted in the CAD environment. Heat losses computed in POLOPT are then passed to STAR-CD, and reinterpreted as electromagnetic heating, raising the metal surface temperatures. This creates a buoyant plume, which has the useful property of convecting the heat away from the surface and bringing cooler air from below into contact with the metal.

How well does the integrated modeling represent reality? Validation of the virtual modeling must always be considered as an essential part of CAE analysis. As the simulation advances in time, one can monitor the temperature of the surface, and compare these predicted temperatures with measurements on the actual apparatus. In fact, to give confidence that the EMC-CFD coupled modeling works over a wide range of operating conditions, different current charges ranging between 2 and 18 kilo-Amperes have been run through the starting switch, then both measured and simulated temperatures compared – they agree to within a few degrees. This gives Design Engineers the confidence to test different design variants in a virtual environment, early in the design process and long before the design space has been fixed. Prof. Andjelic concludes, *"Once a model is validated, the time and resource savings added to the design benefits which are inherent to this form of integrated virtual simulation are very clear."* ■

# Worldwide STAR Conferences

CD-adapco STAR Conferences are more than just another CFD event; they are a gathering of some of the top experts and users in industrial CFD. Taking place annually in your local region, these conferences enable you to keep in touch with the very latest technology and techniques and fully leverage the potential of industrial flow simulation in your engineering process. They also help you to get the best return from your investment in our software.
Hear our specialist Keynote speakers, meet with your Support team and be informed by the presentations given by our industrial clients about real world CFD applications. Our conferences are also accompanied by an exhibition of our hardware and software partners, allowing you access to the most complementary and compatible tools for our software. Our conferences are not only intended for existing users of our software, but for the whole CFD community.

As well as a programme of presentations detailing our Users experiences with our software, you can also take part in Workshops, Training and tutorials as well as hands-on demonstrations of the CD-adapco software of your choice.

## STAR Conference France

Later this year the STAR Conference France, will take place at the **Forest Hill Hotel, Paris on the 19th September 2006**. Airbus, Alstom Transport and Valeo Thermique Habitacle are already confirmed to present, and IBM will sponsor the event. If you wish to submit a paper for this conference, or you wish to sponsor the event in some way, please email your request to **info@fr.cd-adapco.com**.

You will receive your invite and more information on this Conference shortly, but if you have any questions before you do – please email **info@fr.cd-adapco.com** for more information.

Also check **www.cd-adapco.com** for latest information.

## STAR European Conference

Our recent European STAR Conference in London saw the launch of STAR-CD V4, the next version of CD-adapco's flagship CFD code STAR-CD, and Users from across Europe present on a variety of topics.

Keynote speakers this year were: Jon Hilton, Technical Director, Renault Formula 1 Team and Philippe Anrigo, Head of Combustion, Renault France presenting on each of their organization's successful partnership with CD-adapco. We were also joined by Cosworth Ltd, Aerotherm Computational Dynamics, South Africa, Volvo Cars, Audi, PBMR, South Africa, CERN, Airbus Toulouse, ArvinMeritor - ProS3 –MSC, Icon, Van Oossanen & Associates b.v, Electrolux Major Appliances Europe, Sarov Labs, Southampton University and the University of Manchester.

During the conference there were also two well-attended technical workshops on:

Transitioning from STAR-CD 3.26 to STAR-CD V4 and Advanced meshing in STAR-CCM+ and hands-on demo points for the STAR-CAD Series and STAR-Design. We were supported during the conference by our partners from ABAQUS Inc, Adept Scientific, BULL, Fujitsu, Hewlett Packard, IBM, Icon, Intel, Intelligent Light, Microsoft, MSC, P&Z Engineering, SCAI, SGI and UGS.

During the evening on Day 1, all delegates, Partners and CD-adapco employees took part in a Casino evening after dinner with a prize of a 5.2-mega-pixel camera for the shrewdest gambler - kindly sponsored by Hewlett Packard. This was won by Roger Blom, FS Dynamics, who managed to win the most fun money during the evening. Microsoft also sponsored a draw to win a wireless mouse and keyboard, which was won by Carel Viljoen of PBMR and in the CD-adapco "Guess the number of Polyhedral Cells" competition, an iPod nano was won by Christof Hinterberger of Arvinmeritor.

To request a CD of the conference presentations please email; **amanda@uk.cd-adapco.com**



## STAR American Conference

The next STAR Conferences on our calendar will be our American meetings, taking place at the **Sheraton Detroit Novi in Michigan on May 23-24, 2006 and the Hilton San Diego Mission Valley in California on May 25th, 2006**.

We have some leading Keynote speakers joining us this year:

- **David Cole, Chairman, from the Center for Automotive Research in Ann Arbor**
- **Lawrence Wiley, Manager of Compliance, Documentation & Intellectual Property from GE Infrastructure-Wind Systems Engineering**
- **John L. Givens Jr., GM Powertrain Director**
- **Dr. James Laylek, Computational Center for mobility systems - Clemson University**

We would also urge you to join us for our CAE Panel Discussion with our Keynote Speakers and Industry Experts when we discuss:

**"Using Virtual Engineering as a competitive weapon for change"**
**on Tuesday May 23, 2006 at 11.15am**

At the STAR American Conferences you can learn the latest simulation techniques from expert CD-adapco staff and experienced users from North America and meet the people behind the software, from management to developers to support engineers. You can also network with other CFD users and hear how they employ CD-adapco software to get the best return on investment, experience our latest CAE technology, with seminars and hands on demos and listen to presentations from top industrial users, describing new and innovative applications of CFD technology.

You can register online here:
**http://www.cd-adapco.com/ugm_us/register.html**

## STAR Deutschland Konferenz

At our STAR Deutschland Konferenz in Munich last year, Dr. Gangolf Kohnen from Toyota Motorsport GmbH presented our keynote speech, about “The role of CFD in the Development Process of a Formula 1 Car at Toyota Motorsport GmbH”. We also had presentations from BEHR, Bombardier Transportation, promeos GmbH, Wärtsilä Propulsion, ArvinMeritor Emissions Technologies GmbH, FEV Motorentechnik GmbH, Thermotec-cfd GmbH, BMW AG, Germanischer Lloyd AG and TinniT Technologies GmbH amongst others.

Our exhibition of our Partners included IBM Deutschland GmbH, Unigraphics Solutions GmbH, IMAGINE Software GmbH, FE-Design GmbH, Silicon Graphics GmbH, Hewlett-Packard GmbH, LMS Deutschland GmbH and Engineous Software GmbH.

CD-adapco employees, Users and Partners gathered at the beautiful Alte Gärtnerei, in München-Taufkirchen for the traditional Conference dinner and entertainment. After dinner all attendees took part in a Sports quiz, to win leather footballs, tying in nicely with the FIFA World Cup starting in Germany this year!

During the conference Mr. Klaus Vehreschild from AUDI AG was the winner of the conference competition where each delegate was randomly given a key to a padlock on registering, and the winners key opened the box. The prize was a fantastic ticket to the German Grand Prix in Nürburgring.

To request a CD of the conference presentations please email;
info@de.cd-adapco.com

# Upcoming events

At CD-adapco, we regularly attend a variety of trade shows throughout the United States and Europe. We invite you to come and visit us at the following worldwide venues to discuss any of our CFD and CAE products or services and meet our team.

### ERCOFTAC Conference
May 30 - Jun 01, 2006
Porquerolles, France

### Power Gen
Jun 01-04, 2006
Booth A29
Cologne, Germany

### PTC/USER World Event Americas 2006
Jun 01-04, 2006
Gaylord Texan Resort & Convention Center
Dallas, TX

### AIAA Fluid Dynamics
Jun 06-08, 2006
Hyatt Regency San Francisco
San Francisco, CA
Booth 107

### ASME Frontiers in Biomedical Devices Conference
Jun 08-09, 2006
University of California Irvine
Irvine, CA

### DE IDEAS for Manufacturing Conference
Aerospace
Jun 13-14, 2006
Marriott Suites
Anaheim, CA

### DE IDEAS for Manufacturing Conference
Automotive
Jun 20-21, 2006
Sheraton
Novi, MI

### DE IDEAS for Manufacturing Conference
Electronics & Medical Diagnostics Manufacturing
Jun 26-27, 2006
Westin Waltham
Boston, MA

### AIAA Joint Propulsion
Jul 09-12, 2006
Sacramento Convention Center
Sacramento, CA
Booth 700

### MSC 2006
Americas Virtual Product Development Conference
Jul 17-19, 2006
Hyatt Regency Huntington Beach,
Resort & Spa
Huntington Beach, CA

### ICONE 14
Jul 17-20, 2006
InterContinental Miami
Miami, FL

# Training at CD-adapco

We regularly hold CD-adapco product training sessions at our offices in: London, Detroit, Seattle, Nurnberg, Paris and Turin. Other courses as listed on our website can be scheduled to suit your requirements and information can be requested from our training administrators (see below).

To register for a course use the online form, or request a faxable form from your training administrator:

UK
**info@uk.cd-adapco.com** • (+44) 020 7471 6200

USA
**training@us.cd-adapco.com** • (+1) 631 549 2300 x129

Germany
**traning@de.cd-adapco.com** • (+49) 911 946433

France
**info@fr.cd-adapco.com** • (+33) 141 837560

Italy
**info@it.cd-adapco.com** • (+39) 011 562 2194

Note: In most situations it will be possible to register trainees on the course of their choice. However, if requests for places on courses are received too close to the course date, this may not be possible. Availability of places can be obtained by contacting your local office. Please see below for our upcoming schedule of training courses. See our website for most up to date schedules.

| Location | Course | Date |
|---|---|---|
| **London** | | |
| | STAR-CD | July 10 - 12, 2006<br>Sept 11 - 13, 2006 |
| | pro-STAR advanced | June 15 - 16, 2006<br>Aug 17 - 18, 2006 |
| | STAR-CCM+ | July 14, 2006<br>Sept 15, 2006 |
| **Detroit** | | |
| | STAR-CD | Jun 5 - 7, 2006<br>Jul 10 - 12, 2006<br>Aug 7 - 9, 2006<br>Sep 11 - 13, 2006 |
| | pro-STAR advanced | Jun 8 - 9, 2006<br>Jul 13 - 14, 2006<br>Aug 10 - 11, 2006 |
| | STAR-CCM+ | Jun 4 - 5, 2006<br>Jul 6 - 7, 2006<br>Aug 3 - 4, 2006<br>Sep 7 - 8, 2006 |

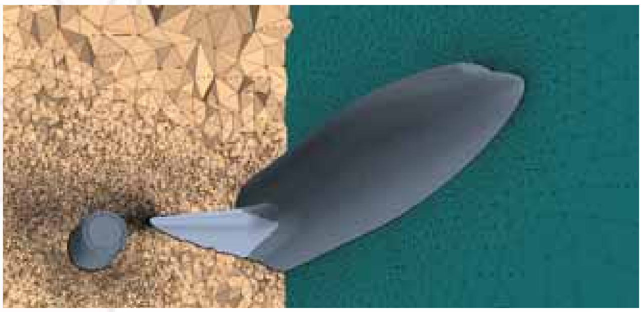
| Location | Course | Date |
|---|---|---|
| **Nurnberg** | | |
| | STAR-CD | Jul 18 - 20, 2006<br>Sep 19 - 21, 2006 |
| | pro-STAR advanced | Jun 27 – 28, 2006<br>Jul 18 - 20, 2006<br>Sep 19 - 21, 2006 |
| | STAR-CCM+ | Jun 29, 2006<br>Aug 24, 2006<br>Oct 12, 2006 |
| **Paris** | | |
| | STAR-CD | Jul 17 - 19, 2006<br>Sept 18 - 20, 2006 |
| | pro-STAR advanced | Jul 20 - 21, 2006<br>Sept 20 - 21, 2006 |
| **Turin** | | |

All training available on request at: info@it.cd-adapco.com

# That's another fine mesh...

**Dr. Mesh, CD-adapco**

If, like me, you come from the "blood, sweat and tears" school of CFD meshing, the recent developments in automatic meshing technology might have left you feeling a little disorientated. The problem is that many of us learnt our trade at a time when the only true measure of mesh quality was the number of hours that you spent building it by hand. Back then the automatic in "automatic meshing" was really just a byword for "quick but dirty."



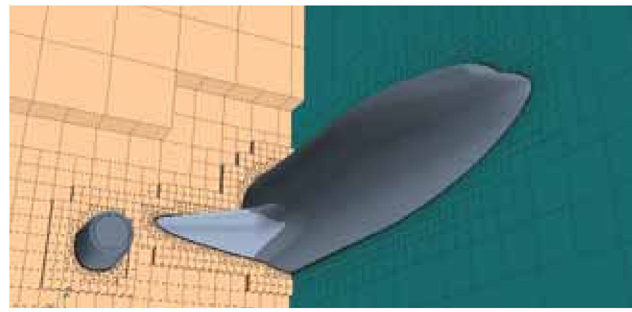
Not anymore. CD-adapco's investment in polyhedral meshing technology has resulted in a meshing technique that is as automatic as tetrahedral meshing but with an accuracy comparable to that of a hexahedral grid.

The most striking thing about polyhedral meshes is that they are so easy to create. If you haven't joined the polyhedral revolution already, I suggest that after finishing this article, you sit down and try to make a polyhedral mesh for yourself. My guess is that whether you are a closet hand-mesher or not, you will be pleasantly surprised by the result.

The easiest way to generate a polyhedral mesh is using either STAR-Design or the STAR-CAD Series. Designed to be used by non-CFD specialists, these products allow you to generate quality polyhedral meshes directly from your solid CAD model with no more than a couple of clicks of the mouse. Meshes are created from within the CAD environment and linked directly to the CAD solid model. Upon any change in the design, the mesh can be regenerated with a single button click, respecting the boundary conditions and resolution prescribed for the previous design iteration.

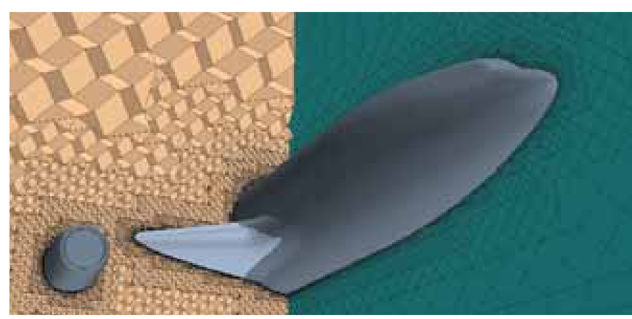
For those of you that prefer vertex-by-vertex control, polyhedral meshes can now also be generated through the Automatic Meshing panel in pro-STAR (for the recently released STAR-CD V4). In common with all of the polyhedral meshing approaches described here, extrusion layers are created automatically as part of the process.

The most powerful method of generating polyhedral meshes, however, is using STAR-CCM+ V2, in which the mesher is linked to the surface wrapper and surface remesher in a so called "meshing pipeline". This provides a fully automatic path from surface import to mesh generation (and then beyond to solution and post-processing). Once set up, the meshing pipeline defines all of the steps necessary to create a directly comparable mesh from a different starting surface. Upon generating a new mesh, STAR-CCM+ automatically maps the solution from the old mesh onto the new one, providing the best possible initial condition for the new simulation.



STAR-CCM+ V2 also includes a brand new trimmer that takes advantage of the fact that the new solvers don't distinguish between trimmed cells and any other type of n-faced polyhedra. Because the trimmer is no longer restricted to just a few trimmed-cell topologies it has far more latitude in its trimming. This means that "unresolved cells" and "couples errors" are a thing of the past (in fact there are no more couples at all – all meshes are now fully conformal).

This also opens the door to new trimming opportunities, users are able to trim from templates of tetrahedral and regular polyhedra as well as the traditional hexahedra.



Of course, you aren't just limited to one approach. Meshes can be transferred between STAR-CD V4 and STAR-CCM+ using the .ccm file, which includes not only the mesh, but also the boundary region definitions. STAR-Design Gateway and STAR-CAD Gateways can also be used to make meshes for either solver.

All of which leaves me with another problem: What to do with all the time that I used to spend creating meshes. Things have become so bad recently that I have even been considering a return to medicine...

Now where did I put those rubber gloves?

*Dr. Mesh*

Dr. Mesh surgery is open. Please send comments and suggestions to **dr.mesh@cd-adapco.com**


# Create a higher profile with CD-adapco.

CD-adapco is always looking for good quality case material as a solid way to outline product functionality and also the business benefits we can bring to our users. They allow us to demonstrate to our more sophisticated customers, how our products can and have delivered tangible benefits in a real business environment.

Our Sales force have a long list of customers who have purchased our products and are happy to endorse us. However often everyone is under the pressure of their next deadline and cannot spare the time to develop the endorsement stories. What if you could deliver the bare bones of a case study to someone and have the full written article delivered back into your lap for approval? CD-adapco has a Specialist Technical Writing Team available for that very function.

So what is the benefit to you, our customer, of investing this time in CD-adapco? We would like to display you in a favourable light within our extensive library of Client Case studies. A good, professionally written case study will portray your company as a forward thinking organisation that has been proactive in addressing critical business needs, responded to potential threats and entered into a contract with a reliable supplier. As well as being promoted in our newsletter, your case study might also feature in our literature and Trade Magazine articles and at Trade shows – creating a perfect opportunity to be seen by your Peers.

If you feel you would be willing to collaborate on a case study with us, get in touch with your Sales Account Manager or email **amanda@uk.cd-adapco.com** and let CD-adapco do some of the legwork for you.

# Global offices of CD-adapco

## Americas

**Headquarters**
**CD-adapco • New York office**
60 Broadhollow Road
Melville, NY 11747, USA
Tel.: (+1) 631 549 2300
info@us.cd-adapco.com
www.cd-adapco.com

Austin, TX
Atlanta, GA
Cincinnati, OH
Detroit, MI
Los Angeles, CA
Seattle, WA
Tulsa, OK
info@us.cd-adapco.com

## Europe

**Headquarters**
**CD-adapco • London office**
200 Shepherds Bush Road
London, W6 7NL, UK
Tel.: (+44) 20 7471 6200
info@uk.cd-adapco.com
www.cd-adapco.com

**France**
Paris office
Lyon office
info@fr.cd-adapco.com

**Germany**
Nürnberg office
info@de.cd-adapco.com

**Italy**
Turin office
Rome office
info@it.cd-adapco.com

## Asia-Pacific

**CDAJ**
**Japan**
37/F Yokohama Landmark Tower
2-2-1-1 Minato-Mirai · Nishi-ku
Yokohama 220-8137 · JAPAN
Tel: (+81) 45 683 1997
info@cdaj.co.jp
www.cdaj.co.jp

**China**
CD-adapco Japan Co. Ltd
Beijing office
cdbj@public.bta.net.cn

**CD-adapco Korea**
Seoul office
info@cdak.co.kr

## Resellers

**Australia**
Veta Pty
info@veta.com.au

**Greece**
ENEFEL
enefel@enefel.gr

**India**
CSM Software Pvt Ltd
info@csmsoftware.com

**Malaysia**
Numac Systems Technologies S/B
nst@numac.com.my

**New Zeland**
Matrix Applied Computing Ltd.
sales@matrix.co.nz

**Russia**
SAROV
info@sarovlabs.com

**South Africa**
Aerotherm Computational Dynamics
martin@aerothermcd.co.za

**Taiwan**
FLOTREND Corp.
buhner@flotrend.com.tw

**Turkey**
A-Ztech Ltd
info@a-ztech.com.tr

CD-adapco

**Your CAE Partner for Success**
**www.cd-adapco.com**