In my short time so far with CD-adapco, I've visited many of our industrial partners to discuss the future direction of our CFD products. Their message is clear, they want easier to use products that are more accessible to the non-CFD specialists that play the most influential role in the design process.

This isn't news to us. We have recently introduced a whole range of products that already offer the most outstanding ease-of-use on the market today. Products such as STAR-CCM+, STAR-Design and the STAR-CAD Series are redefining the way that CFD is being used in industry. We confidently claim that these products offer the simplest front door or gateway into the world's most powerful suite of CFD products.

However, as a company, we have no intention of resting on our laurels. Moves are afoot to improve and refine our customers whole CFD processes from top to bottom. The earliest sign of that is our new surface wrapping functionality, which automatically cleans and de-features dirty or complex CAD geometry with minimal user interaction.

Our strategic focus remains the same. CD-adapco's reputation has always been for tackling the toughest and most business-critical industry problems in flow simulation. This is a leadership position that we are proud of and intend to continue to enhance. We are the only CFD vendor to offer Full Insight, Full Spectrum CFD, able to tackle the full range of CFD problems with a single consistent set of highly-useable tools, integrated into our customers’ CAE processes and workflows.

Dennis Nagy
VP of Marketing & Business Development
CD-adapco.
If you are starting to read this article, you obviously have an interest in CFD and flow simulation. But the real question is: why? CFD has been around for decades and has made great progress in the speed and accuracy with which meaningful answers to real problems can be obtained. The key word here is “meaningful.” The old maxim “the purpose of computing is insight, not numbers” applies here, too. We do CFD in order to understand better thermal/flow-related behavior of products and processes being designed for commercial (read: money-making) purposes. Industry has also come to realize that design (product creation) drives overall product lifecycle costs, profit, and ultimately shareholder value to a much greater extent than the actual cost of the engineering development itself (i.e. a highly leveraged activity).

The better understanding gained through CFD should facilitate better (in the business payback sense) development decisions. For that to be the case with any kind of CAE simulation (of which CFD is becoming a more and more significant part), seven challenges need to be overcome (shown in Figure 1 in the historical order in which they have been encountered over the past decades).

For a number of reasons, CFD has historically lagged structural finite element analysis (FEA) and multibody dynamic analyses (MBD). But all forms of simulation have made enough progress to reveal to industrial management the time- and cost-saving potential of Virtual Product Development (VPD). Computer modeling and simulation can now be used to eliminate many costly physical prototypes (providing in some cases insight not observable from physical tests), shorten the development portion of the product lifecycle, and thus allow time for many more “what-if” examinations of design alternatives, looking further “outside the box” for really innovative product ideas.

This business payback is possible only if we all realize when, where, and how to apply CFD for greatest impact. This is where “hero mountain” and “payback meadow” come in (in case you were still wondering what that strange title meant). In my long career in the CAE software vendor industry, I have observed a situation characterized by the simplified pictorial graph in Figure 2.

Basically, the business impact of solving simulation problems in industry is not proportional to the technical difficulty of those problems. There is, instead, a collection of difficult problems of lesser overall business impact (pathological situations or “grand challenges”) that most R&D groups (and software vendors) have focussed on solving as an “entrance test” for the acceptance of a simulation technology such as CFD into mainstream product development processes. This “Hero Mountain,” which many simulation software vendors are climbing (forced by many companies’ R&D expert “gatekeepers” to climb?), obscures the presence of a much larger number of somewhat easier development problems that occur more often and, if solved regularly and in a timely manner, could have much greater overall (collective) impact.

If integrated, easy-to-use software simulation tools were regularly applied to these problems as part of a company’s established product development workflow, it would actually lead to broader continuous design improvement and realization of this greater...
business benefit ("Payback Meadow"). HP, among other astute companies, realized this phenomenon and created specific technologist positions within their product line organizations. These “Gardeners” had the assignment to find promising technologies and cultivate them for practical use in product development line organizations, as a counter-influence to the centralized R&D groups occupying the “Hero Mountains.”

Bringing this realization back home to our CFD topic, it means that CFD has a wide range of business usefulness across the whole spectrum of problem difficulties actually encountered in industry, as long as the tools are appropriately tailored to the various purposes:

- For quick examination of many more ideas early on in development, design engineers who are not “CFD gurus” need easy-to-use tools intimately linked to their conceptual design tools such as 3D CAD and solid modeling. This is often their first encounter with CFD, their “front door” to applying more advanced CFD more often, as their experience and problem challenges grow. CD-adapco provides the STAR-CAD Series to meet this entry-level “quick-study” need (where timely observations of trends are more insightful than precise answers), right within the most popular 3D modeling environments.

- As development detail evolves, the degree of accuracy needed in the simulations also increases, but these situations are still expected as a regular part of complex technical product/process development. Here, simulation-oriented engineers need specialized tools customized to their particular repetitive workflow. CD-adapco’s es-tools are developed to fill this need and we expect to see a dramatic increase in our development of vertical application-specific templates and application suites. Models developed, and experienced gained, in the early-stage use of the STAR-CAD Series can be migrated into this phase of product development without any loss of time or information.

- And, of course, Hero Mountain is always there. Specialists will always be called upon to tackle the toughest CFD simulation problems, where the combination of complex physics and complex geometry push the limits of available technology, either in the validation or trouble-shooting stages of the product development cycle. CD-adapco’s powerful range of solvers and advanced meshing tools, and our large, highly-experienced consulting services staff, are well-recognized for being able to meet any challenge on Hero Mountain, again without loss of experience or information obtained by the development engineers using compatible CFD tools from CD-adapco.

In short, CD-adapco is the leading provider of the full spectrum of relevant CFD software and experience that any global enterprise can integrate into their VPD processes for dramatic business payback improvement.
The use of CFD in the aerospace community has evolved. No longer viewed as a replacement for wind tunnel experiments, CFD has gained a reputation as a powerful tool for creative design. However, with this reputation comes new responsibility. CFD is increasingly required to “earn its keep” as a responsive mainstream tool, providing critical information within the restrictive timescales of the design process. Whereas, in the past, CFD engineers were able to spend months constructing a mesh for a complete aircraft simulation, competitive pressure means that now the entire aircraft polar needs to be simulated in just a few days. In order to meet this challenge, aerospace companies are increasingly turning to commercial CFD solutions. One such solution is CD-adapco’s newly released flagship code: STAR-CCM+.

In the past, commercial CFD codes have struggled to penetrate the aerospace flow-simulation market that is dominated by a myriad of established “in-house” codes, developed by (and sometimes shared between) the major aerospace companies. Often the comparison with in-house CFD was unfair as, having borne the significant cost of developing these bespoke tools, aerospace companies were often reluctant to consider discarding them in favor of commercial CFD. More often than not, it was indeed true that general-purpose CFD codes, developed to tackle a wide range of engineering problems, did not compare favorably with in-house codes on the specific types of problems that the latter were developed specifically to tackle.

STAR-CCM+ was designed to challenge this paradigm. Although a general-purpose code in spirit, STAR-CCM+ was conceived with the explicit requirement that the needs of the aerospace community had to be fully met. Offering accuracy, which is at least equivalent to in-house aero codes, STAR-CCM+ offers a host of additional benefits and does not require the years of “fine-tuning” which is a feature of many such solutions.

At its heart are two state-of-the-art coupled solvers: an implicit algebraic-multigrid solver and an explicit multistage Runge-Kutta solver. Throughout extensive testing and validation, STAR-CCM+’s coupled solvers have proved exceptionally robust and accurate when dealing with shocks, natural convection and other problems involving a strong coupling between velocity, pressure, and temperature (in both compressible and incompressible flows).

CD-adapco’s aerospace specialist, Stephen McIlwain, explains “Sometimes when engineers first encounter STAR-CCM+, they are so impressed by its user interface and ability to deal with polyhedral meshes that they completely overlook the fact that STAR-CCM+ is based around not one, but two first-rate coupled solvers.”

Unlike many coupled solvers (such as those incorporated within typical in-house CFD codes), STAR-CCM+’s solver is not limited to purely inviscid (Euler) simulations, it is backed up by a comprehensive suite of turbulence and multiphysics models.
STAR-CCM+ also includes a state-of-the-art segregated solver, which is ideal for low-Mach number simulations. Switching between the segregated solver and each of the coupled solvers is a trivial single step process. This means that, for example, the segregated solver can be used to quickly provide a sensible initial condition, from which the coupled solvers can be run to convergence. The major benefit of this is that users can use a single CFD package for all simulation regimes, whether subsonic, transonic or hypersonic.

In a presentation to the 2005 STAR European conference, Matt Milne of QinetiQ reported the progress of their evaluation of STAR-CCM+ as a front line CFD code for the simulation of complete civil and military aircraft configurations. He concluded, "The results of this evaluation indicate that when run on high-quality block-structured meshes, STAR-CCM+ yields high quality results of at least equal accuracy to existing in-house methods and other commercial codes. In particular, excellent agreement has been seen between measured and predicted levels of drag."

Polyhedral meshes offer the best of both worlds, the automatic meshing benefits of tetrahedra combined with near hexahedral accuracy. Extensive testing has shown that a typical polyhedral mesh requires less than one fifth of the cell count of a tetrahedral mesh to deliver an equivalent or improved level of accuracy.

Although, aerospace companies will continue to use their own in-house CFD for much of their work for the foreseeable future, STAR-CCM+ offers a solution that is a practical alternative today, and has the credibility to become the basis for a complete replacement in the future.

A free trial version of STAR-CCM+ can be downloaded from: www.cd-adapco.com

Please contact your sales representative for the full copy of STAR-CCM+. 
With the likelihood that traditional Spark Ignition (SI) and Diesel Compression Ignition (CI) Internal Combustion Engines (ICE) will fail to meet future limitations on engine emissions, engine manufacturers are increasingly focusing their attention to new combustion systems such as HCCI (Homogenous Charge Compression Ignition) and CAI (Compression Auto Ignition). At present, these engine concepts look unlikely to be able to deliver the high levels of specific power typically generated by SI and DI engines. Future production engines will probably rely on a combination of combustion systems; SI and CAI for gasoline, CI and HCCI for Diesel fuel. CAI and HCCI will probably be used to reduce pollution levels at low to medium operating points, while SI and CI will be used in high load scenarios.

The advent of these new concepts has provided a significant challenge for Computational Fluid Dynamics, a numerical flow simulation technique that plays an increasingly critical role in modern engine design. CFD is used in the development of every commercial engine, and is especially useful in promoting an early assessment of the relative value of new combustion concepts. Used effectively, CFD can significantly reduce the development cost of a new engine concept.

At the heart of every CFD engine simulation is a combustion model. Unfortunately, like engines, combustion models have classically been divided into two distinct types; premixed combustion models for SI engines and non-premixed combustion models for CI engines. This distinction has always been questionable as both SI and CI engines can exhibit both premixed and non-premixed combustion under certain operating conditions. However, with the introduction of mixed concept engines and the popularity of Gasoline Direct Injection (GDI), the strict distinction between different mixture regimes is no longer justifiable.

In order to address this challenge, CD-adapco has implemented ECFM-3Z (3 Zone Extended Coherent Flame Model) in the industry leading CFD code STAR-CD. Jointly developed by Renault and IFP (Institut Francais du Petrole), ECFM-3Z was developed to address the specific combustion challenges posed by Gasoline Direct Injection (GDI) engines.
Unlike traditional Port Fuel Injection engines GDI engines are characterized by a high degree of charge stratification. ECFM-3Z accounts for this by splitting each computational cell into 3 zones: an unmixed air zone, a mixed fuel and air zone and an unmixed fuel zone. By overcoming the traditional single mixture regime restriction of other combustion models, ECFM-3Z provides the first general purpose approach to combustion simulation. When combined with auto-ignition and spark ignition models ECFM-3Z is valid across a wide suite of engine concepts, including GDI, HCCI and CAI.

The new model is not only able to represent accurately the combustion behaviour in new engine concepts, but also provide new modeling capabilities for the investigation of traditional SI and CI engines. Using the auto-ignition model, for example, secondary combustion phenomena such as "knock" can be more thoroughly investigated in both SI and GDI engines.

ECFM-3Z is part of a wider trend towards universal modeling. Steve MacDonald, CD-adapco founder and President explains, "CFD is moving away from simple models that address only specific problems. As a CAE solution provider, we are committed to providing more fundamental modeling approaches that allow our partners to simulate the whole system, rather than just small parts of it."

Tom Marinaccio, CD-adapco's head of consultancy is keen to recommend the ECFM-3Z model for engine simulation, "For the first time we have a universal combustion model that we can use with confidence across the complete range of ICE simulation problems."

It’s official:

STAR-CD is the most used CFD code in Automotive Industry

The Top20Auto Survey ranks CD-adapco’s STAR-CD as the top CFD code in the Automotive Industry. The survey, compiled by “Top500 Supercomputer Sites”, places STAR-CD convincingly ahead of rivals Fluent and Powerflow in terms of the number of installed gigaflops for each package. Using this criterion, STAR-CD also ranks as the fourth top automotive CAE application in the Automotive Industry.

Richard Johns, CD-adapco’s Director for the Automotive Sector, is pleased but not surprised. “To have your leadership confirmed in an independent survey is always flattering, but the result comes as no great shock”, he said. “Our aim is not just to be the de facto industry leader, but to have the greatest impact upon the VPD processes of our partners”.

Dennis Nagy, Vice President of Marketing and Business Development at CD-adapco, sees increased dominance in the future, “We are the only CFD vendor to offer Full-Spectrum Flow Simulation software and services. No other CFD solution provider can cope with the full range of flow simulations and Virtual Product Development process situations required by the Automotive Industry. From individual CAD parts, to entire vehicle systems, STAR-CD can handle it”. ■

For more information contact info@uk.cd-adapco.com.
In the words of a famous song: “Birds do it. Bees do it. Even educated fleas do it.” In reality, although bees are very good at it, birds and fleas don’t do much polyhedral meshing – but then up until recently, neither did most CFD engineers.

Whether polyhedral meshes are the best for all types of CFD calculations, it still remains to be proved (although we think that the evidence in their favor is rather compelling). Nature, however, has come to her own conclusions and is full of examples of two and three-dimensional tessellations that bear remarkable similarities to CD-adapco’s latest meshing technology. In contrast, there are relatively few examples of naturally occurring hexahedral and tetrahedral mesh structures.

So how is it that honeybees (average brain size 1g) manage to out-mesh the vast majority of CFD engineers (average brain size 1250g)? The answer is obviously not that bees are more intelligent than engineers (although there are a few notable exceptions).

Whereas CFD and associated meshing technology has been around for just 30 years, bees benefit from several billion years of evolution. From a purely evolutionary viewpoint, the hexagonal structure of the honeycomb is the endpoint of an exercise in energy optimization.

The walls of each honey cell are fashioned from wax and are manufactured to a high tolerance (within 0.2% of their 100 micron thickness). Creating this wax costs energy that could be better-used making honey to rear the next generation of bees.

As Charles Darwin himself wrote:

“With respect to the formation of wax, it is known that bees are often hard pressed to get sufficient nectar...it has been experimentally proven that from twelve to fifteen pounds of dry sugar are consumed by a hive of bees for the secretion of a pound of wax.”

Darwin also described the honeycomb as “a masterpiece of engineering” that is “absolutely perfect in economizing labor and wax.”

Biologists have long contended that the honeycomb was the ideal structure for containing the maximum amount of honey while containing the minimum amount of wax, however mathematical proof of this so-called “honeycomb conjecture” was a long time in coming. The conjecture, which has been a subject of mathematical curiosity since the third century AD, wasn’t finally proved until June 1999, when Thomas C Hales of the University of Michigan, finally demonstrated conclusively that “a hexagonal grid represents the best way to divide a surface into regions of equal area with the least total perimeter”.

As good as honeybees might be at meshing in two dimensions, most practical CFD work requires three-dimensional meshing. (This might offer a degree of relief to any CFD engineers who are nervous about losing their job to a swarm of inexpensive meshing bees.)

In the same way as bees benefit from minimizing the amount of wax used in producing a certain volume of honey, face-addressing CFD solvers benefit from minimizing the number of faces used in a computational mesh for a given mesh resolution. (Using face-addressing, the solver must loop over all cell faces at every solution level – minimizing the number of faces obviously has a huge payback in terms of solver efficiency – see “The Advantage of Polyhedral meshing” Dynamics 24.)

From this point of view, tetrahedra are the worst type of computational cell. As the lowest order polyhedron they fill space less efficiently than any other element. If bees worked in three dimensions they wouldn’t use tetrahedra, as the cost of wax to honey would be too high. Although hexahedra are better from this point of view, they too are far from the ideal.

The obvious question is therefore: Which type of mesh has the fewest number of faces per unit volume? Once again nature has the answer...

In 1887 Lord Kelvin became intrigued by the packing of bubbles in a perfect foam – one in which all bubbles had equal volume. He asked himself “How would bubbles of equal volume pack together, to give the least possible amount of surface film between them?”
His answer was a 14-sided polyhedron that he painfully named the “tetrakaidecahedron”. This element was appealing because it led to a regular symmetric partitioning of space – offering an apparent improvement over nature, which uses a combination of irregular polyhedrons in real soap foams.

The tetrakaidecahedron stood as the best way of partitioning space until 1994 when physicists Denis Weaire and Robert Phelan rejected Kelvin’s symmetrical partitioning in favor of a more nature inspired solution. The so-called Weaire-Phelan structure is a mixture of 12 and 14 sided polyhedra that partitions space 3% more efficiently than Kelvin’s foam.

What does this have to do with CFD? Well, CD-adapco’s polyhedral meshes typically consist of cells of 12 and 14 faces (although the number of faces is unrestricted). This means that they fill space in close to the most efficient way possible. For a given resolution level, a mesh consisting of CD-adapco’s polyhedral cells has fewer faces than a mesh of any other cell type.

Apart from the obvious benefits of economy, polyhedral meshes provide other advantages too. Because each polyhedral cell has more faces, it also has more neighbors than traditional cell types. A tetrahedral cell communicates with only four neighbor cells, and a hexahedral just six. In both cases this limits the influence of each cell to just a few neighbors. By contrast each polyhedral cell has an average 12 or 14 neighbors. The net result of this is that information propagates much more quickly through a polyhedral mesh, ultimately leading to an increased rate of convergence.

In the same way that a polyhedral cell “speaks” to more of its neighbors than other cell types, it also “listens” to information from more of them. Because each polyhedral cell receives information from more of its surroundings, the cell centered values calculated for the cell are more accurate than for other types.

The downside? We’re not sure that there are any. While a flow fitted hexahedral meshes still offer some advantages, they are difficult and expensive to create (and if you know how to make your mesh truly flow fitted you probably don’t need to run the calculation in the first place). Polyhedral meshes can be created at the click of a button and have advantages in efficiency and accuracy.

Sometimes nature knows best.
With the addition of ECFM-3Z — the universal combustion model for Internal Combustion Engine (ICE) calculations — to the existing wealth of ICE functionality in STAR-CD and es-ice, engine designers have never had it so good. But where did it all begin?

We decided to take a trip down memory lane, to tell a story that began over 15 years ago in New York and London. It continues with CD-adapco building on its position as world-wide industry leader, by continuing to deliver the functionality that enables engine designers to model tomorrow’s engines (be they SI or CI engines, even when operating in HCCI or CAI mode) today.

Gerald Schmidt (CD-adapco’s Director of American Customer Support and long-time ICE champion) tells us how it all began, “It was back in February 1990 when we did our first consultancy for Renault. Because of the highly complex nature of in-cylinder flow and the difficulty in obtaining and understanding experimental data, we’ve always had a high demand for running ICE calculations. The Renault work was a steady-state, cold flow, intake port simulation, but by the following year we ran our first transient, moving mesh simulation for DaimlerChrysler. In those days all the mesh motion and connectivity changes had to be defined by hand in newxyz (by user coding). It took between 12 to 16 weeks to complete the model set-up and running.”

Even at this early stage, CD-adapco’s business model of close collaboration with end users and knowledge transfer through ‘expert system’ tools was already in action through the development of PORT, the first generation of CD-adapco’s ICE specific tools.

Steve MacDonald (President and founder of CD-adapco) takes over, “Along side our consultancy activities, we were always looking to increase the level of automation and enable our clients to run these calculations for themselves. With these goals in mind we developed PORT, which allowed users to mesh cylinders for steady-state calculations from a predefined template. The first version came out in 1993, and was followed by NPORT in 1996. Of course the beauty of this is that you know exactly what the user wants and..."
needs because when you’re running a consultancy, you are the user, and if you take a wrong turn, the next time you meet with the OEM [Original Equipment Manufacturer] they set you straight."

With NPORT, users could build complex engine configurations including the 4-valve topology template that many still use today, but they were limited to steady-state calculations; to run transient required considerable user intervention and know-how. The engine designers’ dream of being able to automatically generate the complete mesh motion for an in-cylinder calculation came true in May 1998 when the first version of ProICE was released.

“ProICE took our model set-up and running time from 12 to 16 weeks down to 2 or 3. It was a real step-change in ICE simulations”, says Gerald.

Of course the development didn’t stop there. In July 2002, ProICE became es-ice, and year-after-year it continues to improve. Today, with es-ice’s trimmed cell technique, the user simply defines the geometry and its motion, and es-ice does the rest. The model set-up time for a fully transient, moving mesh calculation has gone from 3 or 4 months, to 1 or 2 days, and we continue to make it faster. Users can increase the complexity of their model by ‘copying and pasting’ geometry to go from a single to multiple cylinders, simplifying it by automatically generating the mesh for a steady, cold flow simulation just by specifying a Crank Angle, automatically generating a sector mesh for faster turnaround, and much more.

Of course, all of this only focuses on mesh generation and setup. “Engine simulations remain one of the most challenging areas in CFD”, says Professor David Gosman (co-founder and Vice President of Technology at CD-adapco). “The challenge is not only to model the wide variety of physical phenomena (turbulence, compressibility, heat transfer, nozzle flow and atomization, droplet dispersal, evaporation and wall impingement, mixing, ignition, combustion, pollutant emissions), but to do so robustly, efficiently and accurately in the extreme conditions found in the combustion chamber, whilst the mesh shape and structure are dynamically changing.”

This is where STAR-CD strides ahead of the competition. Capabilities that are essential for ICE calculations, notably moving-mesh facilities and physics modeling, which have long been in STAR-CD, are only just being developed in some other codes. For example, the highly optimized geometry movement method of cell layer addition and removal (available in STAR-CD since 1990), removes the requirement for cumbersome and slow re-meshing methods. Spray models have been available in STAR-CD since 1992, and combustion models for SI and CI engines have been available since 1995. This means that while others are still working on developing these basic capabilities, STAR-CD is able to focus on delivering the latest state-of-art combustion models, such as ECFM-3Z, in a solver that has been and continues to be the tried and tested solution for engine designers all over the world.

In addition to the above, the benefits of CD-adapco’s policy of close collaboration with industrial and academic partners is apparent: the implementation of ECFM-3Z, in partnership with Renault, and the sponsorship of flamelet-based soot modeling at Lund University are the latest in a series of development initiatives to ensure that STAR-CD has the requisite tools-of-the-trade for ICE simulations.

In the highly competitive and changing world of engine design, one certainty is the continued presence of STAR-CD, helping to drive the engines of tomorrow.
CD-adapco is pleased to announce that Dr. Richard Johns, a globally recognized expert in bringing advanced automotive engine CFD to industry for over 30 years at Ricardo, AVL, and Perkins Engines, has joined the company as Director, Automotive Sector. Dr. Johns will spearhead all CD-adapco activities aimed at further strengthening our global market leadership in automotive CFD.

“Richard represents an optimal blend of proven technical excellence and business marketing acumen that makes him ideal to further strengthen and drive our Automotive Sector leadership position in CFD,” said Dr. Dennis Nagy, Vice President of Marketing and Business Development at CD-adapco. “His well established global relationships with key CAE management at most automotive OEMs and suppliers align very well with CD-adapco’s major customers and prospects to insure that we continue to meet their strategic needs for CFD and related CAE solutions.”

Dr. Johns joined Perkins Engines in 1969 as a student and, after graduating in 1973 with a degree in mechanical engineering, joined their research department. After 2 years of developing cycle simulation and thermal analysis software, Perkins supported his PhD study at Imperial College under Prof. David Gosman in engine CFD. During his 5 years at Imperial College he developed computational methods for the flow in diesel engine cylinders and the first discrete droplet spray model applied to in-cylinder flows. He was awarded his PhD in 1980 for this work.

He subsequently joined AVL in Graz, Austria where he established the CFD software group and was also responsible for the laser diagnostics used for validation of the computations.

In 1987 he joined Ricardo where, again, he established a CFD software group and was subsequently responsible for both the commercial software business and engineering CAE applications. He was appointed President of Ricardo Software in 1995 and, in 1998, to the Board of Ricardo Consulting Engineers Ltd.

Richard is married, with 2 daughters, and is based at CD-adapco’s London Headquarters.
The European CATIA Forum - ECF2005 - represents the first anniversary of the release of STAR-CAT5, CD-adapco’s revolutionary CAD-embedded CFD product for CATIA V5. STAR-CAT5 was the first product to introduce CAD-embedded CFD to the CATIA community at the conference one year ago.

CD-adapco’s CAD-embedded CFD range offers outstanding ease-of-use, and, for the first time, delivers industrial strength CFD to the Design and Product Engineering communities. With STAR-CAT5 the CFD functionality is provided from within the CATIA environment, and accessed as additional items from within CATIA’s specification tree. This intuitive approach to CFD means that even novice users are able to quickly obtain industrial strength flow simulation results. Based on the technology behind the industry leading STAR-CD and STAR-CCM+, STAR-CAT5 is a gateway to the most comprehensive flow simulation tools in the world.

STAR-CAT5 uses CD-adapco’s revolutionary polyhedral meshing technology, proven to be up to 10 times faster than traditional tetrahedral meshing techniques, and delivering faster, more accurate solutions, at a fraction of the computational expense. Unlike the raft of recent imitations, STAR-CAT5 creates meshes specifically designed for flow simulation and is not based on CATIA’s own meshing kernel, which was designed for meshing structural finite element simulations.

CD-adapco launched CAD-embedded CFD in 2001, with the release of STAR-Works, a CAD-embedded CFD package for the SolidWorks community. Building on the success of this original product, CD-adapco’s CAD-embedded products now include all of the major CAD systems, STAR-Pro/E for Pro-ENGINEER, STAR-NX for Unigraphics and STAR-CAT5 for CATIA V5.

CD-adapco’s Vice President of Marketing and Business Development, Dennis Nagy, is confident that STAR-CAT5 is ahead of the competition. “CATIA V5-embedded CFD has been a proven reality from CD-adapco for almost a year now” he said. “Why would anyone want to wait for other vendors’ continual previewing of unreleased products when they can get on with productive use of CFD within the CATIA V5 environment right now, including the proven unique speed and automation advantages of polyhedral meshing, using STAR-CAT5?”
As anyone who has ever ridden a bicycle on a windy day will testify, aerodynamics play a big part in cycling. Perhaps more than any other sport, top-level cycling is dominated by aerodynamics and, more specifically, the art of drafting. Drafting occurs when one cyclist rides in the wake of another, reducing their exposure to the oncoming air and ultimately the energy expended in cycling. The question of “how much energy?” is the topic of some debate. CD-adapco has recently undertaken a comprehensive CFD simulation in order to answer this question.

The influence of aerodynamics is most visible in the time-trial stages of the Tour de France. Here cyclists can be seen riding special carbon-fiber bicycles on which, at a cost of tens of thousands of dollars, each component is specifically designed to minimize the aerodynamic drag. Clad in aerodynamic clothing and helmets, riders are forced to adopt uncomfortable crouched position on their bikes, minimizing their frontal area and reducing their exposure to the oncoming air.

In the Tour de France, the time-trial comes in two distinct flavors: an Individual Time Trial (ITT) and a Team Time Trial (TTT). In the Individual Time Trial, each rider competes alone. With no other riders to draft behind, the ITT is known as “the race of truth”, a brutal contest of man and machine against the clock. In the Team Time Trial, riders compete as a team of (up to) nine riders. Each rider takes it in turns to ride at the front of the line, taking the full force of the oncoming air and providing a wake in which teammates can draft. As each rider tires they swing off the front of the line and drift backwards to the rear, recovering in the wake of the other riders in preparation for the next turn at the front.

Although time-trialing ability alone is not enough to win the Tour de France, each July the final recipient of the yellow jersey is almost certainly a time-trialist of supreme ability. Recent Tours have almost always been won by riders that dominate the time-trial stages; 5 times winner Miguel Indurain and now 7 time winner Lance Armstrong typically based their victories around large time gaps opened up in the time-trial stages.
By performing CFD simulations of a nine-man TTT, and a single cyclist ITT, it is possible to directly evaluate the influence of drafting. Although many individual cyclists have been subjected to wind-tunnel testing, the sheer size of a nine-man chain of cyclists makes physical testing impractical. In the CFD world too, entire TTT calculations were until recently, prohibitively expensive, requiring a great number of computational cells in order to sufficiently resolve the flow past the cyclists.

The CFD models were constructed and run using the STAR-CAD Series, a range of CAD-embedded CFD products. Unlike other CFD codes, which are restricted to using hexahedral or tetrahedral elements, STAR-CAD Series has the unique ability to create and solve upon meshes of arbitrary cell topology. Using specially created polyhedral elements (which typically have between 12 and 14 faces), CD-adapco’s CFD technology can provide near hexahedral accuracy with at least 5 times fewer cells than a typical tetrahedral calculation. The mesh for the TTT comprised almost 7 million polyhedral cells, equivalent in accuracy to a mesh of approximately 30 million tetrahedra.

The results of the simulations were illuminating. Compared with the lead cyclist, the drag of the rider in second place is reduced by 21% - a significant saving. The third rider feels a further small decrease in drag over the second, but from the third rider back all other cyclists experience almost identical drag.

As the riders are continually progressing towards the front of the chain, taking a short turn on the front, before freewheeling to the rear of the line, on average (assuming a constant rate of rider rotation and ignoring the effect of dropping back) the drag coefficient of a rider in the TTT is around 27% lower than experienced by an individual rider.

Perhaps the most surprising conclusion from the CFD simulation is that, despite feeling the full force of the oncoming air, the lead rider experiences lower drag than if he were riding an ITT at the same speed. The drag coefficient of the leading TTT rider is 0.277, while that of an individual rider is 0.285 [drag coefficient is measure of the force each rider experiences corrected for differences in size]. This rare example of “something for nothing” occurs because the second place rider reduces the influence of the lead rider’s wake, increasing his base pressure and consequently reducing the drag force that he experiences.

Despite the predicted reduction in drag coefficient afforded to riders in a team time-trial, the increase in speed over an individual time trialist is not large. In the 2004 Tour, Armstrong completed the 55km individual time trial with an average speed of 49.39 km/h, while his team won the 64.5km TTT at an average speed of 53.71 km/h. This increase in average speed of just 9% seems relatively small in comparison with the effort exerted by eight extra cyclists. However, as drag increases with velocity squared, a 4 km/h increase would require a greatly increased effort for an individual rider. One would expect the difference in speed to be much greater if all nine riders were time-trialists of Armstrong’s ability. In reality even the top teams consist of riders whose competence lies in areas other than time-trialing, so that the contribution of all nine riders is not equivalent. Because of this, the lead rider is also forced to temper his effort, so that the lesser time-trialist on the team can keep up, despite the reduction in drag. If gaps open up in the line, riders are liable to be left behind, cycling in the wind and unable to catch up.

Although CFD calculations are so far not routinely used in perfecting time-trialing tactics for the Tour de France, CD-adapco’s Dennis Nagy believes that they soon will be. “It’s only a matter of time”, he said, “the rewards in professional cycling are increasing all of the time, while winning margins are coming down. In the near future CFD could be the difference between winning and losing the Tour de France”.
Safer UAV landings
Dr. Ulf Specht, IABG mbH, Germany
Performing landing operations of a helicopter or an UAV (Unmanned Air Vehicle) on a helideck of a battle ship is a potentially dangerous situation for both the helicopter pilot and the ship’s crew. Therefore, a detailed understanding of the flow structures and the magnitude of the turbulent fluctuations can be used to reduce the danger; by defining ‘best practice’ guidelines on how to maneuver the ship, prior to a landing, to minimize risk. Taking into consideration the very detailed superstructure of the complete ship, STAR-CD can and has been used to predict turbulent flow fields.

**Background**
Reconnaissance is one of the main tasks of the German Navy. An UAV will ensure identification of objects beyond the horizon of the ship even in poor visibility. Since the landing operation should be a completely automated task, the software of the control system has to take into account anything that might affect the smooth landing of the UAV, i.e. the environmental conditions. Therefore it is necessary to predict the flow field around the ship, by considering different wind speeds and wind directions relative to it.

**CFD simulations**
Starting from the water surface level, a narrow box was meshed containing the complete ship. This included the complicated structure with a refined region near the helideck. The mesh was completed by adding blocks of hexahedral cells in front of the bow, behind the stern, portside, and starboard using STAR-CD’s arbitrary couple methodology above the tetrahedral cells. The final mesh consisted of approximately 2 million cells.

The flow was assumed to be steady, incompressible and turbulent. Turbulence was modeled using the standard high Reynolds number $k-e$ model.

**Results**
A general view of the flow field containing isosurfaces of high values of turbulent kinetic energy is shown in Figure 1. There is a remarkable production of turbulent kinetic energy due to the superstructure. Since most of the turbulent kinetic energy has been dissipated before reaching the helideck, these turbulent fields were considered not to have a considerable affect on the flow field around the helideck.

A large vortex generated by the main flow at the end of the hangar (similar to a backward facing step) has a significant impact on the landing procedure (Figure 2). This vortex interacts with the main flow, generating a shear layer and producing a region of high turbulent kinetic energy.

Simulating many different flow directions, these computations can provide a ‘best practice’ guide on how to maneuver the ship prior to a landing operation. Furthermore, the data sets of the velocities and the turbulent kinetic energy can be prepared as an input for a real time UAV approach and touch down simulation.

**Conclusions**
The detailed information of complex flow patterns obtained by STAR-CD simulations significantly improves the understanding of how turbulent fields are generated. Furthermore, the complete flow field, which is impossible to understand and visualize in experiments, can be used as an input for an UAV simulation environment.

For more information contact; specht@iabg.de

**Figures**
01: Streamlines and isosurfaces of the turbulent kinetic energy (general)
02: Streamlines and isosurfaces of the turbulent kinetic energy (helideck)
Improving hydraulic systems using CFD

Dr. Ulf Specht, IABG mbH, Germany

In full-scale fatigue testing of large aircraft, detailed information about the individual test apparatus can be used to optimize operation: saving both time and money! Such in-depth analyses can be dealt with easily and quickly using CFD, yielding much greater detail and understanding than from an experimental investigation alone. Here, STAR-Design was used to predict the pressure losses in the hydraulic system used to exert large forces on the outer wing, causing the wing to displace and deform (replicating a specified operating condition). The results were compared against experiment and very accurate agreement was achieved.

Testing environment

In order to apply the loads to the wings, a complex hydraulic system with a large number of hydraulic cylinders is required. During normal test operation, the hydraulic cylinders are controlled by servo valves, which are part of the overall control system. During switch off operation, only passive valves control the hydraulic flow and consequently the hydraulic pressure. These have to be individually adjusted during commissioning, therefore obtaining detailed information of the pressure loss between the chambers on both sides of the actuator piston. How to adjust the valves is critical.

CFD simulations

The geometry of the hydraulic channels at the actuator, the valve manifold and the valve itself were designed using STAR-Design. The volume mesh, consisting of approximately 1 million cells, was obtained by triangulating the surface in pro-STAR's surface meshing module and generating a trimmed mesh, with three extruded layers, in pro-STAR. A section of the mesh and a pressure distribution can be seen in Figure 1.

In order to obtain an overview, a steady-state case was analyzed first. The flow was assumed to be steady, incompressible and turbulent. The standard k-ε model was chosen to model turbulence. The fluid under consideration was a special hydraulic fluid. A normal operation temperature of 55°C and the corresponding viscosity of the fluid were used.

A typical flow field is shown in Figure 2. It is characterized by the strong production of swirl at the valve, while at other positions of interest in the fluid channel, the flow is only slightly influenced by turbulence effects. This means that the adjustable valve is the dominant cause of the pressure drop in the oil flow.

Comparison with experiments

Experimental investigations using the same setup have been performed in order to validate the CFD simulations. The pressure losses were measured at different flow rates and orifice diameters.

The agreement between calculation and measurement is very good (Figure 3).

Though the relation between flow rate and pressure loss seems to be almost quadratic, it was essential to understand the parameters that describe this relationship and to evaluate the situation at very low flow.

As the main result, the effects of the flow of the hydraulic fluid during connection of both chambers of a loaded hydraulic actuator were understood to a much higher degree than it was possible only by experimental investigations. The effect of geometry alterations can be understood by numerical evaluation much faster than by experiment, which helps to act faster and reduce cost.
Conclusions
A very accurate prediction of the pressure losses caused by a geometrically complex hydraulic fluid channel including an adjustable valve can be achieved using STAR-CD. It was demonstrated that the combination of a numerical investigation with experimental validation saved time and money.

For more information contact specht@iabg.de

Figures
01: Trimmed mesh and pressure distribution
02: Streamlines and pressure distribution
03: Comparison simulation with experiments
Introduction

There are examples of multiphase flows everywhere. Naturally occurring multiphase flows might include air bubbles rising in a glass of sparkling water, sand particles carried by wind, rain drops in air. In industry, illustrations are the injection of air bubbles in a bubble column, separation of particles in a cyclone separator and the spray drying of milk in a spray dryer.

Equations and models

In order to study flow processes using computer simulation, we first need to describe it using equations. These transport equations are obtained by applying the conservation laws of mass, momentum and energy to each fluid phase in the flow domain. From these transport equations we ascertain volume fraction, velocity, and temperature for each phase. Since the phases are generally moving at different velocities and have different temperatures, there are exchanges of momentum and energy between the phases. Correct modeling of these inter-phase exchanges is one crucial factor in a successful simulation.

Taking inter-phase momentum exchanges as an example, the following forces can be identified: drag, turbulence drag, lift and virtual mass. These are exerted between the phases due to their relative motions. Empirical correlations for these forces are well established. As the particle concentration increases, inter-particle effects become increasingly significant, so that modifications to these forces must be considered. Fortunately, the required equations, models and their solution methods are readily available in STAR-CD from CD-adapco.

The art of formulating and solving the required system of transport equations together with the appropriate interaction terms is known as Computational Multiphase Fluid Dynamics or CMFD. The best way to illustrate the power of this computational technique in flow analyses is by way of examples, described below.

Multiphase mixing vessels

Mixing vessels operating in multiphase flow regimes are commonly found in the chemical and process industries. Examples include: catalyst particles that are introduced into vessels to promote specific reactions, or gas bubbles that are injected in order to provide chemical species for reactions such as oxygen from air bubbles.

Figure 1 demonstrates the computed flow pattern and void distribution in a mixing vessel with a downward pumping, pitched blade impeller. We can clearly see the recirculating flow generated by the impeller in the lower region, and the bulk circulation over the whole tank. The void fraction plot shows that some bubbles are trapped by the recirculating flow resulting in increased gas volume fraction towards the center of the recirculation. Images like this provide valuable information to an engineer, promoting better understanding of the flow dynamics, the spatial distribution of the phases and what these mean in terms of reactions, heat and mass transfers.

Suspension of solid particles in liquids is a common feature of many industrial processes. To prevent settling of particles, impellers stir the mixture to maintain uniform distribution. Plant operators often ask “What is the optimum speed for the impeller to prevent settling?” Researchers at The University of Palermo have carried out a series of experiments [1] using STAR-CD to correctly compute the particle suspension level at different stirrer speeds.

Comparisons between the computed and the experimental results show that the particle suspension levels at three different stirrer speeds (300, 380 and 480 rpm) are in good agreement.

Liquid-liquid extraction column

Liquid-liquid extraction is often used in the petrochemical industry to promote mass transfer between two fluids. To provide maximum contact between the two fluids, a counter-current flow arrangement is used as in the example shown in Figure 2. The heavier fluid is introduced through a central inlet at the top of the column and a distributor screen is used to distribute the fluid. The lighter fluid enters the column through the central inlet at the bottom.
Perforated trays are placed horizontally in the column to provide further contact between the two fluids in similar fashion to a distillation column. The two fluids can leave the column via the bottom or the top outer annuli. The flow inside this column is indeed complex. In the computed solution, we can clearly see the expected collection of the heavier fluid on the trays, the rolling-off at the tips of the trays and the cascade down the column. The computed solution closely resembles the experimental results [2].

**Air-lift reactor**

Air-lift reactors are also found in many applications, usually to provide the oxygen needed in an oxidation reaction in a liquid, to feed the biomaterials in a bioreactor, or to lift or to stir a liquid. In the example shown in Figure 3, the reactor is made of a straight cylindrical column with a central draft tube. Air is injected via a ring sparger placed in the outer annulus formed by the draft tube. The injection of air lifts the air-liquid mixture up the outer annulus (the riser) and the air disengages and escapes through the free surface. The liquid then circulates down the draft tube (the down-comer). Under some circumstances, the downward liquid flow can even be strong enough to pull some bubbles down the down-comer. The quantity of gas bubbles pulled into the down-comer and the depth they penetrate will depend on the speed of the liquid flow.

In this example, measured data is available for comparing the gas hold-up in the riser and down-comer against a wide range of gas injection rates [3]. The results show that gas hold-up in such a column can be predicted reasonably well.

**Settling tank**

The settling of heavy particles from a liquid stream is an important step in separation, mineral processing and in the recovery of catalyst particles in chemical processes. One special feature about solid particles settling on top of each other is that it is not possible for the particles to fill up 100% of the available space. There are always small gaps between the particles. For solid spheres the maximum packing density is around 60 vol.%. The solid-pressure force model is used to represent the inter-particle forces on particles settling on top of each other and ensure the correct maximum packing limit of the settled layer is obeyed.

**Fluidized bed**

Fluidized beds are often found in the petrochemical industry in form of a Fluidized Catalytic Cracker (FCC) and in drying of solid particles. The local concentration of solid particles is often near the maximum packing limit. Since particles are fluidized and densely packed, consideration of particle collision is critical. For fluidized bed applications the kinetic theory model for granular flows is commonly used in CFD simulations.

**Conclusions**

Multiphase flows are generally complex and prominent in many industrial processes. Modeling multiphase flows requires a good handle on the latest numerical techniques and having the appropriate and correct models to represent the different physics involved. Some of the difficult challenges in modeling multiphase flows have now been met with the use of CMFD and we have been able to demonstrate some successes in this article. The CMFD analysis technique is now readily available in STAR-CD.

Engineering analysis tools such as CMFD and CFD will continue to go from strength to strength as engineers across all industries witness the growth in application of this technology and the complexity of the problems it can address.

**References**

2. Total Fina Elf EU Brite/EuRam Project BE-4322.
3. EniChem EU Brite/EuRam Project BE95-2039.

**Figures**

01: Gas-liquid mixing vessel
02: Air-lift reactor and comparison of gas hold-up results
03: Computed solution from Total Fina Elf
When Mario Caponnetto compares Americas Cup Racing to another CFD intensive sport, he does so with certain a glint in his eye. “In many ways, it’s very like Formula 1”, says Luna Rossa’s CFD Team Leader, “We have to contend with the same contradiction. The sponsors and spectators want close races, but as designers, we want to win by any means – and if possible by a large margin”.

While Caponnetto’s comparison is certainly valid – Formula 1 and Americas Cup yacht racing both rely heavily on numerical flow simulation - it betrays a degree of modesty. Whereas Formula 1 CFD engineers need only concern themselves with a single medium (air), Caponnetto’s team (Francis Huber and CD-adapco Thesis prize winner Evan Spong) have to account simultaneously for the influence of both air and water as well as the interaction between them.

From a designer’s point of view, the 32nd edition of the Americas Cup is the toughest ever. In an attempt to reduce winning margins to just a few seconds, organizers have fixed the length, weight and sail area of competing boats. By fixing the critical dimensions of the boat, designers are essentially limited to adding extra appendages and modifying the shape of the hull.

This means that each design team is working to a very similar specification. Although original “ground-breaking” solutions are still possible, yacht design (like Formula 1) now tends to proceed with small and continuous improvements in every aspect of the boat (dynamics, structures, hydrodynamics and materials). It is within these constraints that CFD demonstrates its true value.

Caponetto explains: “Without the help of CAE tools such as CFD, such incremental changes would be very difficult to analyse. Using CD-adapco’s CFD tools we can quickly assess the influence of design changes without having to worry about the influence of scaling.”

Scaling is important. The standard tools of Naval Architecture - towing tank and wind tunnel analysis - operate on a scaled model of the actual boat (competition rules restrict models to a maximum of one-third scale). Since different scaling factors are required for similarity of both Reynolds number (relating to aerodynamics) and Froude number (relating to wave dynamics), comparison with the full-scale design is always subject to a degree of uncertainty.
“Testing of sails in a wind tunnel with the correct Reynolds number is particularly difficult,” says Caponnetto. “This makes it very difficult to test the complete vessel in a situation that simulates the true dynamics of the sea. In order to do this, the test would need to take account of the varying intensity of the waves, the direction and force of the wind and of course the route the boat is taking.”

Until recently CFD was typically used in as a numerical alternative to model testing in naval design. Hydrodynamic and aerodynamic calculations were performed independently. The hydrodynamic calculations accounted for the flow around hull and appendages aimed to assess the impact of the design on resistance and stability. Aerodynamic simulations concentrated on the sails and mast and were concerned with propelling force and tilting momentum.

Although interesting in their own right, these uncoupled calculations provide little direct information on the overall stability of a boat, which relies on a combination of aerodynamic and hydrodynamic influences. Results from the uncoupled calculations were typically combined manually using a VPP (velocity prediction program) that combined the results from testing or analysis and predicted the equilibrium position of the boat. Although this technique has been used to great effect by Luna Rossa and others, it is time consuming to perform and requires a degree of interpolation between conditions.

The Luna Rossa CFD team are busy pioneering a new type of coupled CFD calculations that combine hydrodynamic and aerodynamic calculations in a single model. Using CD-adapco’s unique 6 degrees-of-freedom solver, it is possible to accurately calculate the behavior of the boat design in a number of wave conditions and through a full range of maneuvers. The interaction between air and water is accounted for using a free-surface technique that accurately predicts the shape of waves generated at the waterline.

As well as providing the type of repeatable condition that allows the influence of small design changes to be accurately assessed, coupled CFD also allows detailed flow visualization that can help discover hitherto unnoticed flow phenomena that have a strong influence on the boat.

“It is the future,” says Caponnetto. “CD-adapco’s CFD tools are already making a huge contribution to the excellent performance of the Luna Rossa team. Working together we hope to provide vital design analysis information, which will hopefully prepare Luna Rossa for our first Americas Cup.”

The signs are good. After 5 out of 13 acts, Luna Rossa lead the Louis Vuitton Cup that will determine the challenger for the 2007 Americas Cup.

Details correct at time of going to press.
To meet the needs of graduates entering the work force, the school of Mechanical Engineering, at Purdue University Indianapolis, proposed an introductory undergraduate course in CFD. At the same time, one of the University’s senior engineering students expressed an interest in learning CFD independently, but needed only one or two credit hours to complete her graduation requirements. To meet the needs of this student as well as develop the underpinnings of a full undergraduate course, the University created a guided CFD introduction for this student using STAR-Design.

The aim was to develop a hands-on program of instruction to emphasize how CFD can be used to solve engineering problems and can require little or no CFD background to get started. In the spirit of problem-based learning (PBL), equations, terminology, and CFD concepts were to be covered during set-up and post-processing of actual cases, rather than via lectures beforehand. The University also aimed to enhance students’ understanding of how to use CFD as a virtual reality tool. While the CFD software needed to be simple to use, the student still needed to be able to access advanced capabilities and even to export the solution for post-processing with their own custom software.

The two problem cases were created using STAR-Design and were two-dimensional.

The first case, a semi-tractor trailer truck, exemplifies an external flow that is both familiar and has obvious aerodynamics shortcomings to address. In a typical introduction to CFD, a student might be given the geometry and mesh with which to carry out his or her first computation. Because STAR-Design has CAD modeling built in, a simplified truck configuration was requested and the student was allowed to create the geometry. The automatic mesh generation within STAR-Design is based on a solid model, and the student simply subtracted the truck from the surrounding block to produce the flow domain.
Figure 1 shows velocity vectors for the 2D truck. Using the post-processing within STAR-Design, the student identified separation, boundary layers, the wake, and learned about problem areas of the solution. For example, she could point out what areas of the flow field had high gradients with insufficient grid to resolve them. She also proposed modifications to improve the aerodynamics and computed the resulting flow field.

Velocity vectors for the modified truck are shown in Figure 2. With these two solutions in hand, the University's own post-processing software was used to carry out a control volume analysis and compared drag. This step required that the mesh files and flow data be output from pro-STAR, which was simplified by providing a script.

As an internal flow example, the flow around an axial compressor blade section was chosen. For this case the geometry model was provided to the student after entering the airfoil coordinates into STAR-Design and creating the domain boundaries. This case involved adjusting the mesh parameters to resolve the blade boundary layer and creating periodic boundary conditions. The latter was accomplished in pro-STAR by using a script that was provided. In addition to carrying out the computation, the student studied the relationship between pressure and velocity by examining several flow fields and plotting non-dimensional pressure rise against inlet axial velocity.

Using STAR-Design, CFD concepts can be introduced to an undergraduate student with very little software learning curve. The student learned how to obtain qualitative and quantitative results for relevant engineering situations. Meanwhile, this experience laid the groundwork to develop a full undergraduate course.

Figures

01: Velocity vectors for baseline 2D truck.
02: Velocity vectors for 2D truck after student's geometry modifications.
03: Velocity for 2D linear compressor cascade at low flow condition showing boundary layer separation.
04: Mesh for 2D linear compressor cascade with enlarged view near airfoil leading edge showing STAR-Design customization for boundary layer. Contours show velocity magnitude for high flow condition.
05: Graph - plotted pressure rise characteristic showing eventual decrease for low flows.
Recognizing the importance of supporting the academic engineering community, CD-adapco is offering your university the opportunity to take advantage of a special licensing program. Your college or university has the opportunity to take advantage of STAR-CCM+ at 3 different levels of licensing options.

CD-adapco’s new CFD code STAR-CCM+ is the new standard in the industrial CFD community. State-of-the-art modeling and software technology has resulted in a code that is above all, renowned for its outstanding ease-of-use.

Benefits to the User:

- Tree-structured, step-by-step, graphical user interface
- General polyhedral cell formulation. Delivering more for less: more accurate solutions, using fewer cells
- Guaranteed results from robust, crash-free, STAR-CCM+ solver
- Transparent scalable parallel operation
- Legacy mesh compatibility: STAR-CD, ICEM, GridGen, Gambit
- Visualization and dynamic steering during analysis.

For more information contact: david.neidhart@us.cd-adapco.com
(+1) 734 453 2100 or visit: http://www.cd-adapco.com/support/university.htm

Pricing restricted to the Americas
W hile thumbing through my postbag on these long hot summer days, I’ve noticed a number of you STAR-Design users champing at the bit to discover more about CD-adapco’s easy-to-use CAD package. Never Fear! Dr Mesh is here!

Retake control
A real time-saver is the “Ctrl” key that true to its name, gives you back control. Ever been halfway through defining a piece of geometry, or been operating in “Select” mode and found that you need to re-orient the view, zoom in or out, in order to complete the task? Instead of stopping and restarting, having changed your view, simply hold the “Ctrl” key down. This tells STAR-Design to temporarily revert to “View” mode, allowing you to rotate the geometry with the left mouse button, zoom in and out with the middle mouse button and translate the geometry with the right mouse button. If you are midway through defining a shape, STAR-Design will pause the operation while the “Ctrl” key is down and you change your view. As soon as you lift the “Ctrl” key, you’re back to normal operation.

Pick internal faces
Next, if I have geometry with internal faces, how do I select one the internal face? Right mouse button click does the job. In figure 1, you can see a box with a smaller box shape cut out of it. If I want to pick one of the inner faces I hold the mouse over the face and click the right mouse button. The first click will select the front face (as in figure 1). The second right mouse button click picks the closest internal face (figure 2), the third highlights the next face in line with the mouse (figure 3) and the fourth activates the last face, on the opposite side of the outer box (figure 4).

Running in batch
If you want to run your STAR-Design job in batch mode, there are two different ways of doing this. The first runs STAR-Design entirely as a batch job, so the GUI never gets started. This can be done with the “stardesign --batch < Script.inp > OutFile” command line option.

The second method allows you to run a batch file of commands once STAR-Design has been started. This requires the STAR-Design command line to be activated. All STAR-Design’s actions have an associated “command”, which is echoed to a “casename.echo” file in the run directory. You can activate the command line by using the “--expert” option. Having activated the command line, the user can run batch scripts using the “ifile Script.inp” command.

Until next time,

Dr. Mesh

Dr. Mesh surgery is open. Please send comments and suggestions to dr.mesh@cd-adapco.com
Two more highly successful STAR Conferences took place on May 17 & 18 in Novi, Michigan and May 19, 2005 in La Jolla, California. As always, the STAR Conference featured presentations about the latest features and developments of CD-adapco’s CFD products and services, with special focus this year on:

STAR-CCM+

CD-adapco’s next generation CFD product, and,

STAR-CAD Series

A set of design-centric CAD-embedded tools that bring CFD to the very earliest stages of the design process.

CD-adapco’s President Steve MacDonald kicked off the day by welcoming a full audience to the day’s events. Riaz Sanatian, Director of Project Management, gave a presentation explaining the product roadmap and the future strategy of CD-adapco. Dennis Nagy, Vice President of Marketing and Strategic Planning, then talked about Virtual Product Development and how CD-adapco supports this process.

Keynote speakers

The morning was capped off with two keynote presentations: Monica Schnitzer, Senior Vice President of Market Analysis from Daratech, Inc. who presented “Leveraging Analysis for Competitive Advantage”, followed by Dr. Wolfgang Gentsch, Managing Director of Grid Computing and Networking Services, MCNC who presented “CFD and Grid Computing: A New Opportunity for Research and Business”. The keynote speeches were extremely well received by all attendees and we thank Monica and Wolfgang for taking the time to present at our conference in Michigan.

Special thanks

We would especially like to extend special thanks to all our presenters who made both STAR Conferences such a triumph. These include:

• David Weber from Argonne National Labs
• François McKenty from Brais, Malouin & Associates
• Stefan Kern from Institute of Computational Science, Swiss Federal Institute of Technology
• Chris Zhou, Wright State University
• Buvana Jayaraman, Lawrence Berkley National Laboratory
• Ramu Ramamurthy, Eaton Corporation
• Idil Papila, ISP Chemicals
• Tom Gielda, Whirlpool Corporation
• Norbert Moritz, B&B Agema
• Hany Hakila, Embry Riddle Aeronautical University
• Mohamed Hassan, University of Kentucky
• Nadir Yilmaz, New Mexico State University
• Isabelle Lavedrine from ARUP
• Jiang Luo from Solar Turbines
• Hengchu Cao from Edwards Lifesciences
• Alan Diner from MSC Software

Celebrating 25 years

On the evening of May 17th, CD-adapco invited all attendees to help us celebrate 25 years in the industry, at a highly enjoyable casino night. Great food and fun was had by all. A few lucky participants from both conferences took home prizes, which

STAR American Conference 2005 a great success!
included Paddock Club Passes to the Renault F1 Suite, a weekend in Las Vegas, a digital camera and printer, AMEX gift cards and other gifts. We are very grateful to HP, IBM and SGI for sponsoring our casino night, as well as providing computers for our Training Sessions on the second day. A special thank-you also to AMD and Linux Networx for sponsoring the conference luncheons.

Training sessions
A new format was introduced on the second day of the Detroit conference. Several in-depth, hands-on training sessions were held instead of additional client presentations. Some sessions were for CD-adapco products and some were given by sponsoring partners of CD-adapco. Overall, the feedback about this new type of session within the conference has been very positive.

See you next year!
We would like to recognize and thank all of our sponsoring partners who participated at both locations. Your support is appreciated and we hope to see our partnerships increase over the next year.

The conferences overall, according to the attendees, exceeded their expectations. We are delighted to hear this and we feel this is representative of all the hard work that the team at CD-adapco put in to organize the conferences. Again, thanks to all involved: attendees, presenters, exhibiting partners and CD-adapco team alike. A CD will be available shortly, of the conference presentations. All attendees will automatically receive a copy, but if you did not attend and would like to receive one, please contact loretta.yeager@us.cd-adapco.com.
D-adapco's STAR Deutschland Konferenz will take place on November 7-8, 2005 at the Munich Marriott Hotel, Munich, Germany.

The Conference will be a combination of presentations and workshops given by clients and key CD-adapco employees to provide invaluable knowledge and communicate future developments to the CFD community. The program will include presentations about the latest features and developments of CD-adapco's CFD products and services, with special focus on:

- **STAR-CCM+** The new empowered computational continuum mechanics code with unique benefits in terms of meshing speed and robustness, solving accuracy, quicker turnaround time and multiphysics capabilities.

- **STAR-CAD Series** - A set of CAD-embedded, CFD products capable of performing simple flow and heat transfer simulations for design engineers who do not require the full complex-physics functionality of our other solvers.

Delegates will get the opportunity to meet with employees of CD-adapco and fellow CFD users and to hear presentations given by industrial clients about real world CFD applications. An exhibition of our hardware and software partners will also take place. This invitation extends to anyone in the CFD community and other interested parties, not just existing users of CD-adapco.

To submit a presentation to be considered for the conference, please send a 100-200 word abstract to info@de.cd-adapco.com. Please also use this email address if you wish to receive more information about being an exhibitor or a sponsor.

You can register online by visiting [www.cd-adapco.com/ugmde05](http://www.cd-adapco.com/ugmde05).

**Location:**
Munich Marriott Hotel
Berlin Str. 93
80805 Munich
Germany

Tel: +49 (0)89 36 00 20
Fax: +49 (0) 89 36 00 22

[www.marriotthotels.com](http://www.marriotthotels.com)

Registration cost: Euro 150

Watch your email and our website for further information regarding the STAR Deutschland Konferenz. We look forward to seeing you there!
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:

**Turbomachinery Symposium**  
13 - 15 September 2005  
Houston, TX  
Booth 107  
http://turbolab.tamu.edu/turboshow/turbo.html

**AIAA Unmanned Aerospace**  
26 – 29 September 2005  
Arlington, VA  
Booth 417  
www.aiaa.org

**ASME Design Automation Conference**  
26 -29 September 2005  
Long Beach, CA  
http://swhite.me.washington.edu/~asmeda/

**CPDA PLM Roadmap Conference**  
28-29 September 2005  
Dearborn, MI  
http://cpd-associates.com

**European CATIA Forum (ECF)**  
4-6 October 2005  
Frankfurt, Germany  
www.ecforum.com

**Aachener Kolloqium, Aachen**  
4 - 6 October, 2005  
Aachen, Germany  
http://www.isb.rwth-aachen.de/amus/

**The 2005 Engineous Software International Symposium**  
10-11 October 2005  
Troy, MI  
www.engineous.com

**CATIA Operators Exchange (COE) Automotive Workshop**  
10-11 October 2005  
Detroit, MI  
www.coe.org

**CATIA Operators Exchange (COE) Aerospace Industry Workshop**  
17-18 October 2005  
Wichita, KS  
www.coe.org

**IMExpo (Maritime Expo)**  
George R. Brown Conv. Center, Houston, TX  
19 – 22 October 2005  
Booth 434  
www.worldmaritimetechnology.org/

**PTC Paris Conference**  
20 October 2005  
Paris, France  
www.ptc.com

**PTC Central Europe Conference**  
23-25 October 2005  
Friedrichshafen, Germany  
www.ptc.com

**Daratech Digital Prototyping Strategies (DPS) Conference**  
24-25 October 2005  
Novi, MI  
www.daratech.com/dps2005/

**MSC VPD Conference**  
24-26 October 2005  
Munich, Germany  
www.mscsoftware.com

**Unigraphics (UGS) CAE Conference**  
3-4 November 2005  
Frankfurt, Germany  
www.ugs.com

**Supercomputing 2005**  
14-18 November 2005  
Seattle, WA  
http://sc05.supercomputing.org/

**Global Motorsports Congress**  
17 – 18 November 2005  
Frankfurt, Germany

NB: Attendance at these shows is subject to change

---

**Announcing the STAR European Conference 2006.**

And a little further ahead on your calendar, on the 20-21st of March 2006, CD-adapco will once again hold the highly successful STAR European Conference in London.

This two-day event will include presentations on CD-adapco’s latest CFD software developments and industrial applications, as well as product demonstrations, workshops and an exhibition for hardware and software partners. CD-adapco’s software users in Europe will present the results of their work and share their experiences with other users.

To take your place in next year's Conference agenda or to support us by Sponsoring or Exhibiting, please contact: conference.info@uk.cd-adapco.com by the end of September 2005. We look forward to welcoming you at the STAR European Conference 2006.
Training for all of CD-adapco’s products is available in a range of courses designed to suit your requirements. We regularly hold product-training sessions at our offices in London, Detroit, Paris, Nuremberg and Turin. In addition to these structured courses, we continue to offer customized training courses on request, to companies wishing to expand their CFD knowledge.

More experienced users of STAR-CD can attend a wide range of advanced training courses in areas such as moving mesh, user subroutines, sprays and combustion, advanced modeling, techniques and several other topics. For specialists, training is provided in vehicle thermal management, fluid-structure interaction, applied heat transfer, rotating machinery, es-ice (engine design), es-aero (external aerodynamics), es-uhood (underhood analysis) and other Expert System tools.

### STAR-CD Course description

**Course Duration:** 3 Days  
**Course Schedule:** Monthly

### Who Should Attend

Any new user to STAR-CD or past user requiring a refresher course to improve their experience.

### Course Aims

The course is aimed at giving new STAR-CD users an understanding of the processes involved in setting up a typical CFD simulation, and to give extensive hands-on experience in using the software. This is in the form of working examples, which incorporate the use of the pro-STAR GUI, graphics and command windows and the on-line help facilities. The course also gives the user the opportunity to discuss the application of STAR-CD to their field of interest with an experienced Engineer.

### Course Content

1) **Introduction to CFD:** Gives a general introduction to CFD, what it can be used for and the modeling capabilities of STAR-CD.

2) **Running pro-STAR:** Shows how to start pro-STAR from a terminal window. It then demonstrates the use of the graphics and command windows and the GUI. It also discusses the various files associated with pro-STAR and the STAR solver and the file connectivity between them.

3) **Creating and Running a Simple Model:** Takes the user through the processes of running a complete session to include the building of a simple model, running the solver and viewing the results.

4) **Meshing - Basic Strategies:** Gives an outline of the basic meshing strategies in pro-STAR to include meshing simple geometries, extruding a 3D model from a 2D surface and block meshing.

5) **Meshing - Fundamental entities:** Provides an understanding of the fundamental geometric entities that can exist within pro-STAR and which allow the definition and manipulation of a model geometry. The entities discussed are co-ordinate systems, vertices, splines, cells, couples and boundaries.

6) **Meshing - Working Example:** Gives hands-on experience in building a mesh in pro-STAR using the various meshing techniques previously covered. It also demonstrates the importing of a CAD surface and a volume mesh previously created in other packages.

7) **Thermophysical Models and Properties:** Demonstrates how to set up the physical properties of differing materials that can occur in a computational domain and how these materials are linked to cell types and other settings that can be used to define a model. It also demonstrates the setting of the various modeling options available.