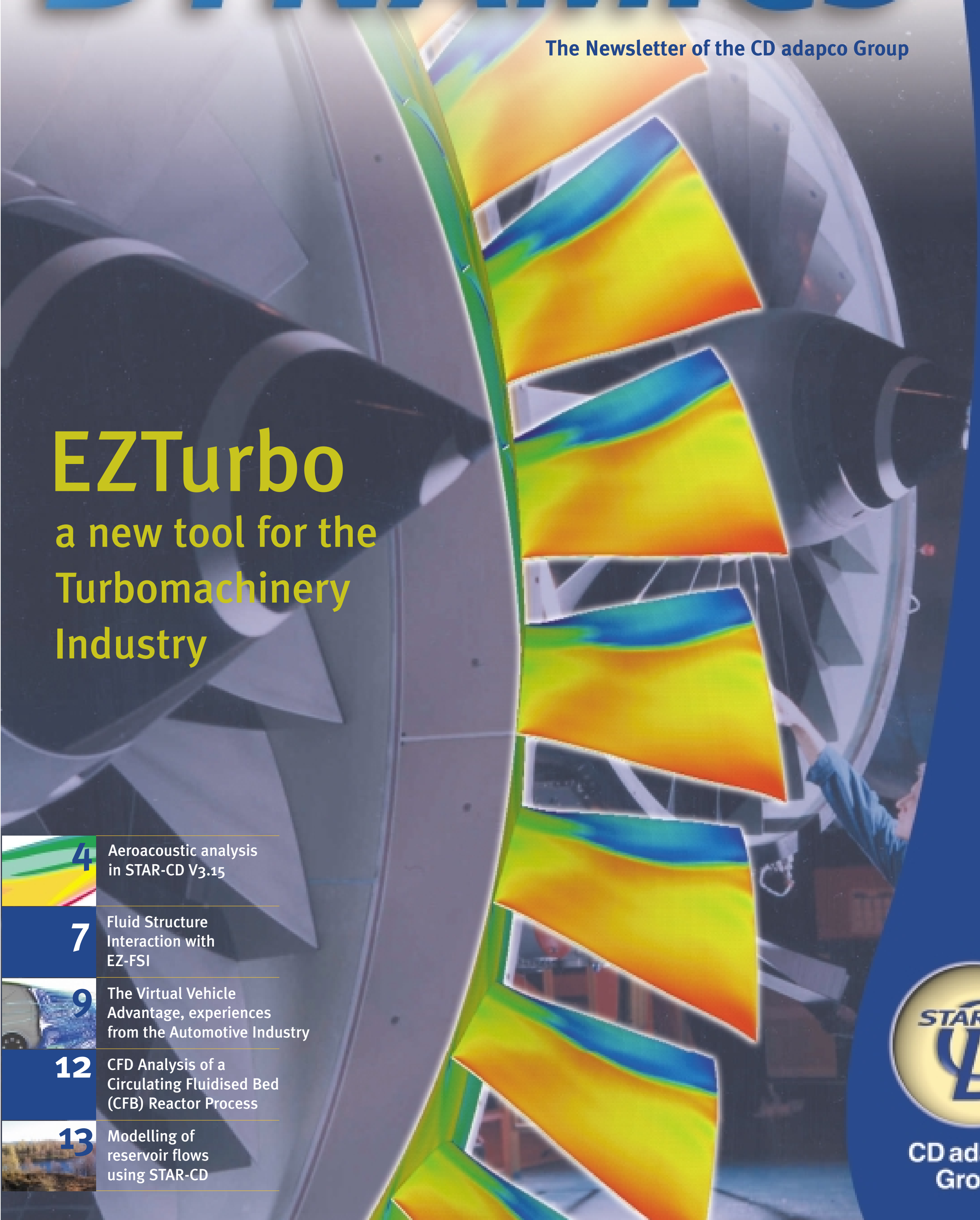
ISSUE 16 WINTER 2001/2

STAR-CD DYNAMICS

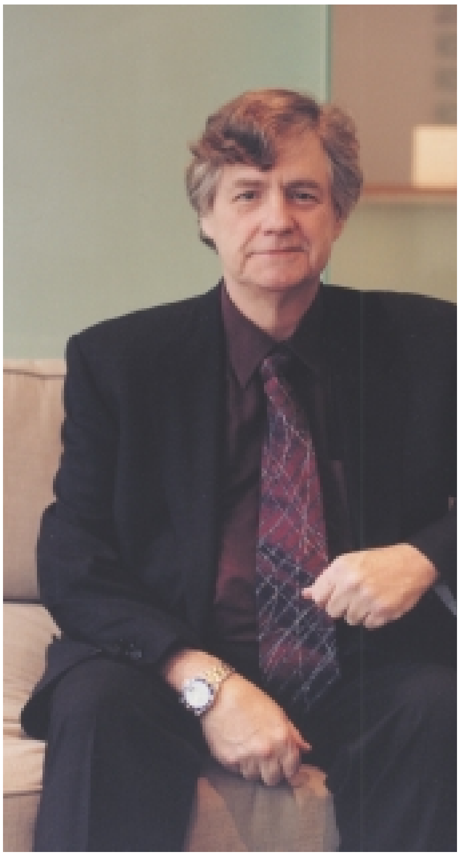
The Newsletter of the CD adapco Group



EZTurbo
a new tool for the Turbomachinery Industry

CD adapco Group

Virtual Engineering - CFD and CAE



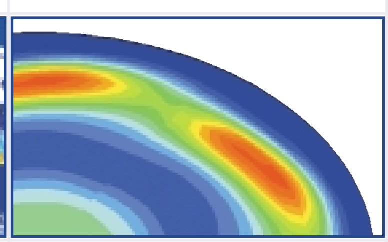
Virtual Engineering is rapidly being seen as the key to productivity and profitability in industry. More and more companies use CAE tools such as STAR-CD together with other CAE systems in preference to physical testing. The benefits are: reduced time to market, reduced cost and improved quality. The emphasis is not just on doing it, but doing it "quicker, cheaper and better"!

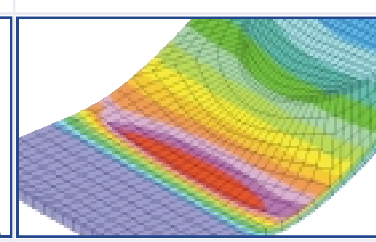
Virtual engineering is not a route that allows half measures. To avoid building multiple prototypes, all aspects of product development must be captured and closely integrated. For example, vehicle design engineers may need to simulate a crash, and at the same time optimise underhood thermal management. Only by working in a virtual CAE environment which integrates all aspects of the design process can prototyping be reduced. Moreover, this virtual approach must be fully integrated within a company's existing design verification process.

Integration for virtual engineering goes beyond making the design tools work together. Success also requires bringing diverse and specialised engineering expertise into the design process. Take power-industry heat exchangers - heat transfer and flow is simulated by CFD experts; however other engineers need to apply their very different skills to evaluate the structural reliability of the proposed design.

Where does the CD adapco Group fit into this picture? Whilst retaining strict confidentiality of client data, we are continually learning how to help clients solve generic engineering problems. We find ourselves enabled like "bees" to "cross-pollinate" expertise and thus achieve effective transfer of technology. To do this effectively, we aim to offer nothing less than a "full service".

contents...





Not only must we continue to develop highly functional, easy-to-use software, but we must also provide appropriate methodology and engineering services. In a nutshell - we want a team relationship with you, our clients, providing whatever you need, from software to engineering applications, filling the gaps in your expertise and training, providing manpower when needed, "empowering, then supporting".

The philosophy of cross-pollination is not new to us. The development of our flagship product, STAR-CD, has always been driven by the engineering applications of our clients, encountered through the engineering services of our offices in New York, Yokohama and London. For example, STAR-CD's moving mesh and spray simulation tools as applied to GDI engines, are now successfully being applied to improving diesel engines.

As the next stage in the establishment of the virtual engineering process, we are developing a set of "EZTools" (pronounced easy-tools). These are problem-specific tools and methodology with associated software, developed via our own engineering knowledge. "Methodology" in this context means everything that is needed to achieve an optimised engineering solution, including: what to consider and what to ignore, how to approach it, and how to judge when enough is done that a design can be accepted.

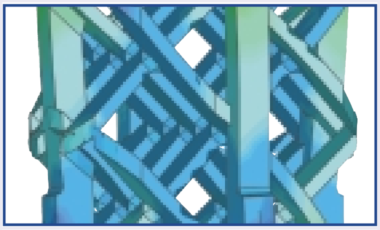
The EZ concept is also being extended to customisation. The CD adapco Group undertakes major simulation projects for clients, for example in external aero and underhood thermal management. This work enables the standard EZAero and EZUhood tools to be customised and allow the same clients to routinely solve similar problems in-house.

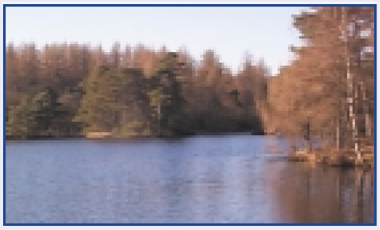
Amongst the new tools, EZ-FSI is one of the first really practicable systems for simulating fluid-structure interactions, whilst EZICE enables moving meshes to be set up quickly for in-cylinder calculations. Meanwhile, EZTurbo offers solution methodology for aerodynamics, heat transfer and stresses for a cooled turbine blade, and is configurable for different turbines. For aircraft engines, it aims to achieve weight reduction with the highest safety, whilst for industrial gas turbines it is used to optimise reliability and is driven by the economics of fixed-price maintenance contracts.

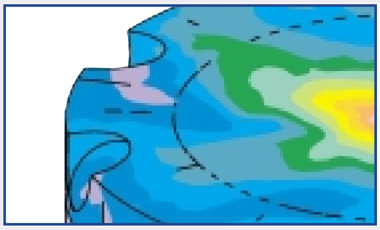
Other EZTools are in the pipeline and will be announced over the next few months and years.

So, that's our vision - in the next decade CFD, with CAE tools in general and engineering expertise in particular, will increasingly be tied together in the exciting new world of virtual engineering. This affects your future as much as ours. We are planning to position ourselves one step ahead of this new way of working. When you are ready, we will be there to help you!

P S MacDonald, Director and General Manager
CD adapco Group







CD adapco Group Corporate News

Partnership with ES.TEC.O

The CD adapco Group has signed an agreement with ES.TEC.O in Trieste to distribute FRONTIER, a design optimisation tool, interfaced with STAR-CD, dedicated to helping our users achieve reduced time-to-market for their products. FRONTIER uses algorithmic and other methods to find sets of parameters that optimise targeted business benefits. For STAR-CD users, FRONTIER offers the possibility of reaching an optimised CAE solution quickly where multiple design variables make intuitive or trial and error optimisation impracticable. See article on page 5 for further details.

Winner of HP prize draw at TCT2001

HP recently sponsored a prize draw on the CD adapco Group stand at the TCT (Time Compression Technologies) 2001 exhibition in Manchester, UK. The lucky winner of an HP Jornada was Mr. Steve Ruff, Head of Design at Martin-Baker, world leader in the field of escape systems technology. Ejection seats are the core of the business and are in service with ninety Air Forces world-wide in some of the latest operational aircraft. During the company's 53 years of continuous experience in the design and manufacture of these seats, nearly 7000 lives have been saved.

" Thank-you to HP and the CD adapco Group" said Steve, " I am sure the Jornada will help with the organisation of my daily activities ".

STAR-CD, A code for all seasons

Loretta Yeager, Trade Shows Coordinator, adapco, Detroit

Autumn, Winter, Spring, Summer - whatever the season, you can be assured that STAR-CD is being presented at a trade show somewhere! As the capabilities of STAR-CD are expanded, so is the range of industries that we serve. In the first year of the twenty-first century, we have demonstrated examples of our work at more than 25 trade shows in the U.S.

The CD adapco Group participates at trade shows for a variety of reasons. It is an opportunity to demonstrate what is new with STAR-CD to a concentrated audience with the ultimate goal of reaching new clientele. Most of the shows are in association with conferences, where technical paper presentations provide an opportunity to see what is new and to learn of new problems faced in each industry.

In the fall, you will find us at Texas A&M's Turbomachinery Symposium or at Medical Design & Manufacturing Expo. In the winter, our presence is found at Power-Gen or AIAA Aerospace. In the spring, SAE Congress is our destination as well as Electric Power. In the summer, ASME's International Gas Turbine Institute Exposition is our focus as well as the newly formed trade show called CoalGen. We look forward to seeing you at one of our trade shows in 2002.

Winter, spring, summer or fall, all you have to do is call, and we'll be there.... I think I feel a song coming on.

New URL for the CD adapco Group web site

In order to reinforce its group identity, the CD adapco Group has combined its UK, US and European (Germany, France and Italy) web sites. The previous addresses www.cd.co.uk, www.adapco.com, www.cd-germany.de, www.cd-adapco.com and www.cd-italy.com are now all directed to the group address www.cd-adapco.com. The site address for www.adapco-online remains unchanged.

Fred Mendonca, Agency Support and Consultancy Manager, UK

Aeroacoustic analysis in STAR-CD v3.15

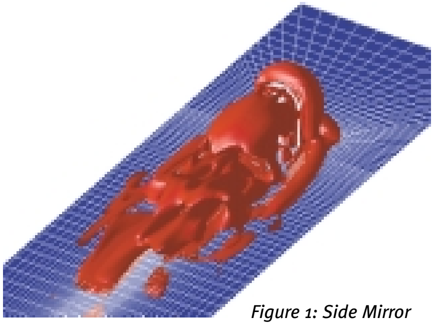


Figure 1: Side Mirror

Significant general interest is currently being shown in the use of CFD methods to capture and thereby understand aeroacoustic phenomena, ultimately towards noise suppression. This is important for many reasons - comfort, safety, avoiding component failure and meeting noise regulations - and is relevant to most industry sectors of which automotive, rail, aerospace and marine are typical examples.

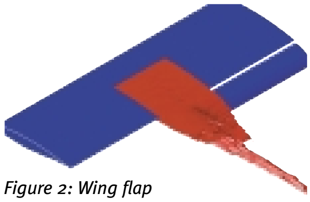


Figure 2: Wing flap

STAR-CD offers two methods for aeroacoustic noise predictions. The first is aimed at the capture of intrinsically broad-band turbulence-based shear noise, quadrupoles, for which a synthesization approach is used. Secondly, time accurate calculations are intended to directly capture the acoustic pressure fluctuations which arise from coherent oscillating flow structures, on resonance.

The synthesization approach uses information of the turbulence field from a steady-state simulation to approximate the turbulence shear-noise distribution in the flow field. Figures 1 and 2 show the aeroacoustic source around a surface mounted side mirror and a wing flap. The noise synthesization method is a new feature released in STAR-CD V3.15.

The second method uses long standing capabilities of STAR-CD which have been extensively exercised mostly for the prediction of coherent flow noise, sometimes referred to as dipole noise, resonance, buffeting or wind-throb. This is mainly caused by the interaction of the fluid (convection and pressure wave combined effects) with surfaces to produce oscillations in the flow. We have demonstrated highly usable simulations using standard features in STAR-CD, namely a standard RANS turbulence models and transient solver. Figures 3, 4 and 5 illustrate the acoustic behaviour of a deep cavity in an aircraft structure with a free-stream Mach Number of 0.85. A cavity vorticity animation can also be downloaded from our web site at www.cd-adapco.com

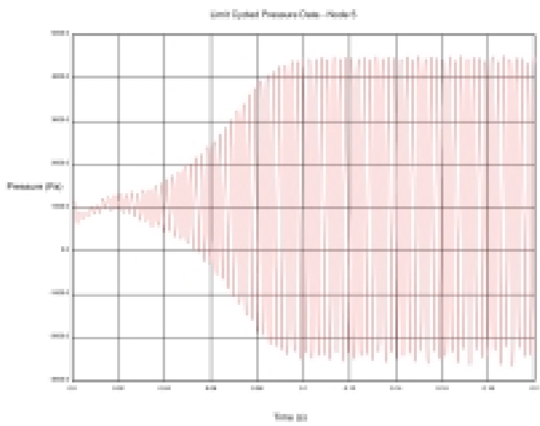


Figure 3: Pressure vs Time

In another example, the benefit of a wind deflector can be seen in the sound pressure level (SPL) plot against frequency. Figure 6 shows that the STAR-CD predictions demonstrate a suppression of acoustic tonal modes and a reduction of the overall decibel level for a sunroof buffeting application.

An important consideration is that run-times for three-dimensional calculations are realisable. Analyses comprising half-million cells takes typically less that 100 hours on eight 400MHz parallel nodes to produce many oscillations which are fully limit-cycled.

In conclusion, whether your noise problem is broad-band based or resonant, we believe that STAR-CD can help you find a solution.

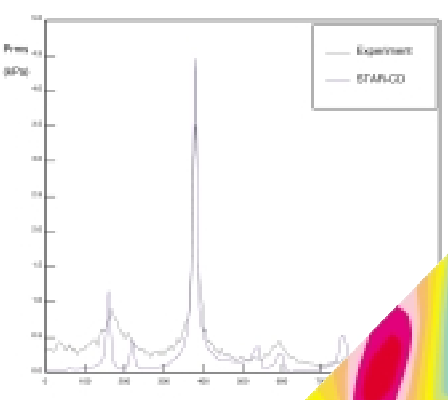


Figure 4: PRMS vs frequency

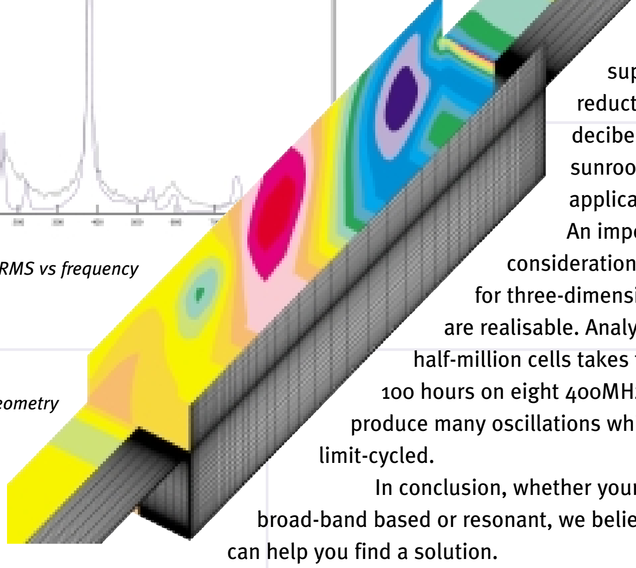


Figure 5: 3-D Geometry

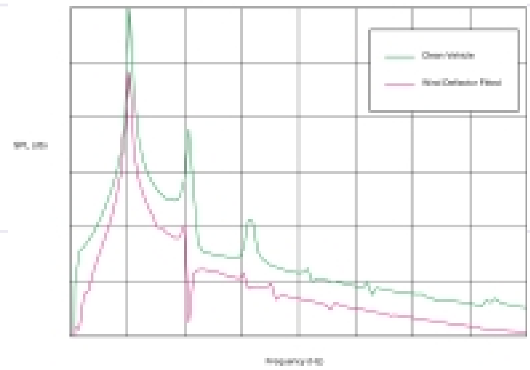


Figure 6: With and without deflector SPL vs frequency

Carlo Poloni, ES.TEC.O. Italy

FRONTIER: a state of the art optimisation tool

for multidisciplinary design

FRONTIER is a design optimisation tool, interfaced with STAR-CD and dedicated to helping our users achieve reduced-time-to market, savings in cost and enhanced quality for their products.

It uses genetic algorithm and other methods to relate "causes and effects" - where "causes" are the set of parameters that define a product or process, and "effects" are the business benefits that result from an optimised design. For STAR-CD users, FRONTIER offers the possibility of reaching an optimised CAE solution quickly where multiple design variables make trial and error optimisation impracticable.

The FRONTIER system enables the designer to search for optimal designs over a number of criteria (e.g. cost, efficiency,...) etc within a given design space. The designs generated during optimisation form the trade-off surface (Pareto Frontier) which is then passed into a selection tool which is used to choose a single design that best suits the designer's aspirations.

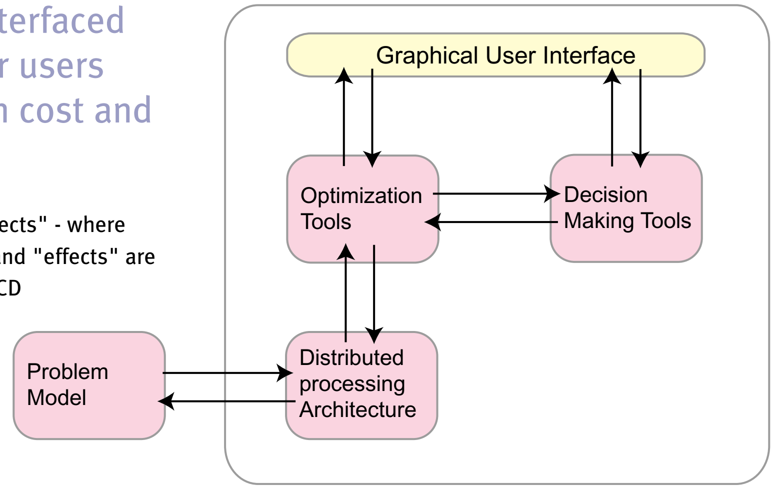


Figure 1

The system is made up of a number of distinct parts, that are integrated to form the complete FRONTIER system (Figure 1):

1. GUI - Graphical User Interface, includes visualisation and set-up tools.
2. Optimization Tools - Several methods used to perform single/multiple objective optimisation.
3. Distributed Processing Architecture - Manages processes on several machines.
4. Decision Making Tools - Aids designer in selection of 'best' design.

Electrolux Zanussi used FRONTIER coupled with STAR-CD to achieve optimised product performance targets by finding the best compromise between a conflicting set of design parameters associated with the insulation of a refrigeration system (Figure 2).

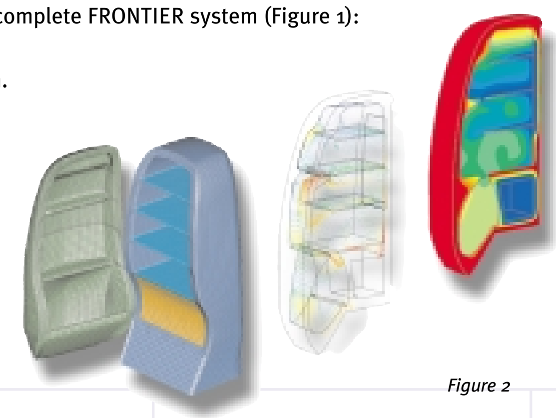


Figure 2

In another example (Figures 3a-d), the application is related to the improvement of the performance of a wing. The geometry is defined by a CATIA-V5 parametric CAD drawing. The geometry is passed automatically to PROSTAR for meshing, then to STAR-CD for the flow solution and to PROSTAR again for post processing.

FRONTIER is developed by ES.TEC.O in Trieste and distributed by the CD adapco Group. A brochure is available on request from info@cd.co.uk

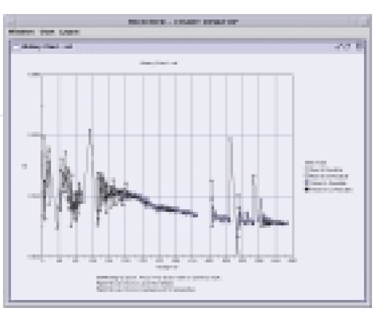


Figure 3a) shows the convergence history of the optimisation run

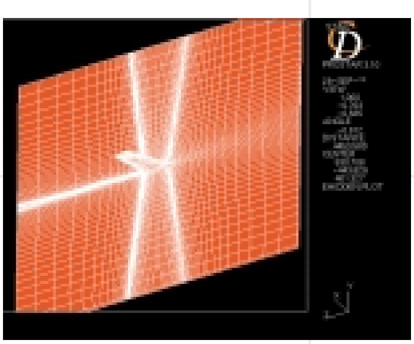


Figure 3b) shows the grid employed

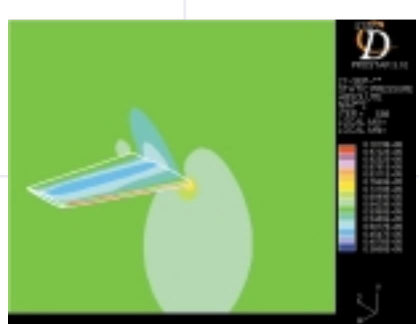


Figure 3c) shows the pressure distribution on the best geometry

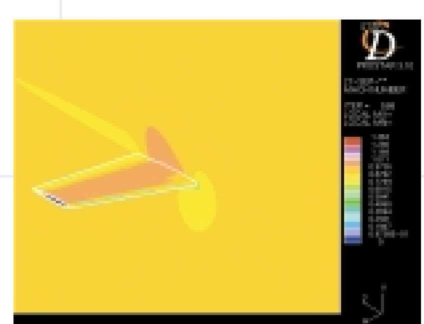


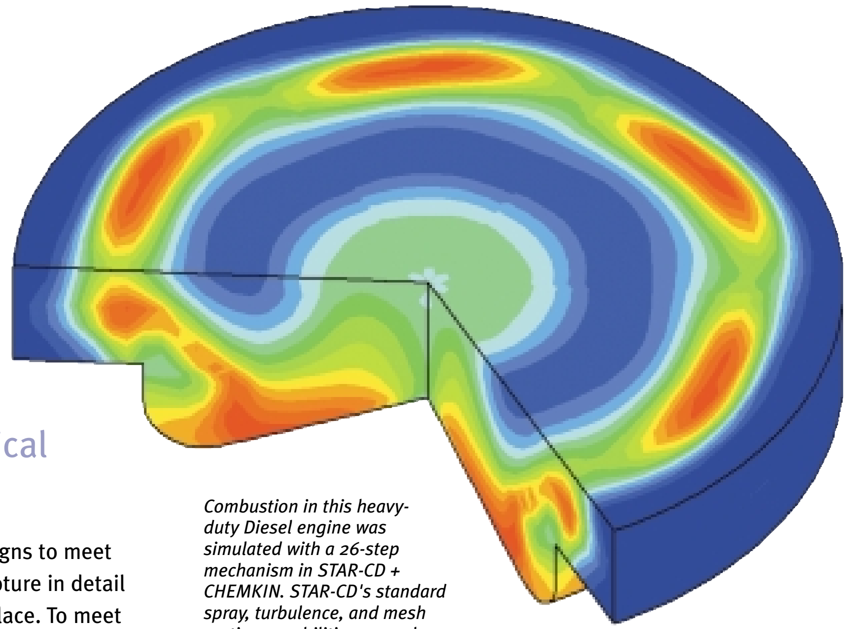
Figure 3d) shows the Mach number distribution on the best geometry

John Deur, adapco, New York

STAR-CD and detailed chemistry

including CHEMKIN

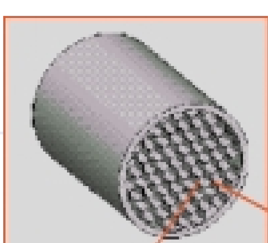
Today more than ever, chemists and engineers face the challenge of increasing the efficiency of chemical processes and combustion devices while reducing emissions of pollutants from them. For example, witness the increasingly stringent environmental regulations being applied to internal combustion engines, industrial gas turbines, chemical processing plants, etc.



Combustion in this heavy-duty Diesel engine was simulated with a 26-step mechanism in STAR-CD + CHEMKIN. STAR-CD's standard spray, turbulence, and mesh motion capabilities were also utilized. Peak chamber pressure was predicted within 5%, and NO_x emissions at exhaust valve opening were predicted within 3%.

Simulation is one tool that can be very effective in shaping designs to meet these new challenges. However, such a tool must be able to capture in detail both the chemical kinetics and the complex fluid flows taking place. To meet the challenge of merging complex chemistry with complex fluid flows, the CD adapco Group has incorporated several new chemistry features in its industry-leading STAR-CD software.

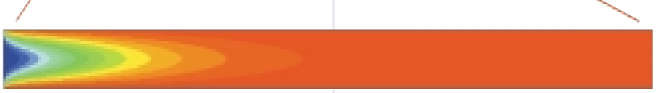
Perhaps the most significant development has been the partnership developed between the CD adapco Group and Reaction Design, distributors of the widely-used CHEMKIN Collection software. The CHEMKIN Collection provides a computationally efficient means of determining species concentrations accurately for complex coupled chemistry in a variety of idealized chemical reactor situations. By extending their highly developed chemistry solvers to STAR-CD, these same complex surface and gas phase chemistries can now be solved for transient and steady-state problems. These involve complex geometries, as well as a variety of physical phenomena including sprays, radiation, turbulence, buoyancy, etc. The resulting joint product, STAR-CD+CHEMKIN, combines the CFD and geometry-handling strengths of STAR-CD with CHEMKIN's leading-edge technology for treating complex coupled chemical kinetics and molecular transport. A beta version of this software based on the current V3.150 of STAR-CD will soon be available; the full version will be with the next general release of STAR-CD. Several other features are also being added to STAR-CD for V3.200 that will complement the STAR-CD+CHEMKIN package:



- The CD adapco Group has revamped the transport property module to include an expanded database of temperature-dependent transport properties, including viscosities, thermal conductivities, and diffusivities based on the NASA Glenn polynomial format. In addition, a general purpose interface that allows users to link with their own transport property packages. Of course, with the STAR-CD+CHEMKIN package, users will also have access to the CHEMKIN Collection's latest thermal and transport databases as well.

- To provide a general chemistry capability, the new general-purpose interface will allow users to couple their own chemistry packages to STAR-CD and simulate a wide-range of reaction systems. A reaction decoupler will also be available to allow users to separately solve reactions that have a negligible impact on the flow field or on other reactions from the coupled chemistry-flow solution, thus saving overall computation time.

- In addition to the STAR-CD+CHEMKIN package, the CD adapco Group has also developed its own suite of gas phase coupled chemistry solvers for transient and steady-state problems. This technology will be reported in the next issue of the Newsletter.



A platinum catalyst is used to assist the oxidation of methane. The surface chemistry capability of STAR-CD + CHEMKIN was used to model this process within one channel of the monolith utilizing a 23-step mechanism developed by Deutschmann of the University of Heidelberg. Of course, several techniques are available in STAR-CD to efficiently extend the analysis to the entire monolith. Such an analysis would naturally include aspects such as heat transfer between the flow field and the monolith structure.

Contact John Deur (jdeur@adapco.com) for more details.

Robert A. Brewster, Technical Manager, adapco, New York

Fluid-Structure Interaction

with EZ-FSI

Fluid-structure interaction (FSI) problems arise in a wide variety of industries. Examples include flow-induced vibration of tubes in cross-flow, liquid sloshing in fuel tanks, pumps and compressors with pressure-actuated valves, and the vibration of fan blades. In this article, we describe the EZ-FSI tool, which allows STAR-CD users to efficiently solve FSI problems involving small linear-elastic deformations.

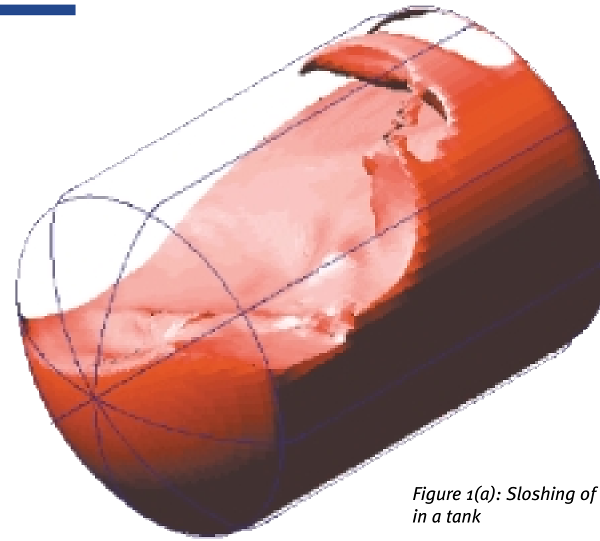
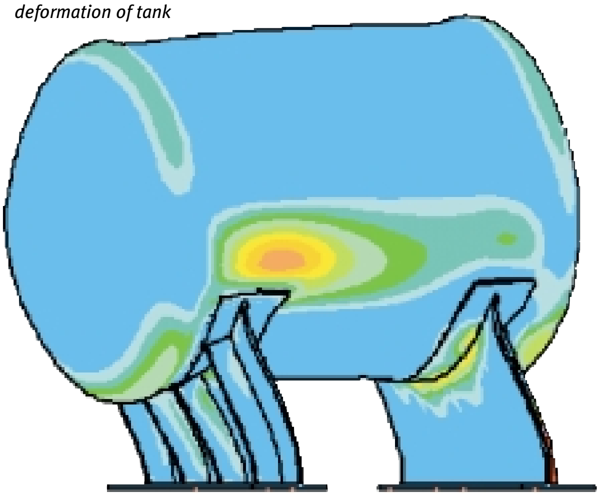


Figure 1(a): Sloshing of liquid in a tank

Figure 1(b):Maximum stress and deformation of tank



EZ-FSI is a new methodology for solving fluid-structure interaction (FSI) problems using STAR-CD. Commercial FEM software is used to generate the mass, damping and stiffness matrices required to solve the transient structural dynamics problem. These matrices are then used by EZ-FSI to solve for the deformations of the structure during the STAR-CD analysis. Finally, an optional post-processing step uses the FEM software to solve for the stresses within the structure.

Unlike the MpCCI approach described elsewhere in this newsletter, the EZ-FSI approach is limited to linear-elastic small-deformation problems. However, this constraint allows EZ-FSI to use a very efficient solution technique, resulting in only a small computational overhead compared with traditional (non-FSI) STAR-CD analyses.

An example of the use of EZ-FSI is the liquid sloshing and structural dynamics of a large fuel tank during an earthquake. STAR-CD's volume of fluid (VOF) capability was used to analyze the free-surface fluid sloshing problem. Some representative results of this analysis are shown in Figure 1. Figure 1(a) shows the shape of the oil free surface in the tank at a particular point in time, while Figure 1(b) shows the maximum principal stress and deformation of the tank (scaled by 7500x).

A second example of the use of EZ-FSI is a fan blade vibration calculation. The geometry of the turbofan is shown in Figure 2(a). Flow around one of the blades was modeled together with the initial bending of the blade due to centrifugal and aerodynamic forces.

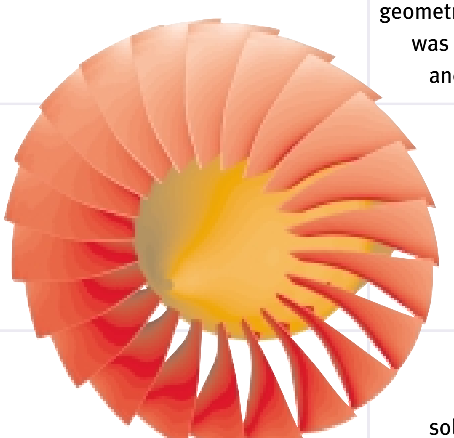


Figure 2(a): Turbofan blade vibration analysis: plot showing geometry of turbofan

The graph of Figure 2(b) shows the bending of the blade tip at the leading and trailing edges as a function of time. Initially the blade overshoots its final position, then oscillates about this point. The oscillations are then damped out by the action of the fluid, and the blade ultimately reaches its steady-state position. Note that the displacement at the leading edge is larger than that of the trailing edge, resulting in 'untwisting' of the blade.

In summary, EZ-FSI provides a way for STAR-CD users to solve FSI problems quickly, using traditional FEM software together with STAR-CD. EZ-FSI is a fast and robust way to analyze your FSI application.

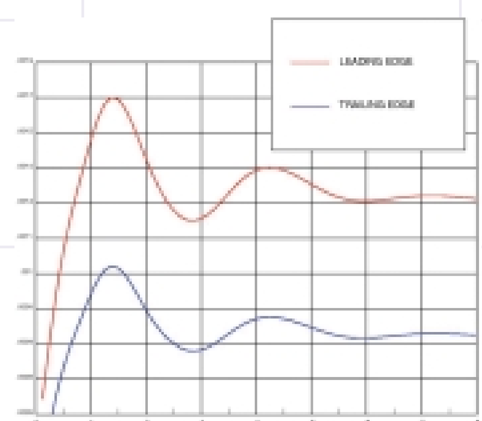


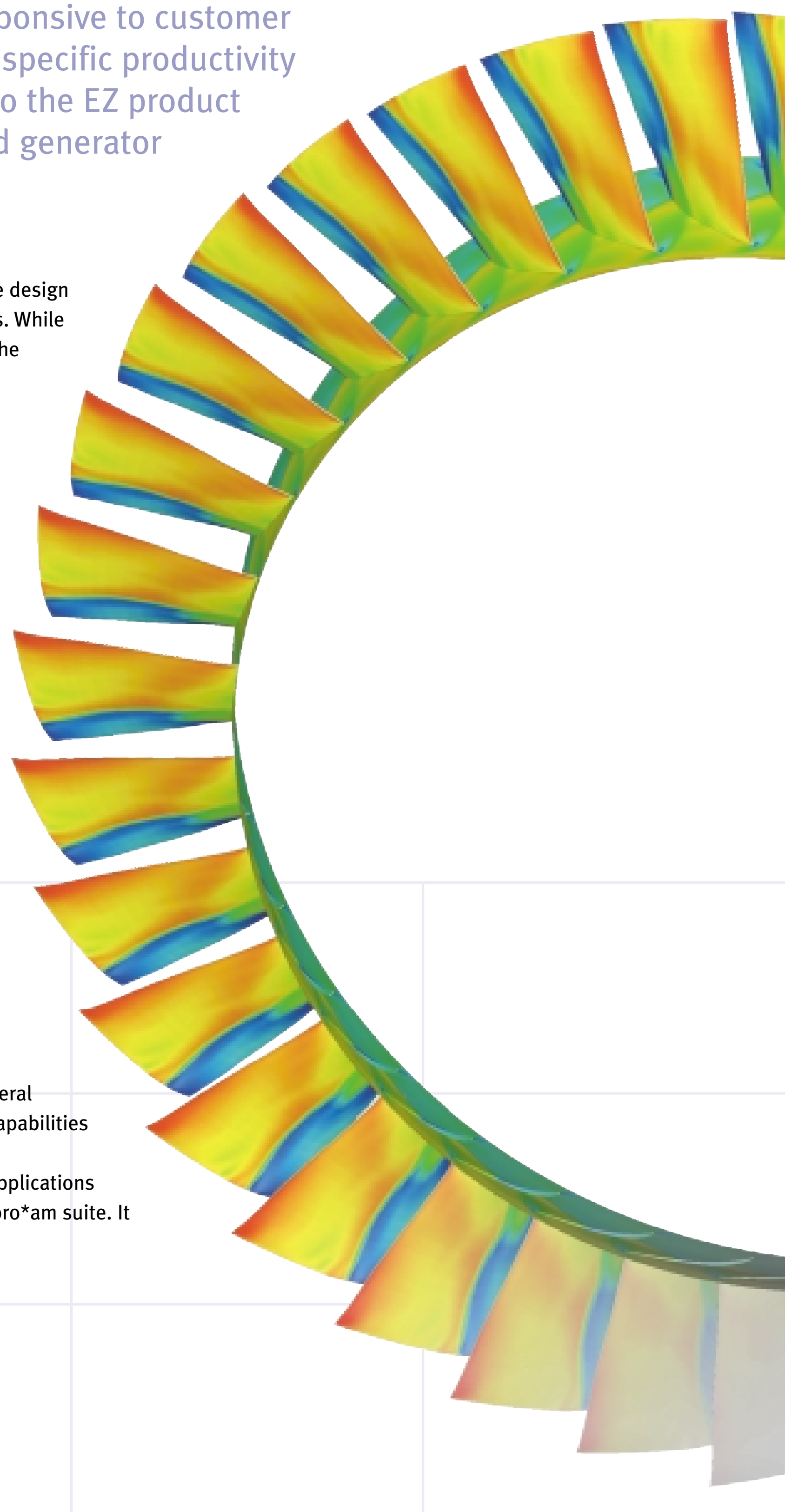
Figure 2(b): Graph of blade tip displacement at leading and trailing edge

Bill Clark, Technical Manager, adapco, Detroit

EZTurbo - a new tool

from the CD adapco Group

The CD adapco group continues to be responsive to customer requests for the development of industry-specific productivity tools. EZTurbo, the most recent addition to the EZ product line, provides a streamlined interface, grid generator and post-processing facilities for the turbomachinery industry.



With EZTurbo, it is possible to interact directly with the client's blade design system and construct a high-fidelity computational model in minutes. While initially developed to satisfy the pre- and post-processing needs of the turbomachinery industry, EZTurbo has been shown to be useful for a much broader range of applications. To date, EZTurbo applications include centrifugal and axial compressors and turbines, turbochargers, torque converters, radiator fans, propellers and blood pumps, to name but a few.

EZTurbo employs a multi-blocked meshing strategy to quickly generate high quality grids around aerodynamic bodies in both axial and centrifugal configurations. Using one of the predefined topologies avoids the confusion associated with many multi-block grid generators and significantly reduces the user input and hence time required to build a computational model. The generation of the computational mesh is achieved through an iterative solution of a set of elliptic equations. These equations are constrained to match the geometric shape being meshed and are subjected to boundary conditions that guarantee smooth variations in grid clustering and orthogonality.

For more complex geometric features, such as part span shrouds, cooling holes and casing treatments, EZTurbo easily interfaces with the automatic meshing capabilities of pro*am. Geometries that would take weeks with conventional multi-block codes can now be meshed in hours.

In addition to its capabilities as a grid generator, EZTurbo also allows the user to impose boundary conditions, specify thermophysical properties and define the numerical algorithm parameters required by STAR-CD. Presently, we are working with several turbomachinery companies to provide specialized post-processing capabilities to further improve engineer productivity and ease-of-use.

If you are looking to increase your productivity and have applications that sound similar to those described above, consider the EZTurbo/pro*am suite. It will be the last and only grid generation tool you need.

Contact Bill Clark (billc@adapco.com) for more details.

Walter Bauer, DaimlerChrysler AG, Stuttgart, Germany

The Virtual Vehicle Advantage

Experiences from the automotive industry

AUTOMOTIVE

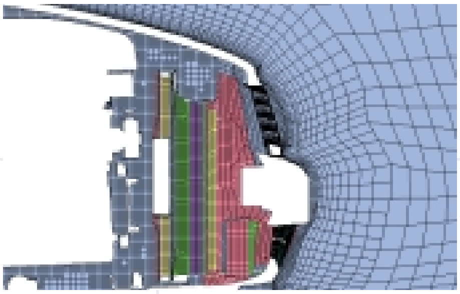
DaimlerChrysler was one of the first companies to integrate CFD (Computational Fluid Dynamics) into their CAE driven development process. They are now reaping the benefits of shorter turn around design cycles, achieved by exploiting to the full the "virtual vehicle" concept.

Since the first real industrial CFD applications in the automotive industry came onto the scene about twelve years ago, the vehicle development process has changed dramatically. The change started with the introduction of CAD to this process about three years before that. CAD was the basis for shorter turn around cycles and with the evolution of CAE tools including powerful mesh generation systems, brought CFD fully into the design process.These developments together with the continually increasing performance of computing hardware and software, made possible even more complex applications. The idea of virtual vehicle development was born.

Today, CFD is completely embedded in the vehicle design process at the Mercedes-Benz passenger car development department. Different facets of the technology are put to optimal use in a diverse range of vehicle development processes including underhood flow analysis, vehicle cooling, passenger compartment flow, air conditioning, brake cooling design and aerodynamics.

It is vital that all CAE activities are performed to an optimum level. As with all vehicle development processes, the Mercedes-Benz Development System (MDS) is very complex, involving multiple iterative loops every time CAE tools are used. As CAD and CAE have come to play an increasingly dominant role in the concept and design phases of the vehicle development process, any inefficiencies in CAE loops are magnified. For this reason, optimisation is built in at every stage and Mercedes-Benz believes that MDS represents the best possible use of CAE tools - and CFD in particular.

Under the Bonnet



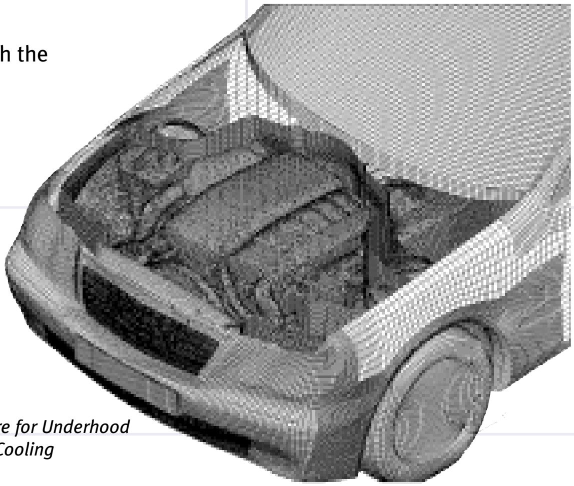
Mesh structure for underhood flow/vehicle cooling

Mercedes-Benz has been using CFD for the simulation of underhood flow since 1990. In that time, more than fifteen different model series (including the A-, C-, E- and S-Class) have benefited from three-dimensional CFD simulations. Today for all vehicles under development, three-dimensional CFD simulation and one-dimensional coolant flow simulation are exclusively used to verify engine cooling design.

The first part of the CFD process - mesh generation - proceeds automatically by creating up to five million "cells" derived from around 200 CAD files (even after simplification these represent some 600MB of data). The mesh generation is performed using CD adapco Group's EZUhood and pro*am software tools.

CFD analysis - all analysis in MDS is performed using STAR-CD - is then undertaken, typically under three standard load cases: with the vehicle driving at maximum speed and the cooling fan off; with the vehicle towing a trailer uphill with the cooling fan on; and finally with the vehicle stationary, the engine idle, the air-conditioning on full and the cooling fan on.

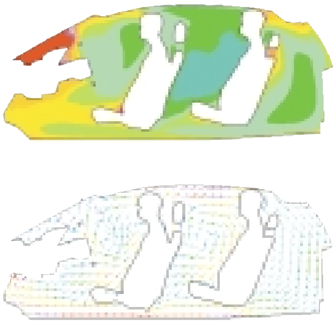
The CFD analysis yields a complete three-dimensional picture of the rates and distribution of airflow through or around all the underhood heat exchangers and components. Moreover, a general picture of airflow and complete temperature contours can be derived. This information - plus a further zero or one-dimensional simulation to give the coolant temperature - is sufficient to provide verification or otherwise for the cooling model.



Mesh Structure for Underhood Flow/Vehicle Cooling

In-car comfort

Around 1998, an approach called TEKOS was implemented in the MDS to predict the thermal comfort of passengers. TEKOS consists of a CFD program and two other simulation tools, one for the heat transfer inside the passenger compartment and the heat exchange between the exterior and interior, and the other for the thermal properties of the passengers. The CFD analysis provides the heat transfer coefficient distribution, temperature distribution and flow structure inside the compartment and therefore gives a true picture of passenger comfort.



TEKOS Results, Temperature Distribition and Flow Structure

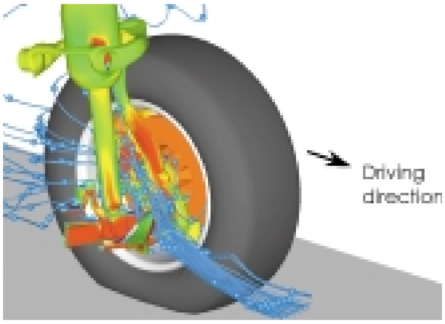
At the same time as the introduction of TEKOS, the use of CFD for the flow analysis inside the ventilation ducts, the heater/air-conditioning box and water separation box was started. CFD is now routinely used to verify interior flow models in all new cars. For the CFD analysis the following standard load cases are applied: winter (steady-state and transient - heating up), summer (steady state and transient - cooling down), windshield deicing and demisting.

For winter and summer (steady-state only) and a given position of ventilation louvres, the inlet mass flow rate, the blow out directions and air temperatures are varied until the optimal thermal comfort for all passengers is achieved. For the transient cases only the inlet air temperature varies in time, the other boundary conditions remaining the same. The transient air temperature is calculated with the help of one-dimensional simulation tools (refrigerant and coolant loop). For the airside of the condenser, the calculated air mass flow rate from the underhood flow analysis is applied.

Brake cooling

As far back as 1992 the first steps were taken to simulate brake cooling. This type of simulation requires first a simulation of the flow inside the wheel arch and secondly a heat transfer analysis for the brake components.

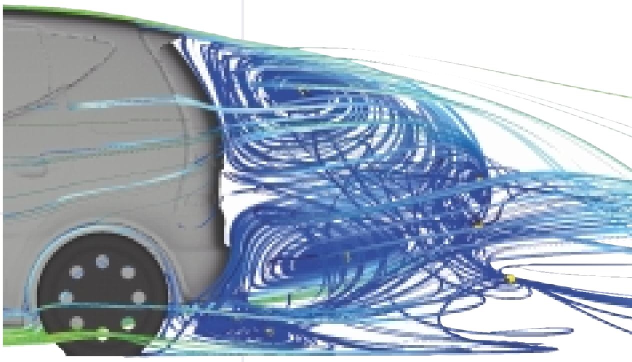
Here the CFD simulation assumes a steady-state speed of 140km/h and for the heat transfer analysis two scenarios are tested: alpine downhill braking and a braking power/distance test based on cyclic braking and acceleration manoeuvres. In addition to the heat transfer coefficient distribution, the CFD analysis indicates how the air in the wheel arch should be guided to cool temperature-sensitive components.



CFD-Results of Brake Cooling Analysis

Vehicle Aerodynamics

The external flow analysis of a vehicle by means of CFD is not yet fully implemented in the MDS but the first steps towards full implementation have been made. For vehicles with a smooth underbody and closed underhood, excellent results for drag and lift can be achieved. For more detailed underbody geometry, additional effort is necessary to improve the results. For better comparison of measured and computed CFD results three wind tunnels were analysed by CFD.



Wake Flow Structure

Conclusion

In order to be approved as a part of MDS, CFD had to prove its worth by showing that the results conform to application-specific expectations and by demonstrating that such results can be delivered within a given timeframe. In our industry even excellent results are more or less worthless if they are provided too late! In most cases (except for external flow and vehicle aerodynamics) CFD is now embedded in a procedure with other simulation tools as part of MDS. The CFD results alone do not fulfil the MDS requirements all full vehicle functions, only the combination with other simulation tools provides the full range of information. Therefore all parts in the simulation chain must deliver accurate results. To be on the safe side, a lot of testing and comparing must be done to find the best combination of tools for a specific optimisation loop.

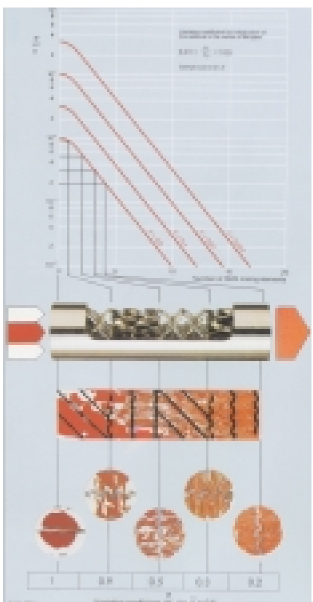
Experiences from the past show that the complexity of the simulation models and the number of optimisation loops increases all the time. Furthermore, more transient CFD simulations will be necessary in the future. To increase today's high level of CFD application and to meet future MDS requirements Mercedes-Benz has identified areas where it would like to see additional functionality from its CFD code. Such information forms valuable - indeed vital - feedback for companies like the CD adapco Group in their ongoing software development.

Torsten Wintergerste, Sulzer Innotec, Winterthur, Switzerland

Fluid-Structure Interaction

with STAR-CD and Permas using MpCCI

Several industrial processes involve mixing of two or more components which have different physical properties. The static mixers of Sulzer Chemtech homogenize the flow within the mixer without the use of any moving part. Differences in concentration, temperature and viscosity are equalized in a continuous process over a flow cross-section.



Coefficient of variation vs. dimensionless mixer length (for laminar flow)

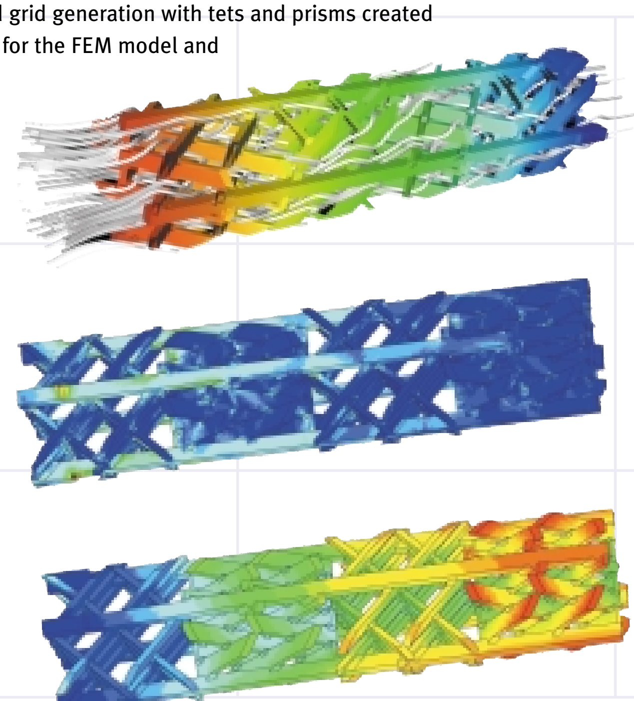
The Sulzer SMX mixers are designed for the mixing of high viscosity liquids under laminar flow conditions. The complex structure of the blades generate layers which drive the mixing process. Special versions of the mixer are designed for applications (e.g. polymer melt blending) where the pressure drop is up to 100 bar or higher. The mechanical forces and stresses in the structure caused by the flow are given by the pressure drop at each blade. The resulting total force is transferred from the mixer structure to the outer pipe e.g. at one end of the mixer. Due to the complexity of the flow materials and different mixer designs, a general mechanical testing under real fluid forces is complicated and expensive. However, mixer optimisation requires consideration of the real fluid forces for the determination of maximum stresses inside the structure.

The coupling of commercial CFD-codes and codes for structural mechanics can be carried out by loose coupling through the MpCCI communication library. This library interpolates the required values such as temperature, force, deformation, etc. from one code to the other, on matching and non-matching grids. The codes are coupled on their outer iterations. A first version of the MpCCI library was used in the CISPAR project. Further development of the library was done in close co-operation between some of the CISPAR partners: GMD, Computational Dynamics, Intes, DaimlerChrysler and Sulzer. Together, we developed the first version of code-coupling between STAR-CD (Computational Dynamics) and PERMAS (Intes) that was used industrially.

The deformations and stresses of the Sulzer Mixer subjected to high-pressure load was investigated by coupling STAR-CD and Permas using MpCCI. The geometry model takes into account all the details of the structure, even welding points. The mixer structure was completely built as a 3D solid model in Unigraphics. The fluid flow domain was generated by Boolean subtraction of the 3D solids of the pipe from the mixer structure. The FEM model was meshed by the specialists at the Mechanical Department using Medina (hexahedral) and/or Patran (tetrahedral). The fluid flow domain was meshed by our colleagues at the Fluid Dynamics Department, using unstructured grid generation with tets and prisms created by GridGen and ICEM/Tetra. The two meshes consist of 35,000 hexahedral elements for the FEM model and 1.7 million elements for the fluid flow model. The FEM model used first-order elements and considered contact conditions between the mixer and the outer pipe. One advantage of the loose coupling approach using MpCCI is that both meshes can be generated independently. The coupling interface for the non-matching grids is very complex and consists of 120,000 triangular elements in the fluid domain and 30'000 quadrilateral elements in the solid domain.

As a first step, the steady-state fluid flow was computed by STAR-CD without any code coupling. The pressure distribution and the streamlines are shown below. As a second step, the fluid forces were transferred from the fluid code to the stress code by coupling the codes.

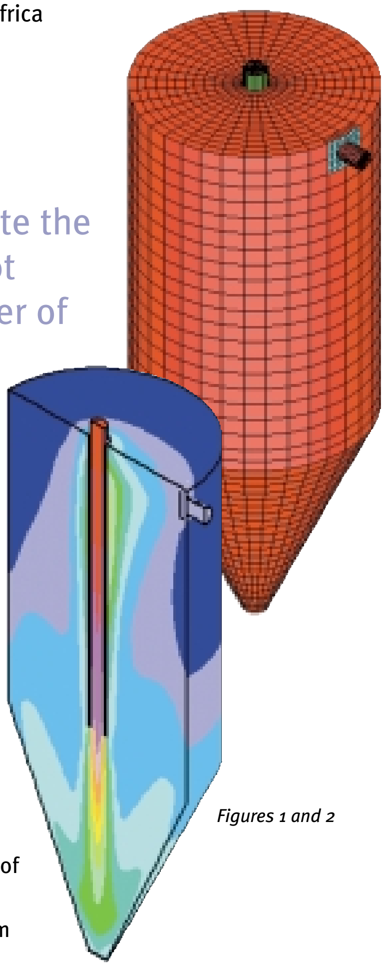
The computed stresses and deformations are shown in the figures. This method (the one-way-coupling) assumes that the fluid flow topology is not affected by the displacement of the mixer, which is realistic for this kind of mixer. The deformations, stresses and rotational movement agree with our experimental observations. We are also working on the full coupling of the flow and stress computations, requiring STAR-CD's moving mesh capabilities. The use of STAR-CD, Permas and MpCCI enables a more realistic computation of the forces on the structure and better design and optimisation of the mixer geometry.



The Sulzer mixer (SMB) used for laminar mixing with streamlines and pressure (top), structural displacement (middle) and stresses (bottom)

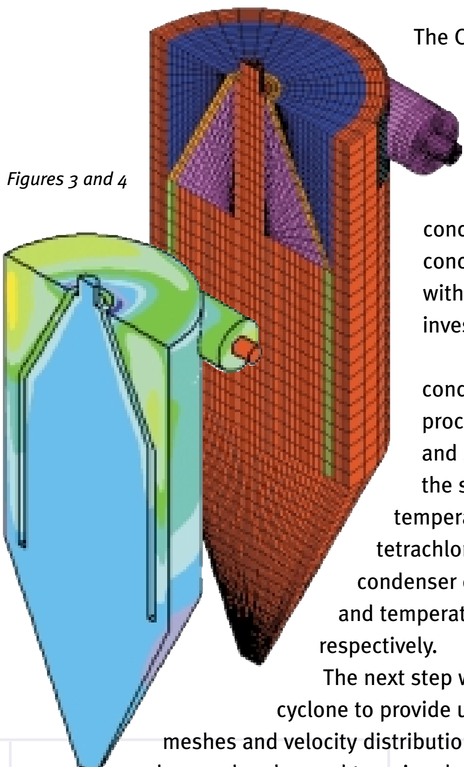
Dr. B.J. Henning, New Business Developments, Bateman Titaco, South Africa

CFD analysis of a CFB Reactor



Figures 1 and 2

STAR-CD has recently been used by Bateman Titaco to calculate the thermo-fluids process in a Circulating Fluidised Bed (CFB) pilot plant on behalf of Mintek, an internationally respected provider of technology to the minerals and metallurgical industries.



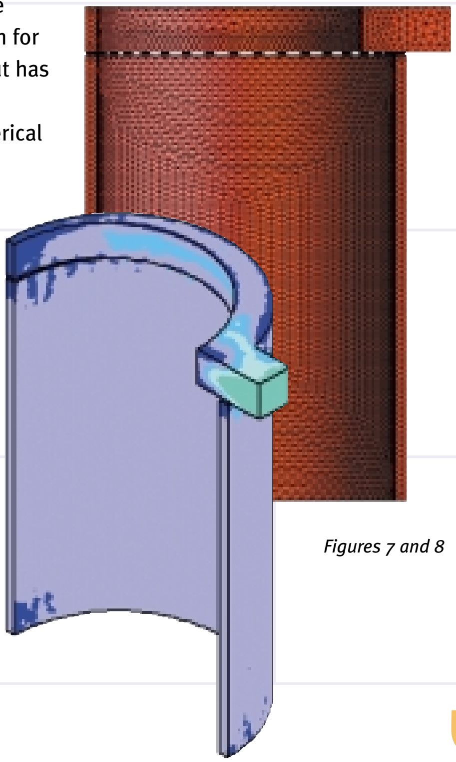
Figures 3 and 4

The CFB pilot plant for chlorinating titanium slag, with coke as a reductant, forms part of a Mintek research project funded by South Africa's Department of Arts, Culture, Science and Techcnology (DACST). This plant will comprise a CFB reactor, gas preheater, feed system, cyclone for dust removal, condensers, prescrubber and afterburner. The CFB concept is an ideal thermo-fluid process to be simulated with STAR-CD, as different designs and variables are investigated.

The modelling consisted of an air-cooled $FeCl_3$ condenser/cyclone and, for simulation purposes, all processes were modelled with nitrogen gas whose density and specific heat values changing with temperature. The aim of the simulation was to determine whether the outlet gas temperature would be between 200 - 250 °C , to prevent titanium tetrachloride gas condensing in the $FeCl_3$ condenser/cyclone. The condenser did not achieve these values and a cyclone was proposed. The numerical meshes and temperature distribution results are shown in figures 1 to 4 for the condenser and cyclone respectively.

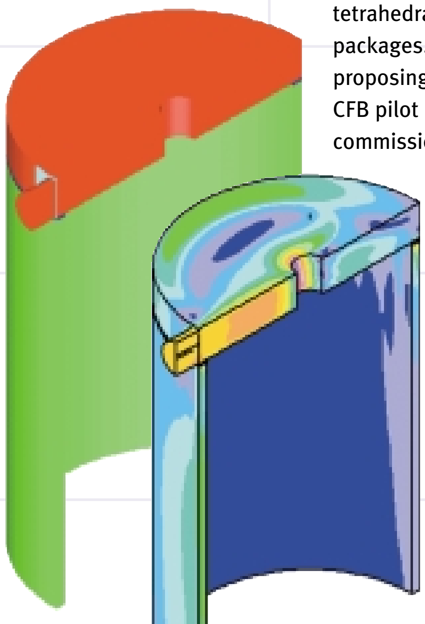
The next step was to determine the best conceptual air-cooled jacket design for the $FeCl_3$ cyclone to provide uniform mass flow distribution around the air-cooling jacket. The numerical meshes and velocity distribution results are shown in figures 5 to 8 for the plenum chamber and tapering duct (preferred), respectively. The best design for each component can be obtained numerically with thermo-fluid modelling, but has to be compared with results from the proposed pilot plant.

To conclude, STAR-CD offers an excellent simulation tool, allowing easy numerical mesh set-up via PROSTAR and testing of process variation. It also enables automatic tetrahedral mesh import from other CAE packages. This assists the designer in proposing the best configuration for the CFB pilot plant before actual commissioning and production of titanium tetrachloride liquid for downstream purposes.



Figures 7 and 8

For further information, please contact Dr BJ Henning, BennieH@tbateman.co.za



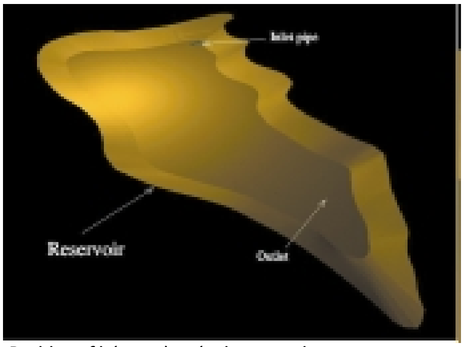
Figures 5 and 6

Isabelle Lavedrine, Arup, London, UK

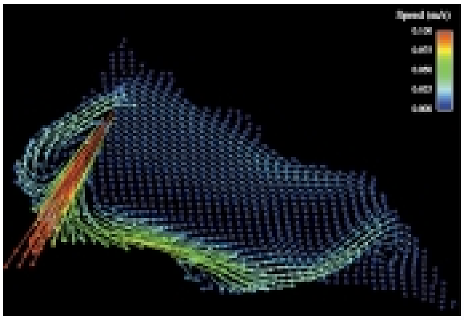
Modelling of reservoir flows
using STAR-CD

Reservoirs constitute an essential part of the water supply capacity of numerous water companies. There are two issues in the management of these resources which water companies have to address:

- The avoidance of "dead" regions of stagnant water is important in preventing the growth of micro-organisms
- If a pollutant reaches the reservoir, risk of pollution of water supply has to be minimised by ensuring that transit time through the reservoir is sufficient for the pollutant to be detected and water from the reservoir stopped from reaching the main supply line.

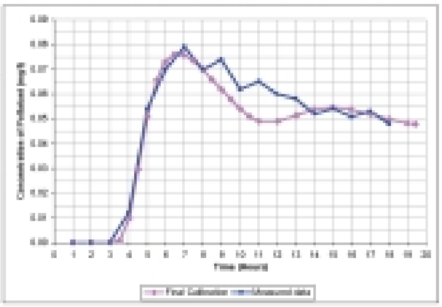


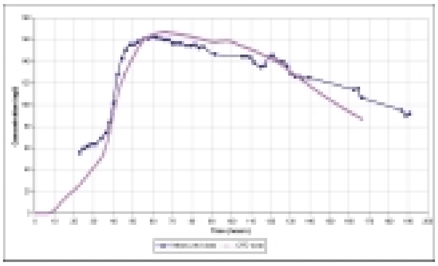
Position of inlet and outlet in reservoir


Velocity vectors in reservoir just under free surface

Large baffles placed within the reservoir are often used as a solution, but the implementation of this method is somewhat haphazard and based mainly on the design engineer's experience and common sense. Moreover, these baffles are not aesthetically pleasing and would hinder the use of the reservoir for recreational purposes. Computational Fluid Dynamics can help the design engineers by providing them with reservoir flow patterns that enable them to place the baffles more strategically, or by suggesting other solutions to resolve the issues. Arup modelled a water supply reservoir where CAD data was available for the reservoir bed topography, as well as a series of experimental measurements including drogue tracking and dye tracing, and concentration of in-going and out-going pollutants during a real-life event. Hence, we were able to first calibrate the CFD model using the experimental measurements, and then validate it against the real-life event.

The STAR-CD model consisted of approximately 275,000 fluid cells. Firstly, flow patterns under steady-state conditions were predicted that matched the results from the drogue tracking experiments. Using these results, a transient simulation was then carried out. For this simulation, the dye tracing data were used, i.e. a low concentration of pollutant was introduced into the inlet water over a period of approximately 35 minutes. The concentration of pollutant within the reservoir, in particular at the outlet, was then predicted over a period totalling 20 hours. Only the pollutant scalar variable was solved for during the transient analysis. No information was available on bed roughness for this reservoir. Because the cell closest to the bed was smaller in size than the estimated roughness height, the bed roughness could not be applied via a user subroutine. Instead, bed roughness was represented by creating a porous medium layer on the bed, and matching the predicted flow patterns and pollutant dispersion rate with the measured ones. Once the porous medium parameters were adjusted, the CFD results matched the measurements quite closely, both for the maximum concentration of pollutant at the outlet (predicted 0.076mg/l, observed 0.079mg/l) and the timing of the peak concentration (predicted 6.5 hours after release of pollutant, measured 7 hours). Finally, to fully validate the calibrated model, a real-life event was modelled where both concentration and flow rate data were available. The results match quite well. The predicted time of peak was 6 hours and 20 minutes later than measured. The predicted peak concentration was 167ng/l, the measured value was 163ng/l. The delay in the time to peak can be explained by the fact that we assumed a constant flow rate through the reservoir, and that this value was smaller than the flow rate actually going through the reservoir in the first part of the pollution event. Overall, the predicted data provide a good fit for the measured data.


Comparison between measured and predicted CFD (final calibration) pollutant concentrations at outlet


Comparison between real-life event and CFD predicted pollutant concentrations at outlet (validation run)

This validated CFD model could now be used to devise means of increasing the pollutant transit time across the reservoir. Various options for increasing transit time in fact became self-evident in the course of completing this project. From the various CFD runs carried out, it seems that increasing the bed roughness in parts of the reservoir would suffice to increase the transit time significantly.

For further information, please contact isabelle.lavedrine@arup.com

Tobias Linke, Jens Scheffermann and Heiko Spekker, University of Hannover, Germany

Designing and maintaining weir structures

Operational safety of hydraulic structures depends on the stability of the foundations, which need to be properly reinforced to avoid collapse. For example, the design of weir structures has in the past required time consuming and expensive physical testing across the complete range of flow conditions to which the structure may be exposed.

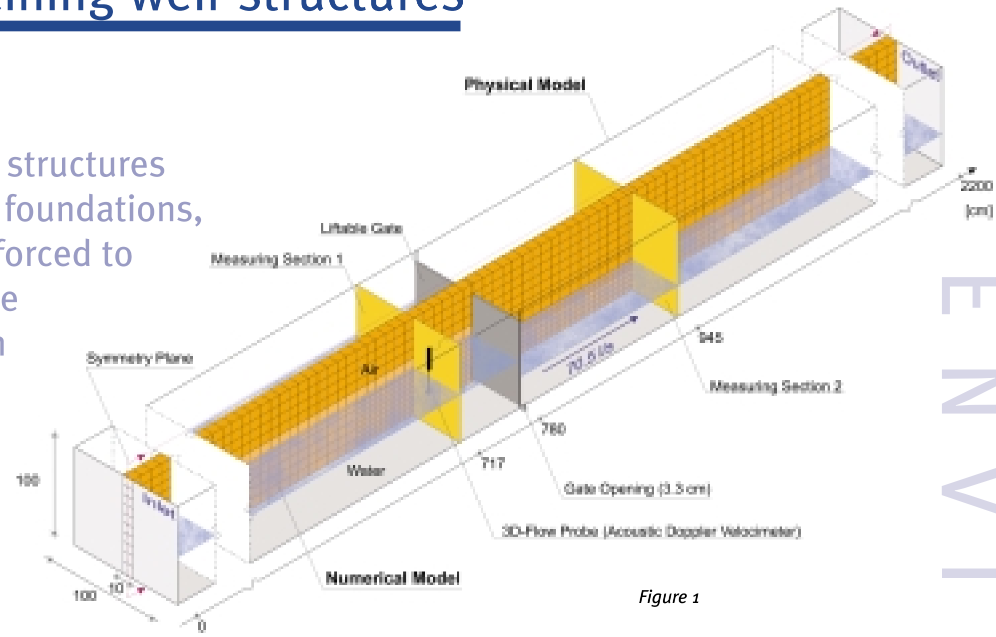


Figure 1

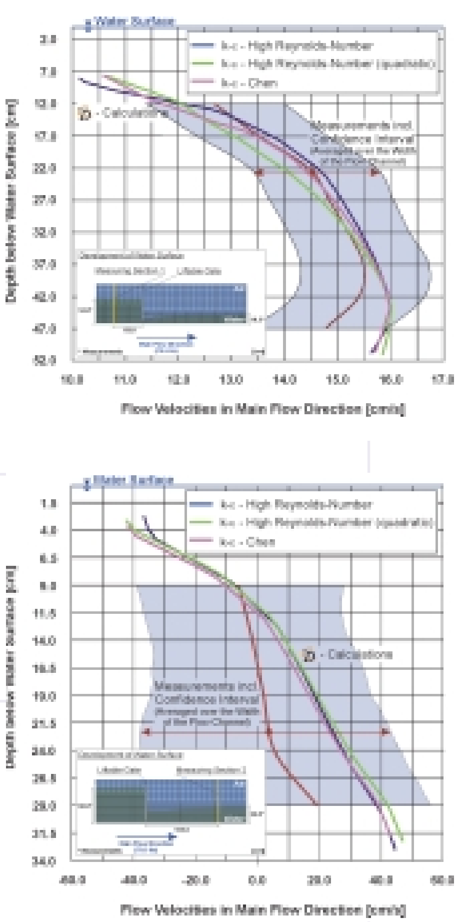
Numerical simulations are an obvious way to optimise the design process, and consequently there is a need for validation. Weirs are used to regulate waterflow in streams and rivers (Figure 1). Liftable gates are placed between piers and the foundations of the structure are reinforced to avoid the effects of erosion. The flow through the weir is characterised as subcritical or supercritical. When conditions allow, a transition from supercritical to subcritical flow occurs and involves the dissipation of energy across an abrupt change in water depth, a 'hydraulic jump', and this may substantially affect the stability of the foundation. The so-called 'roller' at the hydraulic jump can be of two types. The first is a free-surface roller, which confines flow recirculation near the water surface at the hydraulic jump and is more erosive. The second is a surface roller covered from the downstream water which appears immediately downstream of the weir gate. This results in a higher water depth, and in this case the re-circulation occurs over the entire depth of water with reduced erosion effects. The question arises as to how far it is possible to calculate such a complex flow situation using numerical simulations and still make time and cost savings.

To answer this question, physical model tests were carried out in a flow channel (2200 cm long, 100 cm wide and 100 cm deep) at the University of Hannover. A liftable gate with a thickness of 1 cm was located in the middle of the flow channel. By opening the gate at a height of 3.3 cm in a constant stream of 70.5 l/s, it was possible to investigate both types of hydraulic jump using two different loadings. Detailed measurements were taken of the water heights and the velocity distributions at 88 points on two measurement sections, 63 cm upstream and 165 cm downstream of the gate, using a 3D Acoustic Doppler Velocimeter flow probe (Figure 1).

To complement the physical model tests, numerical calculations were run on the university's Silicon Graphics Origin Workstation using STAR-CD. The numerical model was confined to two dimensions, consisting of 5000 cells (Figure 2) and using the Volume-of Fluid-method. Furthermore, the performance of a variety of high Reynolds number turbulence models was investigated, namely the standard k/e model, the k/e-Chen model and the k/e Quadratic non-linear model.

In conclusion, comparing the numerical results with the physical tests, the prediction of the water surface heights differs by no more than 6 % for both loading cases. Similarly, the flow velocity predictions do not differ by more than 6.5 % from the measurements. The effect of changing the turbulence model on the results and on the calculation times were marginal. Consequently, STAR-CD provides a new medium for designing and maintaining weir structures and can supplement the hitherto time-consuming and costly physical model tests carried out almost exclusively in this field (Figures 2 and 3).

For further information please contact: tobilin@web.de (Tobias Linke), scheffermann@web.de (Jens Scheffermann) or spekker@web.de (Heiko Spekker).



Figures 2 and 3

Elisabeth Akoh Hove, Project Engineer,, Danish Maritime Institute, Denmark

Flow and Energy Optimisation

of Electrostatic Precipitators

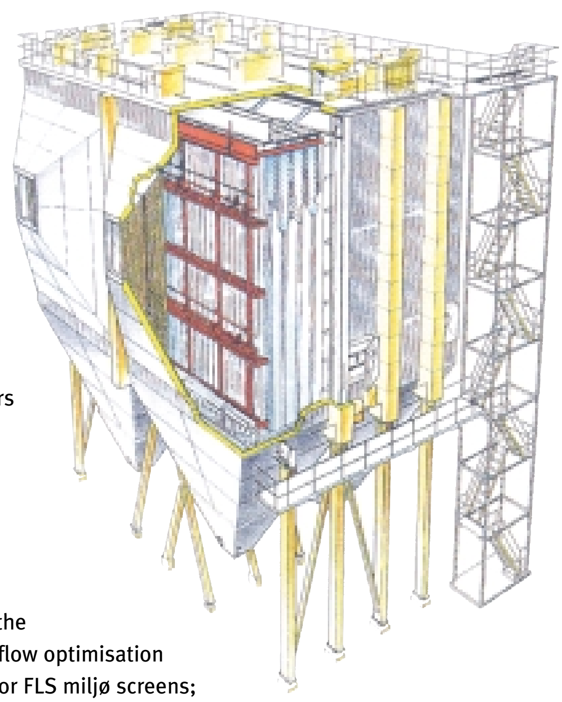


Figure 1: Electrostatic precipitator

The Danish Maritime Institute (DMI) and FLS miljø a/s have carried out a project using STAR-CD with the aim of developing a generic tool for the design of flow-conditioning devices applied at inlets of electrostatic precipitators.

The main objective of the project was to reduce the energy consumption and to obtain a better industrial process by developing and applying modern flow-calculation methods to the design of new and reconstruction of existing flue-gas cleaning plants. Through application of the method, optimisation of the gas distribution in electrostatic precipitators has been achieved resulting in:

- reduced energy consumption
- reduced emission levels
- higher efficiencies

The newly developed model is based on Computational Fluid Dynamics (CFD) with which the three-dimensional flow in an electrostatic precipitator can be calculated and the effect of flow optimisation devices such as screens can be predicted. The implemented model is designed specially for FLS miljø screens; however, the basic model is generic and can be applied with different input data for calculation of the flow field in other designs and applications.

Accurate modelling of the entire screen geometry with CFD would demand more computational power than today's computers can provide. Instead, the effect of screens on the flow was calculated through modelling of the resulting forces acting on the flow due to the screens, reducing the computational effort significantly.

A series of physical model scale tests involving flow through screen elements was carried out, partly to generate input data for the model, partly to verify the model. These tests yielded detailed information about the effect of specific screens applied by FLS miljø. On the basis of the model tests, it was possible to verify equivalent numerical calculations of flow through screen elements, so that input data could be established by numerical means. A detailed database containing the forces acting on the flow due to the screens was set up and used as input data in the model.

The model was used to predict the gas distribution in FLS miljø electrostatic precipitators, and comparisons with experimental scale model data as well as full-scale measurements were carried out. Satisfactory agreement between experimental data and the model predictions was found.

The project has shown that it is possible and advantageous to apply numerical calculations for dimensioning flow-conditioning devices. The developed model is flexible and allows a fast evaluation of different screen configurations leading to easier design and optimisation.

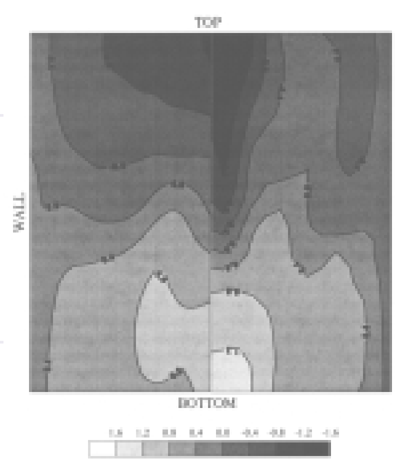


Figure 3: Velocity distribution. Comparison between computed results (left) and experimental data (right) for a FLS miljø experimental screen

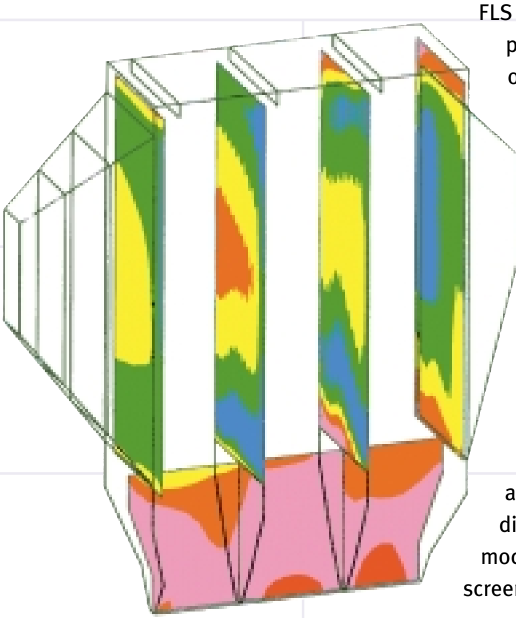


Figure 2: Axial velocity distribution in different planes in an electrostatic precipitator

For further information, please write to the Head of Department, Henning Hassing hh@dmi-online.dk

Peter Stephenson, Innogy plc, Swindon, UK

CFD Analysis of a novel Isothermal Compression process

Innogy plc is developing a reciprocating piston compressor, into which water is injected in order to cool the air and make the compression as near to isothermal as possible. This reduces the work required to compress the gas and therefore increases the overall efficiency of the plant. Possible uses will be in the power generation, fuel gas compression, air compression, and in other process industries.

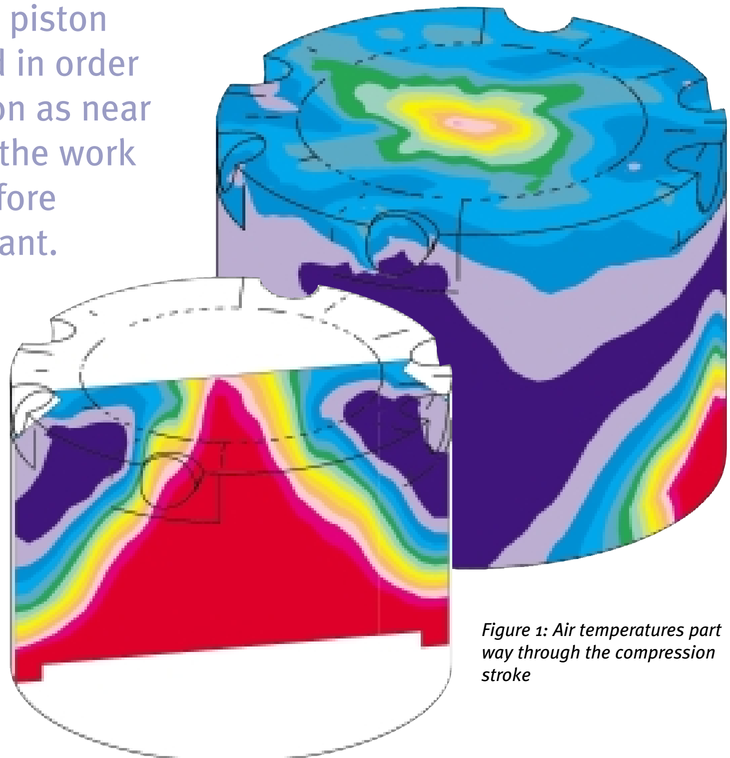


Figure 1: Air temperatures part way through the compression stroke

The plant has been developed in three stages. Detailed measurements have been made at Cranfield University on single sprays. A 'Proof of Concept' compressor has demonstrated that the basic ideas are sound and a full 'Engineering Demonstrator' is being tested at present. Both of these compressor tests are being carried out by Ricardo Consulting Engineers.

STAR-CD is being used to predict the droplet dynamics and heat transfer within the very dense water sprays. Spray models have been developed and validated against detailed single-spray measurements and are now being used in models of the two compressors. These are multi-spray models that use STAR-CD's moving grid capabilities for in-cylinder geometries. The initial validation against the 'Proof of Concept' compressor showed good agreement between prediction and measurement and further validation is in progress. STAR-CD has also been used to predict the performance of the compressor in the 'Engineering Demonstrator'. Among other things, this analysis showed that the initial design of the spray ring did not provide adequate droplet penetration. This helped in the design of an improved ring.

In this way, STAR-CD has enabled some very complex two-phase flow and heat transfer behaviour to be modelled, and has significantly helped with the design and optimisation of this novel isothermal compressor.

For further information, please contact:
peter.stephenson@innogy.com

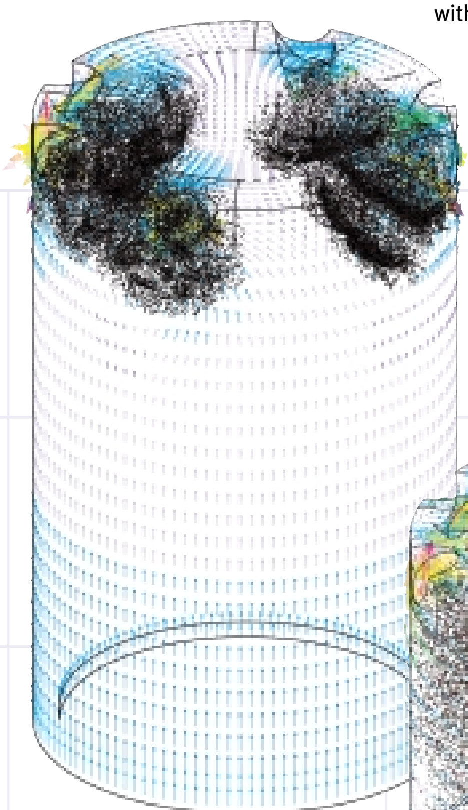


Figure 2: Droplet distribution and air velocities part way through the compression stroke

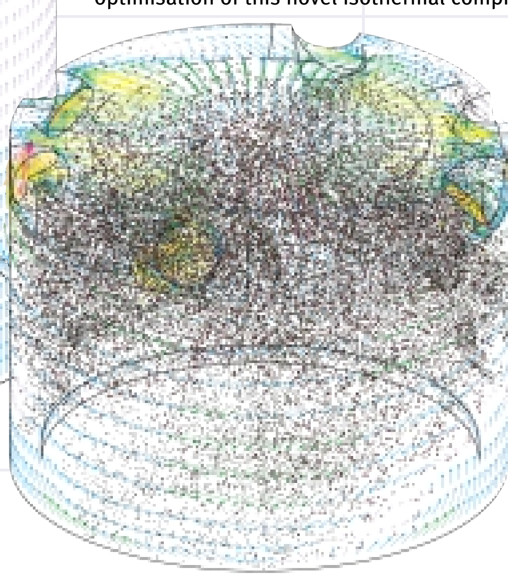
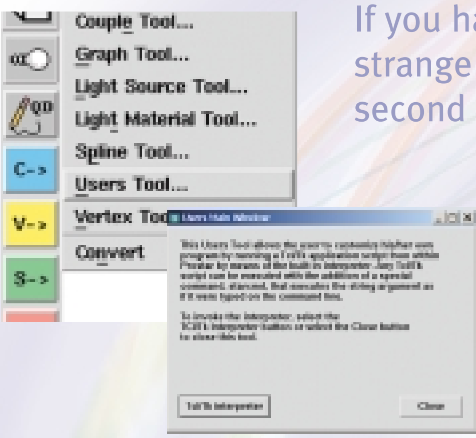


Figure 3: Air velocities near the start of the compression stroke

Dr. Mesh

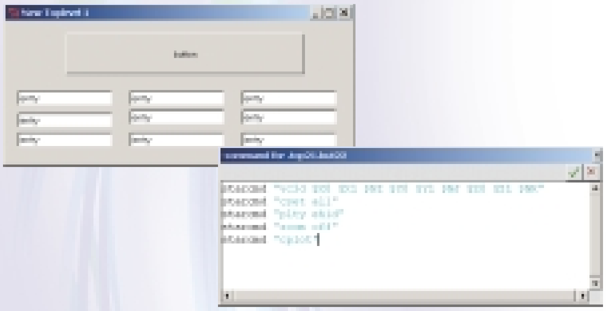
Extending PROSTAR



If you have explored the Prostar GUI, you may have noticed a strange item called Users Tool on the Tools menu which opens up a second panel.

This allows the user to run a Tcl/Tk application (called STARTkGUI.tcl) which gives you the standard Tcl/Tk environment in which to easily develop GUI applications. It has also been extended so you can issue Prostar commands from Tcl/Tk. To show how this happens, let's build a little example. I will use vtcl (http://vtcl.sourceforge.net) to make the tcl part very easy. If you do not have Tcl/Tk installed (most machines do) you can get it from http://www.scriptics.com.

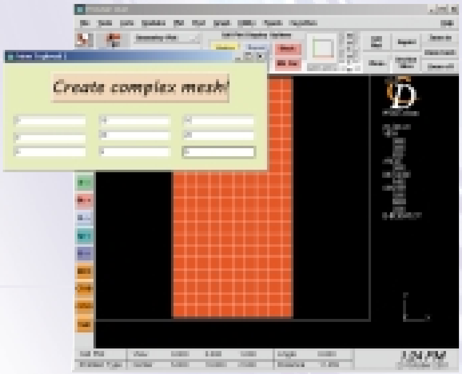
Fire up vtcl, and create a new top level. In the widget toolbar, click on button and then drag a button out in the toplevel, repeat this for nine entries and then after a little sizing you have something like the window on the left.



Now we need to wire up the GUI, click each entry in turn and in the attribute editor find the text var box and type in the name of the variables we are going to use. As this is going to be a GUI to the VC3D command, I am going to use variables Xo,X1,NI,Yo,Y1,NJ,Zo,Z1 and NK. Click the button and find the command in the attribute editor and put in this command:

starcmd "vc3d $Xo $X1 $NI $Yo $Y1 $NJ $Zo $Z1 $NK"
starcmd "cset all"
starcmd "plty ehid"
starcmd "zoom off"

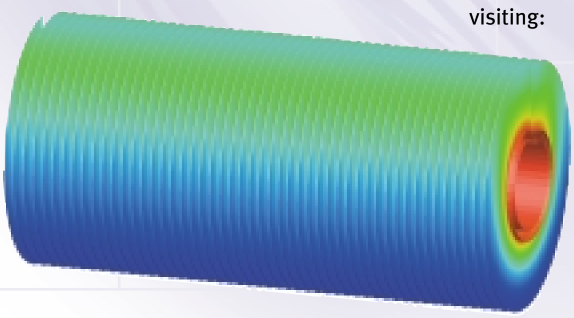
These are the standard Prostar commands except they are quoted for Tcl and where there is a dollar sign ($) the Tcl variable is substituted. In vtcl, switch from edit to test mode and clear the text in the entry boxes; you can also play with colours and fonts.



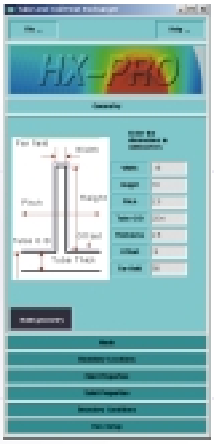
Now run Prostar in the same place where STARTkGUI.tcl is stored, fire up the Tcl/Tk interpreter and the new panel should open. Populate the panel, click the button and Prostar should do what you have told it to do.

This is of course a simple example; a more complex one, a heat exchanger, is illustrated below. Many of the EZ tools are built using the same process so it gives an idea of what can be done.

More information about this case can be found by visiting:
www.cd-adapco.com/support/drmesh.htm

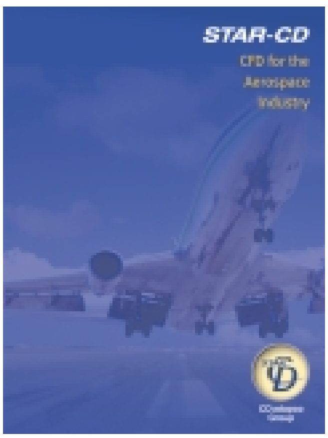


Building a heat exchanger, quickly!



Technical Papers

The following publications are now available and can be obtained by sending an email request to info@cd.co.uk, stating your preferred brochure and contact details, or alternatively you can download them as Adobe PDF's from: www.cd-adapco.com/products/brochuredownload.htm



CFD for the Aerospace Industry

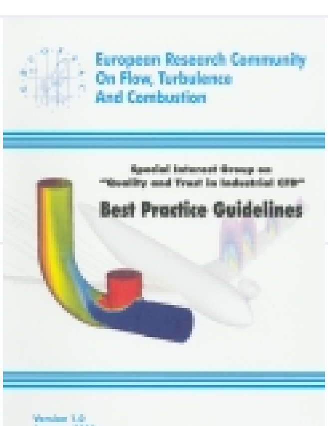
This brochure gives an overview of STAR-CD's applications in the aerospace industry. STAR-CD's ability to deal with flow conditions ranging from subsonic to supersonic, and anything in between, allow it to deal with a wide range of specialised flow problems encountered in the aerospace industry. Typical applications include:

- Aerodynamics
- Rocket nozzle and propulsion system
- Cabin Environment Control
- Aeroacoustics
- Sub- to hypersonic

STAR-NET 2.0

STAR-NET is a utility designed to automate and optimise parallel CFD computation using STAR-HPC on clusters of workstations. A typical application would be to exploit idle might-time CAD resources in an efficient and 'crash proof' way. STAR-NET addresses the most important aspects of running a parallel CFD code on a cluster:

- Network cluster resource management
- Heterogeneous resource load balancing
- Fault tolerance on system failure



The ERCOFTAC Best Practice Guidelines for Industrial Computational Fluid Dynamics

The Best Practice Guidelines (BPG) were commissioned by ERCOFTAC following an extensive consultation with European industry, which revealed an urgent demand for such a document. The first edition was completed in January 2000 and constitutes generic advice on how to carry out quality CFD calculations.

This publication is available from:
A G Hutton, G26/X80, QinetiQ, Ively Road, Hampshire, GU14 0LX, UK
Tel:+ 44 1252 395568, Fax:+44 1252 395322, email:aghutton@QinetiQ.com

The price per copy (not including postage) is:

Non-ERCOFTAC Members	150EUR (discount price for academics 75EUR)
ERCOFTAC Members	100EUR (discount price for academics 50EUR)

STAR-CD

Engineering CFD & CAE Solutions

Global Training Dates

Training dates for UK, USA, Germany and France are listed at: www.cd-adapco.com/support/training.htm

Exhibitions

European and North American exhibitions are listed at: www.cd-adapco.com/company/events.htm

News

For the latest news stories from the CD adapco Group please visit: www.cd-adapco.com/news

Electronic News

If you would like to receive regular news updates via email and your next newsletter in pdf format, please fill in the form at: www.cd-adapco.com/news/newsletter2.htm

New Brochures

The latest STAR-CD and related product brochures including EZICE and STAR-NET 2.0 can be downloaded in pdf format from: www.cd-adapco.com/products/brochuredownload.htm

CD adapco Group

CD adapco Group

Computational Dynamics

200 Shepherds Bush Road
London W6 7NY
ENGLAND
Tel: (+44) 20 7471 6200
Fax: (+44) 20 7471 6201
info@cd.co.uk
support@cd.co.uk
www.cd-adapco.com

adapco

60 Broadhollow Road
Melville, NY 11747
USA
Tel: (+1) 631 549 2300
Fax: (+1) 631 549 2654
starinfo@adapco.com
www.cd-adapco.com

CD-adapco Japan

Nisseki Yokohama
Building 16F
1-1-8, Sakuragi-cho, Naka-Ku Yokohama,
Kanagawa 231
JAPAN
Tel: (+81) 45 683 1998
Fax: (+81) 45 683 1999
info@yokohama.cd-adapco.co.jp
www.cd-adapco.co.jp

Germany

Computational Dynamics (Germany)
Dürrenhofstrasse 4, 90402 Nürnberg
GERMANY
Tel: (+49) 911 94643 3
Fax: (+49) 911 94643 99
info@cd-germany.de
www.cd-adapco.com

France

CD-adapco France
31 rue Delizy, 93500 PANTIN, Paris
FRANCE
Tel: (+33) 1 41 83 75 60
Fax: (+33) 1 41 83 75 66
info@cd-adapco.com
www.cd-adapco.com

Italy

Computational Dynamics (Italy)
Via Ferrarese 3, 40128 Bologna
ITALY
Tel.: (+39) 051 4198 674
Fax: (+39) 051 4198 450
info@cd-italy.com
www.cd-adapco.com

Turkey

A-Ztech Ltd
PERDEMSAC Plaza, Kat: 9/97
Bayar Cad.,Gulbahar Sok., No: 17
Kozyatagi, 81090 Istanbul
TURKEY
Tel: (+90) 216 361 8850
Fax: (+90) 216 361 8851
info@a-ztech.com.tr
www.a-ztech.com.tr

info(+)TRON A.S.
F.K.Gokay No.27
Altunizide
3 81190 Istanbul
TURKEY
Tel: (+90) 216 492 1002
Fax: (+90) 216 343 2132
analiz@infotron.com.tr
www.infotron.com.tr

Czech Republic

SVS FEM s.r.o
Cechynska 25
602 00 Brno
CZECH REPUBLIC
tbachorec@svsfem.cz

Russia

CAD-FEM GmbH
Representation in CIS
Office 1703
77 Schelkovskoe Shosse
Moscow 107497
RUSSIA
Tel: (+7) 095/468-8175
Fax: (+7) 095/913-2300
info@cadfem.ru
www.cadfem.ru

Iran

Sherkat Mohandesin Moshaver
Dynamic Sayallat
No.18, Fifth Floor
Golha Building, Fifth Alley
Northwest of Noor Square
Tehran
IRAN
Tel: (+98) 21 4062528
Fax: (+98) 21 4062660
infostar@cd-iran.com
www.cd-iran.com

India

CSM Software Pvt Ltd.
2nd Floor, Niton Building
11 Palace Road
Bangalore 560052
INDIA
Tel: (+91) 080 2200 996
Fax: (+91) 080 2200 998
info@csmin.com
www.csmin.com

Australia

Orbital Consulting.
1 Whipple Street
Balcatta
Western Australia
AUSTRALIA 6021
Tel: (+61) 8 9441 2311
Fax: (+61) 8 9441 2345
starinfo@orbeng
www.orbeng.com

South Africa

CSIR
Box 395, Pretoria
0001
SOUTH AFRICA
Tel: (+27) 12 841 4843
Fax:(+27) 12 349 1156
cfd@pixie.co.za
www.csir.co.za

Korea

CD-adapco Korea
#905 The Korea Teachers
Pension Bldg.
27-2 Yoido-Dong
Youngdeungpo-Gu, Seoul
SOUTH KOREA 150-742
Tel: (+82) 2780 1760
Fax: (+82) 2780 1763
info@cdak.co.kr
www.cdak.co.kr

China

CD-adapco Japan Co Ltd.
Beijing Office
Room 1208A, Tower A,
FullLink Plaza No.18
Chao Yan Men-
Wai Da Jie Street
Beijing, 100020
PEOPLE'S REPUBLIC OF CHINA
Tel: (+86) 10 65881497/8
Fax: (+86) 10 65881499
cdbj@public.bta.net.cn

Singapore

CAD-IT Consultants
159 Sin Ming Road
03-05 Amtech Building
Singapore 575625
SINGAPORE
ftansb@mbox.signet.com.sg
www.cadit.com.sg

Taiwan

Flotrend corp.
3F, 72 Sungteh Road
Taipei, Taiwan 110
REPUBLIC OF CHINA
Tel: (+886) 2 2758 7668
Fax: (+886) 2 2758 9798
gary@flotrend.com.tw
www.flotrend.com.tw

Malaysia

Numac Systems
Technologies S/B
40-42 Jalan Petaling Utama 3,
Taman Petaling Utama,
6 1/2 Miles, Jalan Klang Lama,
Petaling Jaya
46000 MALAYSIA
Tel: (+60) 603 77 838 188
Fax: (+60) 603 77 832 067
dlimol@pc.jaring.my
www.numacmachine.com

Photographic cover image courtesy of Rolls Royce plc, www.rolls-royce.com