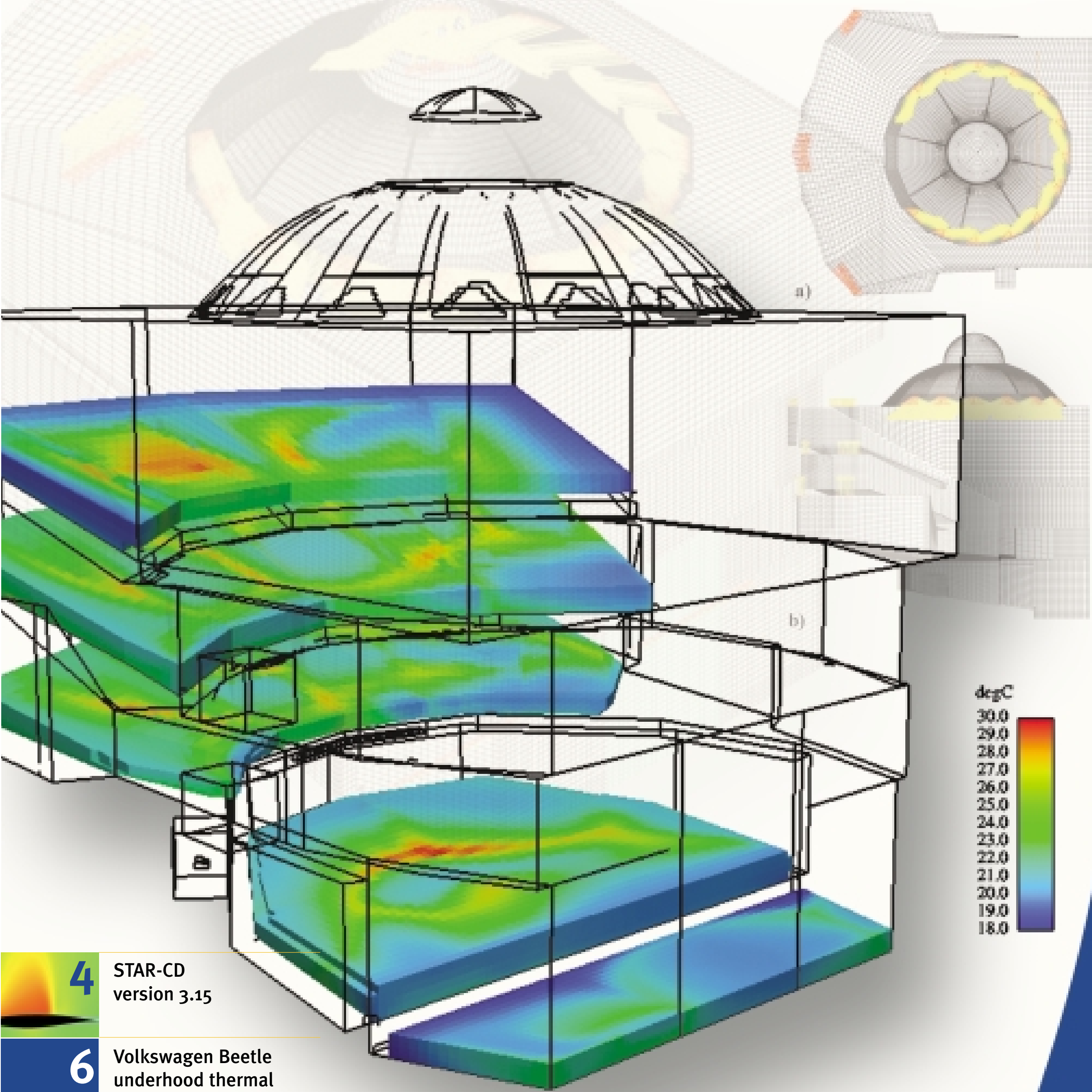
STAR-CD

# DYNAMICS

ISSUE 15 SUMMER 2001

The Newsletter of the CD adapco Group






## 'Arup use STAR-CD to breathe new life into London Coliseum' p. 11-12

STAR CD

CD adapco Group

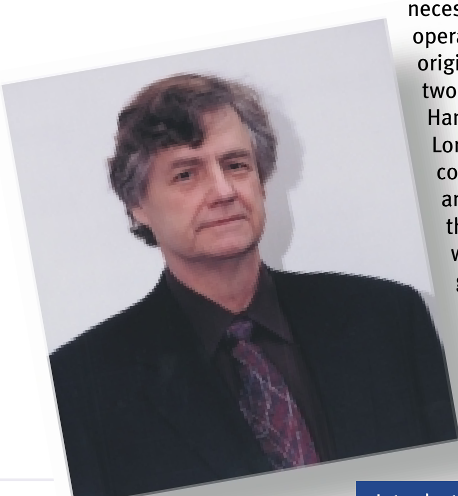
# Changes - What's behind them?

## Dear STAR-CD user,

The CD adapco Group is a closely held set of inter-related companies which are involved in the development and delivery of CAE analysis software (at this time mainly CFD) and other complementary software, engineering methodology, and engineering services to industry. The delivery organization is called an AGENCY. With this background in mind, we have reorganized CD into two separate business units, a Product Development Operation and European Agency Operation, and we have made some new senior appointments.

The Product Development Operation is responsible for STAR-CD and the development of interfacing methodology, and its associated software for coupling STAR-CD to other analysis and optimization products. Professor Milovan Peric has been appointed its director. Many of you know Milovan from his book 'Computational Methods for Fluid Mechanics'. Not only is Milovan technically competent and creative, he also has natural leadership and organizational skills.

These skills are of utmost importance in dealing with our large and highly diverse set of developers on the STAR-CD team. It should be noted that in order to attract such a large and highly capable team it has been necessary to accept a distributed operation. We have the mainstay original team located in London and two other smaller teams located in Hamburg and New Hampshire. The London team is responsible for continued development of STAR3 and maintaining close ties with the two other teams which are working on our new next generation CFD code STAR4. Computational Dynamics is committed to producing the best in class general purpose CFD software with a strong tie to industry.



To meet this commitment the Group is investing all profits and then some in this development activity.

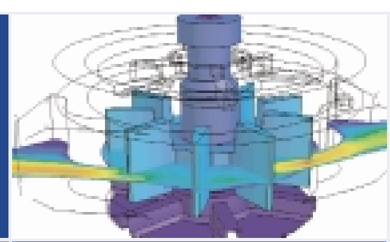
The other business unit (European Agency Operation) which has its offices in London, Paris, Nuremberg, Hamburg and Bologna is directed by Didier Halbronn who joins us from ANSYS Europe. The engineering and support activities of this operation is lead by Marc Zellat, Director of Engineering. This part of the business is modeled on, and is closely linked with, the successful businesses of adapco and CD-adapco Japan. The aim here is to serve you through "value added engineering". This is a neat way of saying to you we know that there is more to CFD than the CFD code - at some stage you will surely require extra help to get the full benefit from the technology. So, in addition to offering STAR-CD with its associated training and support services, our European Agency Operation will be offering EZ-Tools for particular applications and some uniquely supportive engineering services options. These added components aim to blend our engineering/CFD skills with your product knowledge to reduce time to market and increase your competitive edge and profitability. Our aim is to do whatever it takes to help you get the best and fastest engineering solutions.

By reorganizing the company in this way, we hope to serve you better through strengthening both the technology and engineering aspects of our business. Let's see if it works - I think it will.

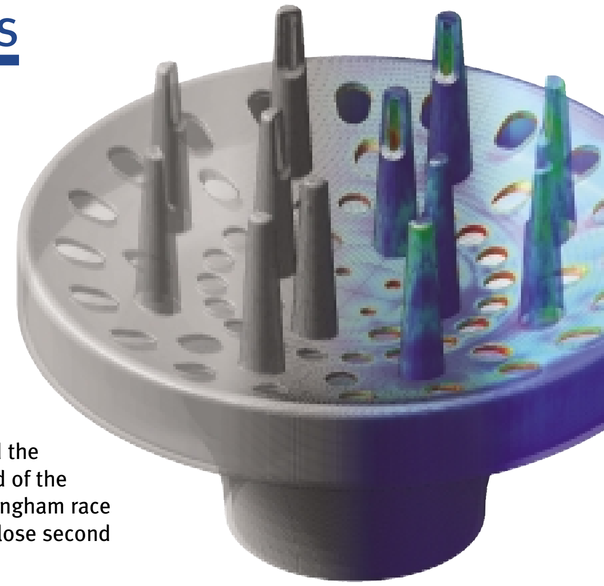
*Steve MacDonald*

CONTENTS





# CD adapco Group Corporate News



## SolidWorks interface for STAR-CD

The SolidWorks interface for STAR-CD has been completed. It is available on request to licensed users of STAR-CD 3.1B for NT. The interface can be used with copies of SolidWorks supplied either by the CD adapco Group with STAR-CD, or as a stand alone product from local SolidWorks VAR's.

## Motorsport Success

Demonstrating its continued support of UK motorsport the CD adapco Group sponsored the Performance Racing team in the latest round of the British formula 3 championship at the Rockingham race circuit, UK. Driver Adam Blair finished at a close second position.

"Adam is the only driver in Formula 3 raising his budget commercially. He has no wealthy parentage to fall back on. Contributions from companies such as Computational Dynamics allow him to compete "

## Professor Milovan Peric joins CD adapco Group developers

Prof Milovan Peric, the well known expert on CFD and co-author (with J H Ferziger) of one the most widely-referenced books in our field 'Computational Methods for Fluid Mechanics' has joined the international development team of the CD adapco Group as Director of Product Development. We are truly honoured that Milovan has confirmed his commitment to us by deciding to give up his professorship at Harburg-Hamburg University to focus his efforts on the development of STAR-CD and COMET. He joins Prof Ismet Demirdzic and Dr Samir Muzaferija within the partnership recently formed between our Group and ICCM.

Professor Peric will be known to many of you through his numerous publications in the areas of finite volume methods, moving grids, free surface flows, multigrid methods and parallel computing. With his knowledge and experience he is destined to lead the CD adapco Group's STAR-CD suite of CFD-related codes to new heights of functionality and performance.

## COMET to orbit STAR-CD

Computational Dynamics (London) has joined ICCM (Hamburg) in a collaboration to develop and sell the latter's CFD code COMET, which has multiphysics capabilities, including an embedded structures solver for fluid/structure interaction simulation. The agreement also involves joint development work on STAR-CD with the possible long-term goal of merging the two codes, which have complementary capabilities and common 'genes'. Existing ICCM contracts will be honoured to ensure continuity of support to COMET users and agents. However, with immediate effect the CD adapco Group has been selling and supporting COMET worldwide as an addition to their STAR-CD suite of CFD tools.

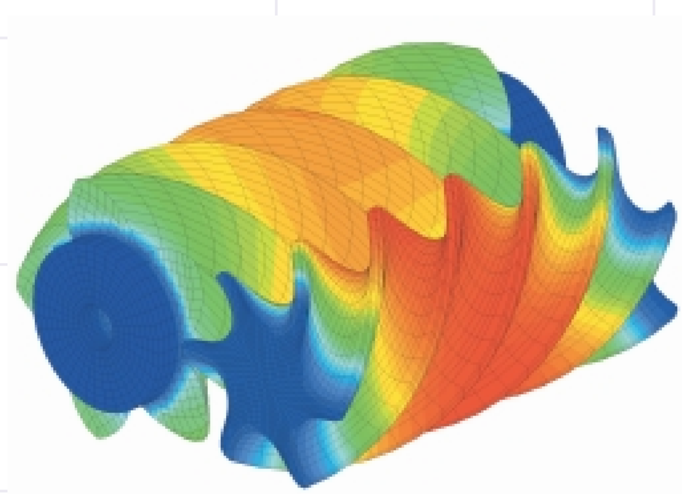


Figure 1:
COMET simulation of displacement magnitude in a screw compressor (Solid body analysis)

# International User Group Meetings

## North America, Korea and Japan

The Northfield Hilton in Troy, Michigan was once again the setting for the STAR-CD North American Users' Conference hosted by the CD adapco Group in May. Over 160 attendees enjoyed presentations given by clients and employees of the CD adapco Group worldwide.

*Users enjoying the reception at the North American Meeting*

A demonstration area featured nine hardware and software companies including Compaq, CEI, Pointwise, Hewlett Packard, IBM, Program Development, Intelligent Light, Reaction Design and VALinux. The CD adapco Group is grateful to Compaq and Hewlett Packard for sponsoring the lunch on both days of the conference.

In addition to presentations on pro*am, STAR-LT, COMET and our suite of EZ Tools, presentations were given by individuals from a variety of industries including automotive, power generation, medical, turbomachinery, gas turbine and architectural. These were good examples illustrating the diverse applications of STAR-CD. The presentations can be found at www.adapco-online.com

The reception, held on the first evening of the conference, was well attended and provided users with the opportunity to meet and share information and to interact with CD adapco Group employees.

The STAR-CD Korean User Group Meeting attended by over 150 users was the next meeting to be held, on June 19th 2001 in Seoul, Korea.

As usual, STAR-CD users shared their experiences of STAR-CD applications. The varied topics included aeroacoustics, EZ-FSI, developments in pre- and post processing and fuel cells analysis. Professor Lee, Soogab, from the school of Mechanical and Aerospace Engineering at Seoul National University, gave an extremely interesting presentation on the applications of computational aeroacoustics. This included topics such as centrifugal compressor noise prediction, research on marine propeller noise, helicopter aerodynamics and aeroacoustics and wall boundary treatment for computational aeroacoustics. Steve MacDonald presented EZ-FSI, which has been developed as an approach to facilitate the solution of FSI problems with STAR-CD.

*Japanese conference in Yokohama*

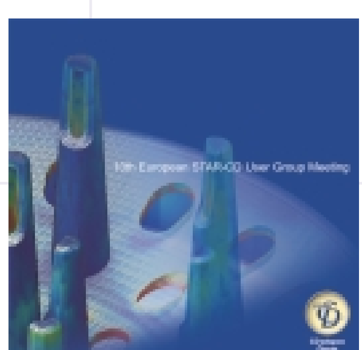
The STAR-CD Japanese User Group Meeting was then held on June 21st-22nd 2001 at the Pan Pacific International Conference Center in Yokohama. The success of the meeting was reflected by the attendance of over 400 users and 15 companies exhibiting hardware and software.

On the first day Professor Carlo Poloni (from Engin Soft, Italy), presented 'Frontier', a state of the art tool for multidisciplinary design optimisation. Several other companies also presented examples of STAR-CD and Frontier usage. Another presentation was given on the first day about a human comfort assessment model with detailed thermoregulation analysis modeling. An evening reception was held at the end of the day including a light meal, drinks and a special magic show!

The last day's key speeches were on the latest fuel cell simulation technology and the detailed chemical reaction analysis tool coupling with 3-D CFD (PEM and GAPC). Aside from the conference room, over 50 demonstrations and mini seminars took place to introduce new products and research, such as Mixpert, Chemkin and Frontier.

A highlight at the end of all three conferences was a presentation by David Gosman (the founding director of Computational Dynamics) on future developments of STAR-CD.

All in all, the three meetings were commended as enjoyable by all who attended.




### European STAR-CD User Group Meetings

The CD adapco Group would like to invite you to attend European User Group Meetings in Paris on the 23rd October and in London on the 19th and 20th November 2001.

For information regarding the French meeting please visit www.cd.co.uk/company/ugmfr2001.htm

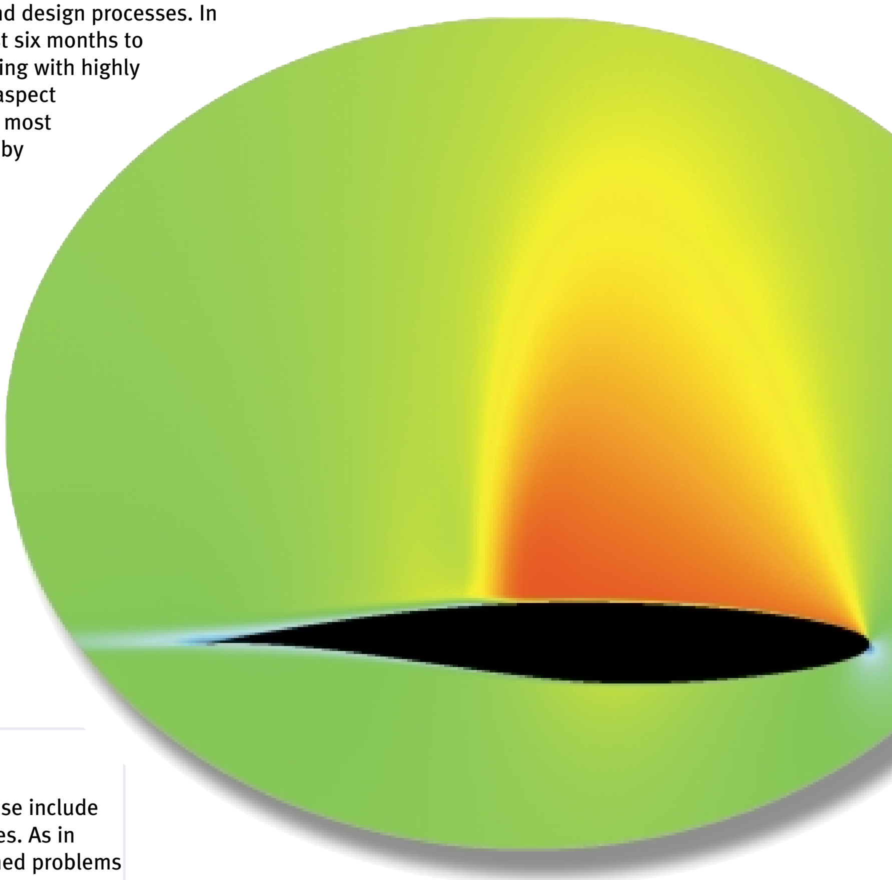
Please see attached invitation for details of the 10th European User Group Meeting in London. If your invitation has not been included with the newsletter, please visit www.cd.co.uk/company/ugm2001.htm for details and registration. If you have any further queries please email us at info@cd.co.uk or telephone on +44 (0)20 7471 6200.

# STAR-CD version 3.15

## Now - even faster!

V3.15's enhancements relative to V3.10 lie in the areas of solver robustness and efficiency, additional modelling capabilities and enhanced pre- and post-processing options.

STAR-CD is applied to ever more complex industrial problems and is playing an increasingly important role in industrial design and development decisions. Therefore, the reliability and robustness of the software are of prime importance to our users as they try to improve the speed and efficiency of their analysis and design processes. In response to their needs, we have made a major effort in the last six months to increase solver robustness and reliability, especially when dealing with highly deformed cells such as those with small internal angles, large aspect ratios and nearly co-planar faces. The resulting improvement is most evident in tetrahedral mesh analyses. This has been confirmed by our test program, run with over 50 industrial CFD cases and additional in-house cases.



Pre-processing, especially mesh building, is often the most time consuming part of the CFD analysis of a complex geometry problem. To reduce the time required for this process, we are offering a new, separately-licensed package called pro*am which offers all capabilities currently in PROSTAR, plus automated meshing with tetrahedral and trimmed cell methodologies.

In addition, a number of other application-specific tools (EZAero for aerodynamics, EZUnderhood for vehicle underhood analyses) are to be made available as plug-ins to the basic meshing facilities. The automated meshing tools, as well as the application-specific tools are each available under additional licenses.

Other improvements specific to PROSTAR include interfaces to a wider variety of external CAD/CAE packages and file formats, such as CGNS, TGRID™ and TECPLOT™.

In V3.15 we are also offering, on a beta basis, a number of new modelling options that are of special interest to our clients. These include aeroacoustic analysis and new turbulence modelling capabilities. As in previous releases, this version contains bug fixes for all confirmed problems reported to date.

# STAR-CD User Competition

With the latest version of STAR-CD, you will have been sent postcards to enter a competition to win one of the latest digital cameras, the HP Photosmart 315. It is not too late to enter! The date of the prize draw has now been extended to 1st November, 2001. For your chance to win, please fill in the prize draw from at www.cd.co.uk/prizedraw.htm, or e-mail your contact details to info@cd.co.uk, stating "PRIZEDRAW" in the subject field.

business partner hp invent

Laetitia Larriau Labree, STAR-CD development team

# Fluid Structure Interaction

## STAR-CD with MpCCI

The CD adapco Group provides engineers with three approaches to Fluid-Structure Interaction (FSI) analysis: (a) the EZ-FSI tool which enables STAR-CD to work with FEA codes to compute linear deformations using a special solver embedded in STAR-CD, (b) the COMET code, a continuum mechanics code which uses finite volume methodology to simultaneously compute both fluid and structure mechanics, and finally (c) MpCCI linkage, which is the subject this article.

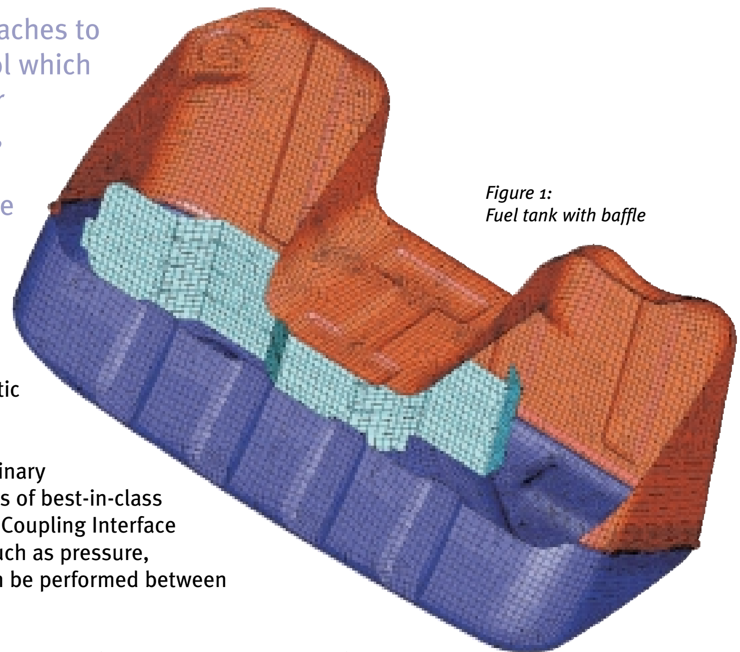


*Figure 1: Fuel tank with baffle*

MpCCI is a universal interface which allows data to be exchanged between a CFD code and (in principle) any other specialized finite volume or finite element analysis code, including linear and non-linear structures and acoustic codes.

The benefit of this general approach is that engineers can solve multidisciplinary problems within their areas of expertise whilst gaining access to the features of best-in-class simulation codes. The linkage is achieved via the Mesh-based parallel Code Coupling Interface (MpCCI) library. At run time, STAR-CD can exchange boundary information such as pressure, displacements, temperature or heat flux with other codes. The exchange can be performed between non-matching meshes as MpCCI will handle the interpolation.

The advantages offered by the MpCCI path have been fully utilized in the STAR-CD implementation. Setting up of a coupled case is identical to that for a non-coupled, i.e. fluid-only, case. All the standard unstructured mesh capabilities can be used. The additional information needed for an MpCCI run consists only of the specification of the boundaries which are involved in the coupling and of the quantities which are to be sent or received.

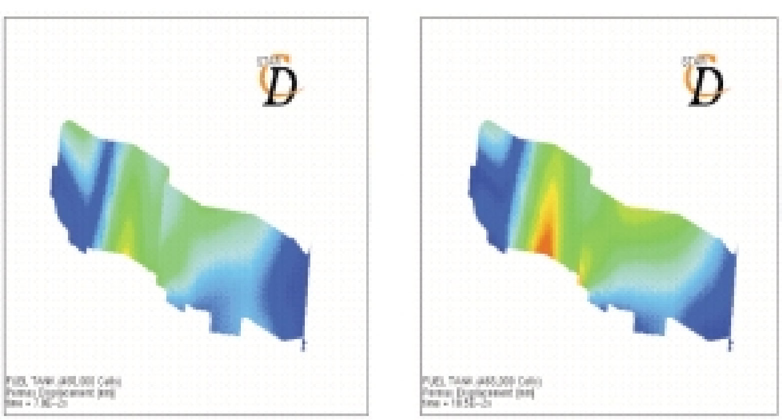


*Figure 2: Displacements of the baffle*

*Figure 3: Deformation of the mixer*

A typical application is a fluid-structure interaction problem involving stress and/or heat-transfer analysis. This requires STAR-CD to send forces and temperatures and receive heat fluxes and displacements at the interface boundaries. Boundary displacements usually require a repositioning of the CFD mesh, which is achieved in STAR-CD with its built-in mesh smoother. Turnaround times for coupled simulation have been kept to a minimum by employing full parallel running capabilities (HPC).

The CD adapco Group is currently collaborating closely with two major industrial partners, Daimler Chrysler AG and Sulzer Innotec AG to apply the method to real engineering problems. Two examples are shown: a static mixer used in the chemical industry where we compute the deformations induced by the flow (Figure 3) and a petrol fuel tank for automobiles (Figure 1) where we calculate the stresses on an internal baffle due to sloshing (Figure 2). Both models were successfully run and used to provide design information.

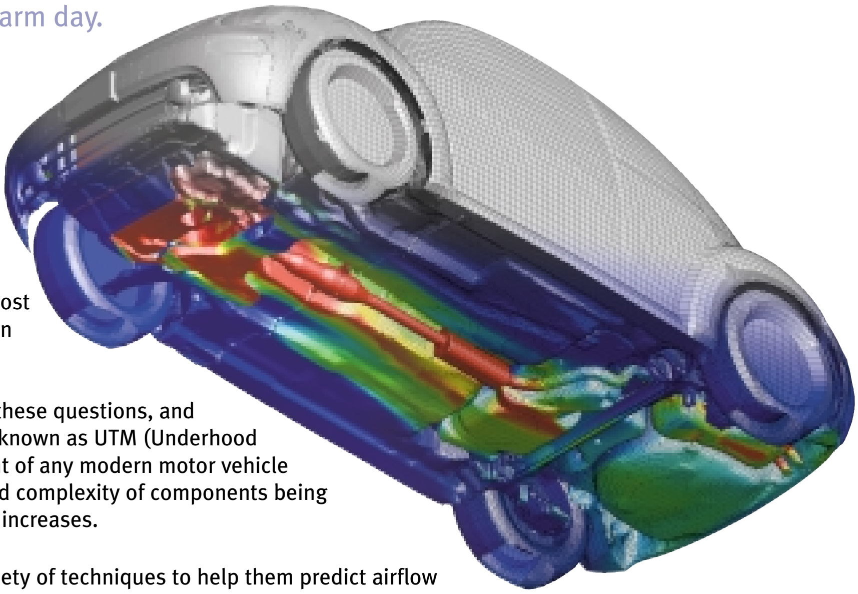
The above MpCCI functionality has been integrated into STAR-CD and will be available in Version 3.2. In summary, STAR-MpCCI coupling can provide STAR-CD users with a powerful new approach to FSI analysis.

Nader Fateh, CD adapco Group

# Underhood Thermal Management

## (or how STAR-CD helps a motor vehicle keep its cool)

Imagine a car moving at low speed under a heavy load (for example when towing a trailer uphill) on a warm day. How does the car keep cool?



Is the airflow sufficient to keep all the components in the engine compartment in a thermally controlled environment? Some of these can be operating at high temperatures (as in the case of a close-coupled catalytic converter), others will be sensitive to high temperatures (for example the engine control unit). How big should the grille be to allow enough air to enter from the outside? Is the ram effect sufficient, or does the air need the extra boost provided by a fan? If so, where is the best place to position the fan? What will happen if a fan fails?

It is the job of the vehicle design engineers to answer all these questions, and many other related issues. This branch of vehicle design, known as UTM (Underhood Thermal Management) is a crucial part of the development of any modern motor vehicle and it is becoming ever more important as the number and complexity of components being packed into the limited space of the engine compartment increases.

Volkswagen's engineers in Wolfsburg, Germany, use a variety of techniques to help them predict airflow rates and temperatures in the engine compartment of vehicles such as the new VW Beetle, shown here. STAR-CD, widely used in the VW-Audi Group for applications ranging from in-cylinder analyses to passenger compartment climate control, is one of the tools applied in this part of the vehicle's development.

In order to simulate engine compartment flow, the following criteria need to be satisfied :

- An accurate representation of the heat exchanger package(s) is essential.
- The effects of fan(s) on the flow must be included in the analysis.
- Reasonable boundary conditions must be applied to the inlets, outlets and walls (e.g. inlet turbulence conditions, wall thermal boundary conditions, moving wall boundary conditions).
- Thermal radiation effects must be included if they are deemed to be of importance. Note that if the goal is predicting the air temperature distribution, and reasonable wall thermal boundary conditions can be determined (i.e. boundary conditions which already include the effects of radiation), then it is not necessary to model thermal radiation.
- Conjugate Heat Transfer may be required – that is, the calculation of heat transfer both to and within the solid components.
- Other practical modeling and solution issues must be considered, such as the turbulence model to be used, the treatment of turbulent wall boundary conditions and the differencing scheme used.
- The user must be able to post-process the results to get the desired information.

STAR-CD can offer the UTM engineer all these features, as well as many others. The CD adapco Group's VTM product EZUhood, a customised tool for exactly this kind of analysis, will allow the time required to get results to be reduced significantly. This allows CFD to become even more of an integral part of the vehicle design cycle.

AUTOMOTIVE

# Benetton Formula 1

## The Race is on with STAR-CD

During the last two years, leading Formula One teams have been racing to get aerodynamic design improvement through whole-car CFD simulation using codes such as STAR-CD.

Benetton Formula 1 is using CFD techniques to gain better insight into the flow field than can be achieved by wind tunnel measurements. In collaboration with the CD adapco Group, Benetton Formula 1's design engineers are using STAR-CD to "fine tune" the aerodynamics of their race cars.

The method starts by importing geometry from a CAD model of the car and then building a CFD mesh automatically, using STAR-CD's engineering-based expert software tool "EZ-Aero". The mesh exploits symmetry for half-car modelling with about 10 million cells. Thanks to scalable parallel computing, these large-scale simulations can typically be completed overnight. By building up fine mesh layers from the CAD surface, an incredibly high flow resolution can be achieved. For example, the flow for the leading edge of a Benetton front wing can be accurately resolved to within 1 millimetre!

This approach to high resolution CFD simulation provides results that capture the car's aerodynamic flow field in extreme detail, including the effects of wheel motion and the thermal air-density effects of the cooling air leaving the engine bay.

The race is truly on at Benetton Formula 1 to use STAR-CD to push the aerodynamic design of their F1 cars to new heights.

N.E. Piersol * et al, University of Illinois at Chicago

# Carotid Bifurcation Hemodynamics

Researchers from the Departments of Mechanical Engineering* and Radiology** at the University of Illinois at Chicago are carrying our research in collaboration with Argonne National Laboratory***, aimed at using STAR-CD to simulate carotid bifurcation hemodynamics.

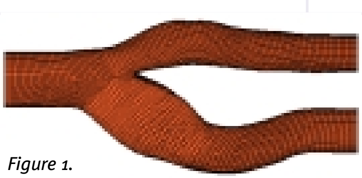


*Figure 1.*

During the past two decades, the fluid mechanics of blood flow has been implicated by medical and biological researchers as a key factor in the pathogenesis of arterial disease, the regulation of cellular biology in both normal and diseased arteries, and in the maintenance of vascular health.

Bioengineering research has demonstrated the importance of wall shear stress (WSS) in the regulation of arterial diameter, with arteries adjusting their diameters to maintain the average WSS within a preferred range. Sites within the carotid artery that experience low and oscillating wall shear stress are more likely to develop an atherosclerotic plaque which could eventually lead to a stoke. Progress over the last decade in both CFD analysis and medical imaging now allows subject specific models of blood fluid dynamics. These may someday lead to risk assessment and surgical planning based on CFD simulation of blood flow.

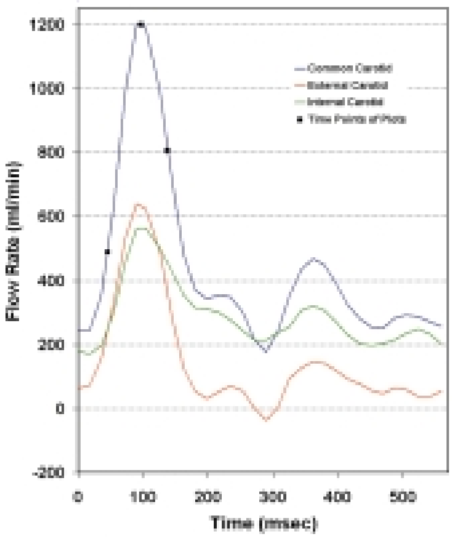



*Figure 2.*

This research presents an automated technique for examination of the blood fluid dynamics specific to a healthy subject's carotid bifurcation. Magnetic Resonance (MR) images of a subject's carotid artery bifurcation were non-invasively acquired and used to determine the vessel geometry. Using PROSTAR a mesh was constructed based on this geometry with 213,000 hexahedral cells generated via an in-house technique (Figure 1). The mesh also contains inlet and outlet extensions, which are not shown.

The CFD solution was determined under steady and pulsatory flow conditions using STAR-CD and the parallel computing system at Argonne National Laboratory's Division of Mathematics and Computer Science. Phase contrast MR (PCMR) was used to obtain the velocity on a cross-section of the common carotid artery (CCA), the internal carotid artery (ICA), and the external carotid artery (ECA) as a function of time for one cardiac cycle. The flow rates in each vessel were obtained by integration of the velocity obtained by PCMR (Figure 2). These flow rates were imposed as input boundary conditions for the CCA and ECA, while a stress-free outlet boundary was imposed at the remaining branch (ICA).

Under steady flow conditions, the numerical results were compared to experimental velocity data obtained using an upscaled flow model and laser Doppler anemometry. Both were conducted at Re=377 which represents the mean value in the CCA during the cardiac cycle. Good agreement was obtained between the experimental and numerical results (Figure 3).

Under pulsatory flow conditions, the simulation results showed dramatically different velocity profiles compared to the steady flow results as seen by the velocity vector plot at systolic acceleration, peak, and deceleration (Figure 4). Wall shear stress patterns also varied greatly during the cardiac cycle as shown by the series of WSS contour plots at systolic acceleration, peak, and deceleration (Figure 5). The time average WSS contour identifies a large region of low WSS (dark blue area) for this subject (Figure 6).

This example case demonstrates how CFD tools can be used to assess the distribution of biomechanical forces imposed on the arterial wall by the blood. The entire process (pre-processing, solution convergence, and post-processing) can be simulated in approximately 48 hours using parallel computing. Future advancements in computer speed will someday allow same day turn around for the simulation of carotid bifurcation hemodynamics. This may provide greater understanding of the relationship between hemodynamics and vascular disease, and may one day lead to a non-invasive tool for early prediction of carotid disease or surgical planning.

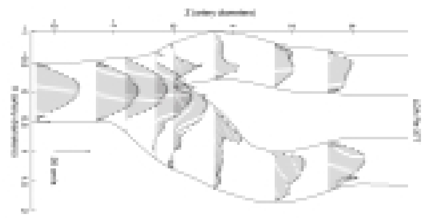


*Figure 3a.*

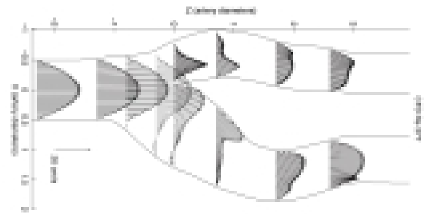


*Figure 3b.*

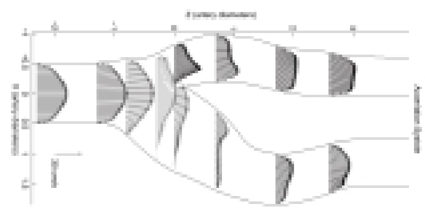


*Figure 4a.*

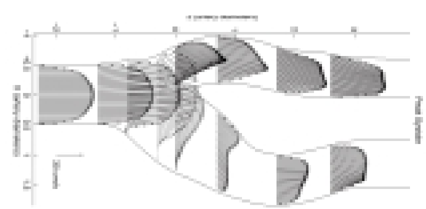


*Figure 4b.*

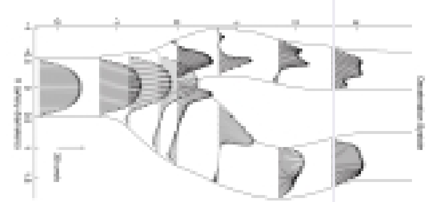
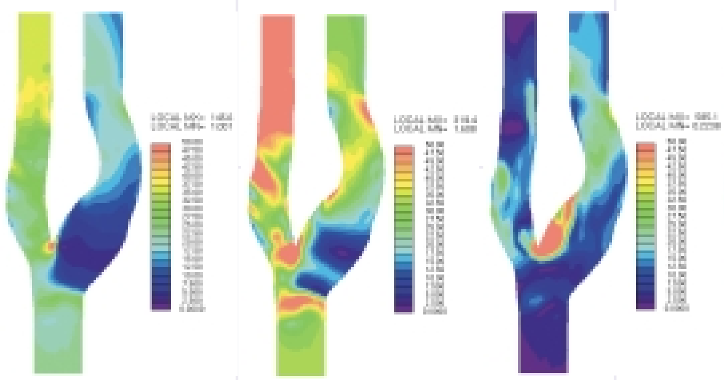


*Figure 4c.*

N.E. Piersol*, E.P. Ghengeaua*, S.E. Lee*, N. Alperin**, P.F. Fischer*** and F. Loth*

*Figure 1.* *Computational mesh of the carotid bifurcation*
*Figure 2.* *Blood flow rate measured by phase contrast MRI*
*Figure 3a.* *Experimental measurements of velocity from LDA under steady flow conditions*
*Figure 3b.* *CFD results of velocity from STAR-CD under steady flow conditions*
*Figure 4a.* *Velocity vectors during systolic acceleration*
*Figure 4b.* *Velocity vectors at the systolic peak*
*Figure 4c.* *Velocity vectors during systolic deceleration*
*Figure 5a.* *Magnitude of WSS during systolic acceleration*
*Figure 5b.* *Magnitude of WSS at the systolic peak*
*Figure 5c.* *Magnitude of WSS during systolic deceleration*
*Figure 6.* *Time averaged values of WSS magnitude over the cardiac cycle*



*Figures 5a,b,c*

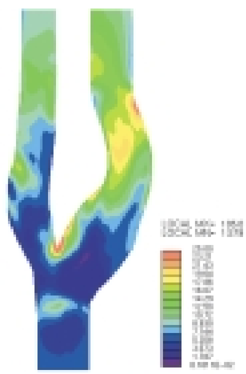


*Figure 6*

Dr. W.F. Schierholz and S. Ott, Siemens Axiva GmbH & Co. KG

# Scaling up of mixing reactors

## From laboratory to production plant scales

CHEMICAL

State of the art process development in the chemical industry uses Computational Fluid Dynamics as a tool for process design and scale up.

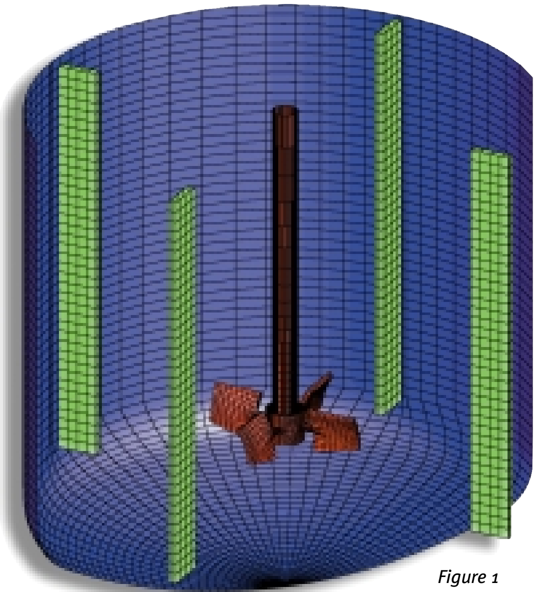


Figure 1

One of the most important considerations in mixing reactor design including the scale up of mixing reactors, is to establish the mechanism of process operation and thus determine whether the most sensitive process criteria are solids suspension, mass transfer, chemical reaction or shear rate phenomena.

Siemens Axiva has the expertise and capability to perform scale up laboratory experiments in combination with numerical predictions. An experimental scale up usually covers the laboratory and sometimes the pilot plant. Because of CFD's wide range of validity, STAR-CD predictions successfully bridge the gap between laboratory- and industrial-scale reactors.

Numerical models of the mixing reactors are created using the MiXpert meshing tool. Figure 1 shows a typical reactor grid with a pitched curved blade impeller. Predictions are made using a rotating frame of reference or a sliding mesh method, depending whether these are inside the reactor or not. Figure 2 shows the velocity vectors and the local shear rates in the fluid on two vertical cuts through the vessel. The dynamic behaviour inside the reactor can be visualised through the motion of a sufficient number of massless particles (Figure 3), which are distributed randomly inside the reactor. Trajectories for these particles were predicted in both time and space. With this method, the data used for scaling up can be recorded. In this example, the shear rate for each particle at every time step, the average over a period of time and the maximum shear rate is recorded over the timespan of the simulation.

Using this information, a typical shear rate frequency distribution histogram can be determined which characterises the process under consideration. The shape of this histogram is used as a measure for scaling up the process in such a way that the distribution remains comparable for the laboratory, pilot plant and production plant scales. A comparison of the power consumption of the measured and predicted plant scale reactors gives information on how to formulate the scale up rule. This means finding a mathematical relation between the relevant physical properties for, for example, constant tip speed or constant power consumption.

To conclude, we have demonstrated that CFD is an essential tool in the search for useful mixing vessel scale up rules from laboratory via pilot plant to production plant.

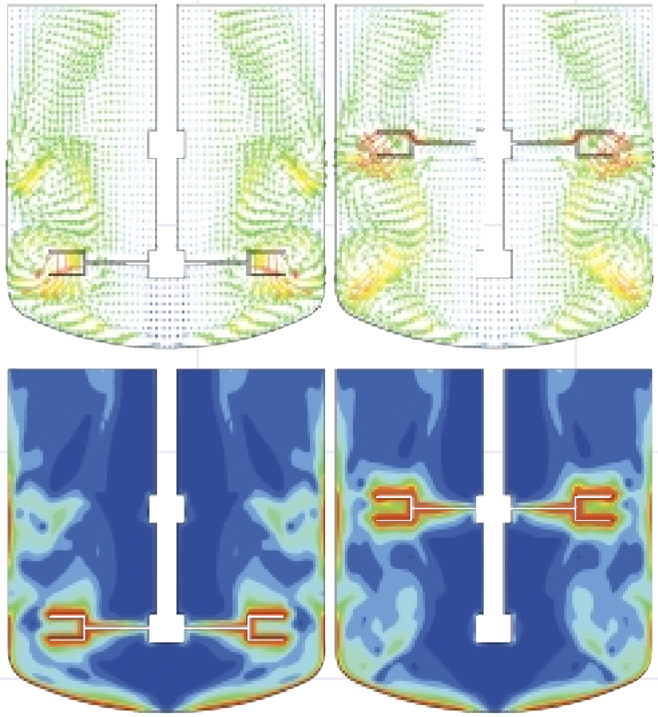


Figure 2

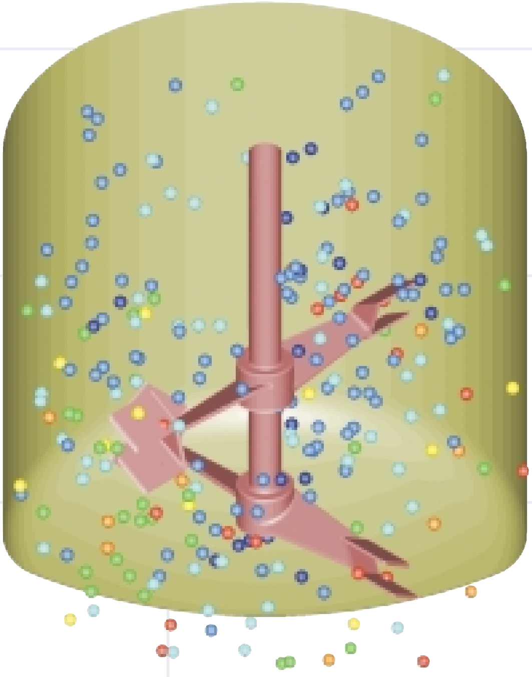


Figure 3

Mr Abouri and Dr Parry, SCHLUMBERGER Industries

# Optimising flow meter design

## with CFD and mechanical analysis

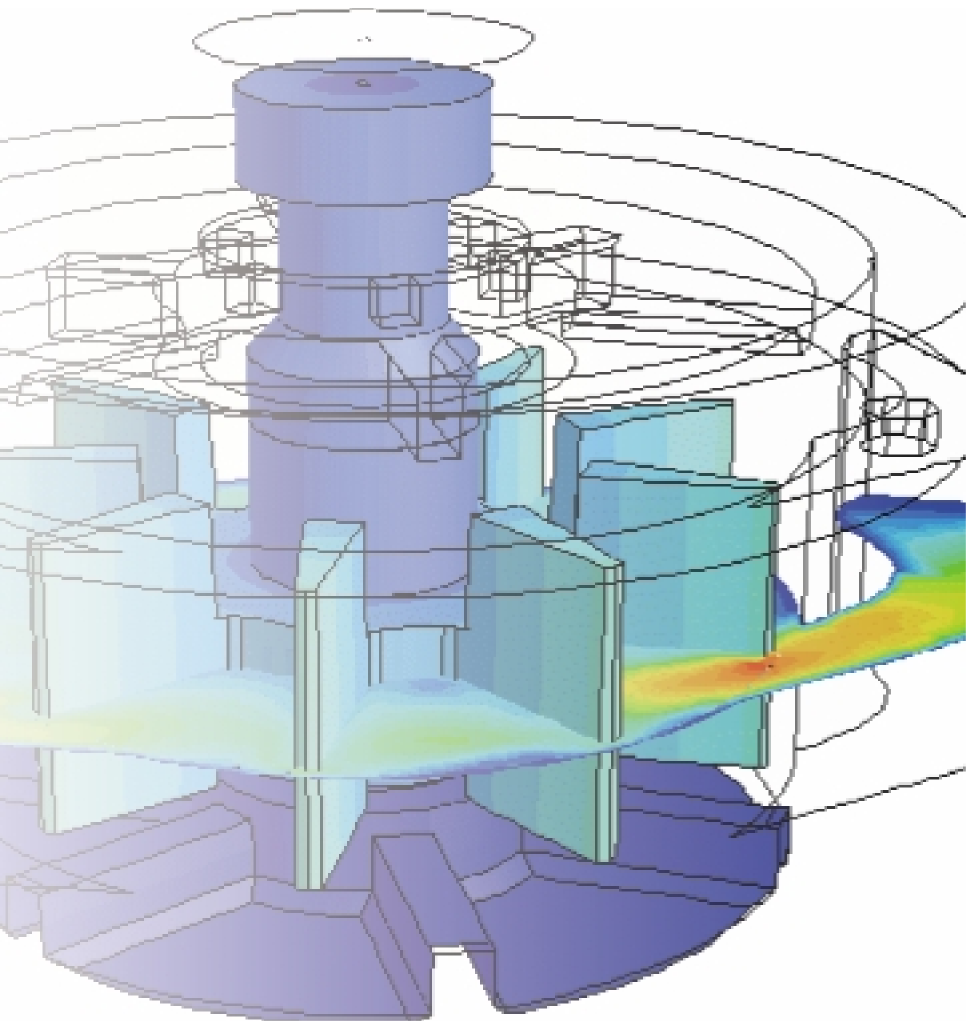
STAR-CD has recently been used by Schlumberger Industries to perform calculations that optimise flow meter design.

Fluid structure interactions are intrinsic to the operation of a wide range of turbine and positive-displacement flow meters. Computational methods enable dynamic interactions between the moving parts of such meters to be calculated. STAR-CD user-coding was used to introduce rigid body dynamics calculations linking pressure to displacement. Two methods can be used, one involving an explicit (forward marching) integration of the equations of motion of the mechanism and the other an implicit time differencing for the integration. Advances in mesh technology including deforming meshes with non-conforming sliding interfaces, open up this new field of application for coupled fluid/structure analysis.

CFD and mechanical analysis were previously applied to several flow meter types for the measurement of fluid flow in closed conduits. In this recent work, the analysis was applied to the study of transient fluid structure interaction for measuring elements and flow control devices with moving parts. The principles developed were general and can be extended to include small or large internal deformations of the moving components. The specific applications considered were a turbine meter and an oscillating piston meter of the positive displacement type.

Comparison of predictions with experimental data for the limiting cases of highly viscous flow and non-viscous flow confirmed the correct functioning of the algorithm. The explicit variant of the algorithm was used for the turbine flow meter. When treating "stiff" systems, numerical difficulties with explicit methods can be expected and the implicit approach is recommended. A successful implicit implementation was demonstrated for the case of piston movement in a tube. This necessitated several repetitions of each step with new values of mesh displacement and boundary conditions. This approach was applied to the oscillating piston flow meter.

STAR-CD was successfully used in both applications. Other work has also shown it to be an effective tool for many other applications including polymer flows, stirred mixing vessels, porous media flow including filters, chemical reactors and heat and mass transfer.



*Turbine flow meter: Velocity magnitude in a plane*

CHEMICAL

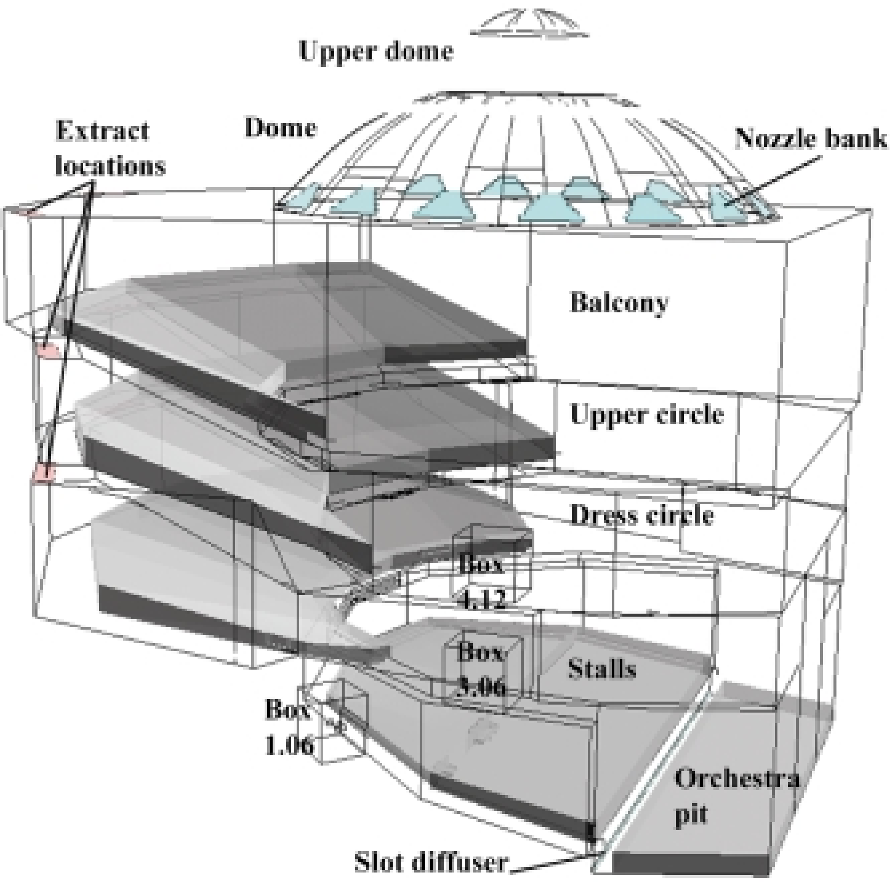
Erin Hatfield, Freelance Writer

# A breath of fresh air

## for the London Coliseum

LONDON, March 29, 2001 – During its near 100-year history, the London Coliseum has played host to many forms of entertainment, including horse races, musicals, varieties, cinema and finally the English National Opera. But while the Coliseum continually reinvented itself as a venue, its infrastructure fell behind the times.






Coming to the rescue of the venerable institution is Arup, a London-based engineering firm. Arup is using STAR-CD and high-end visualization to help ensure that modern-day opera buffs no longer need to suffer for their art. The company's master plan for the Coliseum will improve the standards of support facilities, provide better disabled access and egress, and upgrade the appeal of the building.

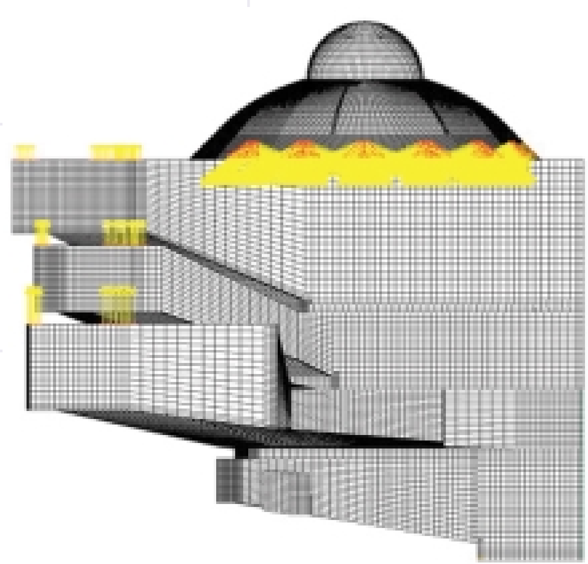
A major component of Arup's work is improving air circulation within the auditorium. Minor adjustments have already been made to improve the existing system, and entirely new ventilation will be installed during the English National Opera's off-season over the next three years.

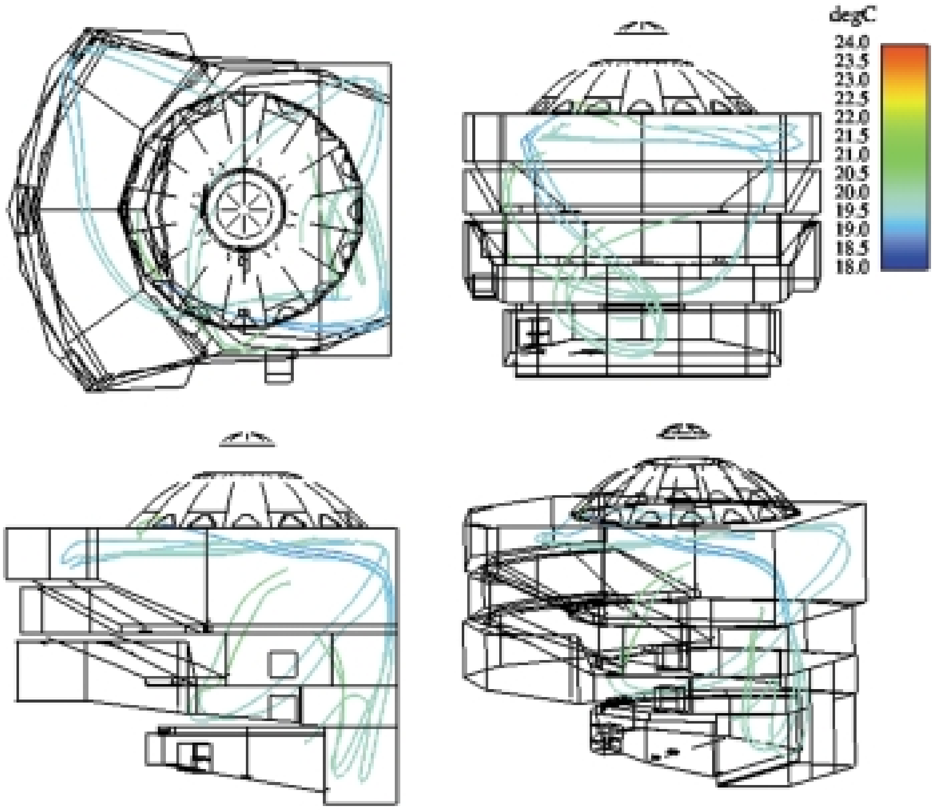
### From Seats to Ceiling

The London Coliseum was completed in late 1904. The original plenum-style ventilation system was reversed in 1932, with air supplied at the seating levels and extracted at the ceiling. The current air handling units and horizontal ductwork distributing air to the risers were installed 50 years later in 1982. In 2000, the system was brought back to full working order at a lower air volume and the addition of cooling supply air.

Arup's new design is a direct inverse of the 1932 system. Air is supplied via nozzle banks in the domed ceiling. This creates a swirling airflow within the auditorium that doesn't affect the quality of acoustics. Air is extracted at the back of the seating regions, creating proper ventilation of the entire space.

Arup's CFD study took approximately five weeks to complete. STAR-CD was used for CFD testing and EnSight from CEI (Apex, N.C., USA) was used for 3D visualisation. STAR-CD solved the momentum, mass and energy equations needed to predict the detailed temperature distribution and air movement within the auditorium.






## Making the Case for Better Air

Arup used three CFD case scenarios to simulate flow and temperature fields. Results from STAR-CD were loaded into EnSight, where animated particle trace paths were used to visualize air flow within the auditorium.

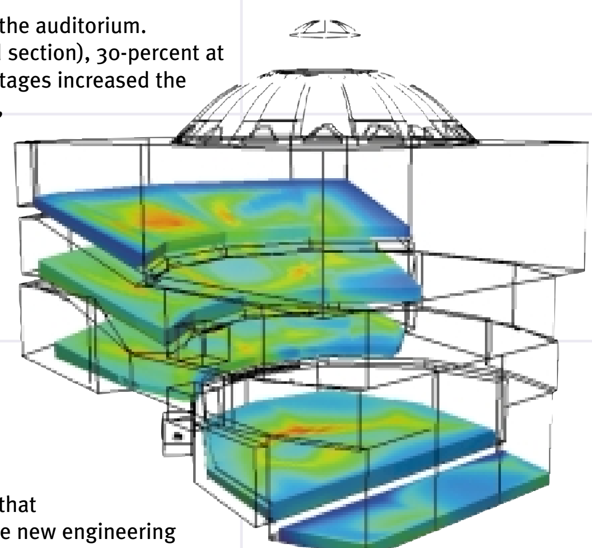
"The air movement is very complex," says Darren Woolf, fluid dynamicist at Arup. "It's driven by jet momentum, air temperature differentials (buoyancy), and wall-to-air temperature differentials that vary spatially and in magnitude. We needed to understand these factors and their various influences in order to improve the design of the system. We also needed to communicate our results. EnSight enabled us to create live demonstration materials and animations that convey the information in a way that's easy to understand."

In all three cases, air from the nozzles was set at 17ºC (63ºF). A convective heat load equivalent to 35W per person for the total 2,364 person occupancy was applied, giving engineers a feel for the atmosphere of the Coliseum under peak conditions. Lighting and other stage effects were not taken into consideration, as a neutral boundary was assumed between the stage and audience areas. Separate considerations were made for the orchestra pit as well.

## Cooler Heads Prevail

In the first case, air was extracted in equal volumes, 33.3% from each of the three sections. CFD testing revealed that airflow in the balcony area was "short-circuited"; that is, the air produced by the nozzles was extracted before it had a chance to effectively cool the occupants. Likewise, air in the dress circle centre stalls and seating areas was not drawn off forcefully enough to cool occupants, resulting in stagnant airflow and higher temperatures.

Using these findings, engineers varied the amount of air being drawn from each section of the auditorium. The second case extracted 50-percent of the air volume at dress circle (the most-populated section), 30-percent at upper circle, and 20-percent in the balcony. Particle trace analysis showed the new percentages increased the scope of the swirling airflow, allowing it to reach the dress circle occupants. Unfortunately, the increased percentage over-compensated for the shallow reach of the air in the first case, and the flow was drawn too low. Temperatures were more evenly spread throughout the audience, however, eliminating the cold and hot zones seen in the first case.



Extraction volumes were further adjusted in case 3, with 40-percent going to the dress circle, 33-percent to the upper circle, and 27-percent to the balcony. Airflow paths were again traced within EnSight. Engineers saw that the swirling air penetrated lower in the seating area, resulting in cooler temperatures in the most densely populated areas. This scenario was determined to be the optimum choice for the ventilation system.

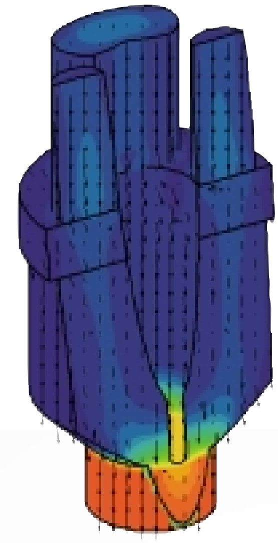
## The Final Act

STAR-CD and EnSight have helped flow engineers at Arup design a new ventilation system that harmonizes with the London Coliseum's acoustics and historical character. Thanks to these new engineering technologies, opera enthusiasts can enjoy their favourite arias in comfort, as the anguish takes place on the stage, not in the seats.

Massimo Masi, Marco Antonello and Giampaolo Navarro, Dept. Mech. Eng., University of Padova

# STAR-CD helps to make better Italian pasta

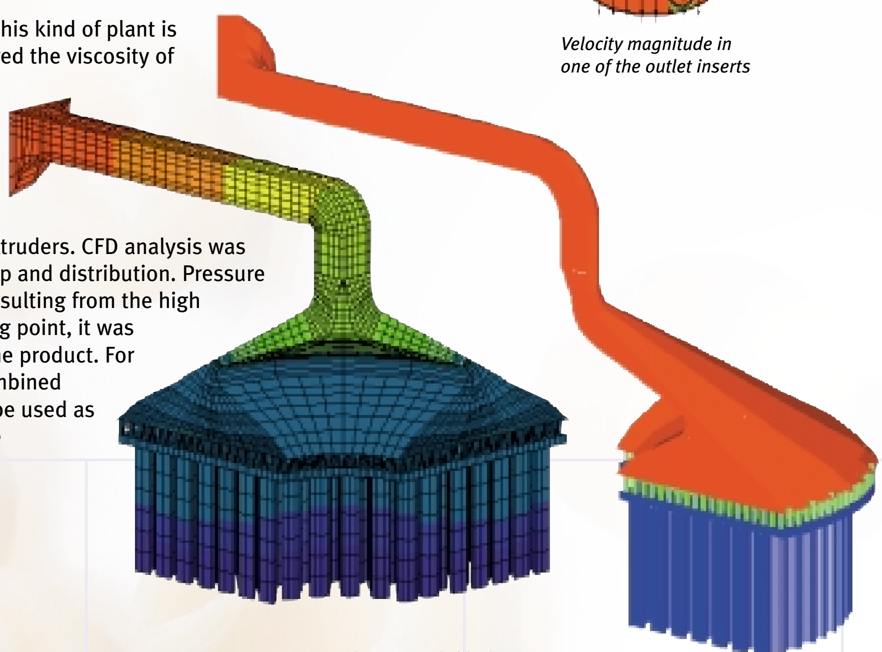
Nowadays, there is a strong demand to adequately model non-Newtonian fluids. Examples of the many applications of these fluids range from the industrial production of plastics or silicones, to the blood circulation in our veins. In response to industrial needs, the fluid dynamics department at the University of Padova, Italy have been involved in modelling non-Newtonian flows and their constitutive laws.



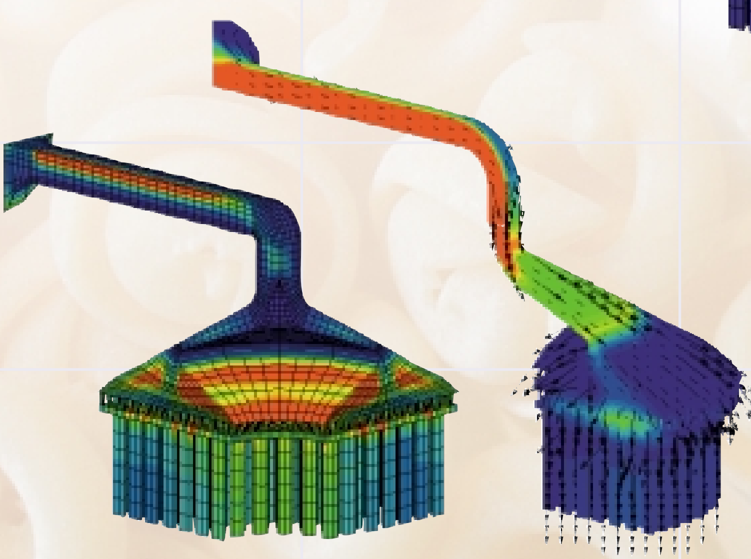
*Velocity magnitude in one of the outlet inserts*

One well known example of an application in the food industry is Italian "pasta". During its production it behaves like a non-Newtonian fluid. Two types of complex machine are involved in its production. The first one moulds the material in a controlled atmosphere. The second, which consists of an extruder coupled with a drawbench, releases the pasta while allowing its constituents to continue to give it nourishing properties. The machine must also ensure that the pasta is correctly distributed between the outlet inserts with minimal energy consumption. These outlets determine the final shape of the pasta, which we know better as spaghetti, tortellini, farfalle, penne etc. It is therefore in these last stages of the production, where the high variability of the fluid molecular properties is required, where benefits can be provided by CFD analysis.

The Italian industry leader in the design and construction for this kind of plant is Pavan S.p.A. Using special instrumentation, engineers measured the viscosity of the fluid constituting the food compound as a function of temperature and velocity gradients. Researchers in the department, who normally use STAR-CD for other applications, then tried out its capability to simulate non-Newtonian flows governed by different constitutive equations. These were implemented in STAR-CD by user coding and were tested for current shapes of moulders and extruders. CFD analysis was seen to be useful for example, for optimising the pressure drop and distribution. Pressure reduction contributes to a reduction in energy consumption resulting from the high viscosity of the fluid. Using the calculated velocity as a starting point, it was possible to modify the geometry to obtain better quality for the product. For these particular fluids, it is also interesting to observe the combined effects of geometry and viscosity. CFD analysis can therefore be used as an extremely effective tool in designing new equipment where performance depends critically on the flow field.



*Geometry and pressure field of a "pasta" moulder-extruder - the drawbench is coloured in blue.*



*Molecular viscosity and velocity magnitude fields*

Norbert Bulten, Lips Jets BV

# Interpretation of model scale test results

## with the aid of CFD calculations

A water jet is used as a propulsion system for high-speed vessels and is modelled with STAR-CD. The main component of a water jet installation is a mixed-flow pump, which includes a stator bowl and a nozzle.

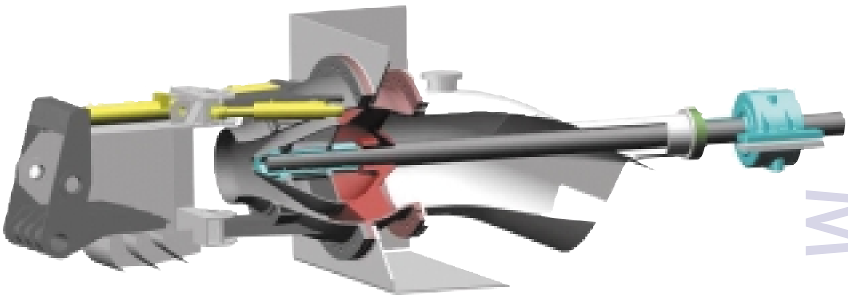


*Figure 1: 3D view of waterjet propulsion system*

The water is supplied through an inlet duct. An example of a water jet assembly is shown in figure 1. The flow through the inlet duct can be characterised by the Inlet Velocity Ratio (IVR), defined as ratio of the ship speed and the average axial velocity across the pump's inlet. It should be obvious that the latter is a function of the mass flow, and therefore related to the applied power of a water jet.

The flow around the cutwater of the inlet is strongly determined by the IVR value. Figure 2 shows examples of the dividing streamline at three IVR values. If the cutwater design is not suitable for the service operating conditions, cavitation could occur at the lip section. A lot of research has been carried out to optimise lip shapes, both experimentally as well as numerically. Experimental determination of cavitation inception is done in a cavitation tunnel, with an inlet model mounted on top of it. Inception pressure is found by continuous reduction of the tunnel pressure until cavities are observed. The difference between an experiment and the full-scale application is not only the size but also the presence of tunnel walls. It is to be expected that the flow patterns will change due to the confinement effect of these walls.

In this article, results of experiments have been compared with two series of calculations. The first series were carried out with boundary conditions representing tunnel walls,and the second with free inflow boundaries. Results of cavitation inception pressure as a function of IVR are shown in figure 3. Calculations with the wall boundary conditions show very good agreement with experimental results. This demonstrates the satisfying accuracy of the CFD model. The second series of calculations show the signifcant effect of the tunnel wall confinement. It is found that the allowable ship speed, where cavitation free operation is still possible, is overestimated by experiments with several knots.

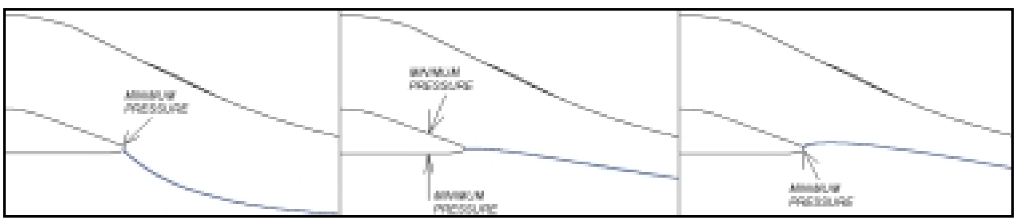



*Figure 2: Diving streamline at three IVR values; low, medium and high ( from left to right)*

Experimental test rig constrains, like tunnel walls at finite distance, can have significant influence on the flow pattern. CFD calculations can be used effectively to quantify this influence, which results in improved flow predictions of full-scale conditions.

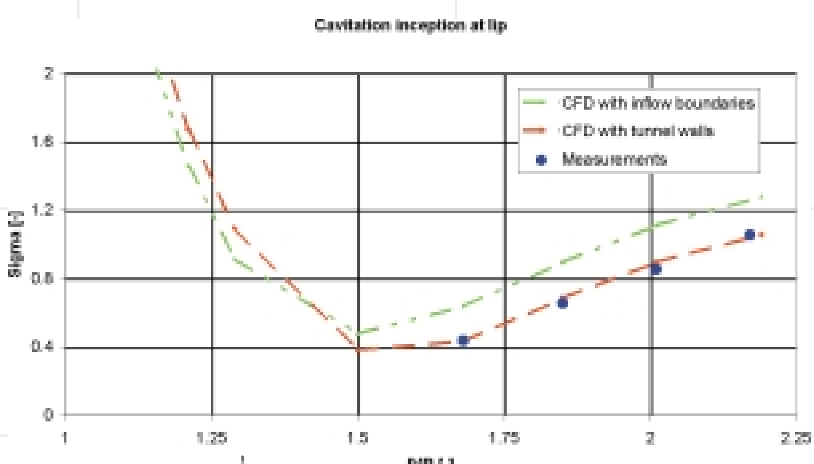


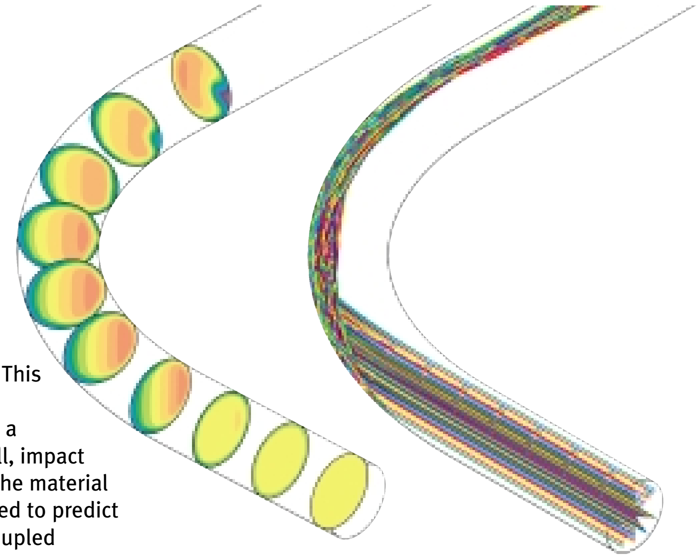

*Figure 3: Cavitation inception diagram at lip*

Malcolm Wallace, STAR-CD development team

# Erosion modelling with STAR-CD

## improving component lifetime and system safety

One of the problems encountered in oil and gas production systems is the damage caused by sand particles entrained within the production fluids. Sand particles impact on component surfaces and remove material by various erosion mechanisms. This has obvious implications for component lifetime and system safety.

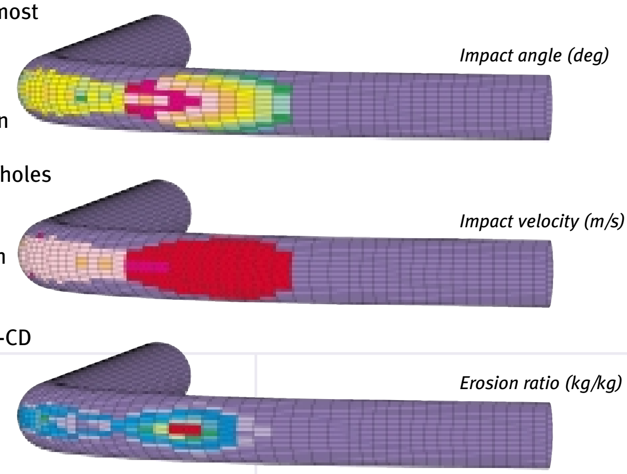


*Step 1: Obtain fluid flow*
*Step 2: Track particles through flow*

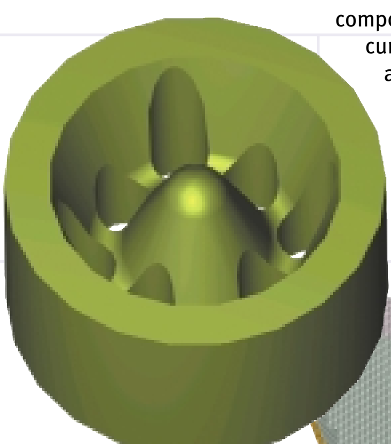
It is possible to use CFD methods to predict the areas of a component that will sustain most damage from impinging particles. Steps 1-3 illustrate the general procedure. Step 1 is to obtain the flow field through the component of interest. This flow field is frozen, and a representative number of particle trajectories are calculated through it (Step 2). This could amount to over 10,000 trajectories for a statistically meaningful prediction. Whenever a particle impinges on a solid wall, impact data are extracted and a semi-empirical erosion equation is applied to predict the material loss resulting from the impact (Step 3). These results can then be post-processed to predict the mass loss for a component. Erosion modelling can also be performed for coupled solutions where the particle loading is high enough to affect the flow.

A component developed by Wood Group Pressure Control Ltd. for use in the outlet of a control valve has been modelled using STAR-CD, and an erosion equation applied to predict the most likely areas of erosion. Figure 2(a) shows the original component and Figure 2(b) a section of the computational mesh. Erosion equations developed at the National Engineering Laboratory (NEL) were used to relate particle impact velocity and angle to erosion rate. Figure 2(c) shows the predicted average particle impact velocities on the surface of the component. When this is translated into mass loss, it becomes apparent that erosion is only expected to take place in specific regions close to the holes (Figure 2(d)). This behaviour corresponds to experimental findings.

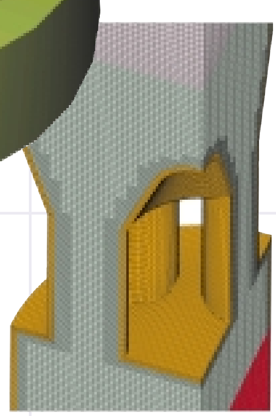
The benefits of CFD-based erosion modelling are that component manufacturers can predict the areas of highest erosion rate, and take steps to either reinforce those areas with a suitable material, or else re-design the component to affect the particle trajectories. STAR-CD currently has a beta erosion-modelling feature available on request.



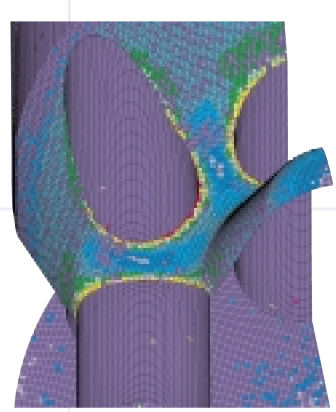

*Step 3: Extract particle impact data and relate to erosion rate*



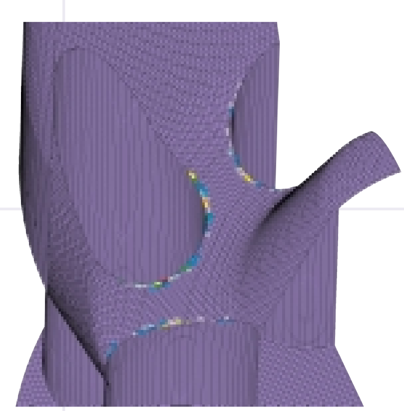
*2a) Original component*



*2b) Computational mesh*



*2c) Particle impact velocity (m/s)*



*2d) Predicted mass loss (mg)*

*Figure 2. Application to valve component*

Dr. F. McKenty*, L. Gravel*, M. Mifuji**

# Optimisation of tangentially-fired industrial boilers

*BMA - Brais, Malouin and Associates Inc, Montréal, Canada, **Cerrey S.A. de C.V., Mexico

Tangentially-fired boilers were originally developed to burn pulverized coal within a confined area by means of an intense fireball in the middle of the furnace. The fireball is created by firing the burners tangentially to a central target circle.

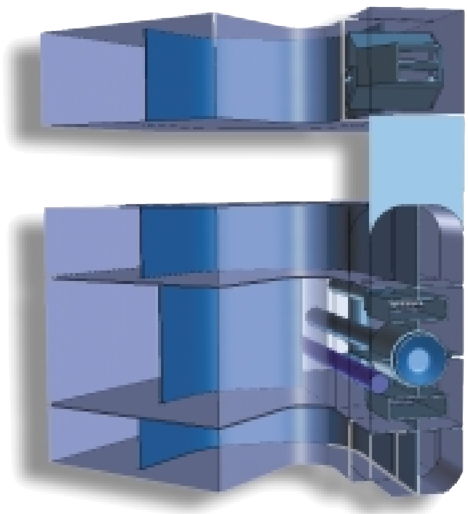


*Figure 1: Tangentially Fired Burner*

There are four burners per level with the number of levels (1-3) depending on total power input, which for this type of boiler ranges from 350 to 1000 million BTU/hr. Since its inception, this technology has also been adapted to burn heavy oil and natural gas. However, several problems such as flame stability, flame impingement, uneven wall heat transfer and pollutant formation have always created operational difficulties.

Due to the impossibility of carrying out any kind of experimental measurements inside the furnace area of an operating boiler, past optimisation of this technology has mainly been by trial and error with limited success. In an effort to eliminate these problems and improve overall boiler performance, Cerrey S.A. de C.V., a major industrial boiler manufacturer based in Monterrey Mexico, has initiated a CFD R&D program to investigate the complex combustion aerodynamics at play in this type of boiler.

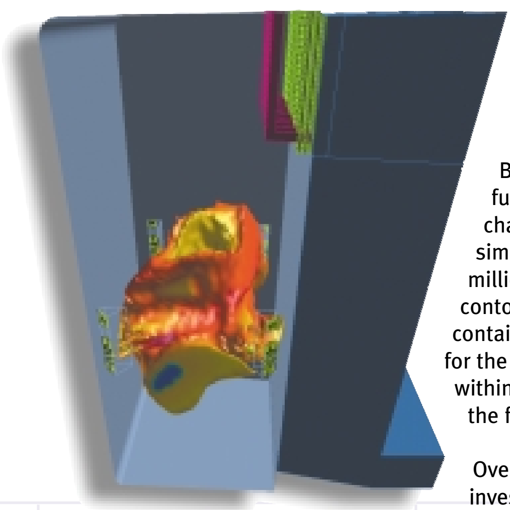


*Figure 2: Luminous Flame Contour - Original Design*

BMA carried out this study in two parts: (1) a detailed burner simulation and (2) a complete full-scale boiler simulation. This was done in order to capture the complex flow pattern and mixing characteristics at the burner outlet and to provide inlet boundary conditions for the full-scale boiler simulations. One of the tangentially fired burner geometries is shown in figure 1. Approximately 3.5 million computational cells were used for the burner study. Figure 2 shows the luminous flame contour of the natural gas flame for the original boiler design where the computational mesh contained some 4 million cells. The streamlines in Figure 3 show which of the burners are responsible for the flame impingement problem. The STAR-CD predictions for furnace outlet temperature were within 2% of measured field values, and the predicted flame impingement locations corresponded with the furnace wall surfaces covered with carbon deposits.

Over the past 6 months, BMA has investigated 9 different firing configurations for Cerrey S.A. de C.V. Figure 4 shows the fireball for an optimised firing configuration, which eliminated the flame impingement problem and lowered peak wall metal temperatures by 40 oF. Stretching of the fireball in the vertical direction has also enhanced furnace heat transfer by a factor of 15%.

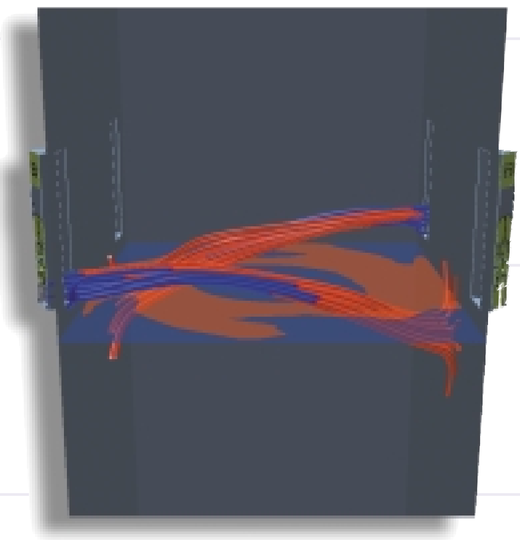


*Figure 3: Flame impingement Streamlines*

Predictions using STAR-CD have allowed Cerrey and BMA to gain greater insight into the complex flow patterns at play inside these boilers and to economically devise solutions to problems that have plagued the boiler industry for the past 50 years.

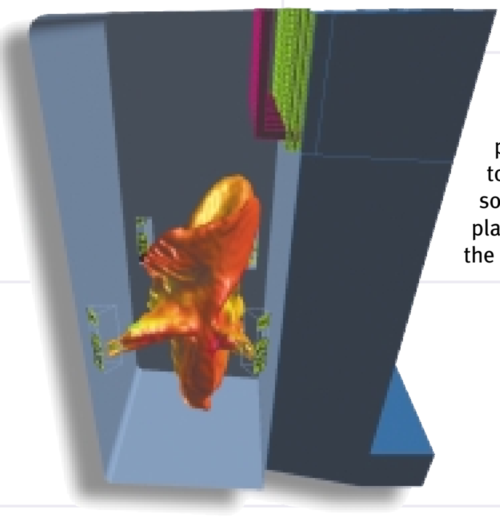


*Figure 4: Luminous Flame Contour - Optimised Design*

POWER GENERATION

# Dr. Mesh

## Programming PROSTAR: or getting the machine to do the boring bits!

In the last article, we saw how the command line worked, but it soon became apparent that commands could get quite cumbersome for repetitive tasks. This is when you need to start to teach PROSTAR a few new tricks.

While reading this article, the user is encouraged to look at the online help and printed manuals for further information. Try out the commands and experiment with variations to learn more.

As a thought exercise, let us create vertices numbered 1 to 10 with an X value the same as the vertex number, the Y value equal to X squared, and Z equal to zero. We do this using commands in a sequence like below:

```
*define noexecute
v iv iv iv * iv
*end
*set iv 0 1
*loop 1 10
```

Let us look at what this does. I use the `*define` command with the `noexecute` parameter, this means that the operations I am doing do not get executed until I loop over them. In the definition we use parameter `iv` which is set before the loop is executed but this can be after the definition. The loop is simple, it creates a vertex numbered `iv` with X=`iv` and Y=`iv*iv`. The loop (or function definition) is completed with the *end command.

The value of `iv` comes from the `*set` command we met last time but this time we see an extra parameter; we set it with an initial value and an increment. PROSTAR works with numbered loops. The number is set by the parameters to the `*loop` command: `*loop 1 10` will execute loops 1 to 10, `*loop -2 2` loops –2 to 2, `*loop 4` loop 4 on its own, *loop 6 4 loop 6 on its own.

The value of a variable is it initial value plus the loop number times its increment, so if a variable has initial value 3 and increment 5 then in the fourth loop it will be set to 3 + 4*5 = 23. If a parameter has an increment of zero (or not set), then its value is constant for all values in a loop. I often use a parameter (such as `iv` above) with initial value zero and increment one so it has the same value as the loop counter.

This simple idea can be extended as far as you want to, but all we have shown so far is how to push out a whole load of data. What we often need to do is to interact with existing data inside the model. We modify our thought exercise to create new vertices which are offset by 100 from the vertex set and have a Y value equal to twice that of the original.

```
*define noexecute
*get iv vset i
*get yold y iv
vgen 2 100 iv,,,0 yold 0
*end
*set i 0 1
*get nv nvset
*loop 1 nv
```

Here we use the `*get` command three times. At the start we use it to get the ith vertex of the vertex set. This means that as we loop, variable `iv` takes the value of each vertex in the vertex set in turn. The second time, we use `*get` to get the y value of this vertex. We then create the new vertex using the `vgen` command (this avoids having to get the x and z values as well). I also use a `*get` to set the number of loops to the number of vertices in the vertex set.

Essential parts of a scripting (or programming) language are conditionals and branches; PROSTAR has the `*if` and `*goto` commands for these purposes. We show their use in this simple example that operates on all the vertices in the vertex set and repeatedly subtracts one from their Y value until this is less than or equal to one.

```
*define noexecute
*get iv vset i
:PT1
*get yval y iv
*if yval gt 1
vgen 2 0 iv,,,0 -1 0
*goto PT1
*endif
*end
*set i 0 1
*get nv nvset
*loop 1 nv
```

We end with a few limitations and a useful trick. The limitations are to do with nesting `*if` blocks can be nested 10 deep but there can only be one `*loop` as PROSTAR works with a single nameless counter. The trick to overcome this is by using the `*calc` command to work out the indices. This is shown by making the vertices of a 10 by 10 patch, but of course the idea can be extended to more than two levels of nesting.

```
*define noexecute
*calc ix,,mod i,,11
*calc iy,,int i  / 11
*set x ix * 0.1
*set y iy * 0.1
v i + 1 x y 0
*end
*set i 0 1
*loop 0 11 * 11 - 1
```

# Technical Papers

Three new publications are now available from the CD adapco Group. To receive printed copies please send an email request to info@cd.co.uk, stating your preferred brochure and contact details, or alternatively download them as Adobe PDF's from www.cd.co.uk/products/brochuredownload.htm

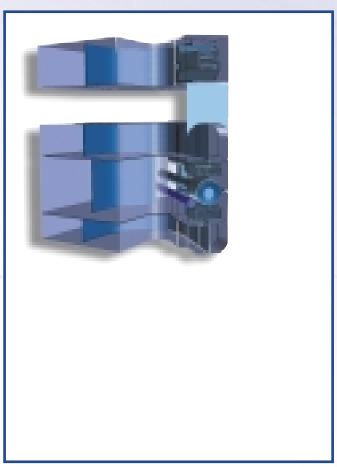

## Fluid-Structure Interaction with EZ-FSI

**Robert Brewster and Peter S MacDonald, adapco, New York**

EZ-FSI is a new productivity tool from the CD adapco Group for analysing fluid-structure interaction problems with STAR-CD. It is suitable for the majority of engineering situations where displacements are linear. It is available to STAR-CD users under an additional licence.

EZ-FSI coupled with STAR-CD provides a general purpose coupled interface between CFD and FEA structural analysis. It functions as follows: Firstly, an FEA code such as ANSYS is used to perform an analysis to characterise the structure by its mass, stiffness and damping matrices. EZ-FSI then works with STAR-CD to import these matrices and calculate the deflections using a multiple degrees-of-freedom solver. Finally, but only if required, the stresses can be computed via a subsequent run of the FEA code.

The advantage of indirect coupling of this type is that communication overheads between the codes are avoided. Moreover, since the EZ-FSI solver is fast, negligible CPU time is added compared with a conventional STAR-CD analysis. The benefit to users is that practical fluid-structure engineering solutions can be achieved with no more computer resource than is required for a typical CFD run. The technique has been validated on a wide and disparate range of applications, including operation of pressure-actuated valves, flutter of turbomachinery fan blades, sloshing in fuel tanks and flow-induced vibration of tube bundles.



## CFD as a Part of a CAE driven Development Process - Experiences from the Automotive Industry

**Walter Bauer, DaimlerChrysler AG, Stuttgart, Germany**

This paper describes how CFD is today embedded in the vehicle development process at the Mercedes-Benz passenger car development department. Through examination of several CAE applications including Underhood Flow Analysis / Vehicle Cooling, Passenger Compartment Flow / Air Conditioning /Thermal Comfort, Brake Cooling / Underhood Component Temperature Analysis and External Flow / Aerodynamics, it illustrates how different the specific requirements are satisfied through optimal use of CFD.


## Advanced CFD for Marine Technologies

**Prof. Dr. Milovan Peric, ICCM**

The aim of this review paper is to present recent applications of COMET for solving problems in maritime technology. Many of the results were obtained at the Technical University of Hamburg-Harburg from diploma projects and PhD studies. Illustrations of flows around ships, sloshing, slamming, submerged hydrofoil and floating bodies are presented. The applications by (or for) industrial users are still confidential, but it is hoped that the range of academic applications adequately illustrate the current possibilities and future trends in applying CFD in this field.

# STAR-CD

CFD joined up with Engineering

## Global Training Dates

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

## Group Exhibitions

European and North American
exhibitions are listed at:
www.cd.co.uk/company/events.htm

## Group News

For the latest news stories from the
CD adapco Group please visit:
www.cd.co.uk/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.co.uk/news/newsletter2.htm

## New Brochures

The latest STAR-CD and related product
brochures including COMET marine and
STAR-CD turbomachinery applications
can be downloaded in pdf format from:
www.cd.co.uk/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.co.uk

**adapco**

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

**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-italy.com

**Turkey**

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

**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

**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.com
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

**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