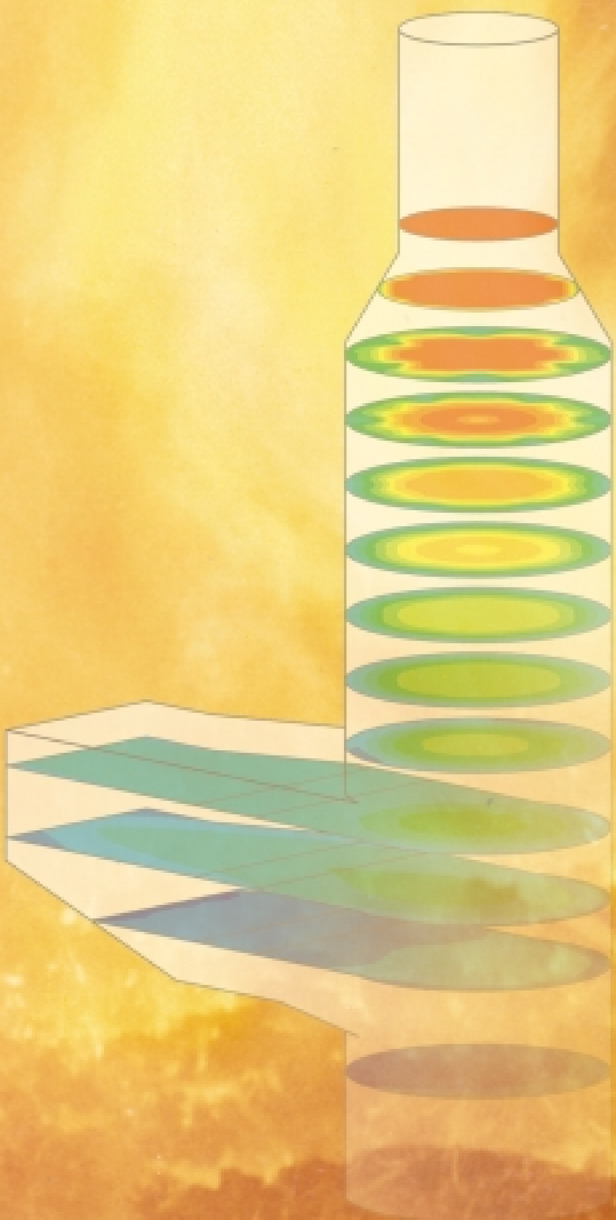
ISSUE 14 SPRING 2001

STAR-CD DYNAMICS

The Newsletter of the CD adapco Group



STAR-CD used to Optimise Waste Incinerators and Flue Gas Scrubbers 7-8

CD adapco Group

HEARTFELT GREETINGS
TO ALL STAR-CD USERS

If you look carefully at our new STAR-CD logo, you will see that we have formally recognized that STAR-CD is a product of the "CD adapco Group". This reflects a new commitment between CD and adapco to work together and with our international partners as a global enterprise on all aspects of CFD, from software development to CFD training, support, consultancy and problem-specific methodology and tool development.

The STAR-CD development team, co-ordinated from CD in London, is being broadened geographically by recruiting the best developers globally, and new teams are being set up at adapco and CDNA in the USA , and at various locations in continental Europe. The goal is to boost the rate of development of STAR-CD, to extend its capabilities in multiphysics simulation and to further integrate it into the CAE design environment. This is an exciting time for us, as the unification of the Group will give us the resources required to work not only on further STAR3 developments, but also on our new CFD codes, designed to serve you in the future.

Another new theme for the CD adapco Group is the development of specialized tools to deliver application-specific methodology, for pre-processing, meshing and post-processing, captured in new products such as adapco's EZAero, EZUhood, EZBart, EZIce and EZBrake. The concept here is to "package" methodology and associated solution capability for the most important industrial CFD applications.

The CFD consultancy side of the group's global business is also being expanded. The aim here is to provide better backup to users of STAR-CD, whether their needs are to get "up and running" quickly or to outsource their CFD problems. We also have plans to further develop our CFD training services.

These changes in the Group fulfil my personal ambition of unifying the activities of CD and adapco. We aim to make the CD adapco Group truly the global leader in CFD, a position that we are already not far from achieving, and to expand in other CAE areas.

As CD's new General Manager, I look forward to working with my fellow Directors, David Gosman and Chingzou Hsu, and with our colleagues throughout the Group to deliver the CFD code and services you need for successful flow analysis and optimization in your industry. You see, although we are running a commercial enterprise, our hearts are in CFD!

Steve MacDonald
General Manager

CONTENTS

BIGGEST UGM EVER

The 9th STAR-CD European User Group Meeting held at London's Millennium Centre was our (or rather "your"!) best attended ever, with more than 200 STAR-CD users, 12 companies exhibiting hardware and software, and representatives from all offices of the global CD adapco Group.

At the heart of the meeting was a series of user presentations covering a very wide range of automotive, marine, power generation and aerospace applications. Here are some examples:

Adam Opel's fuel cell analysis demonstrated that CFD has an important part to play in the coming of age of this alternative propulsion technology.

Two-phase flows and sprays featured in a presentation by Innogy, the leading UK electricity utility.

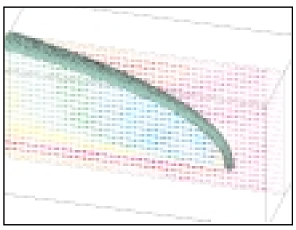
Matra BAe described how they had put STAR-CD through its paces in physical modelling for high speed flow in a ramjet, while Sauer-Danfoss described a smaller scale engineering case in which they customised STAR-CD for valve design.

An interesting "lateral" paper from BOC described how STAR-CD can be used in low pressure molecular flow regimes by exploiting radiation-like particle trajectory modelling techniques.

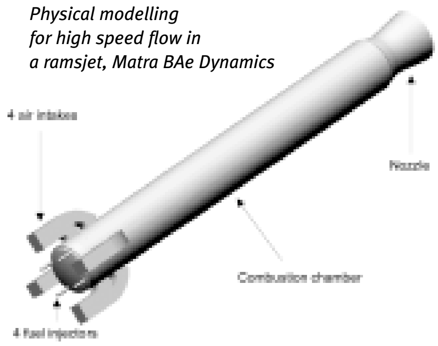
But perhaps the most joyful presentation was by Techno Trans who design ground-effect craft that fly low over the sea with incredibly high efficiency. The videos of scale model tests could not fail to appeal to those of us whose interest in engineering started with model boats and aircraft!

Further presentations by clients are described elsewhere in this newsletter.

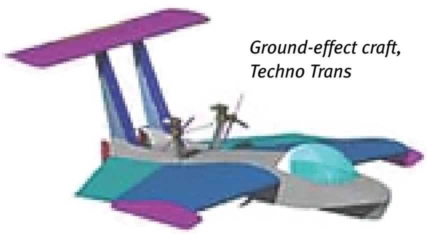
Informal discussions at the European Users Group Meeting



STAR-CD spray prediction for a plain orifice in cross-flow, Innogy



Physical modelling for high speed flow in a ramjet, Matra BAe Dynamics



Ground-effect craft, Techno Trans

USER GROUP MEETING

continued

As always, time was set aside for mixing and mingling between STAR-CD users and CD staff. Perhaps this "quality time" is one of the attractions that has led to more and more users coming each year to our European User Group Meeting.

The buffet-style catering worked well with the increased number of participants and encouraged informal discussions. A reception sponsored by COMPAQ, SGI, IBM and HP was the formal social highlight of the meeting. Informally though, we heard that the Bar Madrid nightclub was very busy later the same evening!

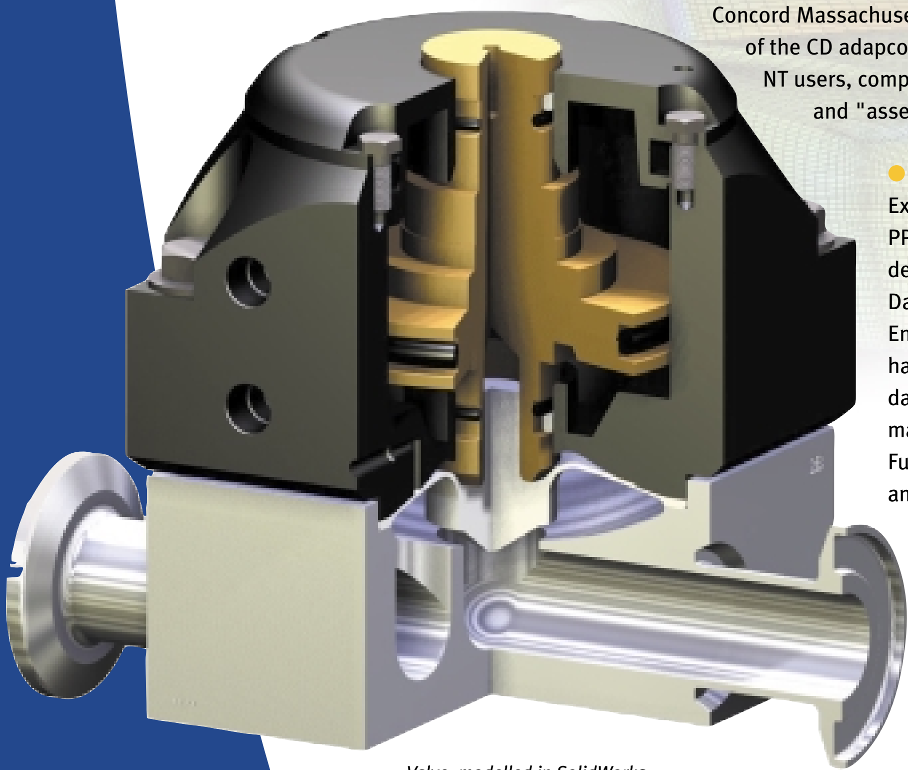
The next European Meeting will take place on the 19th and 20th November 2001, in the Millennium Conference Centre again. We hope to see you all there !

PRESS CUTTINGS

- Board level changes at CD. Following the departure of Dr Raad Issa from CD, the board of directors now combines in a balanced way the global interests of the CD adapco Group. Currently, the board of directors comprises: Professor David Gosman (a founding director of CD), Steve MacDonald (President of adapco) and Chingzou Hsu (Managing Director of CD and also President of CD-adapco Japan). Steve MacDonald also takes on the position of General Manager at CD.

- CD signs Distribution Agreement with SolidWorks. In the last Newsletter we described how CD had developed an interface between STAR-CD and SolidWorks, enabling this advanced solid modelling system to act as a geometry source for our pro*am automatic meshing tool. In the next step of our collaboration, we have signed a distribution agreement with SolidWorks of Concord Massachusetts that enables the global offices of the CD adapco Group to market a "bundle" for NT users, comprising STAR-CD with the "parts" and "assembly" components of SolidWorks.

- STAR-CD Property Database Extended. Using data supplied by PPDS2, a software package developed by the Physical Properties Data Service of the National Engineering Laboratory (NEL), we have extended the property database in STAR-CD to include many more commonly used fluids. Further information regarding PPDS2 and NEL's additional database services is available at www.ppds.co.uk



Valve, modelled in SolidWorks

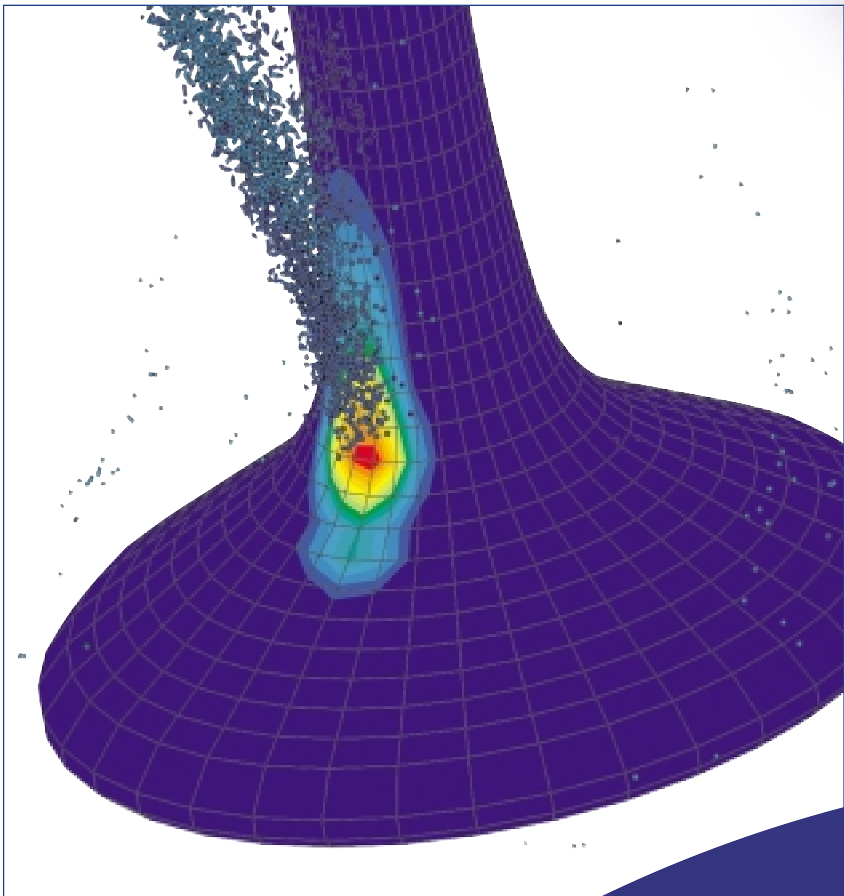
Wall Film Modelling in STAR-CD

Frank Lange and Cedomir Kralj, CD development team

Liquid wall films occur in many industrial processes and in our everyday experiences such as rain on car windows or aircraft wings. But perhaps the most challenging application for a CFD code is in engine design.

Liquid fuel wall films form in diesel and gasoline engines due to impingement of injected fuel sprays on pistons and intake valves and onto the surfaces of ports and cylinders. Once formed, the film develops dynamically under the influence of gas flow and wall movement. At the same time, heat exchange with walls and the surrounding gas leads to evaporation, affecting the composition of the mixture and hence increasing the complexity of the flow and combustion process. The presence of liquid fuel trapped on walls is, among other phenomena, blamed for increased soot formation and unburnt hydrocarbon emission, especially under cold start conditions.

CFD simulation can help engineers understand and optimise these highly complex processes, providing a powerful tool for improving efficiency and reducing pollutant emission in internal combustion engines.



From a CFD viewpoint, the challenge is to model an already complex set of processes and phenomena, with the additional complication of needing to handle different cell types in the dynamically changing geometry and mesh topology of an engine simulation.

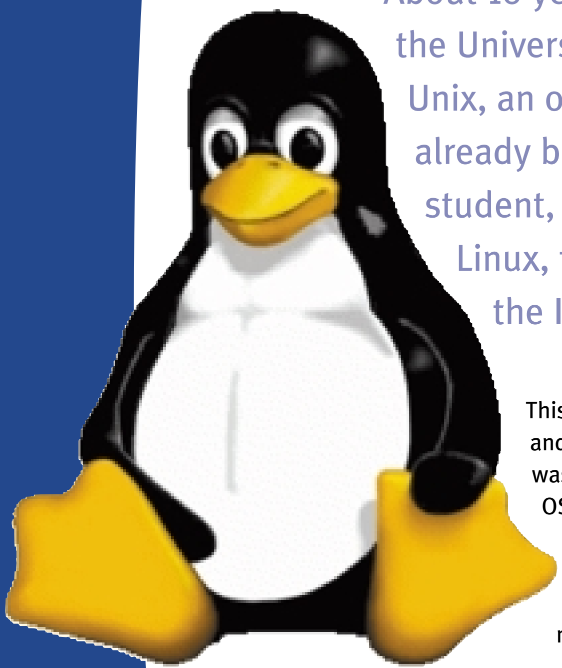
The film model implemented in STAR is based on work carried out by Professor Gosman's research group at Imperial College. It is linked with the existing Bai splashing models which predict the behaviour of droplets hitting walls. These models simulate droplets as they "bounce", "stick" or "splash" and their contribution to a wall film as well as the dynamic development of the film itself. The functionality of this new capability of STAR-CD is as follows:

The model allows for convection in the film, mass transfer with the gas phase, as well as momentum and heat transfer with the walls and gas. These transfer processes are modelled with standard wall functions. The coupling between film and gas is realised through source terms and is completely embedded in the STAR-CD solution procedure. This is accomplished in both the sequential and parallel mode of operation. All the relevant film quantities, such as temperature, thickness, mass and velocity are available for post-processing.

STAR-CD ON LINUX

The Origins of Linux

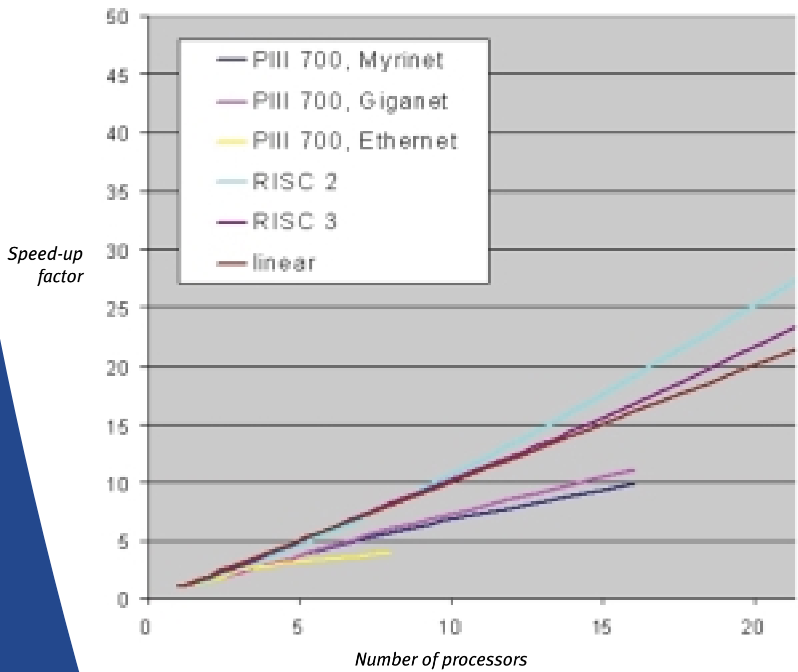
About 10 years ago a young Finnish student at the University of Helsinki created a clone of Unix, an operating system (OS) which had already been in use for over 20 years. The student, Linus Torvalds, called his new OS Linux, the first version of which exploited the Intel 386 chip.



This meant that Linux could be run on personal computers, rather than on the workstations and mainframes to which Unix was tied. In terms of functionality, Linux offered little that was new – it was, in effect, another flavour of Unix. What was revolutionary about the new OS, however, was that it was free: Linux was copyrighted under the terms of the GNU GPL (General Public Licence), written by the FSF (Free Software Foundation). The implication of this was that Linux, together with its source code, became freely available for distribution. Furthermore, anyone modifying the code was required to make that version freely available as well.

As recently as the late 1990s, Linux was still very much in the domain of university groups and enthusiasts; it is only in the last couple of years that it has been viewed as an acceptable alternative to the more established Microsoft and Unix OSs. This reluctance on the part of mainstream users to embrace the new OS was due, in large part, to a "lack of accountability" – if Linux was freely available, then no single person or organisation could provide support should things go wrong.

During the last few years an increasing number of companies have been offering their versions of Linux: companies such as Caldera, SuSE, Red Hat and Mandrake. These companies provide user-friendly installations as well as support – thus addressing the concerns mentioned above.



Case Data:
Passenger Compartment
1,237,000 fluid cells
Turbulent incompressible flow
scalar solver

Benchmark machines:
RISC multiprocessor server (RISC 2)
RISC multiprocessor server (RISC 3)

The advantages of Linux

The main advantage of Linux is that it allows more efficient use of Intel-based computer hardware. Although Linux runs on other architectures, it is the Intel machines which have suffered most from the hegemony of Microsoft. By exploiting Linux's ability as a true multi-user and multi-tasking OS, a PC gains the flexibility and stability of a Unix workstation, making it more suitable for running demanding applications such as engineering analysis software.

Linux is a more robust OS than Windows or NT and is very easy to learn, as it comes with high quality user interfaces and GUI driven utilities. The computers on which Linux runs tend to be significantly less expensive (sometimes by an order of magnitude) than conventional Unix workstations, giving users excellent price/performance.

STAR-CD on Linux

In the mid 1990s CD first ported **STAR-CD** to Linux. Although in those days the difficulties involved in installing Linux made it unsuitable as an OS for commercial applications, university groups were happy to try it out.

Since then however, the introduction of more user-friendly versions from the mainstream Linux providers has made the OS a viable alternative to the Unix machines so widely used for scientific computing. More recently, **STAR-HPC** was also ported to Linux, allowing users to run very large cases on relatively cheap clusters of PCs. During the past year or so, several **STAR-HPC** benchmarks have been run, on a variety of platforms, by the **CD-adapco Group** and by clients and hardware vendors. The results of these tests have been, for the most part, extremely promising.

The results of one particular benchmark, where a 1.2 million cell model was run on two different multiprocessor RISC machines are shown on the left, and the results compared with those from the same case run on a cluster of 700MHz Pentium 3 Linux machines. As could be expected, the type of network used for the cluster was a major factor in the performance of the Linux solution. However, it is still clear from the performance curves that the Pentium 3 machines, costing significantly less than the RISC machines, offer an extremely attractive alternative environment for running **STAR-CD**.

RESEARCH

Thermal Spray Process Simulation with STAR-CD

B.Liu, T. Zhang* and D.T.Gawne***

The Surface Engineering Group at Kingston University* and South Bank University** are carrying out research in collaboration with Computational Dynamics Ltd using STAR-CD with user coding to simulate the plasma spray process. The work is funded by the Engineering and Physical Sciences Research Council and has been very successful in predicting the critical characteristics of the jet profile and in-flight droplets. The research is in progress and is working towards a new design of equipment that will lead to improved properties for engineering products.

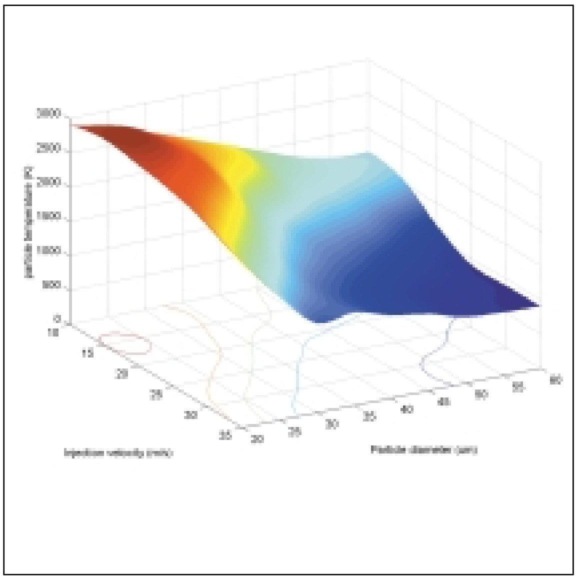


Figure 1: Particle temperature as a function of injection velocity and particle size

Thermal spraying operates by injecting solid particles into a high-temperature, high-velocity gas jet in which they are melted and subsequently projected onto a component to form a coating. The jet acts as a processing medium providing thermal and kinetic energy for heating and accelerating the feedstock particles. There are several types of thermal spraying including plasma spraying, high velocity oxy-fuel spraying and combustion flame spraying. A common feature of plasma jets is that they have a central core temperature of over 10,000 ^{0}C and in principle can melt any solid to produce coatings, provided its melting point is below the decomposition temperature. Plasma spraying is thus widely used to produce wear- or corrosion- resistant coatings for applications ranging from turbine blades in aeroengines to artificial hip replacements.

There are however some problems with the technology. Conventional plasma jets can suffer from high temperature and velocity gradients, resulting in variable coating properties. CFD simulation presents an opportunity to improve the thermal spray process by suggesting ways in which equipment and operations can be optimised. The goal is to achieve uniform high temperatures and spray velocities.

Optimising Waste Incinerators and Flue Gas Scrubbers

Andreas Lang and Dr Jörg Appel, NOELL KRC-ENERGIE-UND UMWELTTECHNIK GmbH, Germany

To ensure capital cost minimisation, process efficiency and environmental friendliness, the design of a modern incineration plant is these days usually carried out with the benefit of CFD analysis. CFD also has an important part to play in optimising the operation, improving the reliability of the process while at the same time reducing operating costs. Furthermore, possible upgrades to the plant can be evaluated using CFD, avoiding costly physical model testing.

WASTE INCINERATORS

Although consumer behaviour has led to a reduction in the amount of municipal waste, there is still an enormous quantity of material to be treated.

Waste incineration has an advantage over landfill sites in that most of the toxic chemicals are destroyed during the process. In addition to the reduction in chemical activity and volume of waste, the energy produced can be used for the generation of electrical power and heat.

Figure 1 shows the geometry of a typical waste incineration plant. Waste enters on the left-hand side and is transported into a water cooled grate system. The unburnt material leaves as slag at the right-hand side. Water and volatile materials evaporate and the latter ignites in the secondary air stream introduced via rows of nozzles. The additional wall-cooling and recirculation gas serves for mixing and cooling purposes. The gas moves along water-cooled walls and through tube-bundle heat exchangers until it passes on to the flue gas cleaning unit.

To ensure reliable functioning and availability of the plant, it is essential to avoid hot regions near the wall that might damage the refractory material. It is also necessary to ensure good mixing and a complete as possible burn up, which is achieved by optimising the geometry, air supply configuration and operational conditions.

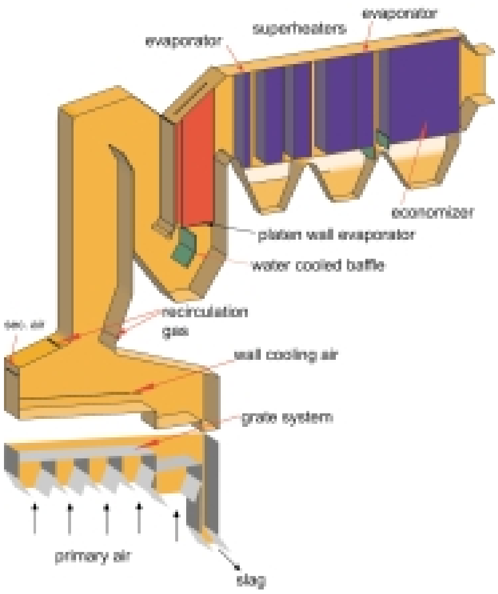
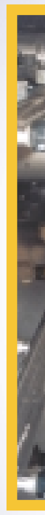


Fig 1: Waste incineration plant



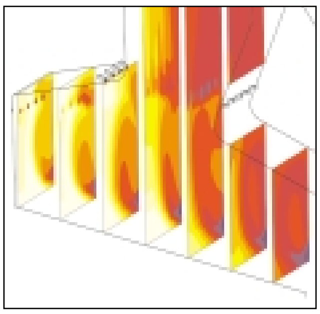


Figure 2.1: Temperature distribution in the plant

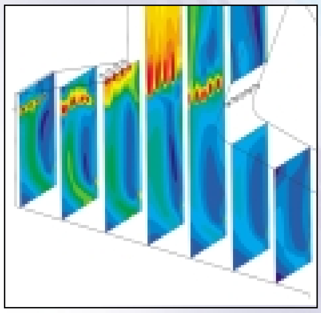


Figure 2.2: Velocity distribution in the plant

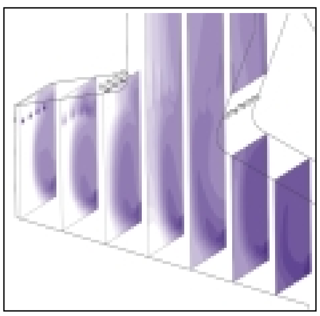


Figure 3.1: Oxygen concentration

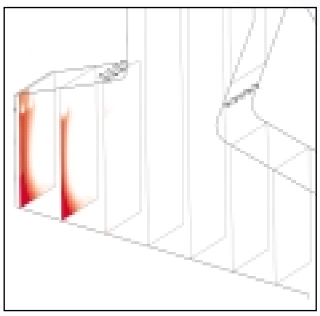


Figure 3.2: C_xH_y concentration

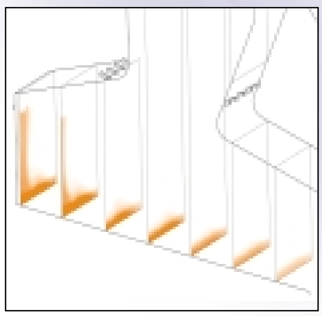


Figure 3.3: CO concentration

As a large number of parameters need to be optimised, care was taken by Noell in their STAR-CD analysis to adopt CFD calculation procedures which were both accurate and time saving. The behaviour of the solid material on the grate is treated with an in-house C++ code which describes the transport and conversion of the solid material. The output of this program serves as input for the gas-phase STAR-CD model in which an eddy break-up treatment with two reactions is employed. Material data is taken from the Chemkin database. Radiation as well as buoyancy effects are simulated. The effect of the tube bundle heat exchangers are modelled in user subroutines.

Figures 2 & 3 show results for half the plant, with the symmetry plane pointing towards the viewer. It can be seen that the wall cooling air induces a vortex that draws the volatiles away from the wall and into the centre of the plant where the highest temperatures occur.

"Comparisons with measured data and observations in existing plants are very encouraging, confirming that STAR-CD is capable of providing realistic simulations and valuable insight into the operations of these incinerators."

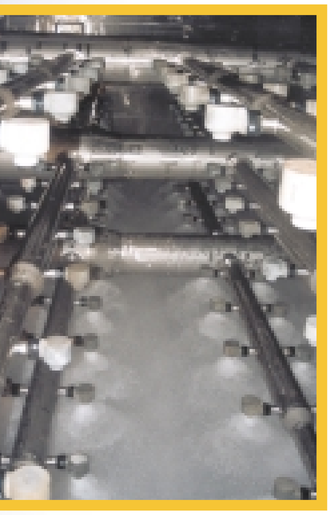


Figure 4: Nozzle arrangement in an SO2 scrubber

FLUE GAS SCRUBBERS

Flue gas scrubbers are an important part of the after-treatment facilities in coal or waste fired power plants. At Noell's plant the scrubbers consist of towers of 5 to 20m in diameter, in which limestone slurry is sprayed into the gas to capture harmful substances such as sulphur dioxide (SO2) and hydrochloric acid (HCl). Large scale SO2 scrubbers have more than 1000 nozzles which spray the gas with several tons of slurry per second. As environmental regulations have become more and more restrictive there is an increasing demand to optimise the design in order to improve cleaning efficiency while minimising the use of energy.

"In the past, we at Noell have carried out many isothermal two-phase Lagrangian calculations in order to achieve uniform holding times, taking full advantage of the available volume and avoiding the escape of untreated flue gas (Figure 5). For accurate modelling, the absorption of harmful substances into the evaporating drops must also be handled, requiring inclusion of inter-phase heat and mass transfer in the simulation. A model describing the separation was implemented in STAR-CD via user subroutines, and a droplet-wall model was introduced to describe the behaviour of droplets splashing into wall films. Figures 6 - 8 show the distributions of temperature, water concentration and HCl concentration in a parallel-flow single-loop HCl scrubber. The STAR-CD simulations showed a satisfactory agreement with our measurements in the plant."

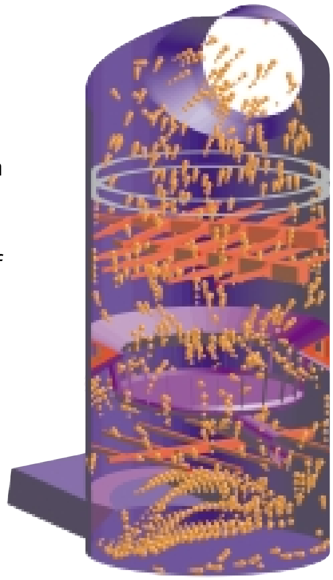


Figure 5: Double loop SO_2 scrubber

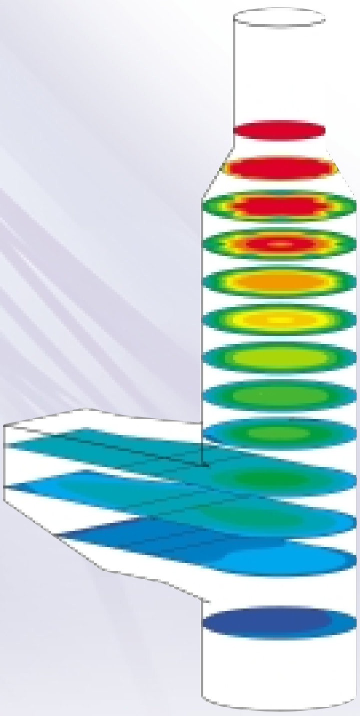


Figure 6: Temperature distribution in the scrubber

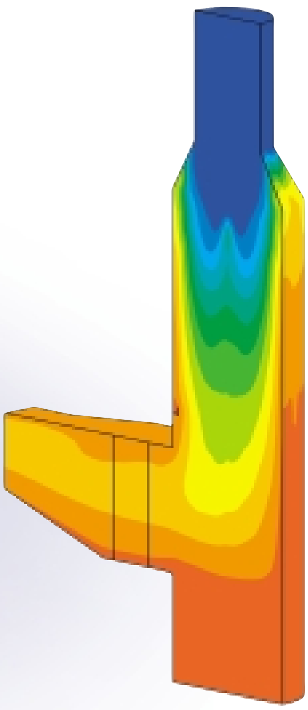


Figure 7: Water concentration in the scrubber

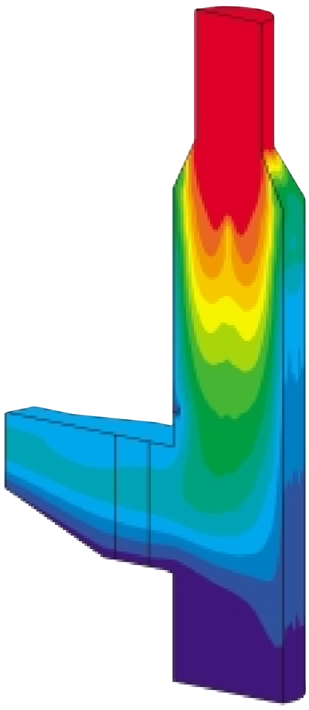


Fig 8: HCl concentration in the scrubber

Understanding how Ships Damage Waterways

Tobias Linke and Jens Scheffermann, University of Hannover, Germany

The passage of ships along inland waterways causes damage to beds and banks from induced sub-surface currents, waves in the wake and direct impingement of jets from the propeller. Very often the damage is such that expensive repair work is necessary.

Figure 1: Operating inland ship

Despite extensive physical testing carried out by the research community worldwide, the complex hydrodynamic forces involved are difficult to predict empirically. Coming to the rescue, numerical simulation using CFD is now being applied to understand, and predict and, we hope, ultimately minimize the types of hydrodynamic flow that cause most damage.

A number of researchers have already successfully applied CFD techniques to these problems but their calculations have mainly been confined to the ship's near field. Furthermore, the shape of the channel and its depth and width have generally been ignored.

In our research at the University of Hannover we have focused on far-field simulations of propeller jets, including the effect of the ship's rudder and taking into account the actual depth and width of the channel. To simulate the rotating jet flow in detail, we used STAR-CD's rotating mesh capability. In order to calibrate, verify and validate the simulations, physical tests were carried out with a propeller jet and in-stream rudder. Measurements were taken using an Acoustic Doppler Velocimeter (ADV).

The results of the numerical simulations show excellent qualitative and quantitative agreement with the physical model test measurements, verifying the suitability of STAR-CD for these applications. We are now carrying out further investigations using STAR-CD's moving-mesh and free-surface facilities. This work extends the simulation to include the effects of currents induced by the ship's motion and the formation of waves in the wake of the ship. The results are very promising and we feel that the knowledge gap in this area is closing so that engineers will be able to take action to better protect our waterways in the near future.

For more details please contact: Tobias.Linke@stud.uni-hannover.de or Jens.Scheffermann@stud.uni-hannover.de

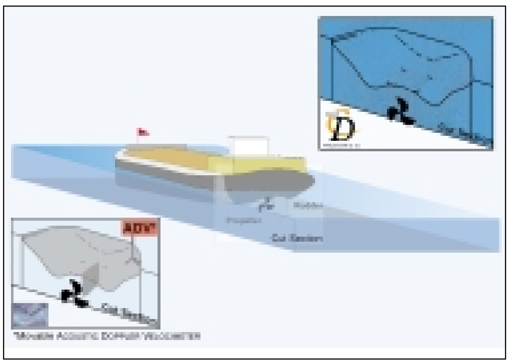


Figure 2: Transformation of prototype (Mid) into numerical model (Top) and physical model (Bottom)

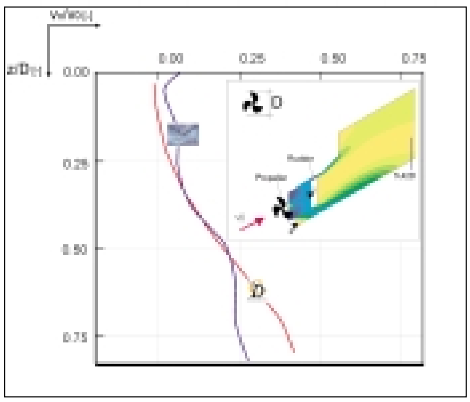


Figure 3: Comparison of current velocities resulting from numerical simulation and physical model test. Distance to propeller = 5,42 D, Rudder Angel = 45°, and Propeller revolution speed = 225 1/min

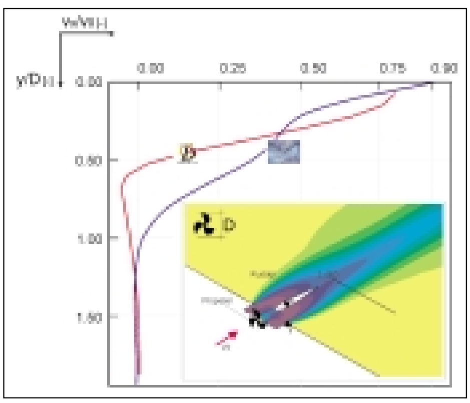


Figure 4: Comparison of current velocities from numerical simulation and physical model tests. Distance to Propeller = 3,48 D, Rudder Angel = 0°, and Propeller Revolution Speed = 150 1/min

CFD Modelling of Large Air Cooled Condenser Systems

Dr. M.P. van Staden and J. J. Janse van Rensburg, Aerotherm, South Africa.

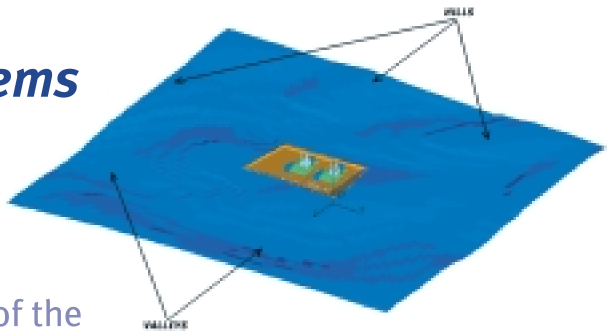


Figure 1: Surrounding Topology

The performance of Air Cooled Condensers (ACCs) used in the power generation industry is of critical importance to the reliability and availability of the power station as a whole. ACCs make use of ambient air as the cooling medium. Changes in the ambient conditions which have a direct impact on the rate of heat exchange by the ACC must therefore be handled appropriately to achieve an accurate CFD simulation.

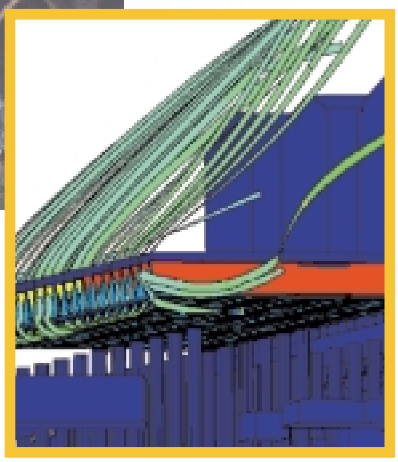


Figure 2: Correlation between actual and numerical flow fields

Aerotherm was contracted to model one of the world's largest ACCs. STAR-CD with additional user coding was applied to simulate five aspects of the process:

(i) Atmospheric modelling of the ambient conditions around the power station buildings and through the ACC; user coding was used to generate the required wind velocity profiles at the atmospheric boundaries.

(ii) A fan performance model was integrated into the STAR-CD code; this produces the necessary source terms for each fan as a function of the prevailing conditions above and below the fan.

(iii) A heat exchanger performance model which included simulation of wet spray cooling and different heat exchanger cores.

(iv) A post-processing subroutine which calculated the aggregate totals of mass flux, heat exchange, platform pressures, etc. for each fan unit and ACC unit.

(v) A turbine performance model which was used to predict the turbine's response resulting from severe change in ACC performance.

Factors such as hot air re-circulation, individual fan and heat exhanger performance and turbine back pressure are predicted by the model. In some instances, construction of ACC systems requires modification of the surrounding landscape, which in turn influences ACC performance (see Figure 1).

Comparative global performance data predicted by the CFD model correlated well with actual operating data as shown in Table 1. Flow visualisation (using smoke) was also compared with flow patterns from the CFD simulation with satisfactory qualitative results (see Figure 2).

	Back Pressure (kPa)		***Unit Load (MW)***	
	Predicted	Actual	Predicted	Actual
Unit 1	20.8	19.1	632	635
Unit 2	20.0	15.6	634	638

Table 1 : Predicted versus actual unit performance for a wind of approximately 6 m/s blowing from behind the power station buildings

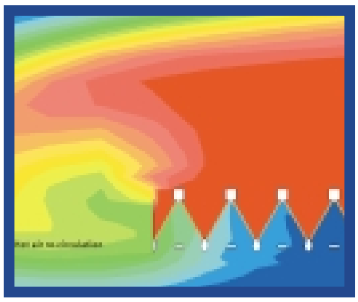


Figure 3: Temperature contour plot of the hot air that is re-circulating into the ACCs

"Due to the relative ease of user subroutine integration into the STAR-CD code, the stability of the solver and the robust manner in which arbitrary mesh interfacing can be applied, STAR-CD proved invaluable in conducting these simulations."

Predicting Combustion Behaviour of Coal Blends

Peter Stephenson, Innogy

Coals burnt in power stations often consist of a blend of various types of coals.The combustion behaviour of the blend is not always what might be expected from the properties of its components, and this has led to operational difficulties.

Innogy (formerly National Power) has been using STAR-CD to investigate the combustion of coal blends. It is essential that the coal burnout is as high as possible, both to maximise the cycle efficiency and to minimise emissions. The coal blend burnout is not necessarily a linear combination of that for the individual coals, so the combustion behaviour is difficult to predict.

As part of a recent collaborative project, funded by the UK Department of Industry, Innogy modelled the combustion of coal blends using STAR-CD. A new feature was specially developed for the project, enabling different chemical kinetic parameters to be specified for the various coal components instead of just assuming average kinetics. In this way, many of the complex interactions within the flame could be represented.

Innogy successfully used this new blending feature to model the behaviour of coals burnt in their Combustion Test Facility (CTF). It was demonstrated that it is possible to include the effects of blend components and valuable insight was gained into the behaviour of coals burnt in the CTF.

The prediction of coal combustion using CFD continues to be challenging and complex. With help from Computational Dynamics, Innogy is currently investigating ways to improve the physical models used in the calculations. The possiblity of reducing the computing requirements is also under investigation.

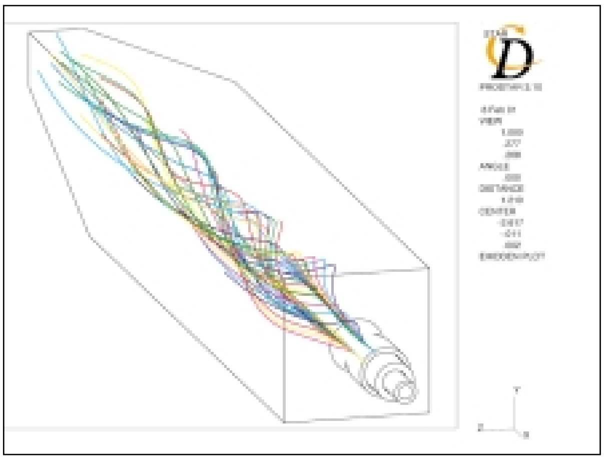


Figure 1: Coal particle trajectory

Figure 2: Combustion test facility

STAR-CD *at Volvo Car Corporation.*

Johan Larsson, Volvo Car Corporation

At Volvo Car Corporation, CFD has become an integral part of the engineering and design optimisation process; its role in the development of Volvo's final products is still growing.

To apply technology effectively, a team-based working approach is adopted, involving people from design, physical testing and CFD groups. This, together with a thorough understanding of the possibilities and limitations of CFD, has increased its impact on the development process.

Figure 1: Aerodynamic design to reduce dirt deposition on the rear windscreen

Today, CFD accounts for a large part of the fluid dynamical optimisation of vehicles. On several occasions, production tooling has been ordered based on CFD analysis alone.

STAR-CD is being used primarily for external aerodynamics, including drag prediction and dirt deposition studies.

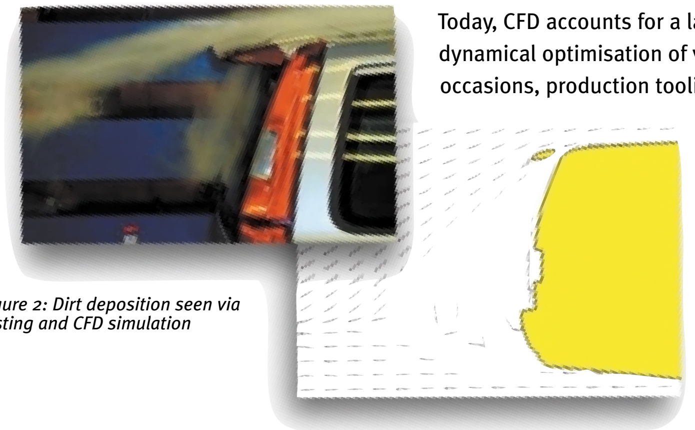


Figure 2: Dirt deposition seen via testing and CFD simulation

DMI Optimises Heat Exchanger Design

Danish Maritime Institute (DMI) with its Industrial Flow Consultancy Services has recently been involved in a project with APV Heat Exchanger where CFD was applied for optimisation of plate heat exchangers.

STAR-CD was used in a selection process to decide between a series of candidate designs for further experimental prototype testing. The CFD model successfully quantified heat exchange efficiency and pressure loss; fouling and product treatment characteristics were also quantified indirectly.

Moreover, the detailed prediction of the flow field gave a valuable insight into the relationship between the specific plate design and flow pattern – an insight that will be utilised in future design studies.

Further information: ***Henrik Hassing,*** *E-mail: nhh@danmar.dk*

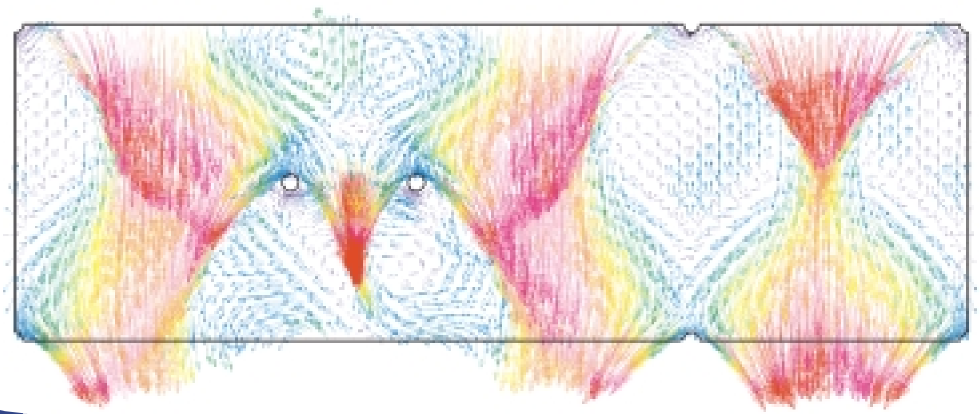


Figure 1: CFD analysis used for flow field optimisation in a plate heat exchanger

Prostar: The Power of the Command Line.

Dr. Mesh

I have a saying, "Do it once, do it again and then programme it." Often repeated tasks can be made much easier by programming them, and doing so leads to faster, more repeatable results.

Whilst Prostar has an extensive graphical user interface, it is backed by a comprehensive command line facility. This article introduces the command line and useful "one liners". Once the command line is mastered, the next step is writing scripts and then the door to customising and extending Prostar is open to you. On opening PROSTAR, two windows will appear: the graphical output window in which the model can be displayed and the GUI accessed, and the output window, which is split into 3 parts. The centre section of this window is where the user enters the command. The upper section is where the output from that command is displayed, and where the command prompts appears, together with any errors, warnings or other information associated with the execution of the command. Finally the lower section of this window contains a history of commands executed, together with a number, in the column to the left of the command. A single mouse click on this number will bring the corresponding command back to the middle section of this window, where it can be edited prior to re-execution. A double mouse click will re-execute the command.

So we have found the command line and know what to do with it, what are the commands we can enter? Prostar has many commands and they are all covered in the online command HELP so will not be covered here. They are grouped together by function, and similar functions have similar names e.g. when working with splines the commands start with SPL, (such as SPLCROSS, SPLDELETE and SPLUNDELETE). Prostar commands and alphabetic input are case insensitive and are abbreviated to their first four letters (with the exception of *ENDIF which is only abbreviated to five letters to avoid confusion with *END). Hence whether CPLOT, CPLOX, CPLO or CPLON is entered, the effect will stay the same: the program will plot cells. Apart from the exception mentioned above, the program will ignore characters after the fourth. Therefore, SPLUTTERED COUGH, CHOKE and DREADFUL WEATHER are valid PROSTAR commands - try them and you'll convince yourself. Any command starting with an exclamation mark will be treated as a comment and ignored. !WOW, did you see that?

Remembering that this article is aimed at eventually programming Prostar, one of the fundamental parts of any programming language is working with variables. Prostar allows assignment of data to named variables with the *SET command. This means that the following are valid:

```
*SET A 10
*SET B 11.2
*SET C A * B
V B A B C
```

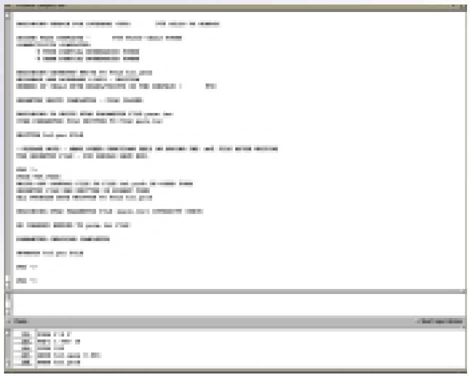
When working with these variables, Prostar forces the right type so the vertex number is assigned the integer value of B and the variable x, y and z are given the floating value of A, B and C. We get a vertex number 11 at location x=10., y=11.2 and z=112.

Now that we have variables, what about their manipulation? Prostar provides arithmetic manipulation on the command line. If I am creating a model which fills one eighth of a cylinder, I can easily create a vertex at 45° (in coordinate system 2, V 1 10 45), but if the model requires one thirteenth then it is cumbersome to type in 27.692307692307° so you can do this instead: V 1 10 360 / 13. Prostar evaluates strictly left to right so you need to take care: 10 + (4 * 5) needs to be written as 4 * 5 + 10. Parameters can be mixed with numbers so if you need to evaluate (4 * 5) + (6 * 7) then you can do it like this, *SET T1 4 * 5 so you have the first part and then use 6 * 7 + T1 where needed.

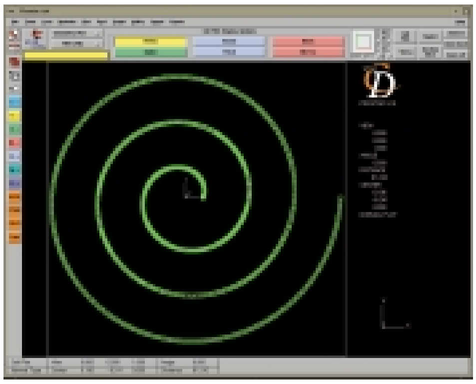
If the maths is more complex, Prostar provides the *CALC function which gives access to all the normal maths operators such as sine, cosine etc. So to set a variable called XVAL to the sine of 60° you use *SET XVAL 1 DSIN 60. (A power function is missing but schoolboy maths reveals that logs get you round this!)

Before we go much further we should look a little more closely at the format of the command line. Prostar works by reading free-format input. It splits this input into items which are separated by blank spaces or commas. There is a limit of 20 items per line so complex input needs to be evaluated piecewise. It then scans these inputs for arithmetic expressions and evaluates these to form the command input. Thus the line V 10 + 1 20 / 4 60 – 12 has 10 space-separated items (to show this we could also input the line as V,10,+,1,20,/,4,60,-,12) which form the command V 11 5 48. This creates a vertex number 11 with x=5 and y=48, and z takes the default value (which is 9) as it is left blank. But what if I want to use the default for y: if I just put more blanks in, Prostar will treat any sequence of blanks as a single blank so will behave the same way. In such cases, the comma is necessary to convey the exact meaning so V 10 + 1 20 / 4, ,60 – 12 creates vertex number 11 with x =5, and z=48 and y set to default. The blank between the commas is not necessary so experienced Prostar users use multiple commas as place holders in commands such as VPROJ VSET,,,0,CSET,,,LOCAL Z.

So far, it all seems to have been learning the fundamentals but where are the advantages? Lets look at one of the quiet heroes of the Prostar command set, RP. The RP command is for repetition, so if you want to repeat the same command but slightly differently, then this is for you. We know how Prostar works by sorting stuff into the right position in the command. Well, RP repeats the command a number of times adding (or subtracting) from the value in the position. An example illustrates this best: creating a spiral. I want to have a spiral of three turns, a base radius of 10 with a pitch of 25. The spiral is made up of vertices every degree. Here is how we do it (remember the lines beginning with a ! are a comment):



```
!set a variable to the base radius*set brad 10
!and the pitch
*set pitch 25
!cylindrical system
csys 2
!create the first vertex and then repeat for three turns with offsets
!the repeats add 1 to the vertex number, 25 / 360 to the pitch to
!give a full pitch of 25 in 360 vertices, and 1 degree to the vertex spacing
v 1 brad 0 0
rp1081 1 25 / 360 1
```



A few things to note:

1. I use the parameter `pitch` but Prostar only uses the first four letters. This is because it is clearer to work with `pitch`, but do not get confused, `radius_inner` is the same parameter as `radius_outer`.

2. The RP command's full name is actually RPnnnn, where nnnn is a variable number. This number can be from 2 to 9999: RP2 means do it once again as the count includes the original. We could have done the command above in its variable form:

```
*SET NUMB 360 * 3 + 1
RPNUMB 1 25 / 360
```

3. This could have been done in other ways: Prostar is a very rich language.

So you can work the command line a little now and you can see some benefits, but looking at the commands you seem to need to know the number of every vertex and cell before you can do anything useful. Is there a middle ground which mixes the power of the command line with some cursor interaction?

Prostar has certain keywords that allow this interaction between command line and cursor; for instance VX when typed into a command lets the user pick a vertex instead of typing its number. If you want to find the distance between two vertices on the screen you can type VDISTANCE VX VX and Prostar pops up the cross-hair cursor twice to allow you to make your picks. If you do this using GUI using Utility menu item Vertex Distance, then you will see that Prostar issues the same command internally. With the exception of a few trivial commands that contol the GUI, every Prostar operation is executed as a command so the user has access at this level to the full functionality of the code. Likewise, you can pick cells, splines, etc with CX and SX and thus go a level further with constructs such as CXT which gives the type of the selected cell. This allows you to say to Prostar, "remove all cells from the cell set with the same type as the one I pick" : CSET DELETE TYPE CXT. Again, the online manual lists all these in more detail.

4. By now, hopefully, you are convinced about the power but may be questioning the length of commands. This is addressed in two ways, the first being the writing scripts. This is covered in the next article, but for now we shall introduce abbreviations. Prostar allows commands to be bound to abbreviations as long as the abbreviation does not conflict with existing commands.

A simple and common use of abbreviations is when working with the cell set. Instead of writing CSET ADD TYPE 10 you make an abbreviation *ABBR CAT and assign CSET ADD TYPE to this abbreviation. The command now becomes CAT 10. Abbreviations are limited to a single line but Prostar accepets a $ (dollar) as a command separator so the following is allowed:
*ABBR HOUSE

```
CSET NEWS NAME ROOF WALLS FLOOR$VIEW 1,1,1$ZOOM OFF$CPLOT
```

European Training Courses

STAR-CD Basic Training Courses

UK

Course Dates	*Course ID*
15-17 May 2001	B160
12-14 June 2001	B161
10-12 July 2001	B162

Germany

Course Dates	*Course ID*
21-23 May 2001	BT0501
11-13 June 2001	BT0601
24-26 July 2001	BT0601

Advanced Training Courses pro*am

UK

Course Dates	*Course ID*
24-25 May 2001	P164
21-22 June 2001	P165
19-20 July 2001	P166

Germany

Course Dates	*Course ID*
30-31 May 2001	PA0301

Advanced Training Courses ICEM

UK

Course Dates	*Course ID*
21-22 May 2001	I180
18-19 June 2001	I181
16-17 July 2001	I182

Further training dates can be found at www.cd.co.uk/support/training

Training courses are provided globally through the CD-adapco partnership.

CD adapco Group

CD-adapco partnership

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

Agents

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
40 avenue des Terroirs de France
75611 PARIS
CEDEX 12
FRANCE
Tel: (+33) 1 56 95 45 60
Fax: (+33) 1 56 95 45 61
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

Sweden
CD Sweden
Odinsgatan 13
411 03 Göteborg
SWEDEN
Tel: (+46) 31 722 01 70
Fax: (+46) 31 722 01 71
info@cd-sweden.com
www.cd-sweden.com

Turkey
info(+)TRON A.S.
F.K.Gökay 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 Engine Co.
1 Whipple Street
Balcatta
Western Australia
AUSTRALIA 6021
Tel: (+61) 8 9441 2311
Fax: (+61) 8 9441 2345
pewing@orbeng.com.au
www.orbeng.com.au

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
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