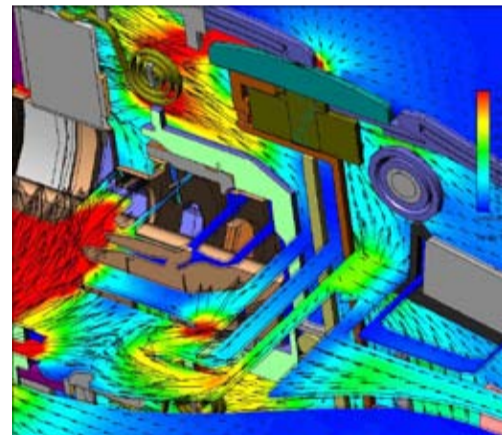


AEG use CAD Embedded Software 'Engineering Fluid Dynamics' to Create Cool Power Tools Faster...

Recognised as a member of the "Top 100 Most Innovative Companies in Germany's SME Sector," AEG Electric Tools GmbH is a market leading manufacturer and marketer of heavy duty portable electric power tools. AEG's product portfolio includes more than 100 different types of tools for the professional market including hammers, percussion and diamond drills, angle and straight grinders, jig and circular saws. The design team at AEG is responsible for solving motor cooling as well as air flow issues. "As power tools are getting more compact, we investigate motor cooling issues to better understand the effectiveness of the cooling method used" explained Markus Wörner, Design Engineer.

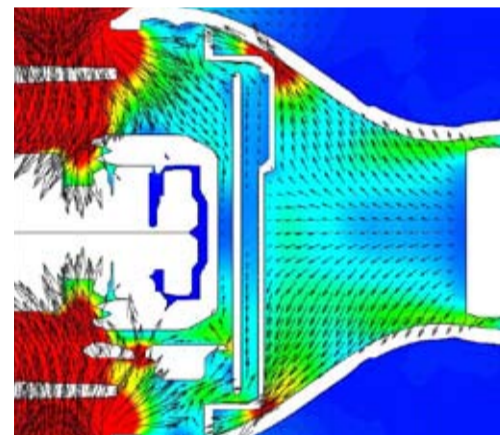


Digital prototype showing heat sources and air circulation

Power tools designed at the South German facilities are engineered for two different brands: AEG Power Tools and Milwaukee Electric Tools. The tools are based on a common platform which makes it possible to modify the machine components for the different needs of the users. The modular design enables AEG to realise two different exterior designs. "Different exterior design and internal components result in completely different airflow in



the machine. Testing each configuration would take a lot of time. But with simulation we can identify the effects caused by all the different design options and ensure proper performance for all machines based on the platform" said Peter Henske, CAD Manager.



Digital prototype image showing air circulation paths

To test design concepts created with PTC's Pro/ENGINEER Wildfire, the design team uses FloEFD for thermal and airflow effects. "On a recent project we used FloEFD early in the design process and we were able to get the engine running

20% cooler, and we obtained this improvement much faster than before" said Wörner. "We chose FloEFD because it is easy to use. For example, with FloEFD we do not need to define the fluid area - the software does this automatically." Most traditional fluid flow simulation programs require users to create 'phantom' parts to represent the (empty) fluid regions - an extremely time consuming process since users need to identify each region manually and then create geometry to fill it. FloEFD saves time and effort by automatically differentiating between solid and fluid regions for internal and external flows to create the fluid domain. Testing prototypes with FloEFD also enables the AEG design team to fine-tune new design concepts by using information that they have amassed over the years as the basic building block for their new design work. "By using FloEFD we are able to fine-tune our designs to reach an optimised design much faster. On a recent project, we reached our goal for improved airflow with the very first prototype" added Henske.

For more information visit: www.flomerics.com/info

News in Brief...

Mentor Graphics Corporation Acquires Flomerics Plc...

On the acquisition, Gary Carter - General Manager of the newly formed Mechanical Analysis Division of Mentor Graphics Corporation - formerly Flomerics Plc said "Flomerics' technology and products are complementary to Mentor's existing suite of electronic design automation software products and services. Mentor Graphics' executive management team recognise that Flomerics' customers are predominantly mechanical engineers and designers rather than electrical designers, and as such Mentor attaches great importance to the skills and experience of Flomerics' existing employees and management. Both Mentor's and Flomerics' management teams believe that with Mentor's global strength and greater financial resources, the success and potential of Flomerics' technology, technical expertise, and customer relationships can be better cultivated and significantly enhanced".

FloEFD is Fastest Growing CFD Tool with Half Year Sales Up a Massive 70%

The Mechanical Analysis Division of Mentor Graphics Corporation (formerly Flomerics Plc) has announced that sales of its FloEFD software increased by a staggering 70% for the first half of 2008 compared to the first half of 2007, and claims that FloEFD is the fastest-growing fluid dynamics simulation software in the world.

"These results clearly demonstrate that our strategy is working." said Gary Carter, General Manager of the Mechanical Analysis Division, formerly CEO of Flomerics. "A major advantage of our Engineering Fluid Dynamics software - unmatched by other CFD codes - is that it is fully embedded with the major Mechanical Computer Aided Design (MCAD) systems such as Pro/ENGINEER, SolidWorks and CATIA. This means that mechanical design engineers can now use fluid flow and heat transfer simulation to optimise their designs in the shortest possible time without leaving their familiar design environment."

New customers during the first half of 2008 included household names such as Bell Helicopter, Camfil, Eltek Energy, Goodrich, Husky, Lexmark International, Lockheed Martin, Philips Medical Systems, Robert Bosch, Siemens, Tatung, Ultra Electronics, Vestas Wind Systems and Zytex.

A - Z of Industries Using FloEFD within Engineering

- Aerospace/Defence
- Automotive/Transportation
- Biomedical/Medical systems
- Chemical/Processing
- Computers
- Construction/HVAC
- Consumer Goods
- Education
- Electronics
- Energy
- HVAC/Construction
- Lighting
- Marine
- Medical/Biomedical systems
- Oil & Gas
- Plastics
- Power generation
- Processing/Chemical
- Refrigeration
- Semiconductors
- Telecommunications
- Test & Measurement
- Transportation/Automotive
- Valves/Pumps/Pipes
- Water Management

In this edition...

Page 1

- AEG Use CAD Embedded FloEFD
- DIALIGHT Plc. Design New LED Lighting Systems

Page 2

- Bronswerk Develop Innovative Fans
- Ventrex Save 4 Months Work
- SAIPEM UK uses FloEFD with Autodesk Inventor
- Trox R&D Investing in CFD Tools

Page 3

- Optimizing Sail Design with FloEFD
- Power LED Overheating Solved
- Wind Load on RADAR Installations
- CFD Related 'White Papers'

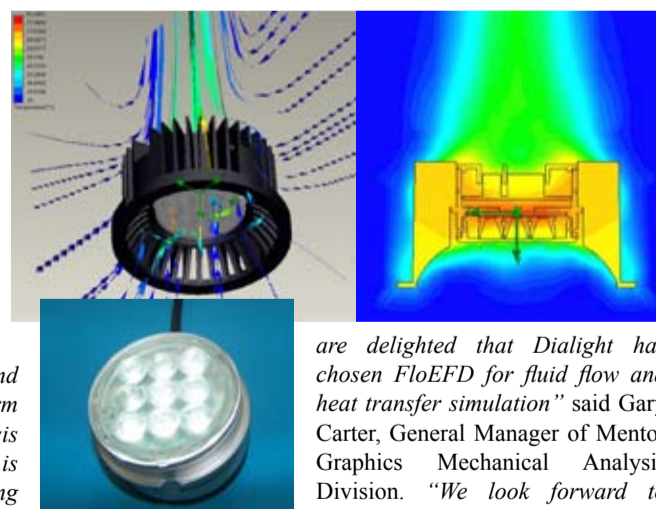
Page 4

- Multi-Processor Ability Added
- NEW Tools and Functionality in FloEFD v9 Available Now!
- The 7 Key Technologies of EFD

LED Manufacturer DIALIGHT PLC use FloEFD to Design Cutting Edge Lumerdrive Light Systems...

Dialight Plc selected Flomerics' FloEFD engineering fluid dynamics software, embedded in Pro/ENGINEER Wildfire, for the design of advanced lighting systems based on light-emitting diode(LED) technology. "Although power LEDs are becoming more efficient, a significant amount of input power translates into heat" said Gordon Routledge, VP of illumination products, Dialight plc. "Cooling of the electronics and LED devices is critical to long-term reliability and so thermal analysis - including airflow analysis - is vital to help us meet our demanding development schedules. We solicited presentations from a number of analysis vendors. Some systems were either too simple or required a degree in maths to produce any meaningful output. Unlike some of the lower cost options, FloEFD is a recognised leader with good support and the software handles all the thermal transfer mechanisms we require."

"We recognise the importance of the fast-growing market for LED lighting systems and



are delighted that Dialight has chosen FloEFD for fluid flow and heat transfer simulation" said Gary Carter, General Manager of Mentor Graphics Mechanical Analysis Division. "We look forward to supporting Dialight's design teams around the world." Dialight Plc is a world leader in applied LED technology, operating in two principal segments: Components, where low brightness LEDs are used for status indication in electronic equipment, and Signals/Illumination, utilising the latest high brightness LED technology for traffic and rail signals, obstruction lighting, hazardous location lighting and developing applications in more general illumination markets.

Bronswerk Develops Groundbreaking Innovative Fans that have a Massive Environmental Impact...

Bronswerk Heat Transfer BV (www.bronswerk.nl) uses FloEFD, an easy-to-use general purpose engineering fluid dynamics software from Mentor Graphics (formerly Flomerics Plc), to develop new air-cooled coolers and innovative fans with the potential to make an enormous impact in industries requiring a 'greener' solution.

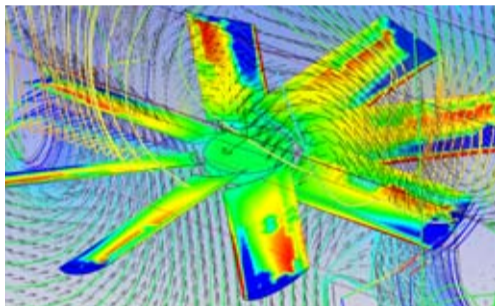
While traditionally such apparatus has a thermodynamic efficiency of up to 60%, their new designs based on the new fan achieve up to 80% efficiency. The potential energy savings worldwide can be staggering as their product can save 15% of the electricity it uses. These savings could result in 20% savings in generation capacity in Holland alone over the course of 10 years. "Engineering Fluid Dynamics (EFD), the foundation for FloEFD, is more practical than Computational Fluid Dynamics (CFD)" said Guus Bertels, Senior Engineer. "EFD is a natural extension of CFD that is more intuitive for mechanical engineers. The information generated with FloEFD especially for the taxing complexity of cooler aerodynamics is far beyond data obtainable through physical measurement and experimentation."



design needed to cope with a wide range of environmental (wind gusts and building interference), fluids and pressure conditions.

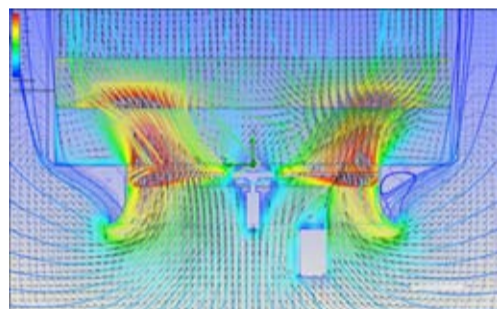
The engineering team at Bronswerk used a combination of Pro/ENGINEER Wildfire from PTC and FloEFD.Pro from Flomerics. Since FloEFD.Pro is fully embedded inside Pro/ENGINEER Wildfire, the combination of the two engineering tools enables Bronswerk to use a single design platform for geometrical design as well as mechanical and aerodynamic analysis without changing their design process. "Using FloEFD.Pro, we were able to make hundreds of variations in the geometry to fine-tune the fan and heat exchanger performance and reduce costs without sacrificing quality" explained Bertels.

After fine-tuning their design, Bronswerk has finally created a new design that is a giant leap forward and years ahead of their competition. "Our challenge was to develop the quietest cooler system. The added efficiency was an additional and unexpected huge benefit. Sometimes when you want to improve something you can't just improve it a little bit. You need a large improvement to breakthrough into uncharted territories." With



Since its inception in 1940, Bronswerk has become a leading designer and manufacturer of heat exchangers, compressors and fans. Bronswerk's air-cooled coolers are used as process coolers in the oil and gas as well as chemical industries. They have become the supplier of choice for companies such as Shell, Exxon Mobil, Bayer, Philips, Unocal and Texaco. Their fans range from 2 to 10 meters in diameter and cool heat exchangers which are extremely large – often as big as a building.

While designing a new generation of these industrial cooler systems and fans, the engineering team at Bronswerk faced a difficult challenge: to develop a new cooler and fan and make them as quiet as possible to meet stringent environmental standards. To further complicate the matter, the new fan



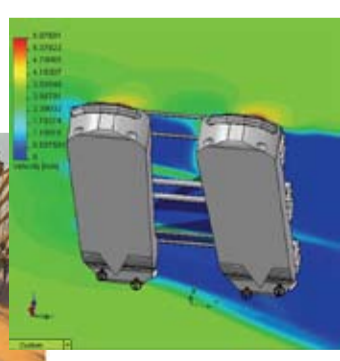
over 30 years of experience in the field of flow analysis, Bertels adds that "we would not have been able to solve this challenge without FloEFD.Pro."

For more information visit: www.flomerics.com/info

Today's mainstream mechanical designer is able to perform fluid-flow simulations directly on his own desktop, dramatically enhancing his design capability for new products.

Saipem UK Selects FloEFD Software for use with Autodesk Inventor...

Saipem UK has selected FloEFD Engineering Fluid Dynamics software for the analyses of transportation barges, construction and drilling vessels, and deck structures. "We have been using traditional Computational Fluid Dynamics (CFD) software to analyse our Autodesk Inventor solid models for the last couple of years. But we found the analysis process too time consuming because our software was unable to meet the rigorous demands of our development timescale. FloEFD on the other hand matches our needs for shortened design time



and ease-of-use" said Dr. Zafer Ayaz, Naval Architect. "As a market leading contractor in the oil and gas industry, Saipem is on the forefront of technological innovation. Their Naval

Architecture Group handles a wide range of projects; therefore, we are very pleased to see that FloEFD fits within their design process effortlessly and offers them the analysis power they need" added Gary Carter, General Manager of the Mechanical Analysis Division and previously CEO of Flomerics.

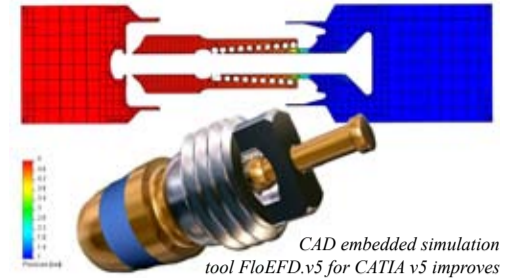
For more information visit: www.flomerics.com/info

VENTREX use CAD embedded FloEFD for CATIA 5 Saving 4 Months Work on Automotive Valve...

VENTREX Automotive GmbH, Graz, Austria, saved four months in the design of an automotive valve by taking advantage of FloEFD.v5 CATIA v5-embedded computational fluid dynamics (CFD) simulation software. "In the past, we would have had to build and test at least 50 valves in order to get the design right," said Peter Pfaffenwimmer, Project Manager for VENTREX Automotive GmbH, Graz, Austria. Daniel Gaisbacher, also a Project Manager for VENTREX, added that: "CAD-embedded CFD made it possible to determine simulation results nearly as fast as we changed the design. The result is that we were able to improve the flow rate of our new valve by 15% while eliminating about 50 prototypes and reducing time to market by 4 months."

Automotive air conditioning systems are beginning to use CO2 as the refrigerant because it does not damage the ozone layer and also has a very low impact on global warming. But CO2 based systems need to operate at pressures 7 to 10 times higher than previous generation systems and this requires redesign of many components. Among the components most affected by the change is the valves used to evacuate and charge the system. Most important, the pressure drops seen in existing valves are too high to enable the required flow rates. But reducing the pressure drop of valves is challenging because of the complexity of internal flow passages caused by the presence of components used to open and close the valve.

In the past, engineers would make design changes based on educated guesses. For each design change they had to build and test a prototype of the valve. This process was time-consuming and expensive and the test results did not provide diagnostic information that would help engineers determine whether or not the design change had the intended effect. CFD makes it possible to build a software



CAD embedded simulation tool FloEFD.v5 for CATIA v5 improves new valve by 15%, reducing prototypes by approximately 50 and lowering 'Time to Market' by 4 months

prototype of the valve that can be solved to determine the pressure drop of any particular design iteration without having to build a prototype. The greatest obstacle to the use of CFD has been that traditional CFD codes require the user to have a deep understanding of the computational aspects of fluid dynamics in order to be certain of obtaining accurate results.

But FloEFD.v5 greatly simplifies the simulation process by extending CATIA V5 functionality to include fluid flow and heat transfer simulation within a single user environment. "We selected FloEFD.v5 because it simplifies the process of performing fluid flow analysis to the point where it can be accomplished by any engineer," Gaisbacher said. "By using CFD software that is embedded into our CAD software, we could evaluate the performance of each new design iteration almost as fast as we could conceive it. This made it possible to quickly improve the performance of the design. We reduced pressure drop to the point that flow rate improved by about 15% in the final design at any given pressure. We reduced the number of prototypes that were required during the design process by about 50, which saved a considerable amount of money but most importantly let us bring the product to market faster. We have already shipped this product to customers and they have verified that it performs nearly exactly as predicted by the simulation."

For more information visit: www.flomerics.com/info

TROX Invests a Further £100K in R&D Facilities Expanding its CFD Tools...

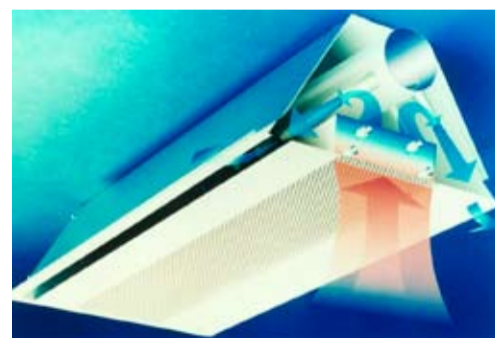
Leading air distribution system provider TROX UK has invested over £100,000 in further extending its test facilities and the purchase of specialist equipment. As well as creating six separate test cells for comfort air conditioning, CO2 testing and product development, TROX has also invested in Mentor Graphics' 'state of the art' Engineering Fluid Dynamics (FloEFD) software.

The FloEFD package is used to obtain computer visualisations of air distribution in terms of products and can provide a wide range of data prior to any detailed testing being carried out.

For example, on a product such as the new high

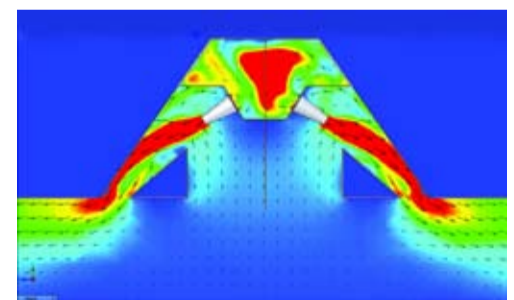
our products without having to make so many prototypes. It will enable us to validate models, perform prototyping within a controlled computer environment, and then validate another model in what is inevitably an iterative design process". "This means we can fine-tune the systems we are proposing, precisely matching the needs of our customers. And, because we do away with the need to manufacture a lot of prototypes, we save time and money for both our customers and for ourselves."

Other features and benefits of the FloEFD package are its pinpoint accuracy and massive analytical power which means that TROX can precisely predict the behaviour of 'real world' products. The program is also incredibly efficient so the simulation task can be performed speedily and can include 'what-if' variations, increasing the confidence of TROX customers that the proposed installation will perform exactly as predicted.



induction chilled beams there were literally millions of combinations of air nozzle diameter, spacing, angle of discharge, air velocities etc., that needed to be tested to improve the induction of the new beam engine. With FloEFD all of these variables could be visualised and more rapidly conclusions drawn as to the most likely high performance configuration.

Cliff Bailey, laboratory manager at TROX UK, said: "FloEFD gave us 6 months testing and results in little over 3 weeks. We were then able to take the recommended options from the FloEFD analysis and conduct detailed full scale actual testing with the beam engine." He went on to say: "We already have a comprehensive physical test facility in the form of our extended six laboratories but this leading edge FloEFD program will help us develop



Mr Bailey added: "Calculating air flow data has traditionally relied on our experience and experimentation which can be laborious and time consuming."

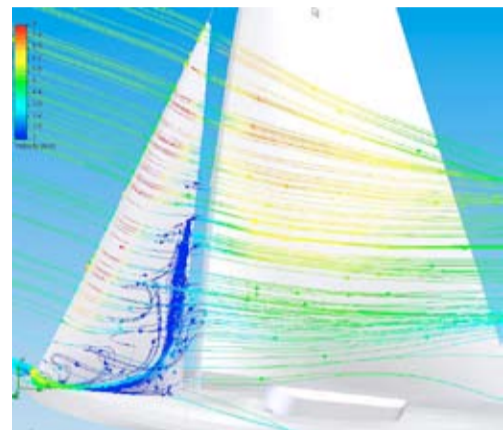
"The package is also excellent for displaying airflow characteristics for use in presentations, which makes it easy for our customers to see how the product will work in practice, and enables us to give everyone involved in the project a common basis for discussions and decision-making over complex technical issues."

For more information visit: www.flomerics.com/info

Optimising Sail Design for the Olympics Using 3D Air Flow Simulation...

Finland-based sail maker WB-Sails used FloEFD fluid dynamics simulation software to develop and optimise sails for Star and Finn class Olympic sailboats. "Our traditional panel codes predict sail forces reasonably well in the upwind case, when flow separation is not a major factor. But we were desperate for something more powerful to predict 3D air flow and sail forces on downwind legs and in the lighter winds that we expected in Beijing," said Mikko Brummer, Head of R&D at WB-Sails. "We had been looking for some time at traditional Computational Fluid Dynamics (CFD)

design scenarios. The detailed predictions coming from FloEFD have made us aware of 3D air flow phenomena that we never knew existed", said Brummer. "Another thing we have learned is that the mast, which is usually considered as a necessary device that adds drag, can actually add to the driving force of the boat."

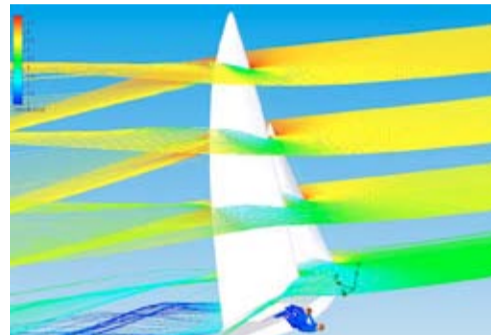


software programs but we found them too slow and cumbersome for our purposes. In FloEFD we found exactly what we were looking for - an extraordinary ease of use provided by features such as "direct CAD-to-CFD technology and automatic mesh generation. This gives us the very fast turn-around times we need for looking at alternative 'what if?'

WB-Sails uses FloEFD to pinpoint problem areas in sails where flow separation is likely to occur. Flow separation reduces the driving force in the sail and increases drag. Simple sail trim changes can be performed directly within the software, and the close coupling with CAD software makes it easy to change components such as sails and rigging. For regions of particular interest a denser local mesh can be created quickly making it possible to investigate detailed flow phenomena in a reasonable time with an ordinary desktop PC.

The FloEFD CFD software allows non-uniform air flow conditions to be specified at the entry plane to represent the atmospheric boundary layer at the sea's surface. When combined with the speed of the boat this results in a complex 'sheared' and 'twisted' wind pattern approaching the sails. The built-in graphical post-processor in FloEFD enables air flow trajectories and pressure forces to be visualised with full 3D animation, making it much easier to understand the results. Driving force, heeling force and turning moments

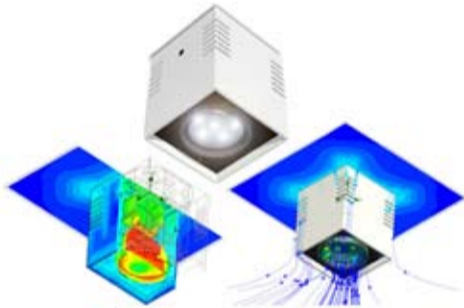
are calculated and output in absolute and coefficient form. These forces are used to predict and optimise the sailboat's performance. "We hope this will lead to medals for our sailors in the Star and Finn class at the Beijing Olympics in August 2008" said Brummer.



Fredrik Lööf and Anders Ekström went into the Star Men's Keel Medal Race on Thursday 21st August in Gold position. However, after a very tense finishing straight, they were pipped at the post by Team GB resulting in Sweden's first medal of the Regatta, a well deserved Bronze Medal.

For more information visit: www.flomerics.com/info

Power LED Overheating Problems Solved in Just One Day..



Voxdale, a Belgium based engineering consulting firm, used Mentor Graphics' FloEFD computational fluid dynamics (CFD) software to solve a power light emitting diode (LED) thermal management problem in just one day. The initial prototype of the LED system overheated to the point that the amount of light delivered by the device and its lifetime were both substantially reduced. Voxdale engineers used the CFD results to redesign the housing in just one day to improve airflow, which eliminated overheating and increased light output and life to the desired levels.

The recent innovation of high power LEDs has the potential to dramatically reduce the amount of power required for lighting residential and commercial buildings. Useful life for white LEDs

can range from approximately 6,000 hours to more than 50,000 hours compared to less than 2,000 hours for incandescent bulbs. Thermal design typically represents the greatest challenge in LED system design projects because 75% to 85% of the energy used to drive LEDs is converted to heat and must be conducted from the LED die.

"The lighting systems manufacturer built a prototype of their design and discovered that the temperature of the LED quickly rose above its maximum operating temperature," said Koen Beyers, President of Voxdale. "The manufacturer was anxious to get their product for market. Who knows how long it would have taken them to solve the problem by modifying and re-testing the prototype until they found a design that worked. The big problem with this approach is that you don't have any idea what is causing the problem so you are essentially flailing around in the dark."

"In the past, CFD required the user to have a deep understanding of the computational aspects of fluid dynamics in order to be certain of obtaining accurate results," Beyers said. "But in the last few years a new generation of CFD software has been introduced that eliminates the need for engineers to master the computational part of CFD and instead allows them to focus on the fluid dynamics of the product. FloEFD, the CFD software that we

use, uses native Pro/ENGINEER computer-aided design (CAD) data and automatically grids the flow space and manages flow parameters as object-based features."

Beyers modelled the prototype using Pro/ENGINEER and entered the materials properties and boundary conditions required for CFD simulation. Defining all of the information needed to convert the CAD geometry to a CFD model took about 30 minutes. Then Beyers gave the command for the CFD software to mesh the fluid path around and through the housing. The simulation results showed the original designers had not provided enough outlets to enable the air to flow freely through the housing.

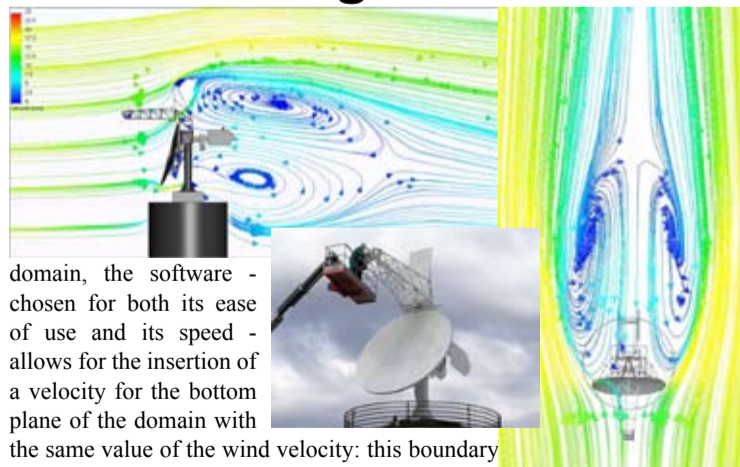
Beyers modified the design to increase the outlet area while keeping in mind that the manufacturers wanted to minimize their size for aesthetic reasons. His new design increased the flow through the housing while ensuring that air stayed in the housing long enough for considerable heat to be transferred to it. He also tried a few different variants of the heat sink to optimise their geometry in order to gain a bit more reduction in the size of the outlets. The new design met all of the client's requirements.

For more information visit: www.flomerics.com/info

C&C of Italy Analyse Wind Loading on RADAR Installations...

CAE & Cooling Systems (C&C) of Rome, Italy, recently carried out a study on a servo/antenna, part of a meteorological radar system for CNR ISAC (National Centre for Research). The antenna, a Polar 55C, is a polarimetric C-band Doppler radar system used for research purposes.

This study required the development of a 3D CAD model to represent the geometry, the masses, and the inertia of the pre-existing antenna system. On the basis of this CAD model, a CFD and a FEM model were computed; the first was to analyse and calculate the loads acting on the antenna due to the wind action, while the FEM model was used to verify the strength for both the main elements of the structure and the kinematic mechanisms under the effects of aerodynamic, inertial and mechanical loads. It should be noted that the most relevant external loading was the aerodynamic wind load. The CFD model was computed using FloEFD. In order to reduce the size of the computational



domain, the software - chosen for both its ease of use and its speed - allows for the insertion of a velocity for the bottom plane of the domain with the same value of the wind velocity: this boundary condition allows the effect of the real distance between antenna and ground to be simulated without the need to simulate the whole structure. Thanks to ease of use of the software and its ability to manage complex geometries without substantial

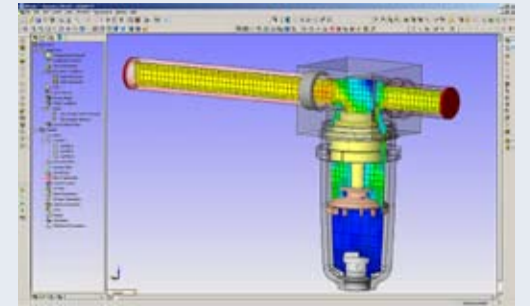
convergence problems, the antenna was analysed in several different configurations, with wind velocities averaging 60 km/h from different directions. For each configuration the reactions (torque and force) acting on the kinematic mechanisms of azimuth and elevation were computed.

FloEFD, thanks to its tight integration with PTC Pro/Engineer allowed C&C to evaluate the effects of possible counterbalancing solutions of the elevation axis loads in order to increase the stability and the accuracy of the system when subject to wind gusts.

CFD Related White Papers

Meshing Techniques for Flow Simulation...

Flomerics, now Mentor Graphics, a world-leading developer of engineering simulation software, has published a comprehensive technical white paper on mesh generation techniques for fluid flow simulation.

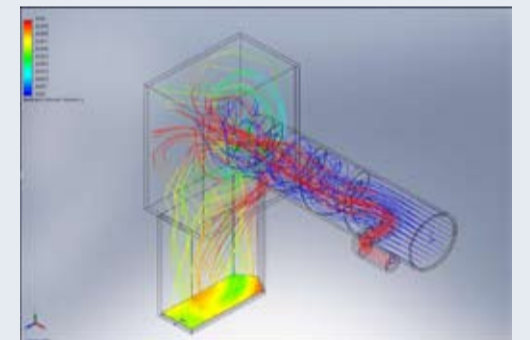


Written for mechanical engineers, the paper explains the need for a computational mesh and the pros and cons of different mesh types. It also explains the factors affecting the choice of grid system such as cell shape, geometry representation, mesh structure and grid generation. "Engineers new to the world of fluid and thermal analysis often find the basic jargon quite daunting" said Dr. John Parry, research manager. "Therefore, we have created a no-nonsense reference document for engineers focusing on meshing techniques. Our intention is to answer typical user questions, and so we have made the white paper freely available to anyone visiting our web site."

Interested readers are encouraged to download the white paper at: www.flomerics.com/info

Gas Mixing - Best Practices...

The mixing of gas and air is important on performance gas burners. The quality of the gas / air mixture determines basically if a burner will function or not. Too much gas or air will have the effect that the chemical reaction of combustion is stopped and the flame of the burner will extinguish.

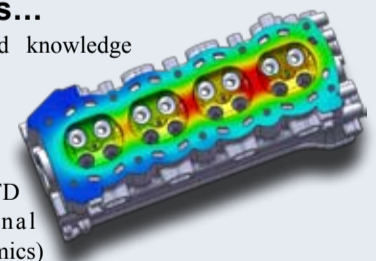


Optimising gas and air mixing to meet the requirements of a specific application is a challenging process that normally involves a very expensive and time-consuming process of building and testing prototypes. Large companies have used computational fluid dynamics (CFD) to simulate gas mixing but its use up to now has been primarily limited to research or troubleshooting existing designs because of the considerable cost, time and expertise required to use CFD technology.

Interested readers are encouraged to download the white paper at: www.flomerics.com/info

Accelerate Automotive Design with Engineering Fluid Dynamics...

It's widespread knowledge that computer modeling and virtual prototyping in the form of CFD (Computational Fluid Dynamics)



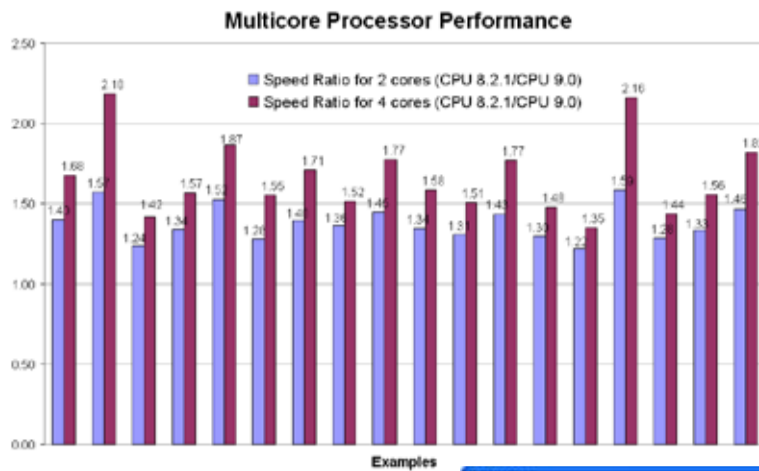
can make major contributions to reducing product costs and accelerating time to market. However, many managers do not yet realize that software advances are making these benefits easily accessible to small- and mid-sized enterprises.

Thanks to a class of software known as FloEFD - Engineering Fluid Dynamics - there is no longer the need to hire or train CFD specialists, out-source analysis to consultants, or conduct tests on multiple physical prototypes; instead, now design engineers with standard training and working in any size company can use their existing knowledge to successfully perform flow and heat analyses and thereby dramatically reduce the number of prototypes needed.

Interested readers are encouraged to download the white paper at: www.flomerics.com/info

FloEFD v9 - The Fastest Just Got FASTER with Multi-Processor Support...

Widely recognized as the fastest Computational Fluid Dynamics (CFD) software available in terms of CAD-to-Solution, FloEFD just got faster! Famed for delivering CFD into the hands of designer engineers with its CAD-embedding, automated meshing systems, the new version of FloEFD version 9 has taken a further leap forward by adding the ability to use the latest in multi-core processor technology. "This time we are focusing on performance, improving the ability to use physical models and improving overall usability." said Ivo Weinhold, product manager for Mentor Graphics' FloEFD range of CFD software. He goes on to say "Over the past couple of years dual core and today quad core processors have become common in the workplace, allowing more and more mainstream design engineers using FloEFD the functional ability to produce many of the results that were once the sole realm of specialist CFD experts." With FloEFD v9, not only has the software the ability to use multi-core processors, it also permits the user the choice of just how many processors



the program allows the user the capacity to have the system work either 100% on the model or, on a dual core or quad core machine, have the software work in the 'background', releasing to the user, the computational resource to continue work on other projects.

to use depending on the processing power required. Weinhold continued "Due to differing industry model complexities, it is difficult to put precise numbers on the speeds that can be attained, however, as a guide, a user working on a large model comprising a complex mesh pattern could typically see speed increases of 80% on a dual core machine and more on a quad core system." This ability to switch cores on and off within



NEW FloEFD v9 Functionality Improves Decision-Making, Reduces Design Times and Accelerates Your Solutions...

Call-Outs for Boundaries Tool

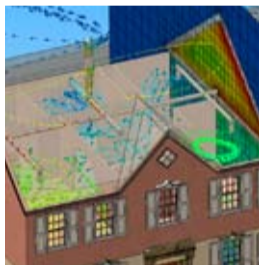
The new 'Callout' feature is great for quickly creating, editing and visualizing 'Boundary' conditions. These are created automatically as a condition is selected for all boundary conditions sources, i.e., fans, porous media, rotating regions and instantly become visible within the display area, but can easily be switched on or off by clicking the 'AutoHide' property. Appearance of 'callouts' can be fully customised: the user is able to edit background colour, its transparency and font type. Adjusting font type of the 'callouts' can be helpful should a user wish to save an image or create a screen shot.

Probe Tool

FloEFD's new Probe tool allows the user to see and obtain parameter values directly from 'Cut Plots' and 'Surface Plots' simply by moving the pointer over the required areas. The user is able to create 'Callouts' showing the values of parameters at the selected points of the plot with a click of the mouse, which are fully editable for background colour, transparency and font.

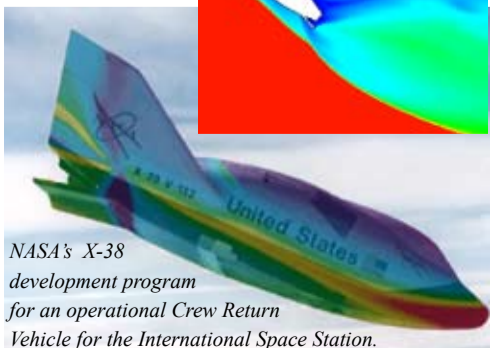
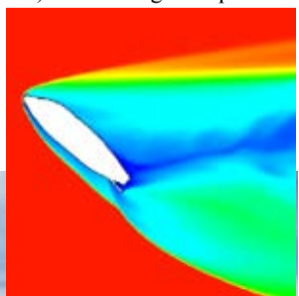
Local Mean Age Tool

The new Local Mean Age (LMA) tool enables the user to easily find areas of dead fluid within a confined space by calculating the average time for fresh fluid to reach a point in space. Low values showing regions with good fresh fluid supply and high value regions of poor fresh fluid supply.



Hypersonic Analysis

The new Hypersonic flow option allows the user to analyse the flow of air at Mach numbers of $5 < M < 30$ and air density of $\rho > 0.001 \text{ kg/m}^3$ (i.e. the flight altitude $H < 50\text{km}$) with the high-temperature air dissociation and ionization effects accounted in accordance with the equilibrium.

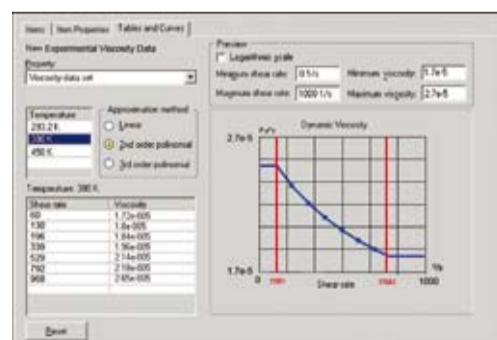


New Models for Non-Newtonian Liquid Simulation

Three new non-Newtonian liquid models are now available within FloEFD v9 for specifying non-Newtonian liquids in the Engineering Database:

- CROSS WLF model
- Second Order model (also known as Klein's model)
- Viscosity table: approximation based on experimental data

In addition the shear rate distribution can now be visualized in the fluid.

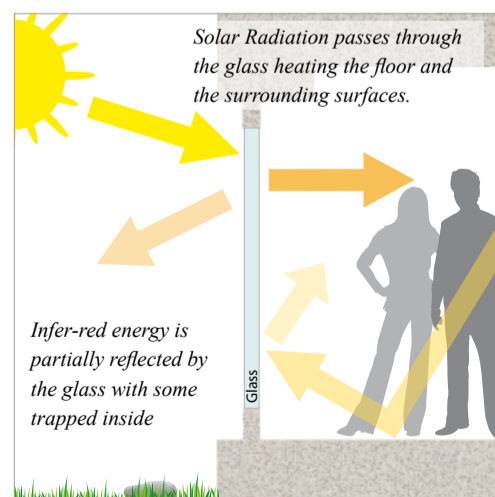


This makes it easier for the user to define the necessary non-Newtonian model for the application's fluid, simulating Non-Newtonian liquids whose viscosity is highly dependent on temperature.

For more information visit: www.flomerics.com/info

Solar Radiation Transparency & Absorbance Tool

A user can now set a component to be transparent for solar radiation only (as well as for thermal radiation only). The absorption factor of a wall (i.e. a portion of incoming radiative heat absorbed by the wall) can now be specified differently for solar and thermal radiation.



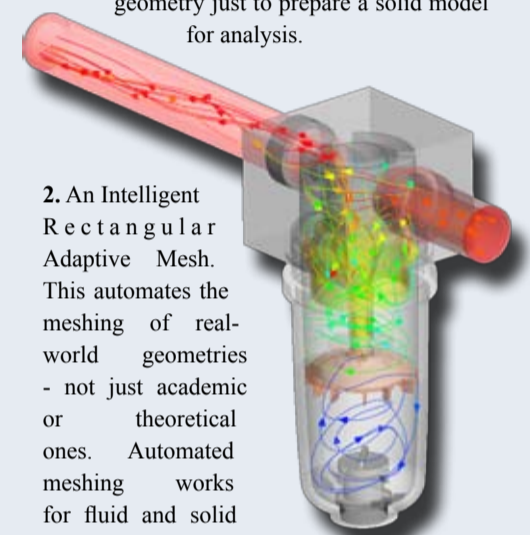
This now makes it possible for the user to analyse 'greenhouse' effect. This new function permits a more accurate simulation when solar radiation is required to be assessed within a project.



The 7 key technologies of Engineering Fluid Dynamics...

Engineering Fluid Dynamics (EFD) shares the same math foundation as traditional CFD code. However EFD features seven key technologies that address the everyday engineer's need to solve everyday problems:

1. Direct CAD to CFD means EFD uses your 3D CAD geometry and solid model information such as features and parameters to help you simulate your designs in real-world conditions. Traditional CFD programs require that you create additional solid parts to represent the fluid (empty) regions. Needless to say, this process takes a lot of time because you must identify each region manually and then create geometry to fill it. But EFD differentiates between solid and empty regions, and automatically represents the fluid domain. Users need not create unnecessary geometry just to prepare a solid model for analysis.



2. An Intelligent Rectangular Adaptive Mesh. This automates the meshing of real-world geometries - not just academic or theoretical ones. Automated meshing works for fluid and solid regions, including mesh refinement where necessary. You can use the automated mesher or further manipulate the mesh manually.

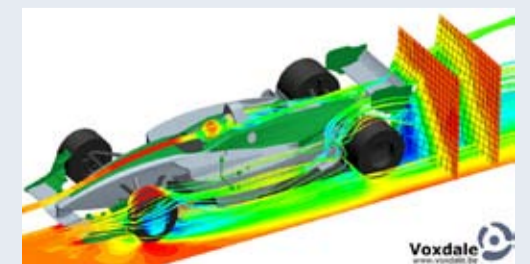
3. Modified Wall Functions. EFD features grid-independent near-wall modeling using Partial Cells technology. This lets you simulate boundary-layer phenomena for fluid flow and heat transfer effects without needing a specifically designed mesh or a lot of time to solve the problem.

4. Laminar-Transitional-Turbulent Modeling. You need not worry about identifying when and where flow characteristics change within your model because EFD supports laminar, transitional, and turbulent flows.

5. Automatic Convergence Control. Cutting-edge numerical methods with multigrid schemes and good convergence behaviour ensure that your engineering questions are answered every time. With EFD, the first run converges without any complications.

6. Design Variants Analysis. EFD makes it easy to conduct "what-if" analyses. Modify your solid model without having to reapply loads, boundary conditions, and material properties. Then simply compare the results.

7. Engineering User Interface. EFD uses the CAD interface, so instead of dealing with technical jargon such as y^+ value and Dirichlet Boundary Conditions, you use familiar terms such as walls, inlets, and outlets.



For more information visit: www.flomerics.com/info