TechTIPS

Digital Ebook | A Design World Resource

CFD Toolkit

Inside

Sponsored by Mentor Graphics®

Computational Fluid Dynamics Analysis & Simulation Explained

Courtesy of Design World

Computational fluid dynamics (CFD) analysis software is a simulation tool that provides engineers, designers and scientists with a non-intrusive, virtual modeling environment with insightful visualization that helps solve and analyze problems that involve fluid flow. The software does this by using numerical methods to solve fundamental nonlinear differential equations that describe fluid flow (the Navier-Stokes and allied equations) for predefined geometries and boundary conditions.

Results can provide predictions on flow velocity, temperature, density, and chemical concentrations for any region where flow occurs. By using CFD simulation software, engineers and scientists can predict the impact of fluid flow behavior on products and evaluate a wide range of design parameters while designs still reside in digital form on the computer.

CFD simulation provides users with an insight into flow patterns that are difficult, expensive or impossible to study using traditional (experimental) techniques, making it a popular tool and a real timesaver for manufacturers trying to shorten design cycles and beat competitors to market. The software's fluid flow analysis capabilities can be used to design and optimize new designs or troubleshoot already existing products.

By using CFD simulation, users can:

- Improve the aerodynamic characteristics of vehicles, planes, rockets, etc.
- Help chemical engineers maximize equipment yield
- Enable petroleum engineers to devise optimal oil recovery strategies
- Help surgeons to study and possibly cure arterial diseases
- Forecast the weather and warn of natural disasters
- Reduce health risks from radiation and other hazards
- Enable military organizations to develop weapons and estimate the damage

Usage of CFD software has seen rapid growth over the last decade and is now being widely used in auto and aircraft design, electronic design, weather science, food processing, power generation, civil engineering, semiconductors, and oceanography.

Part of its growing popularity and increased usage stems from the fact that usage of a CFD simulation solution, is much less expensive than the experiments that would be required to provide the same results. Since the products being tested are not yet built, no physical modifications are required, and changes made as a result of the simulation can be done quickly and at little expense.

CFD simulation and analysis is also becoming increasingly reliable as the numerical schemes and methods upon which CFD is based are rapidly improving. When engineers and scientists put their trust into CFD tools, they gain the insight needed to better understand the complex problems they face daily so they can more easily conceive of innovative ideas to solve them. End result is more innovative products that are designed better, faster, and at less cost to manufacturers.

The 10 Myths Of Computational Fluid Dynamics

Courtesy Drs. Ivo Weinhold & John Parry, Mentor Graphics, Mechanical Analysis Division

HOW THE 5 MYTHS OF CFD HAVE EVOLVED TO BECOME 10

Introduction

Some five years ago we felt compelled to write our whitepaper "The Five Myths of Computational Fluid Dynamics"[1]. Since then, we have had quite a lot of feedback regarding our views, and broadly, our debunking of these myths resonated with people. Through all the feedback and conversations we've had on this topic, it's become clear that the situation is more complex than we first thought. After spending some time to cogitate on this, we felt compelled to write an addendum to "The Five Myths…". Here, we provide a summary of the original myths, introduce four new related myths, and add a completely new one.

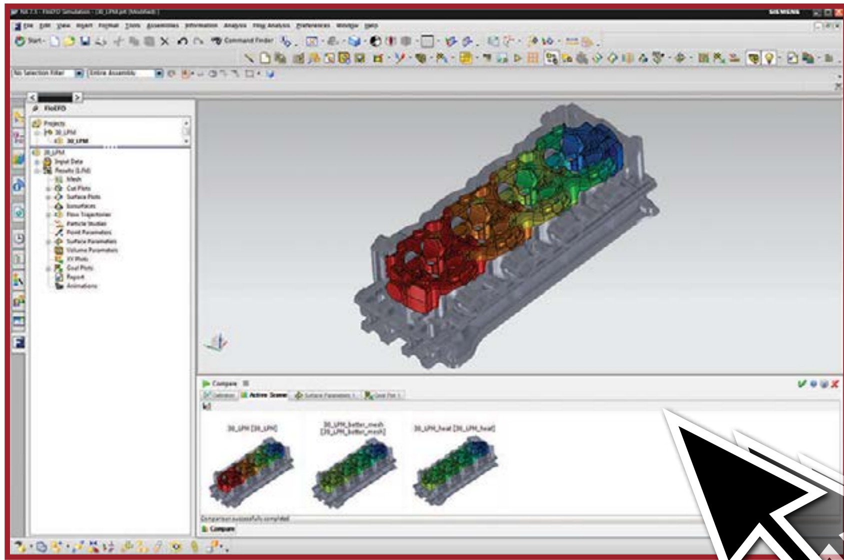
Summarizing "The 5 Myths of Computational Fluid Dynamics "

Since we wrote the "The Five Myths…" quite a lot has happened in the CFD market, so before summarizing these myths we should clarify the scope of this whitepaper. Our comments relate specifically to the broadest section of the CFD market, that of commercial general-purpose CFD software solving the Navier-Stokes equations. We are deliberately excluding so-called meshless approaches using Lattice-Boltzmann methods, and application-specific CFD such as tools for injection moulding, electronics cooling, data center simulation, etc. where their tailored functionality delivers a different value proposition to customers. So, to recap:

MYTH #1: CFD is too difficult to be used in the design process

This myth has a historical basis. Like FEA codes in the distant past, CFD codes of the 1980s and 1990s were difficult to use. Fit-for-purpose meshing, choice of solution numerics, turbulence modeling, achieving and judging solution convergence, assuring result fidelity, and correct result interpretation were all once expert-only activities. Today, the skills a mechanical designer needs to operate the CFD software are simply knowledge of the CAD system and the physics pertaining to the product, both of which the majority of design engineers already possess. This is because the automation and overall usability of the tools has increased so much [2]. However, the importance of usability is largely misunderstood; and in this, we have discovered a new myth: Myth #6 – Usability is not a prerequisite for a reliable and reproducible workflow.

Figure 1: CAD-embedded CFD package FloEFD for Siemens NX



MYTH #2: CFD takes too long to use during the design process

The greatest time sink for CFD has always been the meshing process, with a considerable amount of manual intervention needed to achieve acceptable mesh quality by eliminating gaps and overlaps, reducing skewness, aspect ratio, warpage, and controlling the volume of individual cells (cell size ratio to neighbor cells, smallest cell size, and mesh distribution). As design inherently involves changing geometry, this semi-manual process had to

The 10 Myths Of Computational Fluid Dynamics

[continued]

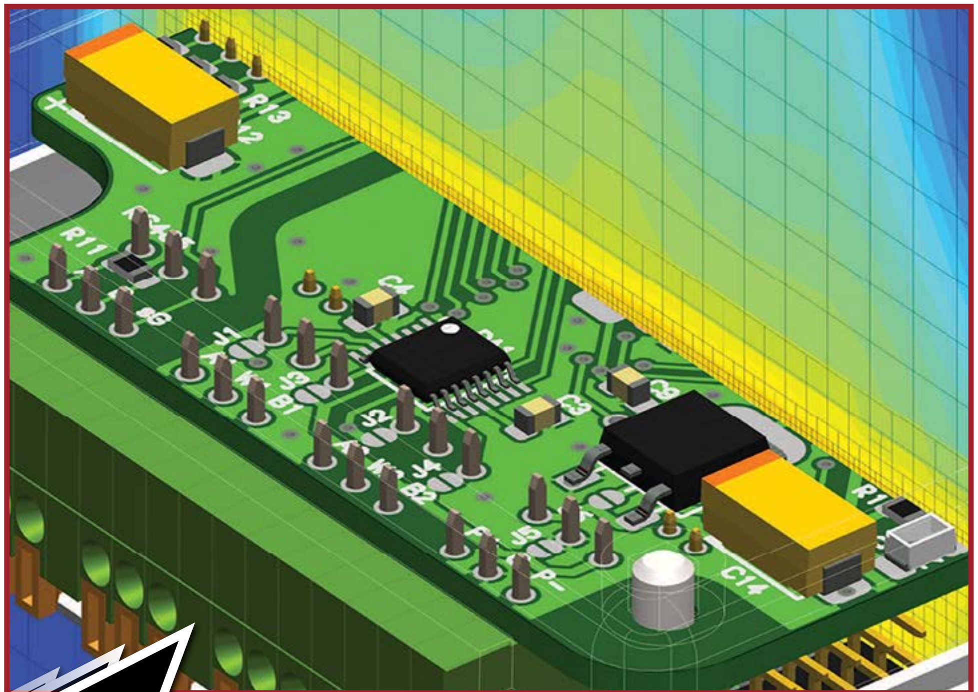


Figure 2: Efficient thermal simulation of an electronics enclosure

be repeated for each design iteration. All of these steps can now be fully automated using native 3D CAD data directly for fluid flow simulations without the need for translations or copies. New parts and features resulting from design changes can be meshed in a matter of minutes, dramatically reducing the time required for analysis. Acceptance of this has, however, revealed another myth: Myth #7 – Accuracy has to be sacrificed to use CFD during the design process.

MYTH #3: CFD is too expensive to be used by mechanical designers

In our original whitepaper, we observed that traditional CFD codes cost in the region of $25,000 to lease for one year. The latest generation of CFD code intended for use during the mainstream design process cost around $25,000 for a perpetual license. The only ongoing cost is a maintenance fee on the order of 18% ($4,500) per year. The cost of ownership is further reduced because it can be used by a mechanical design engineer with minimal training[1]. Novel techniques, such as immersed boundary treatments for fluid–solid surface friction and heat transfer, massively reduce the mesh count required to achieve accurate results, allowing useful work to be undertaken on multicore personal computers and laptops, reducing the cost still further. This myth proved relatively uncontroversial; however, it relates to a third new myth: Myth #8 - Experts are needed to get accurate CFD simulation results.

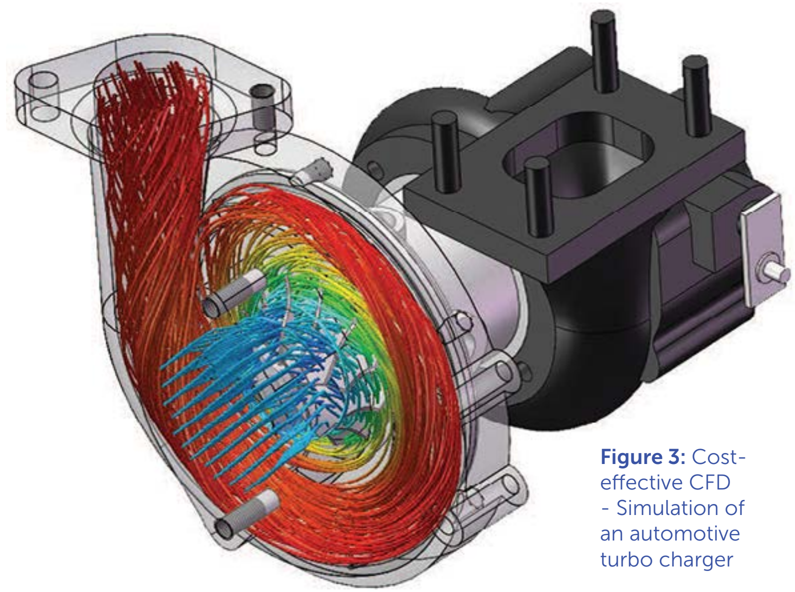


Figure 3: Cost-effective CFD - Simulation of an automotive turbo charger

MYTH #4: You can't directly use your CAD model to do CFD analysis

In the past, it was necessary to copy or translate the CAD model to a different program and then modify it substantially to create the CFD model. Many people found it more reliable and less effort to start from scratch by recreating the geometry within the CFD program, despite this

[1] That is training in the use of the tool. Knowledge of fluid flow and heat transfer appropriate to the application is necessary, but not a detailed knowledge of CFD.

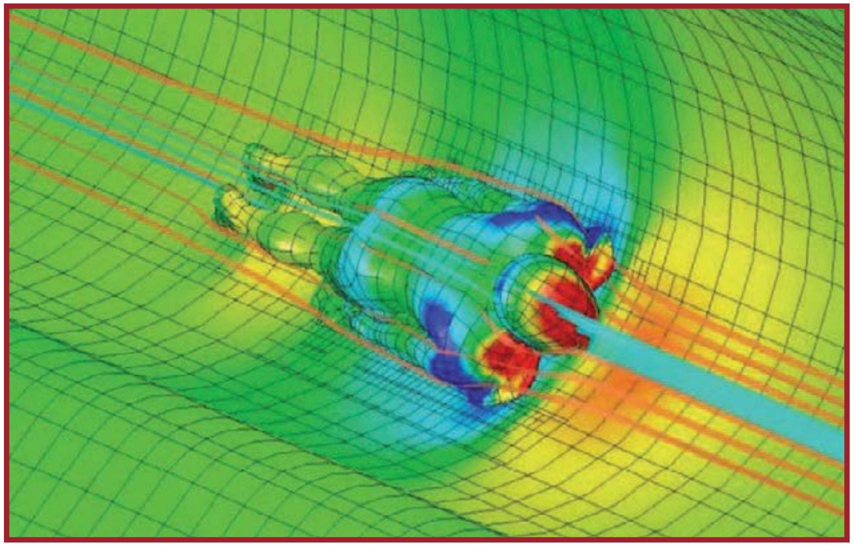
The 10 Myths Of Computational Fluid Dynamics
[continued]

involving a considerable expenditure of time and introducing an additional and significant source of error. Today, native 3D CAD data can be used directly for flow simulations without the need for translations or copies, or creating phantom "objects" in the feature tree to represent the flow spaces. The myth that CAD geometry can't be used directly for analysis persists today, but in a slightly different guise, giving us our fourth new myth: Myth #9 - Production CAD is too complex to use for analysis.

MYTH #5: Most products don't need CFD analysis

We judge this myth to have been largely consigned to history. It is apparent that, today, CFD is used to improve products as diverse as swimming pools, toilets, hand dryers, lawn sprinklers, gas meters, production printing systems, disk drives, and oil filters to name just a few applications. Although not yet complete, the democratization of CFD for use in product design has extended into undergraduate courses and even to high school programs [3, 4]. At this point, we would like to introduce the new, or at least newly identified, myths of CFD that have come to our attention.

Figure 5: CFD simulation of a bob skeleton ride, used to optimize the sled design (Bromley Technologies)



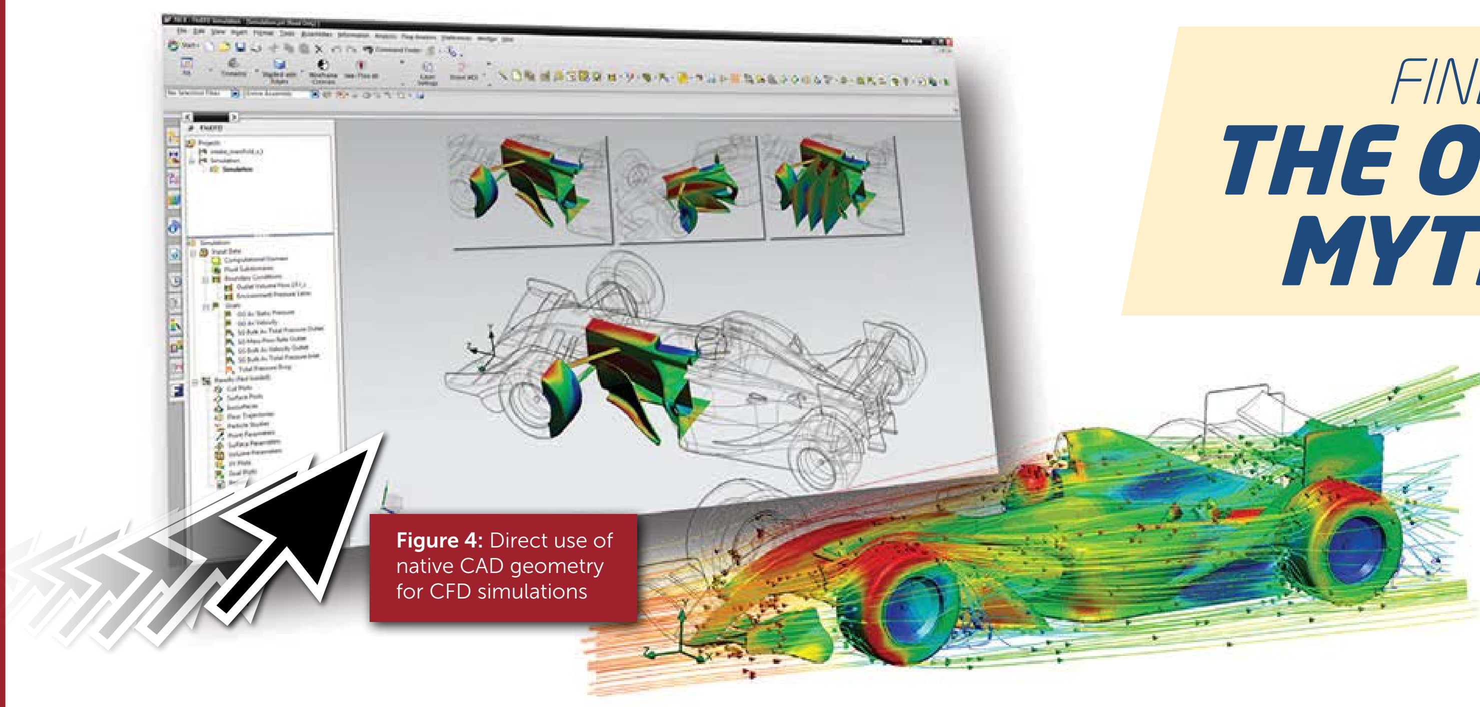


Figure 4: Direct use of native CAD geometry for CFD simulations

The 10 Myths Of Computational Fluid Dynamics

[continued]

References

1. "The Five Myths of Computational Fluid Dynamics", NAFEMS BenchMARK magazine, April 2008, pp. 28-29.
2. "The Third Wave of CFD", Ivo Weinhold and John Parry, Proceedings of the NAFEMS World Congress, Salzburg, Austria, June 2013.
3. Baldwin High School in Kansas Wins Second Real World Design Challenge (RWDC) National Aviation Design Competition Using Mentor Graphics FloEFD Technology http://www.mentor.com/company/news/baldwin-high-wins-second-real-world-design-challenge-using-floefd
4. "High School Students Fly with FloEFD™", Mentor Graphics' Engineering Edge Magazine, Vol 1, Iss. 1., http://www.mentor.com/products/mechanical/engineering-edge/volume1/issue1/high-school
5. "Turbulent Heat and Momentum Transfer in Rough Tubes", M. R. Malin and J. D. Parry, The PHOENICS Journal of Computational Fluid Dynamics and its Applications, Vol. 1, No. 1, January 1988, pp. 59-80.
6. "Flomerics' EFD Meshing Technology: A White Paper", Drs. John Parry and David Tatchell
7. "Enhanced Turbulence Modeling in FloEFD", Mentor Graphics Whitepaper MGC 02-11 TECH9670-W http://s3.mentor.com/public_documents/whitepaper/resources/mentorpaper_65206.pdf
8. "Concurrent CFD: PLM-Embedded Computational Fluid Dynamics for Upfront Product Design", Drs. John Parry, CEng & Ivo Weinhold, Proceedings of NAFEMS World Congress, Boston MA, June 2011.
9. "Back To The Future – Trends In Commercial CFD", Drs. Keith Hanna and John Parry, Proceedings of the NAFEMS World Congress, Boston MA, June 2011
10. "Advanced Immersed Boundary Cartesian Meshing Technology in FloEFD" MGC 02-11 TECH9690-W http://s3.mentor.com/public_documents/whitepaper/resources/mentorpaper_65169.pdf
11. FloEFD and FloTHERM XT, http://www.mentor.com/products/mechanical/

Contact Details

Dr. Ivo Weinhold ivo_weinhold@mentor.com ;
Dr. John Parry, CEng. john_parry@mentor.com

Success Story:

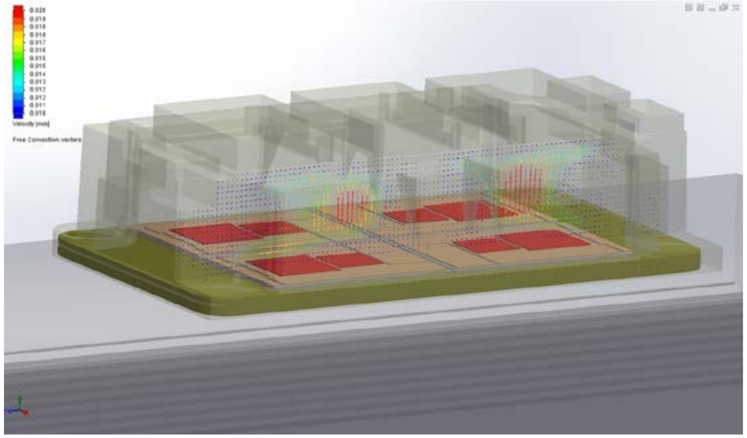
How E-Cooling GmbH Used FloEFD to Mesh Complex Geometrics

Courtesy of Mentor Grapics

E-Cooling GmbH in Berlin is an engineering consultancy founded by Karim Segond. Their expertise lie in providing 3D thermal and flow analysis, enhancement and development supporting electronics, electric engines, and power electronics. When E-Cooling began working on 20 MW motors it was evident that traditional CFD was not up to the job of meshing their complex geometries.

"The accuracy of FloEFD was always good. It is not easy to measure electric motors that run at very high speeds but FloEFD provided good results when compared to the measurements we received from our customers. FloEFD helped me to work on contracts that involved very complex geometries, such as a stator coil end turn support system, which I wouldn't have been able to do with other CFD software."

Karim Segond, E-Cooling GmbH



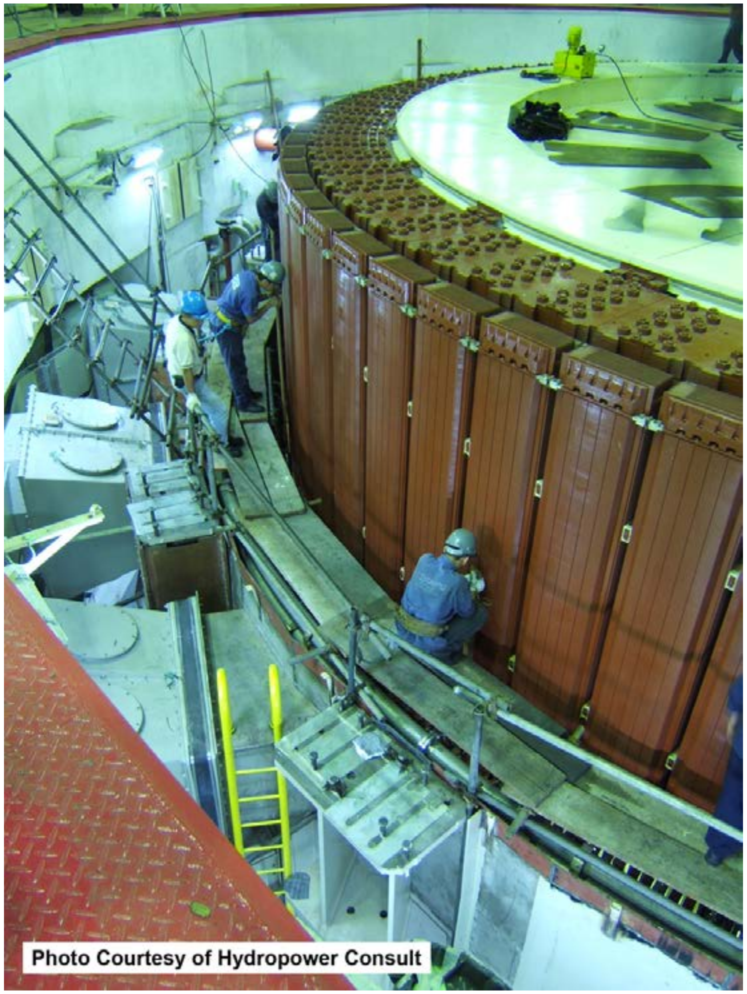


Photo Courtesy of Hydropower Consult

The Problem

Karim undertook the task of finding an approach that could handle complex geometries, wasn't too laborious but still delivered quality results, "The pre-processing with traditional CFD tools is much too slow for the simulation of large complex machines. I decided to look for a better solution that would solve my problems faster. I was specifically looking for software that can mesh such models with a Cartesian mesh and found FloEFD™". "E-Cooling's ethos is to provide detailed, accurate data to their clients at a reasonable cost, therefore the amount of man-hours used to mesh complex geometries is a big factor to consider for Karim.

The Solution

The biggest benefits I got from FloEFD was that it was embedded, I could work within a CAD system and use parametric CAD models. This made it easier to change any geometry and therefore run several variants very easily. Another point that lifted a heavy burden for me is the automatic meshing, so basically the meshing as I knew it became obsolete and I could spend my time with other things than manually mesh the geometry.

How E-Cooling GmbH Used FloEFD to Mesh Complex Geometrics

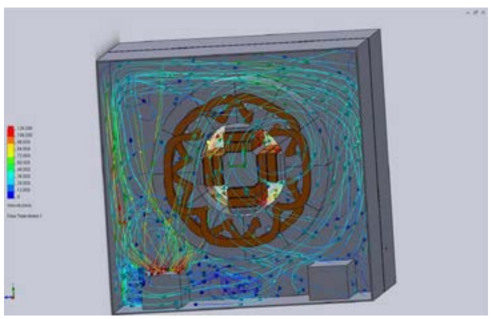
(continued)

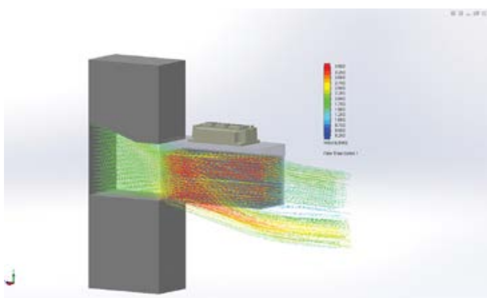
The Results

Karim is supported by his business partner, Guenter Zwarg, for the thermal management of these mega engines; the expertise of Günter covers almost all types of large electrical motors and hydro generators. According to Karim, his customers were always very satisfied with his work and the accuracy of the results has not suffered from the comfort of automatic meshing. Karim is also investigating the thermal management of the power electronics components used to drive such large engines.

Karim says "The cooling of power electronics is very important and should be considered as early as the concept stage of the design. Here CFD can be leveraged to optimize the design and ensure the best possible cooling for the components." Besides simulations of IGBTs E-Cooling has turned their attention to the overall cooling of a system.

Such cooling systems are too big and too complex to simulate with 3D CFD software. Therefore it is necessary to analyze them using a 1D thermal and fluid flow circuit that is modeled in a 1D CFD tool such as Flowmaster®. This combined solution then enables Karim to provide the qualities of both tools and provide the ultimate cooling solution to his customers.

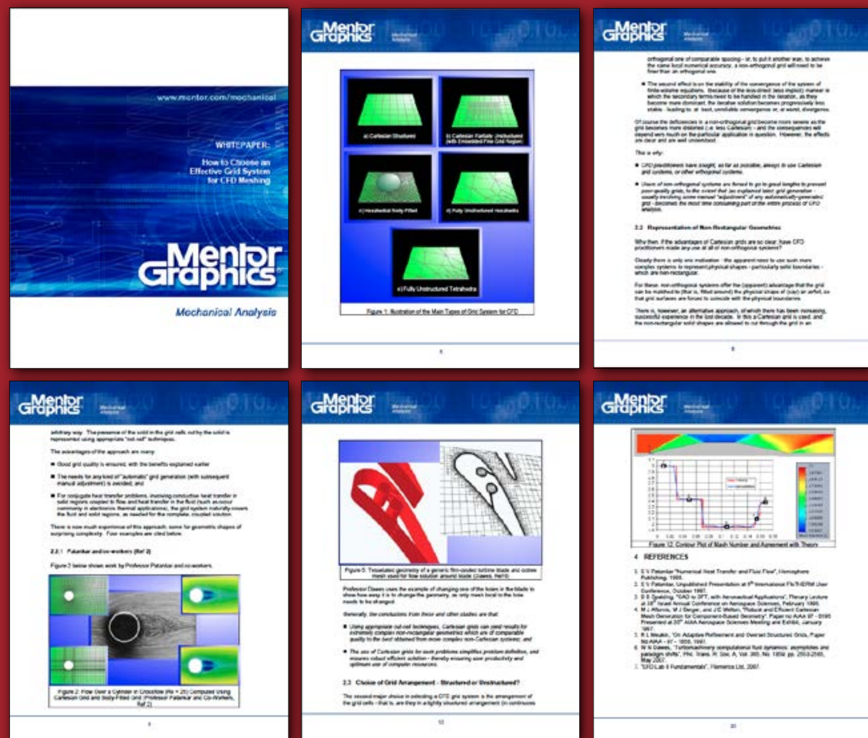




DOWNLOAD PDF NOW

ALSO OF INTEREST

MENTOR GRAPHICS WHITE PAPER:



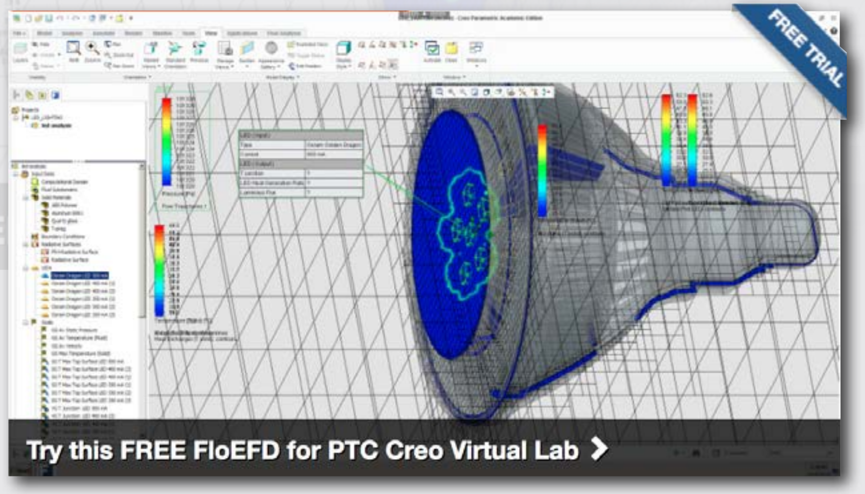
HOW TO CHOOSE AN EFFECTIVE GRID SYSTEM FOR CFD MESHING

Concurrent CFD is a new kind of CFD tool that enables mechanical engineers to simulate the flow of fluid and heat transfer for today's products using 3D CAD models. Download this 26-page resource from Mentor Graphics here.

DOWNLOAD PDF HERE

FREE TRIAL OFFER!

Test Drive FloEFD Free for 30 Days



In this cloud based software eval, you will have access to the fully featured FloEFD software which is embedded into all major MCAD tools. The virtual lab provides comprehensive sample design models & tutorials covering a wide-range of applications including:

- Ball Valve Design
- Conjugate Heat Transfer
- Heat Exchanger Efficiency
- Mesh Optimization
- Radiative Heat Transfer and more...

You can access this cloud based high performance computing environment from any modern web browser and be up and running in minutes.

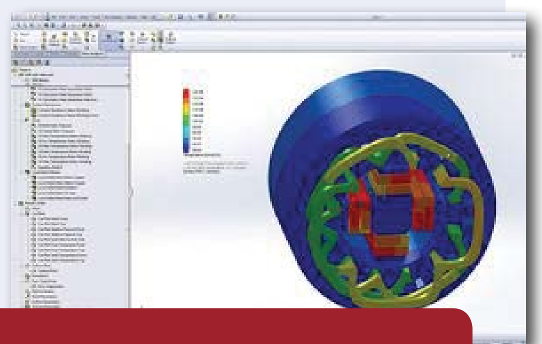
GET STARTED NOW

Cooling of Electrical Machines

ON-DEMAND WEB SEMINAR

You will see how FloEFD can handle the flow and thermal calculations of complex geometries like a large electrical motor.

Duration: 26:36



VIEW ON-DEMAND WEB SEMINAR

For more Information

E-Mail: questions_mechanical@mentor.com

Web: www.mentor.com/mechanical

Phone: 1-800-547-3000

Chat:

Brenda:
Chat with the experts at...

http://www.mentor.com/shared/live_chat?path_context=%2Fproducts%2Fmechanical

You:
Type your message here

Connect:

- Facebook
- Twitter
- YouTube
- LinkedIn
- Google+