Vol. 01 / Issue. 01 / 2012

# ENGINEERING EDGE

## Accelerate Innovation with CFD & Thermal Characterization


### Turn It Up:
Thermal Simulation Design for Automotive Multimedia Systems
Page 8

### How To Guide...
Your complete guide to 1D-3D CFD Simulation
Page 14

### Check, Check and Check Again:
The Babcock Marine mantra when it comes to system behavior
Page 12

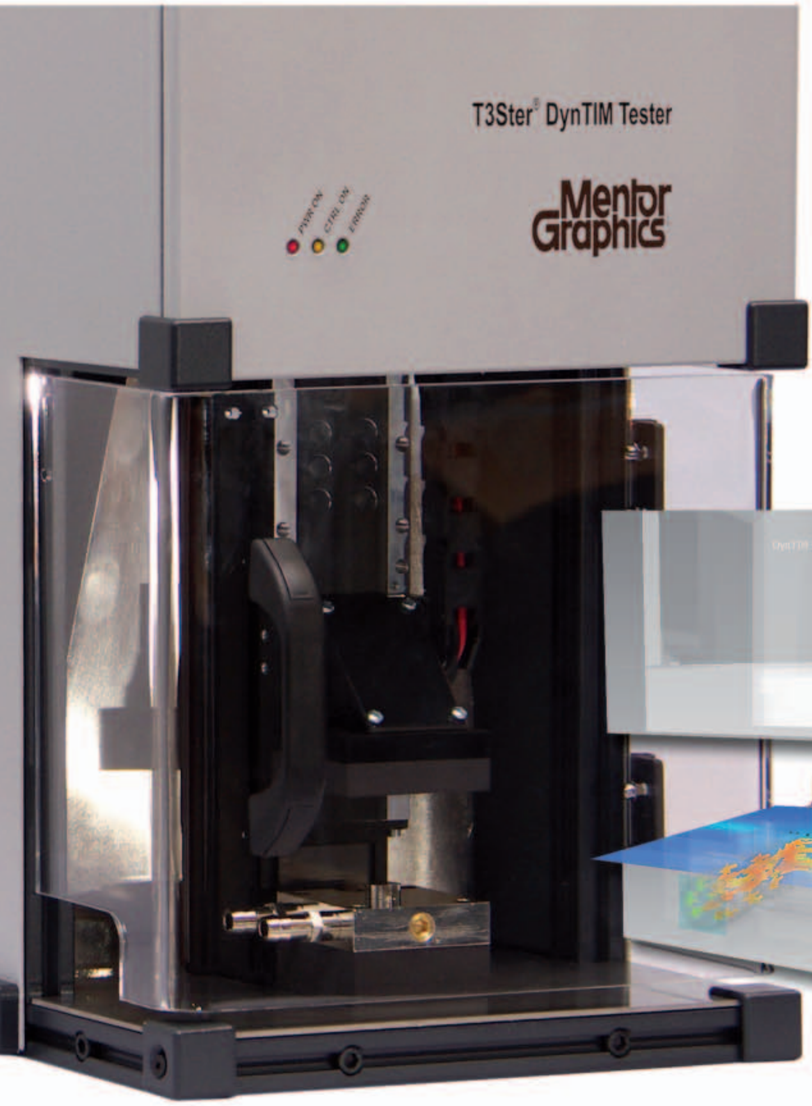
### Small but Mighty:
Micro-Turbine Jet Engine simulation and structural analysis
Page 28

Mentor Graphics®
Mechanical Analysis

mentor.com/mechanical

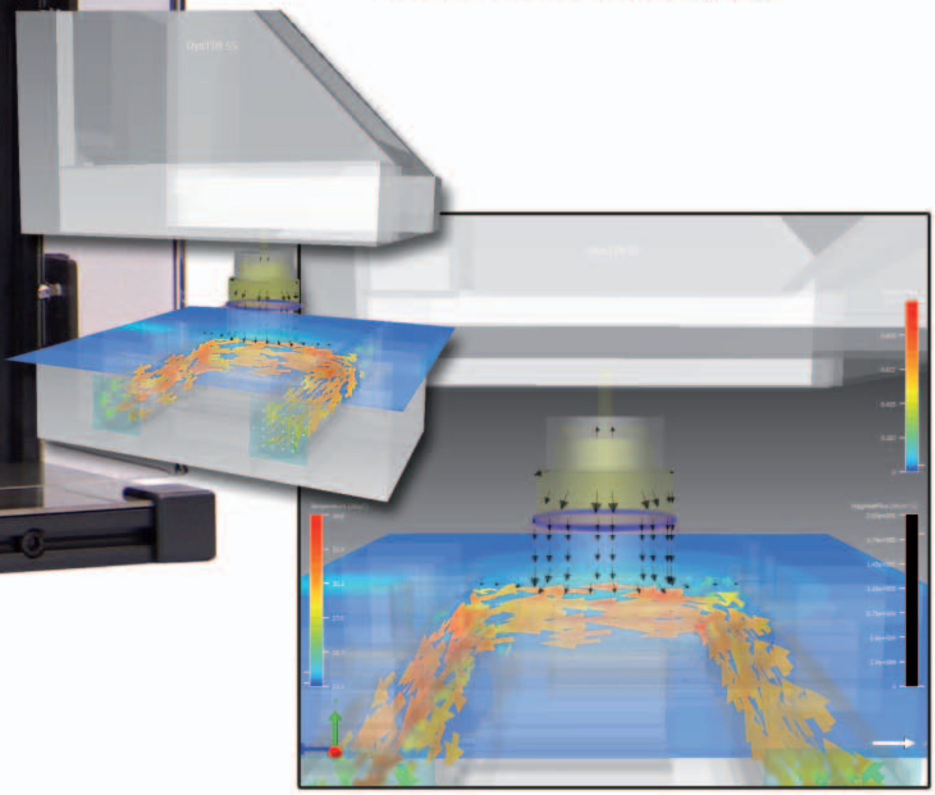
# Introducing the New T3Ster® DynTIM™ Tester

**Delivering Unrivaled Thermal Interface Material (TIM) Measurement Methodology**




**Accurately Measure:**

- ✓ Thermal Greases
- ✓ Gap Pads
- ✓ Phase Change Materials
- ✓ All other soft TIMs and
- ✓ Adhesives cured between high conductive materials to determine the impact of cure conditions and aging



**Enhanced heat transfer in Electrical and Electronic Applications**
**Advanced compliance of the ASTM D5470 standard with direct control of Bondline Thickness (BLT)**

More details on page 5


Mentor Graphics

Mechanical Analysis

**Mentor Graphics Corporation**
Pury Hill Business Park,
The Maltings,
Towcester, NN12 7TB,
United Kingdom
Tel: +44 (0)1327 306000
email: ee@mentor.com

**Editor:**
Keith Hanna

**Managing Editor:**
Natasha Antunes

**Copy Editor:**
Jane Wade

**Contributors:**
Mike Croegaert, Rob Coles, Gennady Dumnov, Keith Hanna, Alexey Kharitonovich, Doug Kolak, Boris Marovic, Travis Mikjaniec, John Murray, Sawako Ofuchi, John Parry, Tatiana Trebunskikh, Andras Vass-Varnai, Jane Wade, Richard Wilton

**With special thanks to:**
Babcock International Group PLC
Bromley Technologies
Continental Corporation
The Real World Design Challenge
The University of Stuttgart
Yaskawa Electric Corporation

**Artwork:**
Buzz Graphic Design Ltd



# Perspective

Vol. 01, Issue. 01

I would like to extend a warm welcome to you, the readers of this, the first edition of our new Mentor Graphics Mechanical Analysis Division newsletter. It will be a platform for our customers and users to share their real world applications and success stories that involve our hardware and software solutions. I also foresee it becoming a forum to articulate our unique vision and worldview for the suite of industries, markets and products that we serve globally.

Having been with Mentor Graphics now for three years, I have no doubt our market leading Computational Fluid Dynamics (CFD) software and thermal characterization tools will continue to accelerate innovation for our customers. Mentor Graphics is the only EDA company to fully encompass electrical, mechanical and software aspects in product design and development. As market forces dictate, electrical, mechanical and software design and development needs to be increasingly and inextricably combined. With further advances in this area Mentor's Mechanical Analysis Division remains well placed to deliver the tools our customers require to compete.

The common theme when I read the articles in this first edition is, I think, simulation and testing solutions for industrial and consumer applications that are driven by time-to-market, reliability and upfront design issues; challenges that must be resolved both quickly and accurately. Concurrency in our solutions is a precursor to how we at Mentor Graphics see the engineering world; our customers are designing, developing and delivering products and solutions with ever-shrinking timescales and budgets.

Many of you will be aware that our division of Mentor Graphics announced the acquisition of the Flowmaster Group of companies in January this year, along with their market leading Flowmaster™ suite of industry specific tools for 1D CFD simulation of systems. Flowmaster has a long and prestigious history itself, having come out of the work by Don Miller at BHRA in the UK in the 1970s and 1980s - see our interview with Don on page 32. Indeed, the software itself celebrates its 20th anniversary this year having been first commercialized in 1992. I believe there is tremendous synergy in the Flowmaster and Mentor Mechanical Analysis worldviews, such that we have already announced to the market a unique 1D-3D CFD coupling. (Page 14)

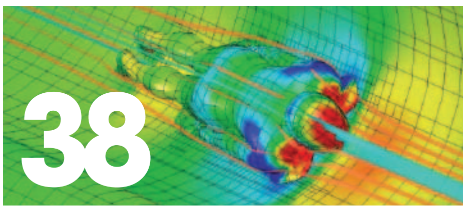
All the product lines feeding into Mentor's Mechanical Analysis offering have been based on the philosophy of freeing up engineers, designers, analysts and testers to focus on what they do best; that is, using their skills to add value to their company's innovations in terms of product design and development. I believe that we have set the pace and direction in electronics cooling, thermal characterization, thermo-fluid systems and upfront 3D CFD with our concurrent design solutions. We aim to fit seamlessly into your design processes and workflows. I therefore commend to you "Engineering Edge" and encourage you to read it, share it and tell us your story.

Erich Buergel

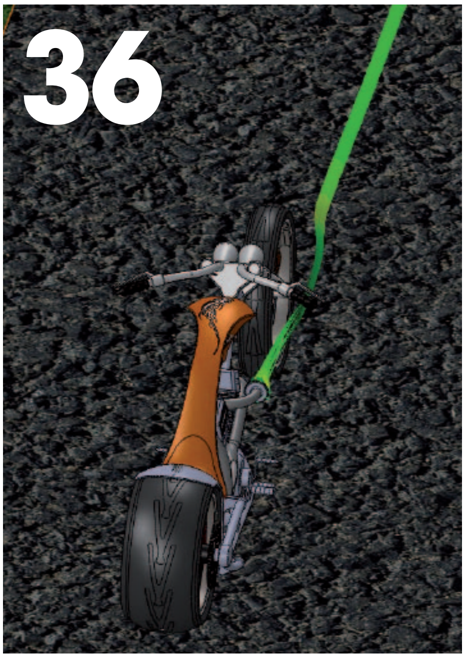
Dr. Erich Buergel, General Manager,
Mechanical Analysis Division, Mentor Graphics

# Contents



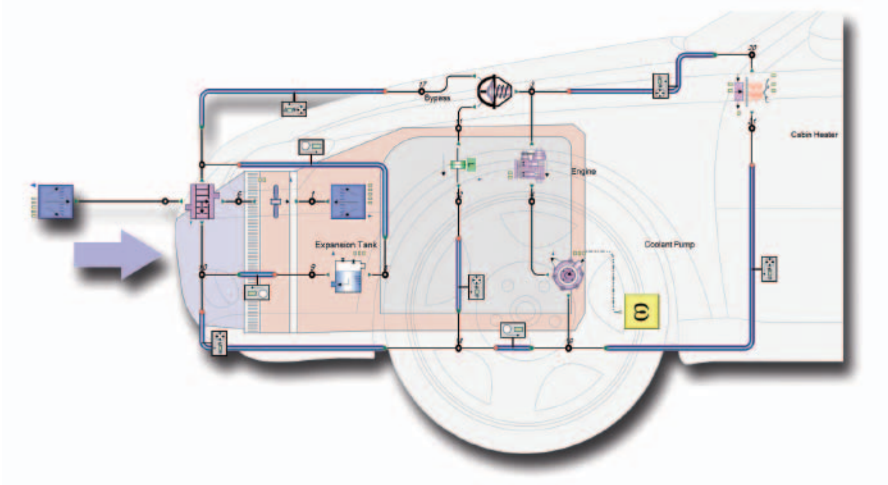


# Mentor Graphics Acquire Flowmaster

At the beginning of 2012 Mentor Graphics acquired the Flowmaster Group to extend the Mechanical Analysis Division product line. Flowmaster, a UK-based organization, produces a 1D Computational Fluid Dynamics (CFD) simulation software for system design.

The first commercially available 1D-3D system simulation tool is now available as a result of this acquisition. The Flowmaster product has a strong customer base in Automotive, Aerospace, Power Generation and Process industries complementing FloTHERM and FloEFD, Mentor Graphics' 3D CFD tools. More on page 14.




"The acquisition of the Flowmaster Group has been a great success, seeing the 1D-3D simulations in action and seeing the cross-pollination of best practise across the organization have been highlights for me this year." **Erich Buergel, General Manager, Mechanical Analysis Division.**

# TIM Characterization now possible with DynTIM

Thermal Interface Materials (TIM) have long been major contributors to the thermal budget of a system and their proper selection can be the key to a successful design. Rising to this challenge, the industry has developed a set of advanced nanoparticle based materials. As a result many highly conductive thermal greases and pastes, phase change materials, thermal pads and gap fillers have flooded the market. The current steady state standard for TIM characterization, the ASTM D5470-06 method is used in many labs due to its simplicity. However, the reproducibility of results among different measurement sites is known to be poor. As an answer to the market's requirement for a high precision TIM tester capable of measuring advanced material with good repeatability, we have created the DynTIM, an enhanced version of the current market standard. DynTIM is cutting-edge thermal interface material test equipment based on T3Ster®, the world's leading semi-conductor thermal testing solution. Materials are tested in a realistic environment, between a real diode package and a nickel plated copper cold-plate. The thermal conductivity of samples is calculated based on the change of the thermal resistance of the TIM as a function of its thickness. This idea resembles the ASTM standard; however the measurement of the temperature is carried out at one location only, at the junction of the semi-conductor diode in the top grip. This fine temperature measurement (0.01°C temperature resolution) and the calculation of the applied power based on accurate electrical parameters is responsible for the high repeatability of the testing solution. Whilst several hundred per cent differences between TIM datasheet values and real thermal conductivity data is possible in the industry, DynTIM can provide accuracy within 5% even in the case of highly conductive TIMs or metals. DynTIM combined with T3Ster allows quick and reliable thermal interface material selection for a number of possible applications. By measuring the real thermal conductivity data, DynTIM helps to identify incorrect datasheet values, providing appropriate input for simulations and making the thermal design safe.

# FloEFD™ v12.0 New Release

Available in November 2012, the latest release of FloEFD v12.0 offers extensive increased functionality and design process speedups for designers, mechanical engineers and Computational Fluid Dynamics (CFD) analysts. Exciting improvements to the automated geometry cleanup and mesher have seen a speed increase of up to 10 times for industrial benchmarks, ensuring that analysis can begin sooner and less workflow processing time is wasted. The best upfront CFD software in the world has just got much better.

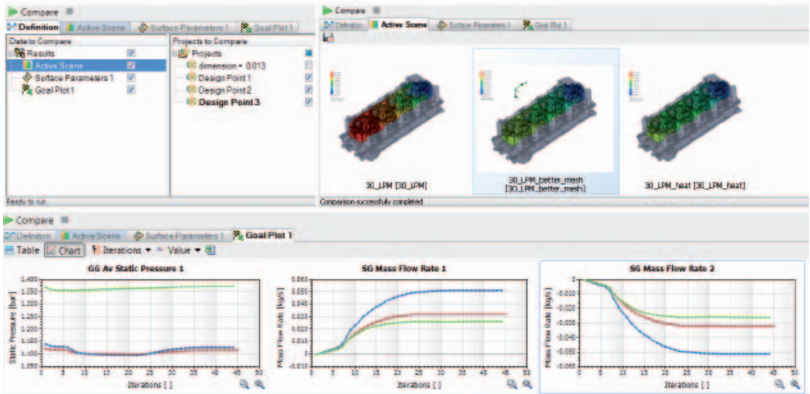
The latest release of FloEFD also includes a Parametric Study tool. This increases the speed of parametric analyses, such as varying boundary conditions and mesh settings, supported by a new Compare Tool that makes identifying the best design variant much faster. Users can compare plots and numerical data across multiple projects. This combination of features provides a very effective platform for performing 'what if' studies in the shortest possible time.

This new Compare Tool coupled with the ability for FloEFD 12.0 users to run multiple meshers and CFD solvers on a Linux cluster means greater and more effective "what if" analysis is now possible. CAD-embedded concurrent CFD practitioners will no doubt find this a powerful tool. A new 'Compare Configuration Tool' in FloEFD allows users to compare complete projects, plots or data with the ability to also save the relevant images. The resultant Comparison Report allows for easy reading with minimum and maximum, show absolute and relative values highlighted.

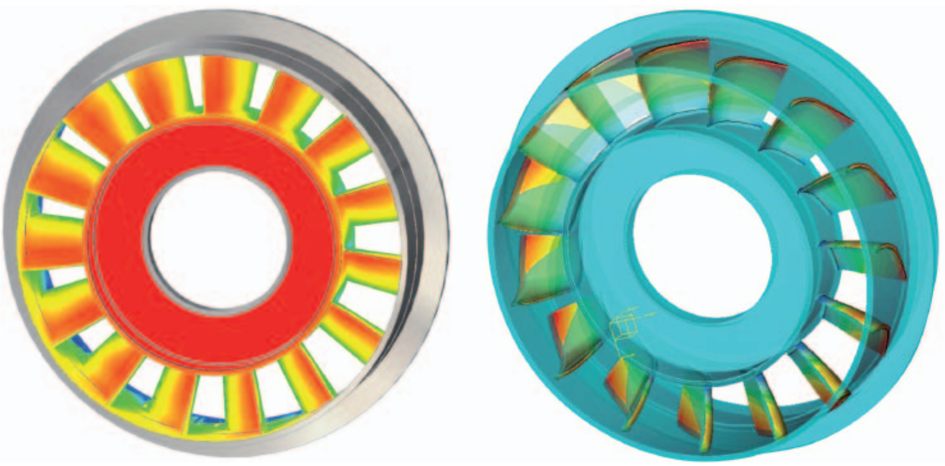
A unique new 'Dynamic Streamlines' feature in V12.0 consisting of "Line Integral Convolution" and "Evenly Spaced Lines" options will dynamically help to highlight flow structures that are difficult to see using standard visualization techniques. The number of streamlines shown are automatically refined when zooming, both speeding and aiding results interpretation. This novel GPU based technology renders evenly spaced surface streamlines in interactive real-time, so that users can zoom in and interpret more details intuitively.

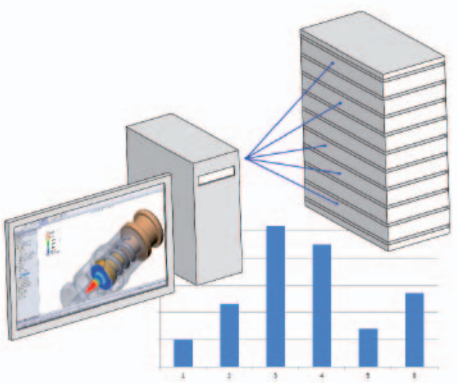
V12.0 includes a solution to the challenge of designing Solid State Lighting (SSL) systems, with a unique LED compact model approach, based on extended real product measurements from T3Ster™ TeraLED™. A "starter pack" of pre-characterized popular LED compact models is provided with the FloEFD 12.0 LED Module covering typical automotive lighting, back lighting, street lighting and general lighting applications. It gives users the ability to uniquely calculate heat power and output luminous flux for the given electrical current value for LEDs and LED arrays in situ. The module has the ability to run transient simulations and users' own LEDs can be added to the extensible database. The LED Module allows for the ability to input what is typically known (forward current) and output what is needed (hot lumens) for optical predictions.

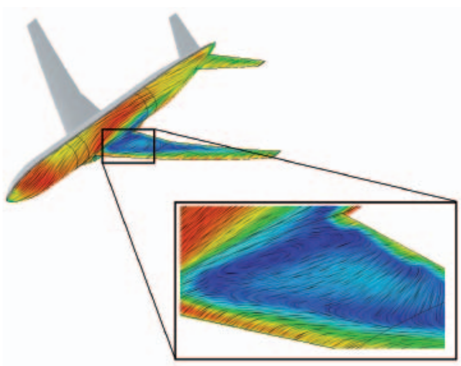
FloEFD 12.0 also now has a new generic structural analysis interface that allows the output of pressure and temperature data from the CFD predictions into a NASTRAN-based solver for fluid-structure multi-physics simulations.



Easy comparing CFD predictions for different analyses

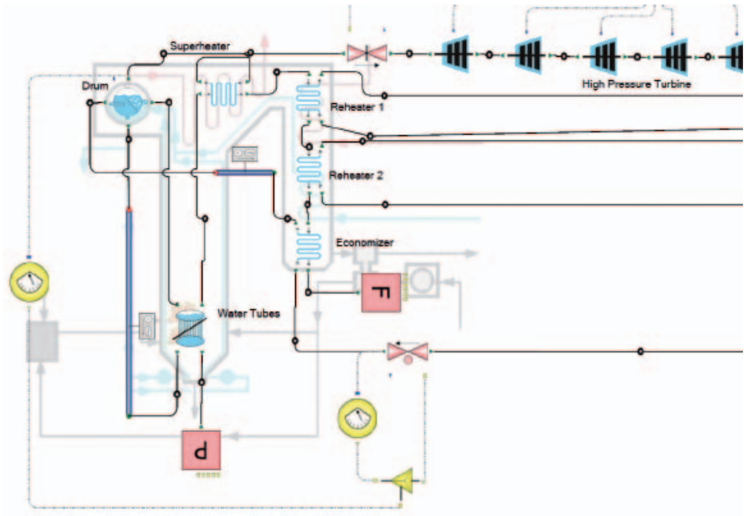

NASTRAN Bridge Calculations


Leverage Linux Cluster for Faster Calculations of Multiple Tasks


Dynamic Streamlines

# Power & Energy remain key focus for Flowmaster V7.9

Energy demand is unrelenting globally and Flowmaster continues to offer 1D simulation solutions to help the industry. The latest release, V7.9, offers the ability to handle two-phase steam and new components for fuel systems, steam systems and cooling water. This enables users to model the entire fluid system of the power plant and consider the interactions of the individual components of the system. Improved accuracy is critical: a 0.5°C improvement can mean a cost saving of $35m for the power plant.




# Mentor Graphics Mechanical Analysis Division in the News

## Top ten as voted for by you!

www.eetimes.com
June 2012

A design to generate UAV electrical power in flight, an article by Guy Wagner, Electronic Cooling Solutions, and Travis Mikjaniec, Mentor Graphics, was selected in the top ten most viewed articles on How to Design on Military Aerospace Designline. A team of engineers at Electronic Cooling Solutions worked with John Langley and engineers at Ambient Micro to build an exhaust-heat thermoelectric generator that can be incorporated into a UAV design to harvest and convert this waste energy into electrical power in flight.

## T3Ster™ TeraLED™ leads the way in Thermal & Radiometric Characterization

http://www.elektronikjournal.de
March 2012

The measurement of thermal and radiometric characterization of LEDs is crucial to organizations meeting industry regulations and guidelines. T3Ster TeraLED is proving indispensable in classifying LEDs to meet new JEDEC and JESD standards.

## Optimizing Fluid System Design with early CFD

www.designworld-digital.com
August 2012

Mentor Graphics couples its 1D and 3D tools for component and system level design. The first Upfront and Concurrent CFD tool on the market is set to change the way mechanical design engineers work. More on page 14

## Dr Erich Buergel's viewpoint in Mil & Aero Electronics

www.militaryaerospace.com
October 2012

Computational Fluid Dynamics (CFD) holds promise and delivers time- and cost-savings for aerospace and defense companies and engineers. Dr Buergel discusses how CFD should fall into the development cycle, comparisons to other CFD methods and why mil-aero executives should consider implementing CFD.

## Mentor Graphics 3rd in Top OEM Employers

www.electronicdesign.com

Mentor Graphics Corporation is ranked 3rd in top OEM employers and $7^{th}$ overall in the Electronic Design's top 50 electronic design employers. More impressively the MGC was described as their "Cinderella Story" having jumped 80 spots from its previous position of $87^{th}$ to $7^{th}$ in just one year.

# Turn it up!

## Thermal Simulation for the Design of Automotive Multimedia Systems

by Dr. Uwe Lautenschlager
Continental Automotive GmbH

Continental's Automotive Group, a system supplier for international automotive OEMs, has three divisions: Chassis & Safety, Powertrain and Interior. The division 'Interior' is itself split into four business units: Instrumentation & Driver HMI; Infotainment & Connectivity; Body & Security and Commercial Vehicles & Aftermarket.

The business unit Infotainment & Connectivity covers Radios, Connected Radio & Entry Navigation, Multimedia Systems, Telematics, Device Connectivity, and Software & Special Solutions, with a particular emphasis on Entertainment, Information, Navigation and Communication.

One of the greatest challenges for automotive multimedia systems is that they have to operate under a wide range of environmental conditions:

- Temperatures ranging from -40°C to +100°C
- Wide range of power supply voltages (-4.5 to 16.0 V)
- Superimposed alternating voltages
- ESD polluted environment (ESD = Electrostatic Discharge up to +/-15 kV)
- Full range of humidity (0% - 100%)
- Chemical influences

And with stringent design criteria:

- Mechanical strength and stability (static / dynamic): Vibration & mechanical shock
- Strong limitations on size and weight
- Strong requirements regarding risk of injury (Head Impact) and product liability

### A World Away from Physical Prototyping

Designing systems to meet such requirements requires much more than physical prototyping after the design is complete. Indeed, Continental's simulation vision is to 'Get the Product Right the First Time', and their strategy to achieve this has been through Simulation-Based Design Decisions. As a result, simulation is now very highly integrated into the design process.

Continental undertakes full 3D system-level modeling, with all thermally-relevant and air flow parts included in the simulation. The geometry representation is based on mechanical CAD data, with export/import of all relevant parts and modules from the CAD-System. However, the geometry representation can involve both simplification and idealization. The complexity used depends on the simulation objective: less detail would be used for the analysis of various initial concepts, whereas more details would be included in the model for a mature design.

### "Know-How" – Continental's Competitive Edge

Continental has developed considerable in-house 'know-how' associated with building and validating thermal simulation models (FloTHERM™ models) of their complex electronic systems and sub-systems, covering both the physical hardware, and potential cooling solutions.

**Hardware-related**

- Housing: 1-DIN, 2-DIN, customer-specific
- CD-, HDD-, DVD-Drives (ROM, Video), Changer
- Shielding frames
- Front displays (CCFL-, LED-Backlight)
- Connectors and cables
- Components: Main-CPU, Processors, Memory
- PCBs
- Modules: Power Supply, Drives, GPS, Telephone, Amplifier, Tuner, Display, etc.

**Cooling solution-related**

- Fan (type, characteristic curve, direction)
- Contact resistances (conductive paste, etc.)
- Vents, filters (free area ratio, pressure loss)
- Heatsinks, heat pipes, heat spreaders
- IC Packages (type, size, thermal model type)
- Thermally conductive gap pads, etc.
- Environmental conditions
- Power dissipations
- Location of sensors for measurement

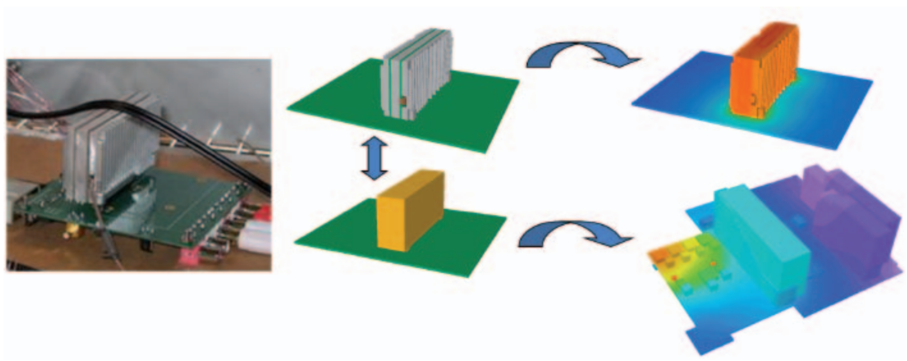
Uncertainties arising from the various unknowns at each stage in the design process, modeling assumptions and simplifications are mitigated through validation, as the following two examples illustrate.

### Examples:
### 1. Tuner-Module Modeling

This can be modeled as a single block with homogeneous power dissipation distribution, as a detailed model with all of the internal electrical components and shielding represented, or anywhere in between. In Continental's experience, a simplified model is often sufficient within a complete system analysis. However, conclusions on the internal component temperatures are drawn from a reference simulation as depicted in Figure 1.

### Example:
### 2. Chassis Model Verification

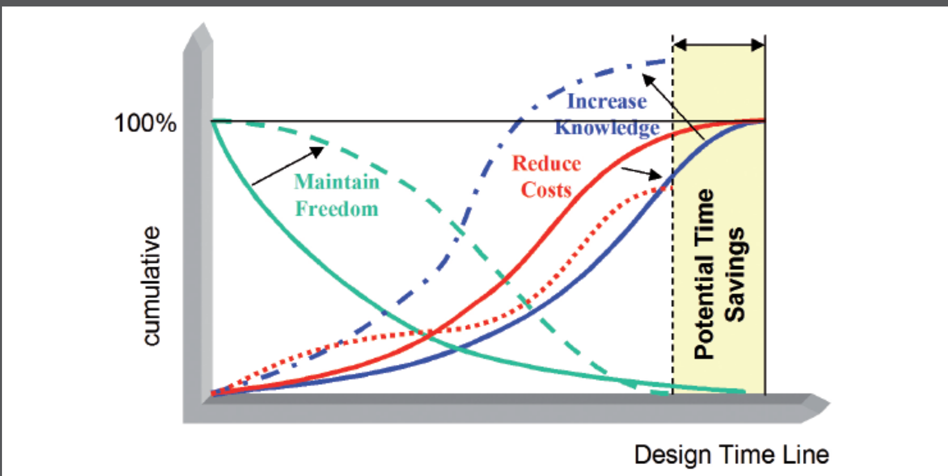
A part of the chassis is used as a heatsink for amplifier cooling. An important aspect of the chassis is the extent to which the brackets on the top and bottom covers are in thermal contact (related to manufacturing tolerances). Verification of modeling assumptions is the key to improving model fidelity. Therefore, a prototype of the product was measured and the results used to compare against the simulation model. Scenario 1 (g1) considers the gap to be fully closed (perfect thermal contact) whereas Scenario 2 (g2) has a gap of 0.3mm, so there is effectively no thermal contact between the covers. By comparison with the measured case temperature the results for Scenario 2 (with gap) seems to be the best representation of the bracket's contact.



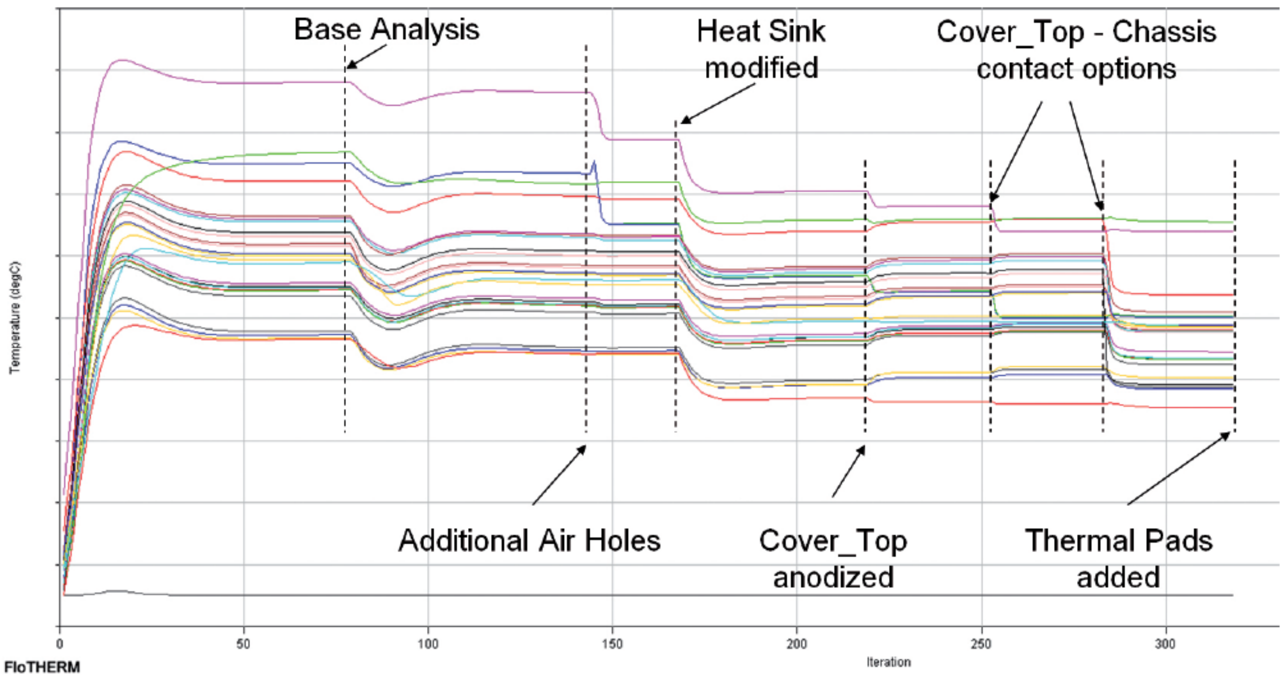
Single Block (System-Level) and Detailed (Reference) Representations of Tuner Module


Multimedia Systems with Audio, Navigation, Internet, Telephone, TV & Rear-Seat Entertainment Functions

"FloTHERM is a key component of our Simulation-Based Design Decisions strategy, ensuring that our thermal design goals are met and we can deliver on Continental's simulation vision of Getting the Product Right the First Time." **Dr Uwe Lautenschlager**



“We selected FloTHERM™ for several reasons but primarily for its robust solution capabilities. FloTHERM’s object-associated Cartesian meshing is instantaneous, fully automatic, and most importantly guarantees a mesh that produces accurate simulation results even when geometry changes are made to the base model. This is something we absolutely need for our Multidisciplinary Design Optimization (MDO) activities. Not to mention its modeling and result evaluation capabilities that in total simply made it the best solution for us.” Dr Uwe Lautenschlager




Sequential Concept/Design Improvements from MDO for Multimedia System

## Beyond High Fidelity Simulation

Continental’s focus on simulation does not stop at building and validating thermal models. Design decisions made to improve the thermal design can impact the mechanical, electrical, and EMC performance of the product. Faced with this problem, a major question facing Continental’s designers is: “How can we achieve design flexibility and enable better design decisions before the freedom for such decisions is eliminated?”

The answer to the challenge requires analysis of the product’s behavior for all disciplines as well as the identification of independent and coupled system variables. Multidisciplinary Design Optimization (MDO) techniques with simulation, optimization (with discipline-dependent objectives and constraints) and Design-of-Experiments (DOE) & Response Surface Methods are suitable means to solve this design problem. Indeed, DOE tools are the foundation for concept exploration and robust design. Two major aspects of DOE are the planning and statistical analysis of the numerical simulations.

The possible number of discrete and continuous design variables is extremely high. Screening simulations support the selection of the major factors (design drivers) to be included as design variables in the MDO. Even so, the computational effort required can be immense.

## Sequential Design Improvements

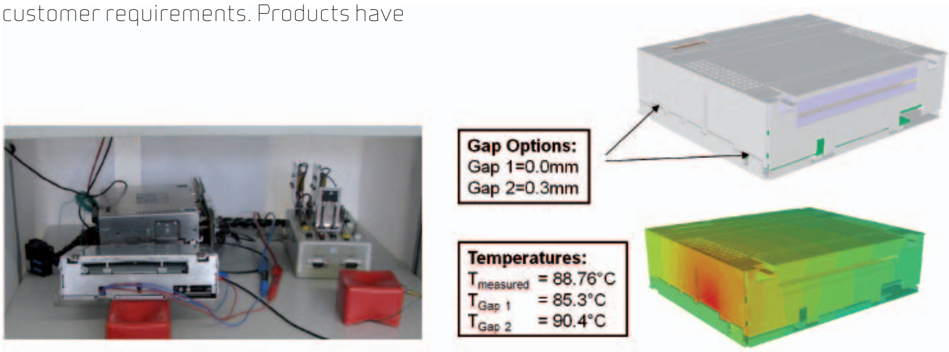
Changes that improve the overall design, i.e. the compromise between the product’s mechanical, electrical, and EMC performance, and additional constraints such as temperature limits, manufacturability or cost allow sequential improvements to be made. For each discipline these improvements can be visualized as improvements against the base design. In the case of the thermal design, these can be visualized as changes in monitor point temperatures that match the locations chosen for sensors that will be used to instrument the physical prototype.

## Business Benefits

Modern radio and navigation systems, i.e. multimedia systems, for automotive


applications are highly complex systems with a large variety of mechanical and electrical components and assemblies. The design of these automotive multimedia systems has to fulfil requirements from mechanical stability and thermal management to electromagnetic compliance and optical homogeneity and therefore requires the interaction of mechanical, electrical and software engineers. Interdisciplinary knowledge is essential.

Continental has to react quickly to changing customer requirements. Products have to be developed with regards to customer confidence and quality, costs and design time. MDO has been found to be a suitable means for finding better design solutions in a multidisciplinary environment. Simulation supports knowledge generation and a deeper understanding of product behavior at lower cost, in shorter time and with increased product flexibility, leading to increased customer confidence in our products.




Verification of Case Thermal Contact Modeling Assumption

# Check, Check and Check Again!

The age old axiom "measure twice, cut once" is as true now as it ever has been.

**PERHAPS the biggest change in modern times is that "measuring" in the engineering context can now cover a range of techniques, all to provide the engineer with an understanding of the performance of the system. In a competitive environment where the drive to reduce through-life costs often goes hand-in-hand with a requirement for increased efficiency, the consequences of not having a thorough and detailed understanding of system response before you've made the 'first cut' can be severe.**

Here the engineers at Babcock Marine demonstrate that applying modern computational techniques affords the opportunity to understand system behavior at a detailed level without the need for expensive instrumentation and tests. It can be employed upfront or, as Babcock must do here in support of the UK's Ministry of Defence (MoD), after production.

Babcock Marine is the UK's premier naval support contractor. It owns and operates both Devonport and Rosyth Royal Dockyards and manages the Clyde Submarine Base at Faslane. It is the sole support partner for the Royal Navy's submarine force and is a major partner in the provision of support to the current surface fleet and to the future aircraft carrier program. Flowmaster® is used within Babcock Marine's Design and Technology Mechanical Engineering Group, which offers front-end design, analysis and through-life support solutions. Babcock Marine was contracted by the MOD to investigate and propose solutions to frequent component failures within the Sea Water fire mains systems onboard both the HMS Albion and HMS Bulwark vessels.

To carry out the required number of system tests as fast and efficiently as possible, Babcock Marine utilised Flowmaster's integration capabilities by linking the software with Microsoft® Excel to cleverly drive multiple real-world testing line-ups and immediately extract results from the Flowmaster model, all via one front-end diagrammatical spreadsheet.

"The ability to integrate Flowmaster into a front end spreadsheet program, enabled us to provide our engineers with a fully capable and adaptable simulation package, able to be rapidly and easily tuned to real-world onboard operating line-ups." **David Millar, Senior Design Engineer.**

## The Challenge

The MoD were experiencing multiple pump and pipe work failures within the Sea Water fire mains on both the HMS Albion and HMS Bulwark vessels. These failures were attributed to various causes:

- Very high cost of maintenance on pumps
- Frequent mechanical seal and motor bearing failures
- Inability to meet system design requirements

Various small targeted initiatives to try and improve system performance were carried out, however it was recognised by the Marine Auxiliary Systems Integrated Project Team (MXS IPT) that a more joined up systematic, system wide approach was required to address the root cause of the problems.

## The Solution

Babcock Marine's Mechanical Engineering Group (MEG) were contracted through MXS IPT HESS (Holistic Engineering Support Strategy) Pumps, to model the entire HPSW (High Pressure Sea Water) system on both HMS Albion and HMS Bulwark to diagnose problems and recommend solutions.

## Design and Integration

The size of the ship's system (5 Fire, 2 Normal Demand and 2 Emergency Pumps connected into a ring main with numerous cooling demands and fire fighting spray systems) meant that Babcock Marine had to develop a custom user interface designed to mimic the look and behavior of the current ship's onboard system management software. This custom interface was developed in Microsoft Excel and fully integrated with Flowmaster, enabling service engineers to run simulations and view results through a familiar software environment. The activation of pumps and valves could be simulated through the spreadsheet by clicking on symbols. The valve and pump status was automatically updated in Flowmaster when the simulations were run.

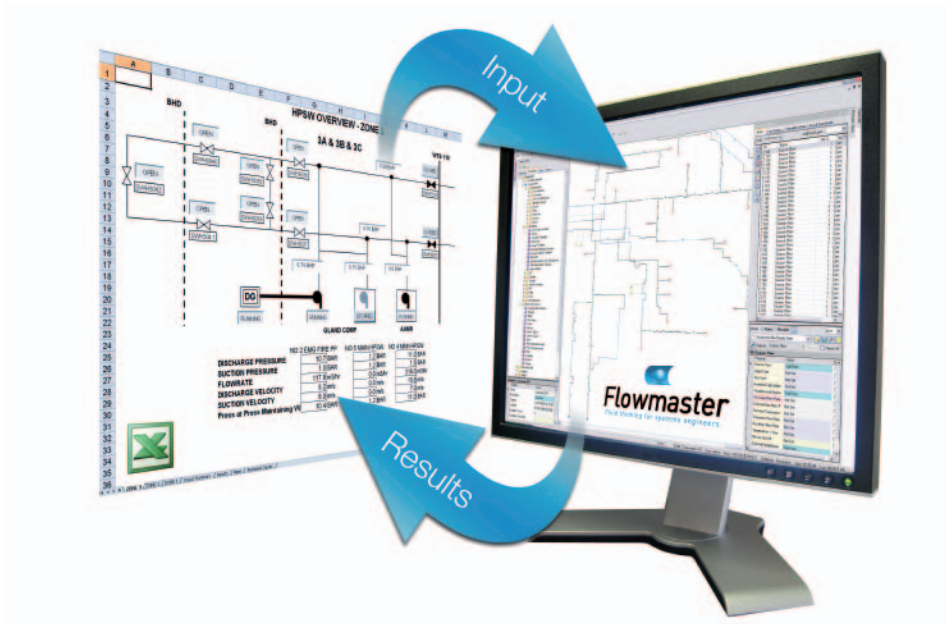
To improve efficiency, certain systems were grouped together so for example; all fire fighting sprays could be turned on in a certain zone through the click of one button, enabling the operator to model different operating scenarios. The integration of Babcock Marine's user interface enabled engineers to validate the system designs in Flowmaster quickly and easily. Aspects, such as the effects of the pump degradation and defective pressure control valves could be quickly ascertained and decisions as to whether the system would perform under ideal conditions and actual "worn" conditions could be made with confidence. To add clarity to the results, pressure readings were automatically extracted from the Flowmaster model at the actual positions of the onboard remote pressure indication. These readings were superimposed onto the model diagrammatic to give direct correlation with the readings visible on the onboard control programs.

## Onboard Simulation and Validation

To measure the performance of the Flowmaster model, a true life level of demand needed to be established. Extensive onboard testing using ultrasonic flow measurement and various system line-ups were carried out to record the actual flow demands on the cooling systems. Flowmaster together with Babcock Marine's spreadsheet based user interface was used onboard to run 'what if' scenarios which were then replicated in the actual 'Live' system, with the predicted results compared to the actual system performance. These tests proved invaluable in building up an understanding of system performance requirements, the reasons behind the problems being experienced and ultimately enabled Babcock Marine to recommend design improvements that would reduce any further component failures.

The impact of design shortcomings being found once the system has entered into service is obvious. Perhaps what isn't so clear is that the same techniques that can prevent this from happening can also aid in planning upgrades, maintenance, and re-fit. This project clearly demonstrates Flowmaster's ability to integrate with third party software, to enable engineers to simulate real onboard operating line-ups, utilising Flowmaster's technical capability through one simple user interface.

Taking advantage of modern tools like Computational Fluid Dynamics takes the age old definition of 'measurement' and redefines it as understanding a system and its response. What doesn't change is the importance of getting this understanding before you make the first cut.



# How To...

## Your complete guide to 1D-3D Simulation

By Doug Kolak, Product Marketing Manager, Mentor Graphics

**OVER the years, Computational Fluid Dynamics (CFD) software solutions have been used successfully for modeling thermo-fluid systems in the automotive, aviation, oil, gas, power and energy industries. Both 1D and 3D CFD enable engineers to improve understanding of fluid flow and engineering designs; and in many organizations, both are used to improve product and system design and to ensure performance targets are achieved throughout the operating cycle of interest.**

When engineers have access to multiple options for CFD analysis, the question arises, "When to use 1D CFD versus 3D CFD?" While there is not a definitive answer, the strengths and weakness of each approach lend themselves to two fairly well defined segments.

When designing a single component or small sub-set of components, every inch of length or degree of curvature can make the difference between efficient operation and undesired fluid flow. In these cases, where small changes to a single part of a system are crucial or there are significant flow variations in multiple dimensions, 3D CFD is the obvious choice because of its ability to analyze complex geometry with extreme accuracy. However, with these benefits come drawbacks, which become more evident as the scale of the design increases. When the design reaches beyond the component level, the computation requirements can become too high and simulations take too long to fit within development schedules.

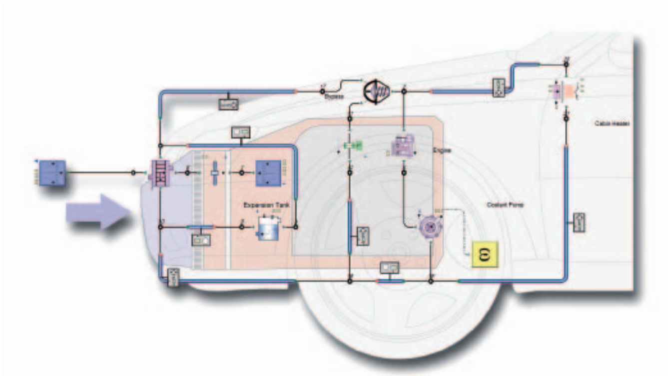
When this occurs, 1D CFD is a good choice because the 1D approach simplifies the 3D geometry to the component level. Usually characterized by some sort of empirical data, this type of analysis uses much less computing power and is significantly faster than a comparable 3D model. One of the biggest challenges with 1D CFD modeling is getting performance data that can adequately define the 3D geometry of the component at the system level. This has historically been done in one of several ways, including data from a supplier, physical testing, or empirical data from text books that is often available for standard geometries such as bends and junctions.

Although these methods are adequate, it can often be time consuming to wait for a supplier to provide data; and one of the main goals of virtual prototyping is to understand the system before physical testing.

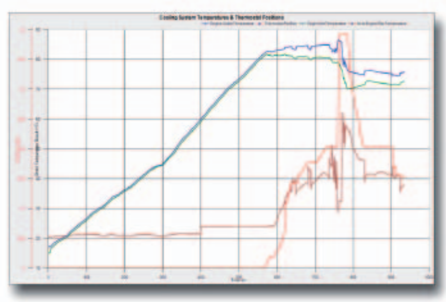
### Coupling 1D & 3D Provides The Best of Both Worlds

**As a result of these challenges, both systems engineers and solution providers are exploring methods for using 1D and 3D CFD together. Mentor Graphics provides a tightly coupled general-purpose 1D-3D CFD simulation software combination. With this combination, engineers can characterize the more complex elements of the system with full 3D and easily insert those models into the 1D system-level model for simulation. Using 1D and 3D CFD tools with a characterization strategy certainly is not a new concept; however, it is the method with which the 1D and 3D tools interact that adds value to the design process.**

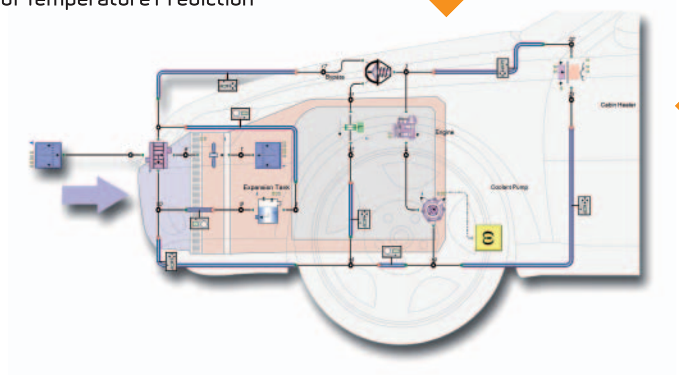
The first step of the process is to characterize the 3D geometry so that it can be used in the system level model. The expected range of conditions that the



Flowmaster™ Design of Cooling System



Improved Accuracy of Temperature Prediction



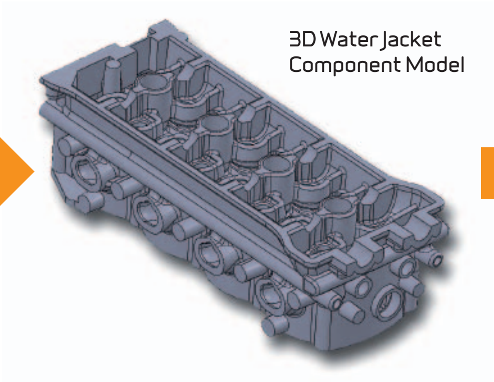
system will likely experience is usually understood by the engineer and this information is the basis for defining a design of experiments to run the 3D CFD model through. By utilizing a Parametric Study tool such as the one available in FloEFD™, the experiments can be configured once and the software left to run on its own. It is important to include a sufficient number of runs so that the characterization of the results is representative of the component's behavior. When all of the analyses are complete, the results are saved for use inside of the 1D tool Flowmaster®. Eliminating the manual transfer of data frees the designer from the tedious and error-prone tasks of extracting all of the required results including temperatures, pressures, flow rates, and fluid properties for each result set and entering it in a format that can be used by the systems engineer.

Once the systems engineer receives the characterization file, it can simply be opened as a new component inside the 1D CFD program. The pressure loss and thermal data should be reviewed and properly fitted before the component is saved in the simulation database. Finally the new component can be added to any appropriate 1D system model (or used with future designs), for analysis and virtual prototyping through a range of conditions.

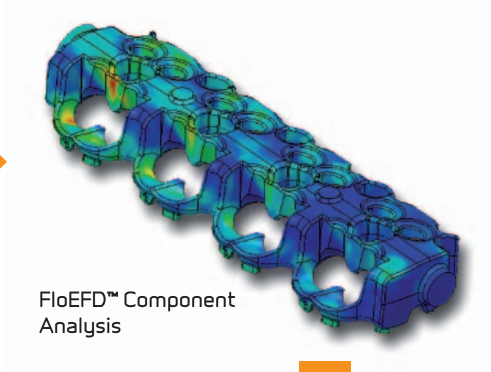
## Conclusions

**FloEFD Concurrent CFD can reduce simulation time by as much as 65–75% in comparison to traditional CFD tools. Concurrent CFD enables design engineers to optimize product performance and reliability while reducing physical prototyping and development costs without time or material penalties. With Flowmaster offering a fast and reliable solution to 1D flow problems, designers can expect unrivalled savings in both time and process compression.**

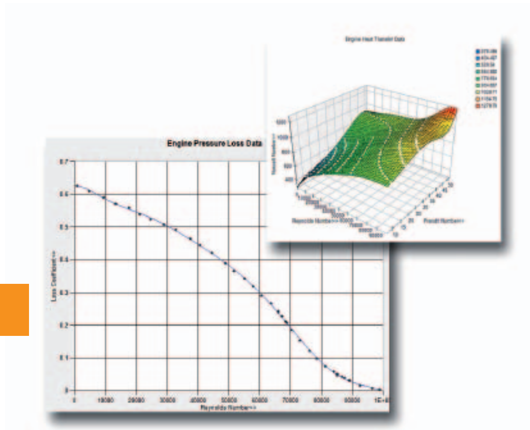
Mentor Graphics 1D-3D solution provides an opportunity to increase digital prototyping information earlier in the product development process. This kind of analysis performed upfront accelerates design and development cycles by orders of magnitude and optimizes product design workflows. It increases engineer and designer productivity and minimizes design risk and re-spins. This 1D-3D combination provides the best of both worlds, reducing time to get essential component information to system engineers and improving system-level accuracy with highly complex geometries. It will ensure the proper tool can be used at the proper time, thus shortening the design cycle.



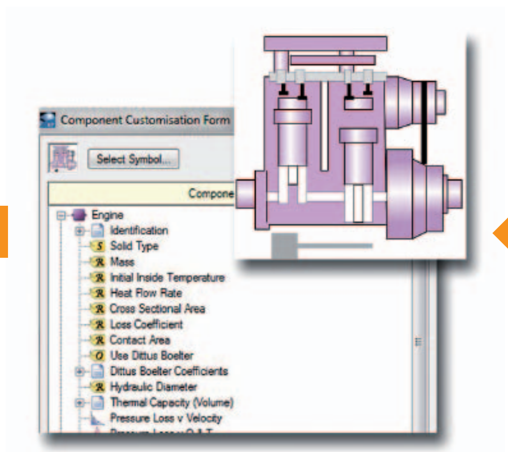
3D Water Jacket Component Model



FloEFD™ Component Analysis



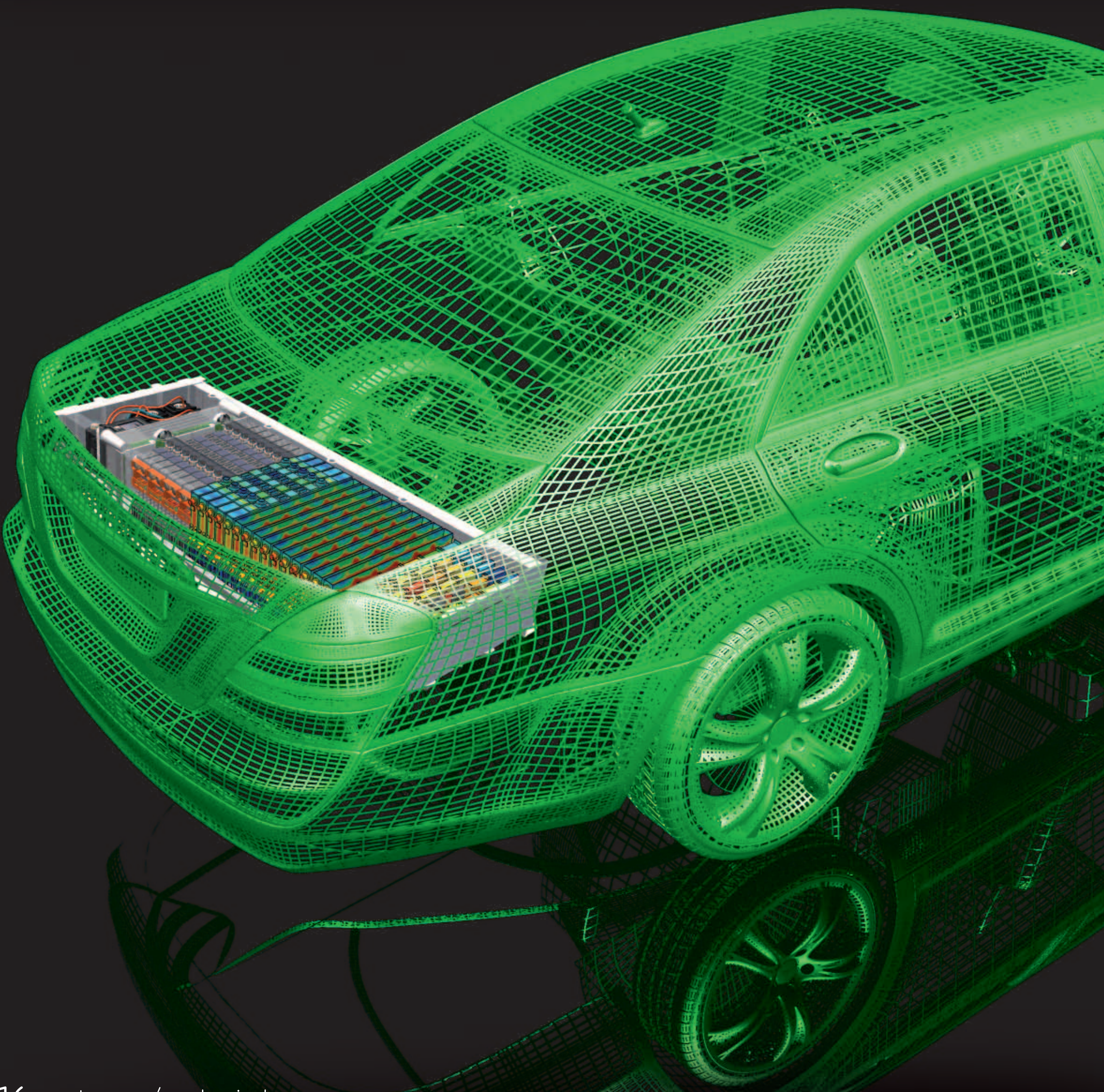

Characterization Results for System Analysis

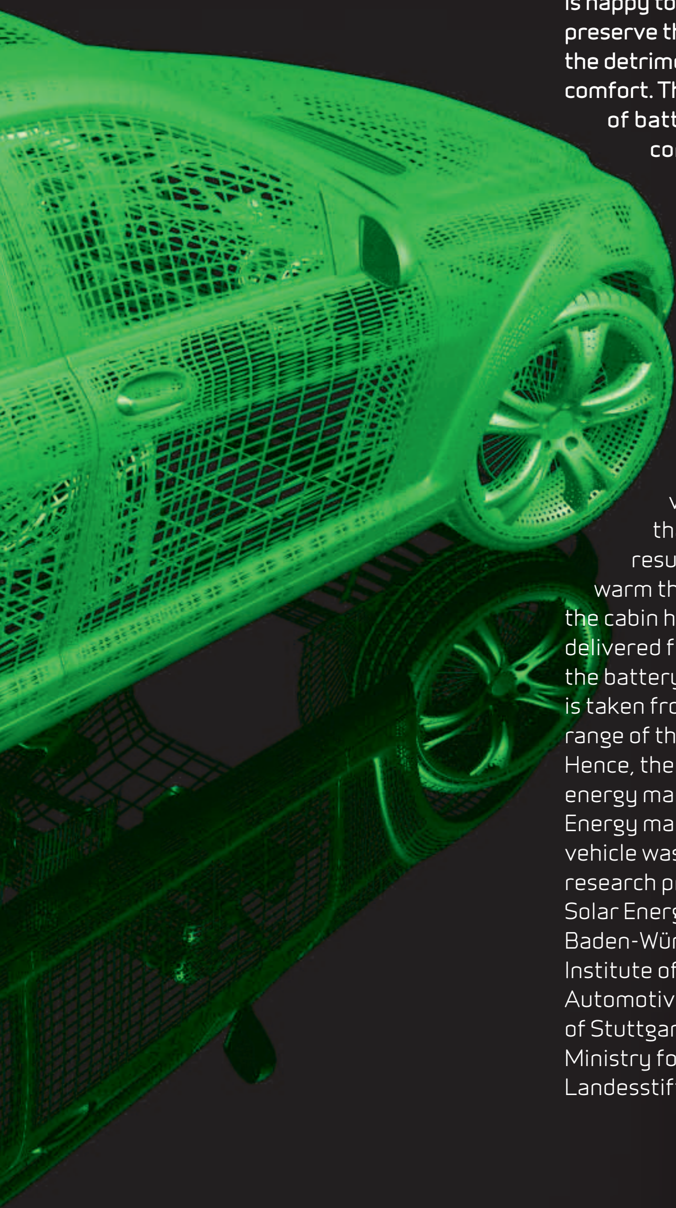



FloEFD Performance Based 1D Engine Model

# Thermal Investigation of a Battery Electric Vehicle with a coupled Matlab® Flowmaster™ Simulation

By Markus Auer, University of Stuttgart





**BATTERY electric vehicles, or BEVs, not only allow for emission free traveling but are also capable of offsetting CO2 emissions in the environment. Whilst the most environmentally conscientious driver is happy to do his duty to protect and preserve the climate, this cannot come at the detriment of speed, endurance and comfort. This article explores the influence of battery temperature and air-conditioning (passenger comfort) on the battery and the range for the vehicle. Several key factors are considered as boundary conditions. A Flowmaster model was set up and linked with Matlab/Simulink to simulate the vehicle, including a complex battery thermal model.**

By design, battery electric vehicles are more energy efficient than combustion driven vehicles resulting in much less waste heat to warm the passenger cabin. Therefore, the cabin has to be heated with energy delivered from another source or from the battery. However when the energy is taken from the battery, the effective range of the vehicle is drastically reduced. Hence, thermal management as well as energy management are very important. Energy management of a battery electric vehicle was the primary focus of a joint research project between the Center for Solar Energy and Hydrogen Research in Baden-Württemberg (ZSW BW*) and the Institute of Internal Combustion Engines and Automotive Engineering at the University of Stuttgart (IVK**). It was funded by the Ministry for Finance and Economics and the Landesstiftung Baden-Württemberg.

In order to determine the effects of different operating strategies and cabin comfort on the driving range of a BEV a simulation model was created. The work of ZSW BW focused on modeling the battery, with IVK modeling the thermal and energy management systems.

The crucial factor for the successful introduction to market of electric powered vehicles is a substantial driving range. Nevertheless, the comfort of the car is an important factor that no car owner is willing to forgo. If the energy for heating, ventilation and air conditioning (HVAC) is drawn from the traction battery, the driving range is affected severely, putting comfort in competition with the driving range for energy. To accurately predict how the driving range is influenced by the need for comfort, a predictive simulation tool is required. To begin the vehicle is modeled via a coupled simulation between Matlab/Simulink and Flowmaster. The desired driving cycle is defined by a velocity over time profile. Based on vehicle parameters, such as drag coefficient and frontal area, the necessary torque and rotational speed of the electric motor are calculated and are used to calculate the heat flux of the varying components. The derived heat fluxes as well as different rotational speeds of the electric AC compressor and electric coolant pumps are fed into the thermo hydraulic solver in Flowmaster. Based on the values and the thermo hydraulic model, Flowmaster calculates the temperatures and required electric power which are then sent back to Matlab (see Figure 1 overleaf).

* Zentrum für Sonnenenergie- und Wasserstoff-Forschung Baden-Württemberg
** Institut für Verbrennungsmotoren und Kraftfahrwesen

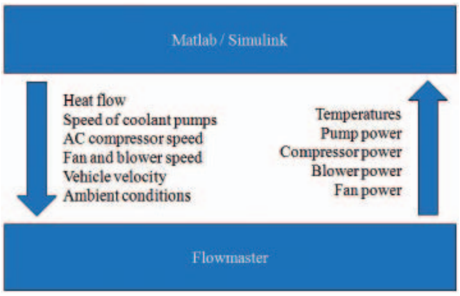



**Figure 1 –** Overview of coupling between Matlab/ Simulink and Flowmaster

As the battery needs to remain within a certain temperature range (15°C to 40°C) [2,3], a liquid conditioning system with the additional capability to heat has been chosen as part of the model. If the ambient temperature is neither too high nor too low, the battery can be cooled via a heat exchanger located in the front of the car. For higher ambient temperatures, the heat exchanger has to be bypassed and the coolant is cooled via a chiller . In case of lower temperatures, the heat exchanger is also bypassed but here the coolant is heated with a resistance heater such as Positive Temperature Coefficient (PTC). Figure 2 shows a sketch of the thermal model used in this case.

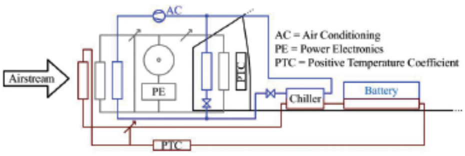



**Figure 2 –** Thermal management system of simulated vehicle

The thermal model was set up to specifically focus on the modeling of the cabin and the battery. To keep the quantity of elements in Flowmaster reasonable, the battery is discretized into 25 packs with five cells each. Figure 3 shows one of these packs within Flowmaster. The pack is modeled with its respective thermal mass and the thermal conductance in all directions. Additionally, there are cooling plates between each pack. Every pack has its own heat source which can be used to account for deviations in heat rate caused by cell to cell variation.

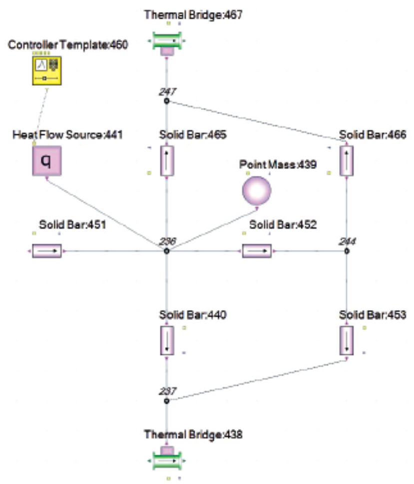



**Figure 3 –** Thermal model of one battery pack (5 cells)

Using the created simulation environment it is possible to study the energy management and its effect on driving range, passenger comfort and the relevant temperatures. To determine the consumption and comfort of the base setup, two consecutive NEDCs (New European Driving Cycle) were simulated. In order to account for the power consumption of HVAC, the ambient temperature was varied between -20°C and 40°C while the temperature control of the cabin was switched on. Figure 4 shows the temperature difference between the cabin mean temperature according to DIN 1986-3 ([4]) and the optimal temperature according to Großmann [5]. For 0°C to 30°C the optimal temperature is reached in less than six minutes. For -10°C and 40°C the requirements are met. To meet the requirements for -20°C, the maximum heating power of the concept, which is currently 3 kW, should be increased.

Figure 5 shows the range calculation based on one respective and two consecutive NEDCs. As shown, the calculated ranges differ. The cause is a reduced power consumption of the HVAC system after the first cycle (1200 s). At -20°C and 40°C there is no difference in consumption, as the HVAC consumption is not reduced between 1200 seconds and 2400 seconds. The single point at 20°C shows the maximum reachable range when only essential loads are switched on while the vehicle still has enough power to follow the velocity profile of the NEDC.

Using the simulation model it was possible to determine the effects of energy management on driving range and cabin comfort. In addition to representing the thermal network, an energy manager has been implemented, which is able to switch off loads if the demanded range is not achievable in the given circumstances. The estimate of energy consumption was improved by route recognition and temperature dependant start value estimation. The different operation strategies make it possible to account for different driver preferences regarding the energy distribution within the system.

By analyzing different drive cycles for various environmental conditions and considering the comfort of the passenger, one thing above all else, becomes the key driver in battery electric vehicle design, the driving range of the car must be substantial if it is to be competitive in today's marketplace. All these parameters influence battery performance which is why efficient battery cooling and heating, is extremely important for the range and battery lifetime. This is especially significant as the efficient working temperature range is narrow. Not to mention additional power consumers such as infotainment systems and charging of any consumer electronics, such as cell

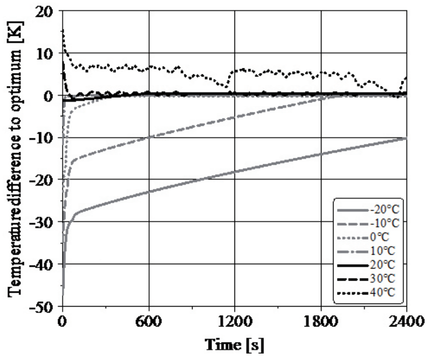



**Figure 4 –** Temperature difference of mean cabin temperature relative to the optimum temperature

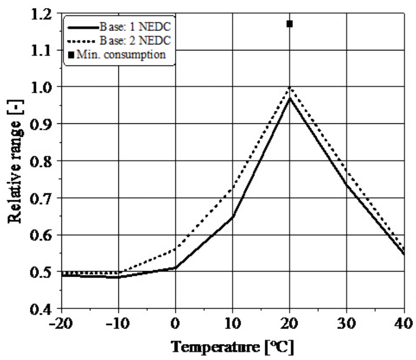



**Figure 5 –** Relative range in NEDC for different temperatures

"By analyzing different drive cycles for various environmental conditions and considering the comfort of the passenger, one thing above all else, becomes the key driver in battery electric vehicle design, the driving range of the car must be substantial if it is to be competitive in today's marketplace."

phones, which were not considered in this study but are ever-increasing in today's automotive environment. This study is an excellent demonstration of the use of system simulation coupled with a complex battery model and an energy manager in Matlab/Simulink to improve the driving range of a battery electric vehicle.

**Acknowledgement**
The authors would like to thank the Ministry for Finance and Economics and the Landesstiftung Baden-Württemberg for supporting this project and Flowmaster GmbH for providing licenses.

**References**

[1] M. Auer, M. Grimm, T. Kuthada, H.-C. Reuss, J. Wiedemann, S. Krug, "Numerical Investigation of Range Achievement with a Generic Electric Vehicle", Stuttgarter Symposium 2012, Stuttgart, 2012.

[2] Naunin, Dietrich: "Hybrid-, Batterie- und Brennstoffzellen-Elektro-fahrzeuge" Expert Verlag, Renningen, 2007.

[3] S. Edwards: "Kühlung für elektrische Fahrzeuge mit erhöhter Reichweite" In: Behr - Technischer Pressetag 2009. Stuttgart, 2009.

[4] Deutsches Institut für Normung: "Raumlufttechnik – Teil 3: Klimatisierung von Personenkraftwagen und Lastkraftwagen" DIN 1946-3, Berlin, 2006.

[5] W.-H. Hucho, "Aerodynamik des Automobils", Wiesbaden, Vieweg + Teubner, 2008, 5. Auflage, ISBN 978-3-528-03959-2.

# High School Students Fly with FloEFD™

The Real World Design Challenge (RWDC) is an annual high school competition in the USA run by a public-private partnership with the goal of assisting in the increase of the Science, Technology, Engineering and Mathematics (STEM) workforce in America.

Real World Design Challenge


Dr. Ralph Coppola, PTC, Director of the Real World Design Challenge

**EVERY year teams of three to seven high school students are asked to address a real challenge that confronts leading engineers. Often students are asked to design an aeroplane or car looking at the forces of lift, weight, thrust and drag with the goal of either enhancing performance or fuel efficiency. It is free for high school teachers and students to participate. Indeed, each participating teacher gets access to $1M worth of commercial professional engineering simulation software to use in the competition, as well as access to professional mentors. To date 45 State governors have supported the RWDC "Governor's Challenge" and following State level competitions, a national competition final is held each year in Washington, DC.**

RWDC helps to address the critical need in the United States for the development of a replacement workforce in the STEM fields, especially as a large proportion of the American aerospace and automotive engineering workforce comes closer to retirement. It is estimated that there are not enough young students with the resources and the enthusiasm to replace the baby boomer generation in these industries. Moreover, throughout America many companies across all industry sectors are increasingly demanding new hires from universities with 21st century skills in engineering simulation allied with creativity, problem solving, collaboration and "real world" knowledge that is comparable to experience. RWDC is therefore an innovative and unique response to this emerging need in America.

It was the brainchild of Dr. Ralph Coppola from Parametric Technologies Corporation (PTC) in Needham, Massachusetts. The idea came to him five years ago, seeking to bridge the gap between high school students and modern engineering simulation technologies and tools with a view to giving school children an aerospace or automotive related challenge that the world is typically facing today. The ultimate goal of the challenge, from PTC's perspective, is to stimulate interest in engineering with 15 – 16 year olds and to illustrate that engineering is a mainstay of the American economy and an ongoing factor behind its GDP growth.

Softer goals for the RWDC were to inspire and engage students in science, technology, engineering and mathematics education and highlight the economic importance of having the world's most skilled and innovative workforce in these fields.

In 2007, Dr. Coppola piloted his RWDC program in 10 States ending the year with the winning teams from each of the participating States competing for national honors at the Smithsonian National Air and Space Museum. Dr. Coppola's vision has always been to create an educational initiative that infuses "real world" experiences, one-on-one communication with industry experts, professional tools and software donated by companies like PTC and Mentor Graphics which is free and accessible to all students in the United States. The Real World Design Challenge aims at being a high-quality national engineering competition that allows any school to participate, regardless of economic limitations or geographic location.

As of today, over 13,000 students in 45 States have participated in the program. Each year the national championship winning team is sponsored by the RWDC to participate in the President's White House Science Fair. In five years RWDC has grown from just 10 to over 700 high schools. Numerous engineers, designers, mentors and judges from across industry, government and academia have been key to the success of the program by donating their expertise, sharing their knowledge with the students and evaluating the students' work.

Executives from an impressive array of 55 governmental, defense and private companies including NASA, the Federal Aviation Authority, the US Department of Defense, Northrup Grumman, Rockwell Collins, Cessna-Textron, SpaceX, GKN Aerospace and Lockhead Martin participate

RWDC 2012 Blue Ribbon Judging Panel including Mentor's Dr. Erich Buergel as well as judges from Lockheed Martin Corporation, Honeywell Aerospace, Northrop Grumman Information Systems and Airbus Americas Engineering, Inc.

in the local and national finals as judges. This year, Dr. Erich Buergel from Mentor Graphics was one of the Blue Ribbon judges.

Boasting a 55% growth in participation last year, the RWDC has now become one of the fastest growing STEM education programs in America, with support from the Aerospace States Association and the National Lieutenant Governors Association. Dr. Coppola chose FloEFD™ from Mentor Graphics because it is a fully-embedded general purpose CFD tool within the popular Mechanical CAD Pro/Engineer and Creo productlines. In addition, FloEFD's unique push-button geometry and mesh generation pre-processing capabilities – the bane of traditional CFD tools – makes it ideal for high school children to pick up and use with minimal supervision.

FloEFD's robustness, built-in convergence intelligence and parametric simulation methodologies make it relatively foolproof for the high school students to use. Its history and pedigree as a well-validated 3D external aerodynamics solver making it ideal for virtual "wind tunnel" simulations. Schools are also offered PTC's Mechanical structural analysis tool, Windchill for their PLM needs and MathCAD as their engineering calculation tool. Dr. Copolla has arguably created the most popular and dynamic engineering design contest in America if not the world. In 2013 schools will be able to compete to design an unmanned aerial vehicle that has been proposed by Northrup Grumman. The true value of RWDC will be in the number of new young engineers it enthuses and stimulates into the profession in years to come.

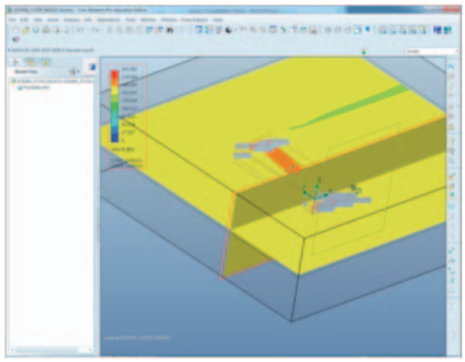
**Want to know more about RWDC or to encourage your high school to participate?**

To get involved or for additional information, contact:
**Dr. Ralph K. Coppola,**
Real World Design Challenge Director
Phone: **(1) 703-298-6630**
email: **rcoppola@ptc.com**
or go to **www.realworlddesignchallenge.org**

Registration is open on the RWDC website.

"Real World Design Challenge has changed his life. I see so much happiness and satisfaction ahead of him and it came from a seed planted by RWDC." **Parent of a RWDC high school pupil**

A Connecticut School's RWDC 2012 Cessna-type Plan Design (in PTC Creo)



A Connecticut School's RWDC 2012 Cessna-type FloEFD for Creo Aerodynamic Analysis of the proposed aircraft wing

# Getting It Right First Time

CFD Simulation saves Mentor Graphics thousands when using their own tool to design two new datacenters

Figure 1: Raised-floor simulation with superimposed CFD grid

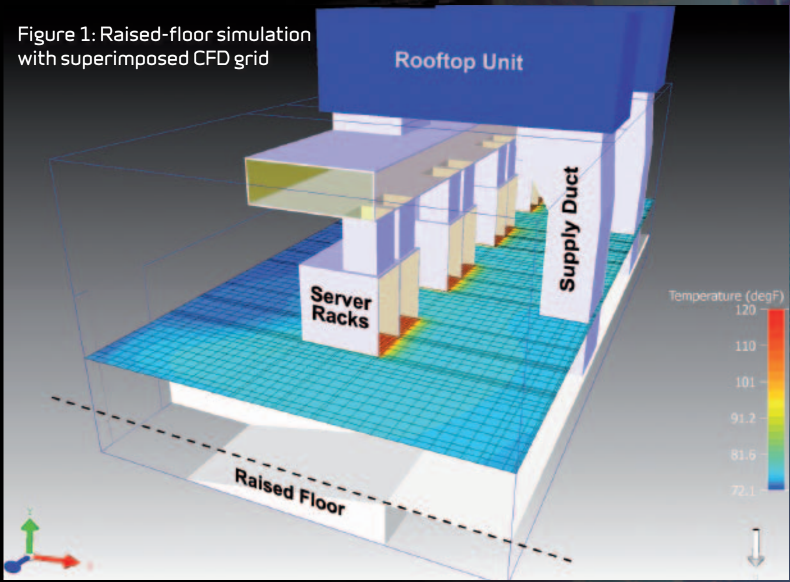
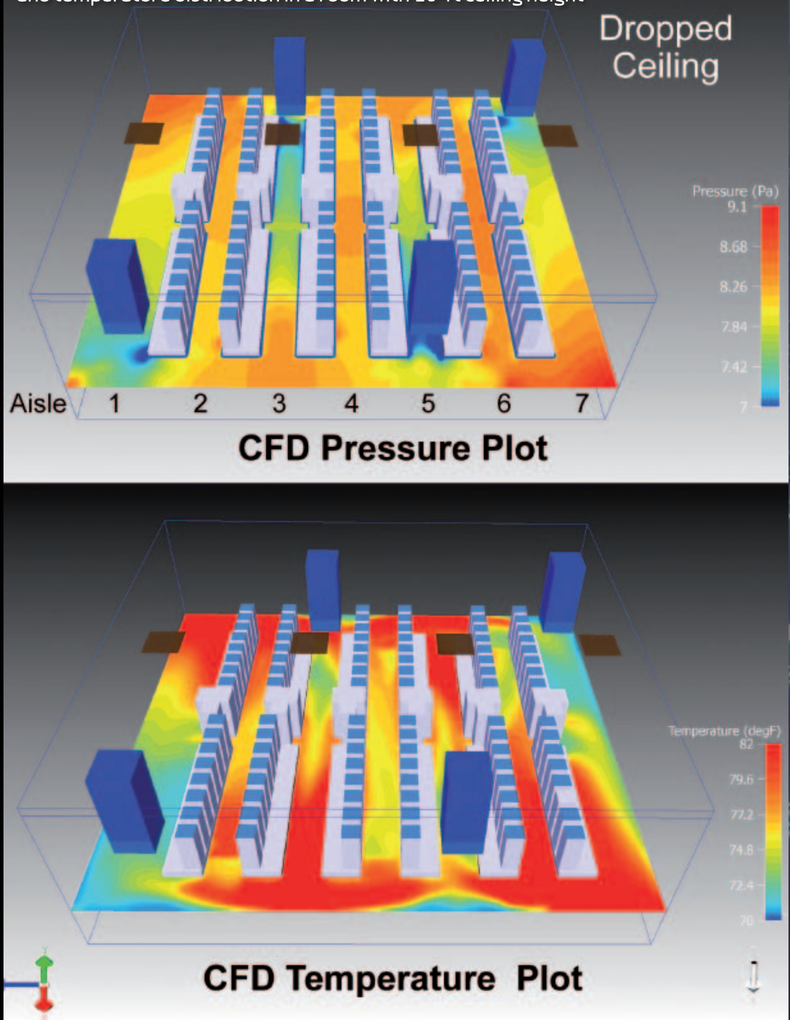



Figure 2: Dropped-ceiling proposal showing non-uniform pressure and temperature distribution in a room with 10-ft ceiling height




BEING a leader in the electronic design automation (EDA) market has its upside when you want to build not one but two centralized datacenters. With a company full of talented engineers and their own highly revered Computational Fluid Dynamics (CFD) modeling tool, FloVENT™, it is no wonder the company decided on a purpose built center as opposed to outsourcing to a third party. The Mentor design team were commissioned to design an effective cooling architecture and then form the configuration of the building around that. While there were some basic guidelines to define the size and shape of the structure, the details of ducting, venting, and internal floor plans were driven by the servers' needs.

The company's primary goal was to save power by minimizing the use of air conditioning to cool their datacenters. With one of the new centers based in Oregon and another in Shannon, Ireland, Mentor Graphics set out to consolidate the resources of more than 20 local centers down to just two. The decision to build custom centers was driven by the steady, costly growth in the company's overall server heat load, which has risen by 33% annually. The climate of both Wilsonville and Shannon helps to keep the datacenters from overheating - Mentor predict they will only have the need to use air conditioning 10% of the time.

## CFD Solutions at Your Fingertips

**To begin, the simulation was gridded into many thousands of very small cells that could be individually analysed and integrated into a composite flow-thermal view. Figure 1 shows such a grid superimposed onto a temperature slice map of a raised-floor HVAC concept. This variable cell sizing provides higher resolution in specific areas such as the server rack interiors.**

Historically designing a building's air and thermal flow involved formulating estimates and judgments based on experience, and then submitting the design to an external specialist for CFD analysis. This approach can deliver the necessary accuracy but is time consuming, often taking weeks for each round of evaluation. Nowadays organizations are bringing CFD in-house, using design teams to perform flow analysis to rapidly model design iterations. Mentor used their own CFD product, FloVENT, in the design of their centers and were able to generate reliable thermal and flow data that guided engineers in proof-testing new ideas.

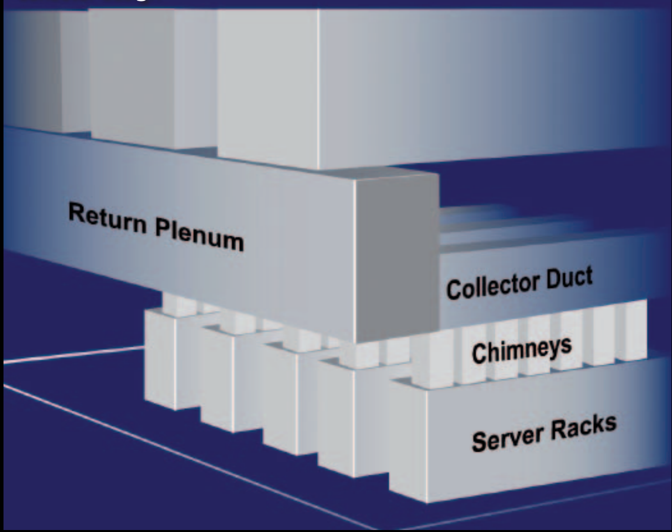
## The Pressure/Temperature Conundrum

**When considering their datacenters, the Mentor team contemplated four methods; the traditional raised floor, a dropped or suspended ceiling, and two further methods that involved hot air confinement. The decision about the basic air circulation approach for the server rooms illustrates the benefit of performing flow analysis locally.**

A simple raised floor design proved inadequate after CFD studies predicted pressure problems under the floor. The raised floor should function as a plenum that regulates pressure and distributes air efficiently. However the volume beneath the floor tiles was insufficient and therefore incapable of acting as a plenum to ventilate the air flow as the air would not reach all the server racks uniformly. To correct this, it would have been necessary to increase the floor's height considerably, adding to an already costly project by as much as 28%.

When examining the suspended ceiling design, the configuration involved hot air from the servers traveling through a chimney system to a plenum space formed by the ceiling itself, and onward to rooftop coolers. The capacity of the plenum was again a problem. To design a plenum of

Figure 3: Initial layout of the chimney/ collector design



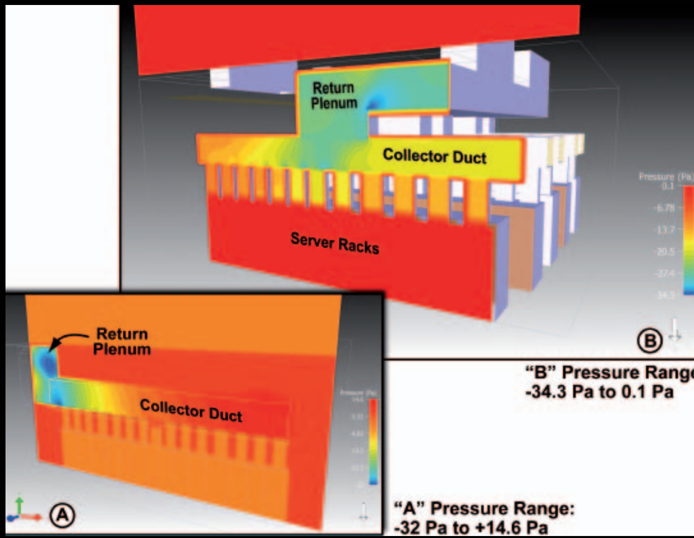




Figure 4: A centered return plenum ensured consistent pressures throughout the collector duct

"Using FloVENT®, a powerful CFD software tool, has been a real bonus for the datacenter design team. After very little training the design team has been able to investigate various permutations virtually, to manage airflow and temperature in the datacenter. We were able to experiment with different layouts and technologies before we finalized the optimized design. Indeed, this process has saved us time and money and it should lead to very significant energy savings over the years the data centers will be in service." Ananthan Thandri, Chief Information Officer, Mentor Graphics

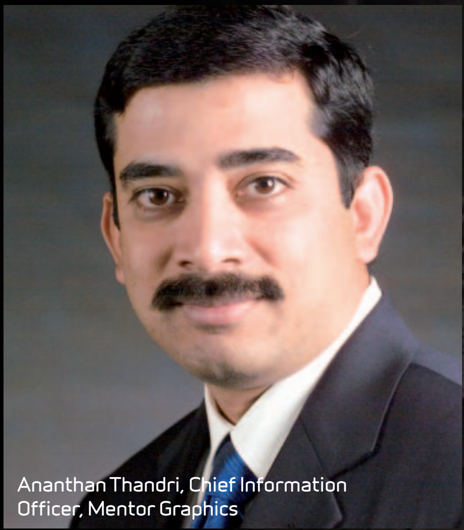
Aerial view of the Wilsonville datacenter under construction

sufficient capacity would constrain the ceiling height in the server room, and the remaining room volume wouldn't allow the pressurized incoming air to distribute evenly. Figure 2 compares a CFD pressure "slice" with an equivalent temperature contour slice, both spanning the room at the 6 ft. height of the server racks. Problem areas in aisles one, three, five and seven were instantly revealed by the simulation, showing areas of high heat in the "cold" aisles. These flaws are a result of uneven pressures throughout the room.

## Capturing Hot Air

**The next two methods considered involved the confinement of the hot air leaving the server racks and redirecting it. This "hot aisle containment" technique forces and confines the air in an entire aisle between two rows of racks and directs it toward an exhaust duct. Early on in the process, this design was abandoned after CFD analysis revealed that the aisle was too hot at 120° F, these temperatures would impact a technician's ability to perform routine maintenance on the servers.**

Figure 3 demonstrates an alternative design incorporating a chimney system that connected chimneys directly to individual server racks, drawing heated air from them and guiding it to a collector duct and then


Ananthan Thandri, Chief Information Officer, Mentor Graphics

into a return plenum. Using CFD simulation, a range of scenarios was tested for the chimney/collector including a worst-case example in which the rear doors of several server racks were open for maintenance at the same time. The results confirmed first that the aisle temperatures were much lower at approximately 80° F making it safer for engineers, and secondly that server racks still supplied at least 66% of their hot exhaust air into the chimney when their doors were left open. Both findings supported the efficacy of the chimney/collector architecture.

Finally, a number of "what-if" simulation scenarios aimed at refining individual components were run. They proved helpful to the Mentor team when evaluating the chimney/collector design. The CFD plots in Figure 4 depict two alternatives for the return plenum layout. As a result of the simulations minor changes to the design were identified that could improve air flow in the duct. Scenario "B" depicts the design changes that resulted from conclusions in Scenario "A", namely the positioning of the plenum in order to relieve the pressure on the servers nearest to it. Further CFD experiments produced a duct design with angled corners that minimized this disturbance.

The Mentor Graphics datacenter project is still underway in Wilsonville, with completion expected in December. Even as the construction progresses the Mentor team are refining HVAC control algorithms based on further CFD studies and real-time data from the already completed Shannon datacenter. With the aid of FloVENT, a CFD tool for HVAC and datacenters, major design issues were investigated and improved. As well as this the project team were awarded a grant from the Oregon Energy Trust for meeting stringent energy compliance targets. CFD simulations have proved invaluable to expose design flaws and dictate the correct design path before any concrete has been poured, and saving time and ultimately money by getting it right the first time.

# Let there be more LED light!

Over the last decade, as the efficiencies of Light Emitting Diodes (LEDs) has increased to the point where they are a viable replacement for incandescent bulbs and compact fluorescent lamps (CFLs) the world has reached a tipping point in the usage of LED Lighting within automotive lighting, street lighting, back lighting (for consumer electronics) and now general lighting applications.

By Dr. John Parry
Electronics Industry Manager
Mentor Graphics
Mechanical Analysis Division

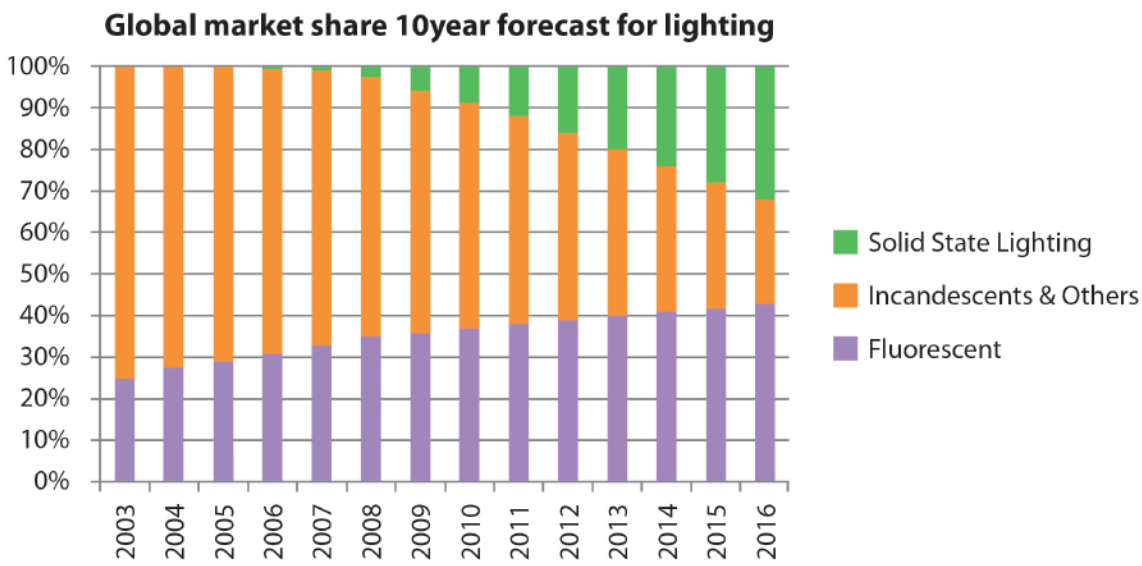
**UP to 95% of the electric input in traditional incandescent light sources becomes heat and is largely dissipated via radiation. LEDs however are significantly more efficient with up to 60% becoming heat and not all of it dissipated via radiation. The challenge for this rapidly growing industry is therefore effective thermal management of these Solid-State Lighting (SSL) products. Heat must be removed by conduction and convection in order to keep the LED cool for quality lighting outputs, reliability and product lifetime. Hence, the key technology barrier to LED adoption for general lighting applications is thermal.**

The LED lighting industry has also been hampered by a lack of standardization on data relating to the thermal performance of the LEDs themselves. As LEDs emit a substantial proportion of the power they consume as light, the power emitted as light has to be taken into account when calculating their real thermal resistance. It was for this purpose that the Mentor Graphics hardware product TeraLED™ was originally developed and released in 2005, and as LED light emission and lifetime strongly depends on temperature, proper thermal characterization of individual LED components is therefore very important. Accurate information on the real thermal resistance of LEDs is critical for lighting system designers to develop proper thermal management solutions of their SSL products. Knowledge about temperature dependence of the light output characteristics (such as luminous flux or color coordinates) of LEDs is necessary for designing luminaires which provide light intensity and light distribution patterns as required by lighting standards. Unfortunately, due to lack of industry standards, LED vendors' data sheets hardly provide any useful information in this regard.

With these factors and trends in mind, Mentor Graphics has released two new productlines. These offer new capabilities for the LED supply chain from manufacturers through sub-assembly to end user SSL products; a larger TeraLED and an LED Module for the CAD-embedded CFD product, FloEFD:

## Extended TeraLED Product Family

The market demand for bigger, brighter LEDs that dissipate more power has necessitated the development and release of a larger TeraLED sphere with greater heat sinking capability. Mentor is now offering both 30cm and 50cm integrating spheres and a range of cooling options up to 50W of cooling. T3Ster® measurement results can be converted into so-called compact thermal models (CTMs) suitable for CFD simulation.



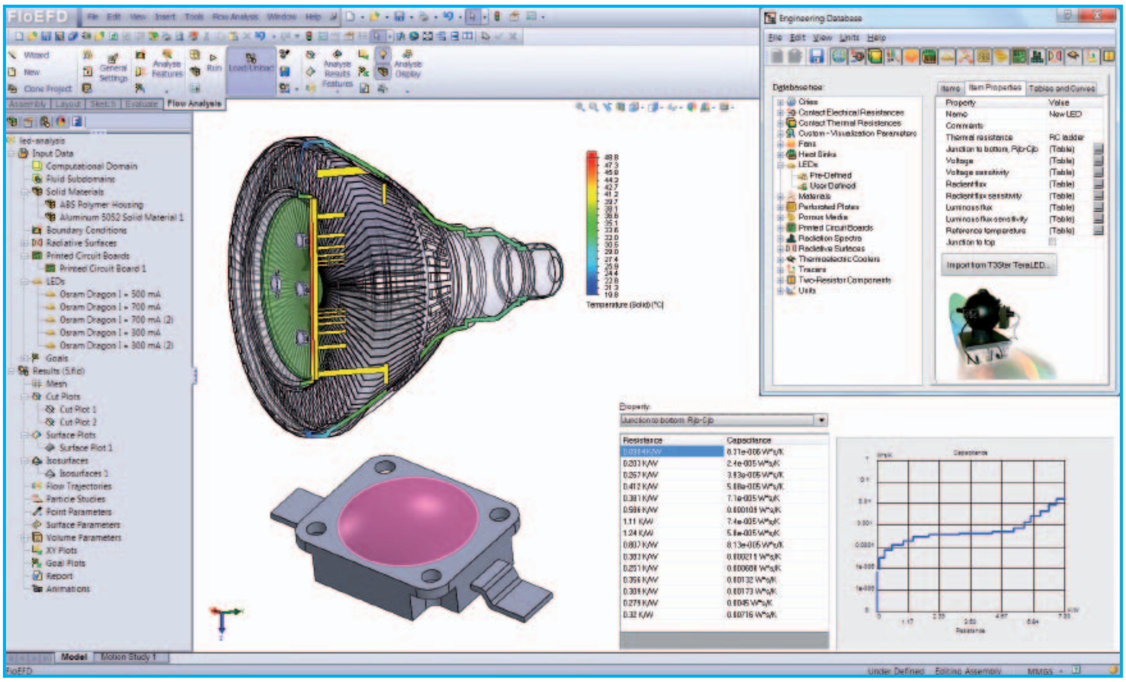

Source: Optoelectronics Industry Development Association (OIDA) Lighting Technology forecast 2007 – 2017

These models have now been extended via TeraLED measurements to include temperature-dependent light output data. These provide SSL designers, for the first time, with the precise models of individual LEDs needed to calculate the 'hot lumens' of their luminaire designs.

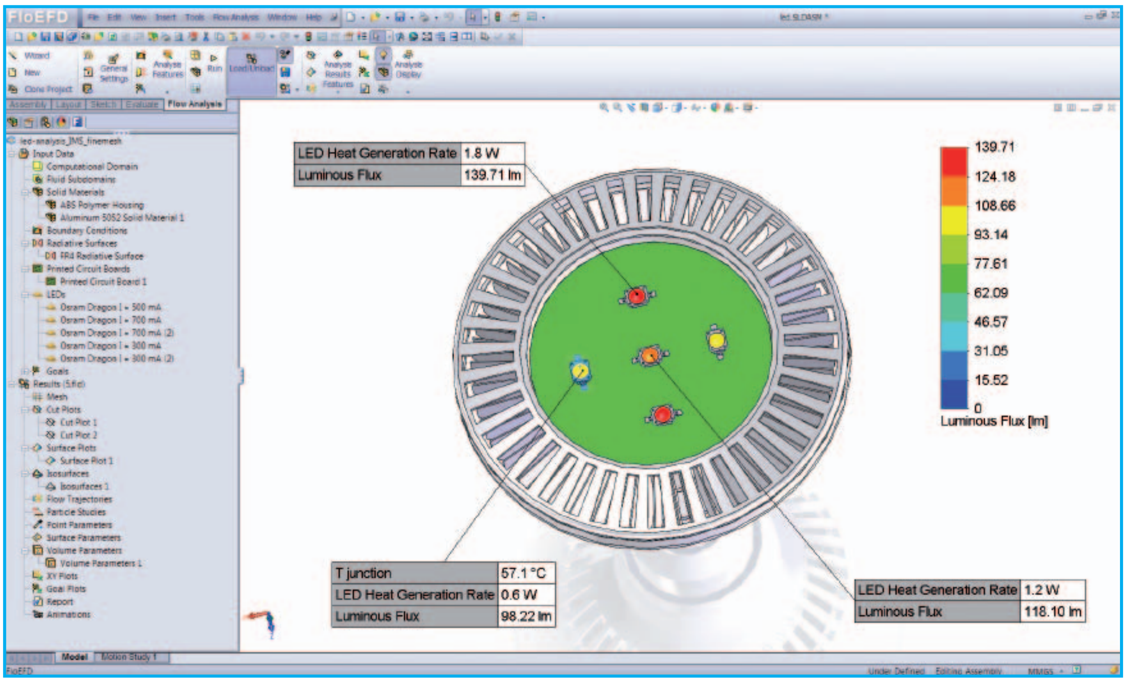
## New CFD Module for LEDs

The FloEFD V12 release includes a unique CFD solution capability for designing Solid State Lighting. The innovative LED Module now allows designers to use accurate CTM and photometric models of LEDs obtained from T3Ster and TeraLED within a general purpose CFD simulation package. These models are uniquely driven at constant current as opposed to estimated power consumption inputs that yield less accurate predictions. This ensures correct accounting for power that is emitted as light when calculating the heat dissipation of the LED. Hence, the temperature, power consumption and light output (hot lumens) of the LED are all predicted accurately by FloEFD. To enable this, a "starter pack" of thermally and photometrically characterized LED models is provided as part of the module. These are: CREE XT-E; Osram Golden Dragon; Seoul P4; and Philips Luxeon REBEL. The module also includes the ability to account for adsorption of radiation in semi-transparent solids such as a lens in front of the LED, and is able to represent a PCB as a compact model with orthotropic thermal conductivity.

New 50cm, 50W TeraLED



Typical LED Module interface panels illustrating the T3Ster-TeraLED workflow in FloEFD 12.0




Typical LED Module "Hot Lumens" prediction of a retrofit SSL bulb in FloEFD 12.0

# Small but Mighty Powerful

## Micro-turbine jet engine simulation and structural analysis using FloEFD™ and Creo® Simulate

by Tatiana Trebunskikh, Andrey Ivanov, Gennady Dumnov

**When talking about jet engines in aerospace, visions of the enormous assemblies that propel passenger aircraft inevitably spring to mind. However just as notable and impressive are the much smaller micro-turbines which propel radio controlled model aircraft and Unmanned Aerial Vehicles (UAVs). Used primarily for military purposes, but also increasingly in civil applications such as firefighting and surveillance, UAVs or Drones are controlled either by computers in the craft or by remote control. In more recent years, jet engine development has been focused on fuel efficiency, reducing emissions and quieter engines.**

These goals go hand-in-hand with the latest component designs, fuel types or utilization of flow behavior. The biggest influence on the aforementioned parameters was achieved by a high-bypass ratio, developed in the mid-1960s as seen today in every passenger aircraft. Reaching up to 115,000 pound (514 kN) of thrust at a bypass ratio (BPR) of 10:1 with a mass flow rate of up to 1,300 kg/s, is enough to impress any engineer. Now of course so called smaller micro-turbine jet engines cannot compete with such numbers but it doesn't make them less impressive or complex. Whilst designers of micro-turbines must also achieve efficiency and power goals, they have an added challenge of doing so on a much smaller scale which poses more problems for materials and components. The best way to efficiently design such high performance engines is by using virtual prototyping such as Computational Fluid Dynamics (CFD) and structural analysis. This article explores how FloEFD is used to simulate the fluid flow, heat condition and combustion of a micro-turbine and how these simulation results apply to a structural analysis model.

Micro-turbine engines are developed for specific flight applications. They are used in UAVs which are designed for short flight duration. Today, lots of different UAVs are operating worldwide for all kinds of mission types. In general UAV missions range from reconnaissance, surveillance, target acquisition, signals intelligence (SIGINT) to scientific research. Another common use for small gas turbines is for auxiliary power units (APU), supplementing aircraft engines

Figure 1. The model (left) and real prototype (above) of KJ 66 micro-turbine engine.

which provide additional power for non-propulsion functions when required. Their small size means, micro-turbine engines have small air mass flow rates and low pressure ratios, but very high rotational speeds of turbine and compressor stage.

For the purposes of this study a KJ 66 (Figure. 1) was chosen, as it is one of the more robust small engines with accessible design data.

Turbojet engines have complex geometries and physical processes. Understanding these processes is very important for designing such a high-performance product. The complex geometry and small size of this kind of engine limits the access of typical instruments used for the measurement of flow parameters as is required for a better understanding of the complex flow structure. As well as this, creating the optimal design for individual parts of an engine during testing can be an expensive procedure, making CFD analysis a very useful tool.

This article presents the CFD analysis of the KJ 66 micro-turbine engine, which is calculated as one unit without any transferred, symmetrical and periodical conditions between its parts. It takes into account the rotation of air in the compressor and turbine, conjugate heat transfer and air/kerosene combustion all within the multiCAD-embedded full-featured general purpose CFD tool FloEFD™. Thermal and structural analysis were conducted using PTC's Creo Simulate together with thermal and pressure load results obtained from FloEFD.

Five cases of varying rotational speeds of 40000, 60000, 80000, 100000 (the normal mode) and 120000 rpm were considered for the compressor and turbine by specifying local rotational zones. The solid parts are specified as aluminum, steel and inconel for the consideration of conjugate heat transfer.

The air mass flow at the inlet of the engine at various rotational speeds of the compressor can be seen in Figure 2. The FloEFD results are compared with the experimental data of Kamps [1], and show the values of mass flow match the experimental data very well with almost no dependence on the number of mesh cells.

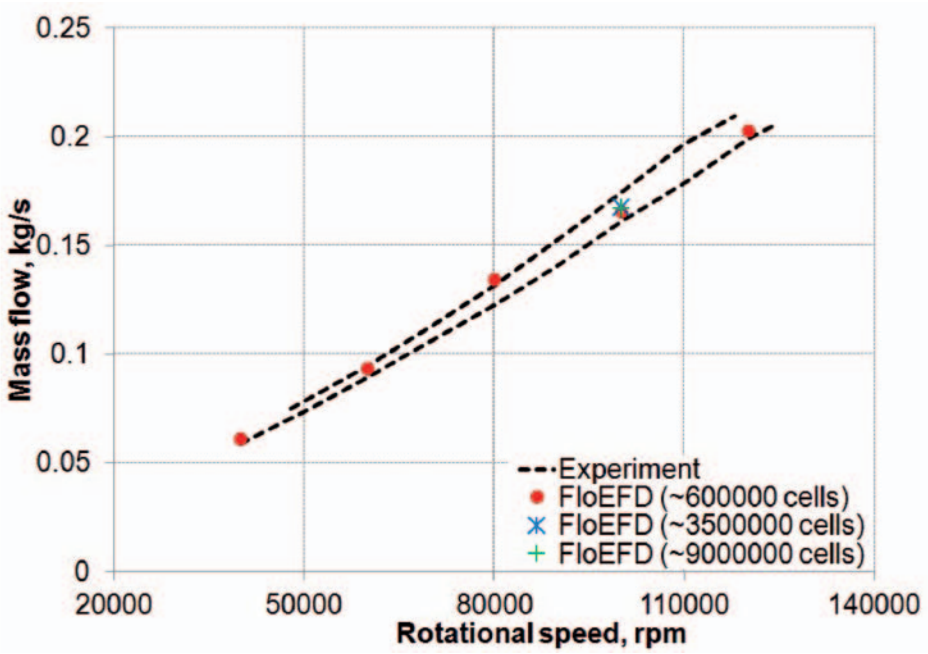


Figure 2. Air mass flow at the inlet of KJ 66 engine.

The calculation results in Figure 3 show flow trajectories colored by velocity magnitude and pressure distribution with Line Integral Convolution (LIC) on the surfaces of the compressor and diffuser at the normal mode. The pressure on the compressor's blades can be lower than 65000 Pa and can reach 180000 Pa on the diffuser's blades.

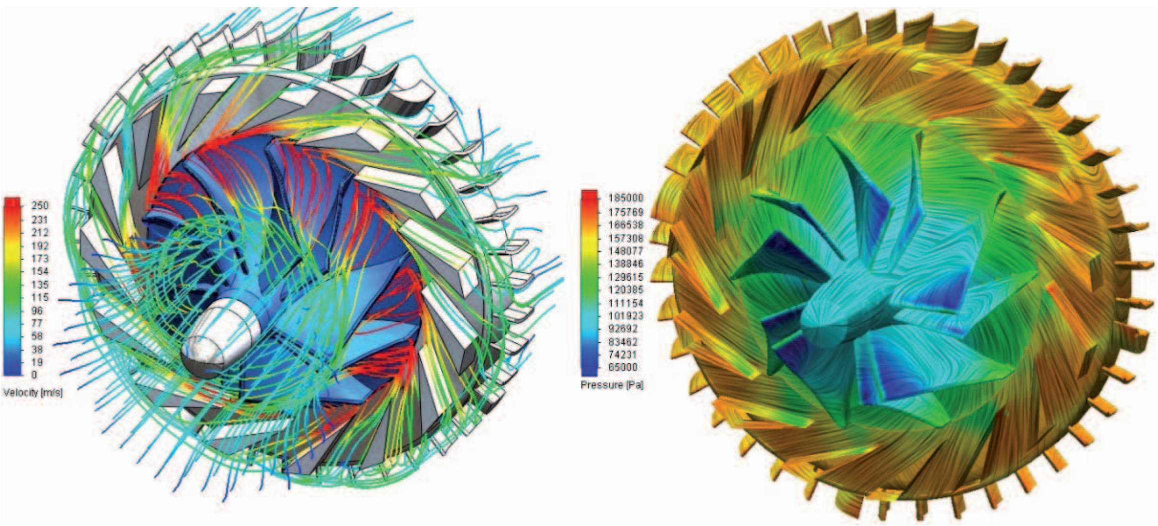


Figure 3. Flow trajectories colored by velocity magnitude (left) and pressure distribution with LIC on surfaces of the compressor and the diffuser (right) at normal mode.

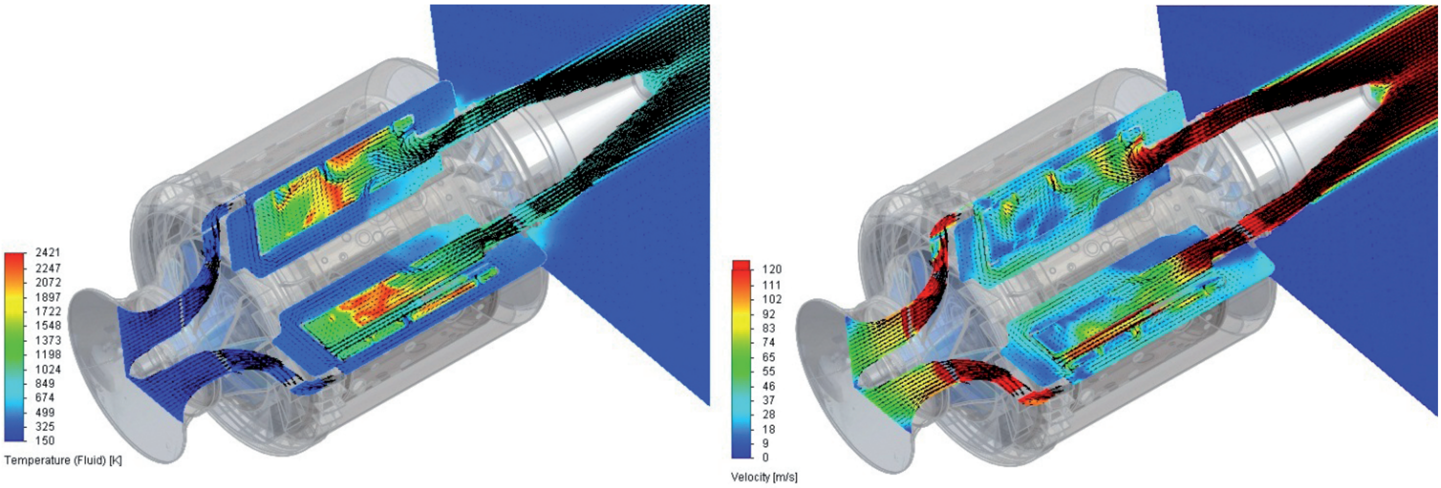



**Figure 4. Fluid temperature (left) and velocity (right) distributions at two longitudinal sections of the combustion chamber with flow vectors at the normal mode.**

The combustion chamber of the KJ 66 engine features direct fuel injection through six vaporizing sticks to ensure complete combustion inside the chamber. Figure 4 presents the fluid temperature and velocity distributions at two longitudinal sections of the combustion chamber with flow vectors at the normal mode. The temperature in the combustion chamber reaches approximately 2400 K. An increase of the velocity in the region of the openings of the combustion chamber can be clearly seen especially on the rear wall of the chamber.

Further evaluation of the results shows a direct comparison of the temperature distribution in the combustion chamber at 120000 rpm obtained with FloEFD (left) and a traditional CFD tool (right) presented by C.A. Gonzales, K.C. Wong and S. Armfield [2]. Both models have been simplified by not examining all parts of the engine, but all features of the combustion chamber have been taken into account. The symmetry conditions are not used in the FloEFD model as they were in that of the traditional CFD model, resulting in some differences in the parameter's distribution which can be seen in Figure 5. Considering these factors, both FloEFD and the traditional CFD tool, show reasonable accordance in their results. It is clearly visible in Figure 5 that the primary combustion zone is located in the central part of the chamber.

Besides thermal simulation, FloEFD also allows some parameters to be exported as loads for structural and thermal analyses with Creo Simulation. In this case the surface temperature was exported from the CFD calculation to run the thermal calculation with Creo Simulation. Then the structural analysis was conducted using the temperature of the previous calculation and the pressure exported from FloEFD. The results in Figure 6 show the displacement distribution of the structural analysis. It can be clearly seen that the combustion chamber is deformed under the loads with the displacement reaching a maximum of 0.001 m.

A pressure and velocity distribution near the surfaces of the engine is presented in Figure 7 and the increase and decrease of the pressure at the compressor and turbine stage is shown respectively.

The overall performance of the engine is usually measured by thrust and Figure 8 shows the comparisons of measured and predicted values of thrust of the KJ 66 engine at different modes. Experimental and predicted values are similar up to 80000 rpm with some divergence at 100000 rpm.

Comparisons of measured and predicted values of the main integral parameters such as air mass flow at the inlet of the engine, thrust and temperatures at the outlet of the diffuser and combustion chamber are almost identical.

This study demonstrates that FloEFD can provide a series of "what-if" CFD analyses and export data values for structural and thermal analyses. With its CAD embedded approach it is very efficient should there be

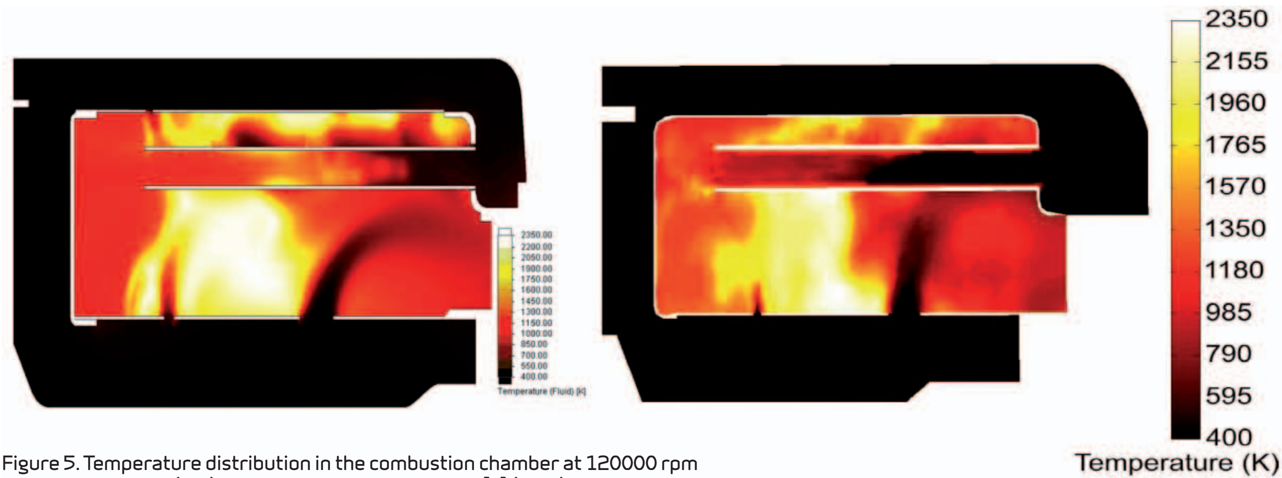



**Figure 5. Temperature distribution in the combustion chamber at 120000 rpm obtained in FloEFD (left) and traditional CFD software [2] (right).**

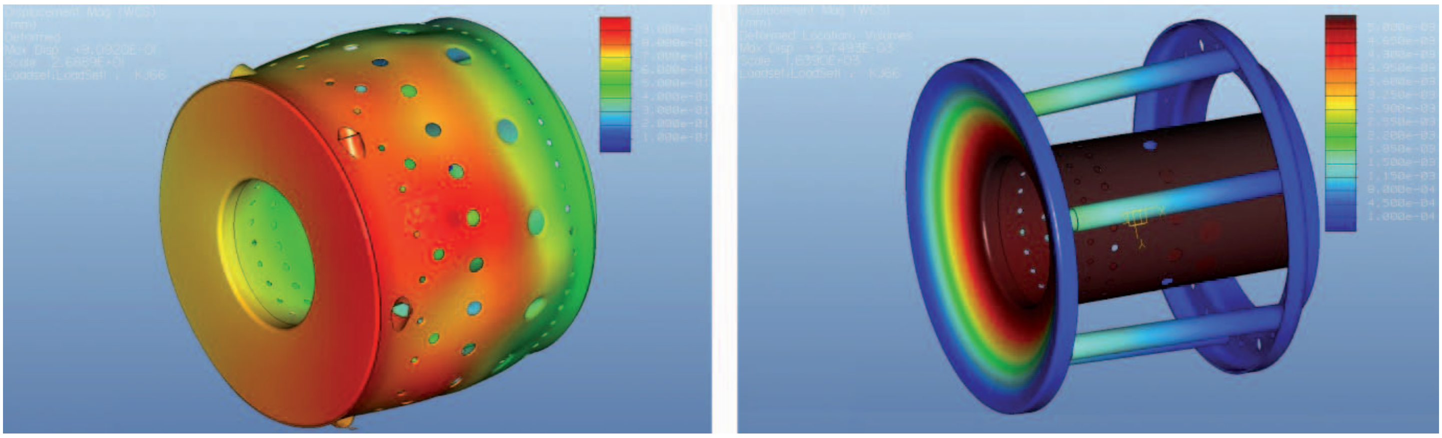


Figure 6. Displacement distribution on the surface of the combustion chamber in Creo Simulate (scaling 20%).

a need to experiment with different designs of any component in the model. By simply changing the CAD parameter of a parametric model, such as the opening diameter of the combustion chamber, multiple simulations can be created in little time, whilst at the same time changing any boundary conditions. The simulation project is always up-to-date with the CAD data.

FloEFD provides high accuracy in high-end applications such as demonstrated in this aerospace example. With its CAD embedded approach, FloEFD allows the user to set up and run simulations and design iterations quickly in order to determine the appropriate design modifications, saving time and money.

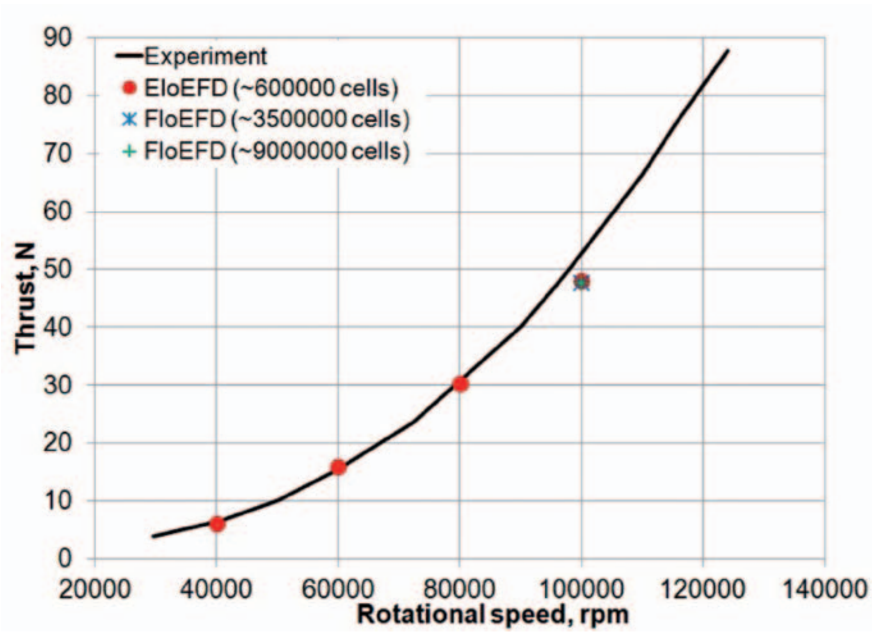



Figure 8. Thrust of KJ 66 engine.

## References


[1] Kamps, T. Model jet engines, UK, 2005.

[2] Gonzalez, C.A., Wong, K.C., Armfield S. Computational study of a micro-turbine engine combustor using large eddy simulation and Reynolds average turbulence models, Austral Mathematical Soc, Australia, 2008.

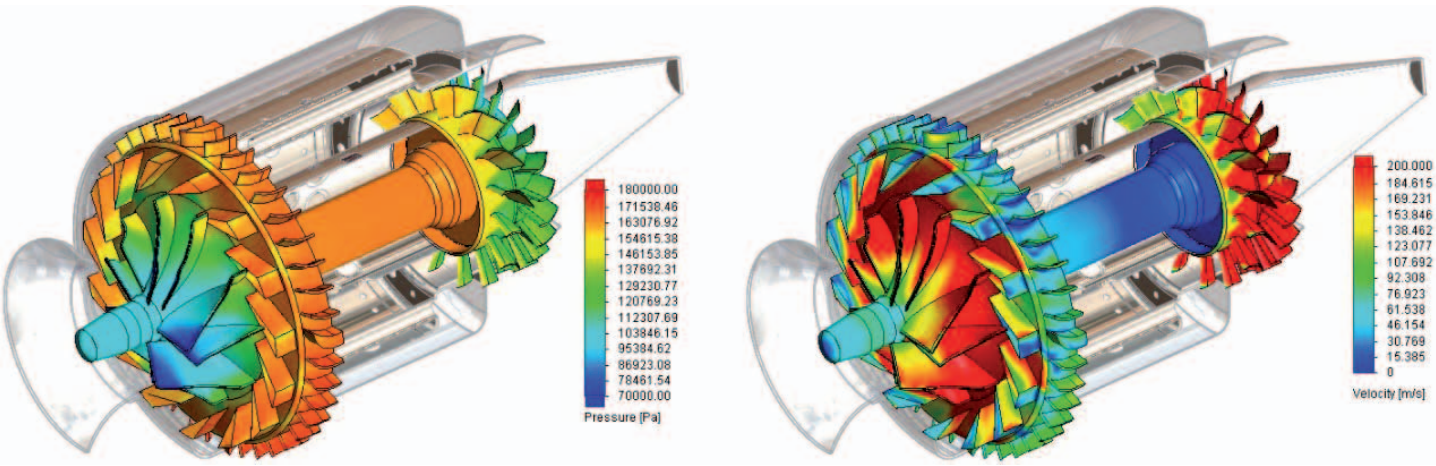




Figure 7. Pressure (left) and velocity (right) distributions.


**Accurate prediction of internal flows is essential for the efficiency, reliability and safety of industrial plant, road vehicles, ships and aircraft. Since its first publication in 1978, Internal Flow Systems has become the standard reference for engineers, the data and procedures described having been adopted worldwide by many companies as an essential part of their design and simulation methods. It was this text that prompted the creation of Flowmaster almost 20 years ago. We met with author Don Miller to find out how it all began and what he believes is the future of 1D CFD.**

**EE: When you were younger, who or what inspired you to be an engineer?**

DM: When I was a school boy I was, what you would today class as, dyslexic. I went to a small village school that did not recognise these things at the time, so when I moved to secondary school, they assumed I was dim and put me in the lowest class where it would take three years to work myself up to a higher class. Those in the higher classes were the ones able to get to grammar school. Assuming I would never get there because I couldn't spell, I was encouraged to do workshops like metalwork and woodwork. I knew I would need a better education if I was to ever get on in life, so I joined the RAF as a boy entrant at the age of 15. After three months we were given an exam of multiple choice answers, I had never seen exams like this before. I got over 90% on that first exam, had I had to write the answers I would probably have failed. When I came back after the Christmas break they told me I was going to RAF Halton to be an engines apprentice along with those from grammar school - just based on ticking boxes on a sheet of paper! After 10 years in the RAF I bought myself out and went to work for Bristol Siddley Aero Engines. Two years later I decided to go to the College of Aeronautics at Cranfield as a student to do the Diploma of Aeronautics. When I graduated from Cranfield, Bristol Siddely Aero Engines offered me a job working on the Olympus Fuel Systems for Concord but at that time the Government had just cancelled the revolutionary TSR 2 aircraft that had Olympus engines and the workers in the Bristol factory were leaning on scrap engines that had cost a million pounds and had no work to do. I decided not to go back to Bristol Siddely and as The British Hydromechanics Association (BHRA) was relocating to the Cranfield campus and needed staff I decided to change from aero engines to fluid mechanics. By the time I joined BHRA I had developed strategies for overcoming some of my dyslexic problems. One of these is writing and repeatedly rewriting text until I understood technical information and could get it in a logical format. In the process I have learnt to present technical information, such as that in Internal Flow Systems, in a form that others find useful.

**EE: How did your world-famous Internal Flow Systems book come into being?**

DM: Well now that is a long story. There were many influencers to the writing of Internal Flow Systems. Shortly after I joined BHRA they got a contract from the UK's Central Electricity Generating Board to provide loss coefficient data for large flow systems. Initially reviews of available loss coefficient data were carried out and a Steering Committee with academic advisors helping to decide on the experimental programme. At that time BHRA probably had the best library in the world for reports on fluid flow systems with thousands of papers that were inaccessible to the general engineering world. As the project went on and we generated new loss coefficient data, I became convinced that there was not a lack of knowledge for the design of efficient internal flow systems but there was a serious and very difficult problem in getting information to engineers and designers in a form they could readily use. A more disciplined and informed approach to flow system design was clearly needed. I also found it very difficult, no impossible, to convince people, including the academic advisors, that the data that they'd been publishing was wrong. I concluded that producing a book on flow systems that industry used was the way forward. Extracting information and data from the literature and putting it in a form to meet a broad spectrum of industrial needs was beyond any contract that BHRA could secure. In view of this and my interests in understanding and adding value to others' work, writing Internal Flow Systems became a `hobby' for ten years, followed later by a further five years in writing the second edition. While this writing was going on BHRA was still generating fluid flow data culminating in work related to the North American Gas distributing system. I was able to feed the output from this industry driven work into Internal Flow Systems.

**EE: What is the most obscure use your book has been put to?**

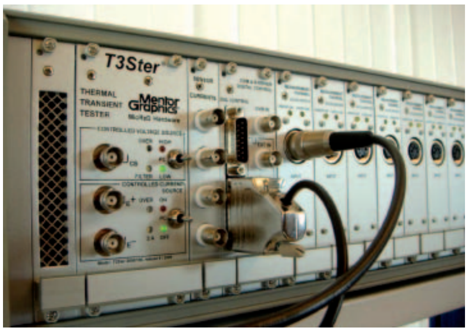
DM: People have used it for blood flow to try to understand how flow patterns affect deposits in arteries and failure of blood vessel walls. Internal Flow Systems has been used in so many applications, from little rocket engines used in space to escape systems from submarines, from sewage treatment plants to ultra clean applications related to fuel cells. Wherever there is fluid flow there is a chance that someone would pick it up.

**EE: How do you think 3D CFD will interact with 1D CFD in future?**

DM: An idea that I find very appealing is to use the power of simulation to be able to "walk" through a system, component by component, and not only see the flow behavior but also be alerted to potential problems. If we go back to the nuclear reactors of the 1960s and 70s, there were billions of dollars lost because they got the fluid flows wrong, usually in a very small area of a flow system. If engineers had the ability to go through the systems looking from component to component and understand where problems might arise, a lot of mistakes could have been avoided. The majority of the mistakes occurred in not conditioning the flow prior to it entering a critical component or fluid machine. Two reoccurring problems are the lack of well-designed diffusers and areas of flow separation resulting in flow instabilities or swirl. You can't expect design engineers to be aware of all the situations where such problems arise so we need to provide with appropriate CFD based tools. Most systems are so strong it often doesn't matter if the flow inside them is unstable but as systems grow in size and are optimised to minimise weight avoiding unstable flows becomes much more important. A combination of 1D and 3D analysis would enable you to do a risk analysis on the full system component - component and determine high and low risks quickly. It will prevent having to retrospectively fix problems and ultimately will save money.

# T3Ster® - Delivering Customer Satisfaction For Yaskawa

Electronics




## About Yaskawa Electric

Yaskawa Electric produces drives and inverters for industrial and air handling purposes, elevators, motion controllers, and robots for industrial and semiconductor applications. These products utilize alternating or rotating magnetic fields from DC current using very high power IGBT modules or MOSFET modules (half bridges, full bridges) as commutators, and making DC from AC, again using high power diode rectifier modules.

## The Need for Thermal Characterization

High power modules are a key component in many of Yaskawa's products, but thermal management poses a significant product design challenge. The majority of products on the market fail because of overcurrent/ overvoltage conditions, so the root cause for burnout is not known. What is known is that in order to reduce field failures, overall improvement of the thermal quality of products is a must!

Yaskawa already used thermal simulations to improve product quality in the design phase, and ensure fast product realization. To further improve the 'thermal power quality' of its products, Yaskawa needed to focus on improvements in its overall thermal design technology, by validating thermal simulations with measurements. Yaskawa wanted to be able to measure the internal chip temperature to verify the theoretical values used in the thermal simulations, to ensure that products perform as designed, and to further improve the thermal quality.

## High-End Product Validation

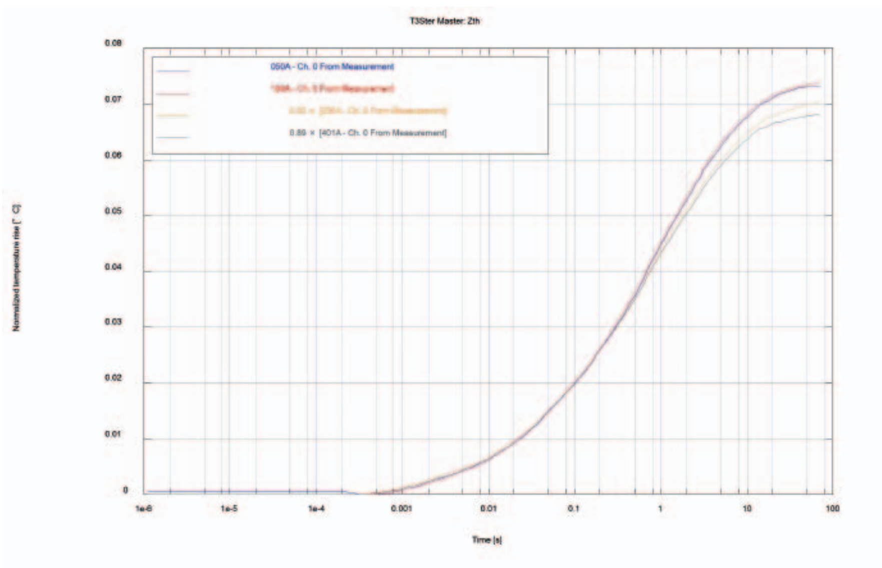
Yaskawa uses a wide variety of power modules (IGBTs and MOSFETS) ranging from 100A to several kA. One of Yaskawa's concerns was whether T3Ster could be used to measure such parts. T3Ster has a 10V/2A range extended by voltage or current boosters, but even so it is difficult to measure kA rated parts at full power.

To determine this, Yaskawa used T3Ster with a 400A booster to measure a commercial 800A IGBT module clamped to an in-house water cooled cold plate, with temperature controlled by the external chilled bath.

The first task was to calibrate the transistor's temperature coefficient (K factor calibration), which was found to be highly linear over the temperature range of interest (room temperature to 80oC). The part was measured at 50A, 100A, 200A and 400A to investigate how the thermal response varied with current.

Yaskawa found that above 100A the normalized temperature vs. time responses had to be factored to get them to match in the early part of the transient (below 0.2s).

T3Ster senses the temperature response of the die. At higher powers an increasing proportion of the total power dissipate occurs off the chip, in the lead wire, and



"Customer satisfaction is an important corporate value for Yaskawa. By testing power modules with T3Ster we have been able to improve our thermal design quality, ensuring that we deliver products that have the highest possible long-term reliability."

wiring etc. within the module, resulting in the response needing to be factored to match the response at lower powers. As the curves match up to the point where the heat leaves the package, T3Ster can be used to test the integrity of the internal structure of the parts at lower power. As Yaskawa's products are used in safety-critical applications, high throughput measurements during production are needed to ensure final quality, and the above results show that lower currents can be used to assess the test the structural integrity of power modules.

## Module Reliability Evaluation

For reliability testing, parts must be subjected to thermal stress ($\Delta$TJ, TJmax) by power cycling. One challenge is how to accurately measure the junction temperature (TJ) to determine the level of stress the part is subjected to. Thermography, or thermal imaging, is not suitable as the junction is located below the surface of the package. Using a thermocouple below the package is less accurate because it is measuring the case temperature, not the chip temperature. Hence it is hard to produce a desired $\Delta$TJ to high accuracy.

To address this issue, Yaskawa again turned to T3Ster. It is possible to transpose the normalized temperature response vs. time for a given power into an equivalent response showing power vs. temperature for any desired temperature rise. An example for an 80oC temperature rise is shown in below in Figure 2.

From the graph it is possible to read off a combination of power and a duration that will give a junction temperature rise of 80oC (e.g. 1500W for 2s as shown on the graph). T3Ster therefore allows power cycling measurements to be accurately planned.

## In Situ Thermal Interface Material (TIM) Measurement

Another issue that Yaskawa faces, along with many other electronics companies, is that the material properties for TIM materials in the application can vary a lot from the data provided by vendors. This is compounded by the thermal properties TIM also vary as a result of processing. Yaskawa needed a way to check the thermal performance of the TIM material as used in the application rather than rely on values obtained when the materials are evaluated.

As T3Ster characterizes the heat flow path from the junction through to the ambient, Yaskawa realized that T3Ster could be used to characterize the TIM material in situ. To evaluate this use of T3Ster they studied a commercial 200A diode module mounted on a cold plate with their existing TIM, a new TIM currently under evaluation, and dry mounted without any TIM present. With T3ster they were able to clearly show that the new TIM resulted in a 0.13K/W improvement in the total thermal resistance of the material stackup:

## Conclusion

Yaskawa's Reliability Technology Center has now successfully adopted T3Ster, providing for the first time the ability to measure chip temperatures directly, allowing them to verify that the design values for thermal parameters are correct, by directly measuring the resulting thermal resistances and capacitances. In doing so they are able to increase the thermal quality of their products.

This article is based on the material presented at T3Ster User Forum Tokyo 2012 and Yaskawa's 2012 Annual Report. For further information visit: www.yaskawa.co.jp/en/ir/ir_doc01.html

## For further information contact

Dr. Kazunori Hakiai,
hakiai@yaskawa.co.jp
Reliability R&D Group,
YASKAWA ELECTRIC CORPORATION

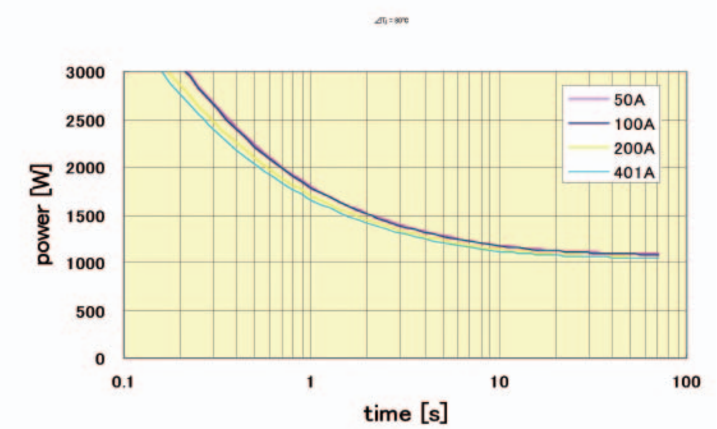
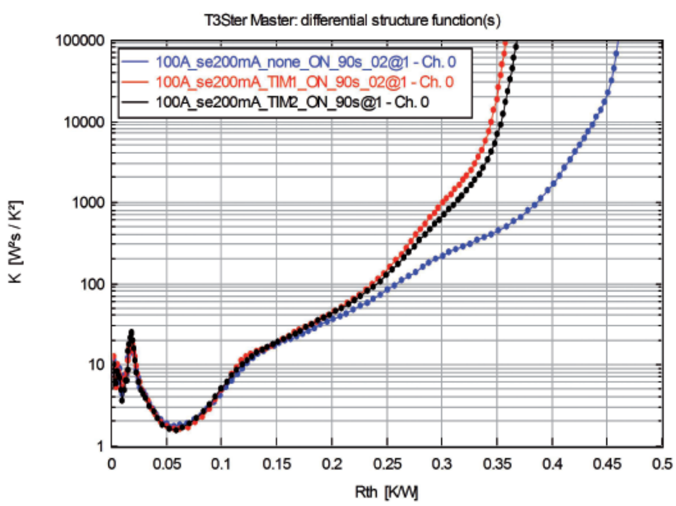



Figure 2



# FAQs

The abbreviation FAQ has become known to all of us who are looking for answers. When we try something new, we often are sent a list of frequently asked questions (FAQs) so that we can get started easily.

**I'VE had the privilege of working for Mentor Graphics for over 20 years. In fact, I joined in the same month that Sir Tim Berners-Lee made a proposal for "an information management system" which nowadays we call the internet.**

Back in those days, Frequently Asked Questions were asked frequently by phone. Sometimes questions and answers would be sent out by post. Working as an Application Engineer, I could come to work expecting that half the questions that I answered during the day were ones that I had been asked before. As an aside, it's worth noting that I HAD to come to work. Working from home was a much less practical proposition than it is now.

These days, when you want to know the answer to something, you go to the internet. That's why, over the last ten years, Mentor Graphics' Customer Support Division (CSD) has enhanced our SupportNet website to be as comprehensive as possible so customers can find answers themselves. As our Application Engineers share their experience with our customers, they continually create and share new solutions and advice.

Of course, all our customers are trying to create something unique for their companies. That means there will always be analysis problems that are unique to their organizations. The ability to discuss their unique analyses with our AEs is one of the most valuable aspects of a maintenance contract, alongside the update releases provided by the Mechanical Analysis Division development teams.

However, alongside those unique questions, we find the FAQs. When our support experts have a discussion with a customer and think "Other customers will come across that issue", then we write a Technote and put it on SupportNet. Furthermore, we email the latest content to our customers every week in our SupportPro technical newsletter. We try to tell customers the answers before they've asked the question.

So what's next in the evolution of support? Gradually, the content we publish on SupportNet is becoming more video and less text based. We're experimenting with putting some content outside the firewall, starting with things like licensing. We expect the Mechanical Analysis Community website to become more active in the future too.

Customers can now get answers to their challenges in more ways, in less time and when they want them. People like talking to people, and our customers will always appreciate talking to an AE who understands our software and can help apply it to their analysis problems. One thing is certain though - the web has changed support forever. www.supportnet.mentor.com.

**Richard Wilton**
**Customer Support Manager – Mechanical Analysis.**

I'm an Electrical and Electronics Engineering graduate of the University of Bristol. Since then I've worked as an electronics designer and in technical marketing before joining Mentor Graphics in 1989 as an Applications Engineer. In 1997 I became a Customer Support Manager and have worked with the Mechanical Analysis Division since 2009.

# Geek Hub

Welcome to Geek Hub! Have you ever wondered what CFD could reveal if it were used to calculate the ideal location for beer in the fridge or the aerodynamics of a hockey puck or the best place to put a radiator in a room? Well Geek Hub is the place to find out.

**OUR team are passionate about all things CFD and love sharing their findings. In this issue, Application Engineer and prolific blogger, Travis Mikjaniec dissects a Chopper motorcycle using FloEFD.**

A number of years ago, I was flipping through the channel guide and saw a show about choppers. At first, I thought I read copters, and being an aerospace engineer, watching a show on helicopters was something I was interested in. Of course the show was about motorcycles, but the first episode I saw was of a fighter jet inspired chopper. From that point on, I was hooked on all these types of motorcycle shows.

The other day, I came across this CAD model on www.grabcad.com, of a person's dream chopper. I wondered about the aerodynamics of a chopper, as just a couple weeks earlier I had seen a "Mythbuster" episode where they tested the fuel efficiency of a motorcycle vs a car. The motorcycle killed on fuel efficiency, but was poor on air pollution. Then they made an aerodynamic shell and tested it again to see if they could improve the fuel efficiency enough to overcome the bad air pollution. Interesting stuff for sure. Of course the motorcycle they used was nothing as cool as this. I'm sure unlike mass manufactured motorcycles, a custom chopper has never been wind tunnel tested or analyzed using CFD…until now!

## Set up

To begin, I set the airspeed to be 75 mph, because just like the song, "I can't drive …. fifty five"! Other than that, the setup was pretty easy. I just needed to figure out the RPM for the wheels at 75 mph, and I took an educated guess at the engine exhaust and intake air flow rate based on assumed cubic inches of engine and an assumed RPM. Then I created some CAD to represent a two lane road and some grass. I didn't get to making a CAD rider for the chopper, as I'm sure most of the drag would have to do with his riding position (leaning forward vs straight up), size, shape etc. and really what I was interested in was the bike itself.

## The Results

Now, the bike looks fast even just alone with the rack of the front forks and the long sweep of the gas tank, but in this first cut plot showing air speed and the mesh, doesn't it look like it's going even faster? If we look at the second contour plot of air speed, this time cropping out any air speed faster than 70 mph, you can see how far downstream the wake of the chopper extends. You can also see, at some point, the wake becomes unstable and oscillates, creating that wiggle at the end. I also like this plot because it shows that the wake is very compact in the width direction. If you were a person on the grass, you wouldn't feel the air move much when the motorcycle passes by, unlike with a transport truck where you definitely feel the air move.

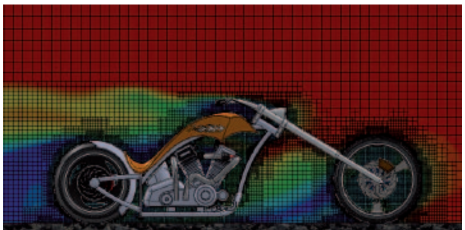


Figure 1. Chopper 75 mph FloEFD Velocity Contours and Adaptive Mesh Plot

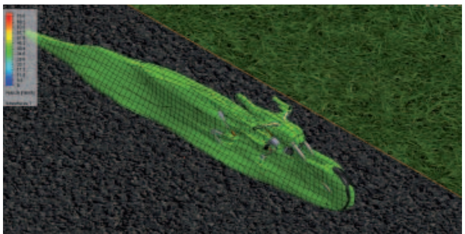


Figure 3. 45 mph Isosurface

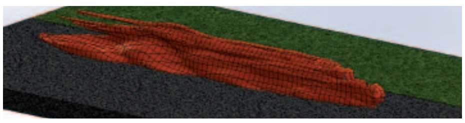


Figure 4. 70 mph Isosurface

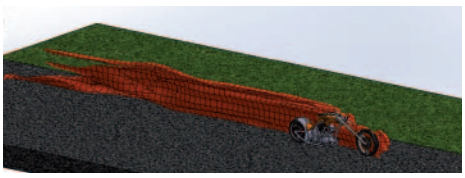
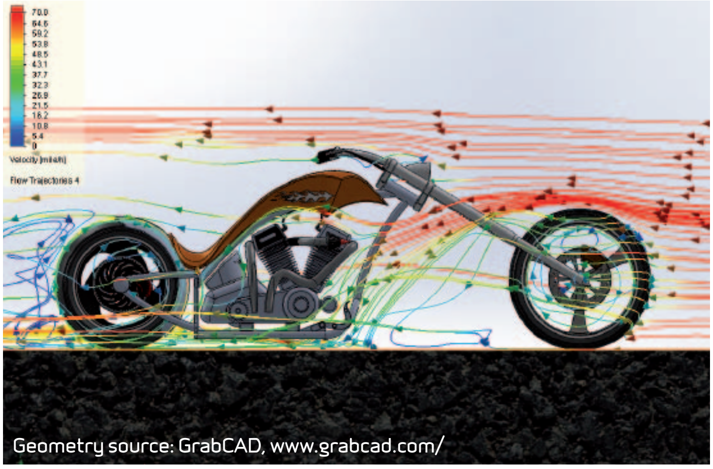


Figure 5. 70 mph Isosurface, cropped down centerline



Geometry source: GrabCAD, www.grabcad.com/

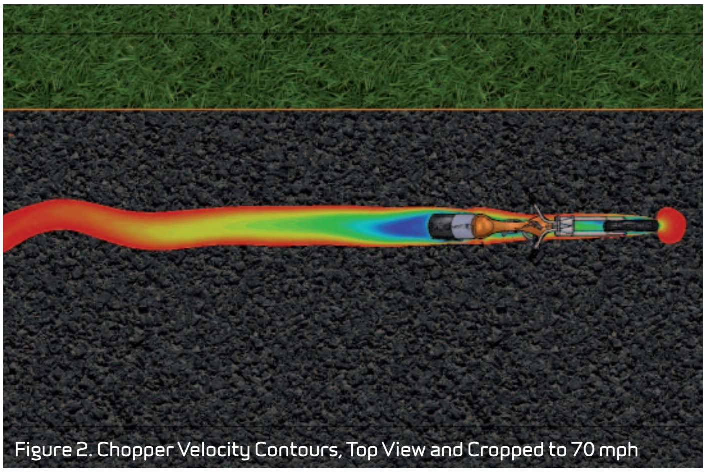


Figure 2. Chopper Velocity Contours, Top View and Cropped to 70 mph

If we change the contour to pressure, and zoom in a bit, we can see areas that are affecting the drag on the bike. Anywhere that there is high pressure on the front side, and low pressure on the rear side is creating a pressure drag force. You can clearly see this on the front wheel. Surprisingly this is also apparent on the headlight. I expected the engine would have a lot of drag, but the headlights? Well the headlights are not the only objects in this space; there are also the metal cross members that attach the risers to the body of the motorcycle to consider. These are solid chunks of metal, where no air can pass. Also, the headlights are experiencing fresh high speed air, so that's creating more pressure. The engine is getting the air that has been slowed when going around the front wheel, so the pressure isn't as high.

Next I made an isosurface connecting places with the same velocity values, one at a speed of 45 mph, and another at 70 mph. (Figures 3, 4 and 5) I added some grid lines to help show the contours of the isosurface. This really helps to show the extent of the wake. The 45 mph isosurface highlights the region that has undergone the most significant deceleration, and 70 mph isosurface shows the extent of the air that has been slightly disturbed. Now because the 70 mph isosurface extends so far, it's difficult to get a sense of its scale as it's obscuring the chopper. To overcome this, I made a cropped isosurface down the chopper centerline to show the isosurface interior. Now we can see the wake seems to extend at least three bike lengths downstream.

Of particular interest to me was the aerodynamics of the wheels. There has been much research into the aerodynamics of exposed wheels in open wheel racing. When it comes to custom choppers, the wheel design is always more about the cool factor than aerodynamics. This can always be improved with design so I created a surface plot of velocity on the front wheel. Obviously velocity is zero on a surface, so this plot is offset into the air slightly to get the near wall air speed. This is demonstrated in Figure 6.

The interesting thing to me, is the air going by the front wheel. Even though the wheel is rotating clockwise, you can see how the air moving over the top of the wheel flows downward, because of the engine. You can also see some clockwise flow within the wheel, but in general you have competing airflows in this area. Now because the air has a downward trajectory as it goes towards the engine this would appear to reduce the drag effects of the engine as there aren't any stagnation points on the front of the engine. The air is being deflected downward and away from the engine.

Another interesting air flow to examine is that of the exhaust air and its path. For the purposes of this demonstration the exhaust pipe geometry was made partially transparent to show the flow inside the pipes. Every custom chopper design has to have cool pipes, so some designs may put the hot exhaust gases where we don't want them. Looking at the animation, we can see some exhaust gas gets entrained with the air circulating around the rear wheel, but the exhaust gas cools very quickly. Again, I took an educated guess for the gas flow rate here.

Similarly, we would be interested in seeing how the air gets to our engine intake. We don't want to put our intake in an area with poor airflow as that could starve the engine and reduce the power output. We can see here, that other than a little wiggle to go around the front wheel, the air has a nice straight line into the intake.

Of course we could dissect this model further, looking at forces and torques numbers on different components or the entire chopper. But I think I've learned a good amount about the aerodynamics of choppers. I hope you did too. This took very little time to analyse as I already had the CAD geometry, and FloEFD works within the CAD system and automatically meshes the model. Until next time!

**Travis Mikjaniec**

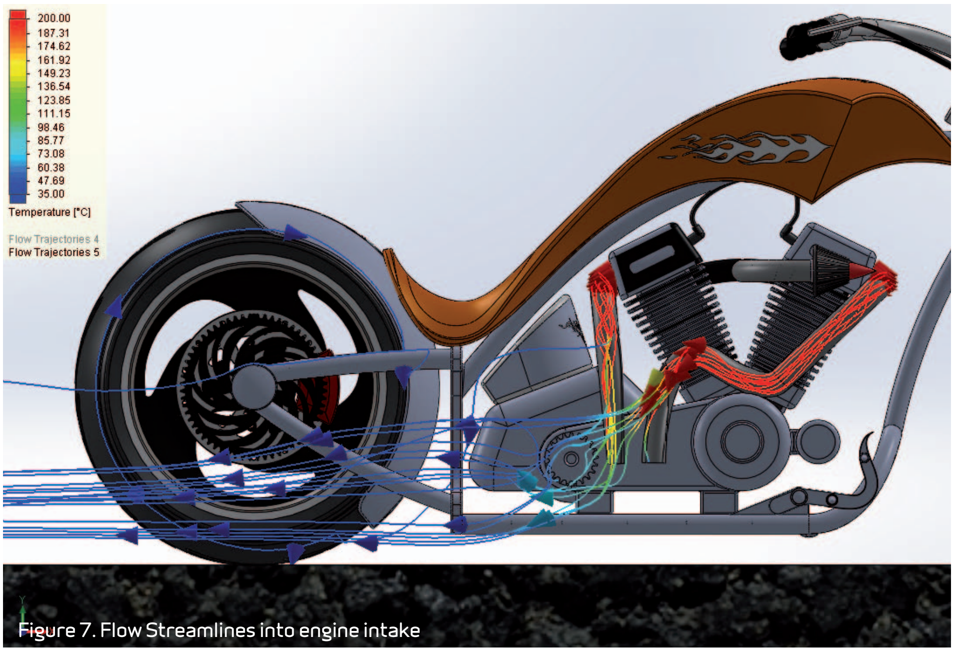



Figure 7. Flow Streamlines into engine intake

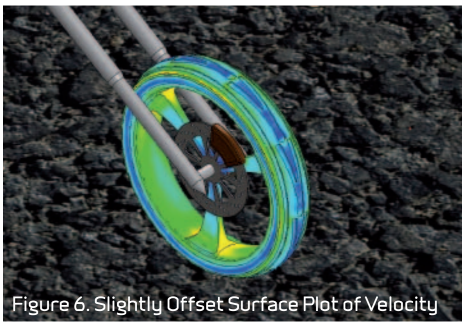


Figure 6. Slightly Offset Surface Plot of Velocity

# Athlete Engineered Technology Fine Tuned by CFD

## The Science that could bring in Winter Olympic Gold Medals for Bob Skeleton Athletes

Bromley Technologies Founders: CEO Kristan Bromley (right), & Structural Dynamics & Systems Director Richard Bromley (left).

Shelley Rudman, currently ranked #1 Skeleton athlete in the world

**A GREAT skeleton ride combines athleticism and courage with the science of physics and meticulous engineering design, according to Professor Kristan Bromley. Over the past 18 years, Professor Bromley, aka Dr. Ice, has used this powerful combination to personally win the 2003-2004 World Cup and the 2008 World Championship titles and three European Championship titles. This winning streak was no fluke, when following the reinstatement of Skeleton at the Winter Olympics in Salt Lake City, Kristan's scientific approach went on to support consecutive medal winning performances as part of a winning streak for British athletes.**

Kristan, with a doctorate in the science and engineering behind the Skeleton sport, is CEO of Bromley Technologies Ltd. which he founded with his brother Richard and supports fellow athlete and 2012 world number one Shelley Rudman. Their impressive offices are located on The Advanced Manufacturing Park in Sheffield, England, where they neighbor aerospace giants such as Rolls Royce and the AMRC Boeing. It is here that Kristan works on developing high-performance sports products with a small but dedicated group of engineers and world-class athletes. Their company focuses on developing and manufacturing cutting edge sports products and technology that give Olympic athletes and professional teams a competitive edge. Bromley has forged a world-leading position in Skeleton Sled sport, however their expertise also extends to projects in bobsleigh, luge, composite hockey sticks, sprint footwear and bobsleigh track simulation. More recently Bromley has been nominated to partner the International Skeleton & Bobsleigh Federation in developing the sport's first Paralympic sled designs.

It's the skeleton bobsleigh that has impassioned Professor Bromley. For those unfamiliar with the sport, to briefly describe it would be to say that is it a 90 mph head-first, face-down hurtle on a narrow one man sled down a 1,500 meter ice track against a roaring 5g-force. Runs are timed electronically to the nearest hundredth of a second, and athletes are continually striving to shave off time in pursuit of the perfect run and medal success.

It's a highly tuned performance, and each sled is customized to the athlete. The athlete must be in unison with the sled and the track physics to successfully complete the course. The track is negotiated by the athlete's body using only shoulders and knee pressure on the sled and the occasional 'toe on ice' to induce subtle steering effects. There are no brakes or active mechanical steering elements. Not only is this a great physical feat but also one of mental strength and courage, because there is no place for fear as athletes hurtle down the track at dangerously high speeds, one tiny mistake could be fatal.

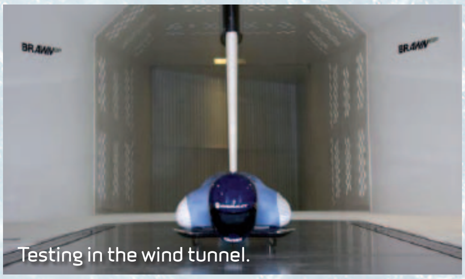
In preparation for the 2014 Sochi Games, Kristan and the experts at Bromley Technologies are putting their sport, engineering, and design knowledge to work to create the fastest sleds on the planet. A meticulous approach to design and testing is being undertaken, consisting of four stages. Stage one, which took place this past summer, included a detailed analysis of the aerodynamics of the athlete-sled system. The focus was on developing equipment that worked optimally with each individual athlete's form. Stages two and three see the team trialing their findings and fine tuning the design and set-up before stage four, the final production ahead of the winter games.

Kristan started his career as a graduate engineer for British Aerospace and is very familiar with Computational Fluid Dynamics (CFD) modeling and simulation techniques, having designed and tested his first sled some 17 years ago for the British Bob Skeleton Association. Kristan uses FloEFD™ modeling and simulation software to optimize the design of the skeletons. Doing the simulation himself, Kristan comments, "When I use a traditional CFD approach to do aerodynamic simulations, it can take weeks to get results back but now I can use engineering feedback within hours."

"An iterative approach is taken with new projects progressing from design to design. During a competitive season I can observe any new developments made by my competitors and combine these with our own ideas hot off the Bromley 'design board'. FloEFD enables me to quickly analyze these ideas to make an initial assessment before further detailed analysis is performed later in the program. It's an extremely efficient way to work in very unforgiving timescales."

For analysis, Kristan uses various flow visualization techniques, including flow lines, pressure contours and maps as well as other data gathered from testing. "I use a mix of visual and numeric outputs to inform me of what's actually happening with respect to air flow around the athlete and the sled. Visualization techniques are extremely powerful, allowing an intuitive approach to design innovation. Numerical outputs enable the direct impact of design changes to be assessed in relation to performance. Drag values from FloEFD are input into Bromley's own track simulation software in order to convert drag saving into time savings. A key part of working closely with athletes is in understanding how the changes made in one area affects other areas. For example, aerodynamic design improvements may have a secondary structural affect on the sled which would change its behavior. This means that design modifications must be made with consideration to how the change will ultimately affect the athlete."

A combination of FloEFD and Pro/ENGINEER Creo™ is used for aerodynamic analysis. In keeping with Kristan's "Athlete Engineered Technology" philosophy, he said "We're testing a variety of shapes that work with the geometry of the athlete. When we're talking about an athlete lying on a sled traveling at speeds close to 90 mph, the first part of the equation is the equipment, (the helmet and the sled), the other parts being the athlete and the track. Our goal is to bring all those components together into one system. We can't change the shape of the track or the athlete's body but we can change the shape of the other components to try and optimize the system. The sport is governed by very tight rules and regulations which limit large advancements. Our aim is to find multiple gains, each with a small percentage improvements that combine to make a tangible difference on the track."



Testing in the wind tunnel.

The engineers at Bromley Technologies test their CFD predictions against wind-tunnel measurements to validate their results and be assured of accuracy. Reserving time in the same wind tunnel used by Formula One racing teams is expensive, so Kristan aims to use the FloEFD data, benchmarked in the wind tunnel with comparable results, to make multiple design changes within hours rather than days.

Over the past two years, Kristan and his team have worked to narrow down the optimal shape for the sled using simulation results in FloEFD. "We needed to combine aerodynamic design improvements with design modification made in other areas. This was one of our biggest challenges because any modification in one area must not negatively impact upon other areas of sled performance."

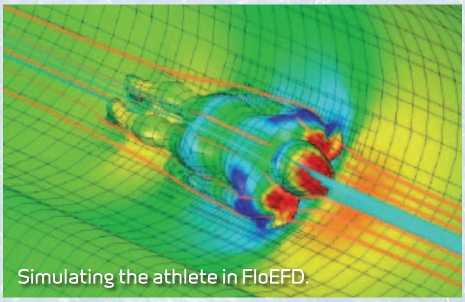
The Bromley team has been painstakingly working on numerous sled geometries over the last two years. This has resulted in a shortlist that was tested on ice at the 1994 Olympic Bobsleigh and Luge Track in Lillehammer, Norway in October. "Our aerodynamics program runs in parallel to other areas of development."

"This is a critical stage of the four year development program, where we find out if the virtual gains highlighted in FloEFD translates into real performance improvements on the track. "We need to get on the ice to test the sled. This is the only way we will know if the innovations result in performance gain."

Following testing in October the team will have a busy schedule in order to move into the final development stages in preparation for the Winter Games of 2014, which are less than 500 days away. We will be catching up with Kristan in the next issue and reporting on their results from the track testing.

### For More Information…

Kristan Bromley: www.kristanbromley.com
Twitter: @kristanbromley
Bromley Technologies:
www.bromleysports.com



Simulating the athlete in FloEFD.

# Structure Functions – The Bridge Between Thermal Measurement and Thermal Simulation

**TRADITIONALLY thermal measurements of electronic components have been done using thermocouples. However, thermocouples come with critical disadvantages, (such as contact thermal resistance between target and probe which makes the measured result very unstable, and the tendency of the thermocouple to conduct heat away from the surface being measured) making them near useless for measuring the surface temperature of plastic packages.**

Increasingly there is a need to validate thermal models during product design to ensure that the product will perform as designed, by confirming material properties and the thickness of thermal interface material (TIM) layers. A major issue with thermocouples is that they simply cannot measure the temperature of the internal structure. Yet by design, the dominant heat flow path is from the junction, through many materials and material interfaces before passing into a PCB or heatsink, whose temperature can be conveniently measured. Even then, a thermocouple only provides a single temperature value. Thermocouples are therefore a 'blunt instrument' when it comes to thermal design verification.

In this article, we are going to introduce the thermal structural analysis method, which is based on transient thermal measurements, that allow the thermal behavior of the system, including heat spreading, to be characterized as a distributed Resistance-Capacitance (RC) network. By measuring the transient response of junction, we can easily observe the heat spreading path inside package, board, TIM and heat-sink, etc.

## The Challenge of Thermal Analysis in the Real World

The best way to study thermal structure is to take a look at the isothermal distribution or heat flux distribution along the heat-spread path. However, in the real world it is impossible to take a picture of heat distribution inside any solid object. The only way to inspect heat flux distribution vitually is by using a software simulator such as FloTHERM™ used in this article.

According to the theory [1], thermal systems are distributed RC systems, which can be modeled by thermal resistance Rth and thermal capacitance Cth. To evaluate a RC system, the most common way is to measure transient response under a step power excitation.

Consider the experiment setup in Fig 1. Ideal heat insulation material prevents heat from escaping to Y and Z direction, the cold plate at the right side of X axis provides an ideal thermal boundary condition. In this setup heat flux will be constrained to X axis which can be considered as one-dimentional heat spreading path starting from the heat source on the left side to the cold plate on the right side along X axis.

Thermal property Rth and Cth on the heat spreading path determines step power response of the system, theoretically we can evaluate the thermal structure by measuring the thermal transient response in an electrical test method as standardized in JEDEC JESD 51-1 in 1995.

In the experiment, we place three kinds of flag material in the middle of heat path.

1. Same as pure copper. (Cu50W)
2. Doubled specific heat against pure copper (Cu50W_2xCth)
3. Halved thermal conductivity against copper. (Cu50W_2xRth)

Figure 2 plots step power responses and Structure Function. In temperature response view, variation caused by different flag material can be seen, however it is not clear enough while in the Structure Function view the structural information can be clearly identified as shown in Figure 3.

## Case Study Closer to a Real PCB Board Application

In a real-world application such as a package mounted on PCB board, heat spreads not only vertically but also horizontally as shown in Figure 4.

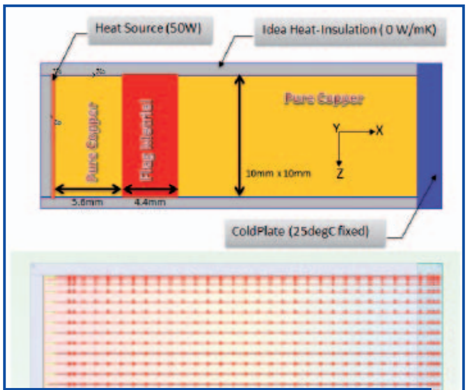



**Figure 1.** Experiment setup and heat flux view

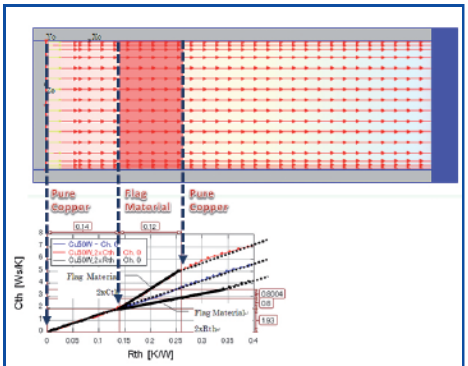



**Figure 3.** Rth in structure function increases along heat path

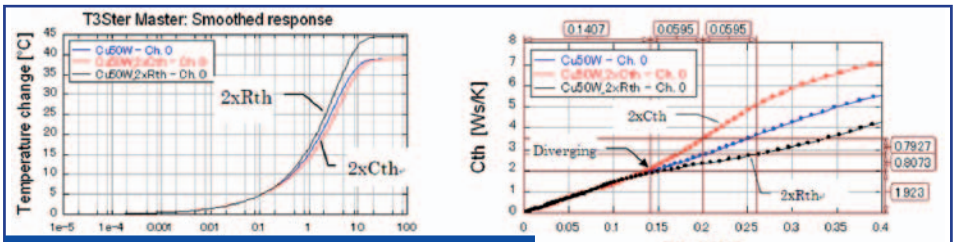



**Figure 2.** Step power response and its Structure Function

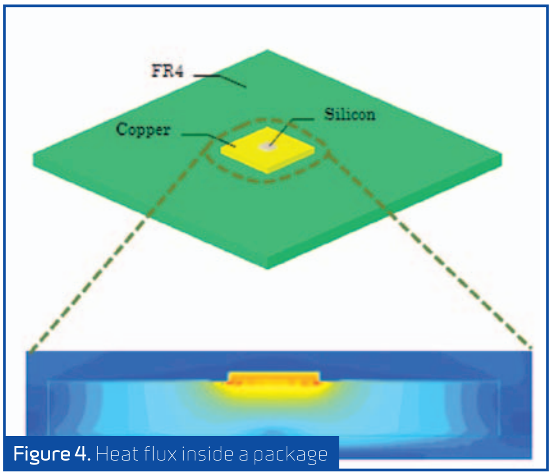


Figure 4. Heat flux inside a package

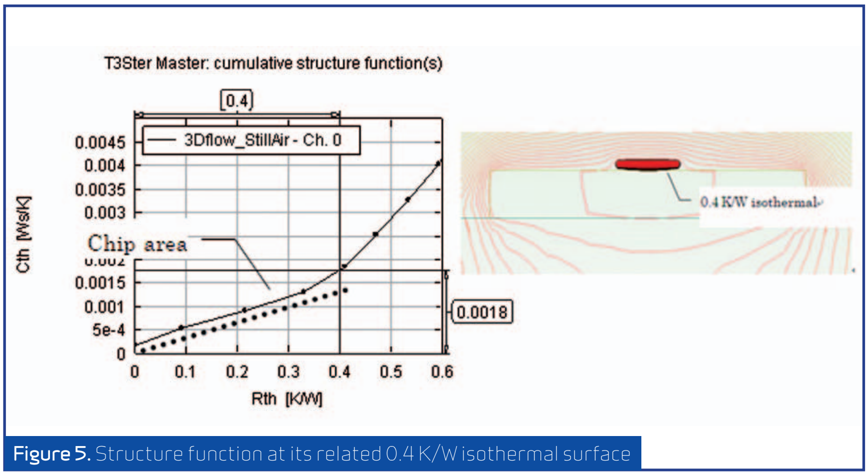


Figure 5. Structure function at its related 0.4 K/W isothermal surface

The heat source (silicon chip) is attached to a metal (copper) substrate and then attached to the FR4 board. Every material is built as a cuboid block and contact thermal resistances are not included for simplicity.

In the Structure Function, a straight line section from 0 - 0.4 K/W can be seen at the beginning. This straight line comes from the nearly 1D heat flow inside the chip as shown in Figure 5. This is because air outside the silicon chip has relatively huge thermal resistance compared to silicon, so that heat is forced to go through the thickness of the chip as discussed in the previous section. After 0.4K/W, structure function curve goes up exponentially which is caused by the 3D heat spreading in the metal substrate as shown in Figure 6.

For the same reason, Structure Function from 0.8K/W - 1.2 K/W also indicates heat spreading in the copper block and the Structure Function curve shows an increasing slope. After 1.2K/W we observe a decrease in the slope of the curve. This is caused by the physical boundary of metal substrate.

## Conclusions

Traditionally when doing thermal analysis, thermal models built in CFD simulation software contain many thousands of pieces of data. The challenge for the user is how to verify the correctness of the model. As Structure Functions can be obtained from both experiment and simulation, we are now able to verify package thermal models against the data for real packages by comparing their structure functions. If there is any mismatch we can easily identify and resolve the problem, thereby increasing the fidelity of any board or system-level models in which the package model is used [2]. As Structure Functions track the heat flow path from the die junction to the ultimate ambient, the technique can also be applied to board and system-level models in late design to qualify electronics products before they go into full production.

## Note

This article is edited from the paper published in the Transactions on The Japan Institute of Electronics Packaging, Vol. 5, No. 1 (Dec. 2012).

## References

[1] V. Székely and T. Van Bien, "Fine structure of heat flow path in semiconductor devices: a measurement and identification method," Solid- State Electron., vol. 31, pp. 1363–1368, 1988.

[2] Andras Vass-Varnai, Robin Bornoff, etc., "Thermal Simulations and Measurements – a Combined Approach for Package Characterization". ICEP 2011 Japan.

[3] Yafei luo, "Structure function based thermal resistance & thermal capacitance measurement for electronic system". CPMT Symposium Japan, 2010 IEEE.

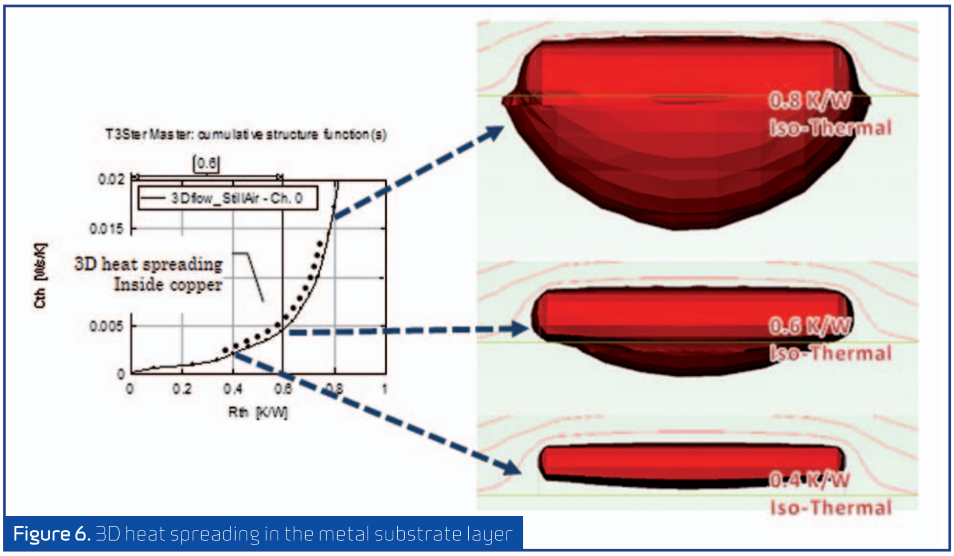
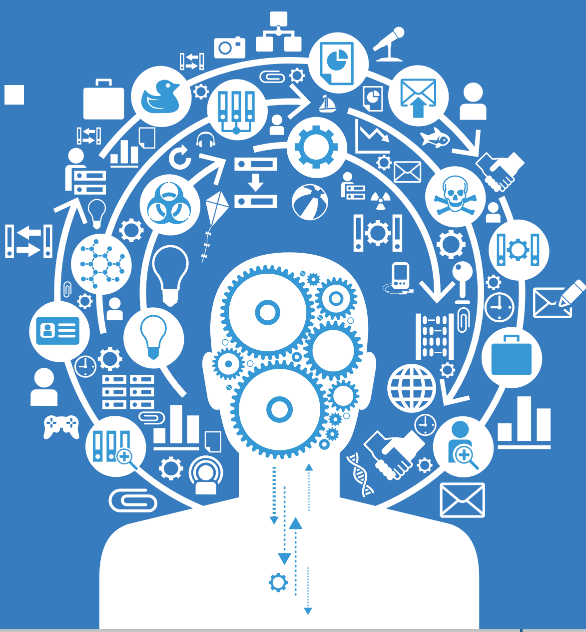


Figure 6. 3D heat spreading in the metal substrate layer

# Brownian Motion...

## The random musings of a Fluid Dynamicist

**Brownian motion** or **pedesis** (from Greek: **πήδησις Pɛdɛ:sɪs** "leaping") is the presumably random moving of particles suspended in a fluid (a liquid or a gas) resulting from their bombardment by the fast-moving atoms or molecules in the gas or liquid. The term "Brownian motion" can also refer to the mathematical model used to describe such random movements, which is often called a particle theory.



# Did someone say supersonic, interplanetary sky crane?

You had me at laser armed, atomic robot-truck

**BEING an engineer means you get used to certain labels or stereotypes; my own wife has been known to ask me to tell her about my working day on those occasions she struggles to get to sleep. Unfortunately for her, another trait we allegedly possess is the inability to pick up on signals such as 'sarcasm', 'painful levels of boredom' and 'regret', so I'm happy to do so, at some length.**

The truth is, I don't so much accept as actively embrace this pigeon-holing. Why? Well, for a number of reasons: I'm a very easy man to buy Christmas presents for (FYI, the answer is clothes. If the engineer in your life is beginning to look like a hobo, that's your fault for buying them a picture frame for their last birthday); thinking of buying something which has at least one of the following components: a screen, a keyboard, wheels, tracks, a laser? I'll give you one guess as to who has already read all the reviews and tech spec's (I will admit that I rarely get asked for laser advice, but the point is I'm ready for it); you will never have any IT issues at home (you may find you're now on Linux, but you'll understand eventually); you need never leave the cinema ignorant of all the technical goofs the film possessed and no, you won't even have to ask. I could go on, but I think you see the point I'm making.

Ultimately, I think that being a nerd is mainly about being enthusiastic about something. The reason it often has such negative connotations is probably down to apathy being such a celebrated trait in popular culture. Don't believe me? Compare and contrast just about any interview with a Hollywood Celebrity with the footage that came out of the Jet Propulsion Laboratory following the successful arrival of the Curiosity probe on Mars. One will be an exercise in studied mono-tone, indifference and intermittent eye-contact, the other… well, if you haven't seen the footage, I can only encourage you to seek it out. I mean, really encourage you: they put a one ton, six wheeled rover on Mars using a parachute that opened at just under Mach two, rockets and a crane that operated while still in flight. You can begin reading that last sentence at just about any point after the colon and it would still contain enough information to make me go slack jawed in wonder.

It also has a laser on board, but I wasn't asked.

Anyway, here's to us who know how to appreciate the big things in life precisely because we're not afraid to celebrate the details.

Turbulent Eddy