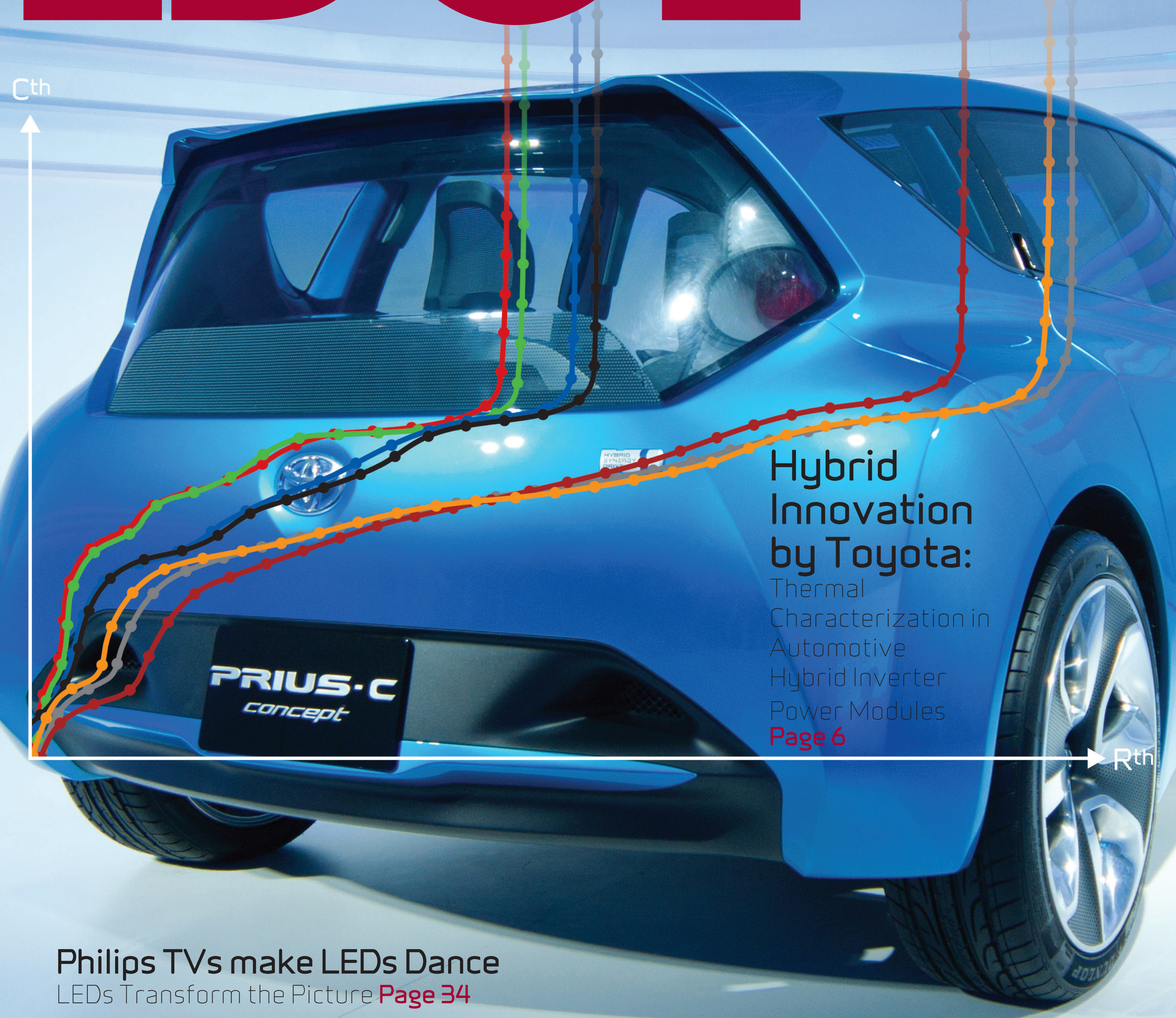
Vol. 02 / Issue. 03 / 2013

ENGINEERING EDGE

Accelerate Innovation with CFD & Thermal Characterization



Hybrid Innovation by Toyota:

Thermal Characterization in Automotive Hybrid Inverter Power Modules Page 6

Philips TVs make LEDs Dance

LEDs Transform the Picture Page 34

Voxdale & Bernico International:

Breaking a World Water Speed Record Page 18

Mentor Graphics®

Mechanical Analysis

mentor.com/mechanical

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:
Robin Bornoff, Mathew Clark, Barbara Hazard, Keith Hanna, John Isaac, Andrey Ivanov, Boris Marovic, John Murray, John Parry, Andras Poppe, Tatiana Trebunskikh, Jane Wade, Richard Wilton

With special thanks to:
Bromley Technologies
Creative Solutions
Curtiss-Wright Controls, Defense Solutions
EnginSoft sPA
Eurotecnica
FAdeA S.A
NMB Minebea GmbH
TP Vision Lab (formerly Philips TV)
Toyota Hybrid Vehicle Development Group
Tupolev
Voxdale BVBA

Perspective

Vol. 02, Issue. 03

Where were you a quarter of a century ago? A tough question for some of us perhaps, and I even guess that some of the Engineering Edge readership may not have been born 25 years ago! I personally recall that in 1988 I was an eager, young salesman with SDRC in Germany, the Cold War was at its peak, Ronald Reagan, Helmut Kohl, Mikhail Gorbachev and Margaret Thatcher were all striding the world political stage, and the earth was three minutes to midnight on the metaphorical 'nuclear clock' we all feared. How the political and technical world has changed beyond belief in the last 25 years. Indeed, for some of us longer in the tooth, 1988 may feel like another country it's so long ago. Whichever it is, we are thinking of the last 25 years in this edition of Engineering Edge because October 3rd 1988 was the founding date of the company that became Flomerics Ltd and thus the foundation date of Mentor's Mechanical Analysis Division, and our world-leading FloTHERM® productline for electronics cooling.

It was in October 1988 that the attic of 148 Elm Road, New Malden, London, saw two visionaries, Dr David Tatchell and Dr Harvey Rosten, who hailed from Brian Spalding's Imperial College and CHAM 'hothouse' of commercial CFD, deciding to found their own company. Their vision was to go after emerging industry-specific markets for CFD, such as electronics cooling and HVAC with design-focused tools for engineers based on 'good science'. This was a game changer for the CFD industry and out of it came the FloTHERM electronics cooling productline and FloVENT® for HVAC. David and Harvey's foresight, wisdom, hard work and, not least, their own personal capital set the ball in motion. They first hired three young, keen engineers; James Dyson, Dave Kirkcaldy and Peter Hall. Three more employees joined the team in 1989, Mike Reynell, Hasan Moezzi and John Parry; and others followed suit, many still with Mentor today. I'm pleased to congratulate James, Dave and John for their longevity in the CFD field and, indeed, they are still in our Hampton Court office at the center of our development and marketing efforts as the FloTHERM productline continues to grow from strength to strength.

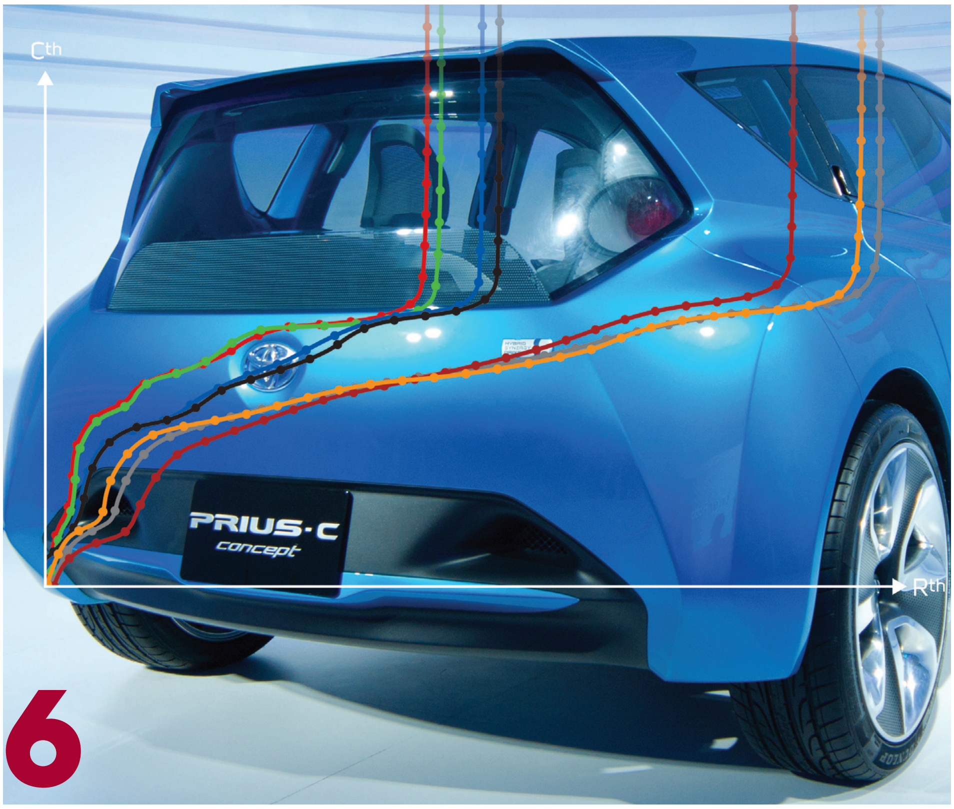
To celebrate the tenacity of the two founders of Flomerics, and the FloTHERM productline thrust over the last 25 years, we have produced a special supplement, Electronics Thermal,that pays tribute to the achievements in this field over the last 25 years as it has expanded to encompass many more co-simulation and test capabilities.David Tatchell shares his thoughts on the last 25 years, whilst the leading EDA Analyst group, Gary Smith Associates, contextualizes 25 years of electronics cooling in the greater trends of Moore's Law, consumer electronics advances and simulation advances. An interesting read that includes our 25 year FloTHERM timeline poster in the centerfold. See also our interactive timeline by visiting www.FloTHERM25.com

Finally, in this edition, we have a wealth of customer application stories led by our cover story from Toyota in Japan where Hisao Nishimori summarizes how thermal characterization is being integrated into their innovative Hybrid Car Inverter Power Module development process to speed up product and delivery efforts. There's a fascinating article on how Philips in the Netherlands develop modern big-screen TVs with LED back lighting, the design of a World Water Speed Record boat, Tupolev aircraft aerodynamics and FAdeA turboprop fuel system design, heatsink design by NMB Minebea, and Curtiss-Wright share how our tools are making a difference in the real world. I commend this, our biggest Engineering Edge ever to you and I wish you happy simulating for the next 25 years with the FloTHERM product line!

Erich Buergel

Dr. Erich Buergel, Vice President,
Focus Products Organization, Mentor Graphics

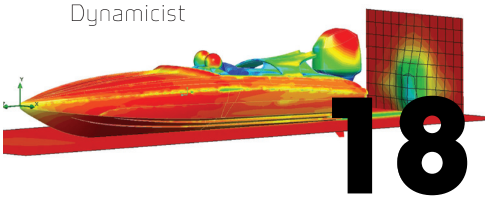
Contents



News

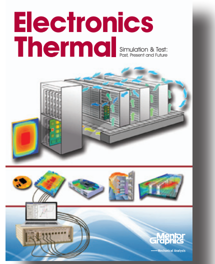
Technology & Knowledge Bank

Regular Features



Engineering Edge

Electronics Thermal Supplement



New Release: FloEFD™ 13

The latest release of FloEFD 13.0, announced in September 2013, offers increased functionality and usability enhancements.

Incorporated into this release is Monte-Carlo radiation modeling for accurate thermo-optic simulation, and advanced multicore meshing of complex geometries which is now orders of magnitude faster than previously available.

Monte-Carlo radiation modeling allows for precise analysis of geometric effects in optics, such as the refraction and reflection of radiation absorptive media; for example, lens focusing. FloEFD 13 provides a unique boundary layer treatment and, when combined with robust meshing capable of capturing highly complex geometries automatically, enables engineers to accurately predict complex thermo-optic behavior. This is ideal for applications such as automobile lighting, consumer luminaire design, and video/photography-related products.

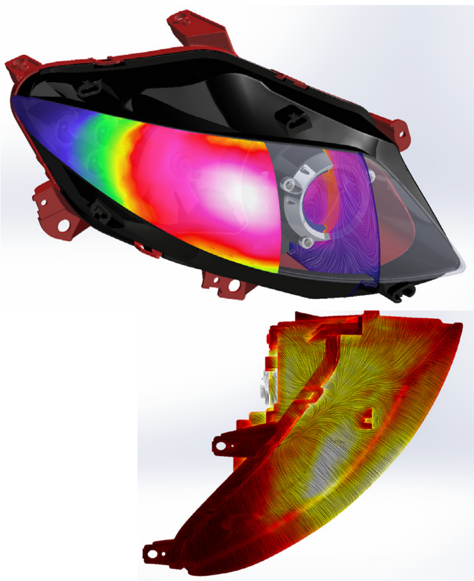


Figure 1. Monte-Carlo Radiation Modeling of an Automotive Headlight

"FloEFD from Mentor helps us to understand and optimize headlamps. Even very complex geometries and test conditions can be investigated with a minimum of effort," stated Peter Jauernig, head of department, Automotive Lighting, Reutlingen, Germany. "New features such as Monte-Carlo radiation and the LED module are especially helpful in speeding the development of very complex products."

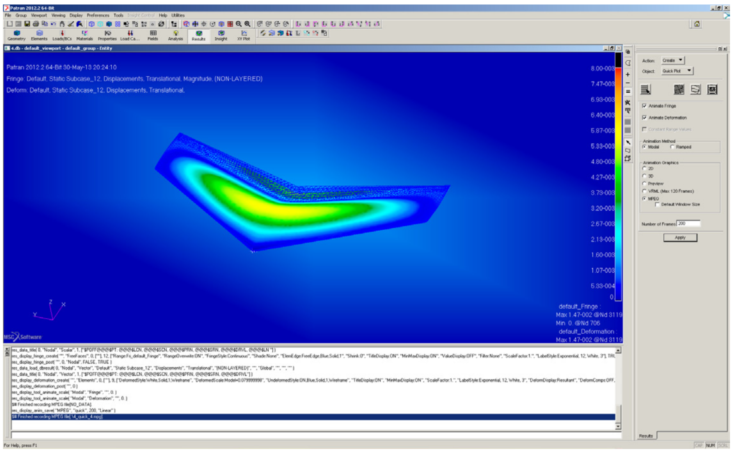


Figure 3. Finite Element Analysis

The multicore architecture meshing feature provides reduced meshing time, up to 20x faster than previous methodologies. (Figure 2). This significant increase in speed allows users to test various design approaches in a matter of minutes, compared to hours. The increased meshing speed provides significant productivity benefits such as conducting 'what-if' testing on various models in less time. The result is delivering competitive products with improved performance, in shorter time-to-market schedules.

FloEFD 13 also supports fluid-structure interaction through a mesh-based parallel code coupling interface (MpCCI) bridge developed at the Fraunhofer Institute SCAI. The new functionality enables engineers to export CFD analysis data for finite element analysis (FEA) in a wide range of popular structural simulation programs thereby enabling users to conduct multi-disciplinary analyses to obtain true-to-life simulation results (Figure 3).

Additional enhancements in this release include:

• Support of Linux solver launch directly from the FloEFD user interface to help users set a series of simulation runs effortlessly;
• Inclusion of multiple parametric study results in the same project tree provides easy access to data while conducting multiple studies on the same model;
• Multi-editing of input data on boundary conditions saves time and clicks for repetitive geometries such as LED arrays; and
• Support of local solution adaptive refinement enables users to specify high resolution only in areas of interest.

This latest version of FloEFD for concurrent CFD addresses the complexities of today's thermo-optic applications with greater ease of adoption and accuracy. As an industry leader with a proven history in CFD technology excellence, Mentor Graphics is continually investing in the FloEFD product line to deliver the best solutions that address customers' technical challenges.

For more information visit our website: www.mentor.com/products/mechanical/products/floefd

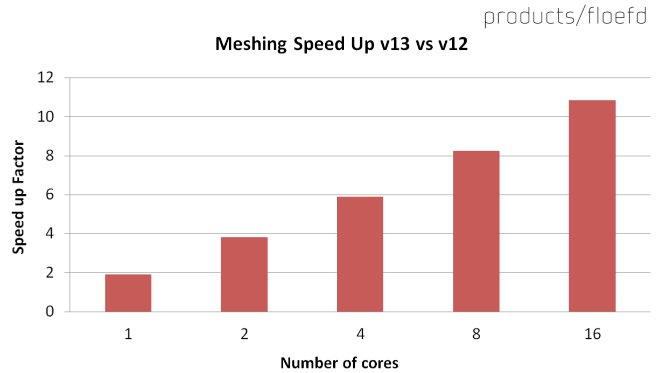
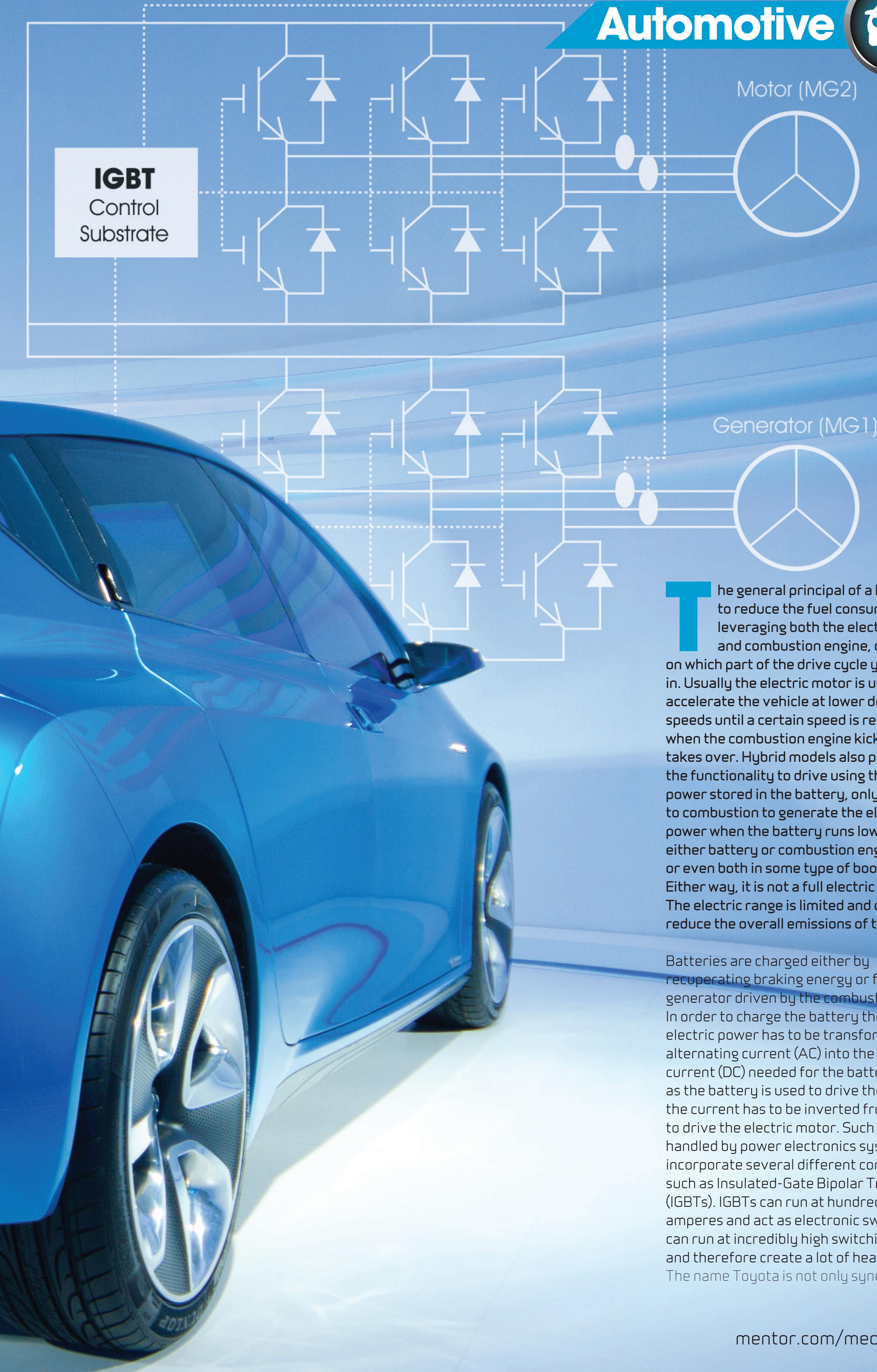


Figure 2. FloEFD Multicore Architecture Meshing

Hybrid Innovation by Toyota

Thermal Characterization in Automotive Hybrid Inverter Power Modules

By Boris Marovic, Product Marketing Manager, Mentor Graphics



The general principal of a hybrid is to reduce the fuel consumption by leveraging both the electric motor and combustion engine, depending on which part of the drive cycle you are in. Usually the electric motor is used to accelerate the vehicle at lower driving speeds until a certain speed is reached, when the combustion engine kicks in and takes over. Hybrid models also provide the functionality to drive using the electric power stored in the battery, only switching to combustion to generate the electric power when the battery runs low, or runs on either battery or combustion engine solely or even both in some type of boost mode. Either way, it is not a full electric vehicle. The electric range is limited and only used to reduce the overall emissions of the vehicle.

Batteries are charged either by recuperating braking energy or from a generator driven by the combustion engine. In order to charge the battery the generated electric power has to be transformed from alternating current (AC) into the direct current (DC) needed for the battery. As soon as the battery is used to drive the vehicle, the current has to be inverted from DC to AC to drive the electric motor. Such tasks are handled by power electronics systems that incorporate several different components such as Insulated-Gate Bipolar Transistors (IGBTs). IGBTs can run at hundreds of amperes and act as electronic switches that can run at incredibly high switching rates and therefore create a lot of heat.

The name Toyota is not only synonymous

with environmentally conscious vehicles, the most well-known could arguably be the Prius, but also with innovation. Hybrid Synergy Drive or HSD is the term designated to the technology Toyota has developed for their range of full hybrid vehicles. (Figures 1 &2)

Toyota's effort in hybrid electric vehicle development is well known with the first hybrids on the road in the late 90´s. Since then the total production has reached millions globally, and is set to grow in the future.

Hisao Nishimori is an Engineering Manager at Toyota's Hybrid Vehicle Development Group. In a presentation given at Mentor Graphics' Tech Design Forum 2012 in Shinagawa, Japan in September, Nishimori discussed the work Toyota has done to characterize and optimize the thermal characteristics of power modules for Hybrids and Electric Vehicles.
IGBTs are power electronic components that need to be optimized to have the ideal thermal resistance to better dissipate the generated heat of the component to the cooling system it is attached to. Applying Mentor Graphics' T3Ster® Transient Thermal Characterization system with the appropriate high-current boosters enables hybrids to reach the required power to heat the component, record the thermal response and derive the structure function of the component.

Prior to implementing T3Ster functionality, Toyota's design process began with the design of the module that was then built into a prototype and tested under different conditions such as switching frequency or grease thickness for the IGBT etc. Measurements were then taken with a thermal camera and the results evaluated. An unsatisfactory or inaccurate result required the team to go back and change the design and repeat the steps.
Nishimori says, "When the design changes again, the re-evaluation of a new prototype took a lot of lead time, effort and cost."

Toyota's goal is to optimize the process to achieve highly accurate and repeatable results in a shorter time and therefore be more efficient. "T3Ster gave us a detailed insight into the structure of the component and its heat flow path. It shows the difference in thermal resistance in the vicinity of the tested element clearly and the measurements can be verified before and after durability tests to identify any changes in the heat dissipation characteristics."
Toyota was also able to detect details in the structure, even in molded components when measuring all elements of the module at once. "We were even able to measure the bonding material under the element that matches the design value."
With the help of T3Ster, Toyota is able to

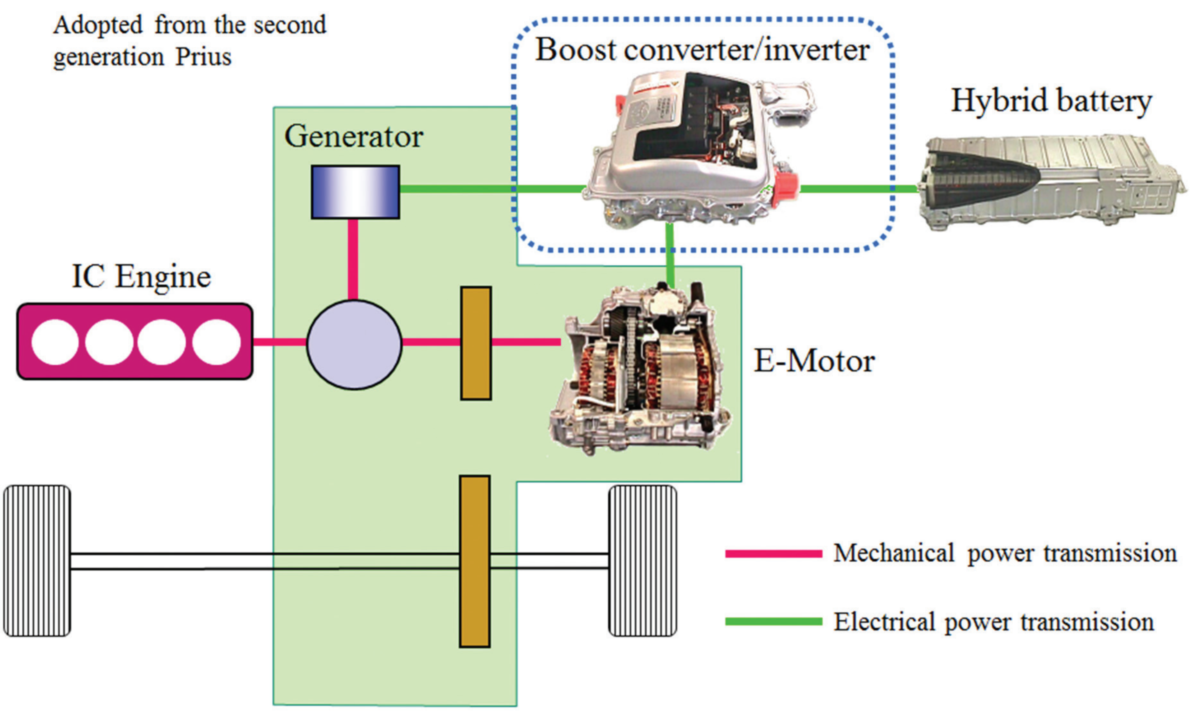


Figure 1. Overview of Toyota Hybrid System

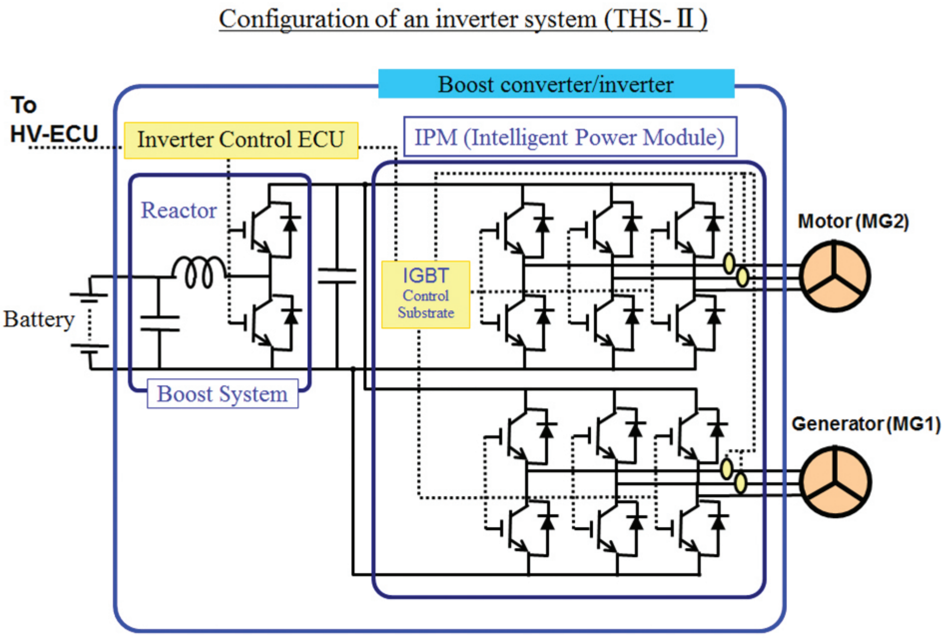


Figure 2. Configuration of an Inverter System (THS-II)

quickly feedback highly accurate results to the designers in order to further improve the design. Everything from grease thickness to bonding state could be optimized. Previously Toyota was unable to verify the status of the product and its heat dissipation in the vicinity of the molded element.

In summary Nishimori says, "By delivering the structure function from the measurement results by T3Ster, the comparison and verification of the design values can be done in more detail. The application of this measurement in an endurance test has helped us to uncover weaknesses in the design affecting its lifetime, and a general front-loading is now made possible because the design values could now be measured. And we can reduce the prototyping costs by reducing the rework on our designs." (Figure 3)

Nishimori expressed his view of the design and evaluation process perfectly in an automotive metaphor:
"Evaluation and Design are the two 'Wheels' of quality assurance and the important 'Engine' gives the driving force to the wheels. With the right combination of CAD, CAE and automated measurements the wheels will bring the combined power of 'Design Quality'onto the road." (Figure 4)

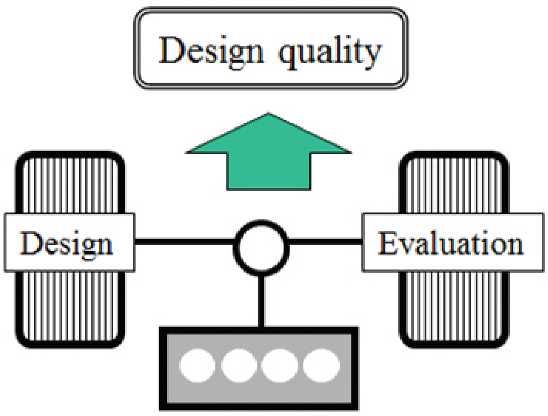


Figure 4. Design & Evaluation Process

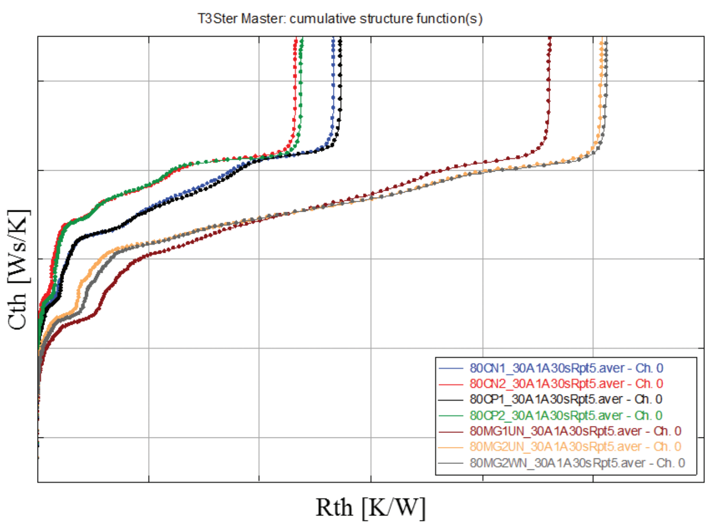


Figure 3. T3Ster Cumulative Structure Functions

Benchmarking takes off for NASA CRM & Tupolev Tu-214

FloEFD™ Simulation of External Aerodynamics

By Tatiana Trebunskikh & Andrey Ivanov, Mentor Graphics

External aerodynamics is a theoretical basis for aerospace technology, and aerodynamic calculations of modern aircraft and other vehicles. This branch of hydro-gas dynamics is becoming increasingly important in modern life due to the development of a new generation of commercial and military aircraft and unmanned aerial vehicles. CFD simulations in this area now play a very important role. Panel and other methods for preliminary estimates are still useful but a modern CFD approach allows for more accurate results and better information on vehicle performance, earlier in the design cycle.

FloEFD™ has been successfully used to simulate external aerodynamics in various studies, ranging from aerofoils to whole aircraft modeling, for physically feasible range of Reynolds numbers including subsonic, transonic and supersonic cases. Compared to a traditional CFD approach, FloEFD arrives at valid results with ease. There are also several additional mathematical models employed in FloEFD for detailed analysis. These are a unique model of laminar/turbulent transition, and, also an innovative and effective model of boundary layer. FloEFD offers various methods for visualization of results, allowing the investigation of a complex 3D flow structure and presenting aerodynamic parameters in an understandable form.

Two models of aircraft will be discussed in this article. The first is the NASA Common Research model in the wing/body/horizontal-tail configuration and the second one is the Russian commercial aircraft Tu-214. The main purpose of this investigation was to obtain aerodynamic characteristics of the vehicles such as lift, drag, pitching moment coefficients and pressure coefficient which were compared with experimental data.

The external flow simulation around the wing/body/horizontal-tail configuration of the NASA CRM with focus on aerodynamic coefficients is presented here. The genesis of an open geometry NASA CRM was motivated by a number of interested parties asking NASA to help develop contemporary experimental databases for the purpose of validating specific applications of CFD [1]. A transonic supercritical wing design is developed with aerodynamic characteristics that are well behaved and of high performance for configurations with and without the nacelle/pylon group. The horizontal tail is robustly designed for dive Mach number conditions and is suitably sized for typical stability and control requirements. The fuselage is representative of a wide body commercial transport aircraft; it includes a wing-body fairing, as well as a scrubbing seal for the horizontal tail. The model of the NASA CRM is presented in Figure1.

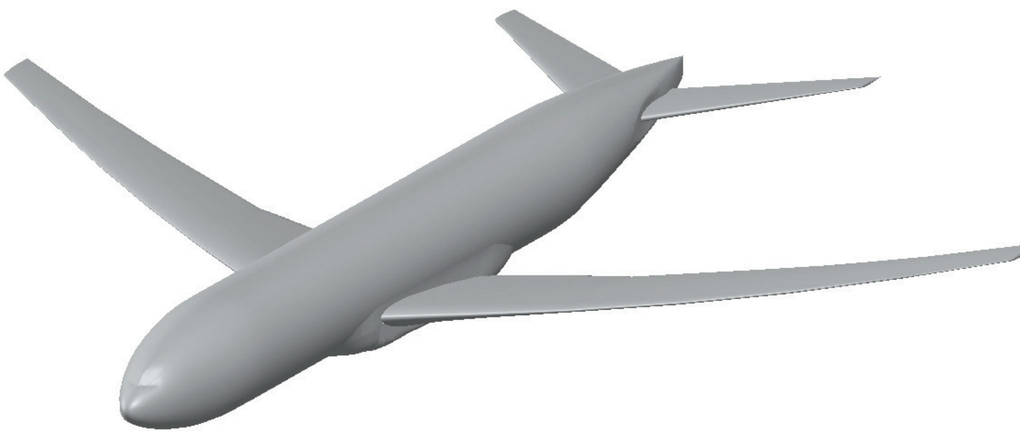


Figure 1. The model of NASA CRM

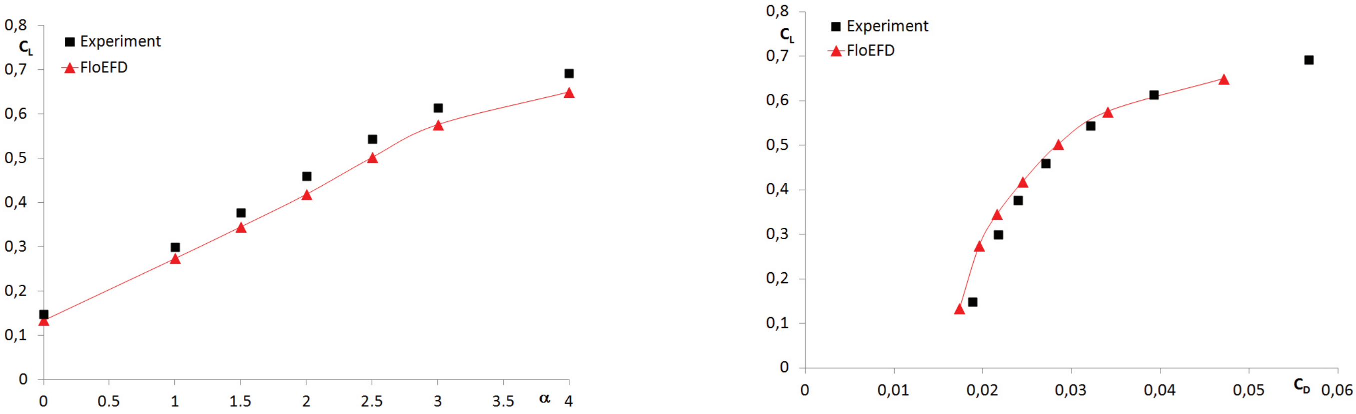


Figure 2. Lift coefficient (left) and polar (right) of NASA CRM

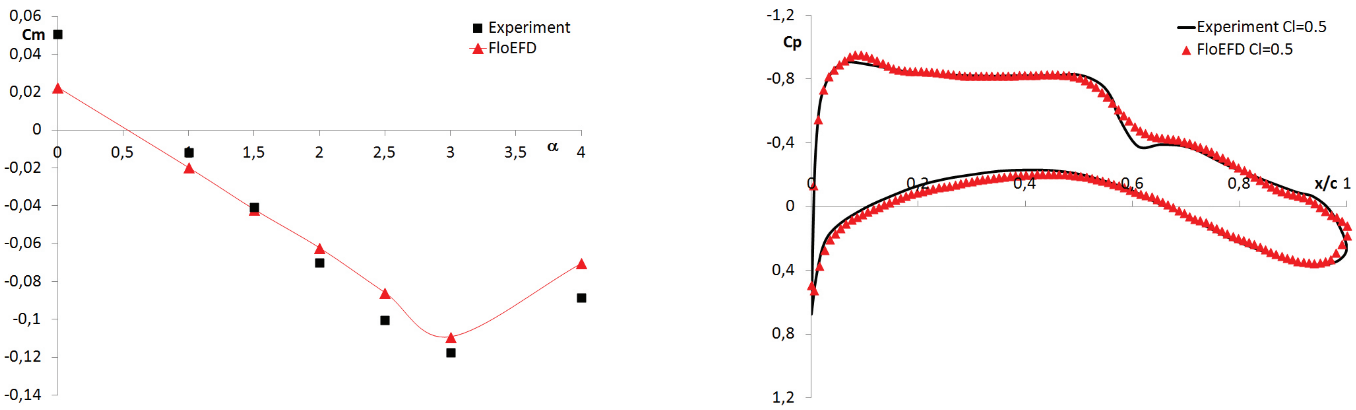


Figure 3. Pitching moment coefficient (left) and chordwise pressure coefficient distribution at 49.9% semispan (right) of NASA CRM

Calculations were provided at the following far field conditions: M = 0.85, P∞ = 201300 Pa and T∞ = 210.9 K. The angle of attack varies in range from 0° to 4°. The best results were obtained on the model with local mesh around the aircraft and several refinements during calculation by Solution Adaptive Refinement (SAR) technology in FloEFD. Attention should be paid to fine mesh resolution in the neighborhood of wing leading edge.

Lift coefficient, polar, pitching moment coefficient and chordwise pressure coefficient distribution at 49.9% semi-span were obtained from calculations and experiments [2] and are presented in Figures 2 and 3. Good FloEFD prediction of the lift and drag coefficients in linear area were achieved. For pitching moment coefficient, discrepancy is bigger. Also there is some insignificant departure in Cp between calculation results and experiment. The pressure distribution with flow trajectories colored by Mach number at M =0.85 and angle of attack 4° is displayed in Figure 4.

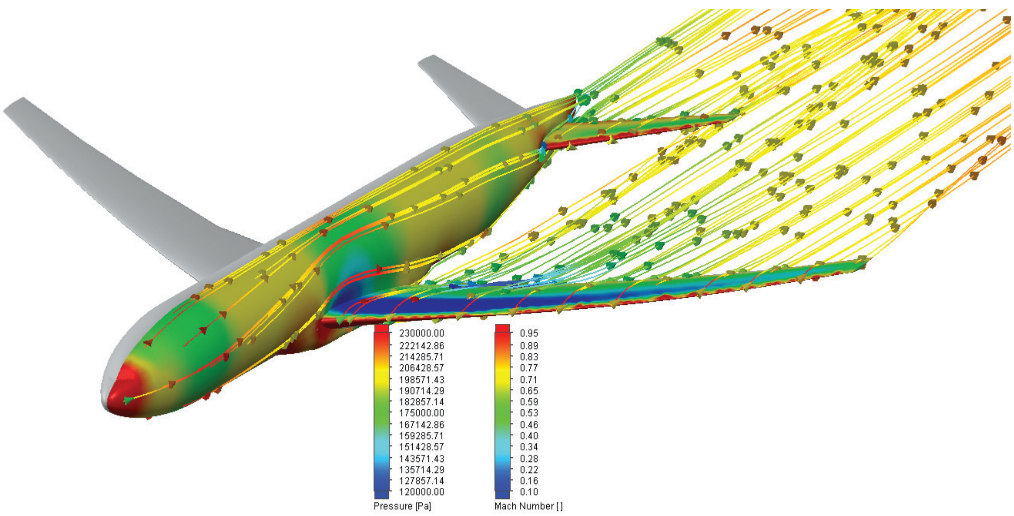


Figure 4. Pressure distribution with flow trajectories colored by Mach number at M =0.85 and angle of attack 4° of NASA CRM

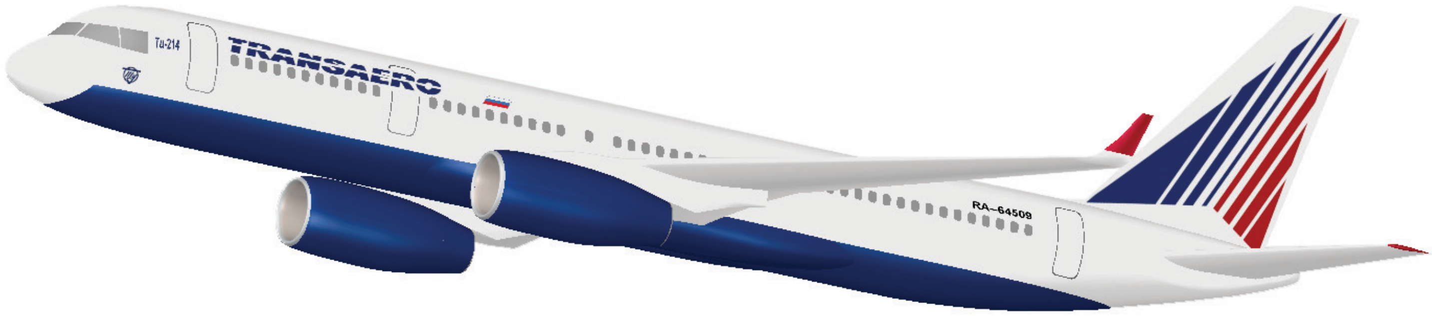


Figure 5. The model of Russian commercial aircraft Tu-214

The second aircraft which is considered here is the Russian commercial aircraft Tupolev-214 (Tu-214) developed by well-known Russian Aircraft Design Bureau PSC 'TUPOLEV' (Figure 5). This vehicle is a cantilever monoplane of a normal scheme with a low-set swept wing and a tail assembly placed on a fuselage with two turbojet engines mounted on pylons under the wing.

The Tu-204/214 was designed as a family of aircraft incorporating passenger, cargo, combi and quick-change variants and relates to a fourth generation of aircraft which have a higher level of reliability and fuel efficiency [3]. For developing this family of aircraft the latest science and technological developments in aerodynamics, strength, propulsion engineering, materials, electronics and ergonomics were applied.

Calculation of this task was provided at the following far field conditions: M = 0.6, P_∞ = 101325 Pa and T_∞ = 288.15 K. The angle of attack varies in range from -3° to 18°. Work of the propulsion system was taken into account in these investigations.
The FloEFD predictions align with the wind tunnel's tests of the Tu-214 aircraft scale model. Comparison was made with respect to integral parameters such as lift coefficient and drag coefficients, etc. All of these parameters were compared with experimental data.

Lift coefficient and polar are presented in Figure 6. It should be pointed out that good FloEFD prediction of these coefficients were achieved at Mach number

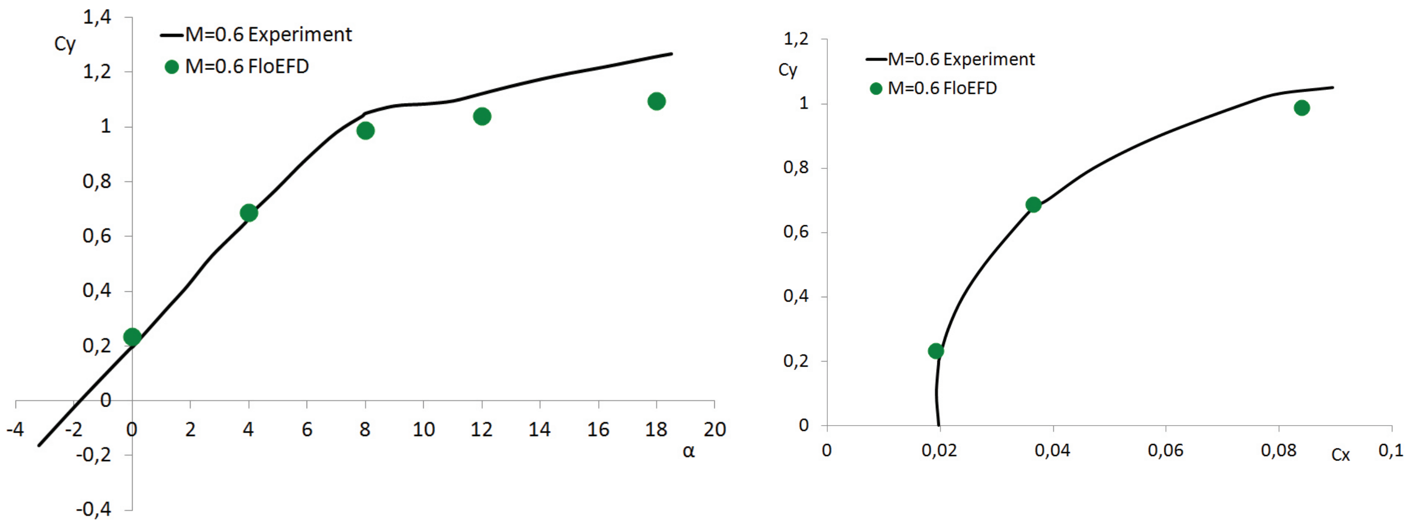


Figure 6. . Lift coefficient (left) and polar (right) of Tu-214 (results were provided by PSC TUPOLEV)

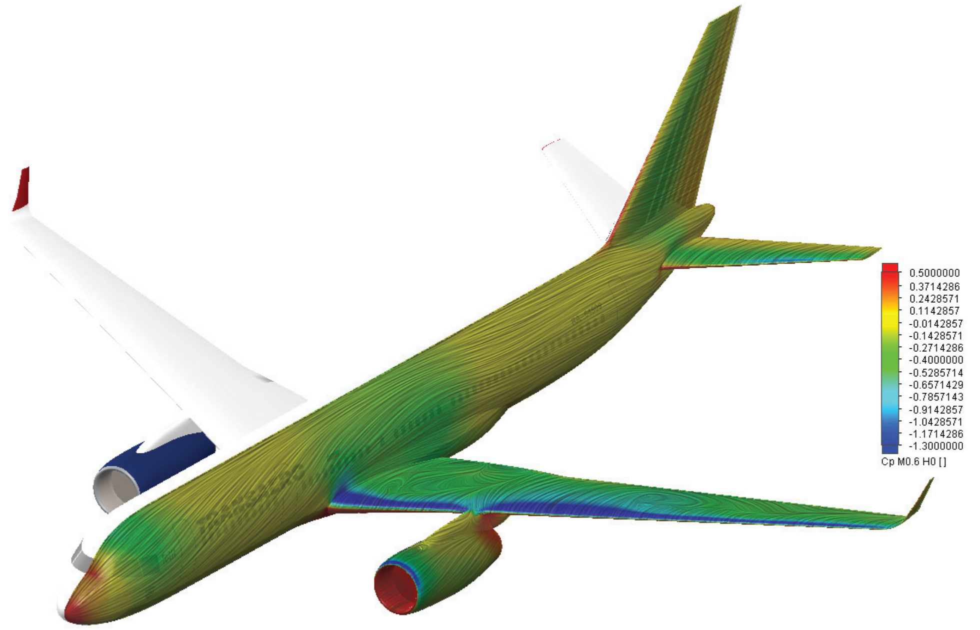


Figure 7. Pressure coefficient distribution with LIC at M =0.6 and angle of attack 10° of Tu-214 (results were provided by PSC TUPOLEV)

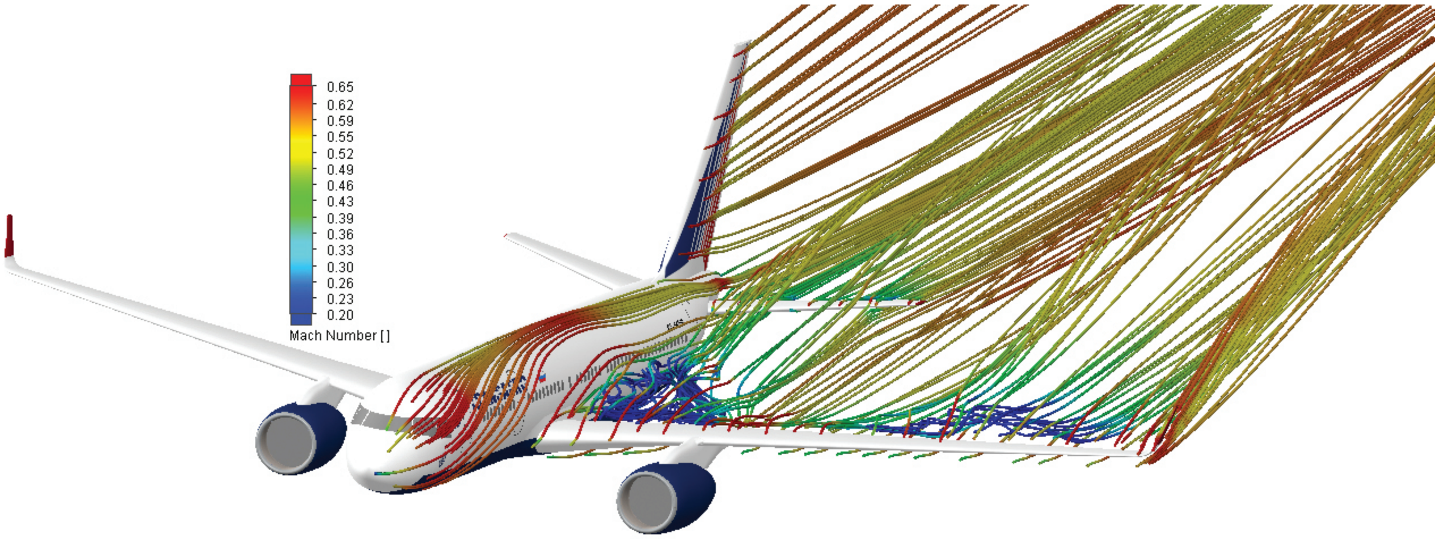


Figure 8. Flow trajectories colored by Mach number at M =0.6 and angle of attack 10° of Tu-214 (results were provided by PSC TUPOLEV)

under study. There is also agreement in aerodynamic derivative obtained in FloEFD and experiments with a discrepancy of approximately 1.7%.
Pressure coefficient distribution with LIC in FloEFD at M =0.6 and angle of attack 10° is presented in Figure 7 as oil flow lines. Using LIC technology allows for good observation of clear flow structures near the aircraft's surfaces.

Comparison of measured and predicted values of the main integral parameters such as lift, drag and pitching moment coefficients show agreement for the investigated class of tasks. Thereby FloEFD yields a series of 'what-if' aerodynamic analyses. It should be pointed out that FloEFD provides export of pressure and temperature as loads for structural analysis, on a structural mesh in NASTRAN format directly, allowing automatic parameter changes rather than a manual approach.

References

1. J.C. Vassberg, M.A. DeHaan, S.M. Rivers, and R.A. Wahls, "Development of a Common Research Model for Applied CFD Validation Studies", AIAA Paper 2008-6919, AIAA Applied Aerodynamics Conference, Honolulu, USA, August, 2008

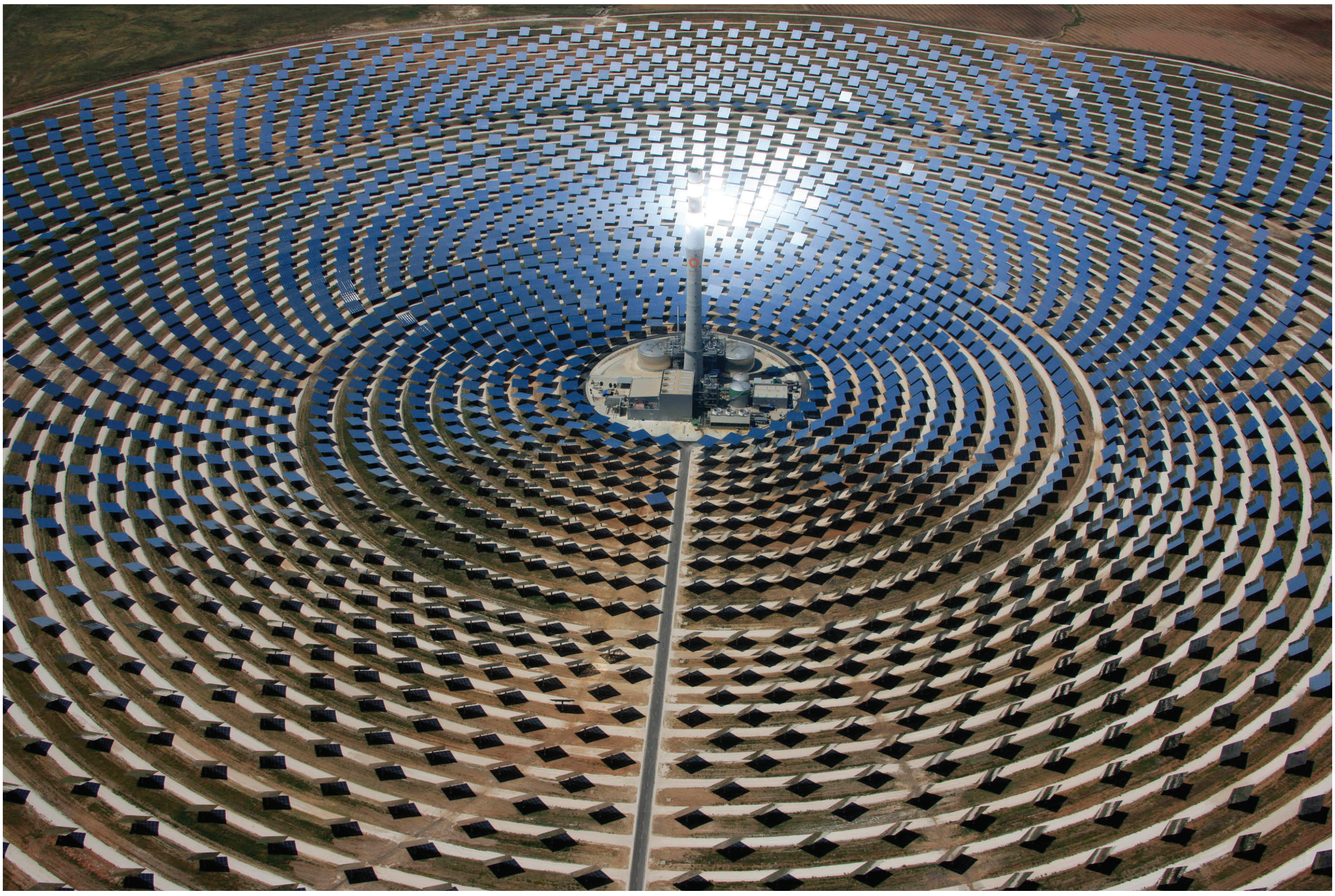
2. 4th AIAA CFD Drag Prediction Workshop: http://aaac.larc.nasa.gov/tsab/cfdlarc/aiaa-dpw.

3. Koshcheev A.B., Platonov A.A., Khabrov A.V. Aircraft aerodynamics of of the family Tu-204/214. Textbook. M.: Poligon-Press, 2009, 304 p. (In Russian)

Harnessing the Power of the Sun

1D Fluid-Dynamic Study of a Molten Salts Thermal Energy Storage System

By Alberto Deponti - EnginSoft
Francesco Castelletta - Eurotecnica



Concentrated Solar Power Plant

Eurotecnica, Italy is an international engineering and contracting company covering a range of sectors from chemical products (notably melamine) to solar power and energy storage. With a core of more than 100 highly skilled employees, it has successfully carried out more than 130 projects all over the world.

A major new area for Eurotecnica is the storage of energy generated from solar power; an obvious characteristic of solar energy is that it is only available during daylight, which leads to a storage requirement more efficient than simply up-scaling standard battery technology. One solution that is often used is to store the thermal energy from the sun is in the form of a mixture of molten nitrates. Held in huge tanks, these molten salts offer a highly efficient manner by which thermal energy can be converted to electrical energy overnight or during periods of lower thermal load (bad weather for example). Heavily insulated, these tanks can hold a capacity of 10,000s of metric tons of salt at temperatures well above their melting point (typically around 131°C) for many days if required. The economic viability of large scale solar projects hinge on the reliability and performance of such systems. Thus, a thorough detailed design process is critical in order to ensure that a reliable system of appropriate scale is ultimately installed.

This paper offers an overview of part of a study, focusing on the investigation of different operating conditions of a molten salts thermal energy storage system. In particular, the emergency closure of a valve is studied at two different conditions, namely the beginning and the end of the cycle. The objective of such an exercise is to find the minimum valve closing time that guarantees the safety of the system, i.e. the minimum time for which the peak pressure is below the maximum allowable pressure for the system. The system is simulated by means of Flowmaster™, which allows many different design iterations to be handled virtually, thereby offering a cost effective and robust means by which such a study can be conducted.

The System

The system to be studied is composed by two tanks of about 15m height and 40m diameter. In each tank there is an immersed pump and a distribution torus. The two tanks are connected by a pipeline with two control valves and six heat exchangers along its length. Each valve is mounted close to a tank (Figure 1). During the day hot molten salt, warmed up indirectly by parabolic troughs via the six heat exchangers, is pumped from one tank to the other one. During the night molten salt is pumped back and releases the heat accumulated during the day through the heat exchangers. For clarity, the tank from which molten salt is pumped will be called Tank 1 and the tank in to which molten salt is pumped will be Tank 2.

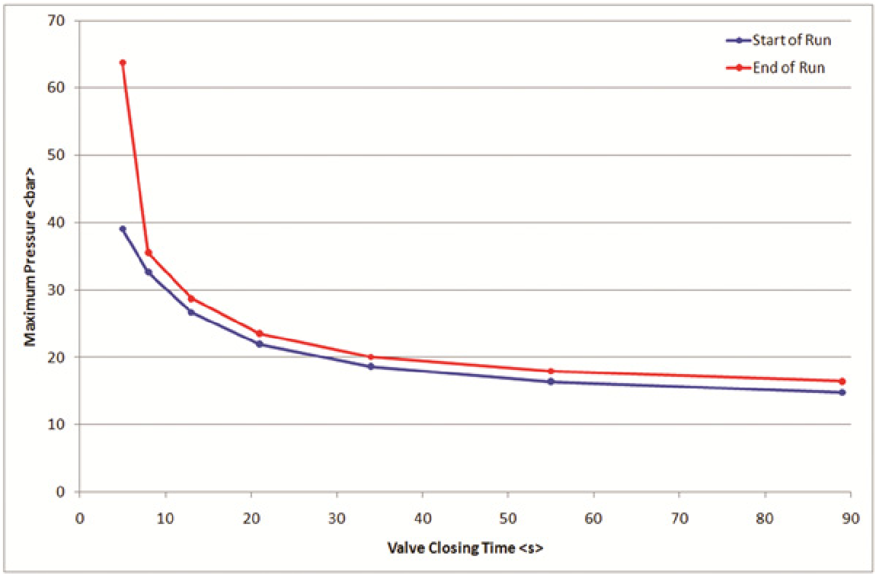


Figure 2. Results of two Parametric Analyses

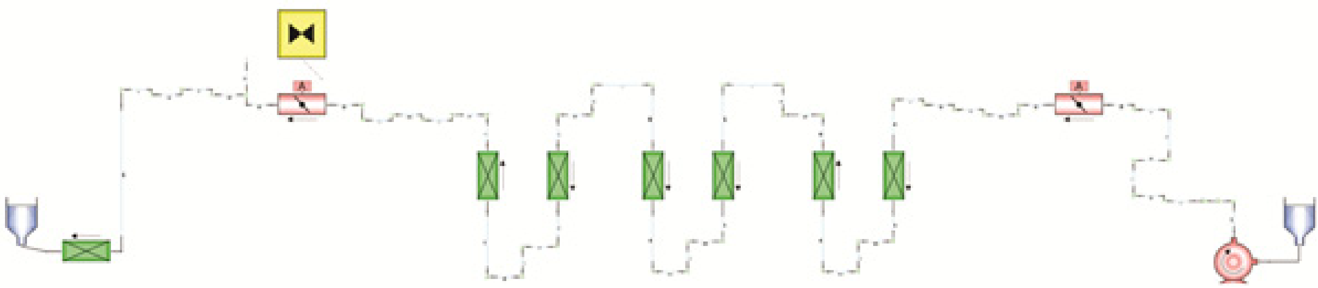


Figure 1. Flowmaster Network Modeling Molten Salts Thermal Energy Storage

Similarly, the valve near Tank 1 will be called Valve A and the valve near Tank 2, Valve B. In the present work the flow of molten salt from Tank 1 to Tank 2 is considered (the reverse flow being symmetrical) and the emergency closure of Valve B in two different operating conditions is studied. The system is studied at the beginning of the cycle when Tank 1 is full and Tank 2 empty and at the end of the cycle when Tank 1 is empty and Tank 2 full. In these simulations molten salt is at a temperature of 286°C and has a density of 1,907 kg/m3. Under these conditions the speed of propagation of a sound wave through the salt is approximately 1850 m/s. The high density and the high wave speed that characterize the molten salt have the potential to produce a severe pressure surge when Valve B closes. For this reason an accurate fluid-dynamic study is essential in order to safely diagnose and design out any serious safety issues. While Flowmaster is perfectly capable of simulating full conjugate heat transfer if required, the focus of this particular study is in the pressure surge phenomena and so heat transfer phenomena that occur in the system will not be considered here.

In Figure 1 the Flowmaster network used for modeling the molten salts thermal energy storage system is presented. Each component of the network is characterized by geometrical and performance data provided by the manufacturer. Since the heat transfer phenomena are being neglected for this investigation, each heat exchanger is modeled by means of a simple discrete pressure loss component (green rectangles in Figure 1). The distribution torus can be handled in a similar manner as its details will not significantly impact the transient response of this part of the network. The closure of Valve B is controlled by a simple tabular controller component (yellow component in Figure 1). Taking into account the elevation changes within the network and the pipe schedules and fittings specified, a maximum allowable pressure of 25.88bar is specified as the design criteria.

The Simulations

In order to evaluate the valve closure time that meets safety standards, two sets of parametric analysis were performed for the start of run and the end of run valve closure conditions. Such studies can be run in batch mode in Flowmaster via the 'Experiments' tab, which allows different combinations of input condition to be run automatically. The results of the two parametric analyses are presented in Figure 2. It can be noted that the maximum absolute pressure decreases significantly as valve closure time increases until about 20 seconds, after that, maximum absolute pressure decreases very slowly. The valve closing time to be used in the case of an emergency maneuver needs to be unique for the entire cycle and needs to guarantee a reasonable safety margin. A valve closure time of 20 seconds guarantees good safety margins for both start and end of run conditions.

In Figure 3 and in Figure 4 the detailed results of the simulations performed with a valve closure time of 20 seconds at the start and at the end of the cycle are presented. In particular the maximum pressure in the system, the pressure at the pump outlet and the mass flow rate at the pump outlet are presented together with the valve closure time. In both cases a strong pressure surge is established, with the largest one occurring in the end of run in the valve closure case. However, the pressure in the system never exceeds the maximum allowable value for the system with a 20 second closure.

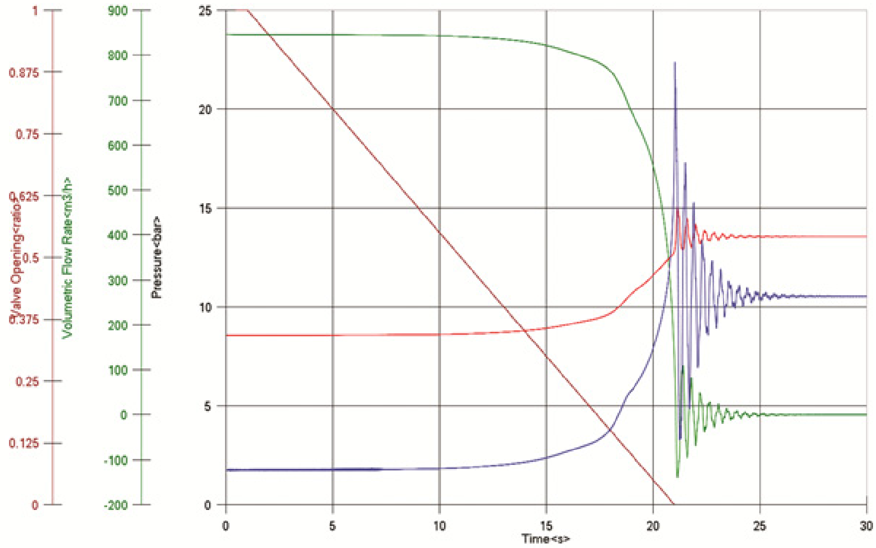


Figure 3. Simulation Results with Valve Closure Time of 20 Seconds at the start of the Cycle

Conclusions

The 1D CFD simulations performed with Flowmaster allowed the study of the detailed behavior of the system early in the design phase considering different operating conditions. Specifically, the present work allowed for the precise definition of emergency maneuvers that guarantee the safety of the system during the entire operating cycle. The precise definition of the valve closure time also allows for the identification of the appropriate motor to be used for maneuvering the control valve. This work demonstrates the importance of numerical simulation early in the design phase of a large plant in which absolute reliability is paramount.

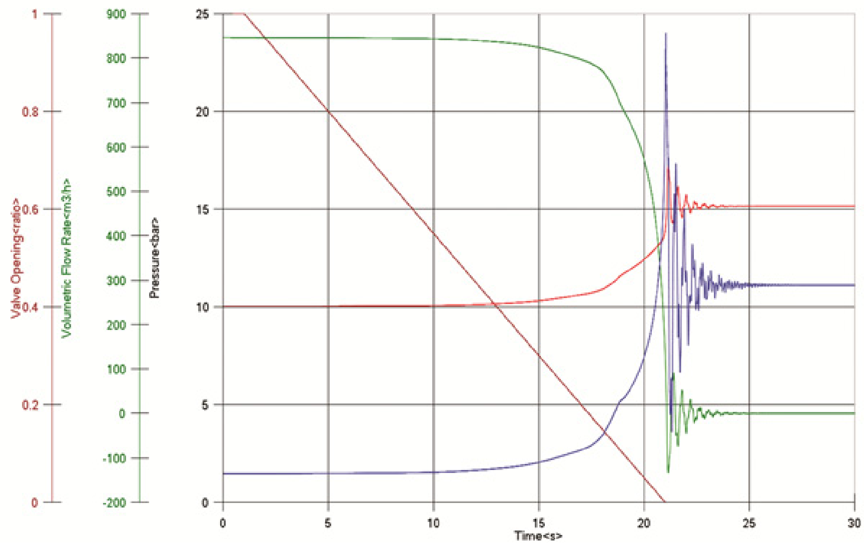


Figure 4. Simulation Results with Valve Closure Time of 20 Seconds at the end of the Cycle

Q: I'm currently working on a model for an electronics box component. How do I determine how many heatsinks I need to control the temperature?

A

Things to Consider When You Need a Heatsink to Cool a Component or Two in Your Electronics Box

Once you've determined that you need a heatsink to cool a component, or several, in your electronics system, there are a couple of things that you should keep in mind. First, knowing that you want to move the heat from your component into the heatsink, pay close attention to that conduction path, and remember to consider your choice of interface material. If you're modeling in FloTHERM®, there's a library of common interface materials from such vendors as Bergquist, Chomerics, Dow Corning, Fujipoly, Thermagon and eGRAF®.

Next, remember to consider the parameters available to you to control the amount of heat that can be transferred from the heatsink into the fluid, most likely air. Remember that:

Q = hA(Tsurface-Tambient)

Where:
Q = heat transferred by convection
h = heat transfer coefficient which increases as the velocity of the fluid (often air) increases
A = surface area of the heatsink
Tsurface = surface temperature of the solid
Tambient = temperature of the fluid (often air) passing by the solid

Many thermal engineers work to make the surface area available for cooling as high as possible. I have seen countless heatsinks with as many pins or fins as could possibly fit on the base. Unfortunately, the resulting pressure drop is often so high that the air velocities passing by all that surface area can end up being very low. As a result, too little heat is transferred into the air. FloTHERM can be a very helpful tool to identify potential designs where that 'hA' product is at a high enough level to remove the necessary amount of heat.

Try to avoid design decisions that allow the air to bypass the heatsink as illustrated in Figure 1.

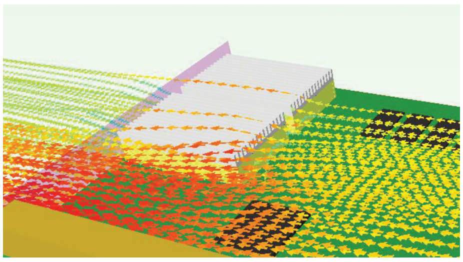


Figure 1

There's a powerful optimization capability in FloTHERM to help identify an optimized heatsink design. Without computational fluid dynamics, it can be difficult to work out that combination of surface area and air velocities that will sufficiently cool those components in the system environment.

Using the Command Center, FloTHERM can automatically build, mesh and solve a series of scenarios that represent a potential design space and then identify an optimum design.

Of course, a thermal engineer might want to explore the design possibilities further. That optimum heatsink that the Command Center came up with might not be available in a vendor's catalog. Then again, there might be a similar one that you can purchase off the shelf that would work well enough. It makes good sense to work out which parameters have the greatest effect on the temperature. The response surface optimization capability in FloTHERM can be used to quickly assess these variations.

In summary,
1. Remember to consider the thermal interface between the heatsink and the component you need to cool.
2. Maximize the velocities of the air passing through the heatsink as well as the surface area of the heatsink itself.

Barbara Hazard
Application Engineer – Mechanical Analysis.

The A to Z of Breaking a World Water Speed Record

It's not every day that a World Record is broken and in speed-sports it's indeed a very rare experience. To do it both safely and cost effectively requires meticulous planning, solid engineering design and quality manufacturing, plus a balance between taking calculated risks versus the accrued speed benefits to be gained.

By Koen Beyers, Voxdale BVBA

Figure 1. Voxdale F2 Speed Boat Design Process - Brainstorm & Conceptual Design (left) Architecture & SDX (center) and Engineering & Optimization (right) - Pro/E, ISDX, AAX, FloEFD

When Voxdale was approached by leading Belgian Speedboat manufacturer, Bernico International, to design an innovative Formula 2 class boat to break the then world record in 2008 and the 100 mph barrier, a new approach was needed. With a combination of PTC CAD/CAE tools, including the 3D CAD-embedded CFD tool FloEFD from Mentor Graphics, we embarked on this challenge.

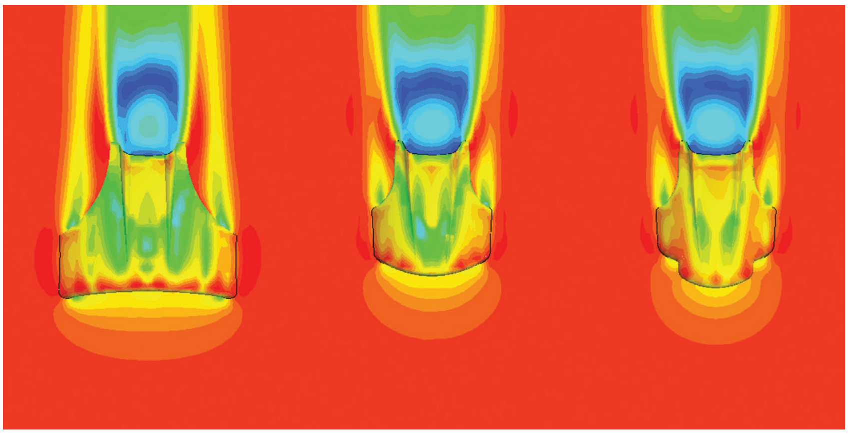
The rationale behind our approach was simple – F2 Racing Boats are very expensive, and typically there is no time or budget for physical prototyping in any design process. The ideal scenario is to 'build and race' and there is not a 'classic' development path for such a process (for example, an alpha version, a beta version, an 0 series and a release version...) For Bernico, the success of the design of a new F2 boat would be commercially essential for subsequent sales of 'cruiser' boat sales. Bernico gave us a simple set of target specifications for the new F2 boat:

1) 300 HP outboard engine;
2) Top speed of 160km/h (~100mph) with an acceleration of 0-100 km/h in six seconds;
3) Cruise at 97% 'above water' for good aerodynamics, and
4) No rear deck and minimal turbulence in the cockpit.

We started the project in early 2009 by taking the then Bernico F2 boat design which could reach speeds of 157.1 km/h (97.6 mph) and 3D scanning the hull and body to reverse engineer the prior design in CAD/CAE as a baseline (see Figure 1). We then did a CFD simulation on the baseline CAD geometry before we could brainstorm several parametric out-of-the-box ideas which we could virtually test in the software and do some design optimization. With the Pro/Mechanica structural analysis tool we could also conduct concurrent structural analysis of our boat designs to yield the optimal solution which we would recommend to Bernico. Within our familiar Pro/E environment we therefore did complex surface geometry modeling of the boat, mechanical integration and engineering, structural optimization, thermal engine management, light-weight material selection, design for cost and assembly, as well as flow analysis in FloEFD to deal with aerodynamics.

CFD simulation helped us to shave off drag components throughout the new boat's shape. We looked at the boat pilot's helmet shape to produce less flow separation and therefore drag. We used a NACA shaped duct to create an overpressure in the cockpit area so there was less turbulence created, and at the rear deck Splitter and Deflector plate we evolved our aerodynamic design (Figure 2) so there was a positive lift on the rear deck and a low flow separation behind the outboard motor. In total we managed to create a drag reduction from our baseline geometry of 240 N, designed

Figure 2. Evolution of Rear Deck Deflector & Splitter Unit



the boat for a positive torque on the rear deck of 450 Nm and when it was built we managed to get to within 3 km/hr of our predicted top speed. Moreover, our design had better acceleration than we needed and crucially more stability and better 'driveability' than what we started with.

This Voxdale-designed Bernico F2 boat broke through the 100 mph barrier at Coniston Water in the UK's Lake District during the annual Power Boat Records Week on its first outing in November 2009 with a stunning World Record of 103.6 mph or 166.7 km/h (Ref 1). Its excellent naval architecture, functional design, optimized aerodynamics, low fuel consumption and lightweight materials proved to be a successful combination. Bernico quickly commercialized the boat design and a Cruiser edition was made available from early 2010 with three orders being taken on the spot during Coniston Records Week! What did we conclude from this exercise? Some boat designs are hard to test but easy to simulate, and CAD/CFD/CAE simulation certainly stimulates radical innovation and leads to workable solutions that yield performance improvement and ultimately in this case a world record breaking result.

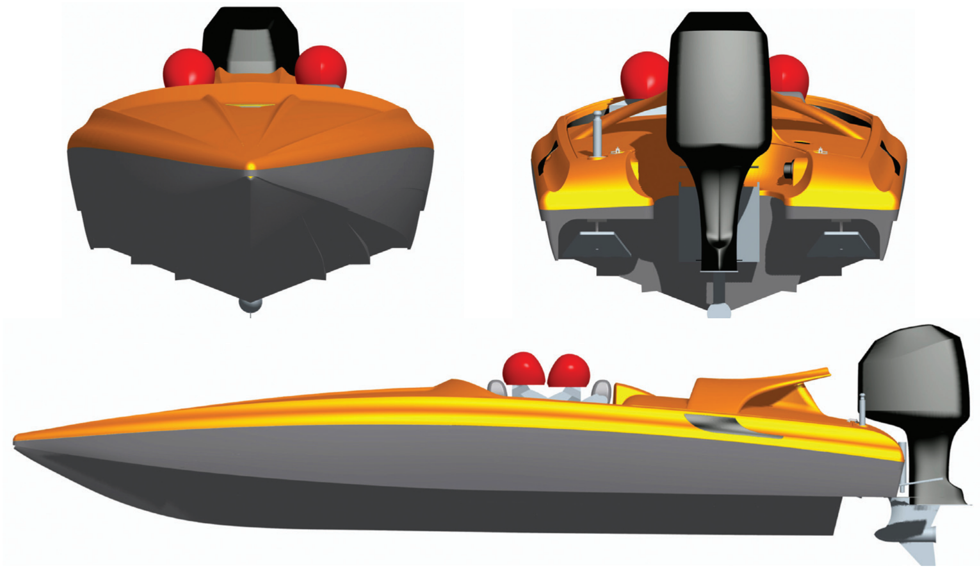


Figure 3. Final Formula 2 Race Boat Design (front, side and rear view)

References & More Information:

1) Coniston 2009 Powerboat Records Week Monohull Record Breaking 103mph run's video: http://www.youtube.com/watch?v=_WBggHZaWCk
2) http://www.voxdale.be
3) http://www.bernico.be
4) https://www.facebook.com/voxdalebelgium

Figure 4. Record Breaking F2 Bernico Boat on Coniston Water in November 2009 with superimposed hull pressure contours from FloEFD simulation prediction

Optimizing Air-to-Air Refueling Systems

A study of a complex aero air-to-air refueling system

By John Isaac, Business Development Manager, Mentor Graphics

Photo courtesy of Wikimedia: F-15C Eagles from the 67th Fighter Squadron at Kadena Air Base, Japan, refueled by a KC-135R Stratotanker from the 909th Air Refueling Squadron during joint bilateral training with other U.S. forces and the Japan Air Self Defense Force Feb. 25, 2010

In the mil/aero industry, systems are getting more complex and the time/cost to design them getting tighter. These systems and components may be mechanical, electrical, or a combination of both. A prime example of a complex aero system is the air-to-air refueling system. It contains not only piping for distributing the fuel, but also complex 3D components such as the fueling nozzle at the plane-to-plane connection.

Three approaches are used in the design and analysis process of these systems. One approach is to do the design, produce a physical prototype, test it, change the design, and then repeat the process, which can be extremely expensive and time-consuming. Another option is to over-engineer, which will result in a safe solution but may be less cost-effective, add weight to an airborne system, and compromise the performance of a system intended to run in a narrow bandwidth.

The third approach, virtual prototyping with computational fluid dynamics (CFD) analysis, early and throughout the design process, can deliver an optimized system at lower cost, and get the system deployed faster.

This approach offers the opportunity for the designer to experiment with multiple design approaches and produce a more optimized design. This is not achievable if each experimental design requires a physical prototype be built and tested.

What Effects Should We Analyze in the System?

Let's assume that we work for an aerospace company that is developing a new refueling system or that we want to analyze some problems with an existing system. In the analysis and subsequent changes to the design, we want to ensure that we have a system that can deliver the following three performance criteria:

1) Will my system be able to deliver fuel at an acceptable rate to the receiving aircraft? Basically, will my system flow rate meet specification?

2) Will my system deliver fuel to the fighter tanks at an even rate? The fighter has tanks in the wings; and if the rates to the tanks are uneven, one wing will become heavier faster and the fighter will become unstable and may break off from the tanker.

3) If or when the fighter disengages from the tanker, either by plan or in an emergency breakaway, will the water hammer effect on the piping cause excessive pressure surges that may damage the system? I know the maximum pressures my system can tolerate, and the analysis can tell me if I am still well within specification.

So we need a CFD solution that will enable us to analyze these effects quickly. First, let's make changes to the design that I think will solve problems in the system. Then, we will quickly re-analyze with the trial changes that gradually focus in on an optimum design to address all of my specifications.

Choosing the Right CFD Analysis Approach

We have two types of CFD analysis tools at our disposal. One can be used to analyze the piping and could be considered a 1D analysis (i.e., the fuel only flows in the axial direction of the pipes). The other can analyze very complex components where the fuel flow is 3D, such as through the fueling nozzle. What CFD tool do I use to analyze this system, which is clearly a combination of 1D piping and 3D complex components?

The 1D CFD tool is much faster than the 3D CFD but lacks the accuracy when simulating the complex nozzle. However, if we analyze the complete system using only the 3D CFD tool, we may get the accuracy we need but the computer execution time will be excessive, defeating the goal of rapid and multiple experimenting with several design approaches. The best approach would be integrating the 1D and 3D tools and leveraging the advantages of both.

Combining 1D and 3D CFD

We will illustrate how such an integrated system works by using Mentor Graphics 1D system simulator, Flowmaster, and 3D simulator, FloEFD, in our analysis. Figure 2 illustrates how this combined 1D-3D solution works for the refueling system.

Initially, the refueling system designer defines a range of operating boundary values (such as pressure and flow rates), that may be presented to the nozzle. They determine this by understanding typical refueling scenarios, and the complete spectrum of possible conditions the system could deliver under normal and extreme conditions.

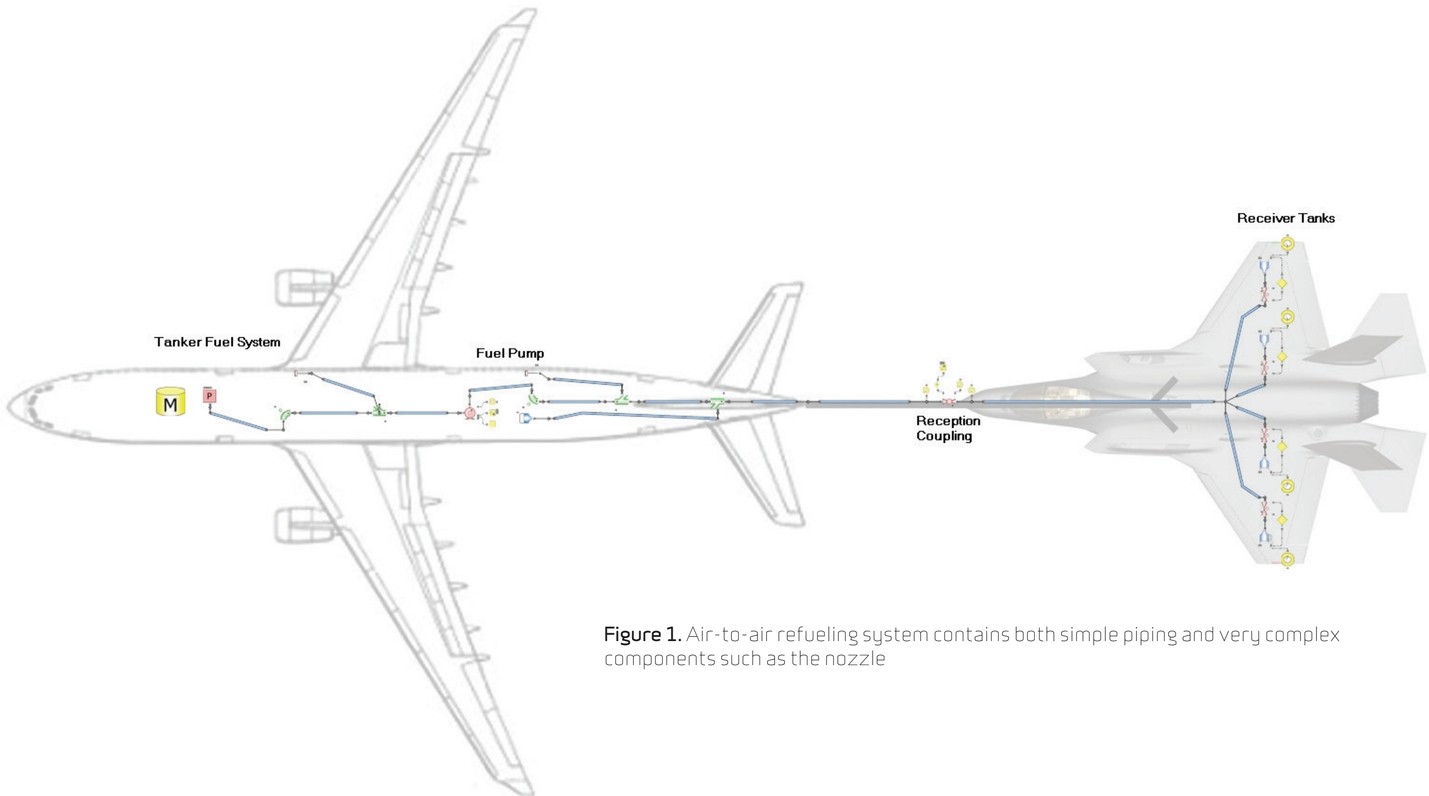


Figure 1. Air-to-air refueling system contains both simple piping and very complex components such as the nozzle

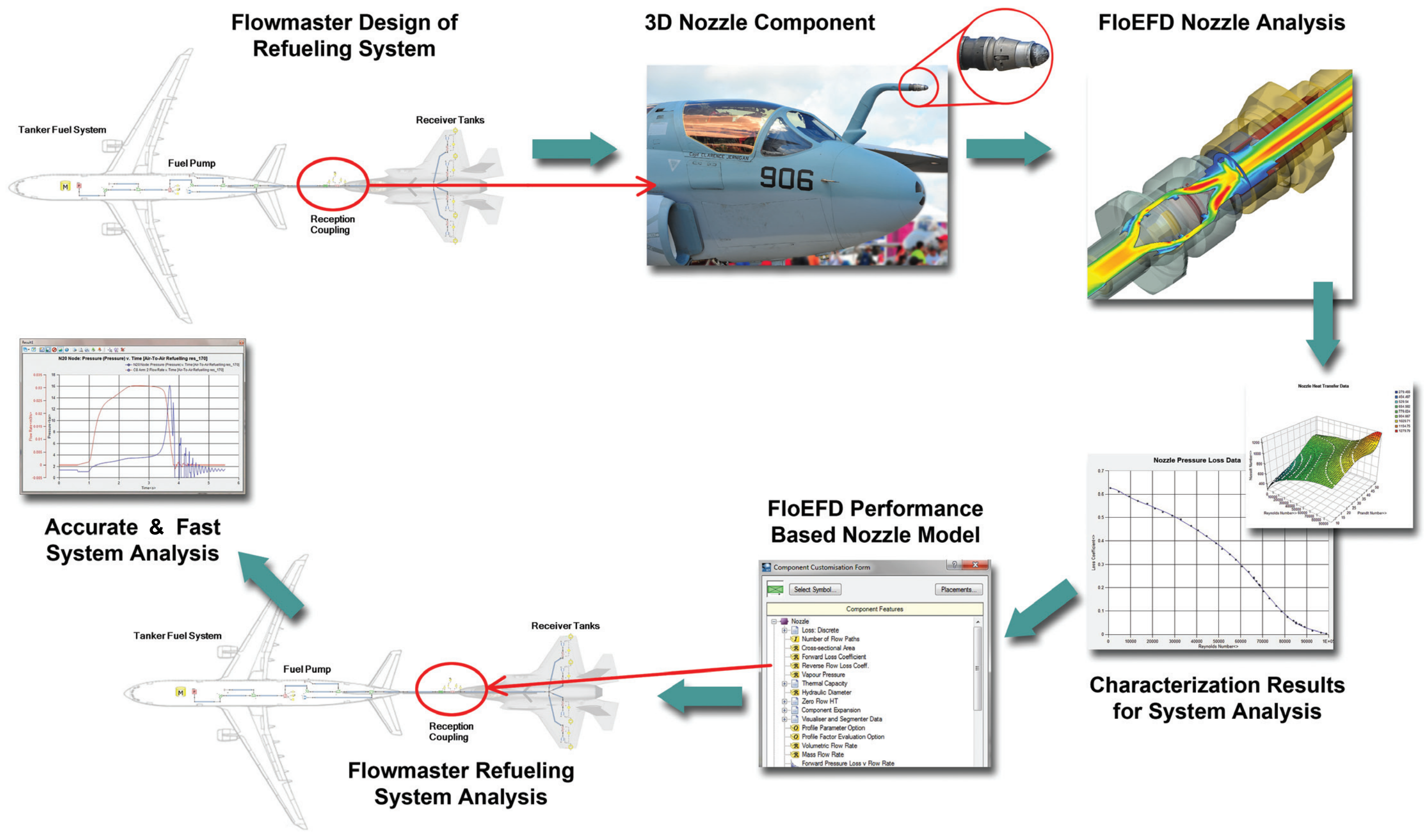


Figure 2. Combining 1D and 3D CFD leverages the advantages of both approaches and provides speed and accuracy in the analysis

The MCAD designer of the nozzle uses the 3D analysis tool embedded in the PTC Creo, CatiaV5, NX, or SolidWorks MCAD system to run detailed fluid flow analysis on the nozzle. Because the 3D analysis is embedded, the designer can perform these analyses directly within the MCAD tool using the same interfaces, an analysis model contrived directly from the MCAD model without external interfaces of data translation, and automatic meshing and convergence.

The nozzle designer sets up a set of analyses based on the nozzle boundary value spectrum presented from the system designer. The designer simply specifies the range of conditions, and the 3D analysis software automatically creates the set of conditions called Design of Experiments. This could result in 30, 40, or even more model batch executions through the 3D analysis, which, for a complex component, might have to run overnight. The resulting data of these runs is automatically condensed into detailed characterization

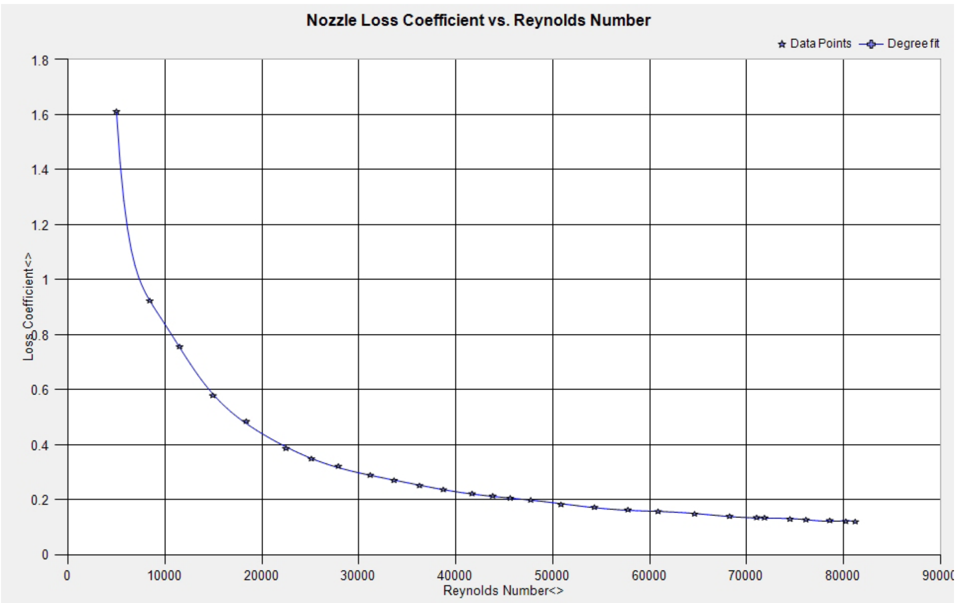


Figure 3. The results of the FloEFD nozzle characterization CFD analyses are captured in a model that spans the spectrum of possible operating conditions

graphs that now represent a complete model of the nozzle.

This model is then simply opened in Flowmaster and saved to the relational database of the 1D system analysis tool. Now, the systems designer can run the flow analysis through the series of refueling scenarios anticipated for the trial design. Design changes can be made to the system and subsequent analysis runs performed. The model of the nozzle remains valid because it covers the full spectrum of possible operating conditions.

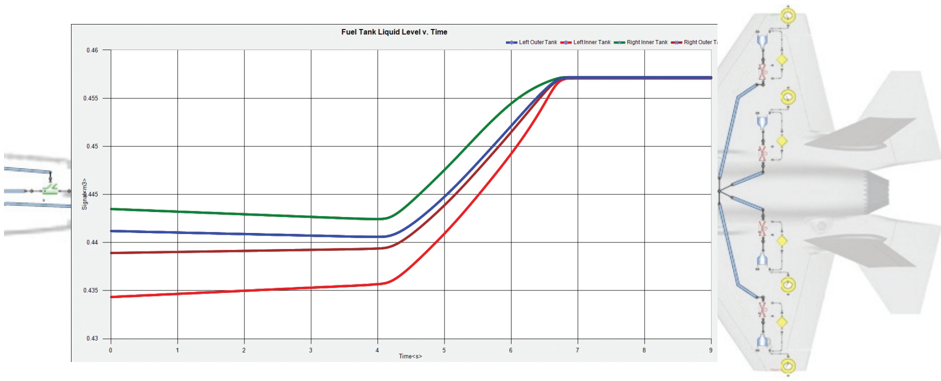


Figure 4. Flowmaster results show how the four fighter tanks will converge to full at a rate within spec that will not cause the fighter to become unstable

The Results

We started out with three criteria for an optimized system: flow rate, flow distribution to the fighter tanks, and the possible water hammer effects of breakaway. The 1D analysis with the 3D-derived nozzle model quickly (in minutes) and accurately creates graphs and numerical data to represent these effects at all nodes throughout the system. The change in fuel levels of the four fighter tanks are shown in Figure 4 and the hammer effects in Figure 5.

1D and 3D CFD Simulation

The accuracy of the 3D simulation of the complex component (nozzle) combined with the speed of the 1D piping system analysis brings the best of both worlds together. With the speed of the analysis, the systems designer is able to try several design scenarios and create a refueling system to run in the small bandwidth of optimum performance. The tanker system could be designed to service different classes of fighter under a spectrum of operating conditions.

This same combination of 1D and 3D integrated analysis methodology can be used for other aerospace systems such as onboard fuel delivery to the engines, engine cooling, interior environmental (air), etc. It also can be applied to industries such as automotive for cooling systems and exhaust, chemical processing, energy, utilities, etc.

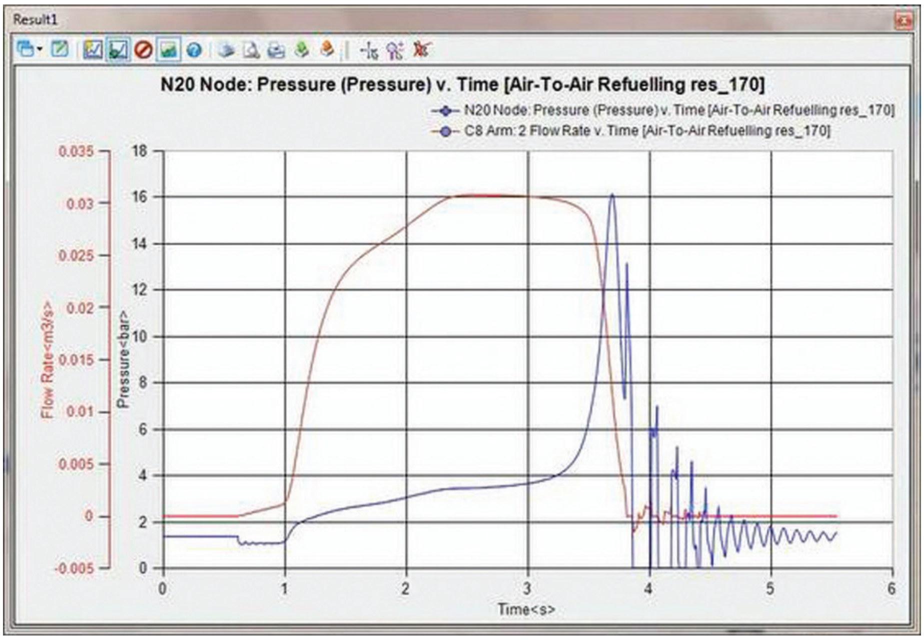


Figure 5. The system designer must analyze the 'water hammer' maximum pressure to determine if it will damage the system

Curtiss-Wright & FloTHERM™: From COTS to Custom Deployment

Curtiss-Wright is a provider of rugged, commercial off-the-shelf (COTS) electronic modules and integrated systems for defense and aerospace applications.

By Andrea Schott, Curtiss-Wright Controls, Defense Solutions

Having both a mechanical design and thermal design background has helped me appreciate the many different aspects of taking a product from concept to production in a short amount of time. Unless you are the sole engineer on the project, doing both thermal and mechanical design, a close working relationship between thermal engineering and mechanical design is critical to ensure on-time, low cost product delivery.

Curtiss-Wright provides rugged, commercial off-the-shelf (COTS) electronic modules and integrated systems for defense and aerospace applications. Our highly engineered solutions, ranging from open standards-based modules to fully optimized systems solutions, are deployed in a wide range of demanding applications, including C4ISR, unmanned systems, mission computing, fire control, turret stabilization, data recording and storage solutions.

As part of the Defense Solutions business unit of Curtiss-Wright Controls, the Littleton facility, Massachusetts addresses a niche market by providing quick turn-around custom electronics enclosures to our customers. Typically, this involves very low volume (quantities between 1 and 15 units), and production delivery in around 20 weeks, which leaves no time for prototyping or testing. We rely completely on engineering experience and thermal simulation to meet our requirements. FloTHERM® allows us to iterate multiple scenarios to optimize our systems for not only thermal performance, but also weight reduction, noise reduction, cost and schedule.

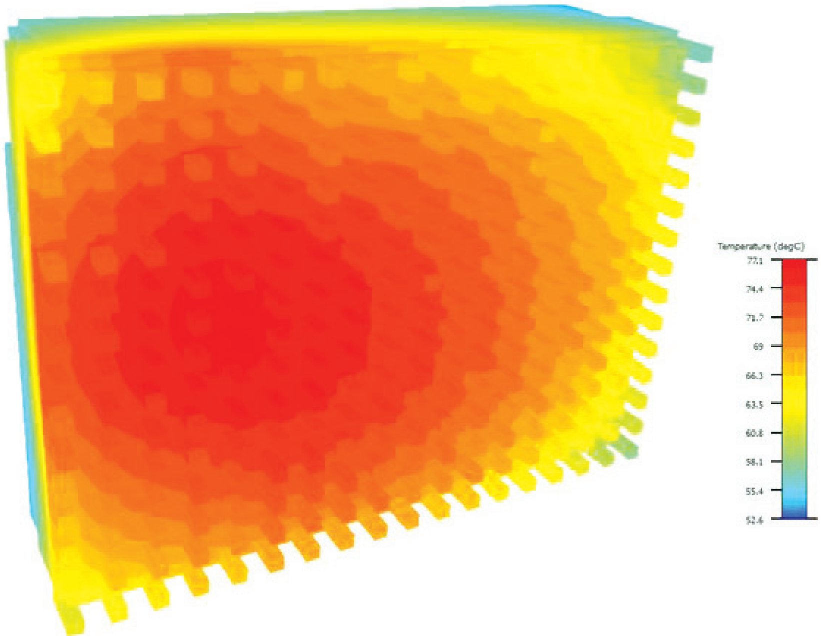


Figure 1. High density, conduction cooled power supply with supplemental air cooling via pin fin heatsink

As a result of FloTHERM's versatility we have the ability to offer our customers a variety of solutions. To further define the operations of Curtiss-Wright's Littleton facility, our activities often involve supplying a metal enclosure (typically brazed aluminum),backplane and power supply all designed to meet specific customer specifications. Our customers populate the enclosure with their own suite (or payload) of electronics. We are usually provided very little information on the design or end-function of the payload. What keeps this process from being straight-forward is the fact the Curtiss-Wright is typically only provided with specifications for the subsystem's overall sizes, power levels, ambient conditions and the temperature requirement for the electronics card mounting. From this limited amount of system detail it is our task to design a solution capable of meeting the required temperature in all environmental conditions at the given power levels. Another challenge that our design team often faces is that in many cases enclosures designed at our facility and are sold to the end customer by a third-party for whom the Curtiss-Wright designed enclosure is essentially a component, not a product level complete system. Because our customers are usually responsible for all verification testing, we rarely receive feedback about results except for the very rare case in which a problem emerges.

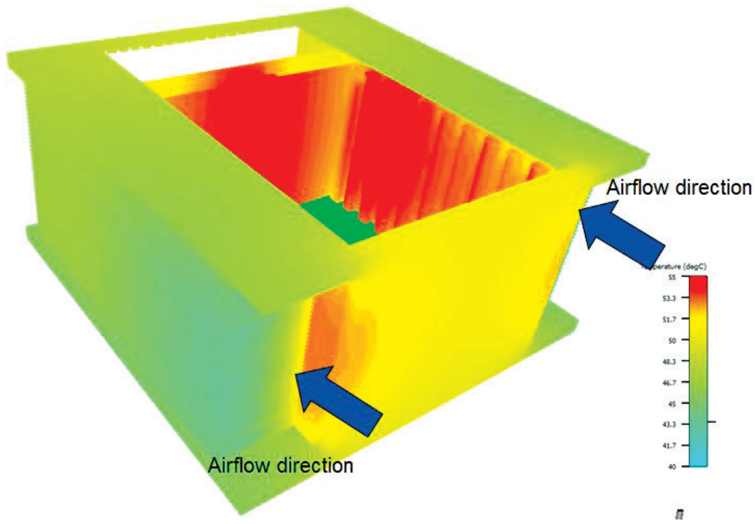


Figure 2. 1300W Custom Electronics Enclosure. The payload is cooled by conduction, Enclosure is air cooled through side walls

As these enclosures are primarily used in military applications, the environmental conditions can vary greatly and are often extreme. New products are often retrofits for older existing equipment and the new higher powered enclosures must be cooled by existing cooling systems. The challenge for our design team, including thermal and mechanical engineers working together, is to meet all of the customer's requirements in a very short timeframe.

The ambient conditions in which the resulting system must perform are harsh, and the power levels required are typically high. Hence thermal simulation is critical for our business, especially since delivery schedules are often very tight.

In order to expedite the design process, Curtiss-Wright has established a consistent method of tracking all thermal design information. This includes tracking the initial customer requirements all the way to documenting the 'as built' configuration. By using a custom spreadsheet template containing the relevant design information we are able to track and maintain information from project to project in a consistent manner. The spreadsheet template contains the initial customer requirements that are used in the quoting process. Since thermal design is one of the highest risk factors in almost every design we undertake, we closely analyze the project's thermal requirements even before we are awarded the job. In many cases we perform some preliminary level of simulation work before the contract has been formally awarded. This initial amount of simulation ensures our customers that we are able to solve their particular design problem. The thermal spreadsheet template also provides our Applications Department a starting point for new designs and saves time in the quoting process.

Geometry used in very early simulation work may turn out to be quite different from what mechanical requirements will later dictate. Frequently, the enclosure space is dictated by mechanical constraints that are not fully defined at the quoting stage. This is because numerous system features such as I/O connections, cabling and air plenum allotment may not yet be determined at this early stage of development. After the results of a preliminary thermal simulation indicate that the customer requirements can be met, the mechanical design process begins. There are often several iterations back and forth between the design engineering and thermal engineering teams to reach a final solution. One design aspect that demands this level of attention is fin optimization. FloTHERM makes it quick and easy to optimize for pitch, thickness, number of fins or base thickness. Each product we design is fully customized so there is very little opportunity for design reuse. For example, the cooling wall (our heatsink) geometry is designed for each particular application to ensure the best design at the lowest cost for each product. While the final solution may be similar to the starting point in the thermal design, it is never exactly the same. Although the time and cost saved by eliminating early prototypes, testing, evaluation and redesign is hard to measure, when delivery schedules are as tight any and all time savings are crucial to our customer's success.

Another key to successful design is a team that works extremely well together. Curtiss-Wright's Littleton facility is staffed with a group of talented mechanical engineers who all work in sync to meet the end goal, which is to deliver a high quality product to our customers every time.

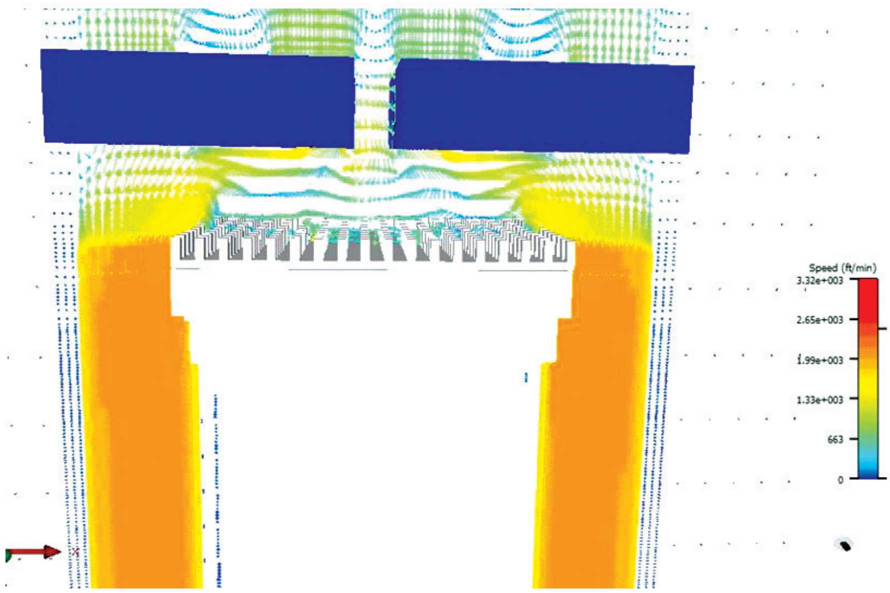


Figure 3. Air flow profile of 1300W enclosure, showing power supply fins and system fans

About Curtiss-Wright Corporation

Curtiss-Wright Controls Defense Solutions (CWCDS) is a long established technology leader in the development of rugged electronic modules and systems for defense applications. CWCDS serves as a technology and integration partner to its customers, providing a full range of advanced, highly engineered solutions from modular open systems approaches to fully custom optimized solutions. Our unmatched capabilities and product breadth span from industry standard based COTS modules to complete electronic subsystems. The company's modules and systems are currently deployed in a wide range of demanding defense & aerospace applications including C4ISR systems, unmanned subsystems, mission computing, fire control, turret stabilization, and recording & storage solutions. Additionally, the company's broad engineering capabilities combine systems, software, electrical, and mechanical design expertise with comprehensive program management and a broad range of life-cycle support services.

For more information visit www. cwcdefense.com.

Making Computers Cool by Design

NMB Minebea use thermal simulation to predict design flaws

By Dr.-Ing. Anton Breier,
NMB Minebea GmbH

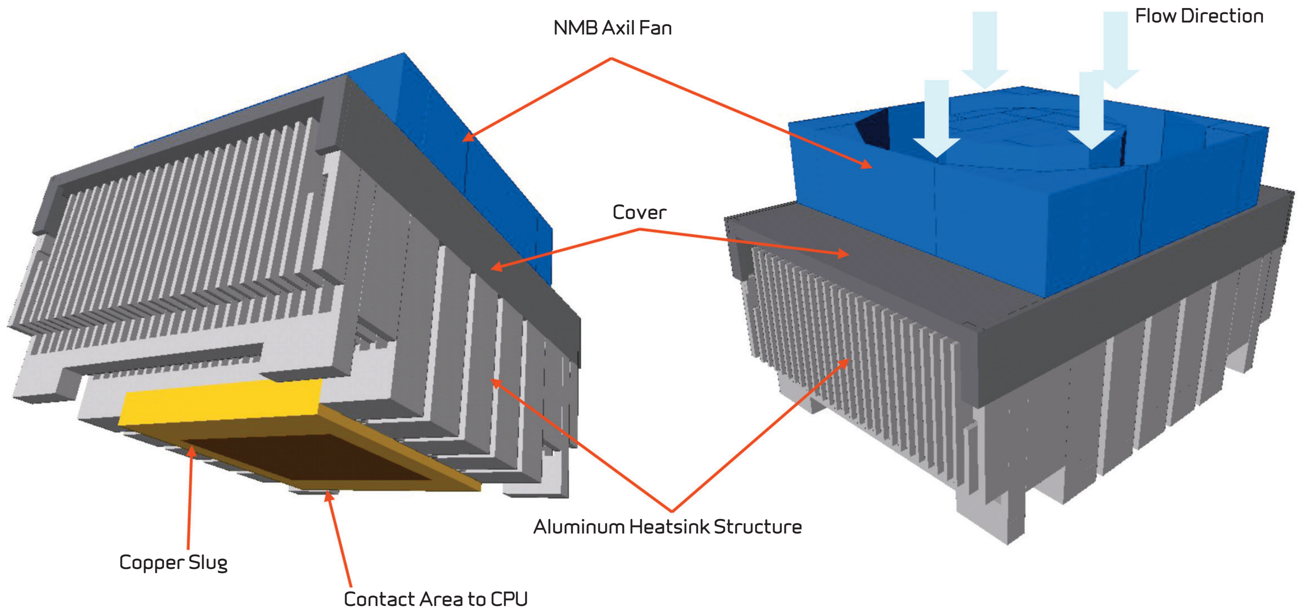


Figure 1. FloTHERM Simulation Model of CPU

CPU cooling is a critical aspect of a functioning computer system, and for this reason the need for forced-air cooling is a significant factor that should be determined at an early stage in system design. Good airflow to heat-generating components, and adequate space and power for the cooling fan are a critical design requirement for any forced convection system. One of the first steps in system design should be to estimate the required airflow. This will depend on the heat generated within the enclosure, and the maximum permitted air temperature rise.

The question becomes, how can we best determine the optimal design for computer cooling, and in particular, for CPU cooling? The answer from fan experts NMB Minebea is to use thermal simulation. Using FloTHERM® from Mentor Graphics, NMB Minebea can anticipate design flaws, and evaluate the thermal behavior of the various components that will ultimately become an important part of the final cooling design. By way of example, here's a quick view of how thermal simulation can assist in predicting and improving the airflow of an average CPU application in order to produce optimal CPU cooling within a desktop computer. The structural design of a typical CPU will look similar to Figure 1.

In this example, the axial fan sits on top of the cover of the heatsink structure. Under the heatsink structure you will find the thermal slug (shown in yellow) and the contact area to the CPU surface. Between the heatsink fin tips and the cover it is necessary to provide a gap in order to allow air to exhaust from the fan.

The FloTHERM results show the distribution of the surface temperature on the heatsink and CPU for this particular heatsink-fan (or 'fansink') combination. As expected the highest temperature is seen directly at the contact area between the CPU and heatsink.

Heat is transported inside the heatsink material by conduction, and from the surfaces to the air by convection. The temperature profile within the heatsink, and the resulting CPU temperature, depend on both the conduction and convection within the assembly. The heatsink has to be designed to deliver the best performance for the chosen fan in order to optimize the entire fansink design. Many factors, including the material, fin design, air velocity and surface treatment all influence the thermal performance of a heatsink.

"Increasing performance gives rise to problems related to equipment cooling. During the development phase, thermal simulations provide us with crucial information about airflow distribution as well as both air and component temperatures. The use of Computational Fluid Dynamics Software not only eliminates the need for thermal redesigns, but also facilitates shorter development times and optimized equipment cooling. We offer our customers complete system solutions, combining fans, heatsinks and power supplies with layout and dimensioning, optimized precisely to the customer's equipment."
Dr. Anton Breier, Deputy General Manager, NMB-Minebea-GmbH

As the cooling air from the fan does not provide a uniform flow pattern, there can be considerable temperature variation within the heatsink. The insight that FloTHERM provides indicates exactly what measures should be taken for optimal cooling. In general, if high temperature gradients are observed within a heatsink then

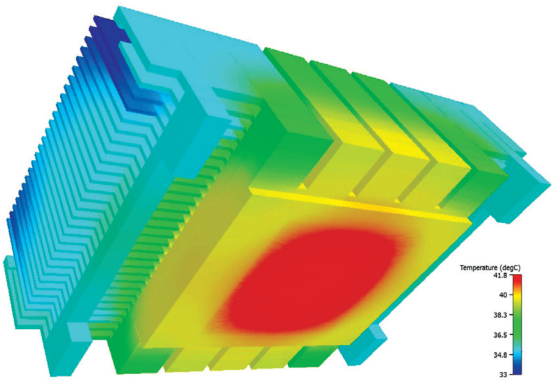
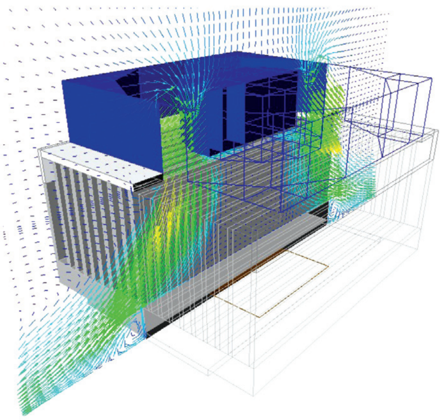


Figure 2. CPU Surface Temperatures

conduction should be improved. This can be done by choosing another material (alloy) with a better thermal conductivity, or by increasing the local cross-sectional area to improve the heat conduction.

Looking at the flow vector field in a center cross-section, above left, FloTHERM reveals a zone below the fan motor with almost no air flow. This behavior is seen for all axial fans used in this design of heatsink. As a consequence of the stagnant air below the fan motor, very little heat is removed from the heatsink fins in this area and so almost no cooling occurs. The fins heat the stagnant air close to the fin temperature as shown on the right hand side of the graphic above.

At the system level, obstructions in the airflow path increase the static pressure drop within the enclosure, reducing the air flow, so airflow obstructions should be minimized. Obstructions in the form of baffles are sometimes necessary to direct the airflow over the components that need cooling. This can lead to unforeseen consequences, such as regions of flow recirculation can occur behind the baffles leading to unexpected hot spots. This is why system-level simulators ,like FloTHERM, are critical for good equipment thermal design. The final system design should



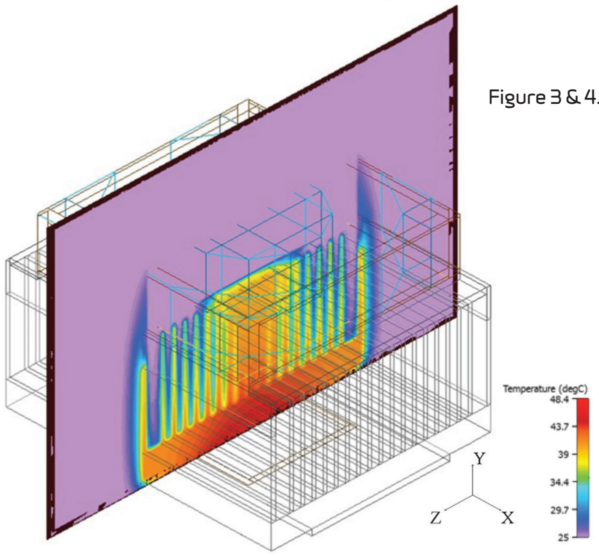


Figure 3 & 4. CPU Cooler Analysis

show continuous airflow through all parts of the enclosure for optimum thermal management of all heat generating components.

To assist customers, NMB offers Thermal Management Consultancy as part of their design services. This has been proven to facilitate the design process of many of NMB's customers by providing key information and solutions for their thermal cooling applications.

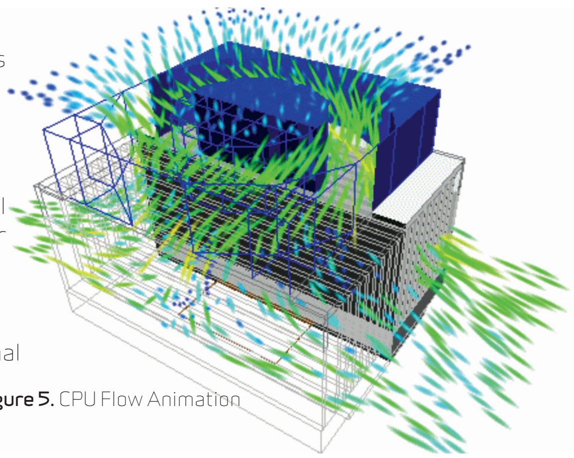


Figure 5. CPU Flow Animation

How To...

Optimize IGBT Design using T3Ster® & FloTHERM® - A salient example

By Mathew Clark, Application Engineer, Mentor Graphics

Several decades ago "Something smells like burning silicon" was the standard exclamation for determining the existence of a thermal design problem. The industry has come a long way since then, using simulation tools, thermal characterization and other techniques to make systems more robust. Sometimes this comes at a cost – making the design more expensive. In response to this, the logical evolution would be to combine all available tools to optimize a system for multiple parameters instead of pointing, looking and checking if something just spontaneously combusted!

Insulated Gate Bipolar Transistors (IGBTs) have been the buzz in industry for a couple of years. Thousands of Watts dissipating through centimeter small surface areas, switching large electrical devices, such as MRI machines or E-Cars sounds like every design engineer's nightmare. But as the devices evolve, so do the engineers, tools and methods.

One of these evolved methods comes from combining T3ster and FloTHERM. Or, in other words, measure, simulate and optimize. Sounds simple, right? It is... to some extent. Using T3Ster® it is possible to obtain the temperature directly from the junction of a semiconductor, giving the advantage of not having to extrapolate data from external thermocouples. When the system is allowed to thermally stabilize, the thermal impact on the entire system can be observed from junction to heatsink in a transient fashion (Figure 1).

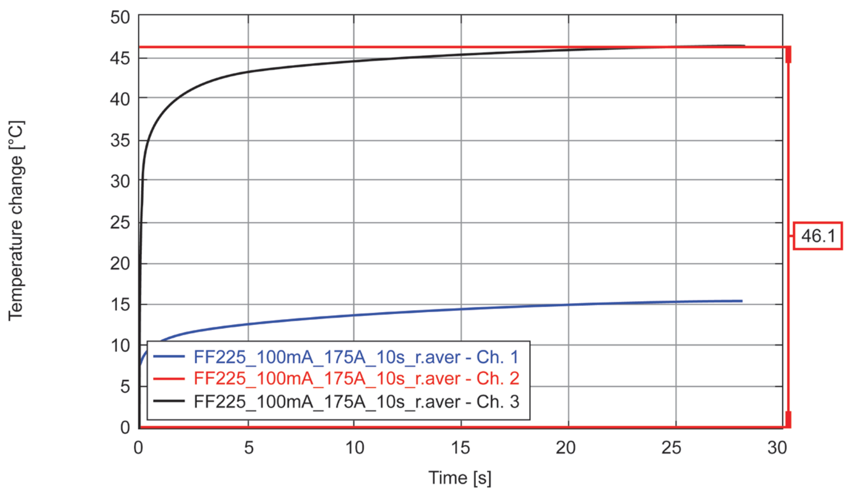


Figure 1. Junction Temperature of IGBT (Add 25°C for ambient conditions)

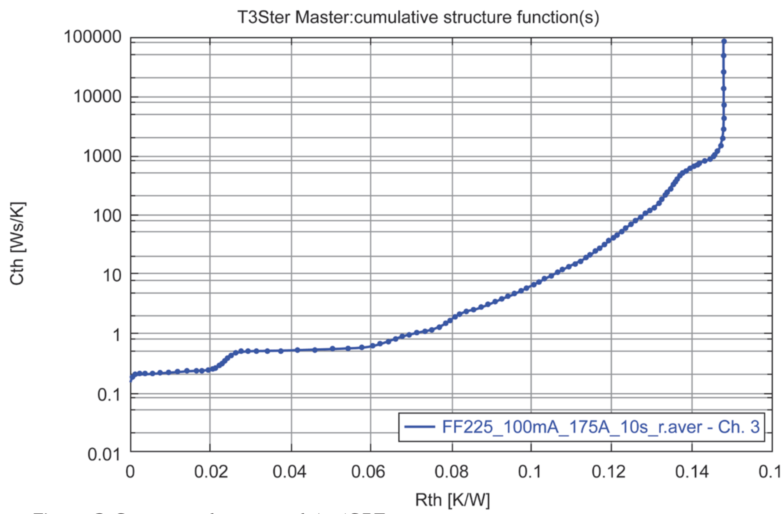


Figure 2. Structure function of the IGBT

Once this has been done, the T3ster Master software can evaluate a so-called 'structure function'. The structure function converts all the obtained thermal information and provides a resistance-capacitance network based on the heat-flow path. Do not worry, this is simpler than it sounds. In essence, every layer through which the heat has to spread from junction-to-heatsink has the capability of absorbing and transferring heat. Smaller layers do this much faster than larger layers, which is why the measurement is transient. When a temperature measurement is combined with time, we obtain our structure functions (Figure 2) – smaller layers (to the left of the graph) have smaller RC-nodes than larger layers (to the right of the graph).

What to do with the information that has been obtained? You could scratch your head, beat the computer in a rage and go home. Instead, it is easier to analyze the structure function for coherence, and then export it to FloTHERM (Figure 3). This can now be done quite simply – click, click, type, click and done.

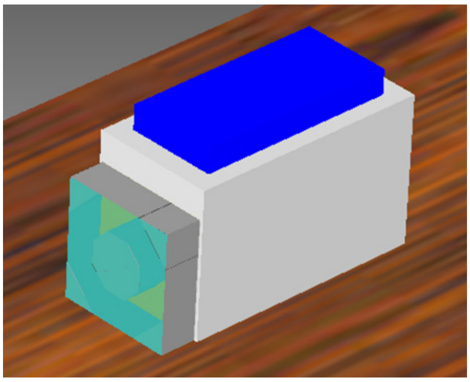


Figure 4. FloTHERM Model of the IGBT

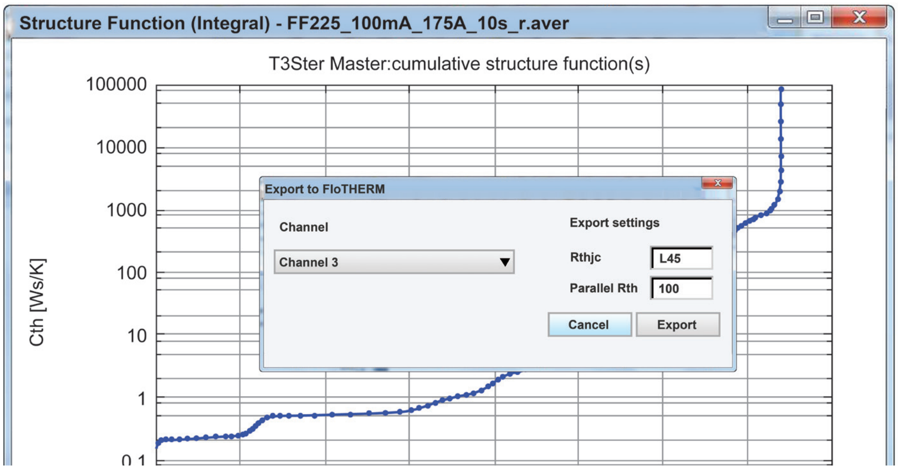


Figure 3. Export Window from T3ster Master to FloTHERM

The RC-Network is now packed into a block-like feature in FloTHERM and contains all the thermal information that was just measured from the small to larger layers. As a sanity check the model is reconstructed similar to the physical example (Figure 4). Now the simulation can be run as a transient, and should yield similar results to the measurement.

Once this has been accomplished the fun part can start. FloTHERM comes with a Command-Center feature. Sound impressive? It is. With this tool, parameters can be selected to create an optimized system. In practice, the example looks as follows: The key parameters are chosen based on what the designer sees as important. Perhaps only the thermal performance needs to be optimized, or the cost and performance need to be combined, depending on budget. In this example, thermal optimization was of the highest concern.

Running the Command-Center optimization, and iterating the number of fins and fin-size, it was possible to lower the junction temperature by 20°C, by increasing the number of fins, and making them smaller (Figure 5).

	Initial Heatsink	After Optimization
Fin Width (mm)	2.2	1.6
Number of Fins	9	17
Junction Temp. (°C)	137.6	118

Naturally, when several parameters need to be optimized, the system chooses the optimal combination by using the response surface (Figure 6).

This unique combination of software and hardware opens up a new world for electronic thermal design. Once the system has been characterized and verified, it can be continuously optimized, changed and tested within the software before building a new prototype of the final model. Not only does this save development time, but with advanced features like the Command-Center, manufacturing costs can also be cut. This creates a ripple effect throughout the supply chain ending with the consumer essentially allowing for robust systems, that are designed fit for purpose.

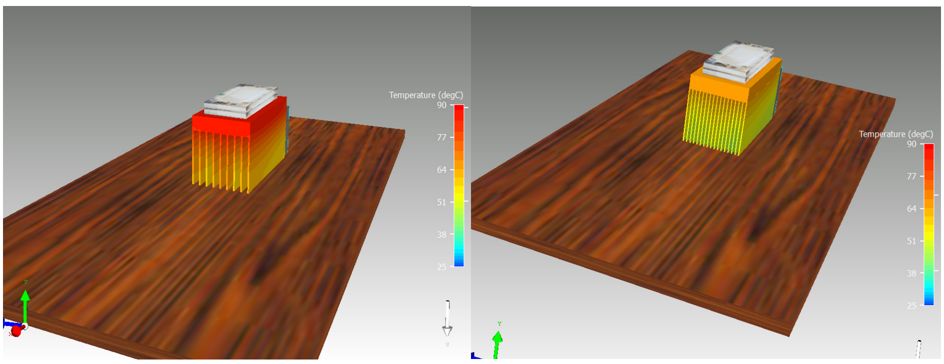


Figure 5. Comparison between the original model and the optimized result

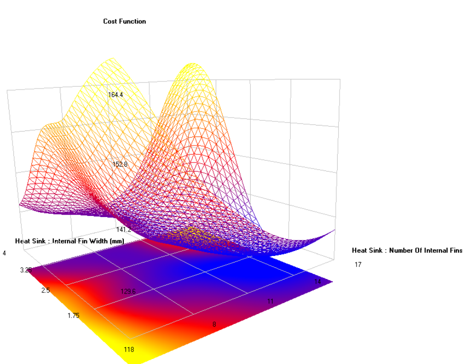


Figure 6. Surface Response Plot for the optimized parameters

Marrying Test & Simulation

The Whole is Greater than the Sum of the Parts for Fuel Systems

Modern Design Engineers have at their disposal a variety of ways to help them in their work. The art lies in ensuring each method complements the others to ensure deadlines are met and costly re-working is avoided. Fábrica Argentina de Aviones have integrated Flowmaster fluid system simulation into their design process in a manner which maximises the return from their test programs and provides additional insight from the earliest possible stage in their design process.

It is widely acknowledged that there is a direct correlation between the cost to make a change in a design and the time since work on it commenced. As the process develops and more man-hours and resources are committed, the relationship becomes decidedly non-linear and even small changes can have large repercussions. Integrating simulation into the design process is a proven and established way of minimizing the risk of having to make a late change to any element of the design.

This principle works best when used in conjunction with 'traditional' experimental methods utilizing test results to validate simulation models, and applying simulation models to help focus experimental programs. The result is a streamlined design process that allows the strengths of all available methods to be brought to bear on a given problem in a cost effective manner.

Guillermo Robíglio, an engineer at Fábrica Argentina de Aviones (FAdeA) explains further, "When designing a new system, or modifying an existing one, we begin by creating 'virtual prototypes' in Flowmaster®. We can enhance the component models with our own test data and then begin the design process on this digital mock-up. This allows us to save a great deal of time and significantly reduce the risk of unexpected problems occurring later on when the design is at an advanced stage."

This principle was demonstrated during the re-engineering of the aircraft and the design stage of the adaptation of the fuel system of the IA-58 Pucará, a twin-engine turbo-propeller aircraft that has been in continuous service with the Argentinian Air Force since 1975.

The ultimate goal was to produce a Flowmaster model that could provide information not only for steady state operation, but also on the transient response of the network under differing scenarios. For example, the design team may wish to know the time taken to fill or empty the inverted flight accumulators at different pneumatic pressures, the behavior of the system at different altitudes and attitudes or its performance following the failure of one or more components.

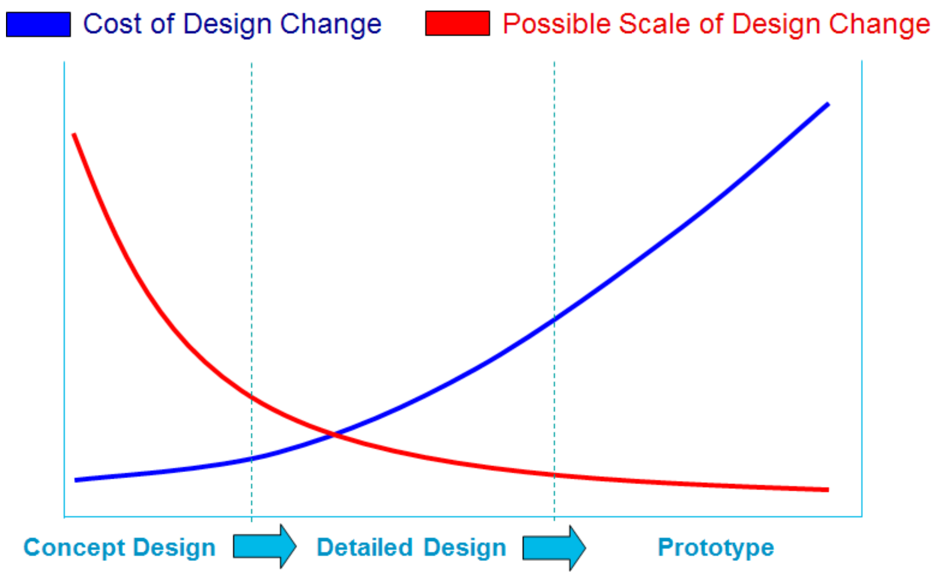
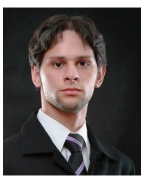


Figure 1. Increasing Cost of Change in Design Process



Guilherme Tondello
Creative Solutions

FAdeA
Fábrica Argentina de Aviones "Brig. San Martín" S.A.

Figure 2. IA 58 Pucará

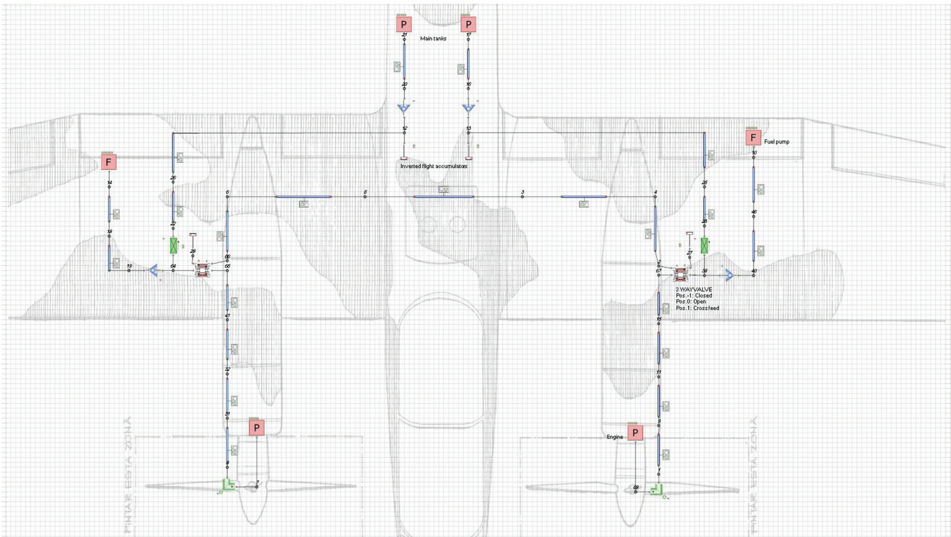


Figure 3. Pressure drop calculation in the fuel system of IA-58 Pucará

In working toward this goal, FAdeA followed an extremely rigorous procedure which took maximum advantage of all the tools at their disposal. Where test results from individual components were available, or made available, they could be used to add an extra degree of reliability to the virtual prototype. In addition, having access to test facilities could be used to supply an extra degree of confidence in the model as a whole. "FAdeA has an extremely capable and high precision test facility at its disposal. This allowed us to validate almost all the simulations we made in Flowmaster. In every case the results were satisfactory and for that reason Flowmaster became an extremely valuable and trusted partner in the design process." explains Martin Blank, Manager of the Systems Team.

Knowing that he had a virtual prototype of such proven accuracy at his disposal allowed Guillermo to confidently deploy Flowmaster across a range of scenarios in order to generate a comprehensive picture of the system behavior. Not only was this achievable in a more cost effective and timely manner than would be possible using physical testing alone (like an 'iron bird' test), it also provided the design team an insight into areas of the system inaccessible to instrumentation. "We were able to identify a localized zone of high pressure in the Flowmaster model of the hydraulic system which was 'invisible' to our system indicator or the relief valves. However, with a virtual prototype at our disposal, discovering the location of this problem was as simple as noting a nodal pressure in the Flowmaster model." explains Guillermo Robiglio.

The integration of Flowmaster into the design loop has allowed FAdeA to increase the efficiency of their extensive test facilities. In turn, the test facilities have allowed the Systems Team to validate their virtual prototypes and so have confidence in using them to predict and understand system behavior across the design envelope. This method of working has proven itself so valuable that it is now the standard for the team, "Actually, every time we need to evaluate or design a fluid system we run lots of Flowmaster simulations," says Martin Blank, "this allows us to understand the behavior of that system very well."
By taking full advantage of all the facilities and tools available to them, the Systems Team at FAdeA have been able to generate validated virtual system models that afford them the ability to test and interrogate their fluid systems to a degree and within timescales that wouldn't previously have been possible, This is also substantially cheaper to set up and run and virtual prototypes have the additional advantage that simulations can be run in seconds or minutes and without supervision.

Hence, batches of simulations can be run over lunch breaks or overnight. Furthermore, the advantages of working in this manner will only increase and become more apparent with time. As the library of test data and simulation results grows, the Systems Team will be in an even better place to predict the performance of new or redesigned systems.

The Latest Philips TVs make LEDs Dance

'Dancing' above refers to dimming and boosting of LEDs in time with the picture

By Wendy Luiten, Philips

In recent years there has been an explosion in LED-LCD displays, and a proliferation of LED TVs in people's homes. They offer a great picture, great styling with a thin form factor, increased functionality like 3D and internet access, and great value for money, but cooling the LEDs, while steadily increasing power densities in ever thinner product enclosures poses a big challenge for designers

In an LED, die (junction) temperature affects both performance and lifetime as LED lumen efficacy is lowered and the color temperature shifted. Heat degrades the epoxy lens, and both the absolute temperature and the temperature distribution over the LCD screen can lead to performance and lifetime issues. Non-uniform LED temperatures lead to unwanted spatial non-uniform lighting in terms of color uniformity and brightness. Effective heat spreading is therefore a key goal of the thermal design.

The temperature of the LCD has to be just right. Too hot (typically 60 - 70°C) and optical materials age , too cold and the switching is sluggish, causing fast changing pictures to display blurred - very performance degrading in modern high resolution 3D TVs! At a system level, the temperature the user experiences when touching the TV is a key safety issue.

While it is entirely feasible to do electrical and optical tests in a standalone setting

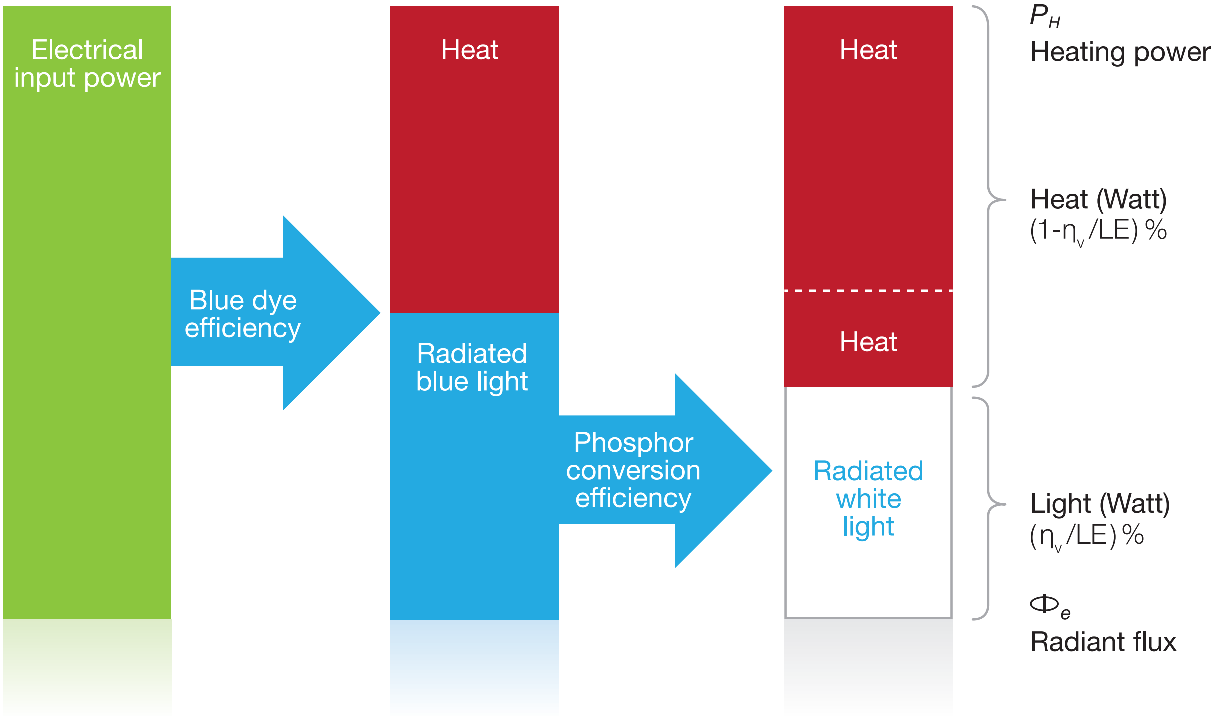


Figure 1. Power Balance of a White LED

and directly translate the results to the system situation, for thermal this is not the case. The presence of the set back cover, which causes an additional thermal resistance from the display back to the room temperature, and second, the direct thermal interaction between the display and boards illustrates the difficulties encountered in co-design.

Philips have been pioneers in the TV industry since the 1940s, so the chances are the first TV you ever saw was a Philips. In fact, it was probably the first TV your grandparents ever saw. Philips has used FloTHERM extensively in the design of TVs since the software was first released in the early 1990s.

Today TPVision, the maker of today's Philips TVs, uses FloTHERM to look at all aspects of the thermal design of LED LCD TVs from the LED package, module, up to the full TV in different environments, starting from conceptual design through to the final product, to optimize all aspects of the thermal design. Factors considered include evaluating different design architectures, such as direct lit display vs. edge lit displays, and optimizing the cost benefit of different cooling solutions.

While LEDs are used to light the display, there is a world of difference between using LEDs for display backlighting and using LEDs in a lighting product. In lighting applications LEDs are typically used in a steady state manner, with timescales in the order of hours or longer. In a TV two different timescales prevail. Heating and steady state behavior of the set as a whole is governed by the average power consumption, and this has a long timescale similar to domestic lighting applications. However, the moment-to-moment changing of the video content happens at a much higher frequency, and this creates an additional much shorter timescale.

A further complicating factor is the thin film transistor (TFT) panel that covers the LCD-LED display. Light emitted from the TV is a combination of the light emitted by

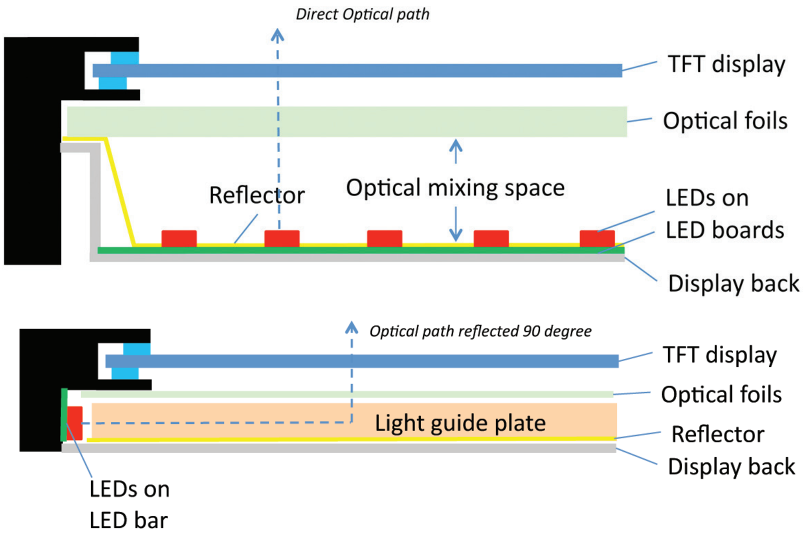


Figure 2. Principle of Direct Lit (top) and Edge-lit (bottom) LCD-LED Display

the backlight and the state of the pixels in the TFT panel. If a TFT pixel is open, light goes through to reach the viewer. If a TFT pixel is closed, light is blocked. In display with a static light emitting backlight, pixels are predominantly open in a bright image, and pixels are predominantly in the 'closed' state if the image is dark. However, from a picture quality and energy consumption point of view, a static light output from a backlight is not optimal.

To improve the picture quality (deeper black) and reduce energy consumption it is common to dim the LEDs in dark scenes. A refinement is to use 2D (or local) dimming, where the LEDs are dimmed not only in time, but also depending on location, providing a further improvement both in picture quality (higher contrast over the screen area) and in energy consumption. However, it is possible to go further still.

The timescale over which the area of a TV lit by a single LED changes is one to two orders of magnitude smaller than typical thermal time constants for display LED packages. As well as dimming LEDs during dark scenes it is also possible to boost their light output for short periods of time. LED dimming and boosting scenarios, while not primarily intended as thermal control measures, are very beneficial to the thermal management of LED-LCD TV sets as the associated LED temperature is more highly correlated to the average LED power, which is much lower than the peak. The result is exceptional picture quality.

Philips TV takes a similar approach for the ambilight feature: the LED temperature is determined by the average LED power dissipation, and large instantaneous LED peak powers can be allowed to increase the immersive experience. Tight thermal management algorithms are deployed to prevent LED boosting from adversely affecting lifetime and reliability of the display.

FloTHERM® simulations were performed on a stand-alone direct lit display, cooled by a heat transfer coefficient typical of natural convection including radiation on the front and on the back. Figure 4 illustrates an important trade-off between the number of LEDs and thermal issues.

The calculated temperature field compares well to the measured temperatures on the front of a direct lit TV set, in Figure 6
In the direct lit TV, the effect of hot air rising is visible, as higher temperatures at the top of the display. Also, the positions of the three boards are visible as locally higher screen temperatures. The infrared picture

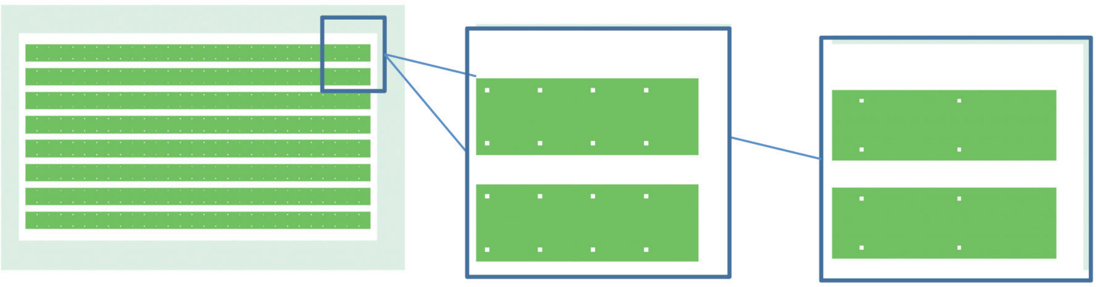


Figure 3. LED Board Geometry

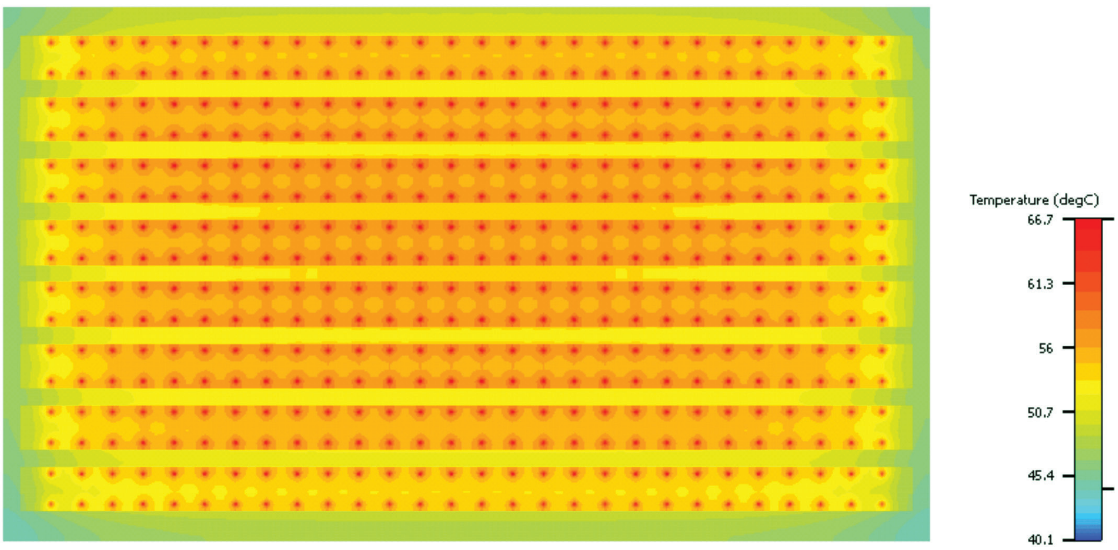


Figure 4. Calculated Temperature Distribution of the LED Boards

"FloTHERM has long been an important tool to our LED-LCD TV thermal design, and is routinely used at module (display) level and at system (TV) level as the temperature distribution in the LED-LCD display is a system-level issue due to the strong thermal interactions. FloTHERM helps us select and optimize the thermal solution so we have confidence from a very early design stage."
G.A. Luiten, Philips Research, Eindhoven, Netherlands

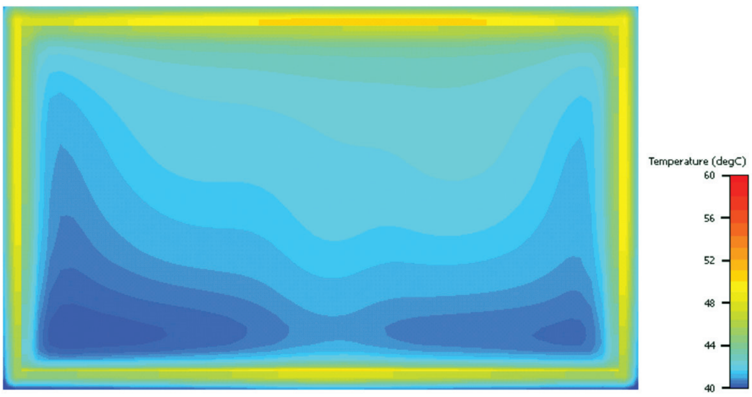


Figure 5. Calculated Temperature Distribution for a Direct Lit TV

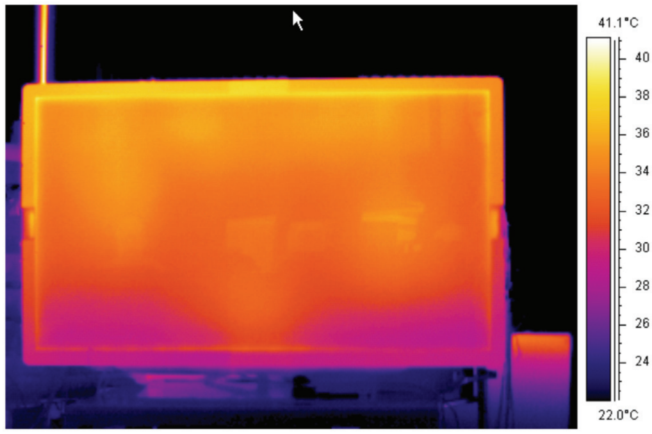


Figure 6. Infrared Picture of a Direct Lit TV

confirms that screen temperatures are well below the aging limit of approximately 60 °C (in 35 °C ambient) and that the temperature difference over the screen is around 8 °C Figure 17 shows the infrared images of the front of a side lit TV, equipped with internal graphite heat spreaders. Comparison of the infrared images with the simulation results shows good agreement. The measurement confirms that the screen is critical with respect to the aging criterion in the zone directly adjacent to the LED bars, and that there is roughly 20 °C temperature difference between the high temperatures at the side and the temperatures in the center.

Acknowledgement

With thanks to TP Vision Innovation Lab, Eindhoven and TP Vision TV Development, Bruges, formerly known as Philips TV

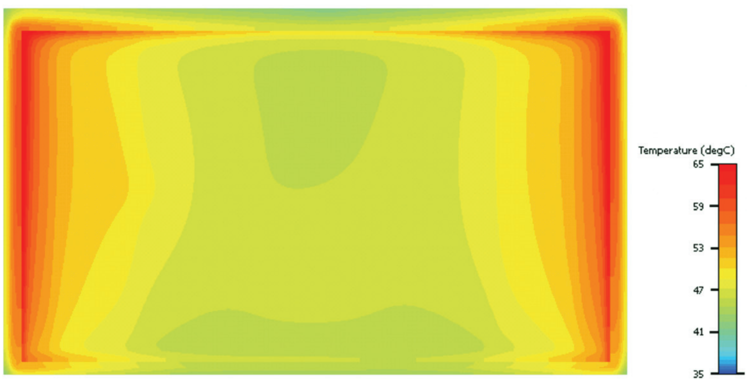


Figure 7. Calculated Temperature Distributions in a Left/Right Edge-lit TV

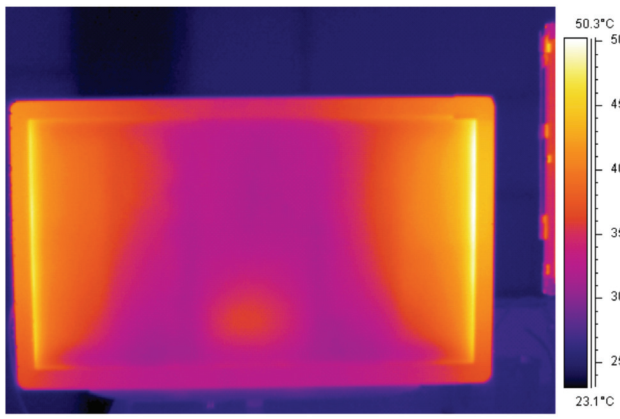


Figure 8. Infrared Picture of a Side Lit TV

Zero to 90 in 166,440 hours

It's what happens off stage that counts

By John Murray
Product Marketing Manager,
Mentor Graphics

In 1994 a young PhD candidate loaded up a transit van, pointed it east and set off for Altenberg. His intention was to meet up with the British skeleton team at one of the most notorious and demanding tracks in the world. In the back of the van was a rudimentary skeleton sled liberally covered with sensors and data acquisition equipment.

Having one of the experienced sliders from the team pilot it down the mountain at Altenberg over a series of runs would fill up the hard disk drives with information on various aspects of sled performance and handling. It was a methodical and scientific approach that would provide crucial and objective information on how a sled performs over the course of a real run. In conjunction with sector and overall times, it afforded the opportunity for real insight into performance.

The cold logic of methodical product development was about to crash headlong in to the extremely focused, tense and introverted environment of a squad of athletes battling for a limited number of team places. "No one would ride it", remembers Professor Kristan Bromley. "At that time, the team was almost uniformly military, so the prospect of jeopardising their chance of making the team in order to help a young engineer who'd just landed up with a fairly unpromising looking sled appealed to no one." None of the athletes would ride the sled, so Kristan did.

Over the next two years, Kristan would be an intermittent and amusing distraction at practice sessions around Europe. Britain didn't make sleds and post-graduate students didn't compete on them, full stop. Except that in 1996 Kristan won the British championships on a sled he designed. The sport had just received the opening salvo from an approach that would upset the established equipment supply chain and lead to the formation of a company that, at the time of writing, supplies the skeleton teams of 23 nations.

Meeting the Bromley brothers, the co-founders of that company, supplies individual anecdotes that taken together offer some explanation as to how it was that a Sheffield based company went on to dominate skeleton bob sled design and produce two world champions from within their stable: the Bromleys are restless. By itself, this might well have left a legacy of half-finished projects constantly superseded by the next big idea. Yet couple it with a grounding in, and respect, for the scientific method and the result is demonstrably a success.

"Our ethos is simple," says Kristan "we need to bring added value through the application of the best technologies." Actually delivering on such a mission statement in a meaningful sense requires focused and intensive effort: computationally, in the wind tunnel and on the track. Asking what was missed, what question wasn't asked, what could be done better. Being restless.

Ask the brothers who they admire from the commercial world and it's interesting that they mention Oakley, Burton, Scott: strong brands with a heritage in their respective sports, each now developed beyond those same sports. Indications of future ambitions perhaps? Kristan relays a quote that has stuck with him "The best advice anyone can get about staying ahead of the copy houses is to build a kick-ass Brand; wear the fastest running shoes in the product development arena; and always be one step ahead. By the time your competitors are copying you; you should have the next product in the market. That is why Apple comes out with so many new models and is so successful - they run faster."

I'm initially surprised to hear the word 'brand' from that side of the table, it feels like too corporate a word from a company founded on putting itself in the full glare of the winter sports media spotlight and 'eating its own dog food' at 140kmh in front of local partisan crowds. But as Richard went on to explain, having a strong Bromley brand isn't about being able to sell their t-shirts for inflated prices; it's about defining what they do better than everyone else. As much to keep their eye on the ball as the company grows, as it is for outside consumption. The best indication of what their brand means within skeleton and what it will come to mean more broadly can be found both in their first foray outside of an elite competitive product.

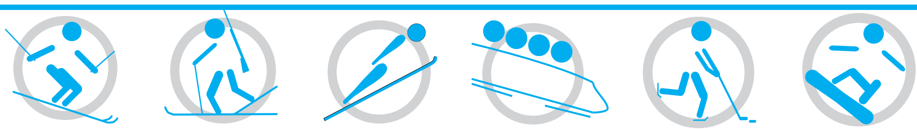
This winter will see the launch of the Bromley Baseboard at a few select ski-resorts. Under development for the last four years, it's fascinating just how methodical, considered and patient the development and launch process has been. What it isn't is at all surprising. Consider the skeleton event itself; what the public sees is 90 seconds or so of action per run. What they don't see is the four years of preparation on the track, in the gym, the hours on the road, the time developing the equipment and in general the devotion to a way of life alien to most people. Teams and athletes used to living and working on those terms are more than capable of taking the

long view and ensuring that everything is engineered such that it comes together at the right time for the main event. If you're successful all you've done is make yourself a target for the competition. Complacency is as much of a foe as injury.

I would suspect that most of us have attended at least one professional training course delivered by a tutor with an armory of sporting analogies at their disposal, most, if not all, of which revolve around the event itself. Perhaps we look in the wrong place for inspiration, maybe the real lessons to be learnt are well away from the field, ring or track.

Is your business ensuring that all your customers see are the 90 seconds of the winning run?

For more information on Bromley Technologies:
www.bromleytechnologies.com



The Road to Sochi 2014

...Continues to be a demanding one. Since Lillehammer, Bromley Technologies has lost no opportunity to continue to test and develop the sled. This is an arduous undertaking which means making sacrifices in a number of areas, but not in athlete preparation. Kristan and Shelley are currently at the wrong end of a long strength and conditioning program which is, to quote Kristan, "horrible!" I'll let the reader reflect on how bad a training program has to be to get that reaction from a man who's been a competitive athlete for as long as Kristan has.

Divide & Conquer

Power Savings from Server to Datacenter

A study by The Department of Engineering at the University of Taiwan to reduce energy costs and improve efficiency in datacenters

By John Parry
Electronics Industry Manager,
Mentor Graphics

Increasingly organizations are choosing to host their data requirements in large purpose-built, energy-hungry datacenters. Datacenters house many racks and a large number of servers where a significant amount of heat is generated from the IT equipment. To remove the heat from equipment so that the electrical components can operate normally, cooling systems must be introduced to provide adequate cooling. Fans, Computer Room Air Handlers (CRAH), and chillers, consume 35%-45% of the total power budget. A big concern for datacenter operators is reducing energy bills and one approach is to improve cooling efficiency.

Figure 1 shows one datacenter room with a single CRAH and 20 racks; each rack is 42U high, and so can house 42 1U-servers. The datacenter includes a raised floor to provide cold air from the CRAH for the IT equipment and a dropped ceiling to draw hot air from the IT equipment into the CRAH. This is a typical datacenter arrangement in which some of the hot and cold air mixes, increasing the room and server inlet temperature. To provide a suitable ambient environment for IT equipment, lowering the temperature may require a higher cooling capability from the CRAH, which will waste more power.

Divided Zone Partitioning

A recent study by The Department of Engineering at the National Taiwan University explored the idea of a divided zone approach to cooling efficiency. A divided zone partition works by concentrating airflow for key components to avoid airflow bypass and controlling different individual zones independently with the aid of a Fan Speed Control (FSC) for the system.

Mixed airflow challenges can be overcome by cooling airflow path management to improve cooling efficiency and power saving [1]. Some datacenters implement hot and cold aisle containment [2, 3], while Zhou et al. [4, 5] propose adaptive vent tiles that can vary their opening for air flow adjustment. Most hot and cold aisle containment systems encompass and seal off the racks in the same row of a datacenter, but providing adequate cooling performance when the loading of one or more racks is much lower than the others remains an issue.

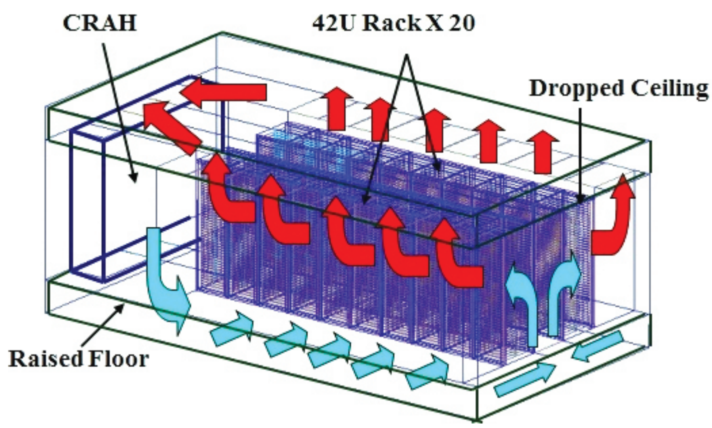


Figure 1. Datacenter configuration and airflow direction

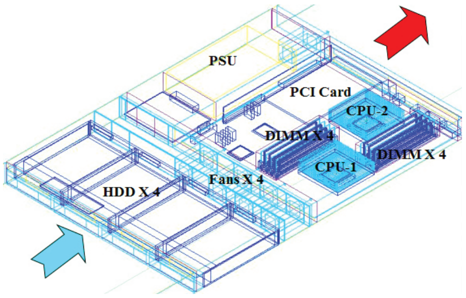


Figure 2. Configuration of the 1U server CFD model and airflow direction

partition shows a significant power saving for IT equipment from the server level up to the datacenter level.

As Simple as One, Two, Three with FloTHERM®

The CFD analysis for the building simulation model is performed with FloTHERM 3D CFD software using a structured Cartesian grid that can be localized and nested to minimize solve times and enable multi-scale modeling for accurate results.

Server-Level Power Savings from a Divided Zone Partition

Figure 2 shows a standard 1U height server CFD model with four fans. Three fans are directed at the CPUs and DIMMs, and one fan towards the PCI card. The PSU includes its own fan, which is located at the rear. We find the thermal solution first to tune the suitable fan curve performance to pass the component thermal specifications at 35°C system ambient. While all the equipment temperatures meet the system thermal requirements, it cannot be assumed that this is the optimum design. Figure 4 shows the airflow distribution. Some airflow does not follow the desired path, providing an opportunity for power savings.

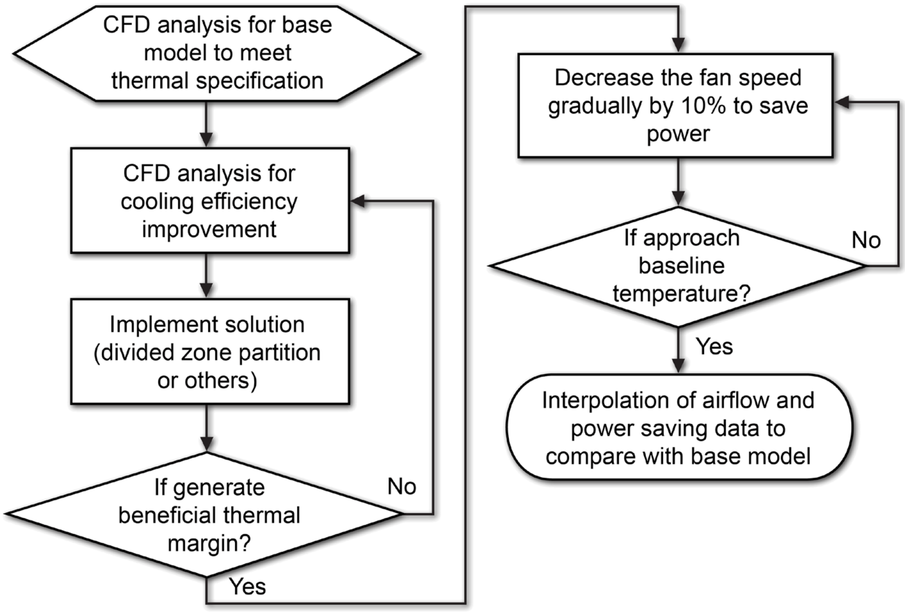


Figure 3. Flow chart of a divided zone analysis for power savings

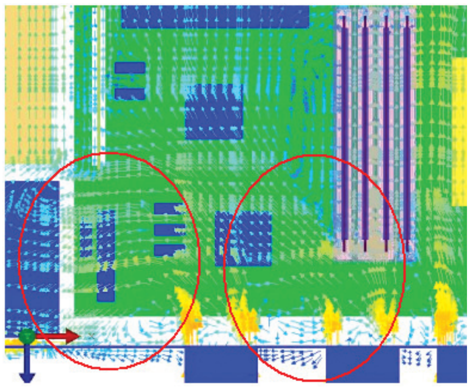


Figure 4. Server system airflow bypass illustration

Servers of the type shown in Figure 2 consisting of Hard Disk Drives (HDDs), Central Processing Units (CPUs), and Dual In-Line Memory Modules (DIMMs), etc., are the main type of IT equipment and can be the target for major power savings in a datacenter, where research [6, 7, 8] into server liquid cooling have shown improvements in cooling efficiency.

In this study, the divided zone method is developed to improve the cooling efficiency for both a server and a datacenter. The effect of a divided zone on airflow management and fan power savings under normal conditions and during a component load change were investigated in detail. Additionally, the utilization of a divided zone

To ensure the design is workable, a fully loaded system is considered at 35°C ambient according to the ASHRAE maximum allowable temperature of the A2 class [9]. First, the thermal solution for the fan and heatsink to meet the fully loaded system requirements that can satisfy all component and device thermal specifications is found for the base model. Second, design optimization is performed to determine whether the divided zone method would improve the cooling. Finally, the solution is analysed to calculate the resulting power savings. The flow chart is shown in Figure 3.

The divided zone partitions were implemented to determine their optimum position for airflow management (Figure 5 overleaf). The resulting effect was that components that generate higher temperatures, such as the CPUs, receive better cooling. The resulting CPU temperature margin allows fan speeds to be reduced, achieving considerable power savings.

The divided zone partition not only saves server fan power directly, but also decreases the system airflow rate requirement, as shown in Table 1. The airflow rate savings can reduce the CRAH blower load. For this case, the divided zone partition can help decrease the server system airflow rate from 59.5 CFM to 51.3 CFM, a 13.8% reduction in the system airflow rate that can further improve datacenter cooling efficiency.

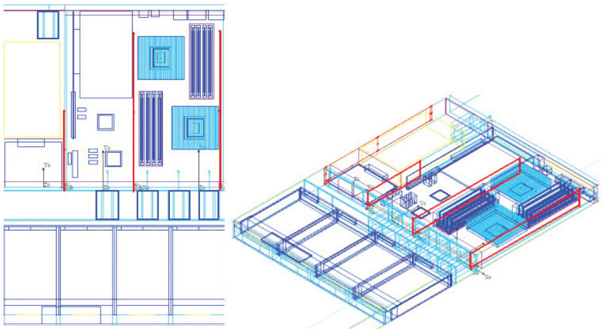


Figure 5. Divided zone partition in the server system

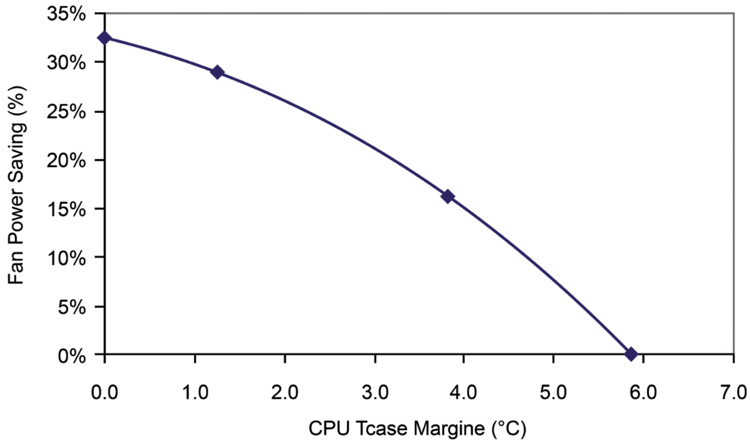


Figure 6. Fan power savings rate vs. the CPU temperature margin of the divided zone partition system

Case	Divided Zone Partition	Fan Speed	CPU2 Tcase (°C)	Fan Power (W)	Fan Power Saving (%)	System Airflow Rate (CFM)	Airflow Rate Saving (%)
1	**No**	**100%**	**79.4**	**66.0**	**-**	**59.5**	**-**
2	Yes	100%	73.5	66.0	-	59.9	-
3	Yes	90%	75.5	55.2	16.4%	56.1	5.7%
4	Yes	80%	78.1	46.8	29.1%	52.5	11.7%
5	Yes	70%	82.0	39.9	39.5%	48.7	18.1%
6	**Yes**	**76.7%**	**79.4**	**44.5**	**32.6%**	**51.3**	**13.8%**

Table 1. Simulation result data of the divided zone partition in the server compared with the original model

Server-Level Power Savings with Regional Load Change Condition

The components in a server system are not always at full load, and the load will not be constant, so there will typically be a thermal sensor in the system to detect component temperature variations caused by ambient temperature or load changes. The fan speed can be modulated according to the thermal sensor data by a controller chip in the server. Figure 7 shows the thermal sensor locations. In addition to the ambient sensor placed in the front of the server, there are component sensors placed in CPUs, DIMMs, etc.

The CPUs and DIMMs are controlled by the three fans shown in the right region, the PCI card and chips are controlled by the one fan shown in the middle region, and the PSU is controlled by its own fan in the left region. In this case, the load in the right region was changed to decrease the CPU power from 95 W to 76 W, a reduction of 20%. The CPU was decreased to 71.9°C, providing a temperature margin corresponding to a fan power saving of 31.4%, as shown in Table 2. When the divided zone partition is added, the power savings can be improved to 46.8%. The divided zone partition is not only useful for typical conditions but also for FSC where the local fan is already independently controlled by temperature in a specific area.

Divided Zone Partition Implemented in a Datacenter

The configuration in Figure 1 follows a typical datacenter layout. Earlier studies have shown that air containment separating hot and cold air can be very effective [2, 3]. In this study a simulation model was created for 20 racks with a total of 252kW of power consumption. Small gaps on each side of the rack were included to emulate a true IT equipment environment. Temperatures were set to an ambient of 27°C, the ASHRAE recommended temperature for this class of datacenter [9].

The hot air from the outlet of the racks and

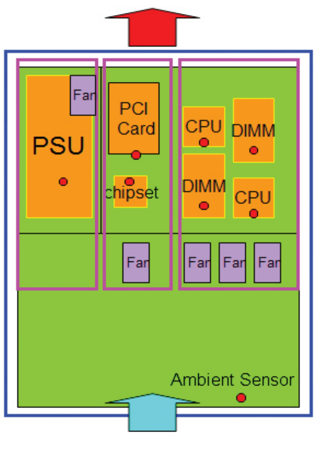


Figure 7. Thermal sensors in the server

the cold air provided to the inlet of the racks are not separated completely, which allows mixing between the hot and cold areas, as shown in Figure 8. To avoid this, some advanced datacenters are implementing air containment, as shown in Figure 9.

At the server level, the component loading is not consistent over a long period of time. At the datacenter level, the rack also experiences different loadings. Although a datacenter with air containment can improve cooling performance, different loadings of the racks in the same row still cause problems. Assume that one of the racks has a zero loading state and that there is no driving fan in the rack. With hot air containment , Figure 10 shows a backflow of hot air from the rear of other racks through the zero-loaded rack to the cold air area. The temperature of the zero-power rack, shown in Figure 11, is seen to be higher than that of other racks.

The partition material could be flexible and transparent plastic sheet for easy access and low cost, with the benefit that every rack can operate independently from the others. From the FloTHERM results, the divided zone has a significant influence on the improvement of the cooling performance of the datacenter under different loading conditions. The study showed that a rack filled with servers with a lower load and correspondingly lower server fan speed resulted in inadequate rack airflow when contained alongside racks containing high-load, high-airflow servers.

The divided zone partition can improve the situation by increasing the airflow rate in the separated region. This can provide better cooling performance for the specific rack and prevent the rack from receiving inadequate airflow, which would lead to an increase in server fan speed and power consumption as the FSC function worked to meet the server's thermal specifications.

Conclusions

A divided zone method has been successfully developed to improve the cooling efficiency for a datacenter. The performance was simulated and investigated under different operating conditions. The major findings are that the divided zone partition can avoid airflow bypass to gain power savings. The partition can save 32.6% of the total fan power consumption and reduce the server airflow rate by 13.8%, reducing the CRAH blower load. For a specific load change case in the server, the FSC function can save 31.4% of the fan power consumption when the CPU load decreases from 95 W to 76 W. Power savings can be enhanced from 31.4% to 46.8% by implementing the divided zone partition with the FSC function. For an advanced datacenter design, the air containment system can avoid the mixing of hot air and cold air to improve cooling efficiency. However, different rack operating load in the same containment region remains an issue. For a 30% load rack case, the implementation of a divided zone partition in the air containment system can improve the airflow rate by 39% for the fan operating in a server case. The divided zone partition shows a significant power savings for IT equipment from the server level to the datacenter level, making it a good choice for datacenter refits to reclaim lost capacity due to cooling.

References

1. M. Green, S. Karajgikar, P. Vozza, N. Gmitter, and D. Dyer, "Achieving Energy Efficient Data Centers Using Cooling Path Management Coupled with ASHRAE Standards2, Proc of 28th IEEE SEMI-THERM Symposium, pp. 288-292, 2012.
2. J. Niemann, K. Brown, and V. Avelar," Impact of Hot and Cold Aisle Containment on Data Center Temperature and Efficiency", White Paper 135 from Schneider Electric - Data Center Science Center.
3. J. Niemann, K. Brown, and V. Avelar, "Hot-Aisle vs. Cold-Aisle Containment on Data Centers", White Paper 135 from Schneider Electric - Data Center Science Center.
4. R. Zhou, C. Bash, Z. Wang, A. McReynolds, T. Christian, and T. Cader, "Data center cooling efficiency improvement through localized and optimized cooling resources delivery", ASME 2012 International Mechanical Engineering Congress & Exposition, 2012.
5. R. Zhou, Z. Wang, C. E. Bash, C. Hoover, R. Shih, A. McReynolds, R. Sharma, and N. Kumari, "A holistic and optimal approach for data center cooling management", 2011 American Control Conference , 2011.
6. M. Iyengar et al, "Server Liquid Cooling with Chiller-less Data Center Design to Enable Significant Energy Savings", Proc of 28th IEEE SEMI-THERM Symposium, pp. 212- 223, 2012.
7. P. R. Parida, "Experimental Investigation of Water Cooled Server Microprocessors and Memory Devices in an Energy Efficient Chiller-less Data Center", Proc of 28th IEEE SEMI-THERM Symposium, pp. 224-231, 2012.
8. M.M. Ohadi, S.V. Dessiatoun, K. Choo, and M. Pecht, "A Comparison Analysis of Air, Liquid, and Two-Phase Cooling of Data Center", Proc of 28th IEEE SEMITHERM Symposium, pp. 58-63, 2012.
9. ASHRAE TC 9.9, "2011 Thermal Guidelines for DataProcessing Environments - Expanded Data Center Classes and Usage Guidance", ASHRAE, 2011.

Divided Zone Partition	Fan Speed	T CPU2 Tcase (°C)	Fan Power (W)	System Airflow Rate (CFM)	Fan Power Saving (%)
No	100%	71.9	66.0	59.7	-
No	77.8%	79.4	45.3	51.7	31.4%
Yes	60%	79.4	35.1	51.3	46.8%

Table 2. Power savings from the divided zone partition for CPU load change situation

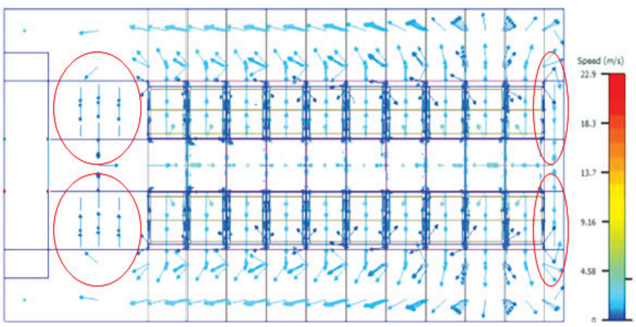


Figure 8. Velocity plot of the simulation results shows the air circulation within the datacenter

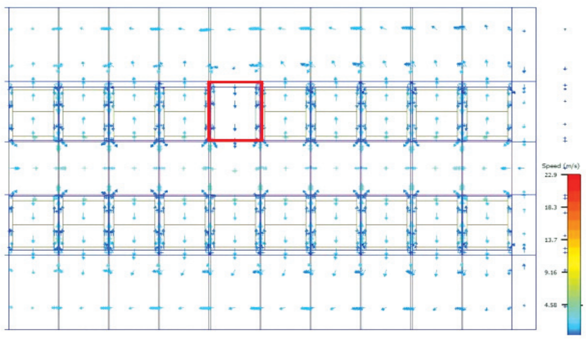


Figure 10. Velocity plot of the simulation result shows that the airflow flows back to the cold air area from the zero loading rack

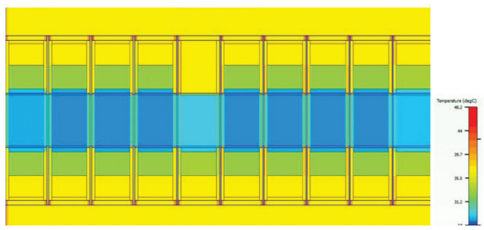


Figure 11. The temperature of the zero power rack is higher than that of other racks

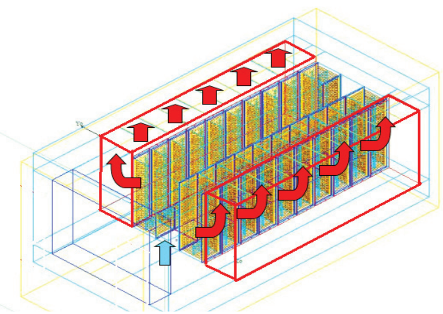


Figure 9. Hot aisle containment within the datacenter

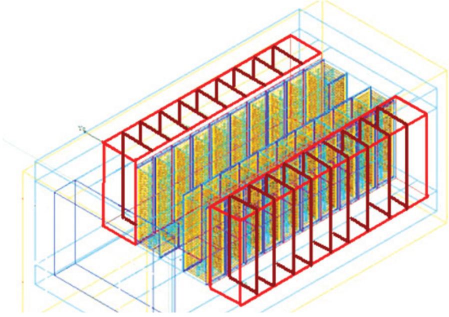


Figure 12. Divided zone partition within the datacenter

Geek Hub

Our team are passionate about all things CFD and love sharing their findings. In this issue, Product Marketing Manager and prolific blogger, Robin Bornoff describes how to bake a games console in an oven!

Video games consoles appeared in the early '70s and since then have grown to be a major industry now worth approximately $30bn per year. Enjoyed mostly by the young, it has not stopped the young at heart from continuing gaming well into middle age and often into the early hours. This includes my colleague Steve Hanslow (FloTHERM® software development project manager) who, together with his love of coding, heavy metal and guitars is every inch a geek. When his games console stopped working it wasn't long before he had determined the likely cause, got his screwdrivers out and embarked on a particularly novel method of repair.

It was estimated in 2007 that such consoles account for 75% of the entire world's general purpose computational power. Such power comes in the form of components that are soldered onto a printed circuit board (PCB). The components are packaged integrated circuits, small wafers of (commonly) silicon encased in a package that allows the micro circuit to be connected to others on the board. It is these wafers that are actually referred to as 'chips'. Modern day CPUs and GPUs are packaged such that an array of solder balls are used to connect that component to the PCB. The PCB manufacturing process sees the component placed on the surface of the PCB, the PCB assembly put through a hot air oven that causes the solder balls to melt such that they form a solid contact with the board, the so-called solder reflow process. The PCB then cools, the solder solidifies, the PCB is then put into a box and the product sold.

In operation the components on the PCB dissipate power and get hot as a consequence. Extreme temperature variations in the component and PCB lead to stress and strain build up so that it can ultimately result in cracks appearing in the solder balls. This break in the electrical connectivity results in the failure of the system to function properly. Gamers will be well aware of RROD and YLOD indication of such failures (if you know what those acronyms stand for without resorting to Google then you too have achieved geek status) and it was this that Steve encountered.

Robin Bornoff
Product Marketing Manager
Mentor Graphics

Commercially available reflow ovens run into the $100ks but the humble domestic oven can reach the temperatures necessary to reflow the solder, reconnecting the component to the PCB and thus restore electrical function. To do this Steve removed the console PCB from its housing, used bolts to ensure an air gap under the PCB (Figure 1) and then placed this on the middle shelf of the oven (Figure 2) and turned it on. Central to achieving a good reflow process is control of the rate at which the PCB heats up, the time the PCB experiences sufficient temperature to melt the solder and finally the rate at which the PCB cools. The key is to make sure all the components experience solder melt temperature but do not get so hot in the oven so as to break them. If all components heated up at the same rate then this would not be an issue. However, larger components heat up more slowly and smaller ones more quickly leading to temperature variations during the reflow process. Lead free solder melt temperature is approx. 217°C and commonly the maximum temperature a component can withstand during reflow is approx. 250°C. This 'reflow window' is therefore about 35°C.

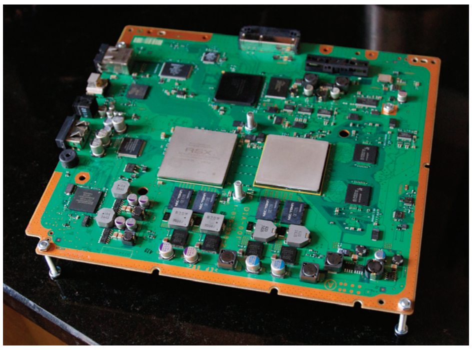


Figure 1. Games Console PCB Ready for the Oven

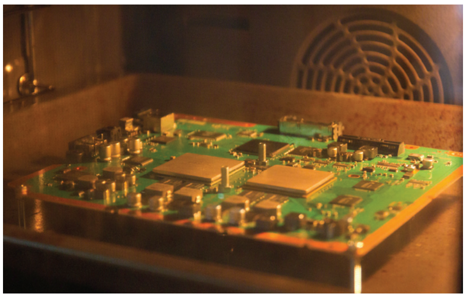


Figure 2. Chips Cookiing in the Oven

Thermal simulation using tools such as FloTHERM or FloTHERM PCB allow for such transient thermal responses to be predicted. Figure 3 shows the temperature variation over the PCB when the CPU component (the one that has cracked solder ball connections) reaches solder melt. One way of controlling this temperature variation is to shield those smaller components that would otherwise exceed their maximum temperature. In keeping with the DIY nature of this repair Steve used bluetack shielded by foil to cover those components, minimising their temperature rise thus reducing the temperature variation at the point of reflow (Figure 4)

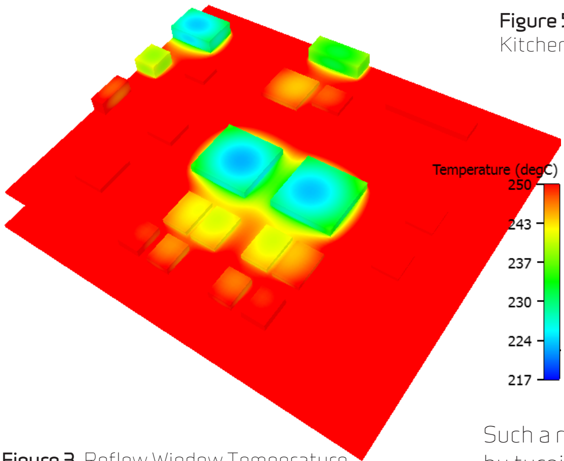


Figure 3. Reflow Window Temperature Distribution as Simulated by FloTHERM

Figure 4. Bluetack and Foil Shielding Small Fast to Heat-up Components

The rate at which the PCB assembly cools after reflow is also important. A fast cooling rate creates a fine grain structure within the solder that improves subsequent mechanical stability.

Figure 5. Cooling the Oven or Heating the Kitchen?

Such a rate is not possible to achieve simply by turning the oven off, nor by also opening the oven door. To achieve the required 4°C/s cooling rate Steve placed a large fan next to the open oven door to help induce convection of cooler air into the oven, accelerating the cooling rate (Figure 5). The location and orientation of the fan will control how much cool air is pushed into the oven. Simulation is well placed to determine this. In this case, having the fan blow parallel to the open oven door induces a recirculation in the oven that pulls in just enough cool air to achieve the desired cooling rate (Figure 6).

Suffice to say that after Steve reassembled the PCB in the console chassis he was not at all surprised to find that it was working properly again. Gaming has continued unabated ever since.

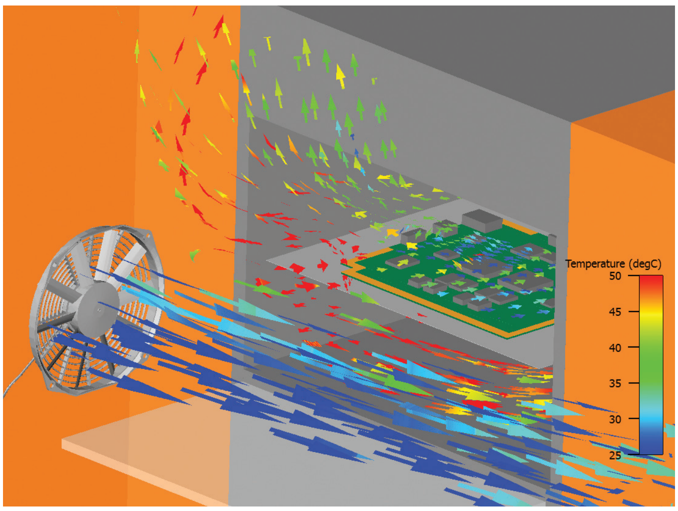
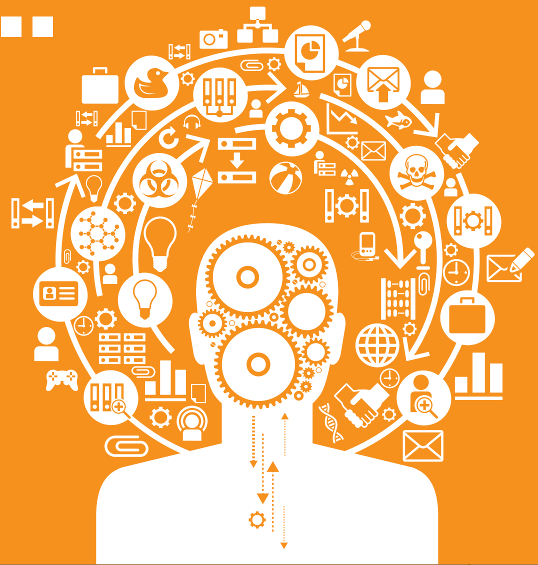


Figure 6. Effects of the Cooling Fan as Simulated by FloTHERM

Brownian Motion...

The random musings of a Fluid Dynamicist

Brownian motion or **pedesis** (from Greek: **πήδησις Pɛdeːsɪs** '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.



I can haz access to source material?*

Why the internet doesn't have to be a world wide waste of time

However you cut it, the internet is a pretty amazing resource. It may have its darker recesses, but the web can only serve as a mirror, it can't actually generate any content spontaneously any more than a cat can take a photo of itself doing something funny and send it round to 20 colleagues.

However, the fact is that one side effect of making everyone their own publishing house has only served to highlight the fact that not everyone should be their own editors. In particular, the comments sections of most online news sources have such a low signal to noise ratio that they usually only serve to prove the rule that those with the least to say shout the loudest. On that point, did people always get that angry that quickly about, well, pretty much everything and anything? I struggle to remember the pre-internet world, but I'm pretty sure it wasn't populated by people teetering on the edge of rage because of a difference of opinion on the history of fountain pen nib development.

Nevertheless, I don't subscribe to the train of thought that blames the web for everything from declining attention spans and social isolation to wrecking pub quizzes. Exhibit One: Wikipedia. I bet you've used it in the last week at least once and statistically it's more likely than not that you've never paid a penny toward it. Secondly, 'build your own robot'. Whether you want to or not isn't the point: you can sleep soundly tonight knowing that there's more useful and free information than you could possibly want on this, or any other, subject only a search engine click away.

Finally, and here's the clincher for me, accessing source material is no longer the preserve of the academic elite. If you want to find out if it's safe to drink this, drill for that or whether burning either will ultimately lead to your house being a whole lot closer to the tide line than you'd like, there is no longer any reason to have this information filtered either by the owners of your favorite daily newspaper, or by the hivemind of the internet. And I say this as a fundamentally lazy person and an expert on precisely no subjects, it genuinely takes less effort than you'd imagine. The worrying thing is just how often it turns out that research is being misrepresented to one degree or another by people or organizations you might have expected better from.

Were you so minded, in the time it took you to read this far you could probably even have checked the veracity of my claim that most of the readers of this column who use Wikipedia have never given it a financial donation (and yes irony fans, I do agree that Wikipedia probably shouldn't be your ultimate source). Or you could be well on your way to building your first robot by now. The internet: it also does funny cat pictures.

Turbulent Eddy

* Resonating http://en.wikipedia.org/wiki/I_Can_Has_Cheezburger%3F