Engineer

INNOVATION

SIEMENS
Ingenuity for life

siemens.com/simcenter

View from the bridge

Siemens PLM Software

Jan Leuridan
Senior Vice President
Simulation and Test Solutions

Engineer Innovation is at the heart of what you are doing on a daily basis using our Simcenter™ portfolio. Hence we found it very relevant to select this as title of our new newsletter. Sharing with you how our customers are successful in exploring the engineering boundaries is what we want to achieve in this new newsletter. The breadth of applications that are covered in this edition is fascinating: SANDEN Manufacturing's approach for bringing water heaters to market faster, Samsonite explaining how to improve their already high quality rolling luggage, Honda improving both cabin comfort and emissions using our engineering capabilities covering both test & system simulation approaches, and Continental increasing the reliability of their electronics.

You will notice that this new newsletter covers a wide variety of industries, each with their own industry challenges but also with a very common one: better products, delivered to market faster.

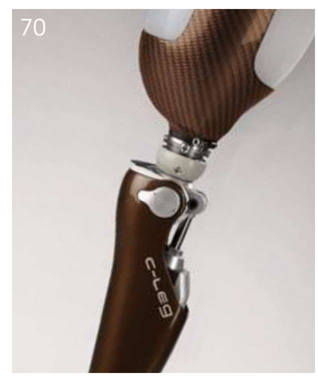
The strong theme from reading these case stories is the increasing role of simulation and controlled testing, with considerably less, but more advanced, prototyping. STX France who have used simulation for many years to optimize hull geometry and ship-wave interactions, large scale simulation, and have now turned this technology to plume dispersion, a far more detailed analysis. Even if you suffer from motion-sickness I urge you to read the full article on page 18 demonstrating the power of simulation.

Continuing that theme are the engineers at Renault Sport Formula One. It must be one of the most time-critical applications, with continual improvement on a day-to-day basis with vast amounts of data to support their decisions on aerodynamics. Paul Cusdin from the Renault team sums this up well, "The computational domain not only augments the physical domain, it also improves it."

Finally I am reminded that behind every challenging customer project is a team of dedicated engineers seeking a better, faster or more cost-effective solution. Engineers are curious, problem solvers and it is always with tremendous pleasure I hear about how they use our solutions to improve lives. That's why in this new newsletter we also make a place for some other aspects. We're adding the Geek Hub article which draws on the personal experiences of one of our team members. In this issue they look at prosthesis and how keen our engineers were to analyze it, and see where they could make improvements. ■

Our cover celebrates some of the many faces of physics, science, mathematics, and innovation that have influenced our lives and the work we do.

Contents



Engineer Innovation

Regular Features

Project: Tailless Drone

CFD simulation and wind tunnel test correlation for a tailless multi-variant sUAS

By Harsh Shah, Research Engineer, National Institute for Aviation Research, Wichita State University

There is a unique class of drone; small unmanned aerial systems or sUAS. These drones have a maximum weight, including payload, of 55 lbs or 25 kgs. They are also restricted to fly under 100 mph. These types of drone are becoming very popular for commercial uses such as aerial photography in real estate; inspection of power poles; and cellular towers. They are even being investigated for package delivery by companies such as Amazon and UPS.

NIAR - the National Institute for Aviation Research at Wichita State University has chosen to design a sUAS to act as a technology demonstrator to highlight the capabilities of NIAR in sUAS design analysis and manufacturing. The design specification they chose is for the sUAS to cruise at 50 mph with a maximum take-off weight of 55 lbs. It uses electric propulsion for vertical take-off (VTOL) flight and an internal combustion engine with pusher propeller for forward flight. Multiple design iterations were carried out on a parametric CAD model to aerodynamically design the wings. The aerodynamic characteristics of the final sUAS configuration are discussed here.

A one-third scale wind tunnel model was fabricated using additive manufacturing techniques and was tested at the NIAR Walter H. Beech wind tunnel. A comparison between the CFD results and the wind tunnel test results were made. Additionally, a coupled CFD-thermal analysis was conducted to understand the cooling performance of the air-cooled internal combustion engine. The results show improvement in the cooling performance of the air-cooled internal combustion engine of the sUAS.

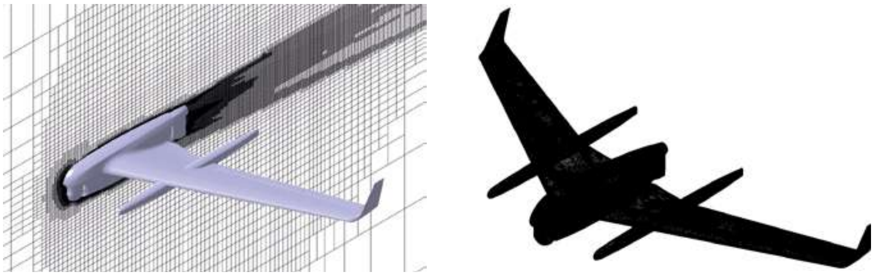


Figure 1: Cartesian Mesh used in the numerical model for the sUAS.

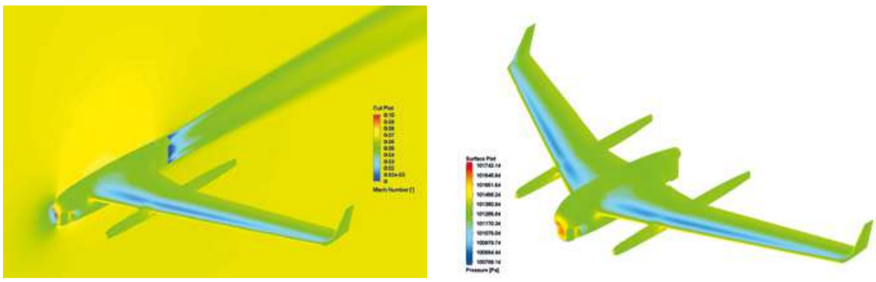


Figure 2: Mach No. contour and surface pressure distribution at Velocity = 50 mph, Angle of Attack= 0°.

The aerodynamic design of sUAS was inspired by the following stakeholder requirements:

- Tailless, multi-variant configuration;
- The ability to perform search and rescue missions, and drop a payload of 5 lbs, cruise velocity of 50 mph;
- Forward flight endurance of five hours; and
- Maximum take-off weight of 55 lbs in order to qualify for the FAA Part 107 certification category.

The commercial CFD tool – Simcenter FLOEFD™ software was used to aerodynamically design the sUAS. Simcenter FLOEFD offers many advantages over other CFD codes. Its CAD embedded functionality allows for automatic detection of fluid regions and eliminates the need to modify or clean-up the geometry. Therefore, the lead-time for evaluating multiple design iterations is shorter. The immersed-body Cartesian mesh allows for quick mesh building for any complex geometry.

The parametric study of the design for different combinations of airfoils, sweep and incidence angles and dihedral angles was carried out to optimize the aerodynamic performance and to achieve a stable configuration.

Simcenter FLOEFD is based on the Favre-Averaged Navier-Stokes model. The governing equations are discretized using the finite-volume method. The pressure-based solver in Simcenter FLOEFD is used in this study. It is based on implicit scheme with second order accuracy of spatial derivatives and first order accuracy of time derivatives. The flow was computed to be fully turbulent and the modified k-ϵ two-equation turbulence model was used. The Cartesian mesh used in this study is shown in Figure 1. The pressure distribution and Mach Number contour are shown in Figure 2.

The wind tunnel model was 3D printed out of PC-ISO in the Fortus 400 printer. It is printed at one third scale to fit within the printer limits. The wind tunnel test was performed at the NIAR Walter H. Beech wind tunnel. It is a subsonic, closed return and atmospheric type with test section of 7’x10’ in cross sectional dimensions. The model configuration is a sting mount with internal balance. The model with the sting mount is shown in Figure 3.

CFD simulations were performed at cruise conditions (Velocity = 50 mph, Reynolds no. ≈ 750,000). Boundary layers were resolved using the Modified Wall Functions approach in Simcenter FLOEFD. Three methods are available: 1) thin boundary layer approach, which is based on integral boundary layer method, 2) thick boundary layer, which is based on Van Driest’s velocity profile and 3) hybrid, which combines the thin and thick boundary layer method. For this study, the thin and hybrid methods were investigated. The comparison of the numerical results with the experimental results is shown in Figures 4 and 5.

Examining Figure 4 it can be seen that good agreement exists between the Simcenter FLOEFD simulation results

Figure 3: Mach No. contour and surface pressure distribution at Velocity= 50 mph, Angle of Attack= 0°.

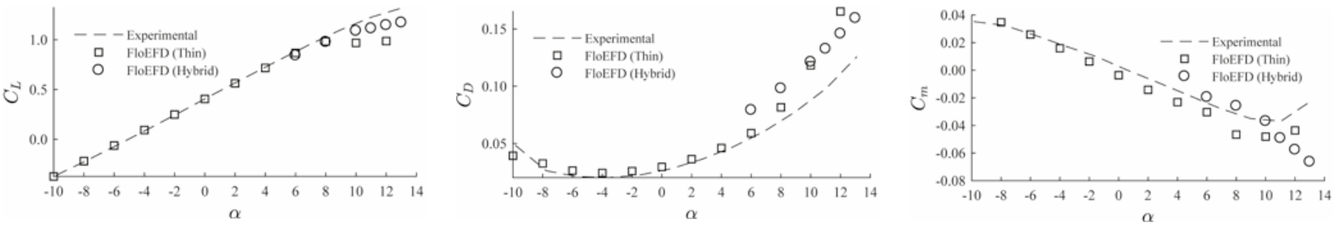


Figure 4: Comparison of Lift, Drag and Pitching Moment coefficients

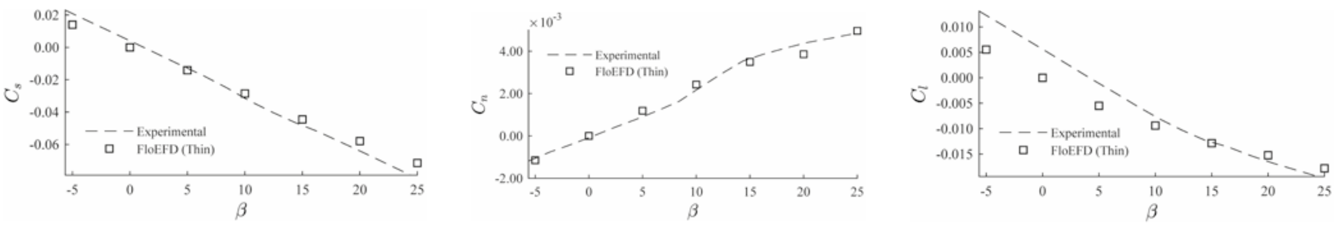


Figure 5: Comparison of Side Force, Yawing and Rolling Moment coefficients

and the wind tunnel results for angles of attack between -10° to 5° using the thin boundary layer approach. This approach also works well for the sideslip angle sweep as can be seen in Figure 5. The hybrid boundary layer approach offers improvement in lift prediction at high angles of attack. Drag prediction at higher angles need further investigation as there was a significant deviation from test.

The thermal management of the sUAS internal combustion engine was the second focus of the CFD analysis. A Conjugate Heat Transfer (CHT) analysis was performed to design ducts for improving cooling efficiency of the sUAS internal combustion engine. The CHT analysis is based on heat transfer (Fourier's Law, Newton's Law of Cooling) due to conduction in solids and heat transfer due to convection in fluids (Navier-Stokes equations). The ability to do this in Simcenter FLOEFD provides a powerful tool to the design engineer where this would normally have to be left to the CFD analyst. The comparison of the temperature distribution of the engine assembly with and without the ducts is shown in Figure 6. A reduction of ≈ 17% in maximum temperature of the engine assembly is obtained with ducts and vents incorporated in the assembly.

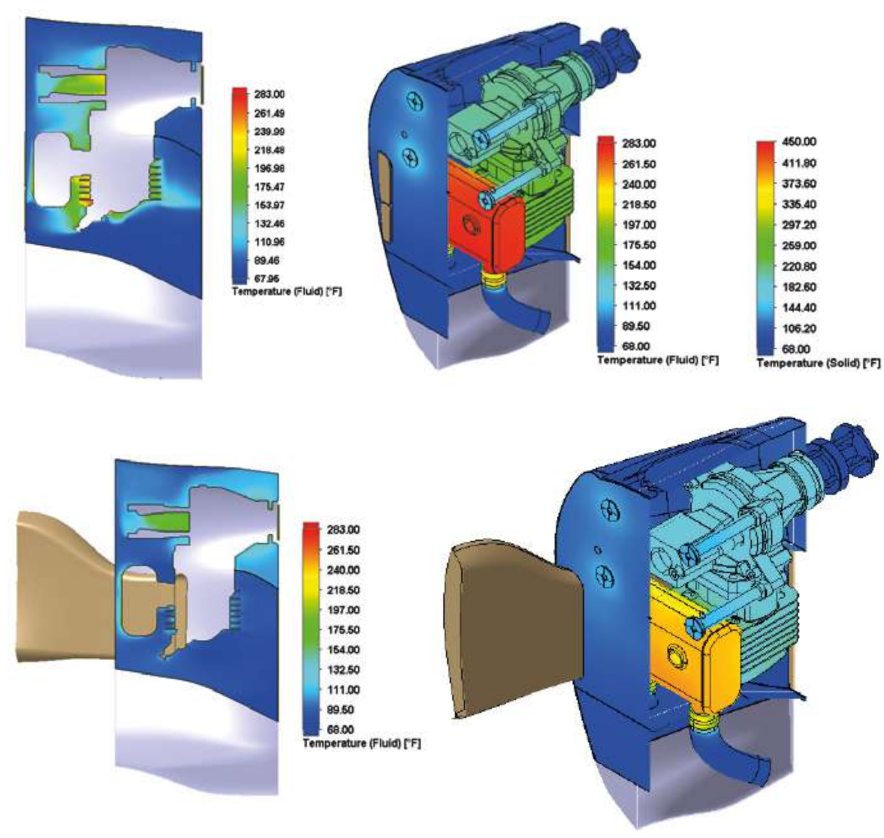


Figure 6: Comparison of temperature distribution (fluid and solid) for the engine assembly with and without ducts .

Overall Simcenter FLOEFD performed well and was a valuable tool when designing the demonstrator sUAS for NIAR. It allowed multiple design iteration to be performed on both the aerodynamic design of the drone as well as optimizing the cooling strategy for the internal combustion engine. Simcenter FLOEFD has proven to be the go to tool for NIAR and has been used on numerous projects. ■

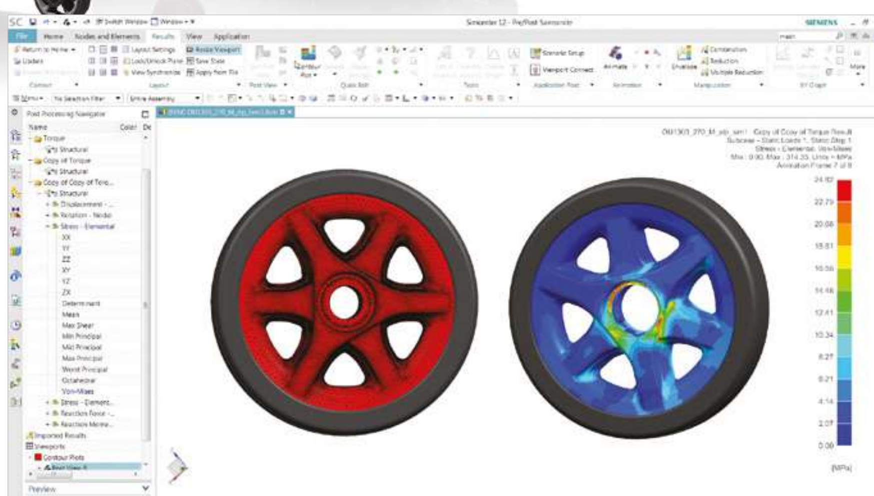
Quality that Exudes Class

Luggage manufacturer uses Simcenter 3D to design lighter and impact-resistant suitcases

Quality in an elegant design
Anyone who travels knows the hassle: arriving at the airport, in a rush and heavily packed, but a bit relieved that your suitcase features a steady pair of wheels that can assist you. You put your luggage on the scale. It's your lucky day. Even though you've packed far more clothing and gear than you will ever be able to use during the course of your trip, you managed to stay within the airline's allowance. The front desk officer tags your suitcase, and then it disappears into a black hole where it bumps along a network of conveyor belts and travels through hands that may not be as careful with it as you are before it gets on the plane. And once you arrive at your destination, you're praying to see your suitcase back in the same state as you left it, and with everything inside intact.

Many people choose to buy a suitcase for its fashionable design. But at the same time, the functional performance requirements are not minor, especially in terms of weight and durability. And the market shows very little tolerance for failure. A bad experience makes consumers instantly opt for another brand, and with online reviews and social media, customers have the power to kill a company's reputation. Among the many manufacturers who bring suitcases to market, only the ones who can deliver consistent quality survive.

For more than 100 years, Samsonite has built a solid reputation in the



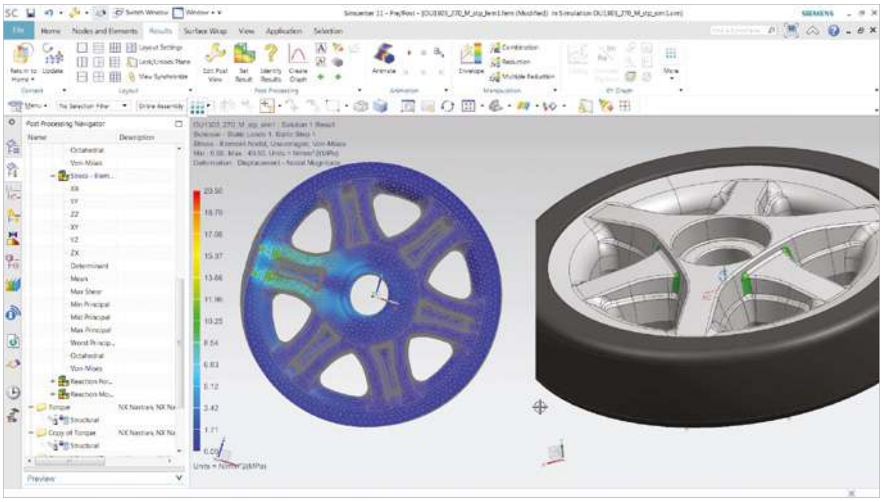
travel goods sector. Running a successful business for such a long time with a rather small variety of products is truly exceptional. Samsonite's secret? Besides creating classy designs, they continually seek out new material technologies and production methods, and have an absolute obsession for quality. Very few products are more thoroughly tested than the suitcases that withstand the torture rooms in Samsonite's factories.

Starting with FEM

In Oudenaarde, Belgium, Samsonite engineers apply the latest material and production techniques to develop the lightest and strongest suitcases. All the models for the European market are made here, and some designs are

"Simcenter 3D helps us dramatically reduce product cost and lead time, while increasing quality."

Vivien Cheng
Head of the Product Development Department
Samsonite



produced and offered in global markets. To become more effective in achieving the desired product quality, the Samsonite team in Oudenaarde selected Simcenter™ software from Siemens PLM Software.

“We were already Siemens customers for a very long time,” says Vivien Cheng, Head of the Product Development Department. “For many years we have successfully used NX software for our CAD/CAM work. So when we realized we could benefit from increasing our knowledge in CAE and FEM, and from deploying a solution inside our organization, Simcenter 3D was the logical choice.”

Samsonite recognized the relevance of using the Finite Element Method (FEM) more than ten years ago. The company outsourced the analysis for a while, before building up the know-how internally and doing it in-house. Now that the company has made the necessary investments, Samsonite has three enthusiastic engineers who perform the bulk of the simulations and are eager to extend the range of the application for the insights it provides them.

Gaining insight

“Most of our CAE work is currently still on an as-needed basis,” explains Gilles Vanneste, 3D Engineer at Samsonite. “From the tumble and the drop tests, we observe damage on, for example, the wheels, a bracket, the carry handle, or the shell, which is unacceptable. Using that knowledge, and by applying the proper boundary conditions, we then try to simulate the behavior in Simcenter. In this way we can discover the locations of high stress concentration, which areas need to be improved, where we need to reinforce, and much more. Sometimes we even learn things we had no idea about, such as the effect of adding ribs on the overall suitcase stiffness.”

These valuable insights gave Vanneste the taste for more. He is quickly building up knowledge of boundary conditions and improving his modeling skills. That doesn’t go unnoticed within his team. “It regularly happens that a designer comes to me with only the drawing,” says Vanneste. “We then evaluate whether the suitcase will be strong enough, or if the entire concept needs to be reconsidered. It’s actually quite thrilling when thinking about it in this way, because it means that we

are doing all the prototyping in a virtual way. The main advantage of using Simcenter 3D is that we quickly get feedback on whether certain components are strong enough, or whether we can make the structure lighter. It helps us to decide where we can remove material, and guides us to where we need to add it."

Reducing physical prototyping
Wim De Vos, Project Manager at Samsonite, confirms that the amount of physical prototyping has been dramatically reduced since the company implemented Simcenter 3D. The main benefit is significant time savings. "In the past we went straight from the design table to creating a physical prototype, which we then tested," says De Vos. "That process took us about 16 weeks, and we weren't even sure if the suitcase would be as strong as expected. "Simcenter 3D allows us to do upfront simulations, reassuring us that the prototype will be okay from the first time, avoiding several loops."

"Simcenter 3D helps us dramatically reduce product cost and lead time, while increasing quality," says Cheng. "It's hard to quantify this, as we are annually busy with more than ten projects on different scales, from makeovers of existing products to creating completely new ones. But since we started using Simcenter 3D, we have an effective process in place to do our job, which is making light and impact-resistant luggage."

Even more FEM in future
Cheng sees the application of Simcenter 3D growing in the future. "We have to increase our simulation capabilities," she says. "More accurate modeling and boundary conditions along with the execution of dynamic simulations will help us create even better suitcases."

Cheng also highlights another important aspect that makes Samsonite walk further down the simulation path. "Simulation will be crucial to innovate our products," she says. "We collaborate with universities to do research on composites. Doing finite element analysis using Simcenter 3D is required to prove that we are serious about it, and to get funding. Thanks to using FEM, we could produce our self-reinforced polypropylene luggage, which is the lightest on the market. This has been an enormous success. We are very determined to continue working on this, and further optimize our materials to keep our number one position." ■

"The main advantage of using Simcenter 3D is that we quickly get feedback on whether certain components are strong enough, or whether we can make the structure lighter. It helps us to decide where we can remove material, and guides us to where we need to add it."

Gilles Vanneste
3D Engineer
Samsonite

Thales Alenia Space

Thales Alenia Space partners with Siemens to explore new tools and methods for acoustic testing in the space industry

Launch survivors

The launch of a communication satellite into space is a traumatic event for its parts and pieces. Components are heavily exposed to the eventuality of breakdown or damage, yet engineers need to ensure that the satellite reaches its orbit in faultless operating condition.

Satellite qualification testing is the ultimate step of a satellite development process. This step certifies that every single satellite element will survive the traumatic launch conditions. Relying on decades of experience in delivering dedicated solutions for dynamic environmental testing, Siemens PLM Software is a preferred partner of many space agencies worldwide for satellite qualification testing. The combined capabilities of Simcenter Testlab™ software and Simcenter™ SCADAS hardware, both part of the Simcenter™ portfolio, ensure safe and efficient qualification testing.

Thales Alenia Space, a leading European space satellite and payloads manufacturer, partners with Siemens PLM Software to explore new methods for satellite testing. Thales Alenia Space is a joint venture between Thales (67%) and Leonardo (33%). Combining 40 years of experience with a unique blend of expertize, talents and cultures, Thales Alenia Space architects design and deliver high-technology solutions for telecommunications, navigation, earth observation, environmental management, exploration, science and orbital infrastructures.

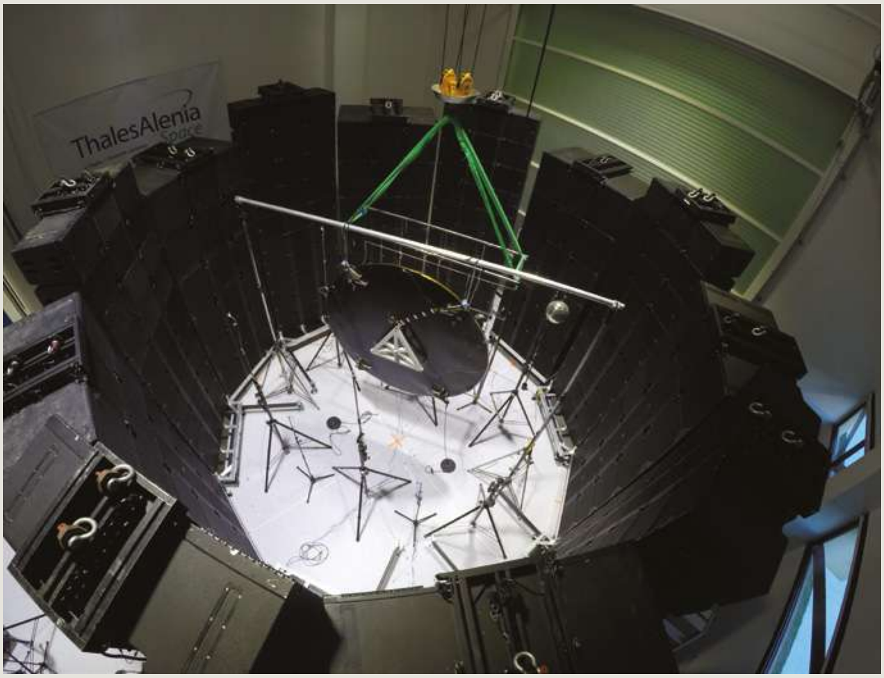
Reforming methods

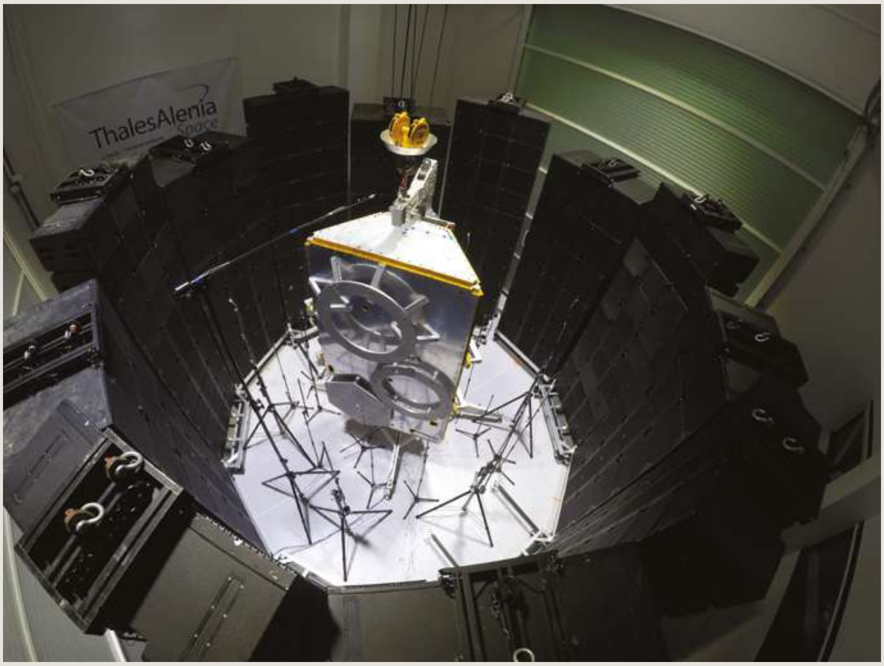
Dynamic environmental testing encompasses a number of essential tests for qualification of space hardware, and acoustic testing is one of those most crucial steps. It subjects an item to intense noise levels while measuring its vibration response. This test is performed on both component (reflectors, solar panels) and system (full satellite) levels.

Satellite acoustic testing is traditionally performed in acoustic reverberant rooms. In most cases, these large facilities (sometimes over 1,000 cubic meters to accommodate large spacecraft) are filled with gaseous nitrogen which has a lower sound absorption coefficient than air. The noise is generated by modulators connected to horns placed in the chamber; the result is a noise level that can reach over 150 decibels (dB). In these facilities, engineers simulate the noise field that excites the satellite in the fairing of the launcher. In addition to its extensive offering for multi-channel data acquisition, Simcenter offers a comprehensive solution to control acoustic signals in the reverberant room.

Acoustic testing in reverberant rooms is a safe, reliable and accurate testing method, while at the same time extremely costly and time-consuming. Satellite subsystems such as antennas or reflectors are also tested according to this method, often in medium-sized reverberant rooms.

Over the past 15 years, the U.S. space industry has been trying alternative testing methods. Research projects evaluate methods that offer a more economical option as well as more flexibility to perform the tests away from sparse and costly-to-operate facilities. A Direct Field Acoustic eXcitation (DFAX) method, also named DFAT in the U.S., has been developed and is partly used today for qualification of North American satellites. DFAX has lower running costs and initial investment and offers the technical benefit of considerably shorter ramp-up time to level or better controllability in the lower frequency range of 20 hertz to 60 hertz (Hz).





> *"Siemens brings its expertize to solve the complex challenge of generating an homogenous sound field around the test item."*

Christophe Fabriès
Project Leader – Antenna Mechanical Analysis
Thales Alenia Space

In 2016, the National Aeronautics and Space Agency (NASA) published the NASA Handbook 7010, which is the first handbook that lays out the guidelines for companies wanting to use the new acoustic testing methodology. Like their North American counterparts, leading European industry companies such as Thales Alenia Space are conducting experiments to explore and validate new methods for satellite acoustic testing.

Pump up the volume

What do Werchter, Belgium; Roskilde, Denmark; and Kourou, French Guyana have in common? Werchter and Roskilde are locations of popular open-air rock and pop festivals that bring crowds of passionate music lovers together. Incidentally, in recent years, performances of modern loudspeakers and amplifiers have been pushed to their maximum to better entertain the ever-growing hordes of music fans. The availability of commercial loudspeakers and amplifiers capable of generating the sound field required in a test has made the development of the direct field acoustic excitation method possible. In a DFAX test, the specimen is placed in the middle of a loudspeaker circle and gets excited by a direct acoustic field. Modern loudspeakers and amplifiers deliver the required high decibels to obtain the target overall sound pressure level (OASPL). The vibration levels measured on the specimen during the DFAX test are comparable with those measured with reverberant field acoustic excitation. In the near future, satellites that are placed on the European launch pad of Kourou might have been partially qualified using rock concert loudspeakers.

Clearly, DFAX lowers overall test expenses, and can be performed (nearly) everywhere bringing more flexibility with shorter test sequences. However, safety, reliability and accuracy of the tests should not be discounted. The nature of the sound field in a DFAX test differs from that of a reverberant room test. This difference needs to be accounted for in order to produce realistic test conditions. The engineers of Thales Alenia Space are relentlessly working to improve and validate the DFAX method.

Homogenate the sound field

The engineering team at Thales Alenia Space in Toulouse, France, develop satellite components that will later be integrated in to the full system. The company owns an acoustic reverberant room facility; however, this facility is located in Cannes, some 500km from Toulouse. In practice, this implies that every newly developed component needs to be shipped to Cannes for acoustic qualification testing, leading to additional costs and delays. With the help of Siemens PLM Software engineers, the team explored a new DFAX method that would permit on-site qualification testing of components. “Thunder” is the name of the project, apropos for a project that generates a 147 dB sound field in an International Organization for Standardization 9 (ISO9) clean room.

The objective of the test campaign is to reproduce the acoustical environment that a communication satellite is subjected to when placed in the fairing of a satellite launcher. The test setup is designed to generate the high acoustic levels that excite the specimen during takeoff. The setup is comprises 96 loudspeakers, stacked in 12 columns and adequately positioned in a circular configuration, and 96 amplifiers that deliver the required high power of 4x5 kilowatts (kW). The specimen being tested is placed at the center of the five-meter cylinder of loudspeaker columns. The challenge is to reproduce a uniform diffuse acoustic field around the specimen. In the described test, the team evaluates the behavior of the specimen, making sure that it is equivalent to the one specimen placed in an acoustic reverberant room.

Christophe Fabriès, Project Leader at Thales Alenia Space explains: “Siemens brings its expertize to solve the complex challenge of generating an homogenous sound field around the test item. The solution uses Simcenter SCADAS hardware fitted with a multiple inputs, multiple outputs (MIMO) controller and combined with Simcenter Testlab software. It requires the measure of 16 microphones positioned around the specimen, analyzes their response and corrects drives. The corrected drive values are reinserted in the loudspeakers to create an homogenous acoustic field.”

Simcenter SCADAS hardware provides the adequate voltage output. Using a closed-loop algorithm, the solution ensures that the output matches the reference profile drive. The method allowed the team to successfully qualify the reflector shell of an antenna subsystem demonstrator. In the second phase, the team performed qualification tests on the mid-sized platform of a Global Star second generation (GB2) spacecraft mockup. The full qualification sequence was realized according to the multi-launcher's requirements.

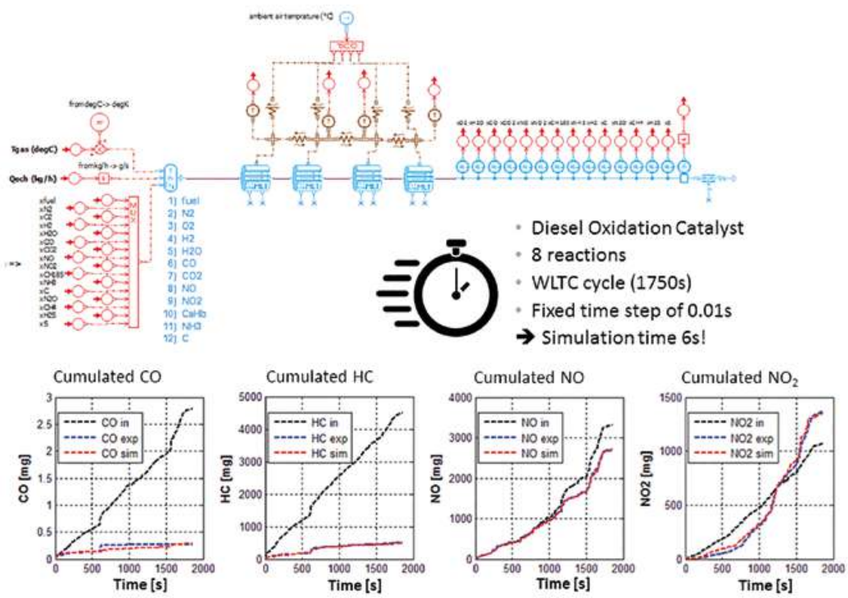
“This phase helped us validate that the testing method is suited for qualification of spacecraft from constellation production lines,” says Fabriès. “With this method, we are able to conduct up 25 test runs within a test session. It is a very efficient way of testing newly-designed hardware. It will allow us to explore more variants with the possibility of performing immediate validation in the lab.” ■

“The project delivered the unmatched performance of generating a 147 decibels direct sound field.”

Christophe Fabriès
Project Leader – Antenna Mechanical Analysis
Thales Alenia Space

Optimizing After-Treatment Systems Performance.

Using a model-based system design approach to support the engineering of exhaust systems in an RDE context



The growing adoption of new regulations for the vehicle tail-pipe emissions has caused significant changes in the product development cycle of automotive OEMs. Rules have changed quite radically within a short timeframe, shifting from regular but smooth evolution of the emissions legal constraints we used to have year after year. The application of Real Driving Emissions (RDE) strongly impacts the way manufacturers engineer vehicles, and this also affects suppliers who can promote new features, tools and methodologies.

The new RDE regulations – whose objective is to evaluate vehicles in real life conditions – generated a more critical need to access efficient modeling and simulation tools supporting the analysis of vehicle emissions at any stage of the design cycle. System simulation used to be only deployed for R&D activities and (pre)design purpose. The new regulation context raises additional requirements, and emphasizes the need to work not only on sub-systems, but also to assess the performance of the system as a whole - i.e. to study the pollutant emissions conversion efficiency at vehicle level, with realistic mission profiles.

When working on the concept phase, engineering departments have a large number of technical options available, and need to make the appropriate decisions when it comes to selecting technologies combined within the optimal architectures. This requires fast simulations to quickly assess the potential of after-treatment devices in their vehicles architectures and variants, always trying to cover the wide range of driving cycles and conditions.

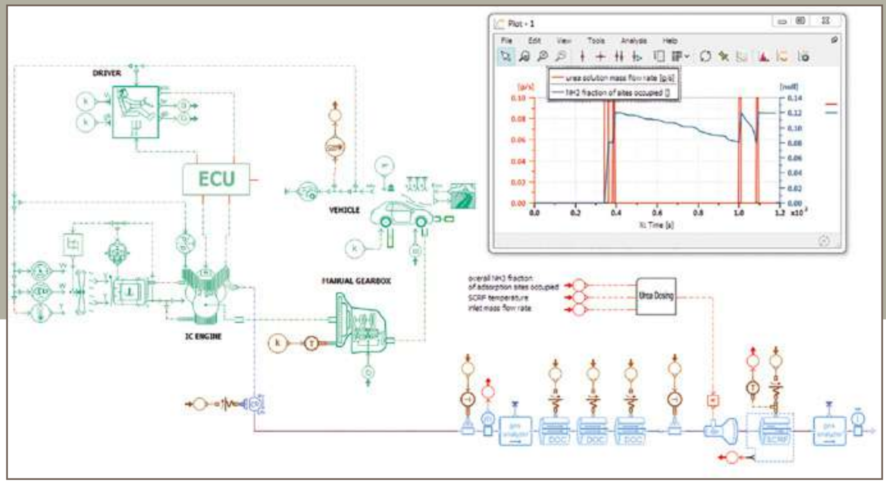
During the design stage, a special focus on the after-treatment control strategies is required, in parallel to hardware detailed analysis at component level. As the exhaust system integrates more and more sensors, actuators and connections with the ECU, plant models are necessary for the development and validation of control strategies using Hardware in the Loop (HiL) environments - which requires models running with fixed step solvers on real time hardware.

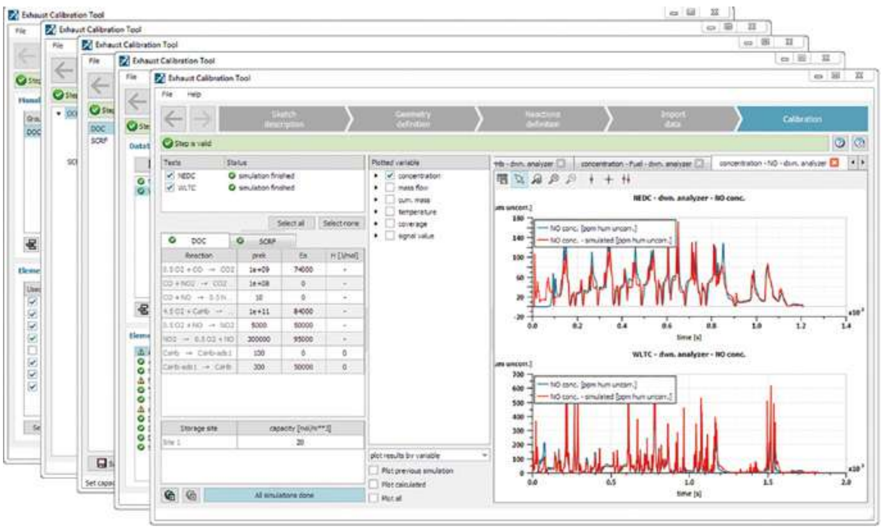
In the later phases of the V-cycle, control calibration tasks are also impacted by the RDE regulations. The calibration tests must migrate from engine dyno and in-vehicle to virtual environments using simulation tools, and cover the new complexity coming with the exploding number of systems and the variety of cycles to be addressed. The change from a calibration workflow based on the optimization of a few engine operation points to a calibration robust enough to tackle real life scenario - i.e. any kind of operation - strongly affects the process with a leap forward in complexity.

Simcenter Amesim™ software answers this growing demand for system simulation involved with the RDE standards requirements.

The after-treatment device (monolith and/or filter) is modeled using a 0D flow approach and thermal components. They can be easily combined to represent a full 1D channel model, to simulate the gradient of temperature along the monolith. This is definitely a "must have" for a fine prediction of the chemistry and the regeneration of filters in particular.

Whereas energy and mass balance equations are used to compute the thermodynamic state of the gas and





wall, a Langmuir-Hinshelwood formalism is used to represent the reaction rate expressions.

Practically, users benefit from multi-level models embedding various physical content, and can enable or disable any sub-models or reaction paths to adapt the model to the physics of their application, including the full details and complexity for an accurate prediction of the reaction mechanism or simplifying the model content for an optimized CPU performance.

As an example, users can activate and model adsorption and desorption phenomena using one or several storage sites. A diffusion model can be added to get more precise results for high flow regimes.

After designing all the after-treatment component models to work with both variable and fixed time step solvers, it is easy to integrate the model later in the design process in real-time targets and drastically reduces simulation times when the step size is increased.

More concretely, common after-treatment modeling approaches find their limits with short fixed steps because of the high dynamics of the reactions to be represented. To answer this issue, we have implemented an advanced reaction rate saturation algorithm in Simcenter Amesim, which guarantees a robust simulation whatever the step size, and ensures a physical consistency to get accurate results. On the other hand, the physical content of the model preserves the consistency of the results and the capability to predict the system performance on extrapolated conditions and driving cycles.

In extreme cases, using a fixed time step solvers of 0.1s, we can execute the simulation of an exhaust after-treatment device over a complete driving cycle in seconds, making it possible to simulate thousands of cycles in a couple of hours.

As a result, Simcenter Amesim enables the engineer to cover the complete V-cycle and can be combined with 3D CAE software for detailed design of catalytic converters. Thanks to a broad range of components offering several modeling options and computational performance, engineers are empowered with a tool providing the right level of model for their application. In addition to its modeling capabilities, Simcenter Amesim also provides tools and methodologies to optimize parameter calibration, reducing the gap toward the adoption of simulation software for after-treatment analysis.

A step-by-step process supports engineers in the definition of their simulation project, thanks to an application-oriented GUI that enables an easy set-up of the monolith or filter geometry and the reaction scheme details, for the loading and pre-processing of the available test data and as a last step for the handling of the tuning parameters. This comprehensive, integrated workflow significantly reduces the effort required for the tuning of the model, and saves incomparable amount of time for the users to focus on their engineering discipline. Those reduced simulation times also allow for the creation of optimization algorithms to automate the definition of chemical scheme parameters. ■

Fresh Air on the High Seas

Predicting exhaust plume dispersion on cruise ships

Modern cruise liners are often called "floating hotels" featuring luxury cabins, a wide range of shopping, dining and entertainment areas, and innovative outdoor swimming and leisure areas. With all of these amenities, passengers expect a luxurious experience. The largest cruise ships can accommodate up to 6,000 passengers and 2,000 staff, the equivalent of a small town floating on the ocean.

While passengers relax and enjoy the various on-ship options, many services are working behind the scenes; these include diesel engines powering the liner, kitchens supplying the food to restaurants, as well as laundry services and incinerators. An unfortunate by-product of these systems are exhaust fumes, which have to be vented to the outside of the ship. If these fumes drift back onto the ship, they can create "smell discomfort" for both passengers and crew. Any area of the ship with unpleasant odors will be avoided or unusable by passengers. The human sense of smell is so sensitive that even concentrations as low as 300 parts per million of diesel fumes will make passengers uncomfortable. Ship owners want to avoid any discomfort or wasted/dead space, but also want to maintain the overall aesthetic and design of the ship.

STX France has extensive experience designing and building cruise ships to meet the highest requirements and standards, and has delivered many vessels operating all over the world. STX France is proud of its heritage of more than 100 years of shipbuilding and of its ability to innovate and create the ultimate luxury experience. Deliveries include Harmony of the Seas – one of the largest cruise ship ever built – which recently received the fuel efficiency award at the Seatrade exhibition. The Mediterranean Shipping Company (MSC) Meraviglia cruise ship makes use of all the latest environmental features developed by STX France in their ECORIZON program.

As part of their drive to create the most energy-efficient and comfortable, best-in-class ships, STX France is increasing its use of computational fluid dynamics (CFD) as part of the design process. CFD has been used systematically for more than ten years to optimize hull geometries and ship-wave interactions. More recently, STX France has started using CFD as a tool to predict exhaust plume dispersion in the vicinity of the superstructure. This method has allowed them to understand the plume behavior in detail and improve the design of funnels and vents to avoid fumes in the passenger decks. STX is using Simcenter STAR-CCM+™ software.

Ship plumes

Gases are vented from the ship at different locations; the exhaust gases are generally warmer than the surrounding air so they rise in a plume away from the vent. The development of this plume is highly dependent on the environmental conditions. Cruise ships run at an average speed of 20 knots, while the prevailing wind direction can vary in both strength and direction. The speed of the exhaust gases at the vent is generally low compared to the external flow, so the plume motion and development is turbulent and highly unsteady. A typical Gaussian plume model will not correctly capture the development of the plume, as it neglects the crosswind contribution to the flow. Instead, it is better to think of the plume as a series of "puffs," or highly unsteady winds moving in three dimensions and growing and dispersing over time.

Plume modeling

To examine the plume dispersion in Simcenter STAR-CCM+, STX France models the entire superstructure of the ship. Each ship has a different

Using Simcenter STAR-CCM+ enables simulation of the plume behavior on the full-scale ship geometry.

three-dimensional computer-aided design (CAD) model, created and maintained by STX France in collaboration with the owner and the architect. This design must fulfill multiple requirements, both aesthetic and practical, and vent placement and design is just one of these. Cruise ship superstructures are complex geometries with detailed features on a range of scales. These features can have a significant impact on the flow patterns around the superstructure that affect the plume dispersion, making it vital to maintain a high level of detail in the CFD model. The Simcenter STAR-CCM+ surface wrapper automatically creates a closed starting surface from the imported CAD geometry. The wrapper does not de-feature the geometry, preserving the full details of the superstructure. Using this makes setting up the model quick and easy, with little manual geometry preparation required. After being imported into Simcenter STAR-CCM+, the full-scale CAD model is wrapped, meshed and set up so that multiple vents are modeled in the same simulation. The same general mesh settings are used for all cases to ensure that the results are consistent and comparable, but additional specific mesh refinements are defined downstream of each vent, with the location depending on the wind direction being tested. The complete mesh has around 35 million cells, surrounding a typical ship size of 350m in length 40m wide and 65m high.

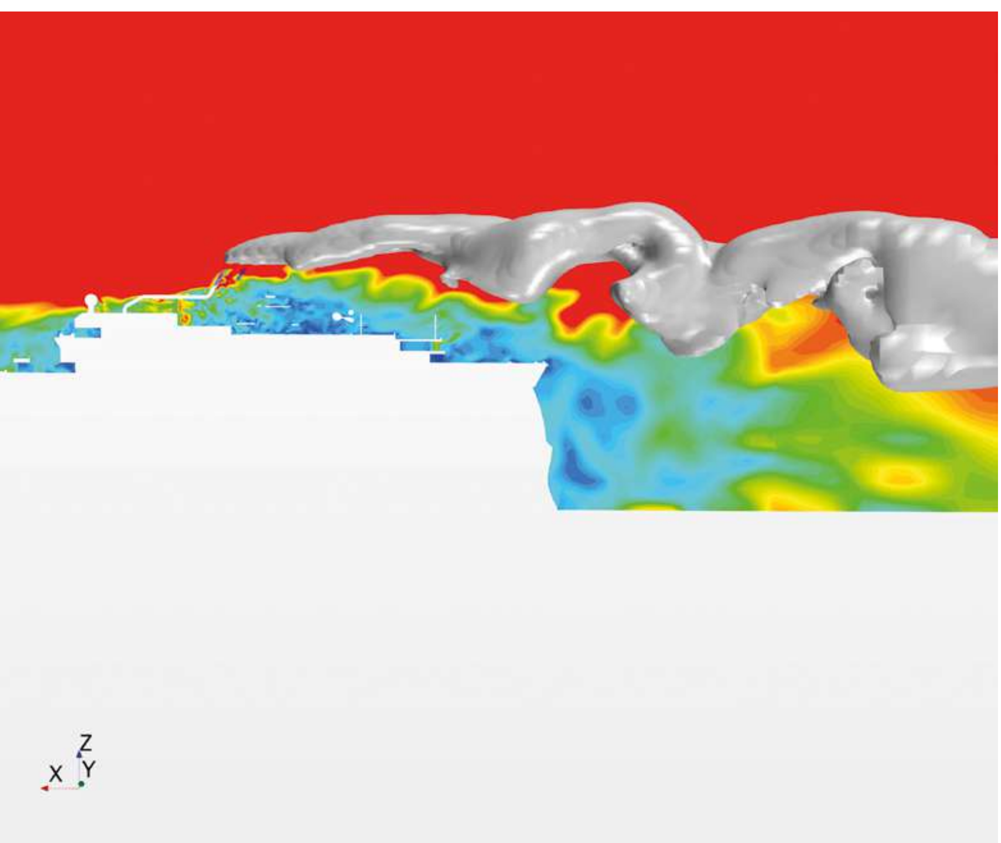


Figure 1: Typical plume dispersion in Simcenter STAR-CCM+, showing wind velocity contours (blue = slowest velocities, red = highest velocities) and an isosurface at a specified smoke mass fraction.

Because of the highly unsteady plume dynamics, a steady Reynolds-Averaged Navier Stokes (RANS) simulation is not suitable. Instead, the detached eddy simulation (DES) hybrid modeling approach is used. The DES approach uses RANS modeling in boundary layers, but switches to a large eddy simulation (LES) model in detached (highly turbulent) flow. This gives higher accuracy in the turbulent contribution to the flow development in the areas needed, for example in the plumes. To track the concentrations of the exhausts, a multi-phase approach is used, with the output from each type of exhaust defined as a different phase. This allows multiple exhaust vents to be analyzed in the same CFD simulation, gaining maximum information from each model run.

In a typical simulation, the velocity field around the ship is highly turbulent, with many vortices forming and shedding from the superstructure.

The plume structure can be visualized as an isosurface of the mass fraction at a specified level. Concentrations of the different exhaust gases can be monitored across the complete model, and any areas that show concentrations higher than the required level can be easily found.

When looking at an initial design, STX France focuses first on the most extreme configurations of wind speed and direction. As the design is refined, a wider range of more detailed configurations is analyzed, covering wind from all directions and at three different strengths. The exhaust speed can also be altered, depending on the ship operation profile. There are more than 25 main outlets on the superstructure of the ship. To reduce the overall number of cases, each simulation can have up to eight exhaust outlets, with four or five different phases being modeled. Even with this, there can be a large number of cases for each design. STX France runs cases in batches on its cluster and has a standard setup for analysis allowing for quick comparison of results.

A standard shipbuilding contract takes up to three years, from signing to delivery of the built ship. The plume analyses are carried out as part of the initial detailed design phase in the first six months of the contract. During this phase, STX France works closely with the ship owner, sharing the results of these CFD simulations and giving feedback on the suggested design. There can be multiple design iterations before a final design is agreed upon; while aesthetics is important, ship owners will not take the risk of bad plume behavior so this feedback and iteration are a vital part of the design process.

Confidence in the results

Prior to using Simcenter STAR-CCM+, STX France performed wind tunnel tests to look at plume dispersion. While it was easy to make quick design alterations in the wind tunnel and repeat results, the scaled model size limited the level of detail that could be captured. This limitation gave some uncertainty on the behavior of the plume, particularly near small spaces on the ship sides. Using Simcenter STAR-CCM+ simulation of the plume behavior on the full-scale ship geometry is possible, preserving all of its detail. The CFD analysis gives a much greater understanding of the full character of the flow, both around the ship and the plume. STX France now routinely uses Simcenter STAR-CCM+ to perform the plume dispersion studies in CFD instead of wind tunnel tests.

Because "smell comfort" is such a subjective measure, it is not easy to validate the CFD studies. STX France has, however, been able to compare its predictions created using Simcenter STAR-CCM+ with readings on a completed ship, and the correlation between the results gives confidence in using CFD in this way.

Conclusion

Ensuring efficient dispersion of exhausts and polluting gases from a cruise ship is critical in providing the experience that cruise passengers expect. A poorly devised funnel design can lead to areas with poor smell comfort, which will be underused by customers. As space is at a premium, ship owners do not want to risk passenger discomfort or create vacant real estate. By running CFD simulations in Simcenter STAR-CCM+, STX France is able to analyze the complete ship geometry at full scale, and therefore predict the plume dispersion and exhaust concentration at any point on the structure. They use this information to provide feedback to ship owners and designers during the early design stage. The process is efficient as multiple exhausts can be modeled in one simulation.

This modeling approach has given a greater understanding of the characteristics of the plumes and their potential interaction with the ship superstructure than was possible before, and has now replaced wind tunnel testing for exhaust dispersal prediction in the design phase. Using Simcenter STAR-CCM+ is helping STX France to develop and build some of the most successful cruise ships sailing today. ■

As part of their drive to create the most energy-efficient and comfortable, best-in-class ships, STX France is increasing its use of computational fluid dynamics as part of the design process.

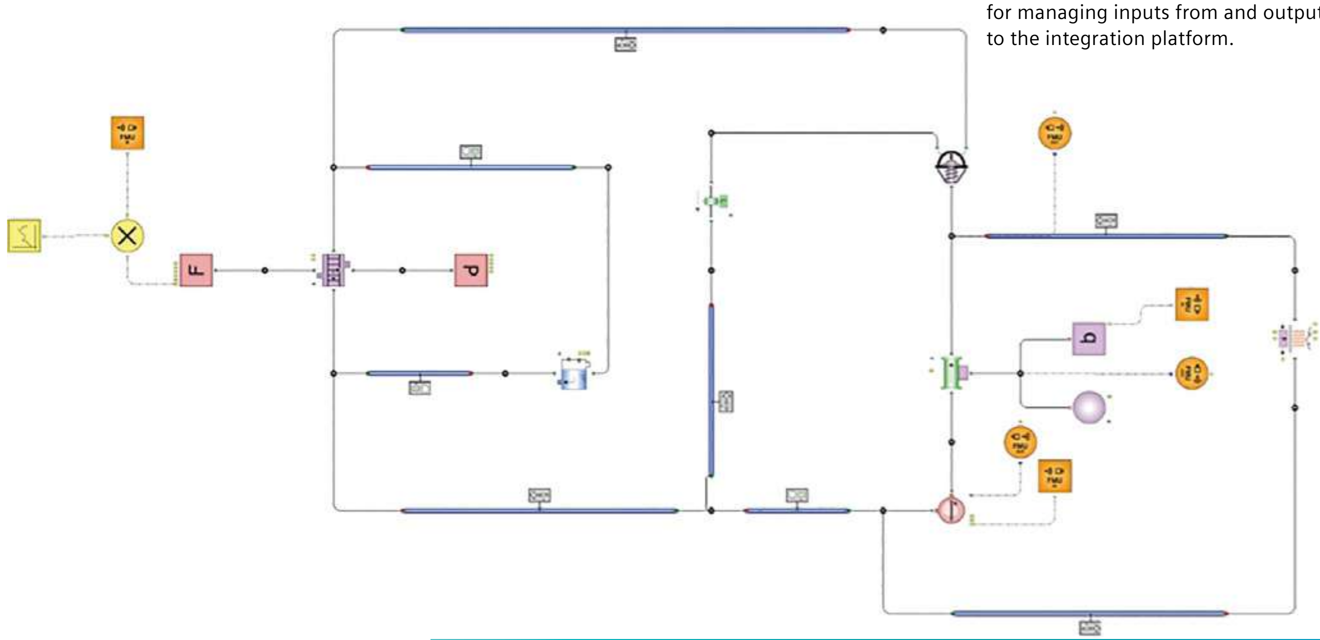


Figure 1: Engine cooling system model in Simcenter Flomaster. Orange components are dedicated controllers for managing inputs from and outputs to the integration platform.

How to...

Model a System-of-Systems through FMI Co-Simulation

By Alberto Deponti, Product Manager, Simcenter Flomaster

Successfully designing complex, sophisticated and efficient systems is not enough to optimize the product. In order to optimize the final product it is necessary to account for system interactions using a System-of-Systems approach early in the design phase and throughout product development. This means that each single system should be modeled using Best-In-Class specialized tools and co-simulation among different tools need to be used to model system interactions.

Functional Mock-up Interface (FMI) is a tool independent standard that specifies an open format for exporting and importing simulation models into a co-simulation framework. FMI is supported by over 100 tools and is used throughout Europe, Asia and North America. An exported model is called Functional Mock-up Unit (FMU). FMU models can be imported in a platform for co-simulation in the wider framework of System-of-Systems analysis where complex system interactions can be properly captured. This opens the door to effective optimization and harmonization of system behaviors. But this is not the only advantage, this approach also allows a consistent reuse of models in different design phases within different company departments which increases the value of model-based design and of the investment in simulation.

Application example

Let us consider for example an internal combustion engine vehicle. To optimize its performance it is necessary to account for the complex interactions of different systems such as the engine and the ECU, the gearbox, the transmission system and the cooling system. This needs to be performed while also considering the vehicle dynamics and a range of different drive cycles.

A comprehensive System-of-System analysis can be set within Simcenter using Simcenter Flomaster™ software and

Simcenter Amesim. Simcenter Flomaster is a vertical solution for accurate modeling of thermo-fluid systems of any size and complexity while Simcenter Amesim is a simulation platform for accurate modeling of mechatronic systems.

The engine cooling system is modeled in Simcenter Flomaster, exported as an FMU and simulated from within Simcenter Amesim, where the interactions among the cooling system, the engine, the ECU, the gearbox and transmission system are accurately modeled.

Preparing an existing Simcenter Flomaster model for export can be achieved in a matter of minutes. Dedicated controllers and gauges are provided for managing inputs from and outputs to the integration platform. Inputs and outputs can vary during a transient co-simulation and effectively account for the complex interactions among the different systems. In addition to this, it is possible to use fixed parameters that will not vary during transient co-simulations but can vary from one simulation to another to account for different operating conditions and/or for different designs.

In this particular case, Simcenter Amesim provides Simcenter Flomaster with computed values of vehicle velocity, engine heat rejection and pump rotational speed. Simcenter Flomaster provides Simcenter Amesim with computed values of engine temperature, fluid temperature downstream of the engine and pump torque.

Importing the FMU into Simcenter Amesim is equally as easy. Once the inputs and outputs of the FMU are connected to the other sub-systems of the Simcenter Amesim model, co-simulations can be run to accurately analyze the interactions among the different sub-systems and to effectively optimize and harmonize system behavior. ■

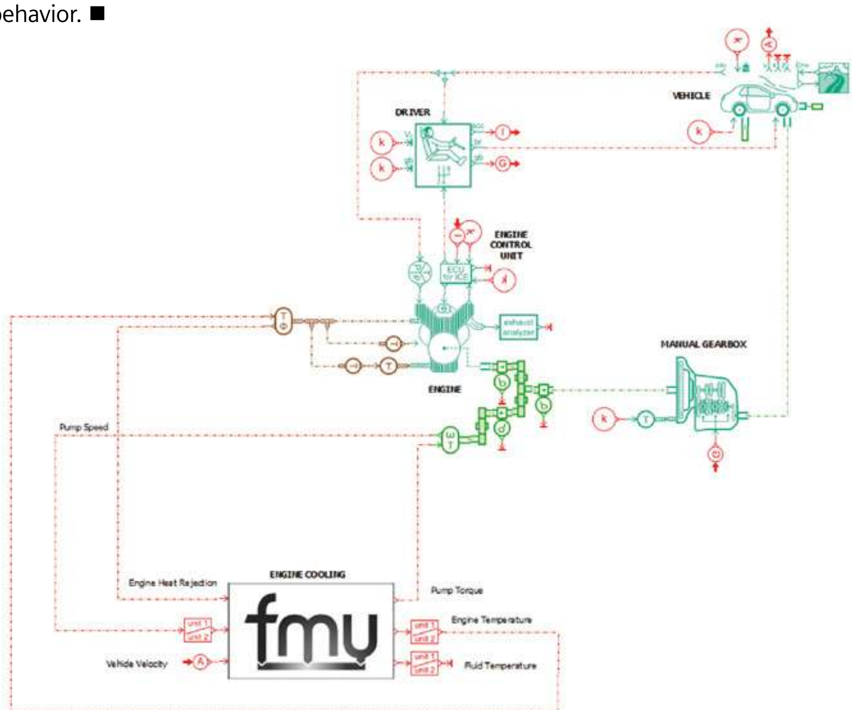


Figure 2: Internal combustion engine vehicle model in Simcenter Amesim. Engine cooling system is modeled with Simcenter Flomaster and integrated into the Amesim model as an FMU.

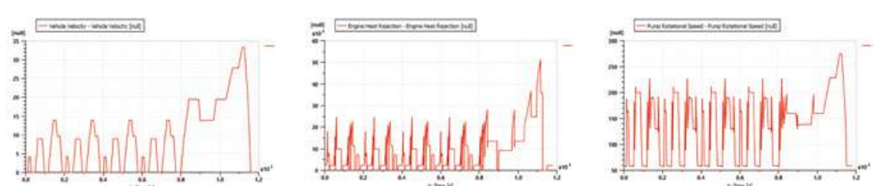


Figure 3: Simcenter Amesim provides Simcenter Flomaster with computed values of vehicle velocity, engine heat rejection and pump rotational speed.

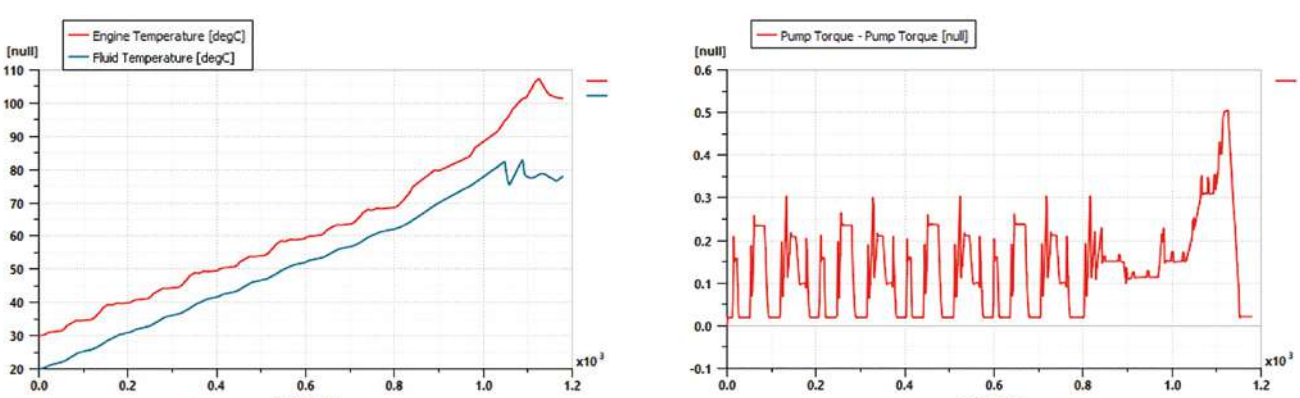


Figure 4: Simcenter Flomaster provides Simcenter Amesim with values of engine temperature, fluid temperature downstream of the engine and pump torque.

Optimization of an Automotive Thermal Management System

By Fabrício Thomaz, FCA Fiat Chrysler Automobiles, Cássio Chamone, Pontifícia Universidade Católica de Minas Gerais, Gustavo Maia, Pontifícia Universidade Católica de Minas Gerais, & Guilherme Tondello, Creative Solutions.

Air pollution and the release of greenhouse gases is a major concern for the automotive industry in Brazil. Following other national governments, in 2012 the Brazilian Federal Government established the "Inovar Auto" program designed to provide tax benefits to car makers who met or exceeded certain efficiency goals. This program recently expired but has been continued with the "Rota 2030" program passed into law in 2018. As a result improved fuel economy and lower vehicle emissions will continue to be a focal point of automotive design in Brazil.

For this reason it is critical to investigate a vehicle's thermal management system. These systems play an important role in maintaining the proper operating temperature of the engine which is crucial to minimizing pollutants. However, these systems also consume a portion of the engine's power and thus need to be designed as efficiently as possible. This need for balancing characteristics of the cooling system means the use of tools such as computer aided engineering (CAE), can help reduce development times and minimize the need for physical prototypes early in the design process.

As shown in Figure 1, decisions made early in the design phase can have the biggest impact on development while doing so at a low cost. If these changes are recognized late the design this will result in much higher costs and possible schedule delays.

One type of software used in cooling system design is 1D computational fluid dynamics (CFD). This type of tool is very helpful because it allows the analysis of many system configurations with low computational effort in a very short amount of time. With Simcenter

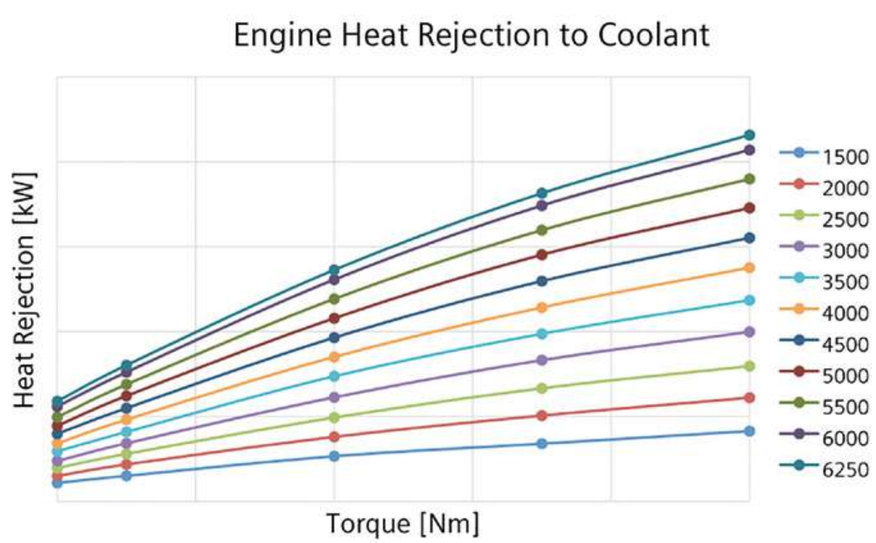


Figure 2: Heat rejection map

Drive Scenarios	Transm.. Gear	Vehicle Speed	Towing	Grade	Amb. Temp.	Coolant limit temp.
A	2nd	65% engine speed on peak power	No	9%	30 °C	101 °C
B	Max speed gear	140 km/h	No	Max	30 °C	90 °C
C	2nd	65% engine speed on peak power	Yes	6%	40 °C	110 °C
D	1st	65% engine speed on peak power	Yes	9%	40 °C	110 °C
E	3rd	89 km/h	Yes	5.6%	38 °C	110 °C
F	Max speed gear	160 km/h	No	0	49 °C	110 °C
G	Engine idling	0 KM/H	No	0	49 °C	110 °C

Table 1. Drive scenarios

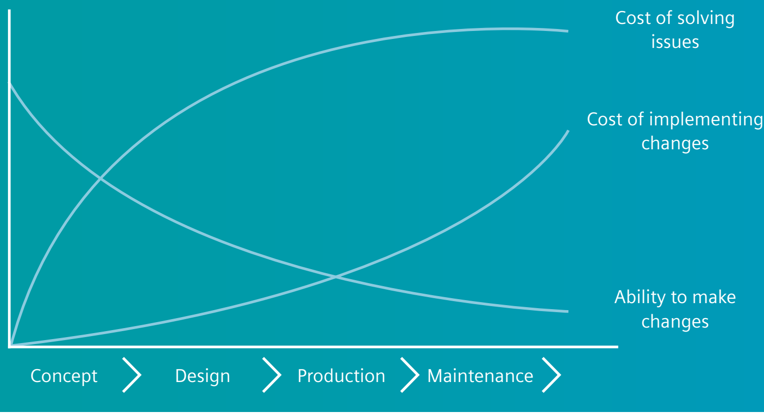


Figure 1: Risk controllability, development costs, and effort to eliminate an error during project development stages

Flomaster there is also is the ability to use Design of Experiments (DOE) to optimize component parameters virtually.

It is important that the cooling system can meet the requirements for extreme vehicle operation which represent the worst case scenarios even though these conditions are encountered less than 1% of the time. To get a good representation of different driving conditions seven scenarios are captured as shown in Table 1.

Cooling system model

Accurately capturing the behavior of the vehicle virtually is often a challenge, for this study four main areas were the focus: engine heat rejection, fan modeling, radiator modeling, and the overall system configuration. The most important of these is properly determining the engine heat rejection since this has a direct impact on the required performance of the thermal management system. The first step was to obtain data for the vehicle, in particular, engine type, weight, gear ratio, coast down, tires, final drive ratio, towing, and payload. This information allowed engine speed and torque to be calculated for each of the test scenarios. Engine speed was determined from the gear ratio, final drive ratio, tires and vehicle speed, while the torque was determined by calculating the resistance to movement using the coast down and calculating the power to overcome obstacles for each test scenario. It is also important to note that when calculating the power to the engine, a drivetrain loss of 8% and an accessory load of 5 hp was added in.

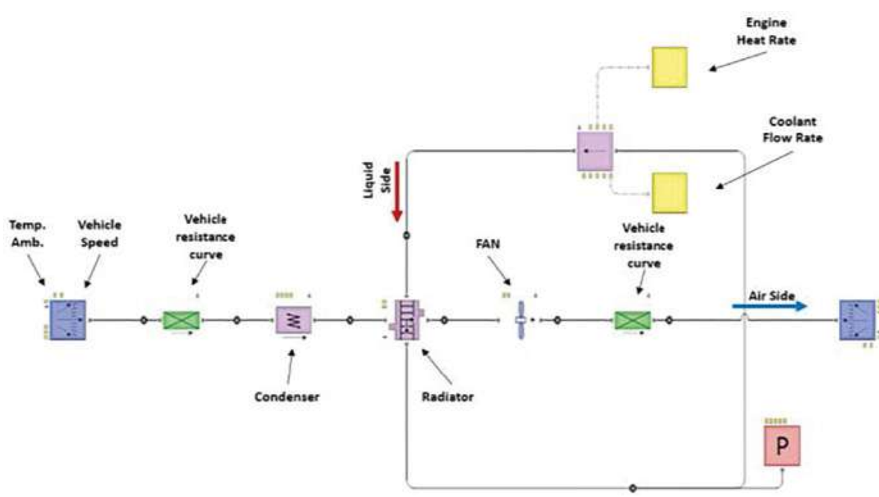


Figure 3: Simcenter Flomaster Cooling Model

Once these values were determined it was very straightforward to look up the heat rejection from the engine thermal characterization map (Figure 2), which was previously available.

The next area to model was the fan and radiator, these were important because they are a major part of removing the waste heat from the engine but also because these are the main components for optimization in this study. Modeling of the fan is straightforward using pressure rise versus volumetric flow rate data along with fan affinity laws. These laws allow the fan to be scaled virtually in the model based on the original data. A similar approach was used for the radiator model with a Nusselt Number versus Reynolds Number for the coolant and airside surface map. Because this map is dimensionless, it allows the test data to be scaled based on size, fluids, and other variations without changing the performance map.

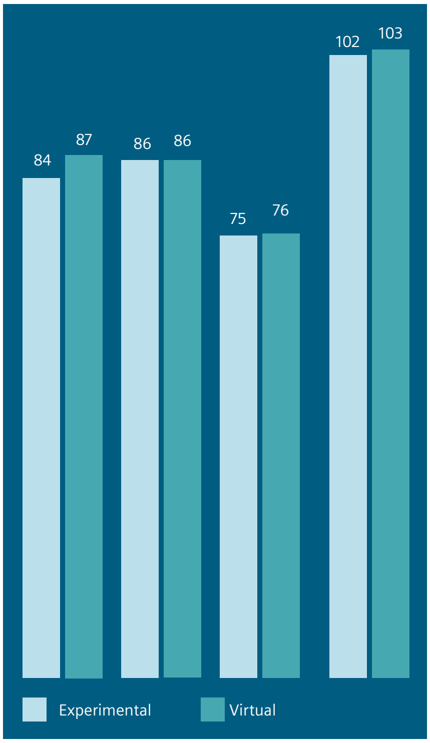


Figure 4: Model validation of temperatures

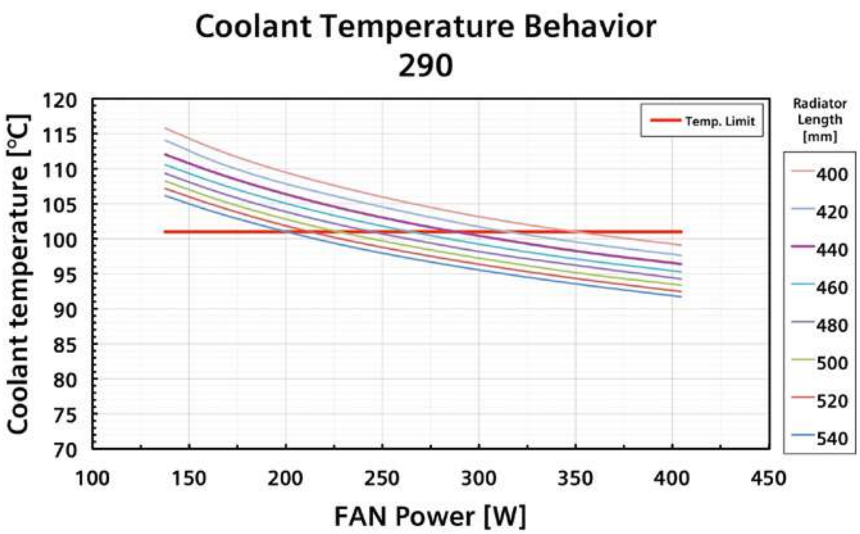


Figure 5: Coolant temperature, A test case

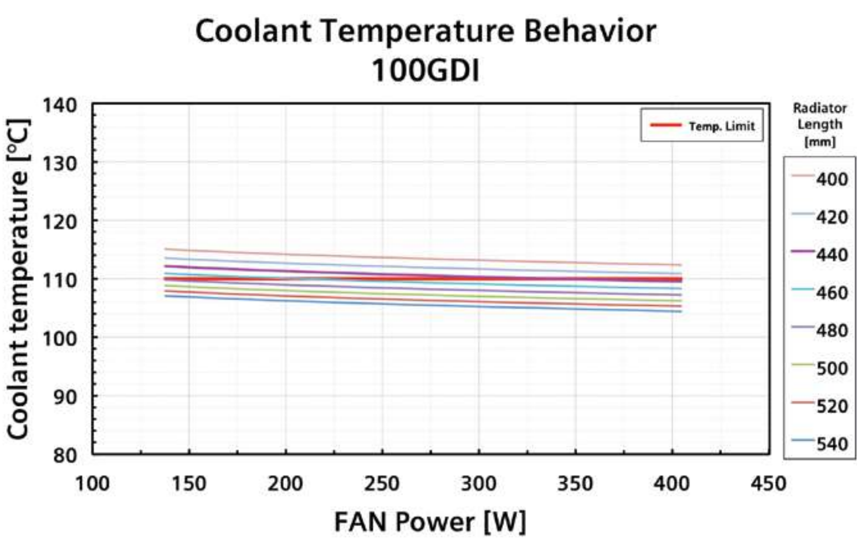


Figure 6: Coolant temperature, drive scenario F

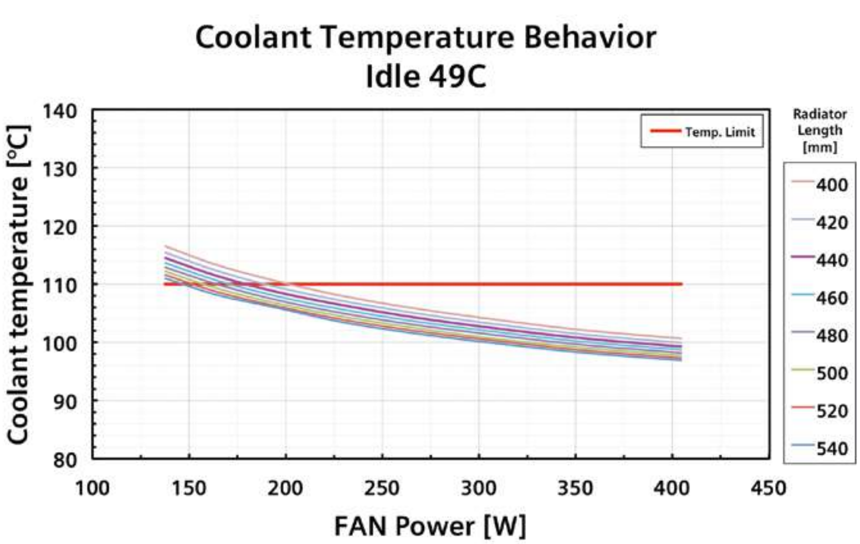


Figure 7: Coolant temperature drive scenario G test case

The final part of the model for the study is the virtual system configuration. Using Simcenter Flomaster, the layout of the cooling system is defined and shown in Figure 3.

All of the necessary data for each component and each scenario was entered into the model with the main output of interest defined as the coolant temperature at the inlet of the radiator. For each driving scenario a Design of Experiment was conducted using the software. It was a full factorial design with respect to the radiator length and the fan power. Each DoE consisted of 64 experiments; with seven different driving scenarios this results in 448 simulation runs in total to determine the optimal radiator length and fan power.

Results

One of the most important things to do when using a virtual model is to verify it against known, accepted data. As shown in Figure 4, five of the test scenarios were compared against real experimental values previously obtained. The requirement was for the model to be within 5°C, and the virtual model succeeded on all five tests with a maximum of 3°C difference.

Focusing on the study of radiator length and fan power, it is necessary to look at each of the different driving scenarios as each needs to perform satisfactorily. As an example Figure 5 shows coolant temperature for drive scenario “A”. In this test, there are several design conditions that could meet the requirements as points shown below the red temperature limit line. Since this is a low vehicle speed test, changing the fan power or the radiator length can have a noticeable effect on the top tank temperature.

This however isn’t always the case as shown in Figures 6 and 7. Figure 6, shows temperatures for drive scenario “F” which is at a much faster vehicle speed test (160 km/h vs. 40 km/h). In this case, the additional airflow increasing the fan power would generate is negligible compared to the ram air already being produced. This means keeping the cooling temperature under the limits will mainly be driven by radiator size. On the other end of the spectrum is drive scenario “G” shown in

Figure 7. Here the airflow generated by the fan is crucial and far outweighs the effect of radiator length.

As mentioned it is critical that all driving conditions are met, Table 2 is a summary of the designs that fail (red), meet within 5°C (yellow), and exceed (green) the requirements.

It is also useful to plot these values as seen in Figure 8.

Conclusions

The results obtained confirm some things we already know: if radiator length is decreased, fan power needs to be increased and vice-versa. Also, results confirm radiator length has more influence at higher speeds, whereas fan power has more influence at lower speeds. The true value of this type of analysis is understanding the sensitivity of each parameter in the final results, and the boundary of viable radiator and fan combination setups.

Ultimately, these results allow engineers to understand the trade-offs between variables and make better design choices amongst the viable cases. Selecting a configuration based on costs, fuel efficiency, reliability, aerodynamics and vehicle performance, etc. ■

Radiator Length [mm]	FAN Power [W]					
	400	350	300	250	200	150
540	OK	OK	OK within tolerance	OK within tolerance	KO	KO
520	OK	OK within tolerance	OK within tolerance	OK within tolerance	KO	KO
500	OK within tolerance	OK within tolerance	OK within tolerance	KO	KO	KO
480	OK within tolerance	OK within tolerance	OK within tolerance	KO	KO	KO
460	OK within tolerance	OK within tolerance	KO	KO	KO	KO
440	OK within tolerance	KO	KO	KO	KO	KO
420	KO	KO	KO	KO	KO	KO
400	KO	KO	KO	KO	KO	KO

OK — OK within tolerance — KO

Table 2: Summary of results

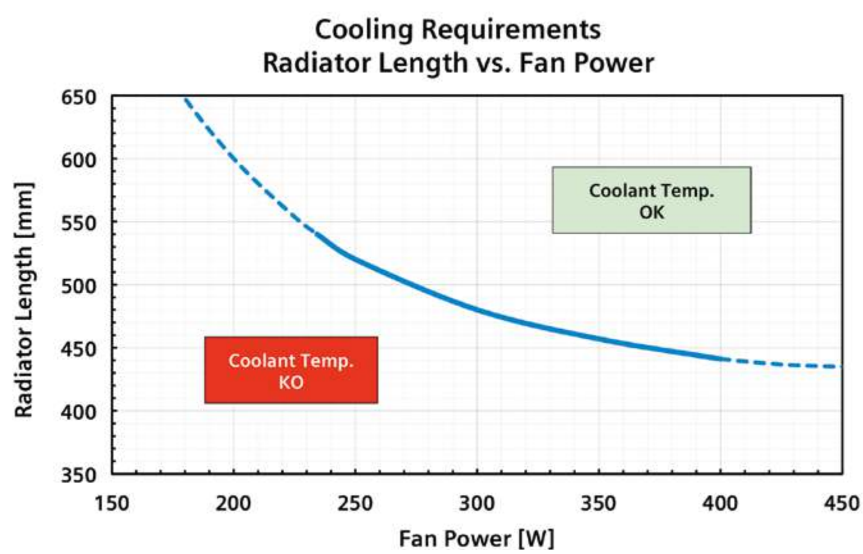


Figure 8: Cooling requirements according to all driving conditions

"Running a Design of Experiments analysis is not always about searching for a single optimal configuration, but sometimes understanding the trade-offs between variables, finding the boundary of viable configurations and making smart decisions. With Simcenter Flomaster all of this can be done very early, even before all geometric information of the system is available"

Fabricio Thomaz
Engineer
FCA Fiat Chrysler Automobiles

Interview

The Route to the Digital Twin

Nicolas Damiani, an expert in simulation and operational analysis in the simulation department at Airbus Helicopters Research and Development in Marignane, France, has spent more than 28 years working in the digital world. Today, he supports the research and development teams at Airbus Helicopters. In this interview Nicolas explains the ever-changing world of simulation and why Airbus Helicopters is well on its way towards developing a digital twin, or virtual iron bird, as experts in the industry like to call it.

Q. Nicolas, what is your role at Airbus Helicopters today?
A. As an expert in digitalization and simulation, I have quite a few roles. I have to keep an eye on the standards and the overall simulation architecture. I also support various project teams when needed and prepare the simulation technologies for the future. Basically, I try to make sure that our overall work in simulation results in successful products – delivering our helicopters on time and within budget that also meet quality requirements.

Q. How has the role of simulation changed at Airbus Helicopters?
A. I have been lucky to witness the evolution of the field over the past 28 years. Starting from simple problem-solving and troubleshooting project work to our groundbreaking model development with Simcenter Amesim, I have seen the simulation side grow brick-by-brick, you might say. Some decades ago, simulation was mainly used in the upper part of the V-cycle for man-machine interfaces and a few research studies. Today, we have a solid simulation strategy and model architecture in place in which simulation plays an enormously important role in our product development. Our simulation products are now part of the test means for performing the verification and certification activities. It is not yet perfect, but it is well on its way.

Q. Why has simulation become so essential?
A. Simulating a helicopter is an immensely complex task. It has even become more complex as the amount of embedded software and number of stakeholders, including suppliers, has drastically increased. Nevertheless, the simulation architecture is constantly evolving as certain systems and subsystem models have gained maturity.

Like other companies, Airbus Helicopters not only uses simulation to iron out development issues, it also plays a vital role in making sure that we can deliver our products on time and within budget. In our industry, reducing the development cycle and detecting design inconsistencies as early as possible have become key factors of success. Simulation is an answer to our time-to-market and design-to-cost constraints.

Q. In your opinion, how mature is your model?
A. Our simulation model as a whole could be considered very precise. Of course, there are some areas that are more mature than others in the overall architecture. When we talk about our simulation model, what people tend to forget is that it is actually a hugely complex model that contains 400-to 500 individual models that work together in real-time and in a validated environment.

Q. That must take an enormous amount of computing power.
A. We are lucky the world of computing is getting faster and faster as well as more efficient economically. When we run the full simulation, we use approximately 24 CPUs. It doesn't take up all of our capacity, but it comes close.

Q. What is a typical trend you see today at Airbus Helicopters?
A. With more precision in our simulation models, we are truly on the right path to predictive engineering in the world of simulation. If you look at this mega-model that we have, composed of 400-to-500 individually validated models, it is a system within

systems. Just think about all the original source code that goes into something like this. It is immense. The source code is validated. It is locked into the model and we don't review it. It is precise without having to question it.

The next challenges for Airbus Helicopters will be our capacity to perform joint development in a simulation environment and to progressively replace physical verification by virtual verification, the simulation being a means of compliance.

Q. Are there specific areas that you are focusing on?
A. Of course, like other industries, we focus on fuel efficiency. In our development process, this happens at a very early stage during the modeling of the hydraulic, fuel and electric subsystems using Simcenter Amesim. To give an example, the actual model physics, be it the pressure or the pump action, are validated in Simcenter Amesim. With this type of validation so early in the process, you can start to answer bigger questions like fuel efficiency using simulation in a predictive fashion. Our system solution for fuel efficiency that we developed with Simcenter Amesim and Simcenter Engineering is certainly beginning to bear fruit.

Q. What are the other ways that you use this model?
A. Our model is critical to confirm the quality of the prototype, what we call Helicopter Zero. Before the first flight we are beginning to use it for supply chain validation. More and more, we are seeing our suppliers involved and using the validated model to create parts and components that are suited for the job.

Q. What do you think you'll be focusing on the next five years?
A. My job is critical considering the enormous complexity of helicopter development. There are still an enormous amount of bad surprises or things that can go wrong. Today, you can't fly without simulation: at the modeling stage, the pilot-in-the-loop stage, the prototyping stage, the validation stage, the flight simulation and the training stage. Simulation is critical to the entire development chain and my main concern in the next years will be to support more engineers as they jump into this new digital way of designing helicopters.

Q. One of the most typical simulation applications that everyone knows is the flight simulator. Is there a connection?
A. What is surprising after all these years is that the answer is 'yes.' In recent years, the simulation world was divided between engineering simulation and training simulation, with limited sharing of models or tools. Basically, a training simulation (like a flight simulator) was developed after the helicopter certification (despite the customer expectation to get trained before receiving their first helicopters) by re-using a limited part of the engineering simulation. Nowadays, the same simulation product is continuously improved from preliminary design phase up to the certification phase. At any moment in time, we can derive from our engineering simulation product a training simulation product (compatible with the helicopter delivery).

Q. So your development model can easily be linked to the final product or the flight simulator. In other words, the digital twin can be recycled for other applications?
A. The beauty of simulation is that it can be coupled with various other types of simulation. All you need is the code. This means that you can link the virtual flight simulator as well as pilot-in-the-loop development to practically any application. This might impact flight tests, other co- simulations like maintenance schedules, and even other in-house development programs. The possibilities are endless.

Q. Will there eventually be a truly virtual test bench?
A. We place an enormous amount of importance on virtual testing at the moment. If you look at how we work with Helicopter Zero and the physical iron bird, this has totally changed! Today, the heart of testing is Helicopter Zero; tomorrow it will be simulation. By using real-time validated models and virtual iron bird (VIB) solutions, we can certainly engineer an extremely accurate helicopter solely in the virtual world.

Q. What does the future of helicopter aviation look like?
A. Lots of technology, but as experts, it is up to us to make the right choices. There is a lot of technology out there and we need to be vigilant and make the right choices as experts when it comes to smart technology and artificial intelligence. There is a side of smart technology in which it becomes artificial intelligence and where machines will really be able to learn behavior. This cannot be taken lightly. Once we open that door, there is no going back. ■

"Our system solution for fuel efficiency that we developed with Simcenter Amesim and Simcenter Engineering is certainly beginning to bear fruit."

Nicolas Damiani
Expert in simulation and operational analysis, Simulation Department, Airbus Helicopters Research and Development

Certifiably Cheaper

Aerospace engineering services company uses Simcenter STAR-CCM+ to reduce aircraft certification costs

"Our goal was to have the most boring flight test program we've ever seen," says Wayne Tygert, Chief Engineer, Boeing, in describing one of the 787-10 Dreamliner test flight programs used for certification.

Boring usually isn't the first word that comes to mind when describing something that took 900 hours across three test aircrafts, thousands of regulations, upwards of 4,000 documents and millions of dollars. And what is the 787-10? A simple extension in the midsection of the already certified 787-9 to accommodate 40 more passengers. The original 787 Dreamliner took eight years from application to certification for the new design, clocking 4,645 flight hours in flight testing, more than 200,000 hours in Federal Aviation Administration (FAA) experts' time and a much higher certification cost.

Getting an aircraft certified, whether new or modified, is a long, expensive and bureaucratic process, albeit one that has led to the safest mode of transportation. From the largest aircraft in history to small two-seaters made of steel and fabric, every plane needs to prove airworthiness and compliance and be certified by regulatory authorities before operation.

The massive cost of certification

Certification is estimated to cost $1 million for a primary category aircraft (three seats or less), $25 million for a general aviation aircraft and upwards of $100 million for a commercial aircraft. Certification costs and delays can run into millions of dollars, sometimes burning up as much currency as it costs to develop the aircraft. This process can often decide profit or loss.

Program delays, missed delivery dates, cost overruns and safety issues due to designs not meeting certification requirements and requiring expensive redesign and flight testing frequently occur. How do companies reduce certification cost and time? Can you reduce expensive testing while still proving airworthiness?

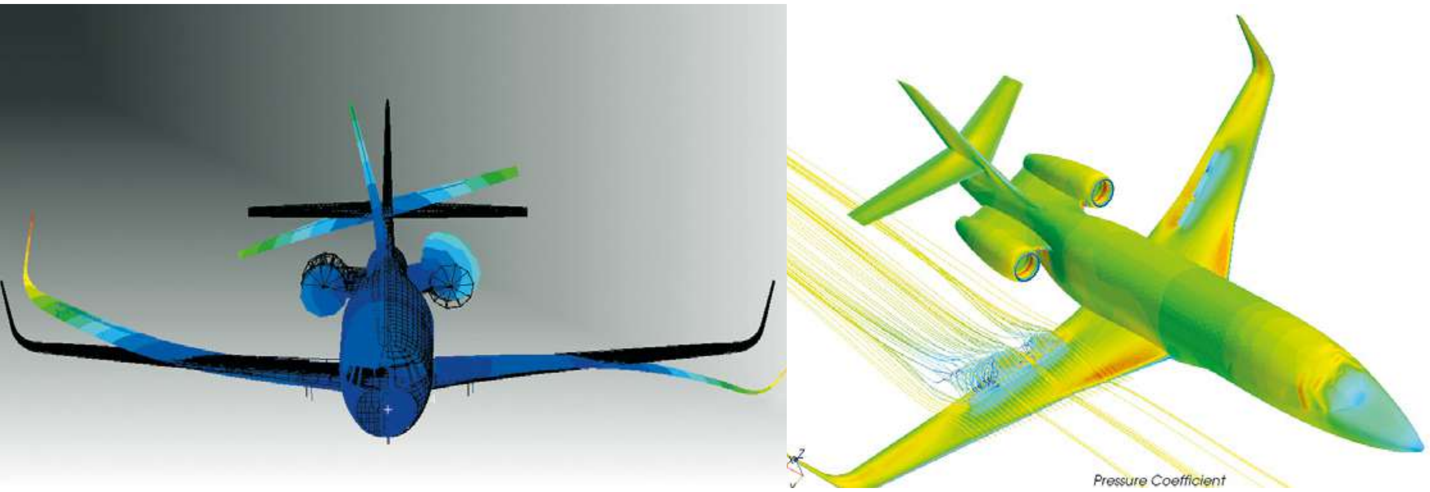


Figure 1: For airplanes where structural flexibility is important, a coupled aerodynamic and structural certification-by-analysis approach is needed.

With 45 years of combined experience in aircraft design, development and certification, Robert Lind and Andrew McComas of TLG Aerospace are no strangers to these challenges. Occupying one floor of an unassuming, six-storey building on Seattle's Lake Union, their modest office belies the experience and expertize that has a client list reading like a who's who of modern day aviation. TLG Aerospace has helped numerous customers receive FAA certification in the U.S. at a low cost and in a short time, something they have achieved with surprising efficiency.

Certification by analysis

FAA aircraft certification involves three stages: design certification, production certification (PC) and airworthiness certification (AC). The design certification stage involves the approval of design safety, operability and durability, with type certificate (TC) for new designs and supplemental type certificate (STC) for modified designs. The PC stage gives approval to manufacture parts, components and systems, while the AC stage gives approval to operate the aircraft.

The process is similar for other worldwide regulatory agencies. The FAA requires certification by test or by analysis validated by test. The industry calls it certification by analysis (CBA). These analyses are made using a full vehicle model that is validated by a flight test over a specified range of the flight envelope, as agreed upon in advance by the certification authority. The full vehicle model includes:

- Aerodynamics: A combination of computational fluid dynamics (CFD), low order methods, wind tunnel and handbook analysis validated by pressure and strain measurements in flight test;
- Structures: Finite element analysis (FEA) and handbook calculations validated by ground vibration testing (GVT) and static load in ground test;
- Mass properties: Computer-aided design (CAD) and weights bookkeeping validated by weighing; and
- Flight controls: Laws of flight control validated by integrated simulation and flight test.

The integrated full-vehicle model is ultimately validated by the flight test and must be shown to be accurate or conservative. The certification authorities ensure that the analysis will yield a safe result. The original equipment manufacturer (OEM) is typically concerned about limiting conservatism to avoid excess weight and missed performance.

The TLG Aerospace approach to certification with CFD

"What has changed is the balance between how much analysis you can do and how much you can use in the certification process," says Lind, Director of Engineering, FAA flight analyst designated engineering representative (DER), FAA flutter DER, TLG Aerospace. "This is a really exciting development in my 30 years in the industry. As CFD codes and computers have become more capable, we can certify faster and cheaper."

> *"Simcenter STAR-CCM+ runs robustly, accurately and repeatedly with simple processes and best practices."*
>
> Andrew McComas
> Engineering Manager and Aerodynamicist
> TLG Aerospace

Figure 2: Certified with TLG Aerospace – (Left) 737NG Split Scimitar Winglets (Right) Falcon 2000 business jet.

Most of Lind's work involves getting customers to type certification with analysis. As one of TLG Aerospace's four resident DERs, he can sign for certain certification functions on behalf of the FAA. TLG Aerospace uses Simcenter STAR-CCM+™ software from Siemens PLM Software for CFD analysis and MSC Nastran® software for FEA to develop full-vehicle certification models for loads, flutter and handling qualities, modeled appropriately for the entire flight envelope.

Andrew McComas, Engineering Manager and Aerodynamicist, TLG Aerospace, notes, "We utilize Simcenter STAR-CCM+ in a certification environment which is different from design. There is a great role for CFD in the certification process. We don't use CFD to get an answer that the FAA signs off on. We use CFD to build a full-scale aero/structure/controls model so we can simulate vehicle response and produce loading and handling information."

To certify a new aircraft, an aerodynamic database is required. To build the entire analysis database would require data for hundreds of thousands of conditions to be available in a short amount of time. The aerodynamic properties of the vehicle are calculated at design and at flight envelope extremes using CFD. The CFD results are mapped to a reduced-order aerodynamic model within the aeroelastic process. TLG Aerospace calibrates the aeroelastic model to develop full-vehicle aeroelastic solutions that are underpinned by the rigid CFD. The final aeroelastic model will reproduce full-vehicle integrated and distributed aerodynamics in rigid mode and yield a converged aeroelastic solution in seconds.

The predictions are now in place to show regulations are met at certain conditions. Flight testing then validates the analysis models. This validation may be limited to something less than the full flight envelope to reduce risk for in-flight testing. Once validated, it can be used to show compliance at other flight conditions. Having a high-fidelity pre-flight test model significantly reduces the amount of required post-flight test model adjustments and calibrations.

Reducing certification cost with Simcenter STAR-CCM+

McComas credits Simcenter STAR-CCM+ and Amazon Web Services (AWS) for the breakthrough in certification cost reduction.

"Simcenter STAR-CCM+ runs robustly, accurately and repeatedly with simple processes and best practices," says McComas. "That has given companies confidence that the code can be used as a source for aero database generation. Elastic computing from AWS, with Siemens' power-on-demand licensing, helps run multiple simulations on multiple compute clusters simultaneously on the cloud in a secure way. If we did not have the POD licensing model, we wouldn't have the capability to take full advantage of elastic computing resources and would incur the large cost of annual licenses."

In short, the entire aero database is built in a shorter time with cost-effective licensing. The added benefit? Reducing wind tunnel tests.

CFD or wind tunnel? The answer is both. With experience in over 100 wind tunnel test campaigns at low and high speeds, TLG Aerospace possesses a significant experimental background in testing. Has wind tunnel testing fallen out of favor then? Not at all.

In common parlance, wind tunnel is still king – but a king who is now increasingly delegating a fair share of royal duties to the trusted advisory council of CFD. Wind tunnel tests are still used for aero database development for new aircraft configurations. However, CFD is supplementing tests at some conditions and replacing testing at others, leading to huge savings. Figure 4 is a notional comparison of legacy CFD codes and Simcenter STAR-CCM+ compared to wind tunnel testing as seen by TLG Aerospace. For a relatively minimal investment, Simcenter STAR-CCM+ can reduce and replace some testing requirements. Considering the usage rate and model cost for wind tunnel tests, this can translate into significant time and cost savings.

Wind tunnel testing is still best suited for incipient separation regimes like high-angle of attack and sideslip handling analysis.

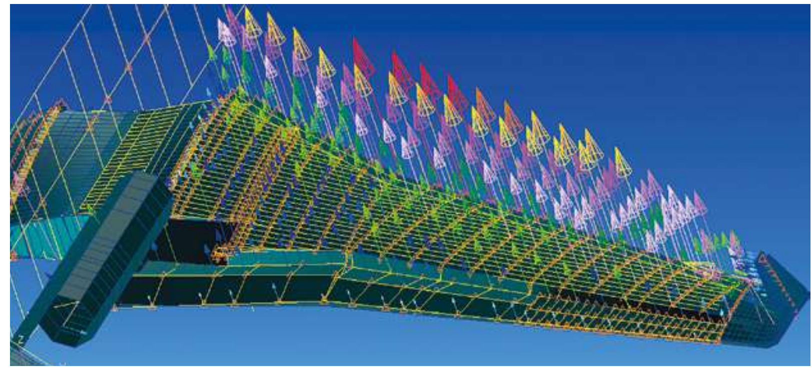


Figure 3: Certification by analysis at TLG starting with the aerodynamic database generation with Simcenter STAR-CCM+,mapping results to NASTRAN with DLM and the final aeroelastic model in NASTRAN

CFD works best for moderate angles of attack and detailed flow-field investigations.

“Large databases can be run in Simcenter STAR-CCM+ today at a fraction of the cost and schedule of legacy methods and wind tunnel testing,” says McComas. “That was not possible only a decade ago.”

With elastic computing and cloud licensing, there is no technical limitation to running large numbers of CFD cases simultaneously. TLG Aerospace is also able to regularly run large, fully detailed simulations, with most models running in under an hour, no matter their size, something which was not previously possible.

Using elastic computing with AWS, TLG Aerospace saves 75% of the total cost per CFD simulation. The technology behind this is the Amazon EC2 Spot Instances, an Amazon offering to utilize unused computing capacity on the AWS cloud at steep discounts.

CFD is not a one-trick pony for certification

Modifications and additions to existing type certified aircraft will affect regulations. Taking older airframes to the limits of the flight envelope with new modifications is hazardous, expensive and time-consuming. The FAA allows CFD as a means to show the compliance of the original aircraft hasn’t changed due to modifications. Companies like TLG Aerospace have adopted this wholeheartedly, utilizing CFD to generate supporting data and arguments to show compliance.

“In the past, engineers had to go to test without any question for any modifications done. Now CFD provides the data to try and eliminate some test requirements,” says McComas.

Other CFD applications in certification include pressure loading on secondary structures, fairings, antennas and radomes, ice accretion, air-data system location, internal flows, winglets and more.

Here’s a scenario: Imagine a new radome is fixed onto the aircraft. To comply with regulations, the manufacturer now has to prove that if the structure comes off the airplane, it will separate safely without impact. Good luck breaking off a radome in flight test! Similar challenges exist in proving icing on the new structure doesn’t affect compliance and operational safety.

“The only feasible option here is to use a validated analysis to show the structure meets safe separation criteria,” says McComas. “For TLG Aerospace, Simcenter STAR-CCM+ has all the tools built in to do these calculations without other third-party software.”

In addition, the onslaught of new, innovative aircraft such as drones and air taxis, military aircraft, the born-again supersonics and others will also benefit from certification by analysis.

The ‘C’ in CFD stands for certification

“Simcenter STAR-CCM+ has contributed

“For TLG Aerospace, Simcenter STAR-CCM+ has all the tools built in to do these calculations without other third-party software.”

Andrew McComas
Engineering Manager and Aerodynamicist
TLG Aerospace

	Model costs	Usage rate (per hour)	Engineers (per hour)	Productivity (conditions/day)
Low speed wind tunnel	$165k	$600	X2	800
High speed wind tunnel	$315k	$4500	X3	600
Legacy CFD	$15k	$250	X1	20
Simcenter STAR CCM+	$15k	$100	X1	80

Figure 4: A notional comparison of wind tunnel versus CFD as seen by TLG Aerospace.

to the receipt of numerous FAA-approved certificates," says McComas. "It has a role in every single certification program at TLG Aerospace."

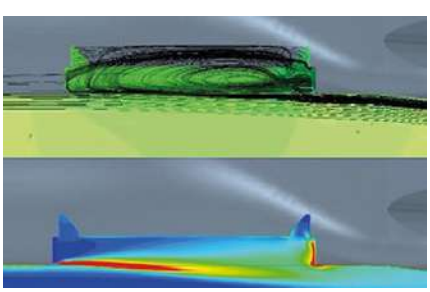
It is unlikely that CFD will ever completely replace wind tunnel testing as computers, codes and licensing continually evolve. Nevertheless, the role of CFD in certification will only increase over time to supplement and complement flight testing.

For now, companies like TLG Aerospace have found a reliable workhorse in CFD for certification, one that can do the heavy lifting of proving compliance at the extremes of flight envelopes, reduce the number of flight test conditions, enable tests to lower loads and predict potential testing hazards. Flexible licensing and elastic computing are further solidifying the case for certification by analysis.

"We can now bid on projects that have greater scope, be more competitive, pass on the savings to our customers and do much more with our dollar," says McComas. ■

"Simcenter STAR-CCM+ has contributed to the receipt of numerous FAA-approved certificates."

Andrew McComas
Engineering Manager and Aerodynamicist
TLG Aerospace



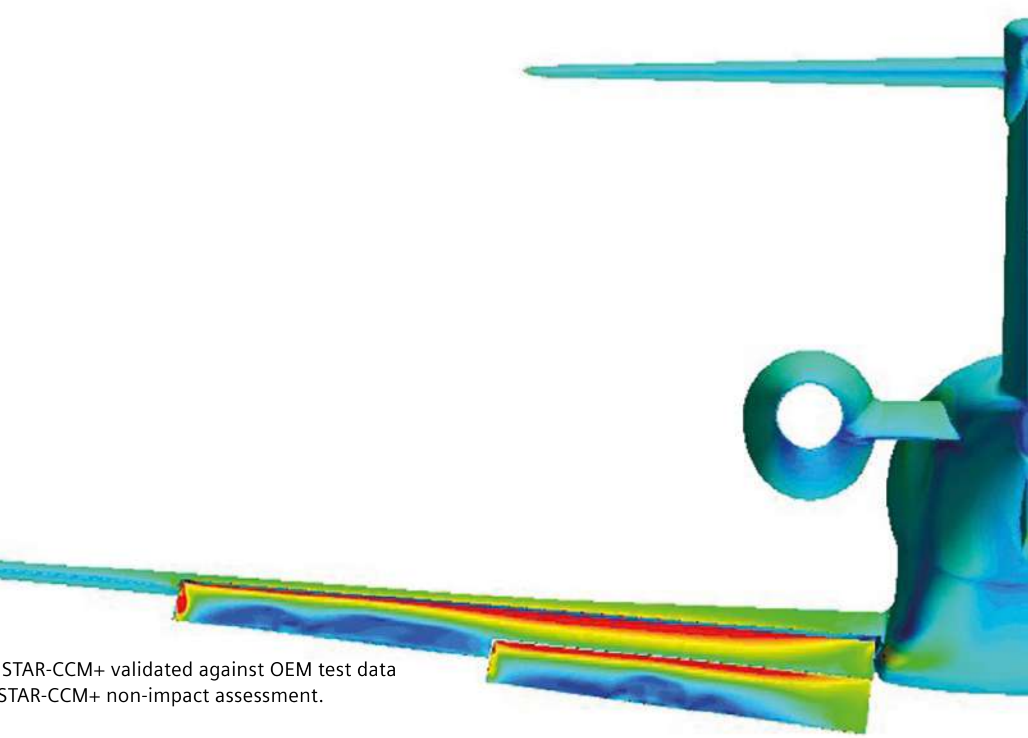


Figure 5: (Left) Flap effectiveness in Simcenter STAR-CCM+ validated against OEM test data (Right) Cavity resonance analysis in Simcenter STAR-CCM+ non-impact assessment.

Avoiding the next major Power Black-Out with Digital Twins

By Hilary Chisepo, MSc(Eng), PhD(Eng) candidate, University of Cape Town, South Africa

Power transformers in bulk transmission networks are vulnerable to the adverse effects of geomagnetically induced currents. Power system instability, major blackouts and overheating of transformers are the main concerns for utilities. To understand the mechanisms that bring about the overheating, a reliable FEM modeling approach needs to be verified with physical measurement data, but data from important large power transformers are seldom available. Model laboratory transformers can be used to investigate the electrical and magnetic repose under simultaneous 50/60 Hz and GIC (quasi-dc) excitation.

Geomagnetically induced currents (GICs) are caused by abnormal space weather that occurs periodically and is known to have adverse effects on power systems. These GICs, having frequencies in the milli-Hertz range that are quasi-DC relative to the power frequency, enter long transmission networks through grounded power transformer neutrals. Power transformers exposed to GICs experience partial half-cycle saturation due to the DC offset of the normal AC excitation. This partial saturation causes the generation of undesirable even and odd harmonics, significant draw of reactive (non-active) power, voltage collapse and excessive stray flux which contributes to overheating inside the transformer.

How do GICs cause overheating of power transformers? The active and structural metallic parts simultaneously experience excessive stray flux that may generate hot spots in the windings, tank and core, leading to the onset of insulation degradation, gassing and in some cases, immediate failure.

GIC-related failures in the Southern African network showed that thermal damage can occur in power transformers even at quite low levels of GIC. As a result, South Africa's power utility, Eskom, now includes DC withstand capability in their power transformer procurement specifications, and so do utilities in many other countries.

Factory verification testing with DC is not viable from a financial and laboratory power capacity point of view and so very few practical results are available e.g. single phase transformer GIC test performed by Siemens, Austria [1]. Approximations of the response of power transformers can be achieved through laboratory testing of smaller transformers and comparing measurements with various modeling techniques, but care is needed when fabricating the test transformers to achieve representative results and reliable findings.

Testing of FEM modeling in Simcenter MAGNET Software

Laboratory tests were performed in a previous study as part of an MSc(Eng) research project at the University of Cape Town [2]. This study investigated the response of transformers of different core structures with GIC emulated in a

Figure 1. Bench-scale transformers single-phase shell type (left) three-phase five limb (centre) and three-phase three limb (right).

laboratory. Numerous physical results involving power, reactive power, and harmonics were recorded under GIC-like conditions. Figure 1 shows the bench-scale transformers used in the laboratory testing. With future studies for power transformers in mind, the same laboratory testing protocol was applied, as far as the FEM environment allowed, in Simcenter MAGNET™ Software to test the validity of the simulation modeling. Figure 2 shows a FEM model of the single-phase shell-type transformer. Simulations were performed in the transient domain that allows for multiple excitation and generation of voltage and currents with different frequency components in the same signal (harmonics). A coupled field-circuit approach was deployed whereby all the simultaneous AC and GIC excitations were triggered in the circuit domain. Analysis of the flux distributions to verify the effect of saturating the core was performed in the FEM domain and the instantaneous voltage and current waveforms were recorded and post-processed in RMS voltage and current calculations, and average power and reactive power calculations. Initially, the FEM model was tested under open circuit conditions with AC excitation only at the nominal voltage for comparison with the measured data. For the single-phase shell type model, the measured magnetizing current and open circuit power (core loss) were 55 mA and 3.2 W, respectively. The 3D solution yielded 41 mA and 3.4 W for the magnetizing current and core loss, respectively. The same protocol was applied using the other two multi-limb core structures and showed that while the core loss calculation (time-averaged hysteresis loss) was fairly accurate against the measured data, the magnetizing currents were consistently lower in the FEM.

Validation of simulations

The key parameters used to validate the FEM models with simultaneous AC and GIC/DC components were reactive power, waveform distortion (harmonics), and time response. Figure 3 shows good correlation between measurement and simulation data for reactive power and increasing levels of DC. The generation of even and odd harmonics in power systems is a typical marker for a partially

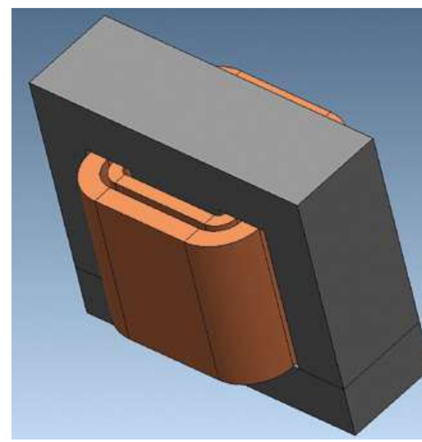


Figure 2. Full 3D FEM model of the single-phase shell type bench transformer

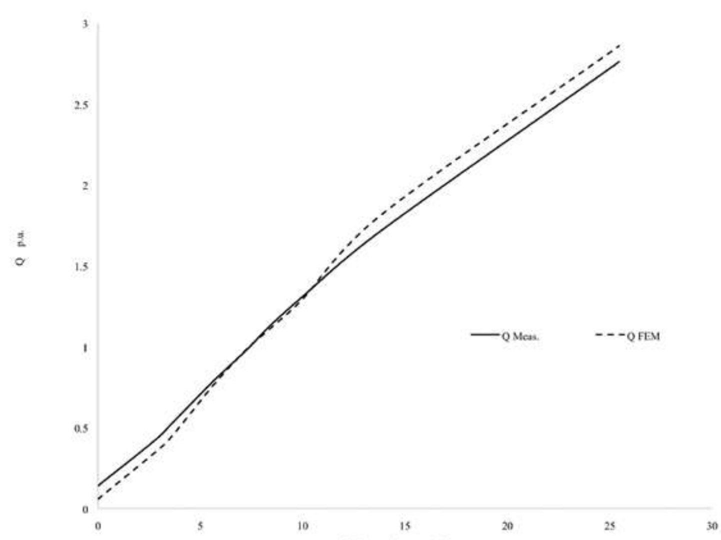


Figure 3. Measured and simulated reactive rower Q in per unit (p.u.) of the VA rating vs. DC in p.u. of the transformer magnetizing current (Imag) for 1p3L laboratory transformer.

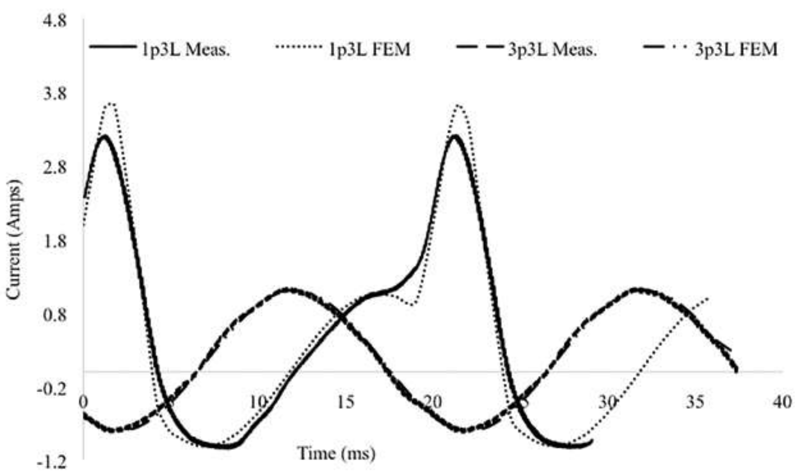


Figure 4. Primary current waveforms for 1p3L and 3p3L test transformers with 550 mA DC and 160 mA DC per phase respectively

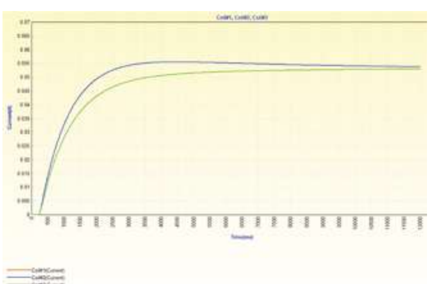


Figure 5. DC time response of the 3p5L model transformer with 160 mA in the neutral

saturated transformers operating under the influence of GIC or simultaneous AC-DC excitation. Figure 4 shows the measured and simulated current waveforms for the single-phase and three-phase three limb transformers. As expected, the three-phase three-limb (3p3L) bench-scale transformer (without a tank) did not generate harmonics with various levels of DC, because a zero sequence path to a tank is necessary for DC flux to flow in and out of this particular core structure, compared with the single phase and the five limb core types that have 'free' return limbs to complete the DC flux return path [3, 4]. The single-phase three limb (1p3L) FEM model generated multiple even and odd harmonics matching the measured distorted current waveform quite closely.

An adjacent study introduced the concept of transformer time response in the calculations of GIC in the Southern African networks [5]. The time response of a FEM model of a three-phase five limb (3p5L) transformer with DC was also tested against laboratory observations, yielding consistent results. Figure 5 shows that three phase line currents are not equal in all the phases during the transient period upon DC injection and that they reach steady state after 12 s. This is consistent, not only with the times recorded in the laboratory, but also confirms the idea that the transformer time constant is a function of the magnitude of the GIC [5]. The FEM analysis, therefore, validates the improved GIC calculation approach that includes the time response of power transformers.

Having tested FEM transformer-GIC models in Simcenter MAGNET and achieved satisfactory correlation with measured recordings (apart from the inexplicable underestimated no load magnetizing current when compared against measurements), the next phase of the research involved the modeling of practical transformers relevant to the manufacturing and utility industries. The main issue with FEM modeling in transformer research with AC-DC excitation is that "Unfortunately, no measurement results are available to validate the [3D FEM] analysis," [6]. This work, therefore, involved exhaustive measurements on larger scale single-phase four limb (1p4L) transformers to investigate key parameters for topological and FEM modeling.

Explicitly modeling individual (0.25-0.3 mm) laminations of transformer cores often results in FEM calculations failing to converge because of excessively large mesh structures [7]. In the context of AC and DC, this problem was overcome using an approach that models laminations explicitly only close to the core surface with the rest of the core treated as solid but without sacrificing the air gap details at the transformer joints with the appropriate boundary conditions [8]. Figures 6 and 7 (1/8th symmetric model of a 1p4L test transformer) show the difference in flux distribution when more detail is added to the core surface and core joints, compared with a solid core. The application of these methods that include air gap details at the joints resulted in more accurate estimations of the no load magnetizing currents and also a better FEM interpretation of the flux distributions with AC only and with AC and DC. The model shown in Figure 6 (right) and zoomed in for Figure 7 gave the results that are closest to all the physical measurement data, including reactive power, "terminal saturation inductance", and even the leakage flux distribution measured with search coils in the air spaces. The terminal saturation inductance is a key parameter in testing topological transformer models for GIC and slow transients, but it cannot be measured readily in any transformer factory. Without the validated FEM or special measurement protocols, this parameter is often wrongly approximated in the literature leading to erroneous results.

Conclusions

The combination of practical testing guided by preliminary FEM simulations yielded some important results regarding the flux distributions under simultaneous AC-DC excitation. This resulted in the submission of a paper at the Advanced Research Workshop on Transformers (ARWtr2016, Spain) which led to a journal publication [8]. The main finding of the study was the importance of details in the transformer core joints for better representation of practical transformers both in the FEM and in topologically derived models. The approach allows for better modeling of stray flux when designing for GIC. This is different from the conventional modeling in industry that assumes solid joints for electromagnetic devices under normal operating conditions. The assumption of solid joints in the FEM model resulted in the underestimation of the no-load currents in the transient calculations performed in the differential core structure bench-scale FEM analysis.

References

[1] J. Raith and S. Ausserhofer, "GIC Strength verification of power transformers in a high voltage laboratory," in GIC Workshop, University of Cape Town, South Africa, 2014.

[2] H. Chisepo, "The response of transformers to geomagneticaly induced-like currents," MSc(Eng) Dissertation, University of Cape Town, South Africa, http://hdl.handle.net/11427/8685, 2014.

[3] P. R. Price, "Geomagnetically induced current effect on transformers," IEEE Transactions on Power Delivery, vol. 8, no. 3, pp. 1002-1008, October 2002.

[4] R. Girgis and K. Vedante, "Effects of Geomagnetically Induced Currents on Power Transformers and Power Systems," in PES T&D, Orlando, 2012.

[5] D. T. O. Oyedokun, "Geomagnetically Induced Currents (GIC) in large power systems including transformer time response," PhD(Eng) Thesis University of Cape Town, South Africa, http://hdl.handle.net/11427/16708, 2015.

[6] O. Bíró, G. Kockza, G. Leber, K. Preis and B. Wagner, "Finite Element Analysis of Three-Phase Three Limb Power Transformers under DC Bias," IEEE Transaction on Magnetics, vol. 50, no. 2, pp. 565-568, February 2014.

[7] N. Hihat, E. Napieralska-Juszczak, J. Lecointe, J. Sykulski and K. Komeza, "Equivalent Permeability of Step-Lap Joints of Transformer Cores: Computational and Experimental Considerations," IEEE Transactions on Magnetics, vol. 47, no. 1, pp. 244-251, 2011.

[8] H. K. Chisepo, L. D. Borrill and C. T. Gaunt, "Measurements show need for transformer core joint details in finite element modeling of GIC and dc effects," The International Journal for Computation and Mathematics in Electrical and Electronic Engineering, vol. 37, no. 3, https://doi.org/10.1108/COMPEL-11-2016-0511, pp. 1011-1028, 2017b. ■

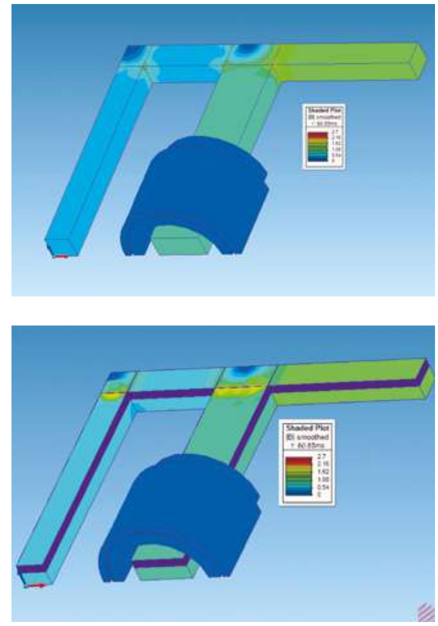


Figure 6. Flux distribution at normal AC excitation for calibrating the FEM model of no load current with measurements. Left: core modeled as solid with air gaps at the joints. Right: core modeled with explicit 0.3 TKES H111-30 grain oriented electrical core steel laminations stacked close to the core surface.

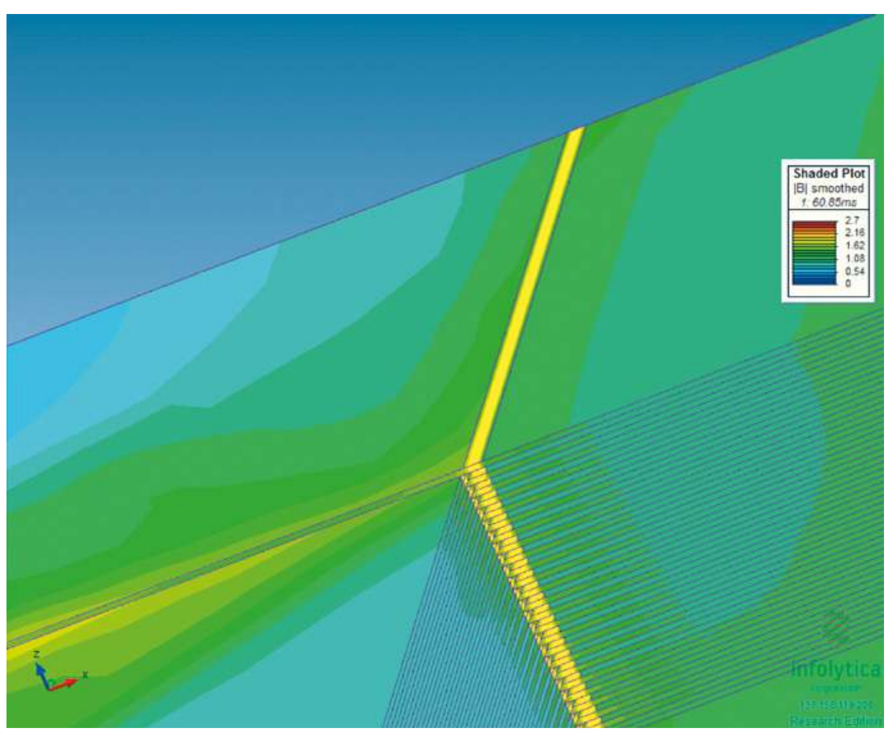


Figure 7. Core stacking lamination detail close to the surface of a mostly solid core.

Lowering Development Costs at SANDEN Manufacturing

Heat pump manufacturer uses Simcenter STAR-CCM+ to rapidly explore design space and bring products to market faster

Old challenges, new solution
Gaining market share in an established, competitive market – like the one for domestic hot water heaters – is always a challenge, even when you're introducing customers to higher-performing, environmentally-friendly product options. This challenge further amplifies the need to reduce the time and costs involved in the design and development of new products. In cooperation with advanced development engineers, the thermal engineers at SANDEN Manufacturing Europe (SME) met this challenge by relying on computational fluid dynamics (CFD) simulations to design a key component of an innovative heat pump that uses carbon dioxide (R744) as the working fluid. They used Simcenter STAR-CCM+ to perform the simulations, resulting in savings of €31,000 and 36 weeks – the cost and time required to build and test two physical prototypes, which were rendered unnecessary by the simulations. Additionally, the SANDEN engineers performed fully automated CFD-based design explorations – using Design Manager, a multidisciplinary design exploration tool that is part of Simcenter STAR-CCM+ – to determine the optimum shape of the passageways that direct air through the heat pump's evaporator to maximize the heat transfer that occurs there. Running Design Manager for six days on 24 CPU cores, the SANDEN engineers explored a much broader range of design possibilities, resulting in a configuration that was not only delivered in less time and cost, but which also significantly outperformed previous designs developed using conventional, manual approaches (including a 60% improvement in the air recirculation ratio – a key performance metric in determining the heat transfer occurring in the evaporator).

Adrien Rochelle, an advanced development engineer at SME, summarized their experience: "The automated and integrated design exploration approach with Simcenter STAR-CCM+ saved us a significant amount of time and expense. Without it, we could not have achieved this level of performance improvements."

Simulation – the key to reducing development time and cost
SANDEN Japan pioneered the development of heat pumps for residential sanitary water heating, using CO_2 as the refrigerant fluid, in response to the ever-present demand for better performance and greater energy efficiency. These CO_2-based heat pumps have emerged as an attractive option due to their superior performance and energy efficiency compared to traditional electric water heaters. Additionally, CO_2 is an eco-friendly, natural refrigerant – compared to the fluorinated gases (F-gases) more commonly used in heat pumps, CO_2 has no ozone depleting potential (ODP), has very low global warming potential (GWP), and is non-flammable and non-toxic. Consequently, CO_2-based heat pumps are an ideal match for use in the new wave of buildings being designed with low energy consumption in mind, as they comply with the latest regulatory requirements for high energy efficiency and use of renewables.

The basic components of a CO_2-based heat pump are illustrated in Figure 1. An integrated pump draws cold water from the bottom of the water tank; the heating of the water occurs as it flows through the gas cooler, which is a heat exchanger (coaxial countercurrent

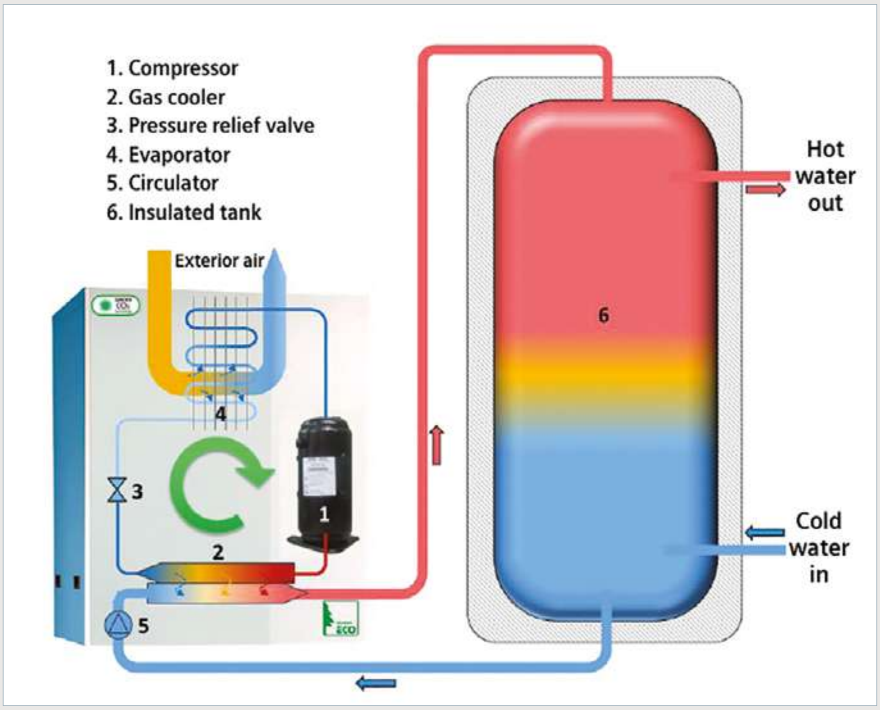


Figure 1: Schematic of a CO_2-based air/water heat pump.

type). This results in an increase in the water temperature, from 10°C to approximately 65°C, after which the hot water is then directed to the top of the tank, near the hot water outlet, allowing an almost immediate availability of water at 65°C.

As a relatively new technology, the cost of parts for these CO_2-based systems is high compared to those in traditional (hydrofluorocarbon) electric heat pumps. As such, the need to minimize the time and costs required to take the technology to market is particularly acute. One way to meet this challenge is minimizing the reliance on physical prototypes in the development process, as they are very expensive (in addition to being slow to build and test). At SANDEN, this meant using Simcenter STAR-CCM+ to automate the search for the best design for a critical component of the system – the evaporator.

As illustrated in Figure 1, the evaporator is a heat exchanger in which exterior air flows over refrigerant-filled passages, resulting in the refrigerant (CO_2) absorbing heat from the exterior air. Better heat transfer in the evaporator results in better performance of the entire system. As such, it is critical to shape the passageways that direct air through the evaporator in such a way as to maximize the heat transfer that occurs in the evaporator.

Figure 2: Detail of ducting geometry surrounding the evaporator and the corresponding computational mesh (evaporator is marked as component 4 in left image, and is highlighted in purple on right image).

Design exploration, not just assessment of individual designs

Based on their years of experience in the design of evaporators used in these heat pumps, the thermal and advanced development engineers at SANDEN had established three well-defined criteria that must be satisfied to maximize the heat transfer in the evaporator:

1. The air velocity on the evaporator interface should be as uniform as possible; specifically, the standard deviation of the velocity magnitude should be less than 0.28 m/s,

2. The velocity of the air flowing through the evaporator should not exceed 1.8 m/s, and

3. The air recirculation ratio should be less than 5%. The recirculation ratio is defined as the percentage of a cross-section through which the air flows in the opposite direction after passing through the evaporator; this is to be minimized as it negatively affects the efficiency of the evaporator.

The geometry of the ducting surrounding the evaporator is illustrated in Figure 2, along with the computational mesh used in the Simcenter STAR-CCM+ simulations. A centrifugal fan (component 6 in Figure 2) draws exterior air through the upstream duct (1), past the top suction section (2), into the suction chamber (3), and through the evaporator (4). The air is then drawn into the shroud (5), then into the fan casing (7) before finally being sent out by the fan (6) to the downstream duct (8). The twisting path the air takes attests to the magnitude of the design challenge, which is magnified by the fact that the evaporator around which the duct is being designed is quite small.

The SANDEN engineers were already aware of the value of virtual prototypes and simulation – they had been running Simcenter STAR-CCM+ simulations of the flow through the evaporator for validating design variants that they had developed based on their own expertize. These were validation runs carried out to understand the effects of individual design parameters, not necessarily to drive the identification of new candidate designs. Even so, these runs had already resulted in the identification of a good "baseline" design that met the maximum velocity criteria listed above (but not the other two).

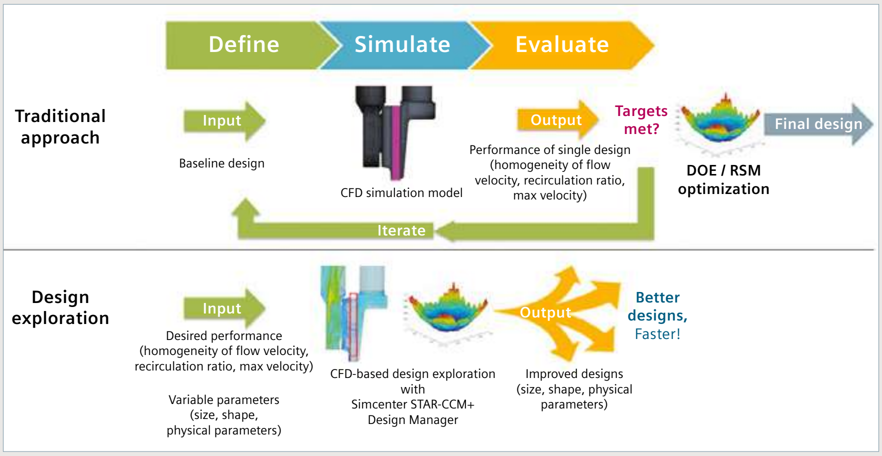


Figure 3: Traditional design approach versus design exploration.

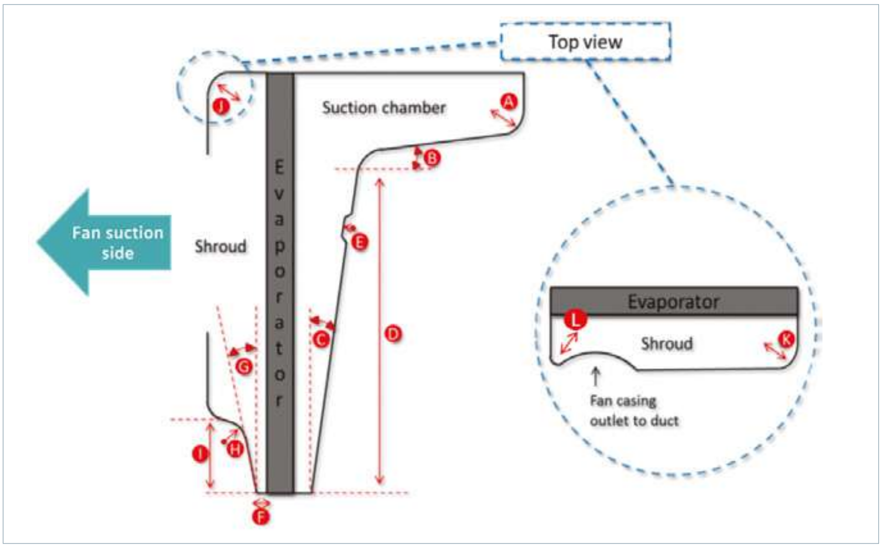


Figure 4: The 12 design parameters used in the Simcenter STAR-CCM+ Design Manager study.

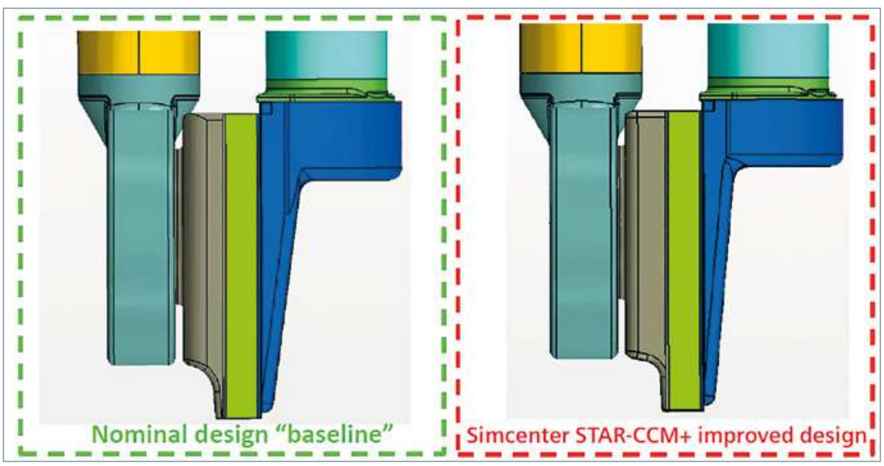


Figure 5: Comparison of baseline design and result of Design Manager exploration study.

	Baseline values	Improved design values	Percentage variation
A	R19	R20	5.3%
B	0,1°	0,1°	± 0%
C	5°	7,3°	**46%**
D	419 mm	390,5 mm	-7.0%
E	6 mm	0,1 mm	**-98.3 %**
F	5 mm	12 mm	**58.30%**
G	3°	4°	-33.3%
H	R30	R48 mm	**37.50%**
I	91,816 m	90 mm	-2.02%
J	R10	R10	± 0%
K	R20	R39	**48.70%**
L	R20	R15	-33.30%

Figure 6: Comparison of design parameter values of baseline design and Design Manager improved design.

The challenge for the SANDEN engineers, then, was to speed up the discovery of duct shapes that met all their performance criteria. Their initial approach was to automate the traditional design of experiments (DOE) plan they would have performed manually in past years to determine the effects of changing a small number of design variables. Using a Java macro, they developed a procedure to automate the evaluation of design points, but found that it was slow and cumbersome, even when considering as few as five design parameters. They then turned to Design Manager – the design exploration tool within Simcenter STAR-CCM+, which is purpose-built for conducting automated exploration of the design space-to guide their design process.

As illustrated in Figure 3, design exploration with Simcenter STAR-CCM+ represents a fundamentally new way of conducting engineering design. The traditional approach (such as the DOE plan mentioned earlier) involves simply simulating the CAD model, assessing whether the performance of a particular design is "good enough," then performing successive simulations on variants of that design until performance criteria are met. This was a perfectly reasonable approach back when computing resources were limited, but it suffers from a number of significant drawbacks, including limited model fidelity and the inability to handle large numbers of design parameters. True design exploration starts by defining the performance criteria (the objectives) upfront, and then using intelligent search algorithms (or better yet, multiple algorithms) to discover families of better designs. This allows for using computational models of high fidelity (to capture the pertinent physics), parameterized using a large number of design variables.

In the case of the ducting for the SANDEN evaporator, the engineers used 12 design parameters in their Design Manager study, as shown in Figure 4.

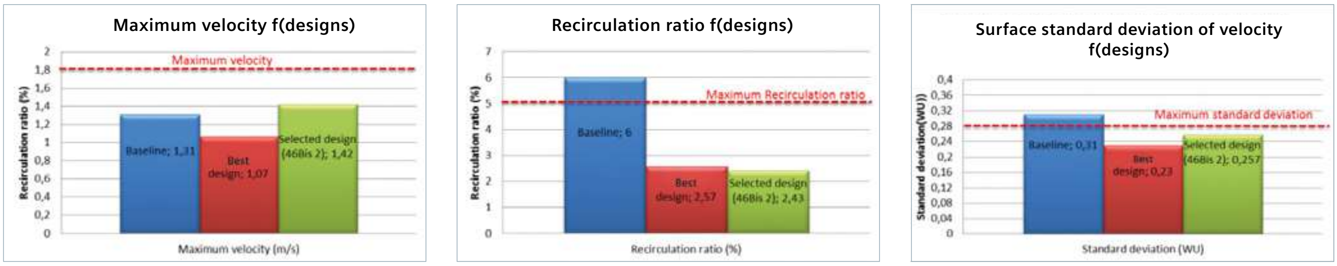


Figure 7: Performance comparison of baseline and Design Manager improved designs, in addition to selected "final" design.

Seemingly small changes result in big performance gains

Figure 5 shows a comparison of the "baseline" duct design and the design found after running Design Manager for six days on 24 CPU cores. While the geometrical differences may not be readily apparent from a casual visual comparison of the two configurations, looking at the changes in the numerical values of each of the 12 design parameters reveals that several of the values changed significantly, as shown in Figure 6.

More importantly, the effects of the changes on the performance metrics are significant, as shown in Figure 7.

Figure 7 highlights an important benefit of the kind of design exploration that Design Manager allows one to conduct: the exploration leads to families of better designs, which allows designers to choose the design whose performance represents the best balance – in the case of the evaporator ducting, the final design that the engineers at SANDEN selected was the one that had the lowest recirculation ratio, even though it had higher values of maximum velocity and velocity standard deviation than the representative design indicated by the red columns. In addition, Design Manager allows you to gain an understanding of which design variables have the greatest effect on specific performance objectives (no small feat when you have several design variables, as in this case).

It should be noted that the design exploration conducted by the SANDEN engineers involved a total of 59 Simcenter STAR-CCM+ evaluation runs. These resulted in 39 designs which satisfied all three performance criteria, 19 that did not meet at least one of the three criteria, and only one that resulted in a code error. The low error rate was the result of close collaboration between the SANDEN engineers and their dedicated Siemens support engineer to develop a process for carefully specifying minimum and maximum tolerances on each of the design parameters in order to reduce the occurrence of geometrically impossible situations.

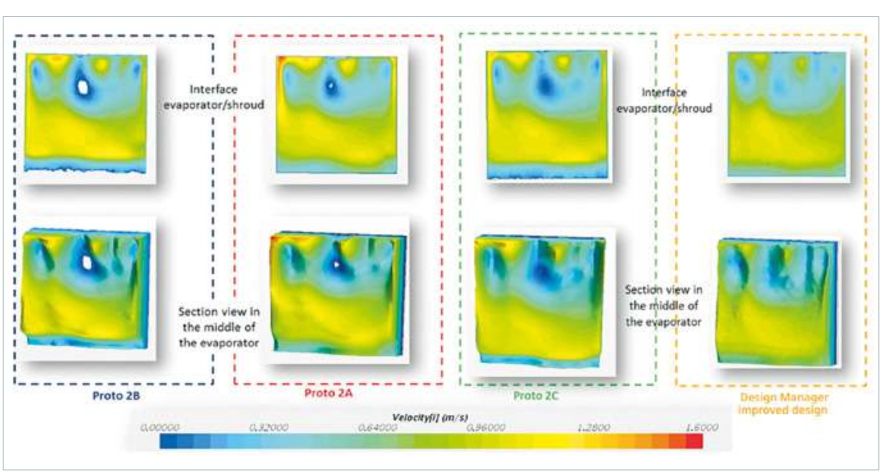


Figure 8: Comparison of velocity distribution at the evaporator interface.

The increased uniformity of the flow velocity at the evaporator interface in the improved design is evident in Figure 8, which shows the velocity variation at that interface. In addition, the areas of recirculation (indicated by the white areas in the velocity distributions) have been virtually eliminated.

Conclusion

The use of Simcenter STAR-CCM+ at SME's Technical Centre of France resulted in significant cost and time savings. Simcenter STAR-CCM+ simulations gave them the confidence to forego the (slow and expensive) building and experimental testing of two prototypes, and Design Manager found significantly better performing designs in a fraction of the time that it would have taken to build and test the prototypes.

Beyond the time and cost savings, there are the gains made in the fundamental understanding of the effects of each design parameter on the system's performance. Adrien Rochelle sums it up this way: "For me, with the design exploration capabilities of Simcenter STAR-CCM+, we've increased our understanding of the performance and the design of our fluid vein design. We now know the effects of the design parameters on the main responses and it represents a fundamental improvement in our understanding. The simulation provides us a kind of insurance in our design development by providing new knowledge and mastery of our design."

What's next for the team at SANDEN? Adrien says, "We are now developing the same process for automotive air conditioning compressors in Sanden Manufacturing Europe." ■

Solving Hybrid Engine Restart Vibration with Honda R&D Co., Ltd.

Resolving NVH of hybrid vehicles

Slowly but surely, the Western world is becoming familiar with cars going into sleep mode when idling – for example, while stopped at a traffic light. Engine start-stop systems (also called idling stop) have been quite common for a solid decade now, and their presence probably will keep growing. The first systems date back to the early 1980s, with the release of the Volkswagen Polo Formel E. in 1983 (in Europe only). Interestingly, disturbing vibrations at engine restart are the reason that start-stop systems didn't quite catch on back then, and are still an impeding factor for widespread adoption today.

Please don't stop the music

The benefit of idling stop systems is obvious. The start-stop feature not only cuts fuel consumption by 10-15%, but also reduces emissions accordingly. With fuel efficiency ranking at the very top of parameters that consumers consider when buying a car, engine start-stop has become the norm for original equipment manufacturers (OEMs) today. According to IHS Automotive, almost 70% of cars driving in Europe (where gas prices are highest) have the start-stop capability on board. But adoption is far behind the curve in other, less urbanized areas of the world, including the United States, with only 7% of cars equipped, and many owners turn the feature off when they have it.

This resistance cannot merely be reduced to laggards, and therefore raises concern throughout the automotive industry. The antagonism stems from the vibrations that come with restart, which are considered annoying because they interrupt or spoil music or silence. In contrast to starting the engine by physically turning a key or pushing a button, restart vibrations occur when passengers do not expect or want them. The vibration noise is especially disturbing in hybrid vehicles, which restart their engines while driving without any driver causal action like shifting gears. Engine restart happens entirely on the car's terms, rather than on those of the driver. As with turbulence on an airplane, the element of surprise can render an experience even more unpleasant than it would be.

Honda R&D Co., Ltd – an early adopter

As customers' wishes are predictors of their buying behavior, Japanese OEMs specializing in hybrid and electrified powertrains are investigating ways to resolve the noise, vibration and harshness (NVH) issues that go hand-in-hand with the restart technology. Honda R&D Co., Ltd wanted to be an early adopter with regards to the technology, which is why the company sought assistance from Simcenter™ Engineering services.

Honda and Siemens have been partners for decades, specifically through the use of Simcenter products and services. In fact, Honda has been a devoted and satisfied user of Simcenter SCADAS data acquisition hardware and Simcenter Testlab™ software for more than 20 years.

From test, through engineering consulting to system simulation

Mr. Satoshi Watanabe, responsible for Model-Based Development (MBD) at the powertrain NVH department at Honda, explains how he believes Simcenter testing tools far exceeded competitor alternatives long ago: "The Simcenter testing tools were multi-purpose; they allowed for transfer path analysis, impact testing, modal analysis and others, whereas the competition was still focused on performing just a single task."

With this versatility, Honda was able to optimize their testing processes significantly. "There was no more need to change the test setup and specialist between measurements, losing precious time," Watanabe explains. "Overall, thanks to the Simcenter tools, our testing process became much more integrated." Honda is still using Simcenter testing solutions for data acquisition and validation today.

Working with the Simcenter testing equipment, Honda accumulated a lot of in-house know-how, especially with regards to NVH. The company currently has this knowledge organized into different operational departments, each pursuing their specific sets of targets. This was the most effective approach to tackle the attribute- and subsystem-specific challenges of vehicle development. But with the advent of hybrid vehicles and their more complex NVH concerns, the company decided to review the departmentalized approach.

"Before we started developing hybrid vehicles, we were doing fine with our established processes and knowledge-base," Watanabe says. "However, we want to explore new ways to shorten our development cycle." Upon encountering the engine restart NVH issue in their Odyssey platform, Honda decided to implement other Simcenter solutions, and to contact Simcenter Engineering services.

"Thanks to our collaboration with Simcenter Engineering, our development cycle time is under better control."

Satoshi Watanabe
Model-Based Development for Powertrain NVH
Honda R&D Co., Ltd

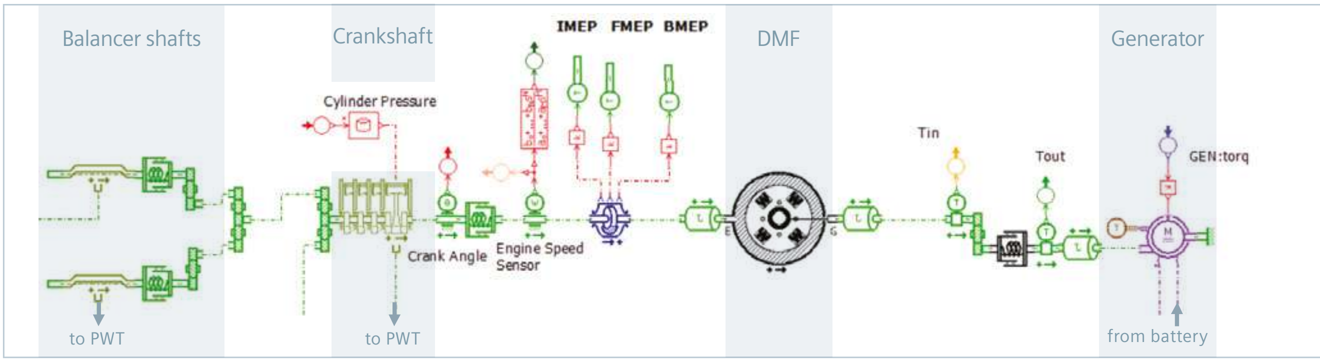


Figure 2: Hybrid driveline torsional model including combustion engine and generator to study ICE restart vibrations.

Simcenter Engineering would help Honda to couple their valuable but compartmentalized knowledge into an integrated solution approach.

"In order to properly capture and resolve the coupling issue as soon as possible, and before prototype availability, Simcenter experts introduced us to the Simcenter Amesim systems simulation software," Watanabe explains. With this software, Honda managed to build fully integrated engine and vehicle models, which were scaled to take into account combustion, mechanics and controls.

A subtle but significant difference

Due to the psychological and largely subjective factors at play, the restart issue is more intricate than science would lead one to suspect. Typically, when an engine starts, body vibrations are caused by the rigid body eigenvalues of the power plant during initial combustion. The difference between normal start and restart however, is that we expect these vibrations to occur in the former case. While expectations are difficult to objectify and therefore generalize, the only effective strategy seems to be to reduce restart vibrations to their absolute minimum, without compromising engine performance.

Conventionally, prediction of noise and vibration phenomena from the powertrain uses a method that inputs the measured in-cylinder pressure values as a source of engine vibration. "However, departing from a fixed in-cylinder pressure value, these models do not consider the parameters for engine cranking and firing, which contribute greatly to engine restart vibration," says Tom Van Houcke, a Simcenter NVH expert closely involved throughout the project.

Using Simcenter Amesim software, Simcenter Engineering experts and Honda set out to develop a new prediction technique and a new evaluation method for engine restart vibrations. Honda can now accurately predict the entire powertrain restart process, beginning with the vehicle controls signals causing a certain in-cylinder pressure, which then results into driveline torque, particular suspension and powertrain bushing interface forces and eventually body vibration shock.

Allowing for the in-cylinder pressure to deviate, Honda can now determine the parts characteristics for the engine restart vibration in the vehicle design

"By predicting systems behavior upfront, the workload afterwards is significantly reduced, which allows us to focus our efforts and resources on other priorities, such as brand image and value."

Satoshi Watanabe
Model-Based Development for Powertrain NVH
Honda R&D Co., Ltd

stage, as well as other noise and vibration phenomena, such as idling noise and vibration.

Technology partners for life

Honda appreciates more than the successful completion of the project today. "Thanks to our collaboration with the Simcenter Engineering team, our development cycle time is under better control," Watanabe asserts. Moreover, after the project completion, Honda received a full technology transfer, enabling them to reproduce all the techniques and methodologies for other purposes than engine restart vibrations. "Because of the technology transfer, we understand how to reproduce the applied techniques and adapt them to resolve all kinds of issues. We can now do this by ourselves, using our own assets to the fullest," Watanabe concludes.

As a result, Honda is now integrating and using MBD throughout their departments, which is allowing them to anticipate a variety of issues and frontload their solution. "Discussions are started earlier and thereby we avoid problems," Watanabe says. "By predicting systems behavior upfront, the workload afterwards is significantly reduced, which allows us to focus our efforts and resources on other priorities, such as brand image and value."

Engineering services as pathfinder to the Simcenter portfolio

Honda considers the collaboration with Simcenter Engineering services of great value in staying ahead in the race towards new technologies and methodologies, such as those developed in the context of this project. "Integrating 1D, 3D CAE and test is the strength of Simcenter Engineering, which has the Simcenter product portfolio at its disposal," Watanabe concludes. "As long as this combination of solutions is available, we will continue to work together in the future." ■

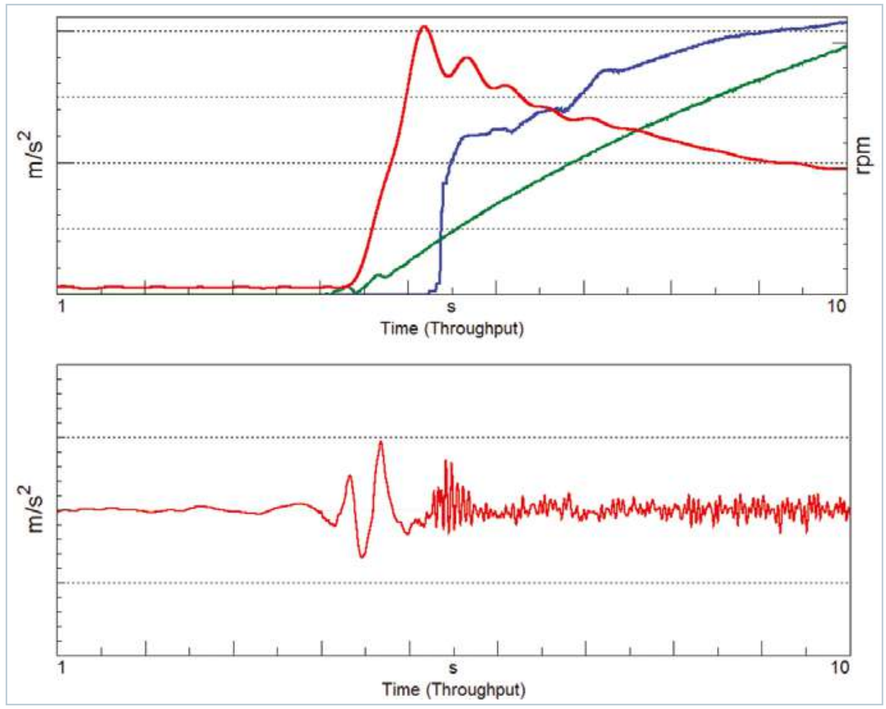


Figure 3: Vehicle acceleration maneuver in hybrid mode: combustion engine restarts to charge the battery after initial pure electric propulsion, generating body vibrations.

"Integrating 1D, 3D CAE and test is the strength of Simcenter Engineering, which has the Simcenter product portfolio at its disposal."

Satoshi Watanabe
Model-Based Development for Powertrain NVH
Honda R&D Co., Ltd

Design Challenges & Opportunities

Bearings manufacturer meets stringent accuracy requirements while improving productivity

Supporting mobility

Humankind has been trying to improve the mobility of people and materials by reducing friction between moving parts for centuries. The creators of the pyramids and Stonehenge were able to move massive structures by placing cylindrical wooden rollers beneath great weights to reduce the coefficient of friction and the force required to move them. These world wonders were made possible by some of the earliest known applications of bearings.

Modern bearings with races and balls were first documented in the fifteenth century by Leonardo da Vinci for his helicopter model. Since then, the design, mobility and precision of bearings have developed dramatically in many application domains. In the semiconductor and medical device industries, miniaturization and increasing product complexity have revolutionized motion systems and their components. The precision and accuracy of motion systems are highly dependent on bearings assemblies and how they are integrated into systems. Precisie Metal Bearings (PM-Bearings) is one of only a few manufacturers in the world that provide high-precision linear bearings.

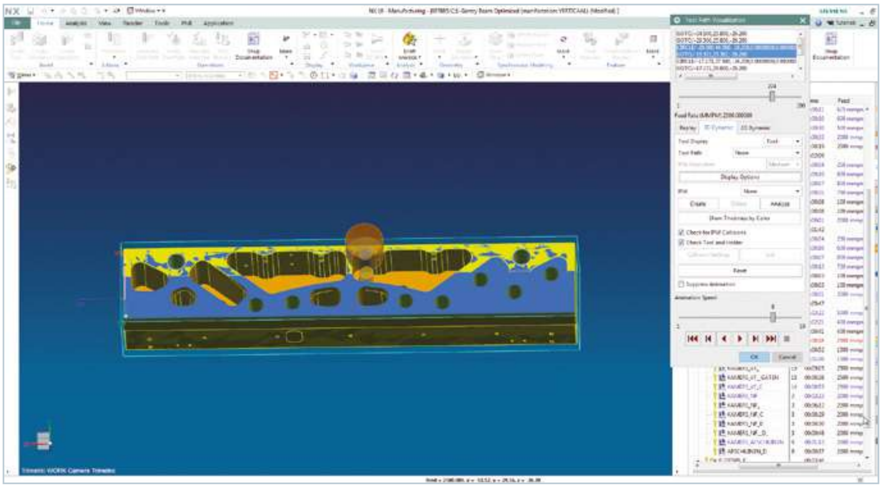
A leader in precision bearings

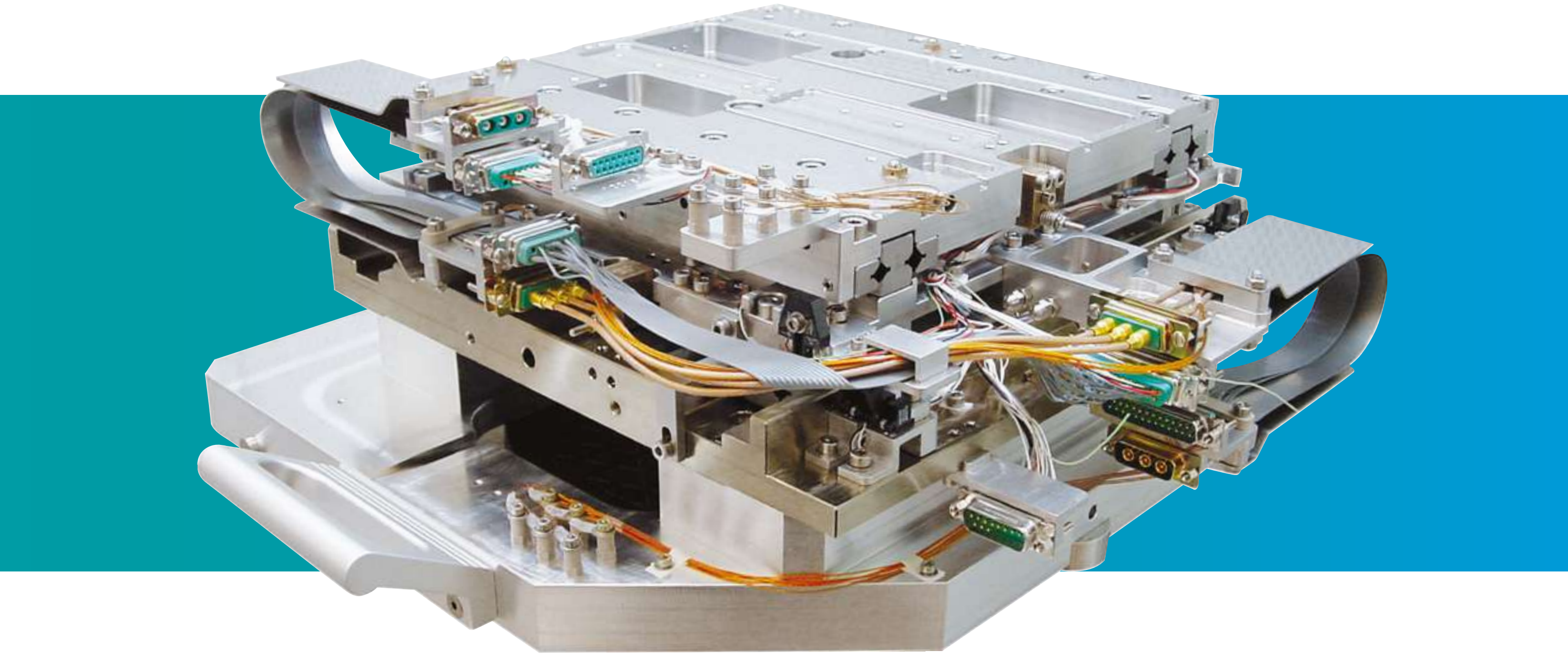
PM-Bearings specializes in the design and manufacture of high-precision linear bearings, motion systems and positioning stages, and supplies the high-end semiconductor, medical device and machine tool industries. The company was founded in 1966 as a manufacturer of linear bearings, and has expanded to include design, manufacturing and assembly of custom-made multi-axis positioning stages with complete mechatronic integration. Located in the Netherlands at Dedemsvaart, the company employs 140 people and supplies customers worldwide. The company's products range from very small bearings (10 millimeters in length) up to systems with footprints of 1.2 to 1.5 square meters with stroke lengths of one meter. The portfolio encompasses linear motion components including precision slides, positioning tables and bearings stages. PM-Bearings is part of the PM group, along with other companies specialized in high-tech machining. Its global customer base extends from Silicon Valley to Shenzhen.

Focusing on customer needs

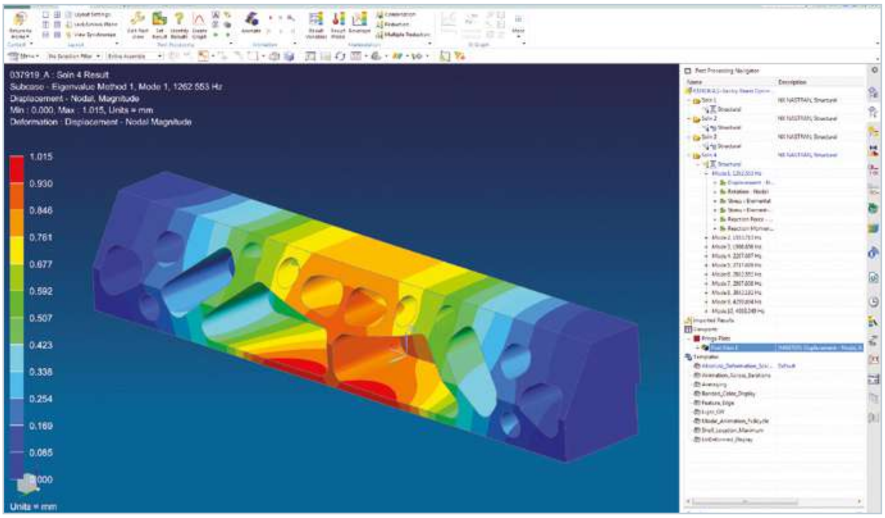
The diversity of customer needs presents a number of challenges for the precision bearing manufacturer. These include requests for customization (up to 80% of the system design in most cases), short time to market, and extreme precision (for example, smooth motion of two microns per meter). Additionally there are requirements for reduced bearing sizes and exotic materials, including ceramics used in non-magnetic or ultra-high vacuum environments.

"When the customer submits a specification, often the deadline is





theoretically yesterday," says Jan Willem Ridderinkhof, Manager of R&D and Engineering at PM-Bearings. "Simple modifications like moving a hole can be implemented with a few operations. But more and more, clients come with a complete new system specification. The trend in the semiconductor industry to make chips smaller and smaller also complicates our work."



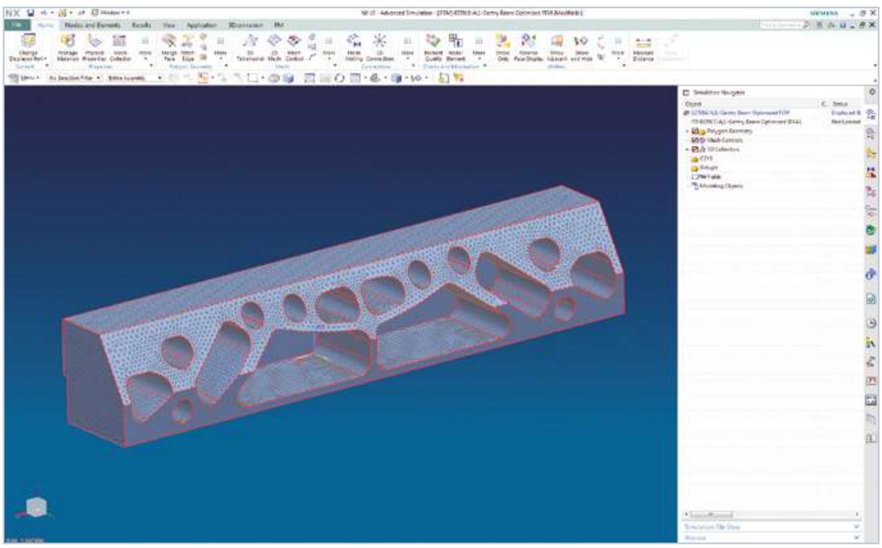
"On one of the wafer inspection machines with a 5-axis system, we had a specification for a settling time of 500 milliseconds and vibrations within 50 nanometers while making a linear displacement," Ridderinkhof continues. "Such extreme precision requirements have strong impacts on our business model. We are moving from a precision bearing supplier to a fully integrated motion system integrator."

To maintain a competitive edge, PM-Bearings knew that complete control of the product realization, from design to delivery, was essential. This is why the company chose a comprehensive set of solutions from product lifecycle management (PLM) specialist Siemens PLM Software. These include NX™ software for computer-aided design (CAD), Simcenter software for performance prediction, NX CAM for computer-aided manufacturing and Teamcenter® software for PLM to allow all stakeholders use the same data and workflows to make the right decisions. PM-Bearings has used these solutions for more than a decade, supported by the Siemens PLM Software reseller, cards PLM Solutions. "Having a local contact is very helpful to us," explains Ridderinkhof. "They did a great job in deploying and customizing Teamcenter. Also for Simcenter, they maintain contacts with Siemens PLM software experts."

"At cards PLM Solutions, our goal is to enable customers make the best possible products," says Erik Burghoorn, CEO of cards PLM Solutions. "We use our expertize to help companies implement software solutions that include best practices to fulfill their requirements. PLM Solutions enables companies like PM-Bearings to increase efficiency by digitalizing their production processes."

Realizing ideas with NX

Does the linear motion bearing roll or slide? Will the sliding motion be purely linear or circular? Is the driving force of the slide motorized or non-motorized? Which material – metallic or ceramic – is best suited for the operating condition of the bearing? Walter Meijerink, Mechatronic Engineer at PM-Bearings, makes such informed decisions every day while using NX to design linear slides and positioning



systems, taking into account the specifications for precision, repeatability, loading and the operating environment. “I am responsible for transforming customers’ requirements into final drawings,” Meijerink says. “NX CAD helps me develop design ideas rapidly and efficiently.”

Instead of classifying parts by alphanumeric codes, the part family feature of Teamcenter enables PM-Bearings to classify parts based on predefined criteria. It helps to embed permitted part variations and to impart defined metrics and design standards. For example, similar parts of different lengths can be grouped under one part family. “Part family is a useful feature,” says Meijerink. “I can rapidly assemble different parts and this saves a lot of time.” The complete model can be easily shared with design analysts and CAM engineers, making product development fast and short. “Another key feature of NX CAD that is important for my job is the ability to read design data coming from customers who use different CAD platforms,” Meijerink adds.

Accelerating analysis iterations

“The integrated NX CAD and Simcenter 3D CAE platform helps us accelerate our daily tasks,” says Mathys te Wierik, R&D Engineer at PM-Bearings. “Once the designs are complete, they must be analyzed for rigidity and performance. Simcenter excels at this. I can easily prepare the geometry for simulation; if necessary, introduce geometry simplification like midsurfacing, mesh it, submit the task to the NX Nastran solver and analyze the computed results.”

“If required, I can perform geometry modifications like shifting stiffening ribs on the fly, directly within Simcenter,” adds te Wierik. “Because all associated parts are updated automatically, I can solve it and compare analyses, which saves a lot of time. After some design and analysis iterations the validated design is sent directly to the CAM engineer, without exporting data to a different format. ”

“In the past, we outsourced the finite element calculation,” says te Wierik. “It took two weeks to get results. Now with Simcenter, we can get the results in a matter of minutes or hours, depending on the analysis complexity. This speeds up many things. Doing simulation in-house gives a lot of insights about our product.”

Nanometer precision

With extreme requirements for dimensional precision and for minimizing vibration magnitude down to nanometers, PM-Bearings faces unique manufacturing challenges. The accuracy, which is highly dependent on the machining process, affects products’ performance and aesthetics. Efficiently machining precise moving parts from 3D designs requires exact manufacturing instructions for the people and machines on the shop floor. NX CAM, with its integrated CAD, NC programming and machining simulation capabilities, enables PM-Bearings to define a complete manufacturing plan long before the first production run.

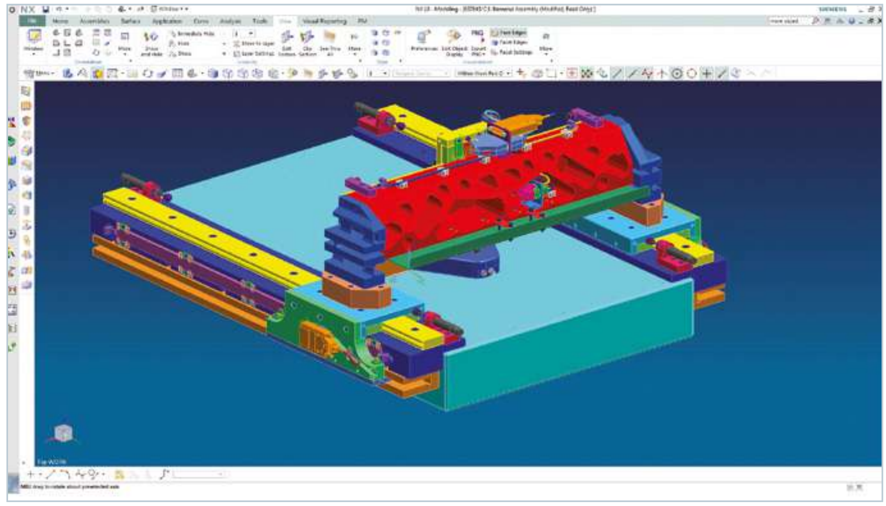
Kenny Prins, Computer-Aided Manufacturing (CAM) Engineer responsible for planning and control of production at PM-Bearings, knows from experience that in order to set up the production run completely right the first time, one must plan it digitally. “In the beginning, all the programming was done at the machine,” Prins says. “For each change we had to check and modify the whole program, which was very time-consuming and prone to errors. The resulting downtime of shop floor personnel and machine tools cost us a lot of time and money. Now, with NX CAM we can reduce programing time by up to 80% using a digital twin of the workpiece, tooling, and machine, which helps us stay competitive.”

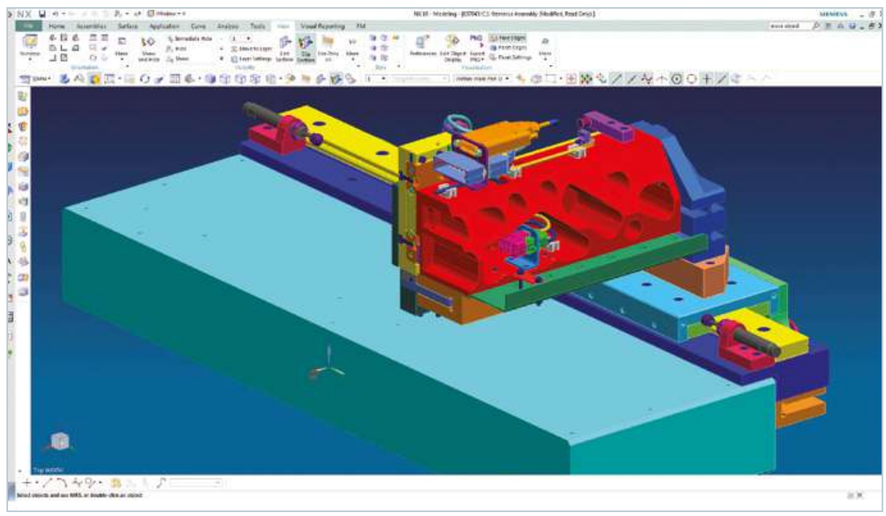
Like many manufacturing companies, PM-Bearings needs to mill, turn, and finish higher-quality products faster, while extending the tool life. “When I receive a model for production from the design team, I first make sure that our tools and machines can handle the job," explains Prins. "A quick verification of the operations in NX CAM software shows whether the physical machine and tooling might have issues handling the designed part. This is a very powerful functionality that helps us easily eliminate any possible manufacturing problem early in the planning process. When we identify a manufacturability issue, we work with the design team to modify the design. Our teams rely on Teamcenter software for collaboration and data sharing. Using this closed-loop process, the design change automatically propagates through the analysis and planning steps, all the way to manufacturing on the shop floor."

"Another functionality that I use every day are the flexible machining strategies in NX CAM," Prins continues. "Not all machining cuts are equal. The precision of the final part is directly linked to the stress induced by cutting operations. The machining strategies are different when milling a large metal block or small parts with 5mm thickness. The software automatically proposes a tool path to remove material, but it also gives us flexibility to use our know-how to adjust the machining strategy for best results. By capturing and re-using our experience, we apply efficient machining processes to achieve excellent part quality while extending tool life."

Precision through digitalization

"3D model sharing and access to the latest information are the greatest strengths of Teamcenter," says Ridderinkhof. "The JT format enables model visualization not only for the CAM team, but also for the rest of the company, including the manufacturing planning and shop floor personnel. Everyone benefits from being able to see those models, turn them around, see hidden components and see how the structure is built up; even engineers in assembly use this information. It touches all the stakeholders, from sales demonstrating products to customers, to educating personnel internally about how to build things. The attributes attached to the items, such as the supplier name and the part number, are used to generate the bill of material automatically in the ERP system, extending our product digitalization process to the purchasing department and reducing any human errors."





"We are committed to remain a high-end precision machining company, so we need to stay ahead of the competition," Ridderinkhof continues. "In order to keep delivering the most accurate bearing and positioning slides, we digitalized our entire engineering and manufacturing process using Siemens PLM Software solutions. The more time we spend with the software at the beginning of the production process, the better the output our company manufactures. The overall efficiency of the company is increasing. We can deliver a range of bearings in less time while eliminating failures and improving quality, which in turn improves our profitability." ■

"The integrated NX CAD and Simcenter 3D CAE platform helps us accelerate our daily tasks."

Mathys te Wierik
R&D Engineer
PM-Bearings

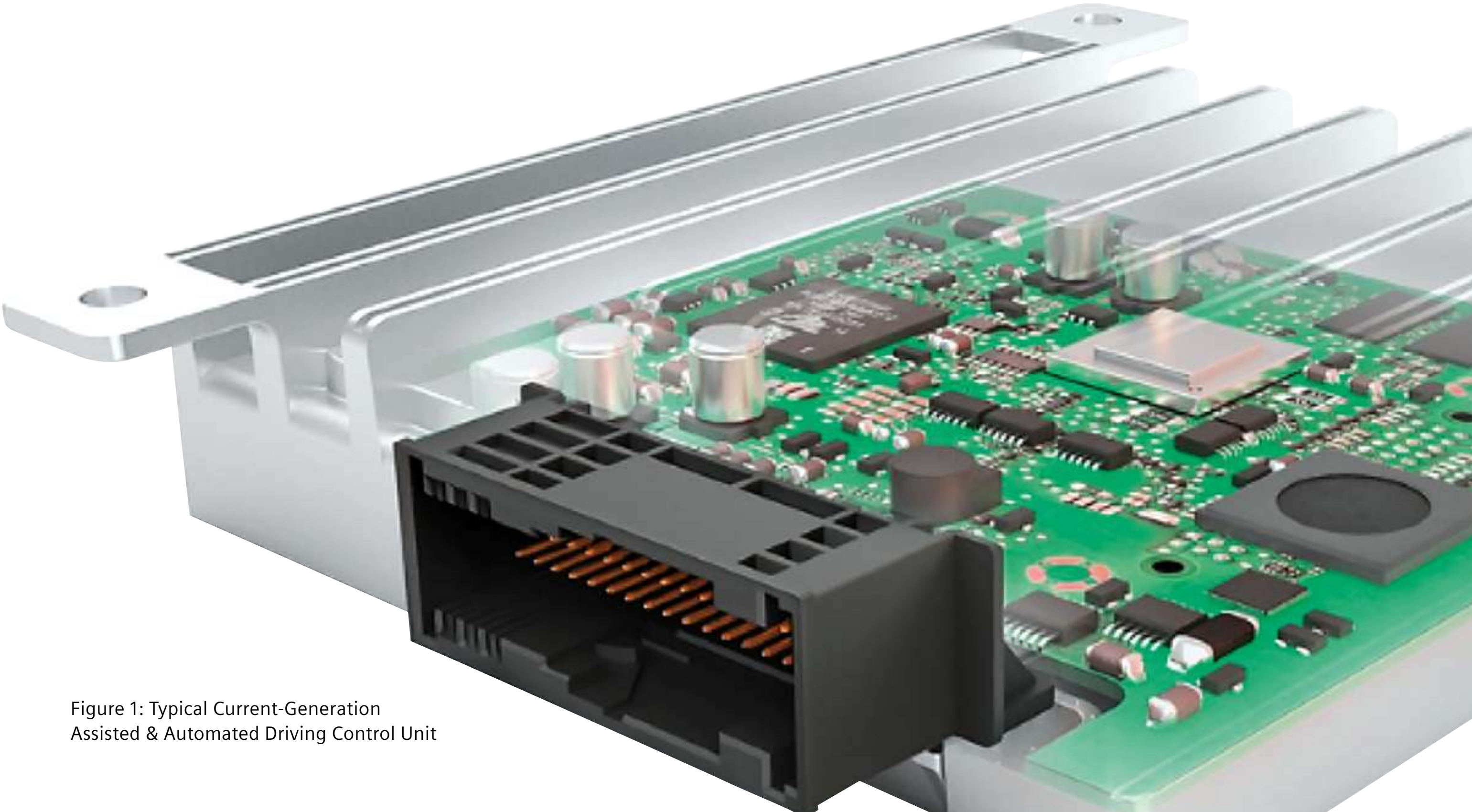


Figure 1: Typical Current-Generation Assisted & Automated Driving Control Unit

Driving up Reliability for ADAS Applications

By George Gabriel Chiriac & Gabriel Ciobanu, Chassis & Safety Division, Continental Automotive AG

Founded in 1871, Continental develops pioneering technologies and services for sustainable and connected mobility of people and their goods. Continental's technologies offer efficient and affordable intelligent safety solutions for vehicles, traffic and transportation and machines. Continental generated sales of €44 billion in 2017 across 60 countries and currently employs more than 243,000 people.

The oldest university in Romania is located in Lasi, home to the country's first school of engineering. Therefore, it's a fitting home for Continental Automotive AG's Chassis & Safety Division, along with the Interior and Powertrain Divisions. Today, the Chassis & Safety Division covers Advanced Driver Assistance Systems, Systems & Technology, and Passive Safety & Sensors, emphasizing the focus that Continental has on-driver assistance technology, as safety concerns propel the automotive industry towards the ultimate goal of fully autonomous, or driverless cars.

Our role in the Chassis & Safety Division is to analyse and understand the thermal, and thermo-mechanical design of Assisted & Automated Driving Control Units, as these are the central 'brain' for the vehicle's assisted and

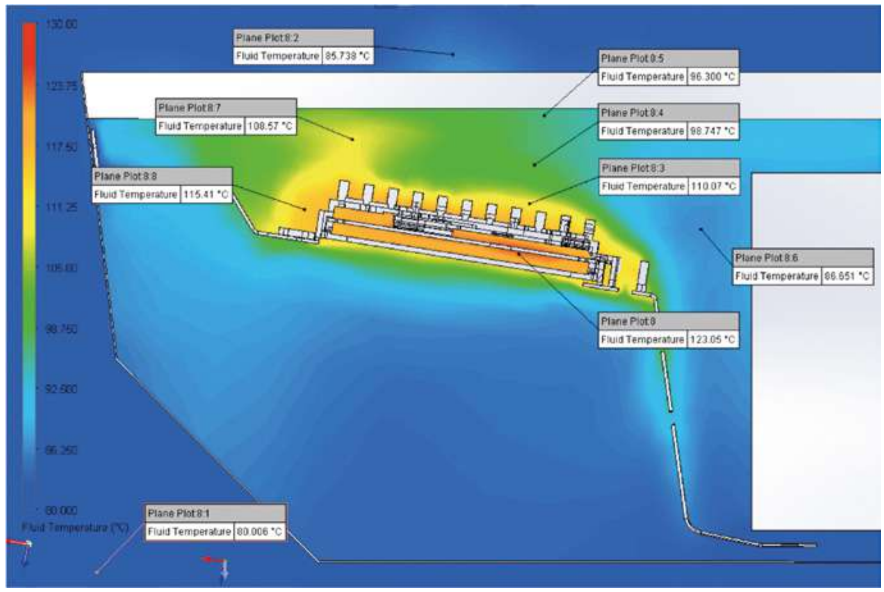


Figure 2: Slice through Assisted & Automated Driving Control Unit showing mounting

automated driving functions. This encompasses all of the vehicle's electronic chassis and safety systems to optimum driver assistance in all driving systems. A typical current generation ADCU us shown in Figure 1.

Figure 1 shows aspects of the cooling solution, being a finned metal enclosure, typically bolted to the chassis of the vehicle to conduct heat from the system. Most of the components seen would be represented directly within the thermal model. A typical mounting arrangement is shown in Figure 2 for a similar unit, which reveals further aspects of the thermal design, through the use of gap pads between key components and the enclosure, and conduction from the edge of the PCB.

The thermal design in such systems is very challenging for a number of reasons. The system has to withstand a maximum underhood temperature of 80°C as its ambient condition. The orientation of the system often cannot be guaranteed, which is why the unit shown in Figure 2 uses a pin fin arrangement for the heatsink. However, the most challenging aspect of the design is that the thermal design needs to ensure the thermo-mechanical reliability of the unit. While components have to be kept sufficiently cool, the high underhood temperatures mean that the electronics are subjected to far more thermo-mechanical stress due to the temperature changes during a drive cycle than would be experienced by, for example, most consumer electronics products. For automotive applications, the thermal and thermo-mechanical design go together hand-in-hand.

Our preferred approach is to perform the thermal design in Simcenter Flotherm™ XT software. This software allows us to build a thermal model that takes into account all the components on the board, usually using 2-Resistor models, plus the board layers with layer copper detail, this information is needed for a highly-accurate thermal simulation using a geometric representation of the board assembly that can be used directly for our thermo-mechanical stress simulations. An advantage of using Simcenter Flotherm XT is that it uses a solid modeler for geometry handling, so we can directly transfer a clean error-free CAD model of the entire board assembly while retaining all the assembly, part and feature information. This allows us to easily attach materials and boundary conditions for the thermo-mechanical simulation.

The thermo-mechanical simulations we perform can also include the stiffening effect of the components on the board, which have an effect on the prediction of the effective, i.e. von-Mises stress, and the out-of-plane deflection of the board, shown in Figure 3.

Using a lumped treatment for the board can significantly under or over-predict the magnitude and location of the maximum deflection. Using a lumped representation of the board also totally fails to capture the variation in von-Mises stress shown in Figure 3.

The high level of geometric detail we are able to work to in Simcenter Flotherm XT also contributes to the accuracy of the stress and deflection predictions we are able to make, as we get a finer grain resolution of temperature throughout the structure.

Generally it is impossible to replicate the CFD simulation thermal results directly in finite element software, as the CFD simulation predicts temperatures throughout the solid and fluid regions, and the local surface heat transfer rate, which typically varies

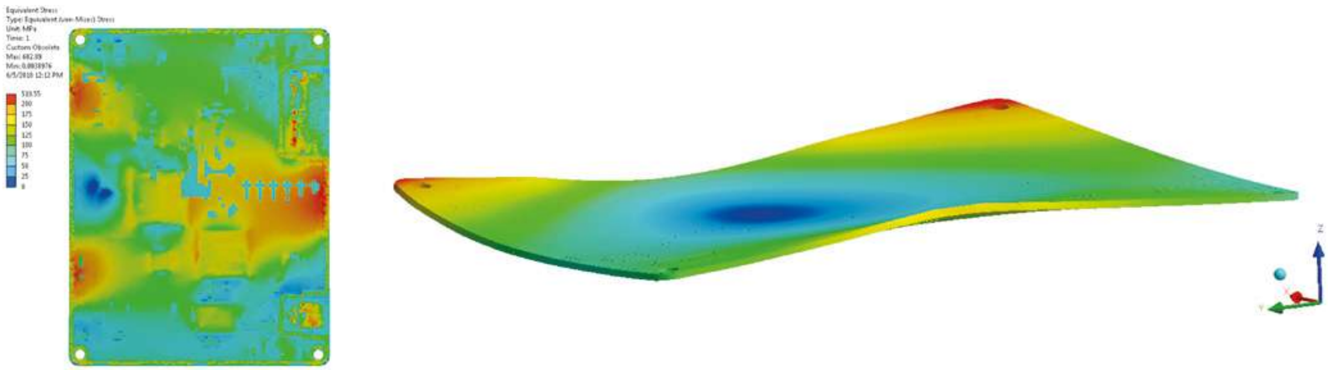


Figure 3: Von-Mises stress and out-of-plane deflection of detailed board with components modeled

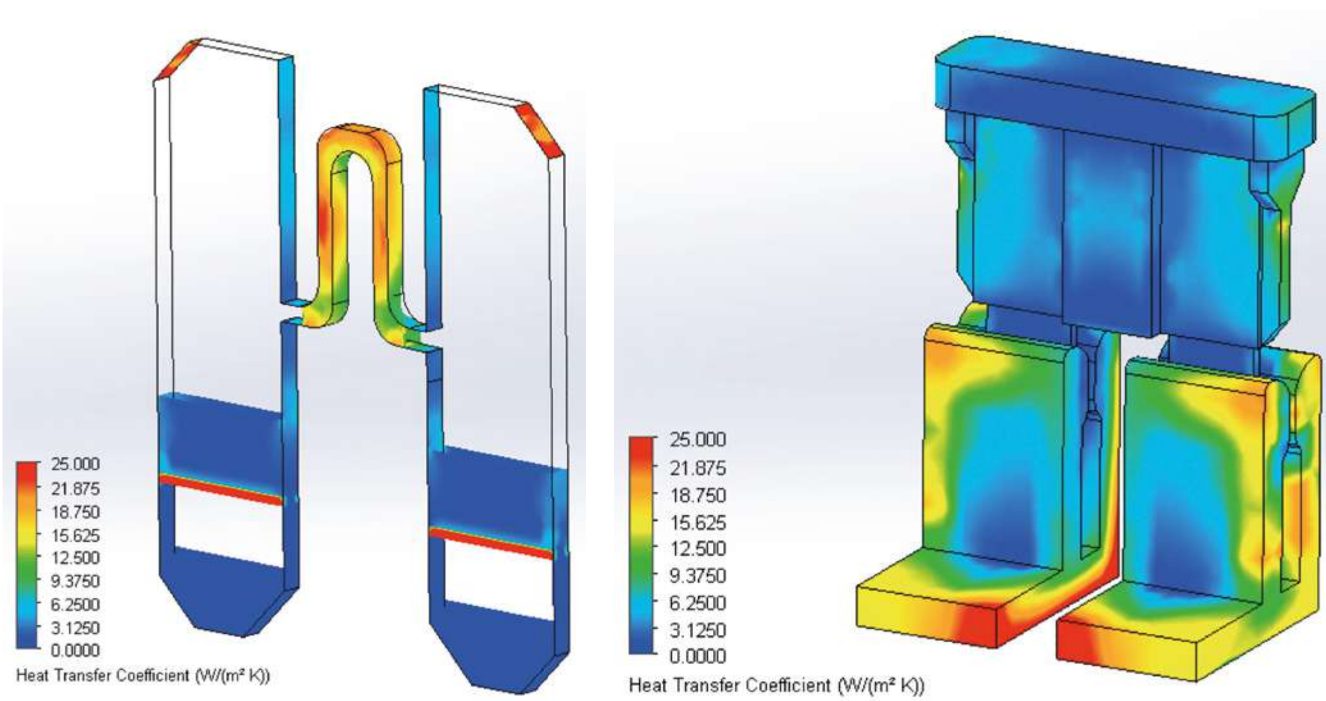


Figure 4: Local surface heat transfer coefficients on blade-type automotive 10A fuse

everywhere across the surface. The traditional approach to represent convection within a finite element tool is to apply a heat transfer coefficient to the surfaces of the finite element model, with different heat transfer coefficients being applied to different surfaces if desired. However, these are global surface-averaged values that link the surface to some remote temperature, typically the ambient, whereas in the CFD software the heat transfer varies locally over the surface and the temperature that drives the surface-to-fluid heat transfer is the local fluid temperature in the near-wall.

Figure 4 shows the variation in the local heat transfer coefficient values on the exposed surfaces of the geometry. These values can be exported from Simcenter Flotherm XT. However, they relate the local heat flux to the local temperature difference between the surface and the fluid temperature in the cell adjacent to the surface. Using them with some other reference temperature, e.g. the ambient, will give the wrong surface heat loss. The meshes between the CFD software and the FEA software are also different, which would also make it hard to map heat transfer coefficients from one tool to the other, given the high level of variation in heat transfer coefficient over the surface shown in Figure 4, and obtain the same local surface heat flux in both tools.

Surface heat flux can also be exported from Simcenter Flotherm XT on a cell-by-cell basis. However, the variation in this also makes it problematical to transfer between tools and get the same surface heat loss due to the mesh differences. Also, imposing a heat source within the solid, and imposing surface heat flux out of it over the entire surface of the structure as a pair of boundary conditions over-constrains the problem, as the temperature field within the structure is decoupled from the ambient.

Fortunately there is a much simpler solution. The most significant consideration for the finite element solution is the temperature field, as it is the change in temperature that drives the thermomechanical stress. By far the cleanest solution is to export the in-cell solid temperatures and their locations from Simcenter Flotherm XT into the FEA tool.

As clean CAD geometry is also transferred from Simcenter Flotherm XT, ensuring that the model is the same in both tools, the location of these temperatures relative to the geometry is correct. While the mesh in the FEA tool is different, it has the ability to map imported temperatures onto its native mesh. The transfer is seamless. We have tested the accuracy of the imported temperature field vs. temperatures generated directly in the FEA tool. For situations where the same boundary conditions can be imposed in each, for example in conduction only cases, and found the thermal results to be identical. Unsurprisingly, the stress results are also identical to within a fraction of a percent. ■

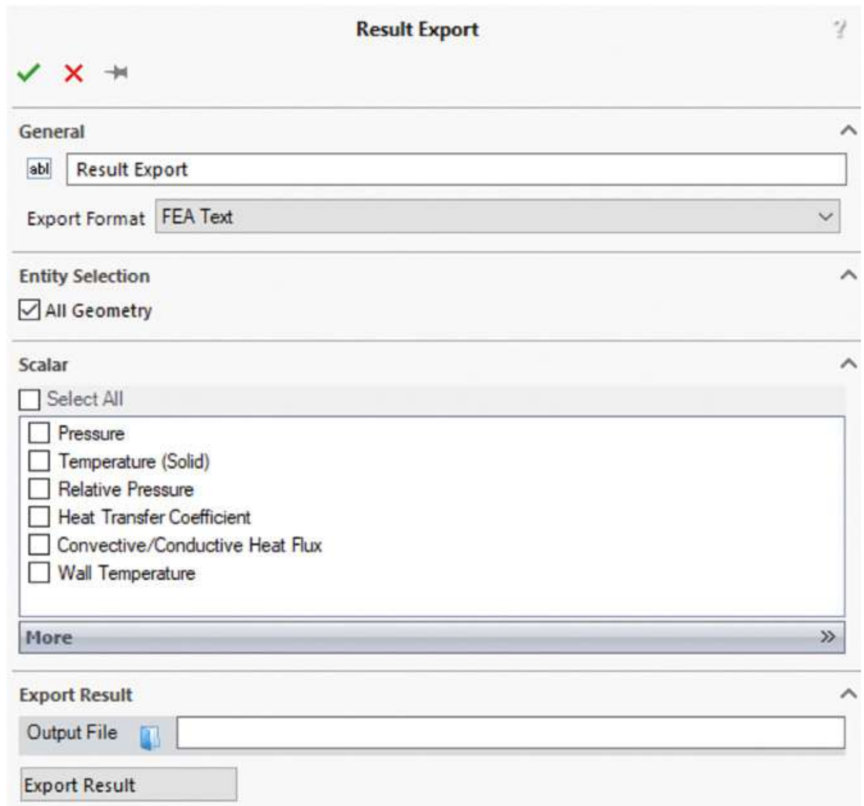
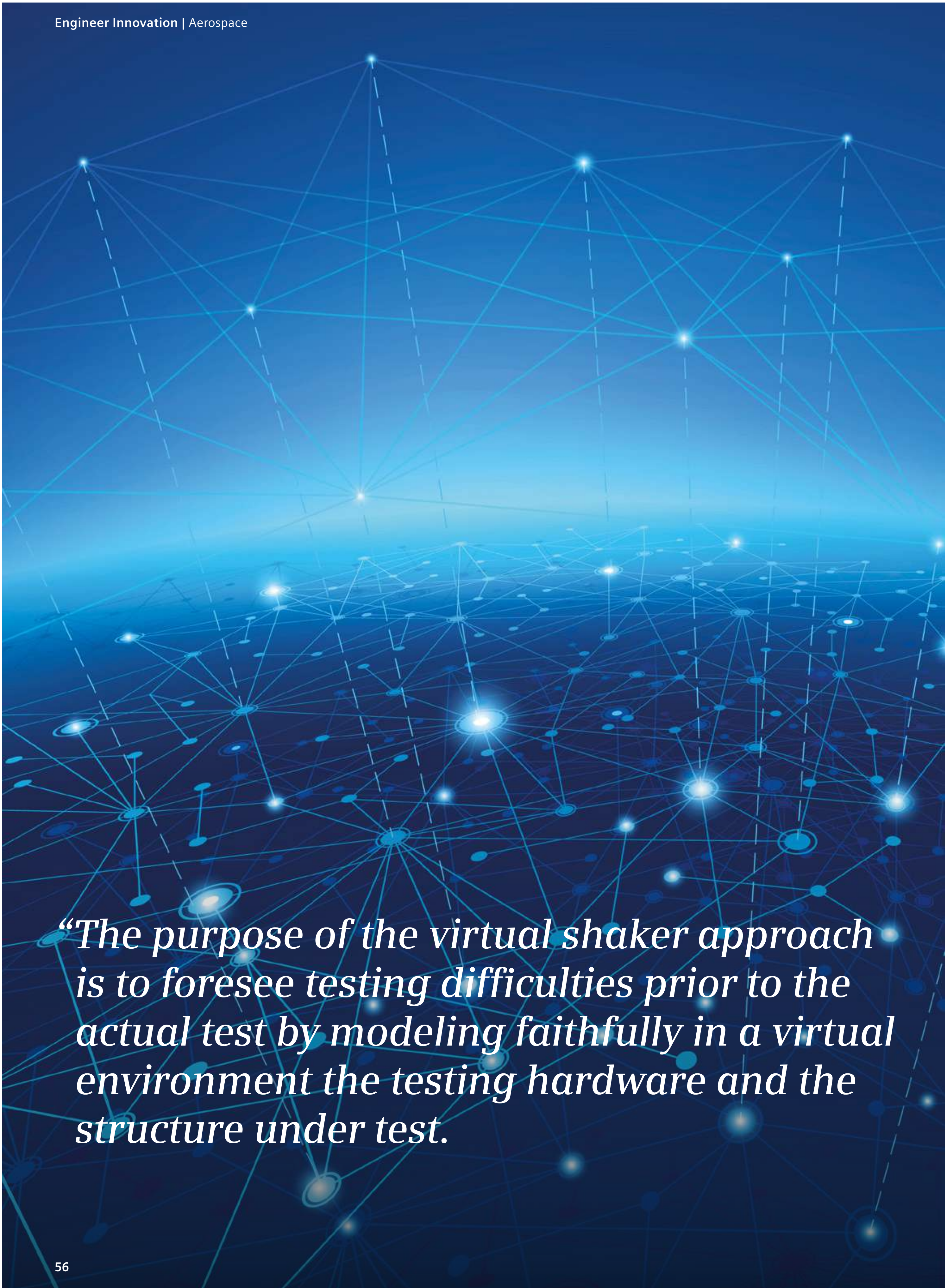


Figure 5: Simcenter Flotherm XT Results Export Dialog

"We find that doing the geometry creation and thermal simulation in Simcenter Flotherm XT and then transferring clean geometry and temperatures to our FEA tool to be the most efficient way to work, halving the time it takes us to do the thermomechanical simulations."

George Gabriel Chiriac,
Chassis & Safety Division,
Continental AG



“The purpose of the virtual shaker approach is to foresee testing difficulties prior to the actual test by modeling faithfully in a virtual environment the testing hardware and the structure under test.

Virtual Testing Through Integrated Sine Control in Simcenter Software

By Gilles Patanchon, ARIANE GROUPE and Flavio D'Ambrosio and Alex Carrella, Siemens PLM Software

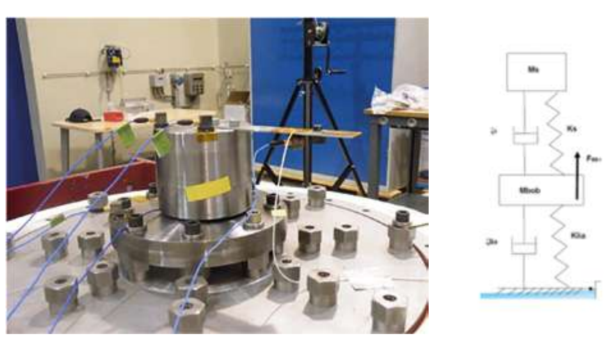
Satellites have to be tested to ensure they have the capability to withstand the extreme conditions encountered during launch and flight operations. This step is called "qualification testing" and is implemented by means of extensive laboratory test campaigns, including for instance vibration, thermal, or acoustic testing. Such environmental tests are mandatory for satellites in order to reduce the risks of failure during launch and mission. Swept-sine tests are considered as a backbone test in order to simulate a low-frequency vibration environment. Regulated by ECSS standards, sine vibration test is implemented by rigidly connecting the satellite to a shaker table and vibrating it with prescribed levels. For the base excitation to be compliant with the targeted level, the drive spectrum sent to the shaker is constantly updated by a control system. One of the difficulties of the test campaign is to optimize the control parameters (swept rate, compression factor, notching values, etc.) in order to validate the ECSS standards requirements. Until now this procedure was done directly on the test rig. The aim of virtual testing is to avoid this experimental research of control parameter using Finite Elements (FE) models to reduce costs, over testing, risks and time of all experimental campaigns.

To reach this aim, a closed-loop control model based on the implementation in Simcenter Testlab software was integrated into the Simcenter Samcef™ solver. Then the virtual controller was validated with some academic and industrial applications, including the Galileo satellite. Finally, an approach was developed to account for structural non-linearities in the virtual testing context. All developments and numerical results have been financed by the AOC (Advanced Operational Certification) Walloon research Program.

Validation and industrial case

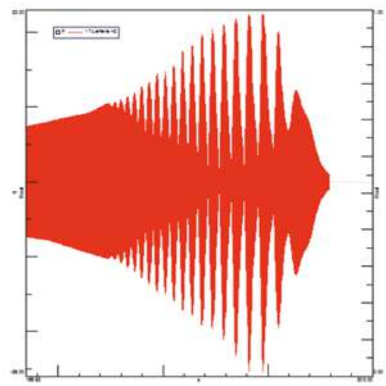
We validated the implementation of Simcenter Testlab sine control in the Simcenter Samcef solver by comparing numerical and experimental results. A free / clamped plate, shown in Figure 1, was selected as the academic structure for this task. The campaign test was executed by Ariane Group in Bordeaux. The numerical model was realized using only two degrees of freedom representing the modal properties of the beam as described in Figure 1.

The standard set-up consists of a piezoelectric accelerometers stocked at the end of the plate as notching sensor and on the shaker table as control sensor. In order to be able to control the system at its characteristic frequencies (resonance and anti-resonance), different scenarios are analyzed with different values of compression factors, sweep rate and profile control level. Figure 1 shows that the numerical



Mbob = 125 Kg
Klia = 19720 N/m
Damping = 2%
Ms = 0.1Kg
Ks = 6630 N/m
Damping = 0.7%

Figure 1: Test rig and numerical model for fixed / free beam
a) Test rig used to validate numerical approach b) Numerical model



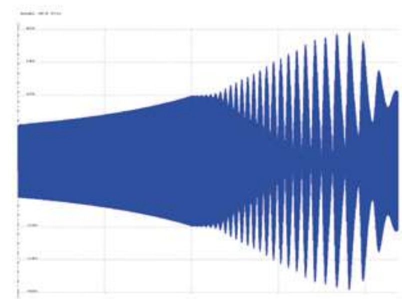


Figure 2: experimental and numerical results for unstable control case
a) Experimental results: unstable control case b) Numerical results: unstable control case

approach is able to validate and reproduce all cases with both stable and unstable control. As a reference, we can report here the instability of the control at the notching sensor (2nd DOF for numerical model) obtained with a very strong reactivity control around the resonance frequency of the structure (Figure 2).

Now we repeat the same approach for an industrial test: the Galileo satellite. The approach is the same. We consider the transversal and longitudinal configurations. The numerical approach is similar to the previous one because we focus our attention only on the first mode shape of the satellite. In Figure 3, we show the experimental set-up for the transversal configuration and the mechanical model. In Table 1, the numerical parameters used in the mechanical model are defined.

Even in this case, we were able to reproduce almost all the experimental results as shown in Figures 1 and 2 using only the linear mechanical model. In some cases, the experimental results in Figure 2 show that a non-linear mechanical model is necessary to reproduce the experimental behavior. Using the example of a transversal configuration for a sweep sine of 0.5 oct/min and without notching, we are able to control in experimental tests the vibration at the base of shaker, which is not true in the numerical approach as shown in Figure 4. Moreover, the experimental signal shows multi-harmonics of the fundamental excitation frequency. This behavior is typical of a non-linear behavior of the structure and this is the main reason why the simulation model cannot reproduce the experimental results. For this reason the next step was to show the approach that can be used in case of non-linear structures.

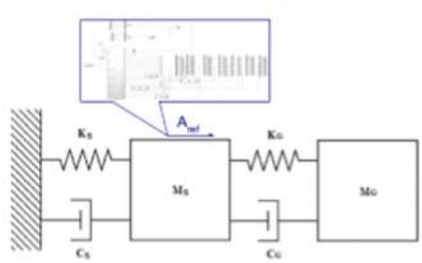


Figure 3: Tested coupled structure
(a) Experimental set-up
(b) Mechanical modelling in Simcenter integrated environment

	Mass (KG)	Stiffness (N/m)	Damping (N m/s)	Nat frequency (Hz)
Shaker	1,500	236.87 x10^3	1,88 x10^3 (5%)	2
Galileo	2,500	22,2 x10^6	3,78 x10^3 (0.8%)	15

Table 1: Physical properties of the 2-DOF linear model

Non-linear approach

Ariane Group has detected the presence of multiple odd and even harmonics in the measured time responses of the notching and control channels both for the longitudinal and transversal testing configurations. During longitudinal tests, the dispenser is connected to the expander head of the shaker by a fixation system comprising a series of fasteners. These fasteners are assumed to suffer from a loss of stiffness when subjected to a tensile load. To represent this behavior, a non-linear restoring force law is added as a parallel spring into the studied 2-DOF structural model. In this case, the values of non-linear stiffness is not validated experimentally and the aim of this study will be only related to the method and qualitative results. Figure 6 shows the modified longitudinal coupled system as well as the total restoring force between the two masses. The non-linearity is defined by a loss of approximately 50% of Galileo's linear stiffness, KG, for positive relative displacements greater than 0.4 mm. Table 2 shows the mechanical properties of the underlying linear longitudinal system. The first step of a non-linear approach consists in the harmonic response in presence of non-linearities. The non-linear harmonic response is computed using the Harmonic Balance Method and the results are shown in Figure 8 in function of the acceleration at the base.

As expected, the resonance decreases from 29 Hz for linear system to 25.9 Hz for a 1-g base level. This softening phenomenon is due to the decrease of the equivalent stiffness in function of the input energy of the system. In this case, the aim was to show how the linear control could work in the presence of non-linear structures. We choose a configuration able to excite the non-linearity within a base acceleration of 0.1 g. To avoid as much as possible controllability issues due to high reactivity, a factor of compression of 12 and a sweep rate of 1 oct/min were also selected. The notching was fixed at 1.65g.

If we study the results of spectrum amplitude, as normally done during experimental test rig (Figure 9), we can expect that the control is stable. The control profile is respected and the notching value is not exceeded. But if we consider the time domain results, we can observe that the conclusions are completely different (Figure 10). We see here that the control channel does not follow the control profile and the 2nd DOF exceeds the notching value. If we perform a FFT of time signal, we can see that the signal has more than one harmonic (Figure 11), which means that in general the control is able to act only on the fundamental harmonic. The presence of other harmonics is not at all considered for the Simcenter Testlab sine control and their presence can have an unexpected effect on the global

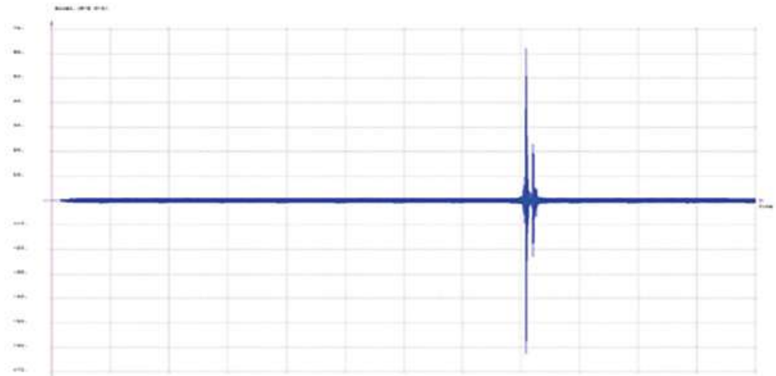


Figure 4: Numerical simulation in case of stable control (Galileo)

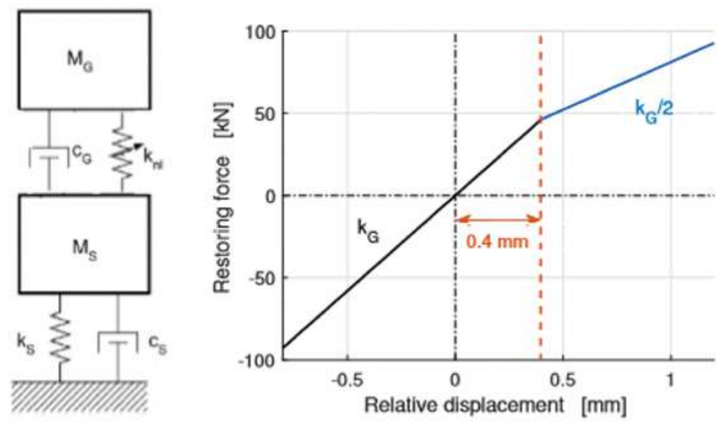


Figure 5: Modification of the structural model to account for the loss of stiffness.

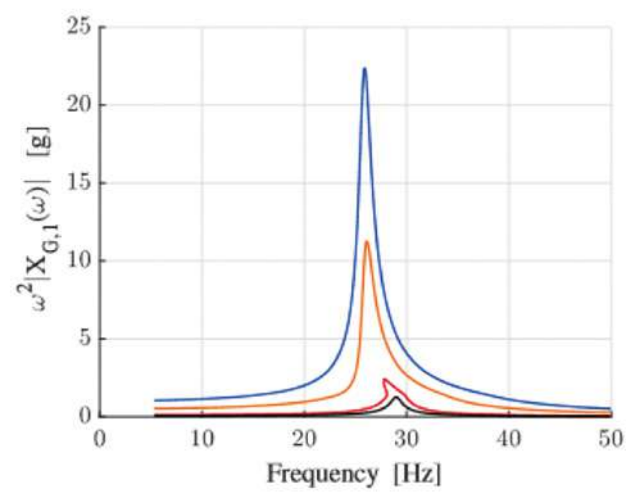


Figure 6: Amplitude of the harmonic forced response of Galileo in terms of acceleration using Simcenter Samcef' Repdyn solver for several levels of base acceleration (black: 0.05 g, red: 0.1 g, orange: 0.5 g, blue: 1 g)

	Mass (KG)	Stiffness (N/m)	Damping (N m/s)	Nat frequency (Hz)
Shaker	1,500	2.37 x10^5	1,88 x10^3 (5%)	2
Galileo	3,500	1.16 x10^8	25,51 x10^3 (2%)	29

Table 2: Underlying longitudinal linear system: physical properties

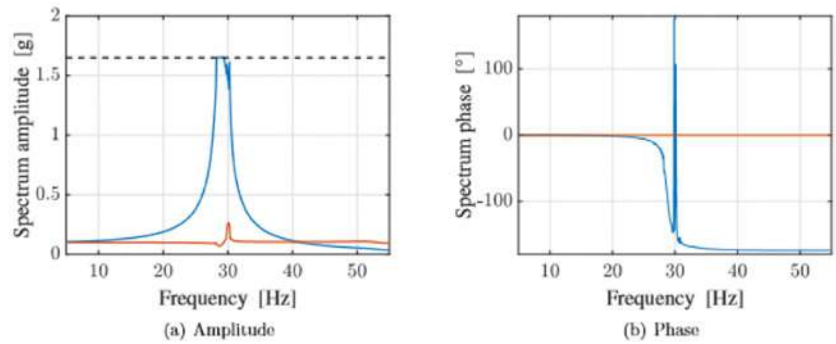


Figure 7: Spectrum verification for 0.1-g base level test using harmonic filtered with Simcenter Testlab

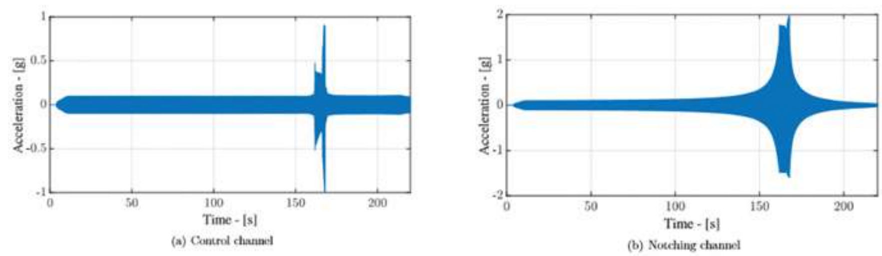


Figure 8: Control and notching channel in time domain

stability of the control. Nevertheless these non-linearities must be modeled in the simulation model if we want to extend the concept of virtual testing, even in the case of non-linear behavior of structure.

Conclusion

The purpose of the virtual shaker approach is to foresee testing difficulties prior to the actual test by modeling faithfully in a virtual environment the testing hardware and the structure under test. For this reason, a dynamically linked library (dll) of Simcenter Testlab sine control was created and fully integrated in the Simcenter Samcef solver in order to numerically predict the experimental behavior of tests. This implementation was validated for an academic case (clamped / free beam) and for an industrial case (Galileo satellite). In the industrial case, we saw that a linear approach is not enough to predict the behavior of the experimental test. In the context of virtual testing, we saw that the non-linearities can completely change the stability of the control. In this scope, we showed how non-linearities can be taken into account in a virtual testing context and how important it is to identify and quantify the non-linearities before starting the virtual testing.

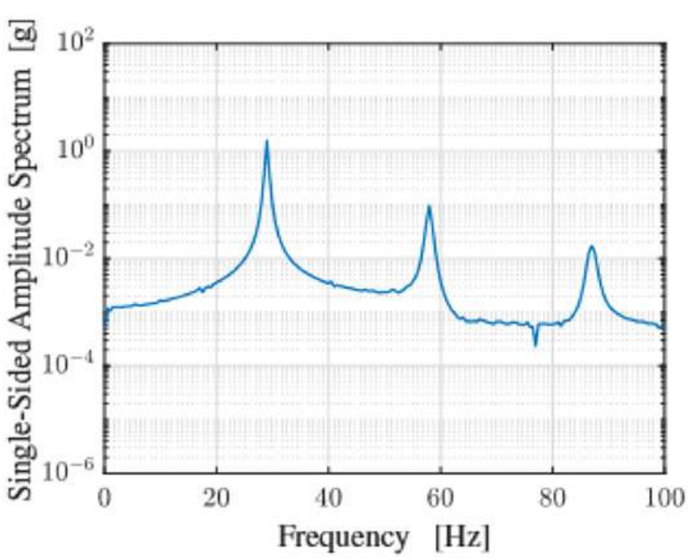


Figure 9: 0.1-g base level test, FFT over [163, 165] s of the notching channel

References:

[1] F. D'Ambrosio, A. Carrella, G. Patanchon: An Integrated Approach for Virtual Testing: Finite Element Methods with embedded sine control for prediction of base-excitation tests in the space industry, NAFEMS 2017 Conference, Stockholm, Sweden

[2] F. D'Ambrosio, S. Hoffait, A. Carrella, G. Patanchon: Integrated Solution for Virtual Testing. ECCSMET 2018, ESA Conference, Noordwjik, The Netherlands ■

Ask the Expert...

No Engineer Left Behind: Supporting engineers in an era of simulation credibility

There has never been a more exciting time to be a simulation engineer. You have better tools, and more computing power, than any engineer in history. What's more is that, while previous generations of simulation engineer were often regarded as eccentric geeks, your colleagues and customers take the results of your simulations seriously, and they play a strong role in influencing the design evolution of your company's products.

However, with acceptance comes accountability and responsibility, which means that there has never been a more difficult time to be a simulation engineer. Engineering simulation involves solving difficult physics problems, using expensive software, enormous computing resources, and highly trained engineers. If the problems weren't difficult, then it is doubtful that anyone would devote so much time and money to solving them. An uncomfortable truth about modern engineering is that there really are no easy problems left to solve.

Whereas previous generations of engineers could take some comfort in the 'safety net' of extensive physical testing to rescue them from the occasional poor prediction, CAE is increasingly the victim of its own success as simulation continues to displace hardware testing as industry's verification method of choice. Although this increased confidence in simulation is well-deserved (and has been hard-earned through many years of successful prediction), it brings with it a great deal of pressure to 'get the answer right' every time.

And while simulation engineers used to be regarded as "specialists" in a particular engineering discipline, modern industry requires a broader set of expertize: in order to meet the demands of industry, it is no longer good enough to do 'a bit of CFD' or 'some stress analyses. Solving complex industrial problems requires simulation tools that span a multitude of physical phenomena and a variety of engineering disciplines. Real-world engineering problems do not separate themselves into convenient categories such as "aerodynamics", "hydrodynamics", "heat transfer" and "solid mechanics". Only multi-disciplinary engineering simulation can accurately capture all of the relevant physics that influence the real-world performance of a product, and can be used to automatically drive the virtual product through a range of design configurations and operating scenarios.

In order to design truly innovative products, engineers like you are continually "pushing back the boundaries of the possible", operating at the very frontier of engineering analysis.

You cannot manage this in isolation, as solving a multi-disciplinary problem will often require competences outside an individual engineer's immediate area of expertize. To be successful, a modern engineer needs to have ready access to a community of simulation experts.

The good news is that becoming a Siemens customer means more than purchasing world-class software or services; it opens the door to an unrivalled wealth of engineering expertize. Our technological solutions are backed by a global team engineering analysis specialists, dedicated to helping you meet the challenges of your industry and exceed the expectations of your market.

Supporting your needs

By maintaining a continual dialogue with our customers, our aim is to identify problems before they happen and to provide immediate resolutions when they do. From the moment you place a call to your dedicated local support engineer, you are accessing one of the world's biggest resources of simulation expertize.

Our aim is to put you in touch with an appropriate local expert in the minimum amount of time, and to provide you with the advice that allows you to deliver top quality engineering analysis on time. By assigning a "Dedicated Support Engineer" to each of our customers, our aim is to develop a professional results-orientated relationship that completely understands your analysis requirements, and how the analysis relates to your business environment.

By developing this comprehensive support model, and seeing the "bigger picture" of overall business goals, Siemens engineers can help you solve more than just technical problems and, in doing so, helps keep your business in front of your competition, and ideally an established relationship with a dedicated support engineer who not only understands the engineer's problems, but can approach the right expert help whenever needed. ■

Continuous Innovation at Renault Sport Formula One

Racing team uses Simcenter STAR-CCM+ and Fibersim

Driving with downforce, drag and danger. The dynamic between grip and drag dictates design preparation for each race in the Formula One calendar. A large rear wing creates the downforce required for tight cornering at Monaco. For Monza, where too much downforce would slow the car on fast straights, a slender rear wing is needed.

"Good grip enables the car to go faster but downforce also produces drag, which must be overcome by engine power," says Peter Machin, Head of Aerodynamics at Renault Sport Formula One™ Team. "The ultimate goal is to generate a vertical force and push the tires into the ground while minimizing drag."

The workflow of a Formula One car design is a 365-day-a-year process. Throughout the season, surfaces are continually being adjusted to accommodate the track, the driver and climate conditions.

"Our car could be seen as an aerospace prototype," says Luca Mazzocco, Head of Technological Partnerships, Renault Sport Formula One Team. "We need to deploy innovation race-by-race if we want to be a credible challenger, and that can be on a weekly basis and on 21 different tracks around the globe."

70% of a car's performance stems directly from its aerodynamic behavior. Incremental improvements are made on a day-to-day basis as stiffness, weight and cost effectiveness are balanced. Not surprisingly, aerodynamics is the largest department at Renault Sport Formula One Team; it commands the biggest budget and its supercomputer produces 60 terabytes (TB) of data each week.

Aerodynamics involves both physical testing and simulation. Aside from the inherent limitations of a wind tunnel, the nature and extent of physical testing is restricted by Formula One regulations. The use of computational fluid dynamics (CFD) software is critical and, for over 15 years, the company has been using Simcenter STAR-CCM+™ software.

As the use of strong, lightweight carbon fiber is critical to a racing car's aerodynamic performance, Renault Sport Formula One Team also uses the Fibersim™ portfolio of software for composites engineering from Siemens PLM Software. This is used to manage the design, analysis and manufacture of fiber-reinforced composite parts.

Correlating data from different sources

Paul Cusdin, Head of CFD for Renault Sport Formula One Team, says, "Our challenge is to ensure that the computational domain correlates with data captured from the wind tunnel, so we can ensure that every design upgrade will actually match up with reality."

The focus is not only on speed; CFD is used for thermal simulation because an overheated car is a potential danger and must be called into the pits. On the other hand, there are clear restrictions on how much cooling can be applied to the engine during a race. CFD is also used to simulate the action on the track, particularly when another vehicle directly in front is creating turbulence that not only makes it difficult to overtake it but could lead to a critical loss of downforce in one part of the car.

Another major question for the CFD experts at Renault Sport Formula One Team is how to get the most from the tires. It is not easy to model the geometry and wake behavior of tires in a wind tunnel, especially in highspeed corners when tire shape fluctuates. Another consideration is

that within a tunnel, wind moves over the car rather than the car moving over ground.

"This area is where we have the least correlation between the physical and the computational," says Cusdin. "Yet we need to extract more from the tires, for example, by placing geometry around the floor of the car in the best way possible for aerodynamics performance."

Revealing the physics of flow

CFD offers a design team insight into what happens in the wind tunnel. "It shows the precise airflow over the car and tells us why we are getting certain results," says Cusdin. "For example, simulation shows whether a vortex is above the wing or below it. It can introduce heat, which we cannot do in the wind tunnel, and illustrate thermal interaction. It tells us more about a specific design, indicating if it is close to optimal. In short, the computational domain not only augments the physical domain, it also improves it."

In one instance, members of the CFD team were looking to incorporate the power of the fast-moving air from the exhaust to enhance downforce, but results on the track were disappointing. By further analyzing the physics, they discovered that modeling the exhaust as a steady jet rather than a series of pulses had inadvertently led Renault Sport Formula One Team designers down the wrong path. CFD solved this

"CFD gives us significant direction and I anticipate expanding our use of Simcenter STAR-CCM+."

Paul Cusdin
Head of CFD
Renault Sport Formula One™ Team

engineering challenge by simulating the pulse aspect and allowing engineers to visualize its repercussions on the airflow.

Enabling speedy innovation
"When we test different geometries in the wind tunnel, we learn whether they are better or worse than the prior design, but we only rarely understand exactly why," says Cusdin.

"Understanding the vortex created by the front wing is particularly important because the rest of the car depends on that; yet CFD has been rather poor at capturing the wake structure at the front of the car.

"However, Siemens PLM Software introduced a turbulence model within the latest enhancements for Simcenter STAR-CCM+ and now we can look at all the vortices shed off the front, side and rear and clearly see how these react with the field downstream."

The CFD team aims to calculate and recalculate design changes within a few hours so that clear information is available for designers.

"Our simulation pipeline is very simple," says Cusdin. "Other CFD software requires extensive coding, but we only write code for each new set of environmental parameters.

This is the unique advantage of Simcenter STAR-CCM+ and it means that we can create templates for the design team."

As a result, design engineers can replace any surface and continuously re-run the same simulation. By accessing templates from within the system, they do not even have to open Simcenter STAR-CCM+ or see the solver. In this way, the CFD team iterates in step with the design department as aerodynamic shapes are assessed for performance. Promising geometries are sent to the wind tunnel and possibly reassessed through CFD. Designs are released for manufacture only when results from the wind tunnel match those from simulation.

In accordance with Formula One regulations, the wind tunnel car is a 60% scale model. Because of the fast testing cycle and shape complexity, most wind tunnel model parts are

made using additive manufacturing, a technology that is also involved in making several full-scale car parts. In some cases, this process can be completed overnight, though more complex parts may take several weeks.

Using composites for lightweight strength and stiffness

Composite components are formed when pre-impregnated materials are layered into a mold and cured in autoclaves at high temperature and pressure. The result is an extremely strong, light and stiff material that contributes to both performance and safety; Composites account for 85% of the volume of a racing car, but only 25% of its mass. The low mass of composite material enables engineers to alter the position of a car's center of gravity, and this can have a significant influence on handling characteristics.

When aerodynamic surfaces are released they lack structural properties, so the first stage of manufacture involves a close liaison between stress engineers and laminate design engineers. They work together to optimize the structure of a part, determine the type of composite material to be used and organize the layers of laminate.

"Fibersim ties departments together through one single digital model, which becomes the baseline for how data migrates through design and ultimately to manufacturing," says Ian Goddard, head of technical partnerships at Renault Sport Formula One Team.

There are over 1,000 pieces of composite material in a race car, and Fibersim was first used by Renault Sport Formula One Team for the most complicated parts, such as the chassis. Goddard continues, "Having evolved the process over many years to gain control of things like the chassis, we decided to apply the same mindset to everything and introduce control methodologies on all components, from the tiniest little bracket up to aerodynamic wings and right through to our crash structures. Having that level of confidence and control through a single 3D Fibersim model, we can really improve the overall quality of the parts we manufacture."

Fibersim enables complete precision during the manufacturing process. It is

used to determine the optimal way to overlap layers and send precise instructions to the machine that cuts flat patterns. It also drives a laser projection guidance system in the clean room. This projects a green laser light into molds, indicating to the laminators exactly how and where to position and cut individual pieces of material in order to create each ply.

"With Fibersim, we really get repeatability and consistency beyond what we could achieve with simple measures and dimensions," says Goddard.

Efficiency through repeatability and consistency

"Our pipeline is a huge shortcut. It saves time, gives repeatability and the use of templates provides consistency," says Cusdin. "CFD gives us significant direction and I anticipate expanding our use of Simcenter STAR-CCM+. Whereas physical testing is both expensive and time consuming, there are far fewer limits on digital experimentation."

Machin adds, "CFD software is absolutely critical; without it we would not know how to use wind tunnel testing to improve performance. Simcenter STAR-CCM+ feeds decision making by enabling us to assess where the biggest performance gain is."

With constant technological and regulatory change, the development cycle at Renault Sport Formula One Team is both dynamic and relentless.

"We need technical partners who really want to embrace our challenge to innovate under pressure and we truly value the relationship we have with Siemens PLM Software," says Mazzocco. ■

"With Fibersim, we really get repeatability and consistency beyond what we could achieve with simple measures and dimensions."

Ian Goddard
Head of Technical Partnerships
Renault Sport Formula One™ Team

Samsung R&D Institute Delivering Safer more Efficient Battery Packs

Developing a better thermal management system

The use of lithium-ion (Li-ion) batteries has made the electric vehicle a reality, so we could see the widespread acceptance of electric mobility in the not-too-distant future. However, there have been more than a few incidents of Li-ion batteries in electric vehicles catching fire due to faulty thermal management systems (TMS) or rough-driving abuse. This underscores the importance of finding new methods for effectively and accurately designing TMS that control temperature and optimize the performance of Li-ion batteries.

To address these challenges, the Samsung R&D Institute in Bangalore, India, in collaboration with the Samsung Advanced Institute of Technology, Korea, recently presented a novel, liquid-coolant-based TMS for large Li-ion battery packs. They constructed a coupled 3D electrochemical/thermal model of the proposed battery pack. The simulation revealed that contact resistance had the greatest impact on the pack's thermal performance.

The role of computational fluid dynamics

Considering the three-dimensional nature of the flow around the cells in a battery pack and the spatial variance involved in heat generation, the practice of simulating battery packs using computational fluid dynamics

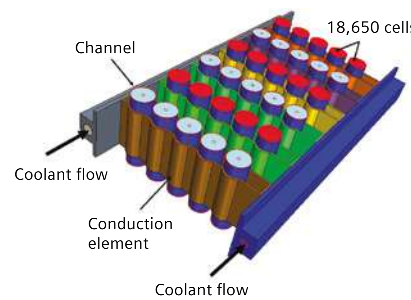


Figure 1: Geometry of the pack and the thermal management system.

(CFD) has evolved to become an effective design and optimization tool to address thermal management problems.

For the large battery packs that operate at the high discharge rates typically used in electric vehicles (EVs) and hybrid electric vehicles (HEVs), CFD studies have shown that liquid cooling is more effective than air cooling, enabling the design of more compact and efficient batteries.

Pack geometry and experimental setup

In the Li-ion battery pack presented in Figure 1, a commercially available 18,650-cell Li-NCA/C battery was used. Elements made of highly conductive metal transferred heat from the cylindrical cells to the coolant channel and, finally, to the coolant liquid (in this case, water). A test pack of 30 cells was fabricated, with six cells in series and five cells in parallel (see Figure 1).

The simulation revealed that contact resistance had the greatest impact on the pack's thermal performance.

3D CFD model

A complete characterization of heat generation was obtained by constructing a 3D CFD-based electrochemical model of the battery that could be validated against experimental results, then used to simulate and evaluate the performance of the TMS under various operating conditions.

This project used two Siemens PLM Software products: Simcenter STAR-CCM+™ software and Simcenter Battery Design Studio™ software. Simcenter STAR-CCM+ was used to simulate flow and conjugate heat transfer, while Simcenter Battery Design Studio was used to obtain electrochemical input data. This combination was used to simulate the performance of the battery pack.

Accurate temperature predictions from a single cell

The 3D TMS model was used to compute the performance of the representative battery pack. It was found the average temperature difference between the hottest and coldest cells was only 0.5 Kelvin (°K). Observing a clear pattern in the temperature rise, the authors realized that a properly defined temperature coefficient could predict the temperature of other cells based on the temperature of just one cell.

Coolant flow rate is critical

In electric vehicles, power for operating the TMS comes from energy extracted from the battery. Reducing the energy requirement for the TMS reduces its drain on the battery, thereby optimizing coolant flow rate, which is essential. The Simcenter STAR-CCM+ model revealed that more heat is stored in the battery pack in lower coolant flow velocity conditions, indicating that at lower flow velocities, less heat is transferred into the coolant.

In most battery packs, maximum temperature variation is limited to 3 °K along the direction of the flow stream. The experimental model easily met the 3 °K limit and could effectively cool the pack even at low-flow velocities.

Materials such as graphene are used in compact TMS, which is a novel but expensive material. The results in Figure 2 show the temperature rise in the battery pack using the experimental TMS are on the same order as those reported in research literature that using graphene as a phase change material (PCM) based thermal management system. Although such PCM-based TMS are compact, this new TMS does not require use of such novel materials and can therefore be produced at lower cost.

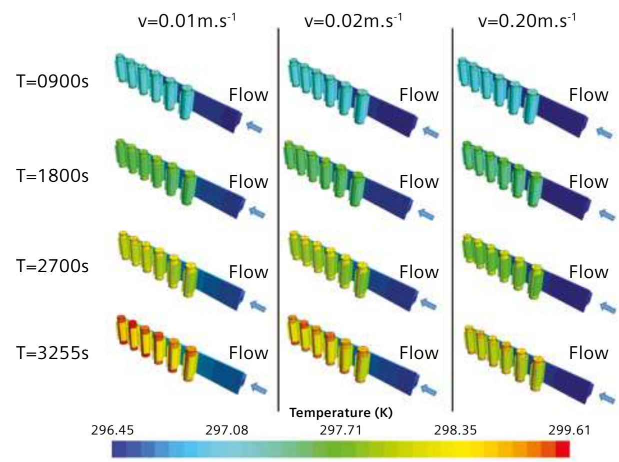


Figure 2: The temperature rise in the first set of series cells in the pack is a function of the 0.9 (°C) discharge rate and contact resistance of 0.0025 (m2.°K)/W.

Conclusion

By using the CFD-based TMS functional model created with Simcenter STAR-CCM+ and Simcenter Battery Design Studio, the results of simulations and experimental measurements were in agreement, validating the model against the experiment with greater than 90% accuracy. Representative battery packs constructed using the symmetry of the total pack were successfully simulated, together with the TMS, to lower the computational cost.

Since the TMS worked effectively and safely under stringent conditions, it is a suitable candidate for large Li-ion battery packs that are used in electric vehicles. ■

The Simcenter STAR-CCM+ model revealed that more heat is stored in the battery pack in lower coolant flow velocity conditions, indicating that at lower flow velocities, less heat is transferred into the coolant.

Geek Hub

Collaboration between two Engineers and a Marketing Specialist to Simulate a Prosthesis.

By Debbie Searle, John Wilson and Mike Gruetzmacher

Figure 1: These images show how we discussed the dimensions of this device

Every July, Hampton Court, UK is turned upside down as the Royal Horticultural Society Flower Show touches down. The usually quiet(ish) little town is transformed into a gardener's haven. Quirky touches along with beautiful planting covers every available space. When Ray Edwards, founder of Limbcare, informed Debbie Searle, Marketing Specialist at Mentor, A Siemens Business, that Limbcare were going to be at this year's event, she championed a petition for Siemens to become a Gold sponsor of the Limbcare stand. Aside from supporting a really good cause, this event gave us a really good opportunity to learn how technology can actually enrich lives.

Debbie lost her leg in 2015 due to complications when she contracted Strep A. Later that year, with a lot of hard work, she was up and walking on a prosthetic with a mechanical knee joint. These prostheses work well, but the big problem with them is that there is a very high risk of falling. Unless you land your stride in the perfect position, the knee has no resistance and will collapse, leaving the wearer on the floor.

Just over two years later Debbie progressed to a micro-processor knee. These sophisticated knees house a PCB, battery, sensors and a resistance mechanism. The software analyzes the motion and speed of the wearer and is able to adjust the resistance to give the wearer a much more natural and economical walking motion. These limbs give the wearer much more confidence to walk without the fear of falling, allowing them to get back to a more normal and active lifestyle.

Before settling on her C-Leg, another prosthetic was trialed, the first one was far too heavy. The knee alone was 1.5kg thus making her prosthetic leg weigh just over 6kg.

The second knee trialed was much lighter and allowed her to make the most of the features provided by the micro-processor. The knee analyzes gait and works out what the knee is required to do. It allows Debbie to walk at whatever speed she needs and catches her if she stumbles. It also recognizes if she is walking on slopes and ramps the resistance up or down. The brain behind all of this is powerful and working constantly. The engineering team wanted to see exactly how hard it was working and if that affected the performance of the prosthetic as a whole.

Mike Gruetzmacher is the Simcenter FLOEFD Software Specialist for Mentor. Mike created a model using the hardware and technical details provided by Debbie, and additional new questions put to her due to never

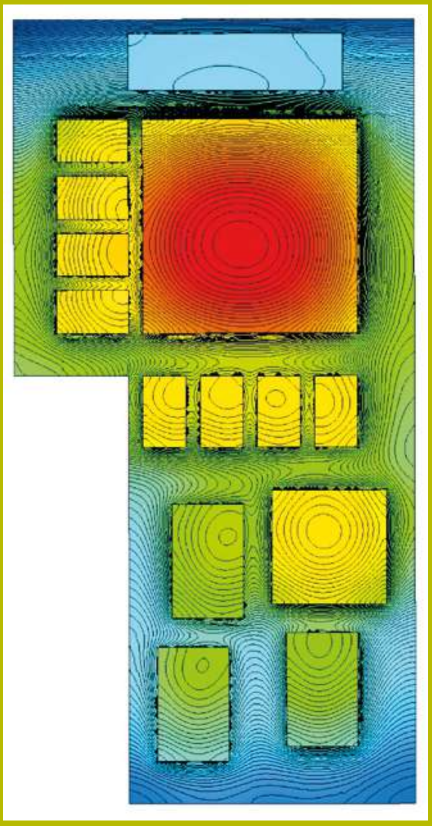


Figure 2: Board created

having to deal with a prosthetic leg before! Debbie found very innovative ways to explain the geometry and help visualize the aim of the project (Figure 1).

With the information gathered, Mike used a model found on GRABCAD and modified it according to Debbie's descriptions.

Once the first version was useable, Mike shared this model with John Wilson who created the PCB. John is the Electronics Product Specialist at Mentor, A Siemens Business.

Rather than starting from MCAD, John started with an image and with the help of Debbie, determined the approximate scales of the board and higher power dissipation components. With electronics that are battery powered there is a need to develop a solution that meets the functional requirements with the lowest power consumption possible. In these situations the critical IC components are well within their temperature requirements (reliability of electronics are strongly related to both peak operating temperatures and temperature cycling). Thermal design often involves ensuring that any exposed surfaces that may be touched are cool enough to eliminate the risk of injury. Many times the design requirements for the outer surface temperature are limited by how hot the surface feels. If a surface feels too warm, the end user could interpret this as a malfunctioning or poorly designed unit.

Mike took a GrabCAD example and performed some modifications until it looked more like the physical board. He also applied some scaling features, because the initial unit was smaller. The IDF file was imported into Simcenter FLOEFD with the Simcenter Flotherm FloEDA Bridge so that the boundary conditions (materials, heat sources and radiative surfaces) were created automatically, see Figure 3. The ambient conditions were set to typical values of 20°C and 1.01 bar. Gravity is enabled because it is cooled by natural convection.

The colored surfaces show the temperature distribution on the PCB. The arrows illustrate the airflow around these components. It is caused by the natural convection, which means the heated air moves upwards causing air circulation in this area.

The air is escaping out through the top, but as our physical example has a lid,

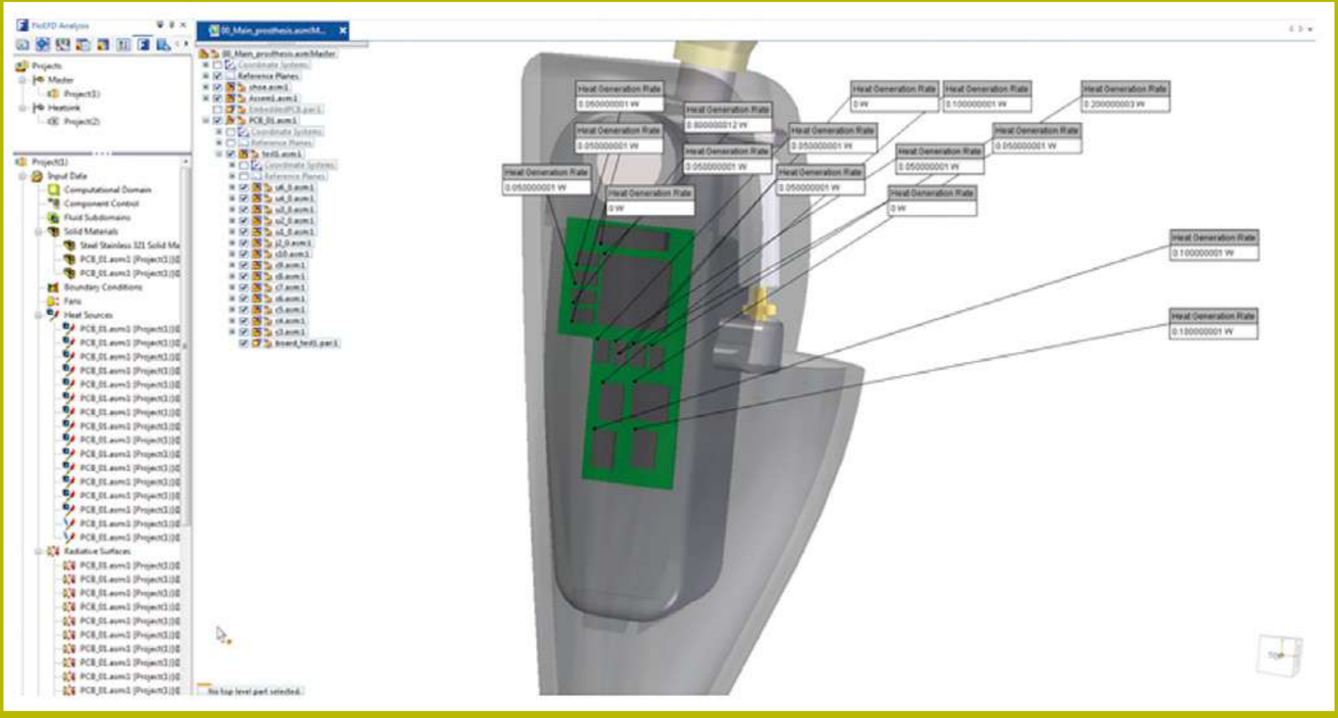


Figure 3: Boundary conditions of the PCB

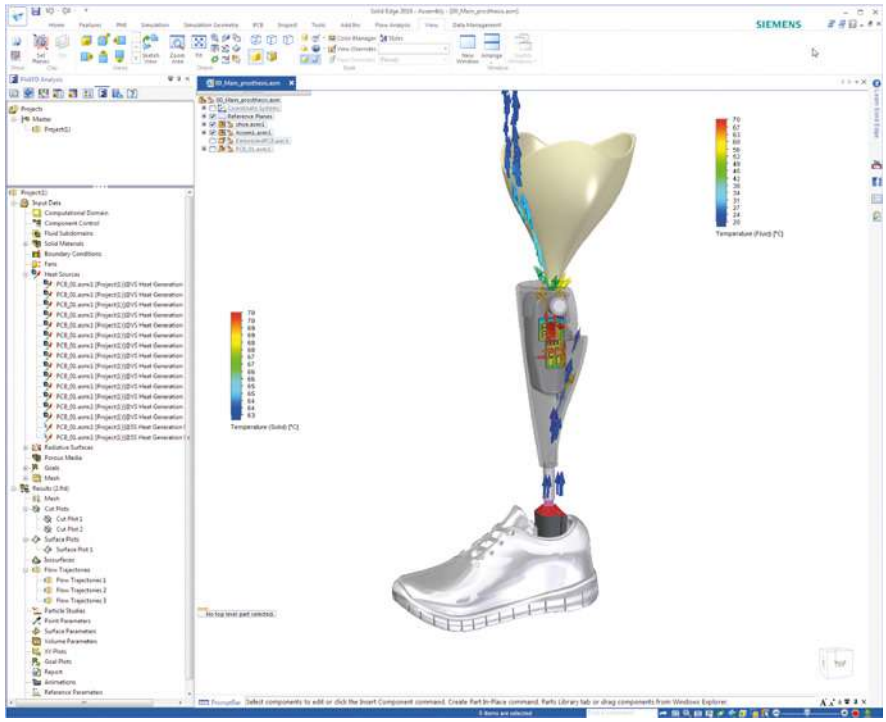


Figure 4: Overview

Mike had to create another simulation with a lid to see how this changed the results. “We expected the temperature to rise, but we had to wait to see if this would be a drastic increase."

When Mike realized there was a lid missing from the model and the air was escaping through the top, he went back into the geometry, selected the open plane and with a few quick clicks created a lid to close the model at the top. In an intermediate simulation, he obtained very high and unrealistic temperatures. As he was making a quick simulation, he’d defined some of the solid components as an insulator material. He also experimented by putting a heatsink on the PCB, and for this comparison the material properties for some of the parts was neglected. Now that the top of the leg is closed, the influence of the material properties becomes more important. The combination of the closed top with insulator material, caused the high temperatures. Once this was realized, a simulation was run for both cases with the leg defined as standard stainless steel. The maximum temperature is almost equal in both cases, it shows only a very small difference. The open version is 0.1°C cooler. What does this mean? Most of the heat is convecting and radiating from the PCB to the outer enclosure.

In summary, putting on a lid and enclosing the casing makes minimal impact. The maximum temperatures reached in either case is 38°C. The maximum advised temperature in a consumer product is around 41°C. So at a little over body temperature, Debbie’s leg, even while working at its hardest, should not cause any burns or discomfort.

The information that can be gathered is invaluable, without having to build multiple prototypes or having to redesign later on in the design process.

For more information:
www.limbcare.org

Debbie Searle Blog:
https://bit.ly/2JJCztV

References

[1] https://grabcad.com/library/below-knee-prosthesis-1 ■

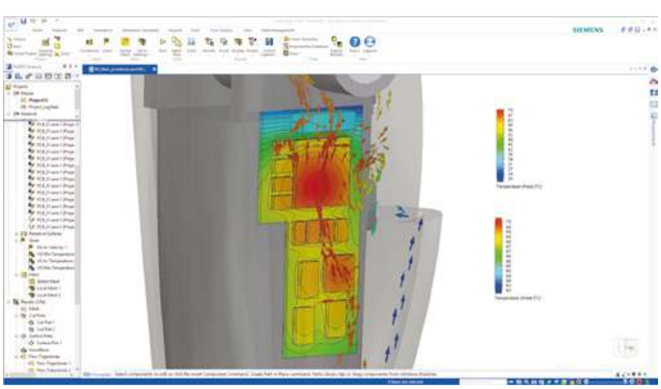


Figure 5: Temperature distribution on the PCB

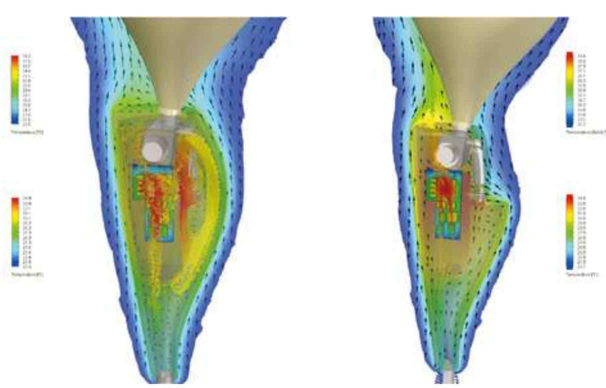
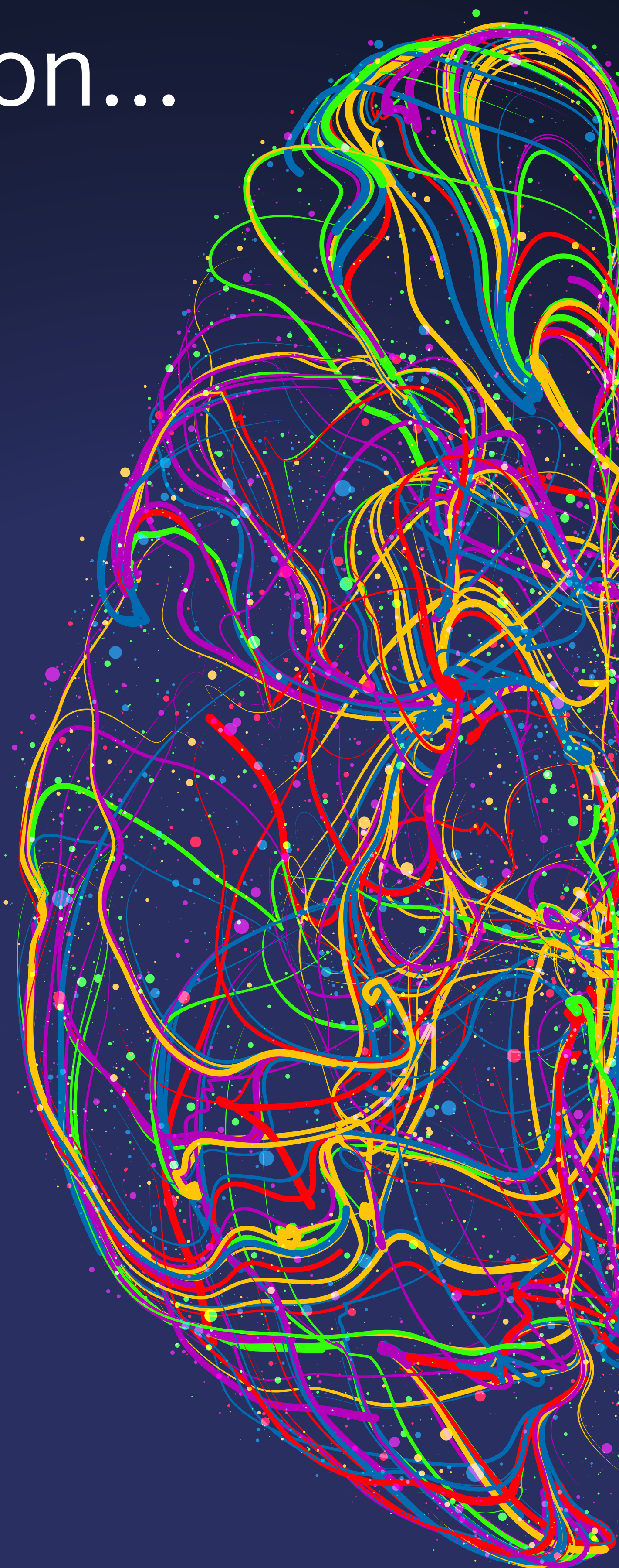


Figure 7: Comparison of the two variants

Brownian Motion...

The random musings of a Fluid Dynamicist



SMART Home gone Rogue!

It's quite exciting living in a SMART world, the utility meter that reads itself so that you never have an estimated bill or the heating that can be controlled while you are on holiday so the pipes don't freeze and you can have a hot shower on your return. The possibilities seem endless and we are apparently now very close to a completely connected home.

The big concern for these devices is their 'hack-ability' whether for honest or dishonest purposes, no doubt you will have read the stories about home assistants listening and then either serving up inappropriate content or maybe sharing it with a third party. This got me thinking back to when I shared a home with toddlers...

In my future home when the toaster makes golden toast, my fridge orders exactly the right amount of milk and the lights go on and off at the right time I am always relaxed and always prepared. Although I suspect it will be rather more like living with a toddler, the knob will have been played with so the toast will be black, the fridge will have been hijacked and contain only chocolate milk and the light will be turned on every night at 3am! ■

Director of publication: Peter De Clerck

Editor-in-chief: Natasha Antunes

75953-01 11/18