ENGIN SOFT

Newsletter

Simulation Based Engineering & Sciences

Year 15 n°4 Winter 2018

Thermal Management of the ALPIDE Space Module for Particle Tracking

Roadmap towards a collaborative multi-physics simulation workflow for **defense applications**

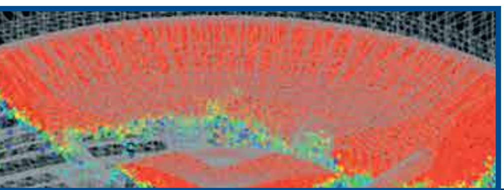
New MPS-based method to design and analyse the **cooling systems** of engine pistons is **faster and more accurate**

The multiscale and **optimization analysis** of a **lattice structure** produced via additive manufacturing

Virtual Reality Application in a **Nuclear Accelerator** Facility

Analysis of **flow characteristics** in an **urban area** using CFD

What future does **digital transformation** predict?



Contents

Newsletter EnginSoft
Year 15 n°4 - Winter 2018

To receive a free copy of the next EnginSoft Newsletters, please contact our Marketing office at: info@enginsoft.it

EnginSoft S.p.A.
24126 BERGAMO c/o Parco Scientifico Tecnologico
Kilometro Rosso - Edificio A1, Via Stezzano 87
Tel. +39 035 368711 • Fax +39 0461 979215
50127 FIRENZE Via Panciatichi, 40
Tel. +39 055 4376113 • Fax +39 0461 979216
35129 PADOVA Via Giambellino, 7
Tel. +39 049 7705311 • Fax +39 0461 979217
72023 MESAGNE (BRINDISI) Via A. Murri, 2 - Z.I.
Tel. +39 0831 730194 • Fax +39 0461 979224
38123 TRENTO fraz. Mattarello - Via della Stazione, 27
Tel. +39 0461 915391 • Fax +39 0461 979201
10133 TORINO Corso Marconi, 10
Tel. +39 011 6525211 • Fax +39 0461 979218

www.enginsoft.it - www.enginsoft.com
e-mail: info@enginsoft.it

The EnginSoft Newsletter is a quarterly magazine published by EnginSoft SpA

COMPANY INTERESTS
EnginSoft GmbH - Germany
EnginSoft UK - United Kingdom
EnginSoft France - France
EnginSoft Nordic - Sweden
EnginSoft Turkey - Turkey
VSA-TTC3 - Germany
www.enginsoft.com

CONSORZIO TCN www.consorziotcn.it • www.improve.it
SimNumerica www.simnumerica.it
M3E Mathematical Methods and Models for Engineering www.m3eweb.it

ASSOCIATION INTERESTS
NAFEMS International www.nafems.it • www.nafems.org
TechNet Alliance www.technet-alliance.com

ADVERTISEMENT
For advertising opportunities, please contact our Marketing office at: info@enginsoft.it

RESPONSIBLE DIRECTOR
Stefano Odorizzi

PRINTING
Grafiche Dalpiaz - Trento

Autorizzazione del Tribunale di Trento n° 1353 RS di data 2/4/2008

FLASH

The 2018 CAE Conference has come and gone and what an interesting event it was. For anyone who didn't get a chance to attend, or who didn't have enough time to explore everything, the EnginSoft Newsletter has collected just a sampling of the most interesting technical papers and leading-edge industrial projects that were presented at this year's conference, which drew delegates from as far afield as Korea, South Africa and the USA, in addition to the numerous European delegates. It also includes some of the highlights of the Research Agorà and some of the finalists of the Poster Awards.

A new aspect of the Conference this year was the extension of its content to appeal to business decision makers. EnginSoft's Digital Twin workshop was a full-day hands-on working session where delegates applied and implemented a digital twin two different real-world problems to support business objectives. Another initiative was the half-day Industry 4.0 Round Table discussion between entrepreneurs and business leaders about the growing pains, challenges and opportunities presented by the dawning of Industry 4.0. Feedback about the business theme was encouraging, so expect this to expand in the following editions.

The CAE Conference's format has changed from year to year in its 35-year long tradition, yet it always combines the three main ingredients (technology, the human factor, and research) from an industrial point of view. The 2019 event will see the launch of the CAE Exhibition, an ample exhibition space to enable the wide range of Engineering Simulation and complementary technologies to be presented. Exhibitors, visitors and conference attendees will be able to jointly engage with and evaluate examples of Industry 4.0-enabling thinking, research and technologies, as well as complementary technologies. EU and EC institutions, industry associations and networks, and the Centers of Excellence and Technological Poles will be invited to host their own aggregative moments around specific themes and offer these to the wider audience at the conference and exhibition.

The theme of digital transformation is fast becoming key to any company, regardless of its sector, activity or size. The implementations may differ, but this transformation cannot be avoided, or a company risks progressive marginalization (if not exclusion) by the market and, ultimately, failure. Uncertainties – and sometimes misinformation – abound in relation to timing and methods of implementation: delays can be costly in terms of loss of market share but, on the other hand, selecting an inadequate solution can negatively impact processes and result in missed objectives, creating mistrust.

The phrase Industry 4.0 has come to symbolize this epochal phase, branding it as the "Fourth Industrial Revolution". The Smart Factory is the ultimate expression of this transformation process and its objectives. It is created using new production technologies that enable collaboration between all the factors of production (operators, machines and tools), has adequate IT infrastructures, and, most importantly, is highly effective in saving energy, making extensive use of sustainable energy paradigms.

Yet, Industry 4.0 is, in fact, the project of digital transformation, which requires the nine enabling technologies of Industry 4.0 to inter-relate. These inter-relationships are the key to any effective and balanced digital transformation that is capable of improving as technology, knowledge and capability improves and that is suitable for its context. The 2019 event will seek to illustrate how and where engineering simulation, which is already pervasive in design and production processes and in the use of products themselves, is inextricably linked to the other Industry 4.0 technologies.

Thus, the 2019 CAE Conference and Exhibition will symbolically illustrate the path towards effective digital transformation adapted to different productive contexts. It takes place from 28 to 29 October 2019, at the Fiera Vicenza. Contact the organizers at info@caeconference.com for more information.

At this point, I would like to wish you and your families everything of the best over the festive season and a successful and prosperous 2019.

Stefano Odorizzi, Editor in chief

What future does digital transformation foretell?

Interview with Roberto Bruschi, head of Digital Transformation and Engineering Authority at the Offshore E&C Division, Saipem

SAIPEM

I've known Roberto Bruschi, head of Digital Transformation and Engineering Authority at the Offshore E&C Division, Saipem, for several years (from his position as senior analyst in the early Eighties, through to when he became VP for Sealines and Subsea Engineering in the new millenium), and I always enjoy the opportunity of exchanging ideas with him on Engineering – a conversation that usually features innovation, challenges and competencies. These three terms also form the fabric underpinning EnginSoft's approach to business. The following article is the result of one of our conversations (and I thank Roberto Bruschi for allowing me to publish it). I invite you to read it, imagining that you are within a Greek agorà.

Q: Currently, we read in the media that the Digital Twin is the engineering of the future. Most readers, having simply translated these Anglo-Saxon terms, stop here and begin to imagine… but could you give us a very extreme synthesis of what form a concrete digital representation of the Oil & Gas sector takes, technically speaking.

A: The digital twin (DT) is a virtual, digital reconstruction - of a physical static-structure, or a dynamic-machine, or the system of integrated machines in a production line - which allows you to monitor, control and operate the most critical functions remotely and to intervene safely when needed, so that physical interventions on site can be done only when necessary.

You therefore create a DT for operational reasons, to reduce the need for personnel in dangerous places, to increase safety by using remotely controlled machines for riskier operations, and to increase the efficiency of more complex operations that require greater experience and specialization.

A DT can become part of the design because, on the one hand, using a virtual reality of the system during its construction provides a way to assist decision making when complex systems and spaces need to be integrated safely. On the other hand, it can also affect the outcome of the design because effective management interventions can use the virtual reality to simulate, prepare and then monitor the condition of systems and machines to allow innovative choices to be made around service security, which could have an impact retrospectively on the design.

To apply a DT philosophy to what is being designed and built from the beginning is perhaps the most innovative and noble aspect of digital transformation and becomes even better if it is accompanied by a balanced use of artificial intelligence to assist – and perform – ordinary and extraordinary services.

Digital transformation involves the whole industry and, as far as I am concerned in the Oil&Gas sector, also the most remote and difficult offshore plants and the operations there. And not only that.

Digital Transformation began in the Eighties with the introduction of computers in engineering analysis, in the administrative and documental management of projects and in the same executive processes.

Today it has accelerated strongly due to the new frontiers of design and the use of advanced technologies, in particular very sophisticated sensors, big data management and connectivity.

In the Oil&Gas sector, following major accidents such as Macondo and after the vertical drop in crude oil prices in 2014, digital transformation represented a new way to reduce costs, increase safety and allow

increasingly complex operations in deep water, remote areas and harsh environments.
This transformation will change the way we work in the office, impact the timing and execution of projects, introduce new ways and means of working in the deep-water ocean and in remote arctic regions.
The DT will increasingly become a necessity for field operators, now targeting the full Life Of Field, for EPCI contractors as well, to meet the Oil&Gas Company's requirements and improve the safety and efficiency of project execution and field installation works.

Q: What about the managerial point of view?
A: It's a crucial topic. Wherever you look, you read that, once digital transformation is accepted by a company to avoid being left behind by competitors, it is the CEO that needs to lead because only this role has the power to drastically change the organization, its people and processes; only this role can decide to include new resources with specific IT training in the processes. Eliminating organizational inhibitions, integrating processes and people, and overcoming cultural barriers are all essential to ensure success and return on investment.
For an EPCI (Engineering, Procurement, Construction and Installation), the management of operations and dedicated assets which must become smart and reliable over time is the aspect that is most affected. Management has to face three types of difficulties:

- Accept protocols for data exchange that allow total transparency with customers and suppliers (how to partialize access?), which contrasts with the rigidly structured management that is necessary to protect the company contractually and to ensure economic margins;
- Protect (up to what point and how?) the technological distinctiveness of innovative and traditional components which are a tool for competition and distinctiveness, although this may contradict the aforementioned requirements for transparency with related standardization;
- Participate in different digital supply chains such as those established with their customers to effectively execute projects while targeting the Life of Field (again, the digital twin). There will be different potencies from field operator to field operator, which will require dedicated solutions, to be carefully reflected contractually. Furthermore, as the EPCI contractor develops and manages technological assets for executing field installation works, it establishes a strategic supply chain that strategically regards its own assets.

The biggest challenge project managers always face is the poor integration of the organizational structure that is a result of the historic parameters which created adiabatic silos that digitalization must now combine using other parameters. Without integration, there can be no digitization – both with regards to the organizational structure and the people responsible for making it work.

Q: In your extensive professional experience and based on these observations, when making requests to your company's suppliers, how has the approach to design evolved with regard to "simulation" metrics?
A: I think most people know that, in the recent past, simulation was the final step in verifying a solution that had been identified through traditional tools and was based on the analytical modeling

of phenomena. Today, simulation enters the design from the very beginning – even to the point of becoming a generator of solutions which have a critical process that often leads to the final solution. Competence is acquired by experimenting and one can classify the objective in data driven engineering. While I don't want to criticize the evolution that has changed the parameters of evaluating competence, I do fear the concentration of this [power] in the hands of a few individuals - and science fiction has helped us to glimpse the potential implications.

Q: So what attitude should companies that produce computer code adopt?
A: It seems evident that the future lies in multidisciplinary software platforms, always based on the logic that a simulation does not depend on a Cartesian prediction of phenomenology and an associated finalization of the model. Today, the approach of using mathematical models to compress behaviors into essential modes and reduce variables seems really outdated; I have heard about software tools with multiple scales that model the continuum starting from the atomic grid right up to our well-known finite element. As the first developers and users of simulation software, we preferred to imagine, then to model at the macro level and then to verify in the physical model, even at full scale.

Q: Finally, what do you hope for the world of scientific technology in its continuous search for equilibrium between creativity and competitiveness?
I would like to start with a thought from Hans Schwab published in the Italian newspaper "Il Sole 24 Ore" on 21/01/18: *"Technology is both a disruptor of conventions and a driving force of progress, hence the need for systemic change while digital transformation reshapes the economy and society".* If I had to extrapolate what I see around us now going in the direction of certain science fiction scenarios, I would be very concerned about the social impact of robots and AI. It is time to think first about creating a different society and then formulate a new code of ethics to accompany the digital transformation.

Interview by Roberto Gonella – EnginSoft SpA

The symphony of numerical simulation

"In carrying out the simulation, the CAE Manager, like an orchestra conductor, must be able to interpret each individual contribution and coordinate and direct all the instruments towards a comprehensive, consistent and harmonious result."

Designing, developing and manufacturing high-performance, ground-breaking products requires engineers to integrate multiple disciplinary skills into an effective and efficient process of design and optimization. Furthermore, the need to bring to market Industry 4.0-ready solutions at competitive production and operational costs, demands a systemic vision capable of implementing virtual prototypes that not only emulate functional and physical behaviors, but also support decision-making in all the product's life-cycle scenarios.

In this context, the engineer - in the broadest sense of the term – guarantees that proven simulation methodologies, combined with new enabling technologies, are effectively developed and productively used. Specifically, this means that anyone involved in numerical simulation and Computer Aided Engineering today must have solid expertise in the specific discipline and must keep their practical experience of the available tools up to date. The ability to use appropriate, proven models that are consistent with the utilization phase and that are capable of providing the necessary information is a prerequisite for contributing successfully to the planning, operational and decision-making processes.

The synthesis of all the models that contribute to represent a whole product – both physically and functionally from a systemic and multidisciplinary perspective – as a digital twin must be dynamic, consistent and flexible enough to evolve coherently with the development program or its alternative uses. In carrying out the simulation, the CAE Manager, like an orchestra conductor, must be able to interpret each individual contribution and coordinate and direct all the instruments towards a comprehensive, consistent, harmonious result.

The digital twin, therefore, needs to be as complex, evolutionary and adaptive as the current product concept in order to provide designers with an accurate virtual representation. This requires the coordinated execution of collaborative simulation processes with models of differing scale and nature and with the infrastructures that allow multiple instruments and specialized tools to communicate.

This edition of the magazine presents three new or existing, but not exhaustive, examples of digital twins from the medical field. The first, RB4F4Artist, was presented in the Research Agorà at the 2018 CAE Conference. The RB4FArtists project aims at developing a platform capable of managing a medical digital twin that provides medical staff and researchers with a reliable numerical model capable of simulating what will happen in vivo. The user sculpts the model geometry interactively using haptic devices and Augmented Reality (AR), and adapts it to each specific patient. Radial Basis Function (RBF) mesh morphing is then used to adapt the shape modifications to the numerical model to produce predictions.

The next medical example is the creation of in silico digital twins based on CAE simulations to help doctors predict pathology evolution and the effect of surgical corrections in cardiovascular patients. In this case, a patient-specific geometry was constructed, then the CFD model was parameterized using the RBF mesh morphing technique in ANSYS 19.1's ROM Builder, after which a Random Order Model was built up by conducting CFD simulations. This enabled the rapid generation of the predicted results for a surgical intervention for an aortic aneurysm in near-real-time without the use of High-Performance Computers.

The final example was a winner in the 2018 CAE Conference's Poster Award competition and involves the development of an in silico tool to reproduce the Cardioband procedure (a specific polyester sleeve fastened to the annulus ring and used to correct Mitral Valve regurgitation) by integrating image processing and Finite Element (FE) simulation. The aim of this study was to evaluate the accuracy of the in silico tool and to analyze how different positioning of the implanted anchors influences the procedure's performance.

Continuing education, constant professional development and the critical investigation of new fields of application are fundamental responsibilities for all users of numerical simulation – from companies, to academia and research.

Marco Perillo
Technical Director for Emerging Methods & Technologies (EMT), EnginSoft

Ensuring Business Benefits from the use of Digital Twins and Digital Transformation

The pragmatic simulation-powered approach, a new methodology from EnginSoft

The media, vendor advertising, academic literature, and more importantly, an increasing number of proactive companies are all discussing the business benefits that can be realized through the implementation of the digital twin and digital transformation, in general.

EnginSoft defines the Digital Twin as a dynamic virtual model (often actually a meta-model) that accurately replicates a real-world physical asset, service or process that is in use in the field. The Digital Twin (DT) changes almost simultaneously with the asset's state as it moves through its life cycle; the data that is collected -- from the asset and about it -- at all these stages is constantly fed back into the DT in a circular loop.

DT and can be used to track that asset's performance: for optimization, to identify critical performance issues for predictive maintenance; to explore the impacts and effects of different variables and dynamics; and for new asset ideation and creation.

As such, the DT represents the new and concrete integration boundary between a real physical asset (product/service/process), its numerical model (or meta-model), and engineering simulation, in a project that involves the entire lifecycle of that asset. The data collected and fed into the digital twin helps companies to deepen their knowledge about their assets' performance and the potential there is for quality improvement and for new product development purposes.

Giulio Cenci, High Value Proposition Manager at EnginSoft, states: "The DT is no longer in the exclusive domain of big businesses: The cost of the enabling technologies for Digital Twins -- sensors, communications, analytics and simulation – have reduced to a point where they can be deployed for almost every asset."

Ensuring a Return on Investment

There are many moving parts in the implementation of a digital twin and to ensure a return on investment is not always easy, which is why it is not a task to undertake lightly or without the right partner.

According to Cenci, the use of digital twins needs to be underpinned by sound business strategy that focuses on a specific business target and its assets before digital modelling begins. He says, "The true benefit of Digital Twins can only be realized by aligning specific business objectives with the right tools and methodology and combining this with strategically choosing the right people with the right skills and experience to advise and assist you in the process."

This is where EnginSoft's approach sets it apart. The company's strategy for assisting clients is called the Simulation-Powered Digital Twin. Cenci explains, "It is deeply effective because it starts with a clearly defined business case for the DT application in the customer's organization. Once the business imperative is defined, a specific methodology is implemented to identify the specifications of the simulation-based approach and its correlation with the real-time data from the operations of the specific asset and the business areas that will be affected."

The phenomena related to the asset are fundamental

"EnginSoft's methodology places great emphasis on the knowledge of the phenomena related to the asset, going beyond purely statistical and data correlation approaches, in order to really add value to the company's growth," he says. "It identifies the key technical factors that create value for the business, and then represents them with digital models that are not necessarily complex. This tradeoff between complexity and business benefits is key. Even short projects can substantially increase business performance by bringing simulation closer to operations," he explains.

He says that the EnginSoft's methodology is only possible due to their staff's engineering and simulation skills. The company's eight Italian business units and six foreign subsidiaries house skills of enormous technical value that can be coordinated in customer-driven, multidisciplinary projects: "Our highly-structured technology transfer process and our digital-twin-driven focus on business, represent the greatest value for our customers."

For instance, the economic driver may be to eliminate unplanned downtime in a production plant. For such a complex situation, EnginSoft

will determine which simulation tools, methods and technologies are required based on an inventory of the state of technology (tools and technologies) deployment across the departments of the business affected, from product development, to supply chain, operations and management.

Another economic driver may be to optimize asset performance to increase competitive advantage. This situation analysis is fairly simple, but always starts with an inventory of the tools and technologies already in place, to determine what needs to be added to enable the creation of a digital twin to effectively assist the company to meet its business objective.

Another complex assessment was required for a company that wanted to create new revenues and generate new profit areas as a result of a new digital value proposition. This involved multiple assets and business units that needed to be integrated into the DT, as well as the third-party partners and other channels involved in the digital supply chain.

To date, actual concrete experience in the use of digital twins worldwide is still limited to the initial phase of a learning curve that offers extreme potential for improvement to those organizations that have embraced these techniques in a business-focused manner.

Due to its technical simulation product development background and its position with important technological partners such as Ansys and PTC, EnginSoft is the ideal company to partner with to implement DT projects because its approach bases these solidly on the customer's historical engineering practices and indissolubly merges this knowledge with operations and business insight.

Cenci adds, "In fact, during the course of 2018, EnginSoft organized round tables and operational workshops with the top management of its largest Italian and European customers to disseminate knowledge about the potential and criticality of the digital twin, encouraging dialogue between these managers and their technicians."

DT is not just a process, but a comprehensive multidisciplinary culture

The DT is not simply a corporate technical process but represents a comprehensive and multidisciplinary culture that combines engineering, processes, data and people into a new ecosystem capable of growing business through substantiated decisions. The DT draws together managers, technicians and operational staff in data collection and decision-making, and thus becomes the basis of multidisciplinary simulation and technical communication platforms that create a Smart Company Environment targeted at dramatically increasing business through an integrated approach.

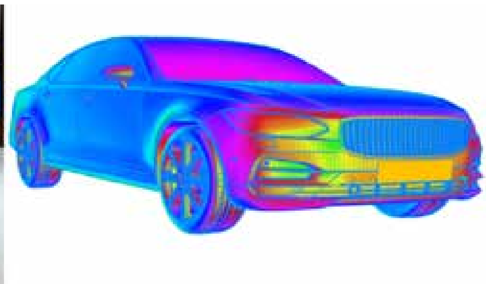
Cenci adds, "Every market sector is already being impacted by these reflections, not least medicine. There were at least six papers presented at the 2018 CAE Conference that addressed the use of the digital twin in the medical field. These techniques have been used in the study, diagnosis and treatment of the human body for years, thanks to the influence of the major international universities."

He concludes, "When the DT is generated with sound simulation-based sciences, it enables all these decisions and their consequences to be virtually tested so that more informed decisions can be made faster and better."

The Digital Twin is an enabler that assists companies to tackle all the current technological challenges that the market imposes while simultaneously effectively supporting product development and the introduction of new business models. Electrification in the automotive sector, predictive maintenance for Aerospace&Defense, and safety and profitability in the oil & gas sector are just some of the challenges that can be confronted with the Digital Twin. The following are some brief highlights of global trends where Digital Twins are already being applied:

Automotive Electrification

An article by Connor Cislo and Nao Sano, published in Bloomberg BusinessWeek, states that the electrification of cars may spell the demise of thousands of jobs in the automotive industry. The article quotes Janet Lewis, an industry analyst at Macquarie in Tokyo: "As with all technological change, there's winners and losers. You'll have some suppliers that are able to develop some of the key components, but if you're making mufflers you may not be one of the winners." The article also states that research in the German car industry conducted by the Fraunhofer Institute for Industrial Engineering echoed this statement, finding that even allowing for all the jobs that would be created by EVs, 9% of Germany's automotive jobs would be lost if electric motors drove just one-quarter of all vehicles.



Aerospace & Defense

Lockheed Martin has a strategy in place to generate digital twins of nearly all its products, processes and tools to enable it to tweak every phase in its aircrafts' life cycles. Digital twins are also being used in the aerospace industry to save resources with prescriptive and predictive analytics, according to Carlo Gutierrez and Alex Khizhniak of Altoros. They go on to say that today's computing power allows engineers to create digital twins of very complex systems and then feed them with real-time data: "NASA has used digital twins to monitor its space stations and spacecraft to ensure crew safety." They add that "digital twins allow developers to break and blow up stuff in the cyberworld without loss of life or resources."

At a meetup sponsored among others by Altoros, Robert Benson, marketing director of Mitek Analytics, spoke about how digital twins are also being used to optimize the performance and reliability of airplanes: "Modeling asset performance across a fleet is much faster than in real time."
He stated that performance digital twins help to collect Industrial Internet of Things data from deployed assets, model asset performance across the fleet based on operational data over many time periods, and then use prescriptive analytics to apply fleet-wide energy efficiency calibrations, condition-based maintenance, availability, and safety adjustments.
He said that reliability digital twins, instead, extract and deeply analyze information based on maintenance and repair (MRO) data that has been gathered over time to create a predictive model that is used to identify negative factors at both system or component level that lead to failure, which allows optimization of MRO to save costs and improve system availability.

Oil&Gas

"Digital Twins are used for operational reasons, to reduce the need for personnel in dangerous places, to increase safety by using remotely controlled machines for riskier operations, and to increase the efficiency of more complex operations that require greater experience and specialization," stated Roberto Bruschi, vice-president for subsea sea lines at SAIPEM ENI Italy, and representative of ENAU, the Offshore Engineering Authority.
He states that a digital twin can become part of the design because, on the one hand, using a virtual reality of the system during its construction provides a way to assist decision making when complex systems and spaces need to be integrated safely. On the other hand, it can also affect the outcome of the design because effective management interventions can use the virtual reality to simulate, prepare and then monitor the condition of systems and machines to allow innovative choices to be made around service security, which could have an impact retrospectively on the design.

In the Oil&Gas sector, following major accidents such as Macondo and after the vertical drop in crude oil prices in 2014, digital transformation represented a new way to reduce costs, increase safety and allow increasingly complex operations in deep water, remote areas and harsh environments.
This transformation will change the way we work in the office, impact the timing and execution of projects, introduce new ways and means of working in the deep-water ocean and in remote-arctic areas.

A collaborative mind set is fundamental to building Industry 4.0

Alfredo Maglione, president of Optoi Microelectronics, was one of the panel members of the Industry 4.0 Round Table session at the CAE Conference from 7-8 October 2018. In this interview he discusses the challenges and opportunities for Italian companies and SMEs around Industry 4.0.

Q. What is the most important aspect of Industry 4.0 to customers?
A: I strongly believe that ROI is the most important criteria. In the first push to Industry 4.0 in Italy, the early adopters achieved ROI by means of the fiscal benefit offered by the hyper amortization tool – in fact, this was the key reason for the first burst of investments. But, the most important long-term ROI that companies should seek from Industry 4.0 comes from cost reduction, increased production quality and quantity. Any Industry 4.0 investment should give the entrepreneur more money back from better quality and higher output from the production line.

In Italy, the sensors that are currently available are not yet able to help us reach these objectives, while in Germany they already can. There are Italian supplier companies developing new sensors, new interfaces and new measuring systems for the Italian market and I believe that in the next two or three years, we will begin to see Italian sensors and systems on the market. At the moment, though, the missing parts are more related to software and data analytics.
Industry 4.0 consists of nine technologies spanning from the individual sensors and the Internet of Things (IoT), robotics, big data, and cloud computing, to additive manufacturing, high performance computing, and artificial intelligence, all of which are more or less available. In my opinion, however, the limiting factor is data analytics – we need innovative software with data analytics to allow predictive maintenance. These are the pieces that are essential ensure the ROI to customers. Only when the full system is available from the sensors through data analytics, to the big data, high performance computing and so on, only at that point will the Italian Industry 4.0 become a reality. Where Germany already began moving to Industry 4.0 about five years ago in 2012/2013, Italy has only started in the past year or two. As a result, German suppliers already have five years of experience and learning and so they have been able to complete their offer. German companies are beginning to offer their products in Italy, but Italy needs more Italian companies to compete in the market and for that we still need to gain some more years of experience with the technologies.

Q. Can you explain in more detail what you mean by the data analytics and the software being the missing pieces?
A. Producers of equipment are designing more and more sophisticated machines with built-in sensors and monitoring systems. When they sell these new machines to customers, it raises the issues of data property and data security. From our experience over the last few months, we have come to understand that you can classify two main needs for this data.
The first is the need of the producers of the equipment to have access to the data about the piece of equipment or machinery that they produced, to see if it is working properly. They need to be able to monitor the equipment remotely, collect the data and then analyze it to determine the need for predictive maintenance to avoid down time for their customer.
The second is the need of the buyers the equipment, in other words the CEO or technical managers of the customer companies. They

need to access, protect and control the specific data about their own products, about their intellectual property or in other words about the product that they use the producer's machinery to manufacture. So, if this customer company manufactures a mechanical part using Industry 4.0-enabled equipment, they need to protect and secure their product-related data about that mechanical part.
This is where software becomes very important – both to support the classification of this data and in analyzing it. A lot of data can be stored in the local memory of the sensors and the software can allow this to be accessed only by the machine's producer, while the software can transfer the other data via wired or wireless communications, to the customer company's servers for analysis where it is accessible to the technical manager or CEO.

It is this software to manage the data security and the data analytics that is one of the missing parts in my opinion.

Q. What do you see as the biggest obstacle to the application of Industry 4.0 in Italy?
A. Quite simply it is the lack of collaboration. The creativity of Made in Italy is world renowned and Italy is a fabulous country, but we have some limiting factors. What is missing is the ability to work as a team – there is a poor culture of collaboration, generally. This is also present in the approach to Industry 4.0. Italy is full of SMEs working by themselves, while the central and northern European models are demonstrating that you need to work together in a team, in a networked approach with other companies that have complementary products and services.
SMEs need to collaborate more instead of viewing everyone as competition. This is a big weakness of the Italian system. We need to support the SMEs and help them to see the value in collaborating with complementary SMEs to offer a full Industry 4.0 system.

A second obstacle is the gap between academia and industry. It is important to bridge the existing separation between the educational system, universities and the working world. Generally speaking, in the past there has not been any substantial communication or collaboration between universities and private industry, although there are obviously some very good examples of exceptions. Italy must be braver in tackling and confronting this gap.

People in the academic world are focused on teaching and on scientific research, mainly to advance each individual's private academic career. Businesses on the other hand focus on creating products and profits and generally are not interested in sharing with the rest of the industry. So there has been this cultural barrier to collaboration between private industry and academia with each following his own path. Instead, in Asia, for instance Japan, this collaboration is very present and active. The same is true in Germany where universities, research institutions and companies work together in public-private partnerships to develop innovative products and services. We also see the same thing in the USA, for instance in California with the universities of Berkley and Stanford etc. When you open your mind and visit other countries, you often see successful partnerships between the public and private sectors. Italy has to make up for lost time and create more of these collaborative projects.

Q. What needs to be done?
A. We need to create the right environment with the right approach and a tested methodology. In order to improve trust, SMEs and entrepreneurs need to see and understand that it is a win-win model. This requires communication. Secondly, the potential partners need to sit around the same table in order to find the opportunities to work together. Then they need to develop a win-win partnership agreement within which to develop products together for Industry 4.0. While this approach is very well developed in western countries, it is not very present in Latin countries.
Optoi has been working with Confindustria to promote the concept that in order to bring the Italian value proposition to the market, SMEs need to collaborate more, and they need the assistance of universities and research organizations and of the public administration.

One of the ways this is being encouraged is through the creation of public-private partnerships in centers of excellence or poles that are proving to be an important way to help SMEs to learn to work together. They are the result of several visits over recent years to many innovating ecosystems worldwide which have shown us, again and again, that the winning factor lies in collaboration between public and private organizations.
In some restricted areas of Italy, like in Trentino, there is a strong culture of collaboration. Trentino is a local autonomous province where the public sector has invested heavily over the past 25 years in building collaboration between top universities and local and national industry for the joint development of innovative solutions.

One of these projects is Meccatronica in Rovereto. This has been a five-year project of intense public-private collaboration between private sector companies like Optoi, together with industry associations, the university and research centers in Trentino, to help SMEs in the mechatronic sector, and it is now delivering some important results: around 40 innovative companies and startups have chosen to open and base their R&D departments, offices and teams there.

Another project in Trentino is "Progetto Manifattura" which is a public-private partnership focused on green and clean technology involving all the main private associations such as Confindustria, the artisan associations, and ConfComercio, together with the University of Trento, and the research organizations the Bruno Kessler Foundation and Trentino Sviluppo, all of whom are working together on a project to create the technology for smart buildings. We have seen some great progress from comfort sensors, to the IoT, and energy efficient systems focused on Building 4.0 technologies, the equivalent of Industry 4.0 in this sector.

As a result of these efforts to increase collaboration and partnership over the past 25 years, the province now boasts one of the best universities, many smart specializations, several centers of excellence and many innovative projects. It is winning many Horizon2020 projects and it has created many more innovative startups. So, this collaborative ecosystem has successfully created more win-win projects, more open innovation projects in existing industry, and more innovative startups.

Q. Why has Trentino been so successful at encouraging collaboration?

A. In my opinion, it is because the province represents a good compromise between the positive aspects of Italy and the positive aspects of the Germanic countries. As a border area, there is a sort of more open mentality that seeks to take advantage of the very strong creativity of Made in Italy on the one side, and of the very systemic vision and strongly team-based approach to networking between the public and private sectors that we see in Germany and central Europe. I think this is mainly a matter of mentality and of the systemic vision and culture. For geographical and historical reasons, Trentino has both these aspects and so we were able to build up a local ecosystem that is quite good.

Q. What can an SME or entrepreneur reading this article do to get started?

A. Any SME that is really interested in collaborating should start visiting the technology poles and centers of excellence around Italy, such as Meccatronica in Trentino, the IIT in Genoa or the Area Science Park in Friuli Venezia Giulia. These are centers where you can see how collaboration works in action. At these centers you find laboratories and research centers where universities and industry collaborate and often there are also startup accelerators. For instance, Industrio is the privately funded startup accelerator based at Meccatronica that focuses on helping startups in hardware, mechatronics and industry 4.0, in collaboration with Trentino Sviluppo, the Bruno Kessler Foundation and Confindustria. It has built up 10 innovative startups over the past four years, helping them to prototype and develop their own products through collaborative efforts. There are now ten industrial groups investing in innovative startups, so the first and most important step is to visit these poles and see first-hand what they have to offer.

A next positive step is to visit your own local associations like Confindustria and become involved. Often, they already have industry initiatives in place that investigate ways to collaborate to create strategic projects.

Q. What is your outlook for an Industry 4.0 Made in Italy?

A. I am very, very optimistic. Italy is very beautiful and has many positive points, but it is lacking some parts. I believe we can improve the economy and grow our companies if we use this approach. But we definitely need to do a lot of mental work. While companies in Italy are well positioned when you measure them in terms of productivity the number of pieces produced by SMEs is very high, but we need to improve the systemic and mental approach. I believe the way to help Italy to grow is to keep its positive aspects – the creativity and design strengths – and import the soft skills, the mental approach and the more systemic vision and culture that characterizes Germany and central Europe. It is faster and easier to copy these, than try to develop them here by ourselves. I think an ultimate solution to many of the limiting factors lies in a real "United States of Europe" where we try and join the positive aspects of the different European countries together to grow the economy for everyone.

Roadmap towards a collaborative multi-physics simulation workflow for defense applications

Multiple redundancies in the current use of CAE analysis, the lack of priority criteria setting among different physics, and the concurrent running of disjointed models lead to costly and time-consuming loops and reworks, belated design changes and significant loss of data, which increases with the number of revisions.
A stepwise approach was adopted to implement a collaborative simulation workflow for Defense and Aerospace applications. Firstly, the team assessed the existing use of CAE, defined the objectives and issues, and identified the gaps to be overcome to achieve the desired outcome.
Then, a multi-physics workflow, spanning from the requirements to the design validation, was defined. The use of parameters was enhanced. Test cases including thermo-mechanical, electromagnetic and fluid-dynamic phenomena were used to experiment with the emerging practices.. Once mastered, this workflow is expected to scale rapidly and to improve the way products are designed and developed.

Introduction

Defense and Aerospace applications greatly benefit from rapid workflow simulations of the different physics that influence product performance in very challenging operating environments. A typical defense application demands that engineers consider the multiple limitations of different physics and find the optimal trade-off among them. The simultaneous presence of electromagnetic, thermo-structural and environmental limitations calls for a Simulation Process and Data Management (SPDM) tool that allows the numerical models from different physics disciplines and workgroups to be managed concurrently.

An SPDM coupled with a PLM system allows users to share the same starting CAD model as well as all the analyses' results during the design and simulation processes. In this way, all the design engineers (e.g. electromagnetics, mechanical, etc.) can share the same models, avoiding the costly loops and reworks, belated design changes and significant loss of data and time that often increase with the number of revisions (workflow optimization).

Simulation Workflow

The existing workflow has each group work without the establishment of any hierarchy of priorities between the simulations of the different physics (i.e. giving greater priority to an electromagnetic simulation over the mechanical one). Another problem is that an electromagnetic model may be based on a CAD revision that is older than the one being employed by a mechanical model: in this way, the workgroups end up using different revisions to analyze the product, which causes errors and lost time in product development.
This existing workflow was analyzed after understanding the interactions among the workgroups and a new workflow, depicted in Figure 1, was proposed that combined both the PLM system and the SPDM.

The proposed new workflow would be divided into five main stages:
Preliminary Design
- Stage 1: Preliminary design;
- Stage 2: CAD merging and parametrization for numerical analyses;

Fig. 1 - Proposed new workflow for using SPDM and Multiphysics simulations.

Design and Numerical Analyses

- Stage 3: CAD simplification and construction of the numerical mono-disciplinary models by each design workgroup;
- Stage 4: Identification of a common post-processing methodology for all design workgroups;
- Stage 5: Integration of the mono-physic models to construct a multi-physics workflow;

Stage 1: Preliminary Design

The preliminary design phase is carried out using ELT's existing workflow. Once the project requirements are received from the System Engineer, the workgroup most interested in these requirements designs a preliminary CAD model. If there are already conflicting requirements in this phase, a preliminary study using the main project objectives is carried out synergistically by the groups most concerned. This study generates a functional geometric model (.stp). The System Engineer coordinates this phase. Based on the main functional requirements to be achieved, it is possible to identify the technical group – and hence, technical manager – that will be most involved. In the subsequent phases, this technical manager will coordinate the interfaces with all the other technical groups' managers, verifying the informatics compatibility of the various CAE models, and facilitating their integration for post-processing and for the possible implementation of a Multiphysics workflow.

Stage 2: Geometry unification and parametrization for numerical analyses

In this phase, the Mechanical Designer starts with the functional geometry realized in Stage 1 and uses it to define the areas/volumes of relevance within which each group can then operate, as well as the areas of interface between the different sub-systems. Furthermore, all the operational units discuss the general objectives of the developing project. In this phase, all the data are collected from the system, its components and sub-systems with the necessary information and prerequisites to enable the design teams (PCB, RF, MW, Mech., etc.) to start their detailed analysis. Using the acquired information, the Mechanical Designer produces a more detailed assembly CAD and, if necessary, also defines a set of preliminary parameters for each part of the system. The assembly CAD is then published on the PLM system for the individual design groups to use or modify according to their needs, to conduct their own numeric analyses and then check the performance of the product (see Stage 3). The CAD generated in this phase is not directly aimed at performing Multiphysics analyses (although this is not a possibility to be excluded, a priori). In fact, if the engineers recognize the necessity for a direct Multiphysics approach at the preliminary design phase, a properly constructed CAD can be used to develop a dedicated Multiphysics analysis and verification workflow.

Stage 3: Geometry unification and parametrization for numerical analyses

In this phase, the design groups independently perform their single physics numerical analyses on the Unique CAD baseline model provided by the Mechanical Designer.

Stage 4: Post-processing and review of the multi-physics simulations

Phase 4 consists of identifying a common methodology for collecting, interpreting and
manipulating the results obtained from the various numerical simulations, based on which key decisions can be taken to overcome the two problems that were foreseen for this stage. This activity aims to control the whole project at the system level, implementing and managing the analyses in a single workflow to find better solutions by automatically moving from the mono-disciplinary to the multi-disciplinary level (see Stage 5).

Stage 5: Integration of the mono-physic models to construct a Multiphysics workflow

Phase 5 consists of the construction and execution of the Multiphysics numerical model. The complete workflow is processed by importing the individual ANSYS projects from the different workgroups into a single ANSYS Workbench project. If the analyses in the previous phases of the design were performed without ANSYS, these models can be integrated into an ANSYS Multiphysics workflow by means of appropriate work-arounds, interchange text files or customizations.

Test case 1: workflow for mono-physics simulations

The test case presented below demonstrates a multi-physical approach for a submarine antenna. This test case served to evaluate the effects on electro-magnetic and thermo-structural performance from varying the geometry of the external frame, by following the new approach shown in Figure 2. In this case, the System Engineer defined the functional and performance parameters of the whole system, identified the interfaces with the external elements, and indicated the modeling dimensions and desired performance required by the various operating units. Since the monitored thermo-structural and electronic performance were completely independent of each other, there was no priority for any of the technical workgroups. The operating units could carry out their simulation activities autonomously, respecting the input data defined upstream (materials, dimensions, interface areas, etc.) and the output data to be monitored. The Antenna Engineer and the

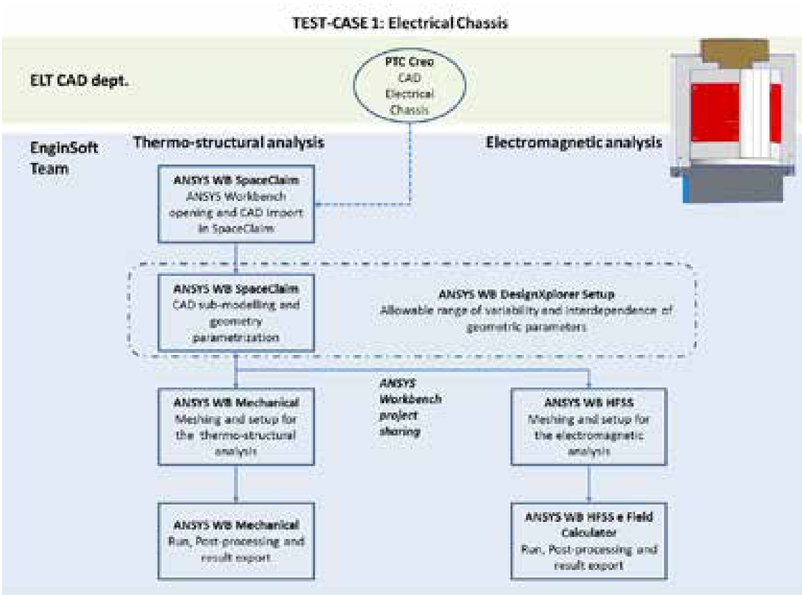


Fig. 2 - flow-chart of process for the test case 1.

Mechanical Designers discussed the functional aspects of the module and provided data to the various operating units for the mono-disciplinary analyses. In this specific test case, the Electromagnetic Design and the Mechanical Design workgroups shared only the incoming parametric CAD. Each workgroup independently performed the CAD changes necessary for their CAE analyses.

Test case 2: workflow for Multiphysics simulations

The workflow for test case 2 was similar to that of test case 1 for generating the Workbench project with relation to the import, simplification and sharing of geometries. However, where the High Frequency Structure Simulator (HFSS) module and the Mechanical module only shared information at the CAD level in test case 1, in this case communication among the various physics was established at different levels:

- Geometry: the thickness of the radome (shown in green in Figure 3) represented an important variable both electronically and structurally;
- Materials: the composite materials chosen for the construction of the radome had significant relevance both for the electromagnetic response and for the structural one;
- Results and boundary conditions: the results obtained in the HFSS environment were used by the Mechanical model to set the thermal power field generated by the printed circuit as a boundary condition.

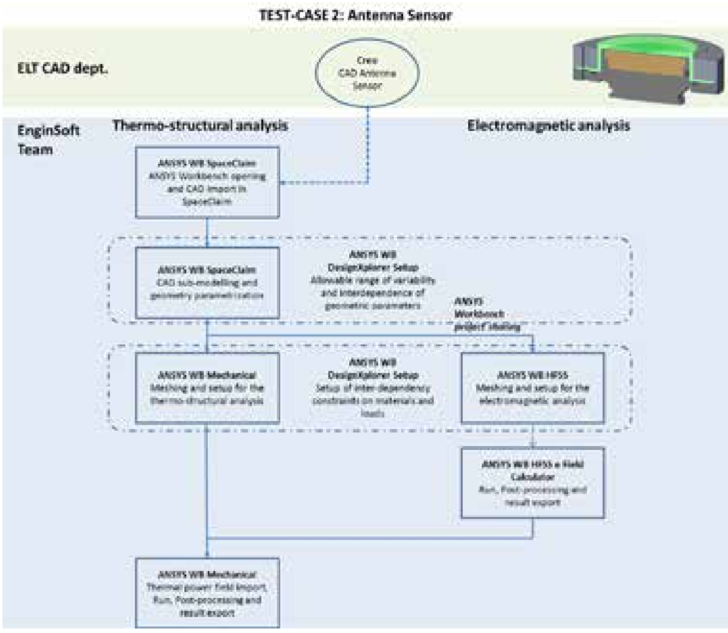


Fig. 3 - flow-chart of process for test case 2

From the electromagnetic point of view, the generation of the output data followed a similar methodology of test case 1 (i.e. the processing of output data took place in the “Field Calculator” and “DesignXplorerSetup”). In particular, since the value of the k-th percentile of the Radar Cross Section was determined, analogous quantities were generated and shared via “DesignXplorerSetup” by means of the HFSS module’s post-processing tools. In terms of electromagnetic analysis, this was a standard calculation of the radiative parameters of an antenna for which it was not necessary to provide different guidelines to the “single physics” analysis method currently implemented in ELT.

ELT’s existing electromagnetic analysis in ANSYS HFSS already contains the basic calculation guidelines. As explained in the previous paragraphs, the bottleneck caused by the timings and means of sharing the models in the past was mitigated in this case by the use of a shared model, both in the preliminary CAD phase and in ANSYS Workbench. In this case, the Antenna Engineer defined the functional parameters for the antenna, and verified the performance based on the surrounding containment materials that could potentially generate a shield factor in the electromagnetic signal. This shield factor was established as the priority, hence the electronic workgroup (driver) took priority over the mechanical workgroup. In any case, the fundamental aspect in implementing this procedure is for every update to be made available to all workgroups in a timely manner. We wish to emphasize that the optimal configuration for the product must be identified by finding a compromise between the electronic and mechanical requirements, regardless of any precedence that may be granted to one workgroup over another in terms of a project’s requirements. Compared to test case 1, the following operations were implemented in addition to the Multiphysics I/O parameterization and post-processing for system analysis:

- shared management of the material properties;
- parametric definition of the radome’s structure in composite material;
- interpolation of the thermal loads from the global model (TC1) to the particular model (TC2).

Conclusions

This paper describes the work undertaken to develop an operational procedure to strengthen and intensify the use of computational simulation in ELT, and to enable the introduction of Multiphysics analysis as a guide for product development. The ultimate aim of this work, and of the ongoing collaboration with EnginSoft, is to maximize the use of all the engineering simulation tools and skills available in ELT for the development, performance improvement and validation of solutions that integrate multiple components and different physics. Beginning with the results of the initial assessment, concept and planning initiative in the Simulation Process and Data Management (SPDM) system, a more detailed analysis of the existing workflow was carried out. A proposed new workflow from the preliminary design phase to the design phase was outlined in the SPDM system to make CAE analyses functional to the product development process. A description of the operations performed at each stage of the proposed new workflow has been provided, with notes concerning the objectives, actions, users involved and the deliverables. The proposed new workflow was then applied to some ELT-provided test cases related to the main activities requiring the participation of multidisciplinary teams in product development.

Daniele Santoro, Barbara Orobello, Umberto Lecci, Mario Fiorito, Marco Pietrobono, Christian Canestri - Elettronica Group, Elettronica S.p.A.

Alessandro Mellone - EnginSoft

New MPS-based method to design and analyse the cooling systems of engine pistons is faster and more accurate

This paper presents a method for calculating the temperature field of a medium-speed engine piston. The method combines simple empirical formulas for calculating the heat load of the piston, the Moving Particle Simulation (MPS) method in Particleworks' software for solving the cooling effect of the oil, and finite element analysis (FEA) for calculating the temperature field of the piston. This method makes it possible to quickly evaluate several different piston cooling designs in the early stage of the piston design process. The paper shows the validation results and compares the simulation times between the MPS method and the Volume of Fluid (VOF) method.

To keep up with the competition in internal combustion engine manufacturing, all the key components of an engine must be designed and optimized very precisely. During last 50–60 years, the brake mean effective pressure (BMEP) of Wärtsilä engines has increased from around the 5 bar level of naturally aspirated engines to the over-30 bar level of the recent W31 engine. Higher BMEP (i.e. more work per cycle) also means more mechanical and thermal load to the piston and the other components in the combustion process.

High thermal load generates a need for effective piston cooling. Piston temperature must not exceed a certain level in order to maintain the steel material properties, to prevent hot corrosion and the build-up of carbon deposits. Too high a piston temperature causes thermal expansion which can lead to seizure between the piston and the cylinder liner. Figure 1 demonstrates the principle of piston cooling by oil jet, which is one method used for controlling piston temperature. Oil is sprayed from below through an inlet hole in the piston. The oil then splashes inside the cooling gallery of the piston, cooling it by means of the so-called shaker effect. The hot oil finally exits the piston and flows back to the oil sump.

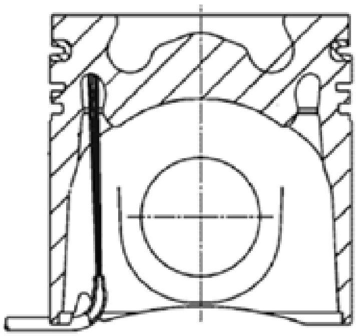


Fig. 1 - Piston jet cooling principle [1]

When a new piston is developed, the main focus of the computer-assisted engineering (CAE) work is usually on how to calculate the mechanical stresses and dimension the piston against fatigue. Nowadays, mechanical stresses can be calculated very accurately with sophisticated simulation tools. An aspect that often receives less attention than the solid mechanics is the thermal analysis of the piston. The heat transfer-related boundary conditions that are being used might be very inaccurate, which means that the temperature field of the piston is likely to be far from real. The ability to simulate the temperature field of the piston realistically is important as the starting point of the stress calculation. Uneven temperature distribution causes thermal stresses. High temperatures affect the materials' mechanical properties, as mentioned earlier. All these elements have a direct effect on the outcome of the structural analysis. The reason for this inaccuracy in thermal boundary conditions is that they are often quite difficult to determine.

The objective of the work was to create a reliable and flexible tool that can be used to evaluate many designs which are different in their piston geometry and their cooling arrangements. Thermal analysis must be undertaken in the early stage of the design process to ensure that temperatures will not exceed general limits. The philosophy was to keep the process quick by avoiding a time-consuming and detailed computational fluid dynamics (CFD) combustion analysis. Instead, simpler experimental formulas to determine the thermal load were chosen. The most difficult part of the problem was how to calculate the cooling effect of the oil splashing inside the piston cooling gallery. Convective heat transfer is always coupled to fluid flow; the existing flow scenario consists of a two-phase free surface flow, which is extremely difficult to handle. As said, to keep the simulation time short, there was no point in using traditional CFD methods here, either. Particleworks' mesh-free Moving Particle Simulation (MPS) method offers a great solution to the simulation of fluid flow and determining the cooling effect. Once the boundary conditions are established, the temperature field is easily solved.

Heat transfer in a piston

A general transient heat transfer problem can be formulated as below [2]:

$$\begin{cases} -\int_{\Omega} (\nabla\psi)^T \underline{k} \nabla T \, dV + \int_{\partial\Omega_q} \psi q'' dS + \int_{\Omega} \psi(\dot{E}_g - \rho\dot{u}) \, dV = 0 \\ T = T_0 \text{ on bounday } \partial\Omega_T. \end{cases}$$

This equation forms the basis of solving a temperature field with FEM.

The boundary condition of the convective heat flux of the second term in the equation above is expressed as [3]:

$$q'' = h(T_1 - T_2),$$

where h is the convective heat transfer coefficient.

Thermal boundary conditions

Figure 2 summarises the combined heat transfer problem of piston cooling.

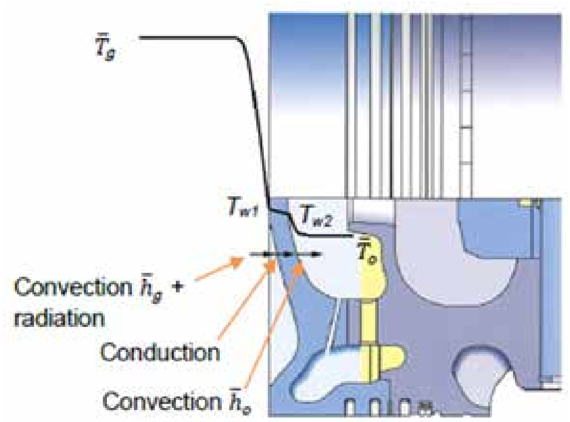


Fig. 2 - Combined heat transfer in a piston

Heat is transferred by convection and radiation from the hot gases inside the cylinder (temperature T_g) to the piston wall, which is at temperature T_{w1} by heat transfer coefficient h_g. Temperature T_g and coefficient h_g are assumed to be constant for the whole top surface of the piston crown. This is a reasonably good approximation for gas engines where the heat load is more uniform due to premixed combustion. In diesel engines, the spatial distribution for the thermal load must consider the injection spray pattern. Surface heat from the top is conducted through the metal to the cooling oil on the side wall that is at temperature T_{w2}. There, the heat is transferred to the oil which is at temperature T_0 by convective heat transfer coefficient h_0.

In this application, the empirical correlation chosen for the thermal load is based on the so-called Woschni [4] formula which relates the instantaneous heat transfer coefficient of the cylinder to the piston movement, the cylinder gas temperature and pressure. The formula can be written as:

$$h_g = \frac{k_f}{D_c} Nu = K_1 D_c^{b-1} p_c^b w^b T_g^{K_2 - 1{,}62b}.$$

Information about valve timing and the compression ratio are also included in the Woschni formula. The gas temperature inside the cylinder is calculated using ideal gas law, and cylinder pressure is measured using a cylinder pressure sensor, which is available anyway in Wärtsilä gas engines.

The instantaneous heat transfer coefficient and the reference temperature are time-averaged over one full engine cycle using the following formulas:

$$\begin{cases} \bar{h}_g = \frac{1}{\Delta\phi} \int_{-360°}^{360°} h_g \, d\phi \\ \bar{T}_g = \frac{1}{\Delta\phi \, \bar{h}_g} \int_{-360°}^{360°} h_g \, T_g \, d\phi \end{cases}$$

Particleworks 6.1.1 implements a model for the calculation of the heat transfer coefficient. The model is based on the analytical results of a flat isothermal plate in parallel flow. Software is used to solve the cooling effect of the oil i.e. the spatial distribution of the heat transfer coefficient h_0 on the cooling gallery surfaces. Unlike the heat transfer coefficient, the reference temperature T_0 is considered to be one constant value over the area.

The solution procedure

The general principle of the iterative solution procedure and the coupling of Particleworks with FEM is shown in Figure 3.

First, the basic geometry and parameters of the engine being analysed must be determined. For example, the cooling oil mass flow $\dot{m}_0$, the oil temperature at spray nozzle $T_{0,in,}$ engine speed, piston geometry, piston movement, cylinder pressure, and the mass flow rate of the intake air are needed.

Then a simulation is performed using Particleworks, which provides a time-averaged spatial distribution of the heat transfer coefficient inside the cooling gallery. The reference temperature of the oil T_0 is guessed for the first round of the iteration.

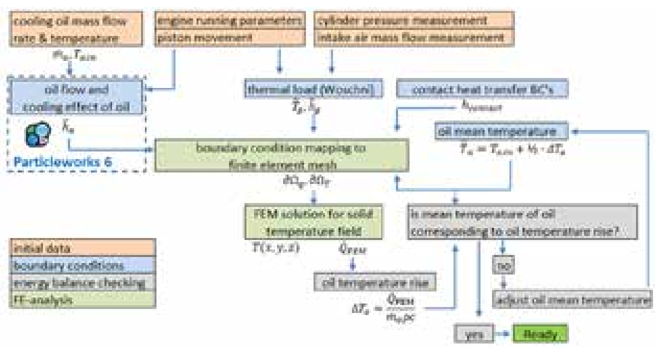


Fig. 3 - General solution procedure

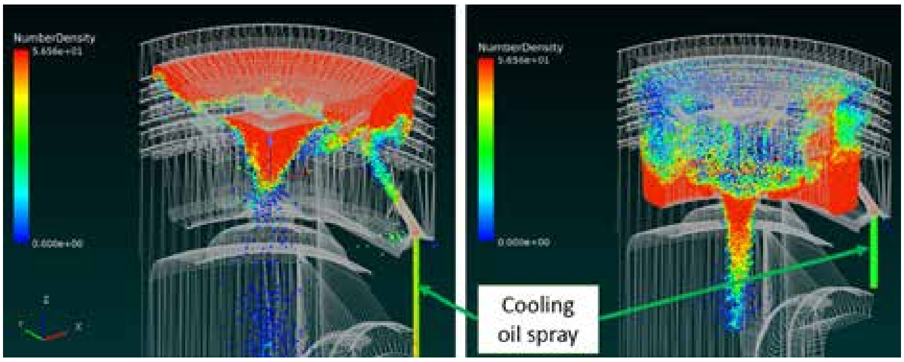


Fig. 4 - Oil sloshing inside the piston cooling gallery

At the same time, a script based on the Woschni correlation, which was developed in-house, outputs the thermal load, T_g and h_g, for the piston. The gap conductance $h_{contact}$ between the piston crown and the piston skirt is acquired by using simple methods like those described in [5] and [6], for example.

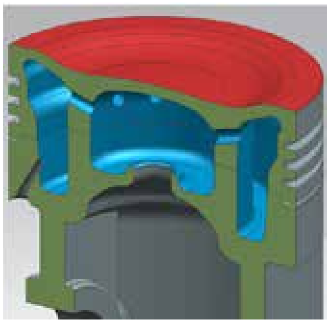


Fig. 5 - Boundary conditions on different surfaces

Once all the boundary conditions have been determined, they are mapped to the surface of a finite element mesh that describes the solid geometry of the piston. Figure 5 shows that the thermal load is applied to the red area and the cooling effect is mapped to the blue areas. The grey area is treated as an insulated surface because it is assumed that the major part – at least 90% – of the heat flow goes through the red and blue surfaces. While there is obviously also heat flow through some grey surfaces, like the top land area, only major contributors are taken into account to keep the model simple.

After mapping the boundary conditions, a steady-state FE analysis is performed. The outputs of the solution are the solid temperature field of the piston and the heat flow $\dot{Q}_{FEM}$ through the piston. At the next stage, the rise in oil temperature equivalent to the heat flow $\dot{Q}_{FEM}$ is calculated. The new mean temperature of the oil based on this rise in oil temperature is then compared to the initial guess used for the first round of iteration. If the two values are different, the mean oil temperature is adjusted, and the FE analysis is performed again. This loop will continue until these two temperatures match. Usually two rounds are enough to obtain converged results. The ultimate result after the final loop is the piston's solid temperature field T(x,y,z).

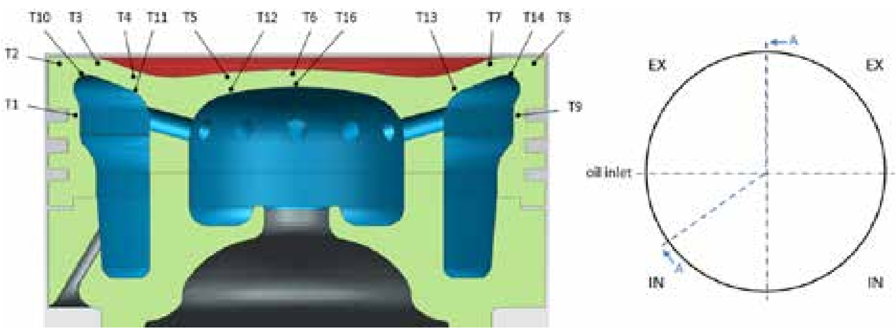


Fig. 6 - Temperature sensor locations inside the piston

Validation results

To validate the model, a simple test series was run on a laboratory engine. This is a medium-bore Wärtsilä spark-ignited gas (SG) engine with a mean piston speed of 10,75 m/s and a specific output of 18,5 kW/litre. One cylinder of the engine was equipped with a special piston with thermocouple sensors. The schematic picture on the left hand side of Figure 6 shows the locations of these temperature sensors. The picture on the right hand side shows the oil inlet hole and the valve locations. All the sensors are located in section A–A.

A test series was run according to Table 1 to see if the calculation model could handle changes in the engine load, the cooling oil's feed rate and the engine speed.

engine load (BMEP)	cooling oil mass flow rate		
	high	Medium	low
100 %	nominal speed	nominal speed	nominal speed
75 %	nominal speed	nominal speed	nominal speed
50 %	nominal speed	nominal speed	nominal speed
10 %	nominal speed	nominal speed	nominal speed
100 %	-	-	0,95 x nominal
100 %	-	-	0,90 x nominal
100 %	-	-	0,85 x nominal

Table 1: Test matrix

Figure 7 shows an example result set from a test point that was run with a 100% engine load, a medium oil-feed rate and a nominal engine speed. The simulated temperatures match well to the measured ones. The root mean square (RMS) error for 15 measurement points is 21,36° C which can be considered a good result.

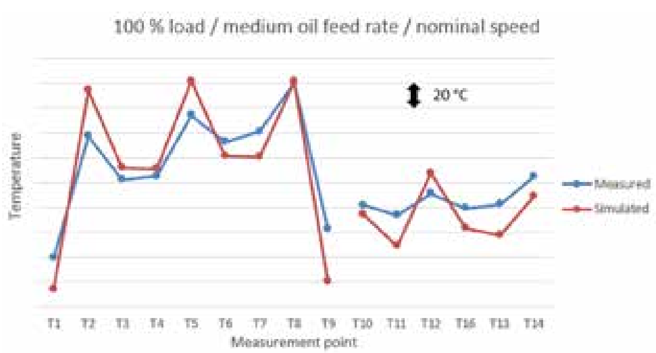


Fig. 7 - Example result set from the validation measurements

The model underestimates the piston ring groove temperatures (T1 and T9) – probably because no heat load was applied to the top land and the 1st ring groove areas. The simulation gives a substantially hotter temperature T2 than was found during measurement. This was caused by using uniform heat load in the simulation, which doesn't take into account the fact that the inlet-valve side of the piston is usually cooler than the exhaust-valve side. A

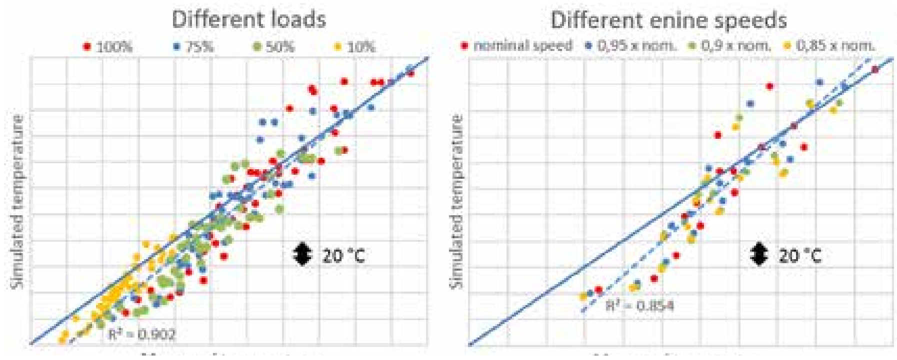


Fig. 8 - Correlation between the simulation and measurement

future task will be to develop the heat-load distribution to include the effect of the valves. Temperatures T10–T16 which are located closer to the cooling gallery surfaces were generally hotter in measurement than in simulation. Sensor T15 wasn't working properly during the test run.

Figure 8 shows the correlation between the measured and the simulated temperatures. One dot represents one temperature measurement point. The different colours represent different engine loads, but they all include all three values for the cooling oil feed rate. On the left hand side, the engine load is varied and on the right hand side the engine speed is varied. The solid blue line shows the ideal correlation. The dashed blue line is a trendline fitted to all data points.

It can be noted that the model is able to follow the changes in engine load and the changes in oil feed rate. Especially the highest temperatures, which are usually the most interesting and important, are very near ideal blue line. On the other hand, the prediction of temperatures for varying engine speeds is not as accurate. Overall, the model provides reliable results. The RMS error in all cases is in range 7, or 24° C.

Conclusions

Particleworks CFD software, which utilizes the MPS method, is an effective tool for the simulation of free surface flow. While the heat transfer coefficient model implemented in the software might seem very simple at first, it calculates the cooling effect of the splashing fluid very accurately. The validation measurements showed that the model can predict the effect of the main parameters related to piston cooling.

Thanks to the mesh-free nature of the method, the pre-processing time of the simulation is minimal. The whole pre-processing workflow, starting from the importation of the piston's 3D-geometry from the CAD software to setting up the simulation using Particleworks' GUI, can be done in only a couple of hours. Instead, traditional mesh-based methods require at least a few days for the pre-processing. ParticleWorks' actual simulation time is short, too. The post-processing was done with the features included in the program and self-made Excel macros.

Table 2 compares the simulation time between Particleworks and another commercial CFD software that uses the Volume of Fluid (VOF) method. Basically, a similar case was simulated using both types of software.

With Particleworks, the simulation time is 80% shorter and uses only a fraction of the CPU cores. This simulation time can be shortened even further by using Graphics Processing Unit (GPU) computing. In additional, the VOF method seemed quite unstable at the beginning so extra iterations were needed to get converged results. Even then, the method couldn't beat the accuracy of the MPS method. The time required to perform the analysis with the new model that was developed is quick enough to allow the evaluation of different piston cooling designs to be done parallel to rest of the piston design process.

The new method has already been used in practice to solve the cooling problems being experienced with a specific design project for a new piston. The piston cooling gallery's geometry was unfavourable, and the oil wasn't flowing properly through the piston causing the piston to heat up more than expected, which ultimately led to a piston seizure. Once this design was simulated with Particleworks, the problem was discovered. A new design was simulated, and a geometry design change was implemented after which the problem disappeared.

	MPS-method Particleworks	VOF-method
physical time (one piston stroke)	0,1 s	0,1 s
simulation time	5 h	24 h
number of CPU cores	12	160
particle diameter (MPS) average element size (VOF)	1,75 mm	2 mm

Table 2: Comparison of simulation time

References

[1] Mahle GmbH (2012). Pistons and engine testing, pp. 284.

[2] Bergheau, J. & Fortunier, R. (2004). Finite element simulation of heat transfer, John Wiley & Sons, Incorporated, Hoboken, New Jersey, USA, 279 p.

[3] Incropera, F. & DeWitt, D. (2002). Introduction to heat transfer, 4th ed. John Wiley & Sons, Inc., New York, USA, 892 p.

[4] Woschni, G. (1967). A universally applicable equation for the instantaneous heat transfer coefficient in the internal combustion engine, available at: https://saemobilus.sae.org/content/670931.

[5] Mikić, B.B. (1974). Thermal contact conductance; theoretical considerations, International Journal of Heat and Mass Transfer, Vol. 17(2), pp. 205-214. http://www.sciencedirect.com/science/article/pii/0017931074900829.

[6] Hasselström, A. & Nilsson, U. (2012). Thermal contact conductance in bolted joints, Chalmers University of Technology, Department of materials and manufacturing technology, 81 p. Available at: http://publications.lib.chalmers.se/records/fulltext/159027.pdf.

Sami Ojala - Wärtsilä Finland Oy

For more information, email: sami.ojala@wartsila.com at Wärtsilä Finland Oy (http://wartsila.com/), based in Vaasa, Finland.

Medical Digital Twins used to help predict pathology evolution and the effect of surgical corrections in cardiovascular patients

Reduced-order model and mesh morphing techniques facilitate the generation of results in near-real-time without High Performance Computers

Nowadays in silico analysis tools in the bio-medical field are moving from the research context to patient-specific treatment and prevention. Hemo-dynamics is receiving great attention, and accurate Computational Fluid Dynamics (CFD) modelling can be adopted to produce a digital medical twin capable of reliably predicting the pathology evolution, and the effect of surgical corrections. The availability of in silico digital twins based on computer-assisted engineering (CAE) simulations is one of the key enablers; the parametric shape of vessels and reduced-order models (ROM) are promising solutions. The ROM approach usually requires the use of high-performance computing (HPC), but it can also be executed almost in real time and outside the standard CAE tools as well. This paper demonstrates the concept using the new ROM Builder available in ANSYS 19.1. We developed a pipeline for an aortic aneurysm to study the effect of a bulge-shaped progression on the flow field. First, a patient-specific geometry was reconstructed, then a CFD model was created with a bulge shape that was parameterised using a radial basis function (RBF) mesh morphing technique. Finally, a ROM was suitably built up by conducting CFD simulations. We provide examples of fast evaluations that were achieved off-line by using the ROM results.

Tackling cardiovascular disease, the world's leading cause of deathCardiovascular disease (CVD) is the main cause of death in the world, resulting in 17.9 million deaths (32.1%) in 2015 alone [1]. It is estimated that up to 90% of these casualties could be predicted and prevented [2]. In silico analysis can help in this regard, improving these figures by analysing patient-specific tailored systems for prevention. The evolution and growing availability of computational power has extended this concept by enabling the use of high-fidelity CAE simulations and the development of medical digital twins that provide a safe environment in which to predict the pathology evolution and to explore the impact of potential surgical corrections. For CVD pathologies, the hemo-dynamics of the vessels can be thoroughly examined by using high-fidelity CFD techniques and tools. The large quantity of degrees of freedom inherent in a real clinical case, however, make this approach computationally expensive and less suitable for testing patient-specific cases virtually and in real time. While several methodologies, such as the use of stochastic approaches for geometrical variations, material properties and uncertainty quantification [3][4][5], have been developed to overcome this problem, as can be seen in the literature, none of them can cope with the full field of variables of interest to provide quantitative output in real time. It is here that ROMs are emerging as a viable technology that can provide complex, multi-degree-of-freedom, full-scale solutions while drastically reducing the overall computing cost.

The use of ROM systems is well-known and has received significant attention in the last decade, but this use has mainly been limited to classical mechanics [6] and dynamics [7]; their implementation in the field of biomechanics is very recent and is limited to only a few works [8]. Furthermore, the ROM's ability to investigate the effect of an enormous number of parameters in real time can be synergistically combined with mesh-morphing techniques that can provide a vast number of geometrical variations. Radial basis functions [9] have been demonstrated to be powerful morphing tools in several fields, including the medical one [10]. In this paper, a workflow envisioning the use of the ROM implementation available in ANSYS 19.1 together with the commercial morpher RBF Morph is proposed and applied to the investigation of the hemodynamic fields for a wide set of ascending thoracic aneurysmatic aortic (ATAA) morphologies. The following section briefly gives the background of ROM and RBF. The proposed workflow is then introduced and demonstrated for execution on an ATAA test case.

A brief background of reduced-order model (ROM) and radial basis function (RBF)

ROM: Reduced-Order Model

Model Reduction is a technique that allows cost-efficient evaluation of systems with a large degree-of-freedom (DOF) by reducing the number of variables involved in the problem. This simplification, while preserving the essential characteristics of the system to enable an accurate and efficient representation, makes real-time control over the parameters of the problem feasible. ROMs have been successfully applied over the years to controls [11], fluid dynamics, structural dynamics, thermal analysis, multi-physics [12], medical [13] and optimization problems [14]. Several techniques to obtain a ROM representation have been explored and are available to the scientific community in the literature. The technique employed in this paper is based on Proper Order Decomposition (POD) which is implemented as commercial software in ANSYS Workbench 19.1 [15]. A typical workflow that includes the use of ROM, shown in figure 1, is composed of two main tasks that are typically referred to as creation and consumption. The first task -- the training of the ROM using a simulation software and extracting the most important modes using a POD technique -- deals with all the processes necessary to generate and extract the information from the full DOF system. The latter task uses these extracted modes to produce a reduced order system in real time. This can be accomplished either by using the ANSYS ROM viewer, a standalone software designated to ROM consumption, or by importing the reduced system into ANSYS Fluent.

Since the reduced system is a direct consequence of the variation in the input parameters and of the generation of several Design Points (DPs), the ROM environment implemented by ANSYS is designed to be strongly entwined with the Response Surface (RS) concept. Indeed, the generation of the training data is produced by evaluating a given number of DPs, called snapshots, at full scale. The higher the number of snapshots, the more faithful the reduced order model, which then captures all the nuances of the system produced by a parameter variation. The quality of the reduced model is also influenced by the number of modes employed for its discretization. ANSYS Workbench allows the result of the ROM creation task to be saved as a standalone file that includes the reduced system as well as the numerical grid of the problem. When necessary, this file can then be imported into the consumption task by exploiting the standalone ANSYS ROM viewer, or it can be loaded into ANSYS Fluent as an alternative to the *.cas and *.dat files.

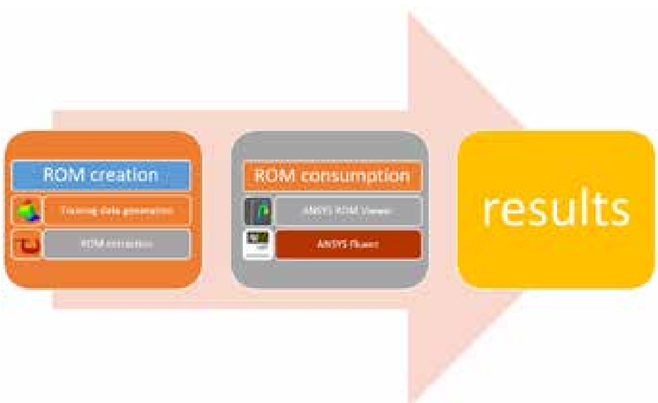


Fig. 1 - ANSYS ROM usage workflow

RBF: Radial basis function

From a mathematical point of view, the solution of an RBF problem consists of calculating the coefficients of a linear system of order equal to the number of source points (De Boer et al., 2007), by means of which the displacement of an arbitrary mesh's node (target) can be expressed, and then imposed, as the summation of the radial contribution of each controlled node (source). In this way, mesh smoothing can be rapidly applied by maintaining the mesh topology expressed in terms of the total number and type of elements.

In particular, the RBF Morph tool uses the RBF interpolant s composed of a radial function that contains the RBF φ and a multivariate polynomial corrector vector h of the order m-1, where m is said to be the order of φ, introduced with the aim of assuring the uniqueness of the RBF solution and its compatibility for rigid motion. Specifically, if N is the total number of source points, the formulation of the RBF interpolant is

$$s(\boldsymbol{x}) = \sum_{i=1}^{N} \gamma_i \varphi(\|\boldsymbol{x} - \boldsymbol{x}_{k_i}\|) + h(\boldsymbol{x}) \quad (1)$$

where x is the vector identifying the position of a generic node belonging to the surface, and/or volume mesh x_{ki} is the ith source node position vector; ||·|| is the Euclidean norm.

The fitting RBF solution exists in the case that the RBF coefficients vector γ_i and the weights of the polynomial corrector vector β_i can be found such that the interpolant function, at the source points, possesses the specified (known) values of displacement g_i, whilst the polynomial terms give a null contribution. In other words, the following relations

$$s(\boldsymbol{x}_{k_i}) = \boldsymbol{g}_i \qquad 1 \le i \le N \quad (2)$$

$$\sum_{i=1}^{N} \gamma_i q(\boldsymbol{x}_{k_i}) = 0 \quad (3)$$

are simultaneously verified for all polynomials q with a degree less than or equal to that of polynomial h (Beckert and Wendland, 2001). The minimal degree of polynomial h is dependent on the choice of RBF type. It can be demonstrated that a unique RBF interpolant exists if the RBF is conditionally a positive definite (Van Zuijlen et al., 2007). In the case that this latter condition is established, and if the order is less than or equal to 2 (Jin et al., 2001), a linear polynomial applies

$$h(\boldsymbol{x}) = \boldsymbol{\beta}_1 + \boldsymbol{\beta}_2 x + \boldsymbol{\beta}_3 y + \boldsymbol{\beta}_4 z \quad (4)$$

enabling the exact recovery of the rigid body translations.

In the event such assumptions are verified, the interpolant has the form

$$s(\boldsymbol{x}) = \sum_{i=1}^{N} \gamma_i \varphi(\|\boldsymbol{x} - \boldsymbol{x}_{k_i}\|) + \beta_1 + \beta_2 x + \beta_3 y + \beta_4 z \tag{5}$$

and the values for γ_i and β_i can be obtained by solving the system

$$\begin{pmatrix} \boldsymbol{U} & \boldsymbol{P} \\ \boldsymbol{P}^T & \boldsymbol{0} \end{pmatrix} \begin{pmatrix} \boldsymbol{\gamma} \\ \boldsymbol{\beta} \end{pmatrix} = \begin{pmatrix} \boldsymbol{g} \\ \boldsymbol{0} \end{pmatrix} \tag{6}$$

where U is the interpolation matrix whose elements are derived by calculating all the radial interactions between the source points, as follows

$$U_{ij} = \varphi\left(\|\boldsymbol{x}_{k_i} - \boldsymbol{x}_{k_j}\| \right) \qquad 1 \le i \le N, \qquad 1 \le j \le N \tag{7}$$

and P is a constraint matrix that arises from balancing the

$$\boldsymbol{P} = \begin{pmatrix} 1 & x_{k_1} & y_{k_1} & z_{k_1} \\ 1 & x_{k_2} & y_{k_2} & z_{k_2} \\ \vdots & \vdots & \vdots & \vdots \\ 1 & x_{k_N} & y_{k_N} & z_{k_N} \end{pmatrix} \tag{8}$$

assuming that the source points are not contained in the same plane (otherwise the interpolation matrix would be singular).

Radial Basis Functions (RBF) with global support	$\varphi(r)$
Spline type (Rn)	$r^n, n\ odd$
Thin plate spline (TPSn)	$r^n \log(r), n\ even$
Multiquadric (MQ)	$\sqrt{1+r^2}$
Inverse multiquadric (IMQ)	$\frac{1}{\sqrt{1+r^2}}$
Inverse quadratic (IQ)	$\frac{1}{1+r^2}$
Gaussian (GS)	e^{-r^2}
Radial Basis Functions (RBF) with compact support	$\varphi(r) = f(\xi), \xi \le 1, \xi = \frac{r}{R_{sup}}$
Wendland C0 (C0)	$(1-\xi)^2$
Wendland C2 (C2)	$(1-\xi)^4(4\xi+1)$
Wendland C4 (C4)	$(1-\xi)^6\left(\frac{35}{3}\xi^2+6\xi+1\right)$

Table1: Typical RBF

As has been described, by satisfying the displacement field prescribed at the source points, RBF Morph smoothes the mesh nodes by using the following formulation of the interpolant

$$\begin{cases} s_x(\boldsymbol{x}) = \sum_{i=1}^{N} \gamma_i^x \varphi(\|\boldsymbol{x} - \boldsymbol{x}_{k_i}\|) + \beta_1^x + \beta_2^x x + \beta_3^x y + \beta_4^x z \\ s_y(\boldsymbol{x}) = \sum_{i=1}^{N} \gamma_i^y \varphi(\|\boldsymbol{x} - \boldsymbol{x}_{k_i}\|) + \beta_1^y + \beta_2^y x + \beta_3^y y + \beta_4^y z \\ s_z(\boldsymbol{x}) = \sum_{i=1}^{N} \gamma_i^z \varphi(\|\boldsymbol{x} - \boldsymbol{x}_{k_i}\|) + \beta_1^z + \beta_2^z x + \beta_3^z y + \beta_4^z z\,. \end{cases} \tag{9}$$

RBF enables great flexibility in acting on the radial functions, which can be compactly or globally supported. The common options are summarized in Table 1. RBF Morph allows four radial functions.

The distance function (global support and bi-harmonic in 3D) is used by default and performs very well in volume morphing, as it allows one to obtain very high quality and it is accelerated so that it can handle RBF problems of beyond 1 million centres (source points). Wendland functions (C0, C2 and C4) are available for

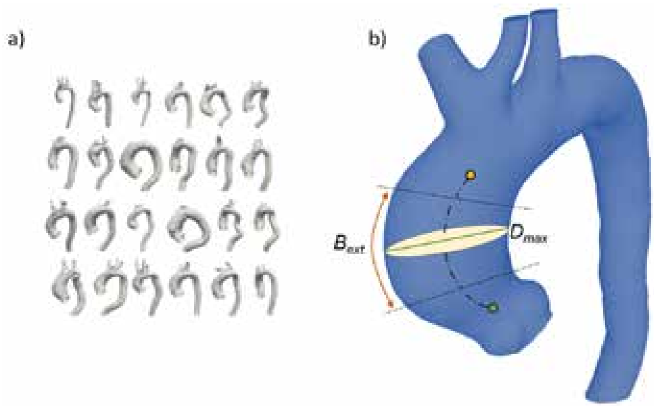


Fig. 2 - Statistical aneurysmatic aorta

surface sculpting since the high level of continuity can be used to check a surface using just a few control points.

Application description

The application selected to describe the proposed workflow is an ascending aorta aneurysm. In the following sections, the ROM and RBF set-ups are provided in detail.

CFD-model generation

The geometry considered, and its shape variations, are the result of processing the data pertaining to an aneurysmatic aorta SSGA, which has been defined according to the procedure described in Capellini et al. [16]. The 3D-model was obtained from the statistical analysis of the aortic morphological shapes of 45 aneurysmatic patients (Fig.2a) whose CT datasets were retrospectively segmented and elaborated using VMTK software and Python custom scripts. The

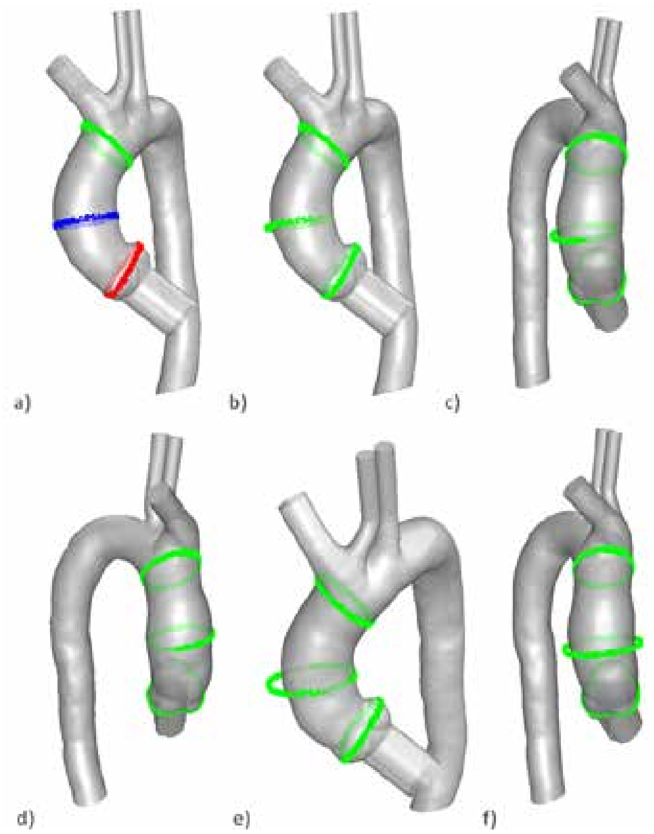


Fig. 3 - RBF set-up. Source points and their displacement

SSGA model was parameterized to create a CFD model that could cope with many ascending aneurysmatic aortic configurations through shape deformations. Three of the parameters evaluated in the previous statistical analysis work were considered to follow the bulge variations: the maximum diameter of the aneurysm (D_{max}), the tortuosity value (T) of the centreline of the ascending portion of aorta, and the bulge extension (B_{ext}). These parameters were assumed to vary at specific intervals that were extracted from the results of the statistical analysis reported in Capellini et al. [16]. We considered a range of average values $\pm$ standard deviations for each parameter (D_{max} = [45.69-55.95 mm], T= [0.0776-0.1484], B_{ext} = [17.93-41.69 mm]). Figure 2(b) depicts the SSGA model including the ascending centreline, the D_{max} and the B_{ext}.

RBF set-up

The mesh morphing action was accomplished using the commercial morpher RBF Morph. A set of source points on the ascending thoracic aortic region was selected to generate five shape-variations representative of the statistical geometry explained in the previous section. Figure 3 shows the RBF set-up. Three sets of RBF points were generated, as shown in Figure 3a, by maintaining fixed the extremal sets (green and red in the picture) in order to circumscribe the morphing action. The central set was translated into the plane over which the points were defined (Figure 3b, 3c and 3d), and was scaled over the same plane along the principal directions (Figure 3e and 3f). This set of five shape-modifications was chosen to generate the representative variations shown in Figure 2a. The time required to morph the 1.5 million-element tetrahedral CFD mesh for each DP was 43s using an i7 laptop with 16 GB of RAM.

ROM set-up

As previously described, the ROM extraction process requires the evaluation of several high-fidelity snapshots. This problem considered five shape parameters. Consequently, a number of snapshots had to be chosen to capture all the important characteristics of the system: 40 snapshots, selected via the Kriging method, were taken to seed the DP in the parameter space. The first five orthogonal modes were extracted with the POD to obtain a model that was reduced to 5 DOF. It was decided to only extract 5 modes because the error that would have been introduced by using 10 modes was below 1%, as shown in Figure 4, where the pressure results obtained using 5 modes (top row) are compared to the ones gained employing 10 modes (bottom row). This comparison was made by varying the flow according to three levels of accretion of the aneurism.

Results

To assess the quality of the proposed methodology, the results obtained using ROM were compared to those achieved using plain CFD evaluations. Figure 5 shows the pressure contours and streamlines obtained through ROM for the three degrees of accretion of the aneurism (in the first row), while the ones calculated using a full CFD computation are shown in the last row. The maximum discrepancy between the two methodologies is in the order of 2.5%. Although this error could be further decreased by enriching the number of snapshots and employing a higher number of orthogonal modes, it was judged to be acceptable because each new ROM evaluation could be visualised in almost real-time whereas the CFD calculations required about half an hour.

Conclusions

The activities described in this paper deal with a critical aspect of digital twins: the need to gain computational results in real time. Since high-fidelity CFD simulations, which have already proven to be useful in predicting the growth and evolution of CVD pathologies, require a lot of computational effort, a methodology to overcome this problem was proposed. ROM has proven to be very efficient in reducing the computing costs for complex multi-degree-of-freedom models (like CFD models). On the other hand, RBF mesh morphing has proven to be highly efficient in generating different shapes for numerical models, thus reducing the time required to generate new ones. The proposed methodology envisages the synergetic use of ROM and RBF mesh morphing to generate a new shape for the CFD model including its numerical results from a discrete set of shape configurations. The proposed methodology was developed in the ANSYS® Workbench™ 19.1 environment by exploiting the ROM functionalities provided by this release of the software together with the RBF-Morph™ Fluent® add-on. The complete workflow was tested in a CFD study of an ascending thoracic aortic aneurysm. The results obtained with the proposed approach were then compared to regular CFD evaluations, which required the full CFD solution of the new shapes of the aneurism. A good agreement in terms of the monitored variables was finally obtained. The successful application of the proposed methodology led to the conclusion

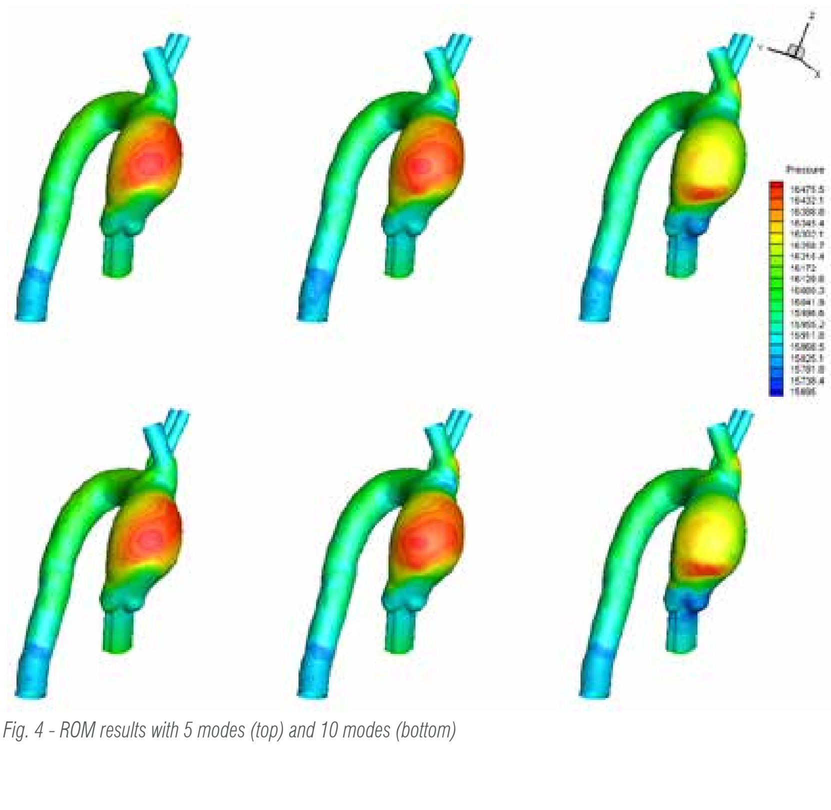


Fig. 4 - ROM results with 5 modes (top) and 10 modes (bottom)

that it can be successfully exploited to assist in the generation and fruition of digital twins in medical applications.

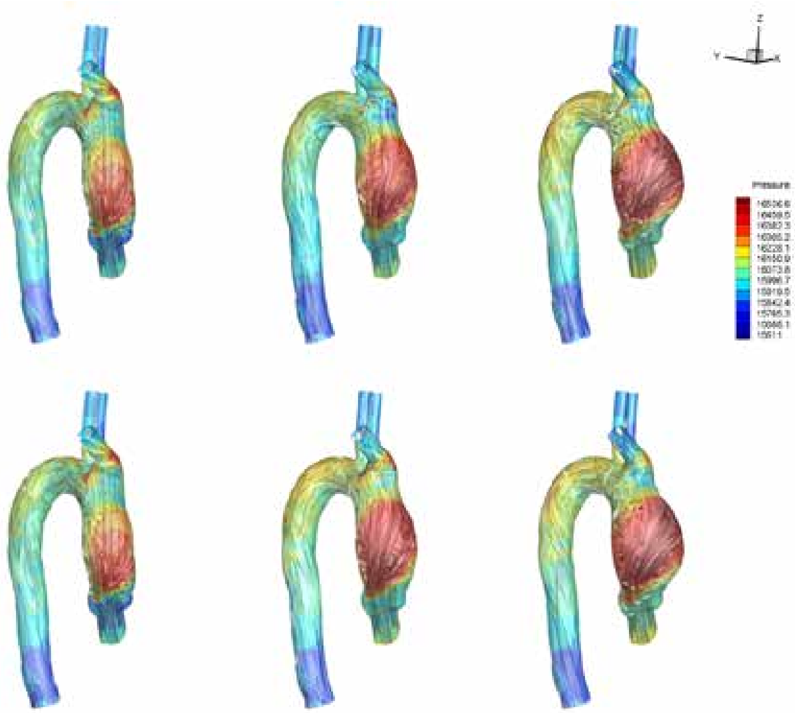


Fig. 5 - ROM results (top) compared to CFD (bottom)

References

[1] GBD 2015 Mortality and Causes of Death, Collaborators. (8 October 2016). "Global, regional, and national life expectancy, all-cause mortality, and cause-specific mortality for 249 causes of death, 1980-2015: a systematic analysis for the Global Burden of Disease Study 2015". Lancet. 388 (10053): 1459–1544. doi:10.1016/S0140-6736(16)31012-1.

[2] McGill HC, McMahan CA, Gidding SS (March 2008). "Preventing heart disease in the 21st century: implications of the Pathobiological Determinants of Atherosclerosis in Youth (PDAY) study". Circulation. 117 (9): 1216–27. doi:10.1161/CIRCULATIONAHA.107.717033.

[3] Celi, S., and Berti, S., 2012. "Biomechanics and FE modelling of aneurysm: Review and advances in computational models". In Aneurysm. InTech.

[4] Celi, S., and Berti, S., 2013. "Three-dimensional sensitivity assessment of thoracic aortic aneurysm wall stress: a probabilistic finite-element study". European Journal of Cardio-Thoracic Surgery, 45(3), pp. 467–475

[5] Boccadifuoco, A., Mariotti, A., Celi, S., Martini, N., and Salvetti, M., 2018. "Impact of uncertainties in outflow boundary conditions on the predictions of hemodynamic simulations of ascending thoracic aortic aneurysms". Computers & Fluids, 165, pp. 96–115

[6] Alonso, D., Velazquez, A., and J.M., V., 2009. "A method to generate computationally efficient reduced order models". Comput. Methods Appl. Mech. Engrg, 198(-), pp. 2683–2691

[7] Gennaretti, M., and Mastroddi, F., 2004. "Study of Reduced-Order Models for Gust-Response Analysis of Flexible Fixed Wings". Journal of Aircraft, 41(2), pp. 304–313.

[8] Niroomandi, S., Alfaro, I., Gonzlez, D., Cueto, E., and Chinesta, F. "Real-time simulation of surgery by reduced-order modeling and x-fem techniques". International Journal for Numerical Methods in Biomedical Engineering, 28(5), pp. 574–588.

[9] Buhmann, M. D., 2000. "Radial basis functions". Acta Numerica 2000, 9(247), Jan, pp. 1–38

[10] Biancolini, M., Ponzini, R., Antiga, L., and Morbiducci, U., 2012. "A new workflow for patient specific image-based hemodynamics: parametric study of the carotid bifurcation". Comput Model Objects Represented Images III: Fundam Methods Appl.

[11] Bailey, E. A., Dutton, A. W., Mattingly, M., Devasia, S., and Roemer, R. B., 1998. "A comparison of reduced-order modelling techniques for application in hyperthermia control and estimation". International Journal of Hyperthermia, 14(2), pp. 135–156

[12] Gennaretti, M., and Mastroddi, F., 2004. "Study of Reduced-Order Models for Gust-Response Analysis of Flexible Fixed Wings". Journal of Aircraft, 41(2), pp. 304–313

[13] Ballarin, F., Faggiano, E., Ippolito, S., Manzoni, A., Quarteroni, A., Rozza, G., and Scrofani, R., 2016. "Fast simulations of patient-specific haemodynamics of coronary artery bypass grafts based on a POD-Galerkin method and a vascular shape parametrization". Journal of Computational Physics, 315, pp. 609–628

[14] Karpel, M., 1999. "Reduced-Order Models for Integrated Aeroservoelastic Optimization". Journal of Aircraft, 36(1), pp. 146–155.

[15] Chaturantabut, S., and Sorensen, D. C., 2012. "A State Space Error Estimate for POD-DEIM Nonlinear Model Reduction". SIAM Journal on Numerical Analysis, 50(1), pp. 46–63

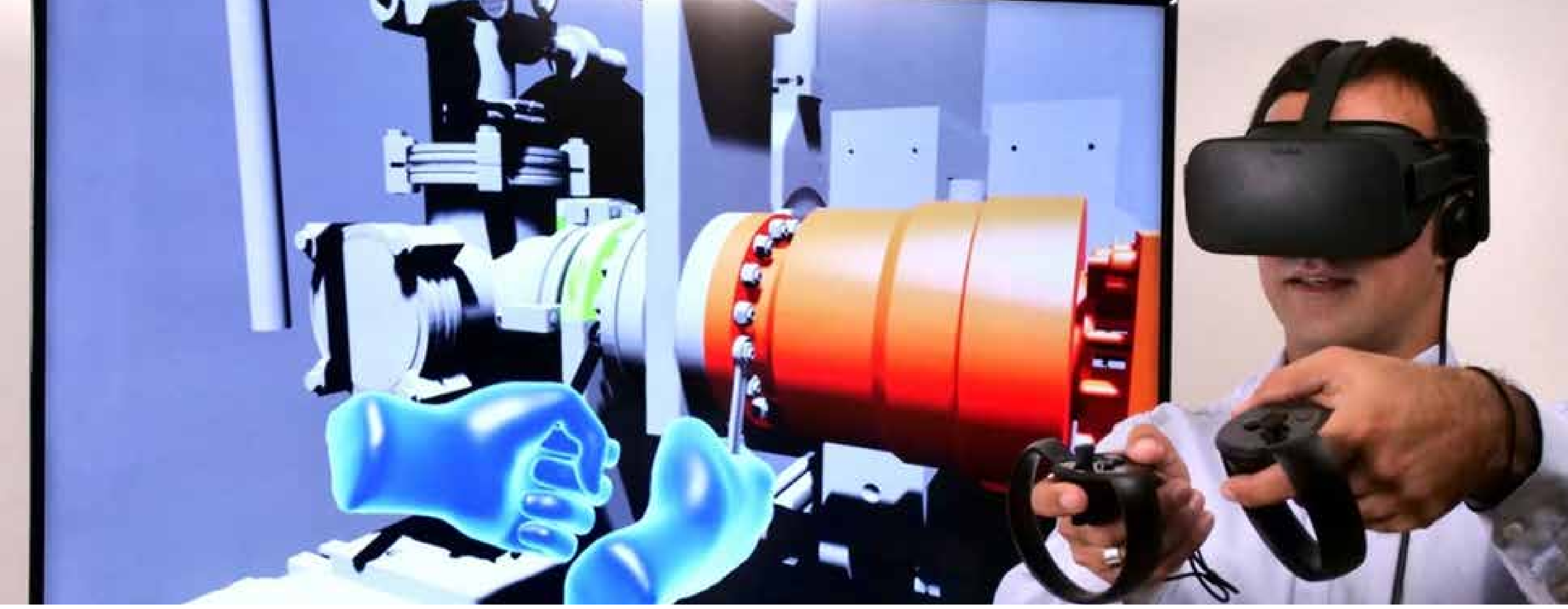
[16] Capellini, K., Vignali, E., Costa, E., Gasparotti, E.,Biancolini, M., Landini, L., Positano, V., and Celi,S., 2018."Computational fluid dynamic study for ATAA hemodynamics: an integrated image-based and RBF mesh morphing approach". Journal of Bio-mechanical Engineering. in-press

Corrado Groth, Stefano Porziani, Marco Evangelos Biancolini
University of Rome Tor Vergata's Dep. of Enterprise Engineering in Rome

Emiliano Costa - RINA Consulting S.p.A

Simona Celi, Katia Capellini
BioCardioLab's Bioengineering Unit at the Fondazione Toscana "G. Monasterio"

Michel Rochette, Valery Morgenthaler - ANSYS France



Virtual Reality Application in a Nuclear Accelerator Facility

In a complex facility like a nuclear research laboratory, a high level of integration among the different sub-systems is required and allows the proper operation of the experimental lines, coordination, organization and planning (crucial aspects for maintenance and facility upgrade). The group studied virtual reality (VR) technology and used it to design different solutions, with the aim of optimizing these key points.

The use of virtual technology in the research

A nuclear accelerator laboratory can be described as a complex system composed of several functional sub-systems such as a vacuum system, cryogenic plant, control system, RF system, etc. which have to be coordinated properly to provide the service (the beam) to the end user. These systems have to work correctly and require continuous maintenance to guarantee the beam for experiments. In addition, operators have to be fully trained to work and control every single part of the accelerator. Because of the nature of the experiments and the kind of facility, different problems and limitations related to area accessibility are experienced during and after operations. These limitations cause delays and uncertainty in the planning process at every level (management, logistics and operations).
It would be ideal to access all the restricted areas irrespective of the limitations and the machine's operation. The aim, therefore, was to find a method and a tool that would overcome these constraints via a realistic representation of the area, while allowing regular operations in the facility to continue. VR is the key technology that was used to design and implement the solutions.

Virtual Reality Technology

This technology is an interactive computer-generated experience within a simulated environment that incorporates mainly auditory and visual, but also other types of sensory feedback. It is rapidly diffusing in several different professional environments, such as medicine, architecture, the military and industry, with different levels of interaction, based on the user experience required. For the developer, VR technology is a powerful tool which allows one to reproduce environments and objects with a high level of detail. The principal characteristics of a VR experience are:

- Photorealism
- Real time
- Immersive
- Interactive

Study of the Process

As previously stated, this kind of technology can be adopted to help different users (managers, developers, operators) with different critical tasks. The hardest part to achieve is how to integrate a VR-based solution into the normal design and organization process: users have their own sets of tools and methodologies to execute their tasks, and these two factors must be taken into account by the developers because of the heterogeneity of the people involved.

We focused on the following three macro areas for the first test bench:

- **Training**: one of the biggest challenges in a physical laboratory is training all the operators properly to work with the accelerator machine and its ancillaries. As technology evolves and new solutions are continually implemented in the facility, operators have to be trained cyclically. This can be securely done in a simulated environment, independent of the physical machine. In this scenario, the trainer can also simulate emergencies and evaluate the feedback about behavior, to estimate response times in a hazardous environment;
- **Machine operations and maintenance**: using the same virtual environment provided for training, it's possible to evaluate and prepare the planning of ordinary and extraordinary maintenance and machine upgrades. The entire environment is rebuilt starting from CAD so the final 3D virtual model guarantees sub-millimeter resolution. This means VR users can operate in the virtual simulation to evaluate, for example, device positioning for machine upgrades;
- **Data collection:** as already mentioned, the virtual model is defined starting from 3D CAD models. Since different groups manage the different sub-systems in the facility, the tool can be very useful to test and verify data integrity and coherence. This reduces design errors and optimizes communication and data exchange among the groups.

The control process and its integration

An important preliminary consideration relates to the set of hardware and software tools and the process required to integrate a VR experience into the regular control process. In considering the normal flow of a control process cycle (shown in Figure1 below) and the virtual reality tool's potential impact, it was determined that it would provide the greatest benefit if it was integrated into the design stage. Consequently, a secondary cycle was defined and implemented to execute the following steps:

- **Data collection and harmonization:** in this phase, VR developers collected the data (3D CAD models) provided by every group involved and verified its coherence before designing the virtual experience. A huge but important effort is required at this stage to check misalignments and dimensional errors;
- **Virtual prototype:** this phase is where the real design of the virtual experience is created. The complexity and the effort required depend on the kind of experience desired (training, planning, show room, etc.): in some cases, a low-level software program is required to produce a completely "real" virtual experience, such as for hazard training. In this stage, the type of hardware used for the experience has an impact on the design and implementation
- **Virtual test and debugging:** in this phase, the virtual experience can be used to verify the application itself (logic, design, etc.) for internal debugging, and to acquire the first feedback from the target end user:
    - o In a training session, the end user (such as the operator) can be guided and evaluated by the trainer. In this case, the feedback comes from the operator's experience.
    - o In machine operation and maintenance, the feedback is based on the end user's evaluations about executing a particular real upgrade to the accelerator machine and facility. This feedback makes it possible to test objects' positions and task procedures and, in case of errors, to redefine the prototype at the design level or take corrective decisions

The goal of this secondary process cycle was to optimize the design stage in the main control flow to minimize design and decision errors and optimize the entire process' execution time.

The most crucial steps in the secondary process cycle are data collection and virtual prototype design. The second step, in particular, is highly dependent on the hardware and software solutions chosen. There are several solutions on the market for implementing a virtual experience at both the hardware and software levels. While the hardware has an intrinsic cost (purchase and maintenance), there are both commercial and open-

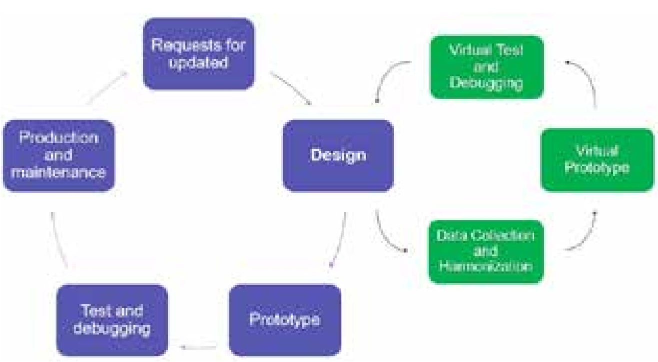


Fig. 1 - Double Wheel Flowchart: how to integrate the Virtual Reality tool into a common design process. The VR experience provides immediate feedback to the design process, helping developers and designers in their task

source software solutions. Our team's goal was to determine the best set of software to minimize license costs (excluding the licenses already held by the Institute) and, at the same time, to achieve the best resolution in terms of photorealism and real-time execution.

As indicated in Figure 2, the software used in this process can be divided into these sub-categories:

- **Licensed software:** 3D CAD – different groups in the Laboratory use different suites, and the related costs are already carried by the Institute. The key point here was to determine the best file format to use to export the models.
- **Open-source software:** Computer graphics framework and real-time development engine, importing 3D models directly into the development engine, badly impacted performance in the final VR experience due to the very high polygon count, even using the latest hardware available on the market. It was necessary to insert an intermediate step with additional software to simplify the 3D models.

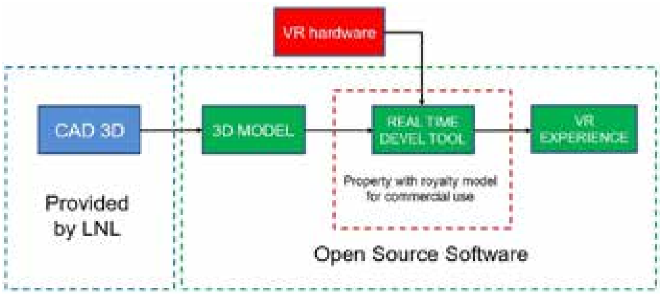


Fig. 2 - The virtual reality process: the flowchart defines the software and hardware required, as well as the correct flow required to provide the VR experience to the end user

Several VR hardware solutions are available on the market. As a result, the VR platform is fully compatible with all commonly used head-mounted displays (HMD): a virtual reality system that completely immerses users inside virtual worlds by means of a dedicated headset. All devices are fully integrated into the real-time engine which provides good photorealistic immersion at a high frame-rate to avoid common motion-sickness problems.

The VR solutions implemented

Data collection and design

Because this technology is completely new in our facility, a preliminary proof of concept was required. The double objective here was:

- to demonstrate the validity of the VR process flowchart
- to acquire the first important feedback about dimensional and data coherence

The collection of data from several sources (mechanics, plant schemas, etc.) allowed the replication of an immersive virtual environment that faithfully represented the real one, see the first test was extremely useful because it proved the incoherencies among the data provided by the groups involved and made it possible to correct these during design and documentation.

Training

The first case was used to verify the procedures required to mount and unmount parts of the accelerator line where operators can come into contact with the residual radiation produced by activated objects. In this case, photorealism was not key so the environment was defined with dedicated colors to emphasize the functional object with which the user

Fig. 3 - the virtual room has a 1:1 scale and every single detail (geometry, textures, object interactions, lights, physics) has been completely reproduced

had to interact; instead, the dimensions and object sizes were deeply verified and respected. The results of the first trials were remarkable: the experience is ready to be used to train operators to work in new parts of the particle accelerator allowing them the opportunity to become familiar with the system while the line itself is being used for preliminary tests. In normal conditions, the maintenance people would have to wait for the tests to end first; furthermore, they will now be confident and prepared to operate immediately in the actual configuration.

Machine Operation and Maintenance

The machine operation and maintenance planning processes were verified in parallel to the training. The first user case was the preparation of a new area under construction in the facility where several different parts developed by different groups must be placed in a limited space. The complete area and all the systems, devices and ancillaries were reproduced with the same precision of the 3D CAD models and positioned with an error less than 1mm.

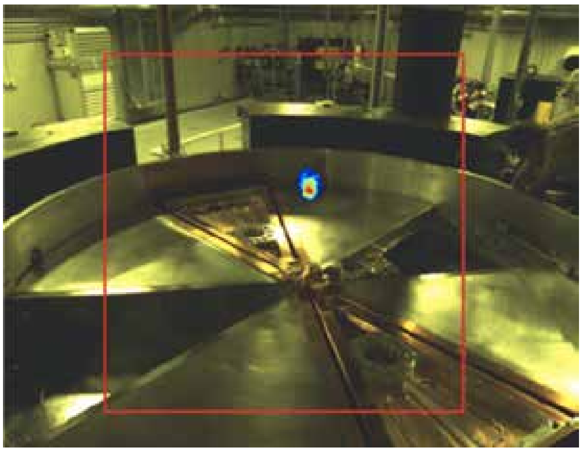


Fig. 5 – Example of AR: information provided by the Radioprotection Service (D. Zafiropoulos, L. Sarchiapone) can be used to generate a live map of the radioactivity in a particular area

Data integration – VR, AR and MR

A further step was to study the integration of the heterogeneous data and information into the VR environment to produce additional details for the end user: this is a new way to use and visualize common information already available in our laboratory for physical experiments and accelerator setups.
Different approaches and technology were considered:

- **Data integration in VR:** data provided by software other than 3D CAD was manipulated and imported into a virtual simulation to visualize different details. Figure 4 represents an example of the data integration: the information produced by a laser tracking system was used in the virtual environment to visualize and verify objects' positions and alignments. This kind of information can be useful for machine operation and maintenance.
- **Augmented Reality (AR):** augmented reality is an interactive experience of a real-world environment where the real-world objects are "augmented" by computer-generated perceptual information, sometimes across multiple sensory modalities. In this scenario, the user utilizes specific data to become aware of dangers. For example, data produced by the Radioprotection Service can be used in an AR solution to create a live radioactivity map for users that work near a particular apparatus (Figure 5). Different hardware is required for this solution.
- **Mixed Reality (MR):** mixed reality, sometimes referred to as hybrid reality, is the merging of real and virtual worlds to produce new environments and visualizations where physical and digital objects co-exist and interact in real time.

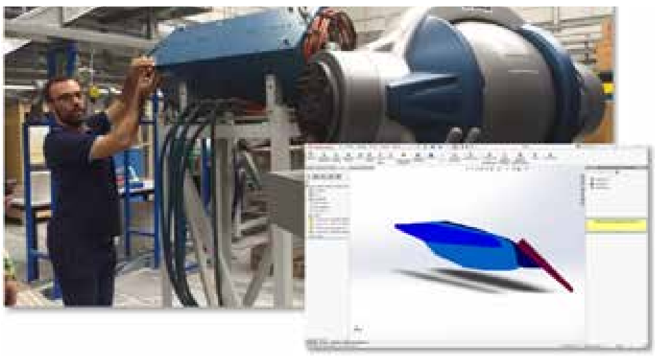
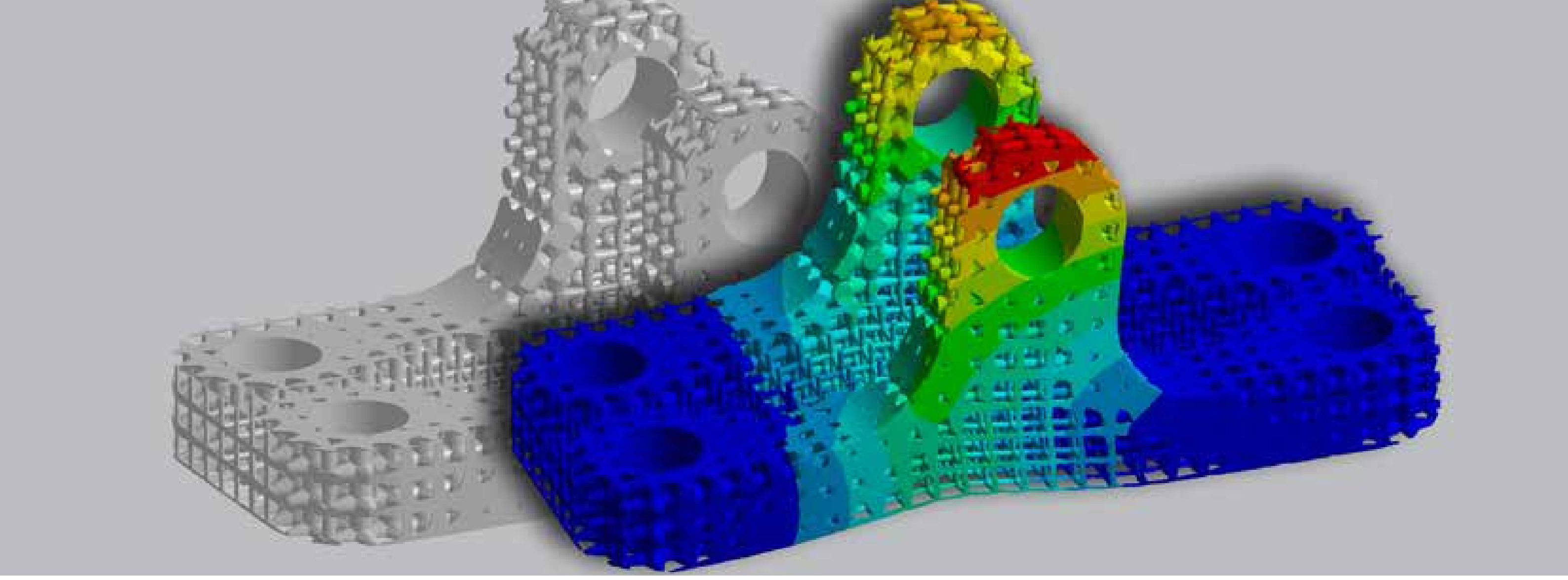


Fig. 4 - Heterogeneous data used in the virtual reality: information provided by the laser track system was imported into the virtual environment to verify the objects' positions and alignments

Conclusions

Virtual Reality technology and its derivatives are mature enough to be used in environments like a nuclear research laboratory to provide interesting tools and procedures that let developers and operators optimize their tasks. Particular emphasis has been placed on training and maintenance planning -- critical aspects that can be improved. Preliminary tests were performed to understand the feasibility of this technology when applied to an environment and the feedback that was obtained is very promising: the VR helped us to evaluate the solutions proposed. Additional studies have been started to extend the functionality and the first prototypes for local and remote multiuser experiences are being implemented.

Lorenzo Pranovi, Maurizio Montis
National Institute for Nuclear Physics (INFN) - National Laboratory of Legnaro



The multiscale and optimization analysis of a lattice structure produced via additive manufacturing

CYBERNET

Products of additive manufacturing have a non-homogeneous structure and can be characterized in three types of scale, namely macro, meso and micro. A macro-scale structure consists of many meso-scale inhomogeneities that are known as a lattice structure. The merits of a lattice structure are that it is lighter, and the material properties are more controllable. The behavior of a macroscopic material is, therefore, strongly dependent on the shape of its lattice structure. This paper discusses an example of the optimization and comparison of a multiscale analysis between a macro-scale and a meso-scale lattice structure. The macroscopic, an-isotropic material constants of the lattice structure were predicted using CMAS. CMAS is an add-on tool for the ANSYS Workbench environment. It was developed by CYBERNET SYSTEMS Co. Ltd under the direction of Professor Terada of Tohoku University and is available in Europe from EnginSoft SpA.

A lattice structure

Over the past few years, many engineers have shown an interest in the lattice structures created with additive manufacturing by 3D-printer equipment. A lattice structure has a non-homogenized shape created by many pillars. In this, it differs from a traditional truss structure. The lattice is constructed of pillars whose angles can be set freely, while a truss consists only of pillars which are limited to right-angled triangle, or equilateral triangle shapes, in most cases. Moreover, a lattice structure of various sizes -- ranging from the order of meters to micro meters -- can easily be made with 3D printers.

Mechanical characteristics

There are many benefits to be obtained by replacing homogeneous materials with lattice structures, as listed below.

- Feature 1: Less weight for the entire component. The volume-fraction of the pillars in a lattice structure can be easily controlled by changing their thickness, so, one can control the weight of products at any size. A lattice structure is also much smaller than a macro topology (see Figure 1(a).)
- Feature 2: The lattice density can be distributed. The material density of products can also be distributed according to their position. The lattice structures can take on a different density based on their stress state: high elasticity or strength is not needed at points that experience low stress. This feature means that products can be optimized for weight saving while maintaining the elasticity of the whole structure (see Figure 1(b).) ANSYS19.2 should be used to take advantage

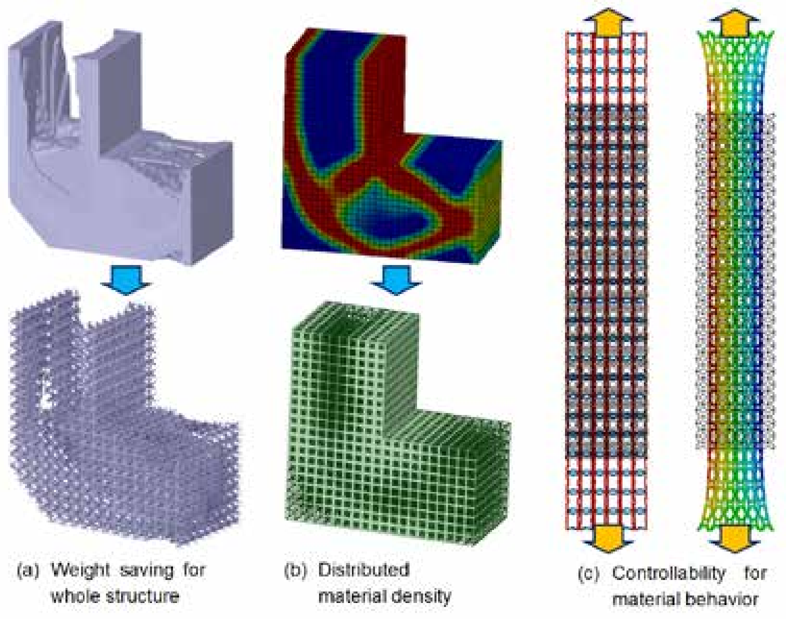


Fig. 1 - Benefits to using a lattice structure.

of this feature because it provides a lattice optimization analysis function for lattice density.

- Feature 3: Control material behavior without changing raw materials. The macroscopic material behavior of lattice structures can be flexibly controlled without changing the raw material. This feature is not very widely utilized for product design, even though there are many application possibilities. Figure 1(c) shows an example of numerical material testing on two types of lattice structure whose pillars have the same material and volume fraction. The specimens were pulled in a vertical direction. The tests confirmed that the change of thickness in cross machine direction was extremely different. In fact, the macroscopic Poisson's ratio was estimated at 0.00 and 0.55, respectively.

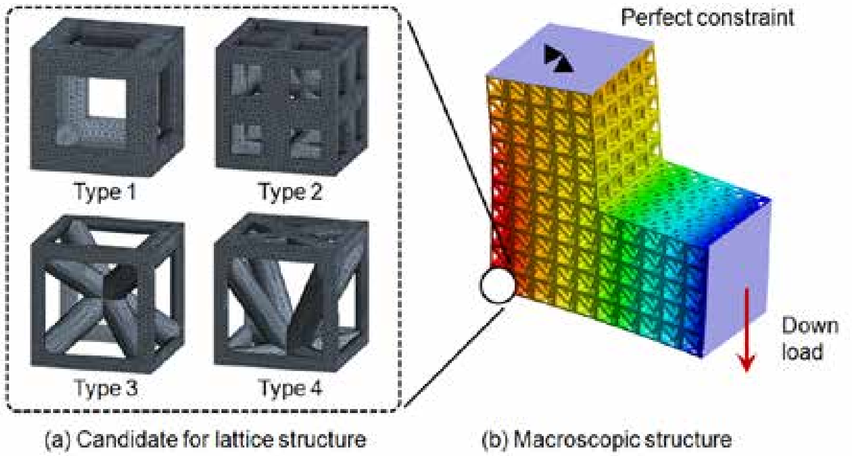


Fig. 3 - Overview of the analysis example

We have to obtain the material behavior in order to utilize these features of a lattice structure. This paper introduces analysis examples to estimate the material constants for lattice structures by means of the homogenization technique to then optimize lattice density.

Lattice optimization analysis

Analysis flow

Figure 2 shows ANSYS Workbench's analysis flow for lattice optimization. As a first step, the equivalent material constants for a lattice structure are evaluated by means of homogenization analysis using a unit cell model.

This can be performed with CMAS, an add-on tool for the ANSYS Workbench environment. As a second step, the structural analysis of a macroscopic model composed of a lattice structure is performed. The macroscopic model is then created as a homogenized solid that has equivalent material constants, as evaluated in the first step. In order to observe the detailed results in meso-scale inhomogeneity, we have to perform a sub-modeling analysis. This is the third step and should be performed as necessary. Finally, the lattice optimization is performed. The optimal lattice structure is mapped based on the topology density taken from the topology optimization analysis results acquired in the second step.

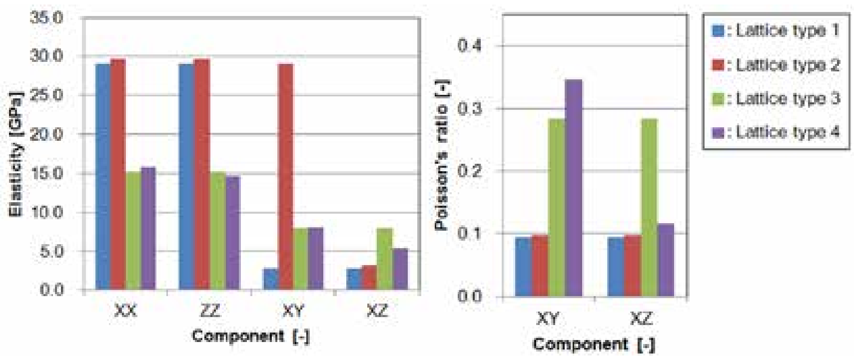


Fig. 4 - Equivalent material constants of the lattice evaluated by means of a homogenization analysis

Analysis example

Figure 3 shows an overview of the analysis example for the lattice optimization. The macro-scale object is an L-shaped bracket made of a lattice structure. We prepared four types of lattice shape, as shown at the left of Figure 3.

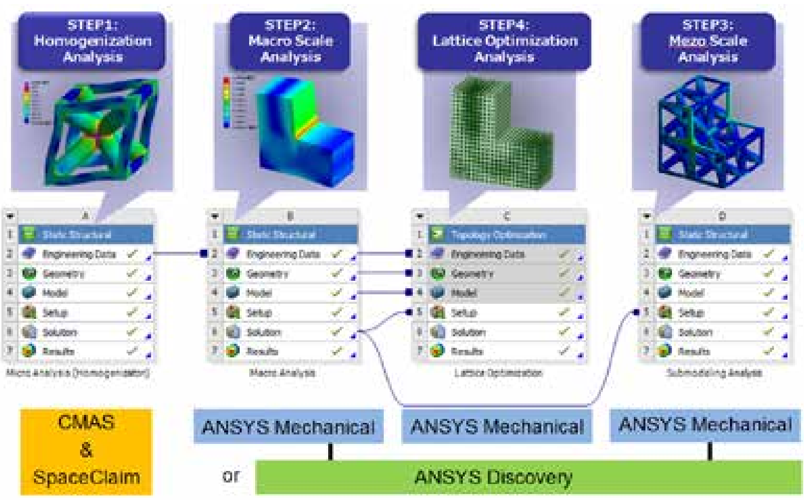


Fig. 2 - Analysis flow for lattice density optimization

All the candidates for the lattice structure have pillars with the same volume fraction, meaning that the weight saving effect is also the same.

First, a homogenization analysis of the elastic constants for each lattice structure was conducted and then a macroscopic analysis was performed by assigning the equivalent material constants resulting from the previous step.

To decide the optimum type of lattice structure for the L-shaped bracket, we need to compare the results of the macroscopic analysis using the four types of material constants.

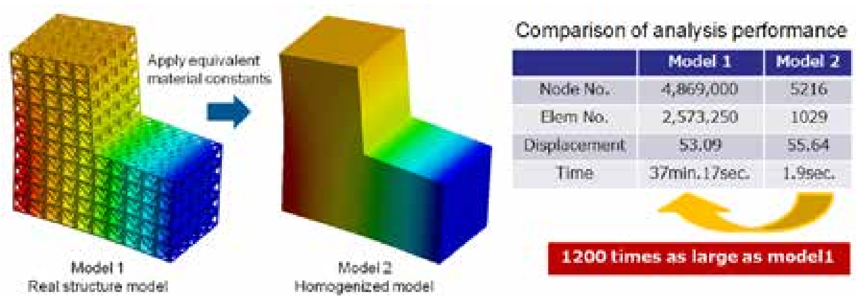


Fig. 5 - Comparison of analysis cost and accuracy between two types of models

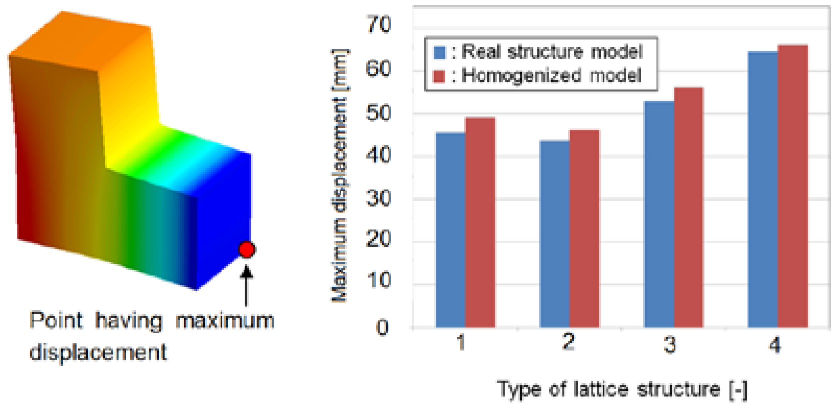


Fig. 6 - Maximum displacement as a result of the macro-scale analysis for each lattice structure

The ideal would be to change the shape of the lattice depending on its location, however, we had to specify the lattice type in advance because, in lattice optimization, ANSYS 19.2 does not provide the ability to change the lattice type, but only the thickness of the pillar. Figure 4 shows the macroscopic elastic constants and Poisson's ratio. The findings confirmed that the elastic constants and Poisson's ratio can be increased by a factor of two and three respectively, even though the type and content of the material is not changed.

Generally speaking, elasticity can be increased by orientating the material parallel to the load direction. In other words, the pillars should be oriented parallel to the direction of maximum principal stress. Therefore, we should set the orientation of the pillar to the x,y,z direction in order to obtain a high longitudinal elastic modulus, and to a 45-degree direction on the xy,yz,xz plane in order to obtain a high shear elastic modulus.

As a next step, we performed a macroscopic analysis using a homogenization solid. It is inefficient to create a meso-scale inhomogeneity in a macro-scale model because of the increased analysis cost. Figure 5 shows the analysis cost and accuracy for two types of models. Note that the node and the element number are very large in comparison to the homogenized model, as shown in the table at the right of Figure 5.

As a result, the analysis cost for a real structure model is 1200 times as large as for a homogenized model. This difference will become more significant if we consider any nonlinear effects.

Figure 6 shows the maximum displacement result of the macroscopic model for all the lattice structures. Using a homogenized model does not create practical problems, even though the results of the two types of model are slightly different.

The best choice to maximize the elasticity of an L-shaped bracket in a macro-scale model is lattice type No.2. Lastly, we performed the lattice optimization analysis. ANSYS19.2 can distribute the lattice density based on the topology optimization results, as shown in Figure 7.

As mentioned previously, it is important to select the best lattice structure type at the macroscopic analysis phase because ANSYS19.2 does not permit optimization for lattice shape but only for the thickness of the pillar.

Conclusions

Lattice structures created by additive manufacturing have several merits, namely the controllability of the material density and behavior. The lattice optimization function in ANSYS19.2 and the homogenization analysis in CMAS are very useful to optimize lattice structures by exploiting these merits. Lattice optimization can distribute the thickness of the pillar based on the topology density. It is very important to first decide the optimal lattice shape by means of homogenization analysis, since lattice optimization does not handle the shape elements of the lattice, such as angle, connection point and so on.

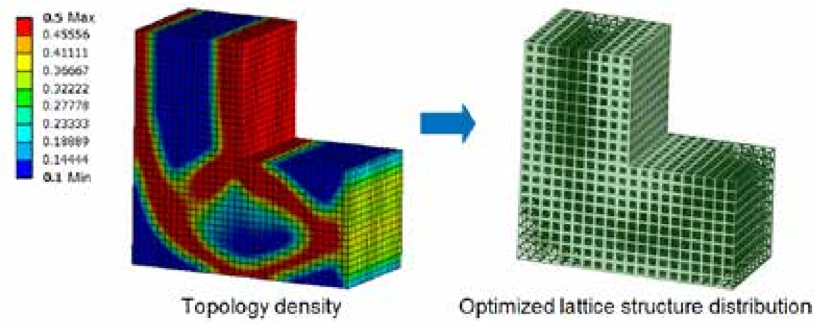


Fig. 7 - Lattice optimization results.

Koji Yamamoto - CYBERNET SYSTEMS CO

For more information:
Alessandro Mellone - EnginSoft
a.mellone@enginsoft.com

Materials Modeling Session of the International CAE Conference 2018
proceedings2018.caeconference.com

Analysis of flow characteristics in an urban area using CFD

High-rise buildings in urban areas are likely to have an important influence on both the residents and on the pedestrians walking by. Since these buildings greatly change the air-flow patterns, this irregular flow may have a negative impact on the neighborhoods. Therefore, it is important to evaluate the flow characteristics of the wind around high-rise buildings and in their vicinity in urban areas. Traditionally, the characteristics of wind flow have been evaluated by means of wind tunnel experiments. Recently, these are quickly being replaced by Computational Fluid Dynamics (CFD) simulation. For this paper, we selected a real urban area where low-rise houses and high-rise buildings are situated closely together to assess the wind flow characteristics for specific urban areas.

We used the ANSYS CFD program to evaluate the flow characteristics of these real urban areas and compared the simulation results with the test results of the measured wind speed at pedestrian level in the experiment's main location.

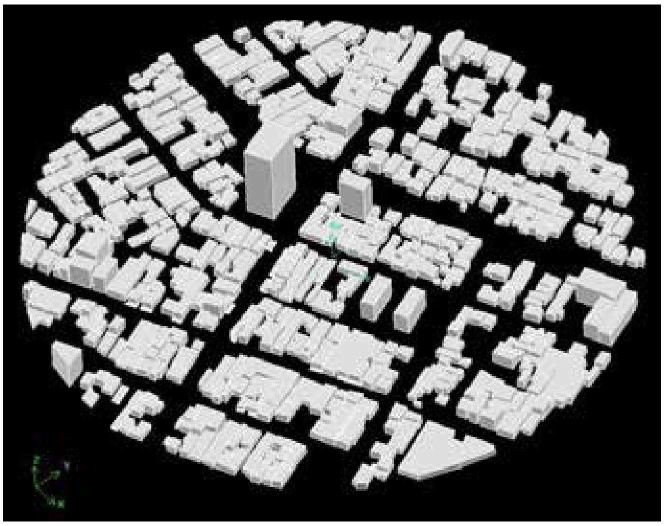


Fig. 1 – Arrangement of houses and buildings

Urban area geometry for CFD

The real urban area is shown in Fig. 1 and consists of many small low-rise houses and two high-rise buildings – one of sixty stories and one of eighteen stories – which are arranged closely together. The size of the simulation domain for the CFD analysis of the urban area is shown in Fig. 2. The proposed model was 500m (x-axis) X 500m (y-axis) X 400m (z-axis) and was constructed using ANSYS Preprocessing.

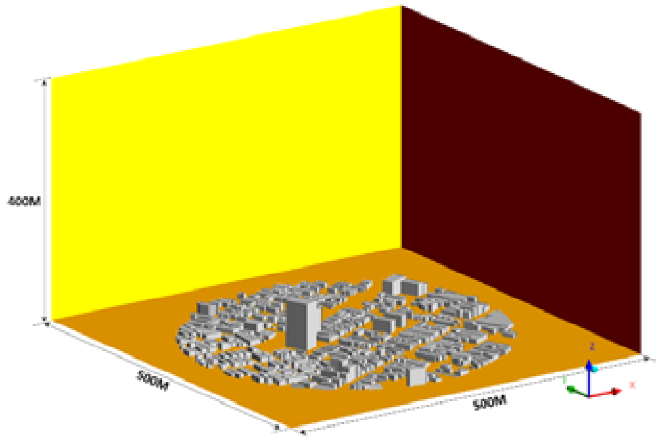


Fig. 2 - Urban area domain size for CFD

Grid distribution for CFD

ANSYS Preprocessor was used to generate the grids for the flow analysis of the urban area. Considering the size difference between the low-rise houses and the high-rise buildings, as well as the density of the buildings in the urban area, a tetrahedral-type grid system was suggested for creating the shape of the grids. We also carefully controlled the size of the grid around the region of interest in an effort to minimize the deviation of the numerical solution due to the grid's size. The total number of grids composing the analysis domain was about 4.5 million. The shape and distribution of the grids are shown in Fig. 3

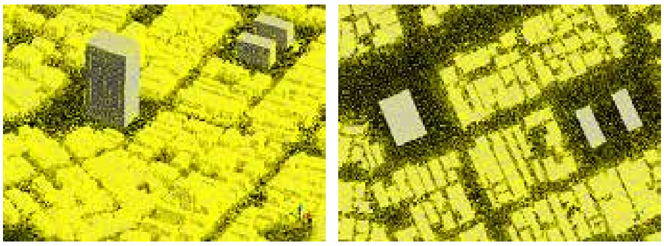


Fig. 3 - Grid distribution of the interested area

Numerical method for CFD

We used the CFD governing equations that solve for continuity, momentum, and energy conservation; the Finite Volume Method (FVM) was used to discretize these algebraic equations.
The SIMPLE method was applied to analyze the governing equations, while the turbulence model was based on the Realizable k-ε model.

Atmospheric boundary layer model

The velocity of the wind was not uniformly defined but was defined using the atmospheric boundary layer model method by applying the velocity gradient to the height (altitude) according to the ground surface's roughness characteristics, such as surface, sea surface, topography and building height.

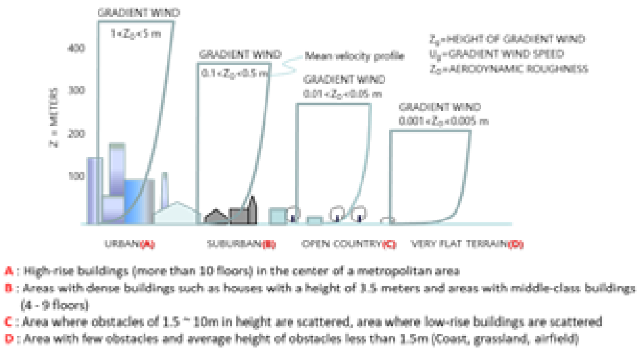


Fig. 4 - Surface roughness classification

Surface Roughness classification	A	B	C	D
Z_b (m)	20 m	15 m	10 m	5 m
Z_g (m)	500 m	400 m	300 m	250 m
α	0.33	0.22	0.15	0.10

Table 1 Z_b, Z_g, α according to surface roughness classification

(Note)
Z_b: Atmospheric boundary layer starting height
Zg: The reference gradient wind height
α: Wind speed altitude distribution index

The criterion was based on KBC2009 (KOREAN BUILDING CODE 2009). From Fig. 4 and Table 1, the surface roughness of the analyzed object could be defined as "B".
Next, the vertical directional velocity distribution had to be defined. Vertical directional velocity distribution is generally defined by log-law, power-law and NPD (Norwegian Petroleum Directorate) law (based on NORSOK standard). In this case, the power-law in the form of an exponential function was used and can be expressed as follows:

(1) u, κ, ε profile

$$U(z) = U(Z_1) \times \left(\frac{Z}{Z_1}\right)^2$$

$$k(z) = \frac{(U^*)^2}{\sqrt{C_\mu}}$$

$$\epsilon(z) = \frac{(U^*)^3}{\kappa z}$$

$$U^* = \left(\frac{\kappa(U(Z_1))}{\ln\frac{Z_1}{Z_0}}\right)$$

(2) Wind profile exponent

$$\alpha = 0.24 + 0.096 \log_{10} z_0 + 0.016(\log_{10} z_0)^2$$

Here,
$U(Z_1)$: wind speed at Z_1 [m/s]
κ : Von Karman Constant (0.4~0.42)
Z_1 : Height of meteorological station [m]
Z_0 : Aerodynamic roughness length [m]
α : Wind Profile Exponent

Boundary condition for CFD

The wind inflow condition to the urban area was set using the atmospheric boundary layer model and the User-Defined Functions (UDFs) were compiled for the application in ANSYS Fluent.
When the direction of the inflow is + x and U(Z_1) in the power-law function formula is defined as 4.5 m / s, the vertical directional velocity profile was confirmed as illustrated in Fig. 5.

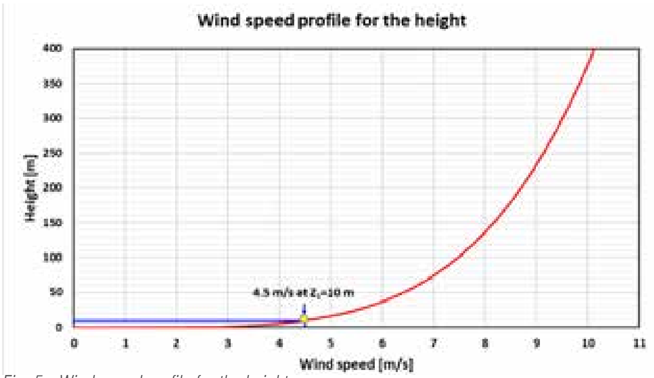


Fig. 5 - Wind speed profile for the height

Results

Wind flow analysis was carried out to evaluate the flow characteristics around high-rise buildings and in their vicinity.
Figure 6 shows the pressure distributions for the low-rise houses, high-rise buildings and ground surface.
The high-pressure region around the low-rise houses and the high-rise buildings can be identified. In particular, the pressure difference was relatively large to the front and to the rear of the high-rise buildings.
Figure 7 shows the velocity distribution around the low-rise houses and the high-rise buildings.
As a result of the pressure difference and the flow separation at the high-rise building, we could find the low speed region (top) and the velocity in the opposite direction of the main flow

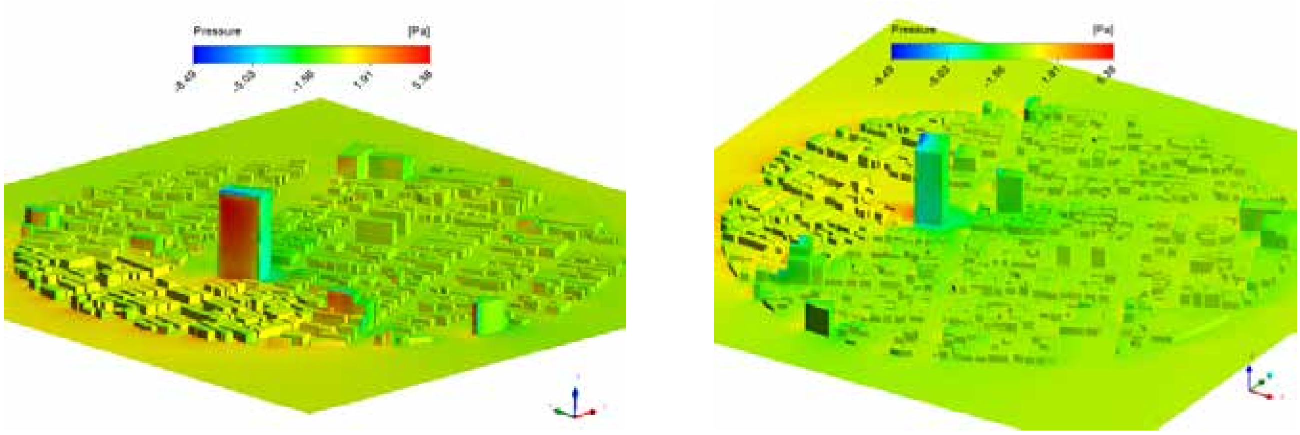


Fig. 6 – Pressure distribution (top, bottom) at the buildings and the ground surface

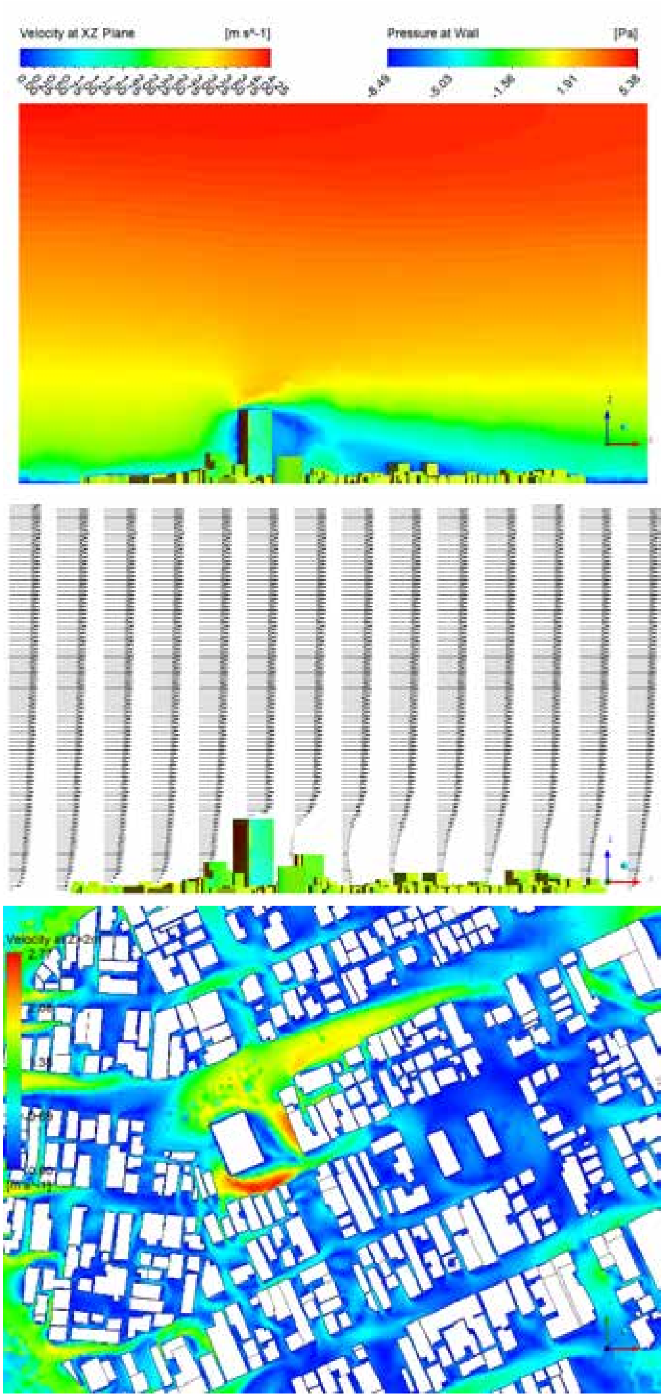


Fig. 7 – Velocity distribution (top, middle, bottom) around the buildings

direction (middle). The contour plot (bottom) is the result found at pedestrian level. The high-rise building divided the wind to both sides and a strong velocity magnitude was observed around the high-rise buildings.

Figure 8 is the streamline result around the high-rise buildings 20m above the ground.

Due to the wind separation at the high-rise buildings, the flow at the front of the high-rise building shows a strong downward flow characteristic. At the rear, a strong swirl region was observed.

There was no decrease in wind speed as the height increased due to the ground effect; free-flow disturbance occurred only in the vicinity of the high-rise buildings due to the size of their cross-sectional area. Figure 9 shows the streamline characteristics according to the height (2.0m, 5.0m, 10.0m, 15.0m and 20.0m) above the base of the high-rise buildings.

Figure 10 shows the wind speed measurement at 2.0m from the ground. The CFD results were compared to the wind speed measured at the eighty locations.

Comparing the results of the wind tunnel experiment and the CFD results, the CFD simulation results display a reasonably good match with the measurements.

Summary

Using the ANSYS CFD program, the flow characteristics in real urban areas were compared to the wind tunnel experiment results. Through these studies, the wind flow characteristics and the wind

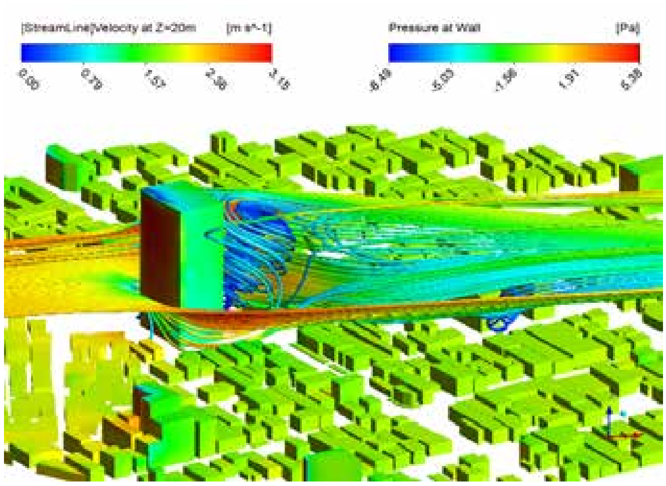
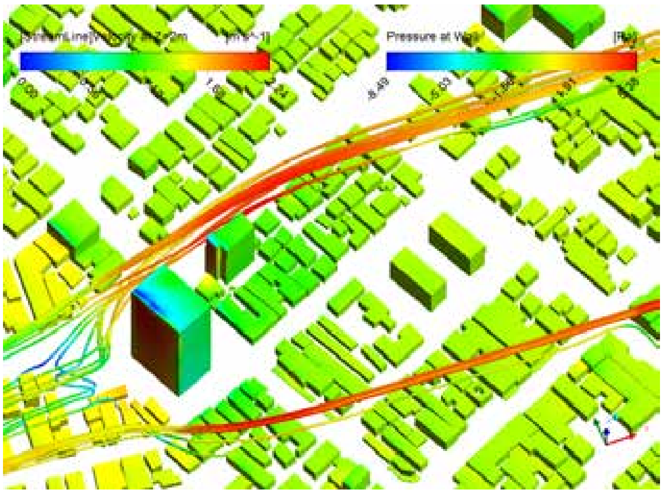
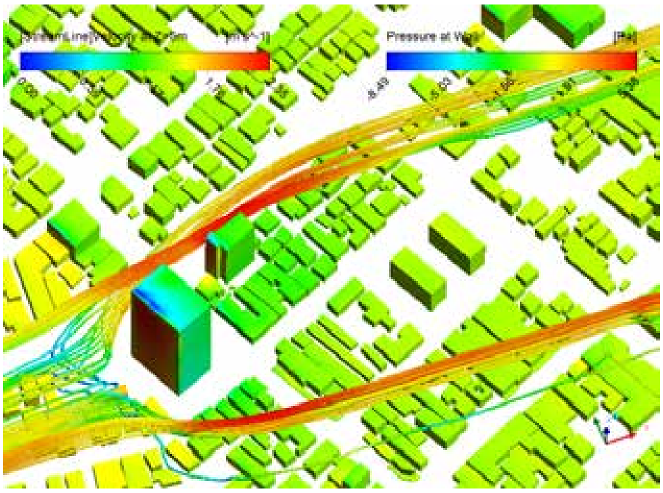


Fig. 8 – Streamline around the high-rise building

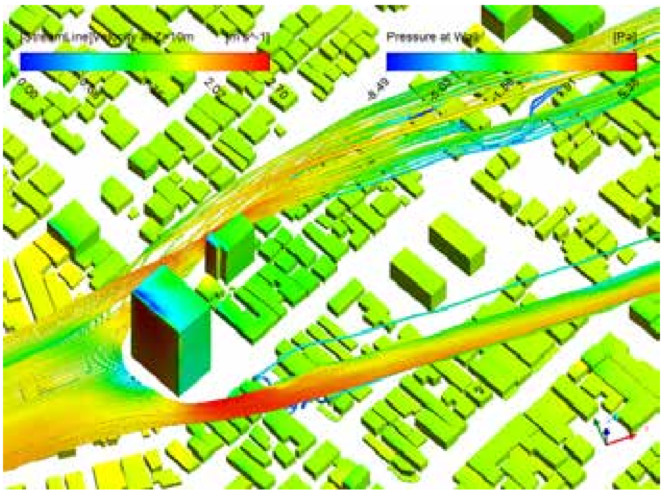
Streamline at 2.0m from the ground



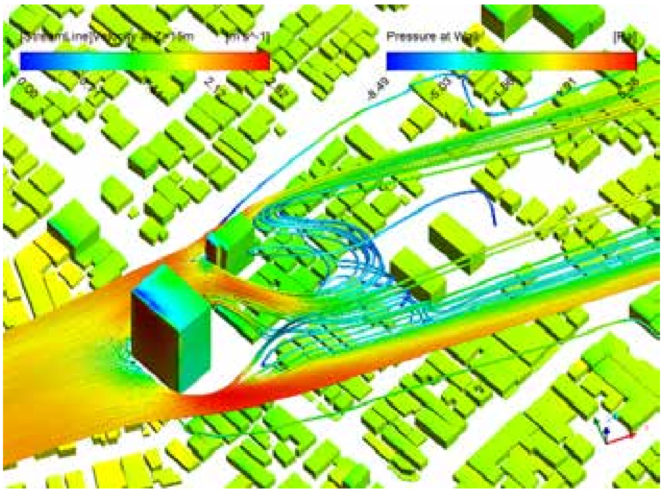
Streamline at 5.0m from the ground



Streamline at 10.0m from the ground



Streamline at 15.0m from the ground



Streamline at 20.0m from the ground

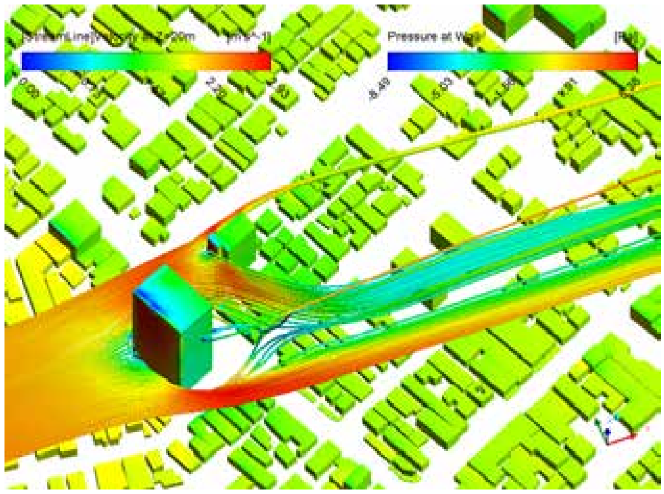


Fig. 9 – Streamline around high-rise buildings by height

characteristics at pedestrian level were confirmed for an area where low-rise houses and high-rise buildings are closely located. Finally, we confirmed the validity of CFD and calculated the wind flow characteristics of real urban areas with high accuracy.

Jinwon Seo

* Jinwon Seo is Senior Application Engineer at Tae Sung S&E, Inc. (www.tsne.co.kr.). For more information, email: jwseo@tsne.co.kr

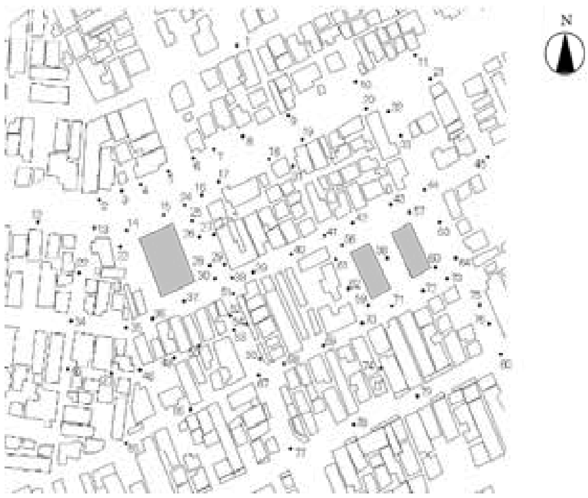


Fig. 10 – Wind speed measurement at 2.0m above the ground

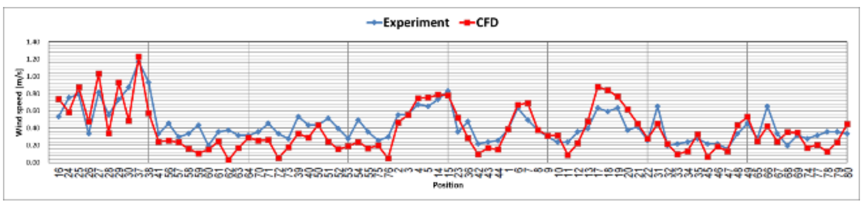


Fig. 11 – Comparisons of experiment and CFD

Thermal Management of the ALPIDE Space Module for Particle Tracking

Silicon complementary metal-oxide-semiconductor (CMOS) Monolithic Active Pixel Sensors (MAPS) are used for high-energy physics experiments. These sensors, which detect charged particles, are gaining prominence in the fields of nuclear, radiation and medical physics. MAPS are very thin (down to 50 um) and are made using cost-effective, commercial CMOS technology. The ALPIDE sensor, which was developed at CERN for the ALICE Inner Tracking System (ITS) Upgrade, has recently been proposed for use in space applications (ALICE is the name of CERN's Large Ion Collider Experiment). The major design issues with this sensor concern the thermal management of the power needed by the detector modules that group several sensors. We discuss how to deal with this by using ANSYS thermo-structural simulations, as well as discussing the module design and its integration with light-weight/high thermal-conductance composite Carbon Fiber Reinforced Plastics (CFRPs). With regard to heat dissipation, we demonstrate the possible integration of an ALPIDE-based module into a new Chinese Seismic Electromagnetic Satellite in the near future.

The China Seismic Electromagnetic Satellite (CSES) mission studies the near-Earth electromagnetic, plasma and particle environments in the transition zone ionosphere-magnetosphere. Its main objective is to search for correlations between the occurrence of large magnitude ($M > 6$) seismic phenomena and electromagnetic perturbations in the inner Van Allen belt. The existence of a time correlation between particle bursts (PBs) and large-magnitude earthquakes has been suggested by several authors, and experimental results from the METEOR-3A and AUREOL-3 satellites [1] are still under debate, making it necessary to develop new satellites to detect the phenomenon. The CSES mission is a joint effort between the Chinese National Space Administration (CNSA) and Agenzia Spaziale Italiana (ASI); its first multi-instrument satellite (Figure 1) was successfully launched on 2 February 2018 from the Jiuquan Satellite Launch Centre in the Chinese Gobi Desert. The CSES mission's multi-channel approach aims at extending the exploratory studies of seismic-electromagnetic phenomena already carried out in space by other missions (DEMETER).

The instrumentation on board the satellite consists of several detectors with different measurement functions:

- Electrical and magnetic fields and their perturbations in the ionosphere: search-coil magnetometer, fluxgate magnetometer, electrical field detector
- Disturbance of plasma in ionosphere: plasma analyzer and Langmuir probe
- Flux and energy spectrum of the particles in the radiation belts: High Energy Particle Package and High Energy Particle Detector
- Profile of electronic content: GPS occultation receiver and tri-frequency transmitter

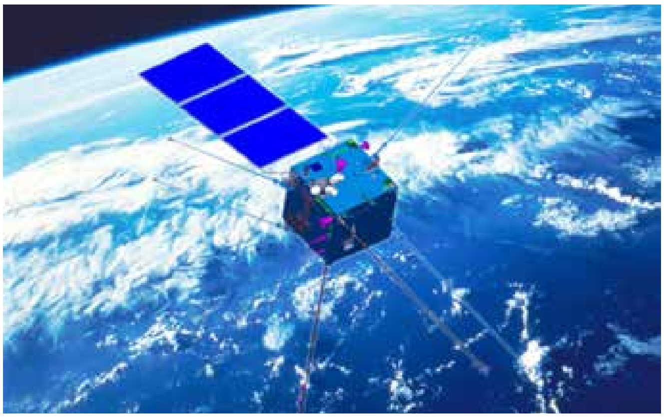


Fig. 1 - The CSES satellite orbiting at an altitude of 507 km (low earth orbit). Courtesy of [1,2]

In this paper, we focus on a specific module of the CSES satellite: the High Energy Particle Detector (HEPD). Developed by the Italian collaboration LIMADOU, HEPD is sensitive to electrons in the 10-200 MeV energy range and to light nuclei in the 30-300 MeV/nucleon energy range. HEPD is very sensitive to:

- Seismic-induced perturbations of the inner Van Allen belt (particle precipitations)
- Solar-terrestrial environmental changes
- Solar impulse activity (e.g. Solar Energetic Particle (SEP) events)
- Solar modulation of low-energy cosmic rays
- Low-energy cosmic rays (p, He, Li, Be, B, C)

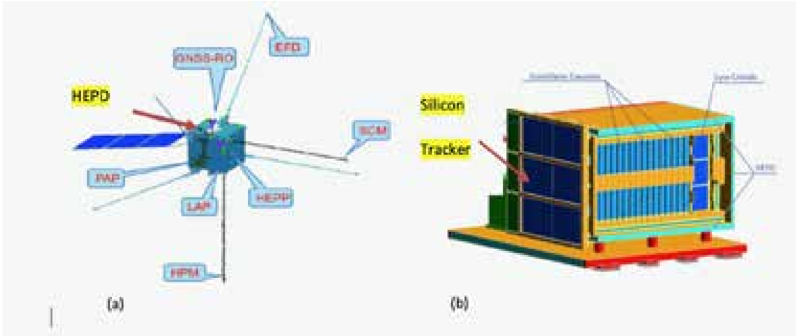


Fig. 3 - (a) The instrumentation on board the CSES satellite; (b) The CAD model of the HEPD instrumentation. Courtesy of [2].

Figure 2 shows an example of a particle trajectory trapped in the geomagnetic field, and the HEPD energy range interval. CSES is intended to be the first of a constellation of satellites and the design and construction process of CSES-2 has already started. This paper discusses the thermal issues related to the attempt to improve the HEPD-2 payload's performance in terms of energy threshold, pointing resolution, read-out rate and data compression. One major change could see the replacement of the silicon tracker module – currently made of two planes of double-sided silicon micro-strip detectors – with a new tracker based on the Monolithic ActivePixel Sensor ALPIDE, designed at CERN for the ALICE ITS upgrade [2].

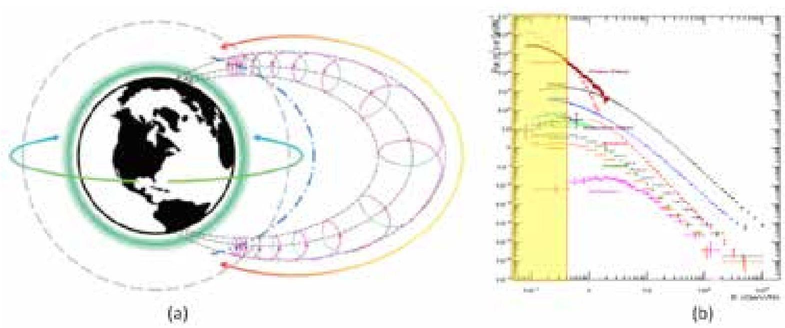


Fig. 2 - (a) Particle path trapped by the geomagnetic field; (b) Particle flux with HEPD energy window (in yellow). Courtesy of [2].

ALPIDE is very thin (reducing the effect of multiple scattering), is low-cost, features low power-consumption for standard applications in ground-based High Energy Physics, and improves the capabilities in terms of read-out rate,. To operate ALPIDE in space, however, the tracker modules must be redesigned to account for space constraints. One of the main differences from its use at CERN pertains to the possibility there of dissipating the power to a heat-transfer circuit based on water convection. This solution is not viable in HEPD-2 so heat will have to be transferred by conduction only, while keeping the thermal gradient along the tracker modules as low as possible to guarantee the uniformity of the detector response. This paper reports on the estimation of this power dissipation and describes how to transfer the thermal power to the heat sinks via a Carbon-Fiber Reinforced Plastics (CFRPs) module. This module must comply with the mechanical precision-assembly process at facilities of the Italian Istituto Nazionale di Fisica Nucleare. The module is was analyzed using the ANSYS Workbench Multiphysics software tool.

The HEPD Module with ALPIDE sensors

The CSES-HEPD module is made of:

- a Silicon tracker that provides the direction of the incident particle;
- the Trigger: a layer of thin plastic scintillator divided into six segments (200 mm × 30 mm × 5 mm each); and
- the Calorimeter: a tower of 16 layers of 1 cm-thick plastic scintillator planes followed by a 3 x 3 matrix of inorganic scintillator (LYSO).

In the new CSES-HEPD-2, three layers of ALPIDE will be used to lower the energy threshold, and to improve the angle resolution and the read-out speed.

The new CSES-HPDE-2 will replace the previous module that contained the silicon-strip tracker. Silicon-strip trackers rely on a mature technology, so substituting this module with one containing the ALPIDE Monolithic Active Pixel Sensors (MAPS) will require a new tracker design.

The ALPIDE pixel sensor

The ALPIDE sensors were designed at CERN and were produced by Imaging Process at the TowerJazz facility to exploit 180 nm technology with 3 nm-thin gate oxide and six metal layers. The high-resistive epitaxial layer is grown on a p-substrate thin wafer (total thickness 50 um). Being a quadruple-well process,

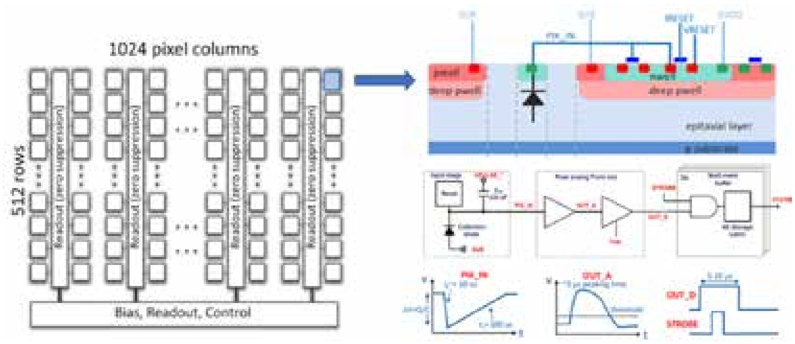


Fig. 4 - The ALPIDE pixel matrix architecture (left); Sketch of the cross-section of the pixel cell showing the collection diode and the front-end CMOS electronics (top right). Circuits for detection, discrimination and digital conversion and storage of the signal produced by the particles (bottom right). Courtesy of [3].

it offers a deep PWELL, which shields the NWELL of the P-type metal-oxide-semiconductor (PMOS) transistors. Those transistors form the front-end electronics making it possible to use the full CMOS circuitry in the pixel area without the drawback of parasitic charge collection in the transistor NWELL. The application of a moderate negative voltage (around − 3 V) to the substrate can be used to increase the depletion zone around the collection diode which helps to improve both the charge collection and the Signal-to-Noise ratio by decreasing the pixel capacitance. The interconnections between the transistor and the diodes will be realized using six metal layers to allow a large number of control functions to be integrated into a single pixel. Data can be transmitted using two different readout ports. A 1.2 Gbps serial output port with differential signaling is intended to be the largest capacity data readout interface. A bi-directional parallel data port with single-ended signaling is also present, with a capacity of 320 Mbps. The ALPIDE chip measures 15 mm x 30 mm and includes a matrix of 512 x 1024-pixel cells, each one measuring roughly 30x30 μm2. Analog biasing, control, readout and interfacing functionalities are implemented in a peripheral region of 1.2 × 30 mm2. Figure 4 shows an image of the pixel matrix forming the ALPIDE chip, together with a cross-section of the ALPIDE pixel and the scheme of principle for the amplification and digitalization of the signal produced by particles crossing the p-epitaxial regions. Circuits for the amplification, discrimination and storage of the signal are also shown in the figure. The double-column architecture has an intermediate Priority Encoder that sequentially provides the addresses of all hit pixels; there is no activity if there are no hits, thus lowering the power consumption to 3 mW for each hit. Their stable temperature and low power consumption, together with their detection efficiency > 99%, their fake hit rate << 10-5 /event/pixel, and their position resolution < 5 um all make ALPIDE chips a good potential platform for space applications

Power constraints

The power budget of a satellite payload is defined at the beginning of the mission and cannot be exceeded. Therefore, it was necessary to reconfigure the ALPIDE chip's functions with respect to those used in the ALICE ITS Upgrade to adapt the ALPIDE chip to the low power-consumption requirements of the satellite.
We only focused on the strategy that was used to save power digitally:

Readout line/ operation	Max current in digital line	Average current In digital line	Max Power consumption
Power on*	Variable, up to 120 mA	12 mA	216 mW
DATA	134 mA	114 mA	240 mW
CTRL	67 mA	51 mA	120 mW

Table 1: The current flow in the digital lines, and total power consumption, when test signals are applied to the DATA line, the CTRL line, and in the Power-on operational mode of the chip ("Power on" is the most critical phase and should be checked in the future).*

(1) to use the control line only to send commands, read/write registers, and read data;
(2) to use lower clock frequencies (the control line works up to 5 MHz);
(3) clock gating between trigger commands.

In practice, we used restricted functions of the ALPIDE sensors with a low scanning rate (trigger rate < 1k Hz) and a bi-directional serial line (control line – CTRL at 40 Mbps) not only for sending sensor commands and reading registers, as used in ALICE, but also for reading data. In this way, we could completely bypass the High-Speed Serial Data (HSSD) available on the ALPIDE sensor, which is very power intensive due to its massive use of Phase-Locked Loops (PLL). To this end, the test set-up was based on a high-speed communications board called MOSAIC that uses a specific software that enables it to act as a Data Acquisition (DAQ) interface with the modules (Figure 5). The ALPIDE chip is designed with a special feature that allows a self-injection charge in the collection diode using a capacitor located inside the pixel. Using this feature, it is possible to test all the collection's electronic chain without irradiating it with charged particles. The ALPIDE chip's power consumption was estimated using two current probes. The system is fed with 1.8 V during routine operations in the chip and using both the CTRL and the fast DATA lines. We estimated the maximum power dissipated by the chip by combining the measurements of both probes. The results for three modes of operation (Power on, DATA, CTRL) are summarized in Table 1 below.

By imitating a configuration of the ALPIDE chips that is similar to that of the InfiniBand (IB) Host Interface Card (HIC) as used in the ALICE ITS Upgrade, we considered a 5+5 layout for the mechanical unit, where the chips working as the master are glued onto a rigid stave. They have to be connected to an electronics control mounted on a Flexible Printed Circuit (FPC) containing the data and control buses. This chip pad will be wire-bonded to the FPC bus. In the following part, we only considered the dissipation related to the chips according to the worst case reported in Table 1. While strategies for lowering the power dissipation are under investigation (i.e. distributing the clock for the readout), we used the CTRL line to estimate the temperature gradient across the mechanical staves.

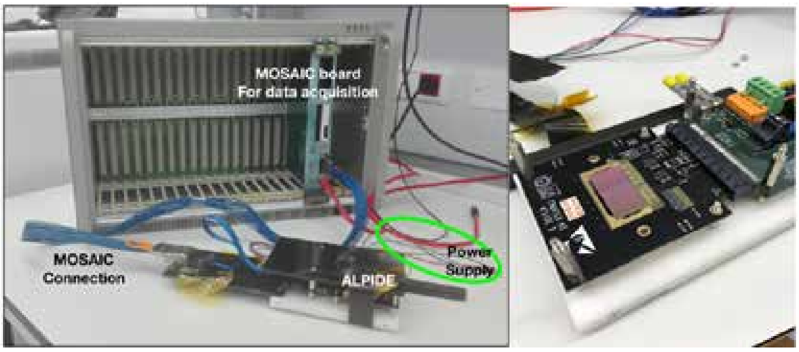


Fig. 5 - The set-up for measuring the power dissipation (left); A detailed view of the ALPIDE chip mounted on the test board which is connected to the MOSAIC high-speed board (right).

Particle Tracker thermal models

To improve reliability and prevent premature failure of the sensors, we proposed a mechanical layout that must satisfy two fundamental requirements:

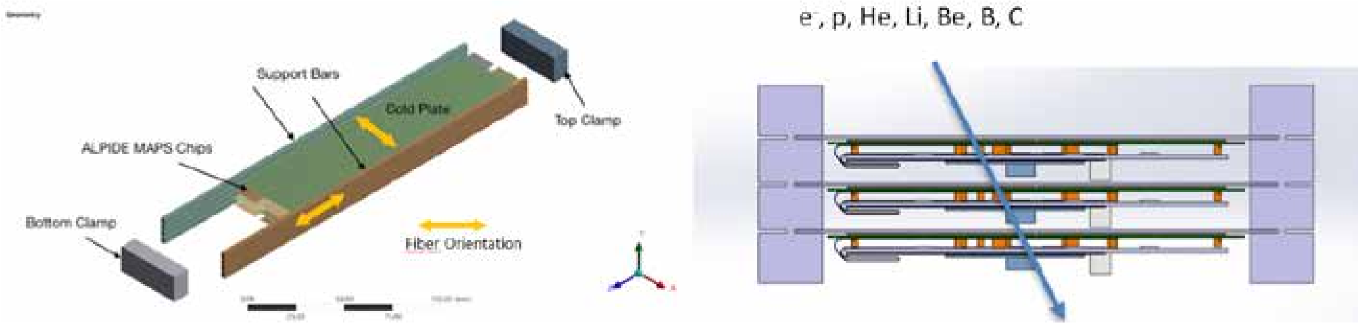


Fig. 6 - (left) The exploded-view drawing of model 1 showing the mechanical unit with the CFRPs fiber orientation; (right) The cross-section of a mechanical unit showing the three cold plates clamped to the support bars and a bunch of particles crossing the three layers.

mechanical stability in a harsh vibrating environment (such as the launch and operating phases), and the thermal management in electronic circuits. Non-optimal temperature gradients between ALPIDE chips could affect their performance and this must therefore be considered in the design. The basic design consideration to satisfy both requirements is to derive the structural stability and thermal management from the Alice ITS Upgrade's cooling system [4]. In this system, staves made from Carbon-Fiber Composite-Reinforced Plastics (CFRPs) work as high-conductive thermal links between the chips and the cooling pipes integrated into the stave. The CFRP staves also provide a high-strength mechanical component. In the Alice ITS Upgrade, heat exchange is facilitated by the cooling pipes integrated into the staves, but this solution is not allowed in the satellite. Therefore, our main task was to minimize the thermal gradient to within 5° C on the cold plate which was quite a challenge. We addressed it by developing 3D geometries of the tracker, together with a combination of materials that maximize the heat rate flow from the chips to the heat sinks, which was made possible by an extensive use of -ANSYS Workbench's thermal analysis and optimization cycles. The detail of our analysis follows.

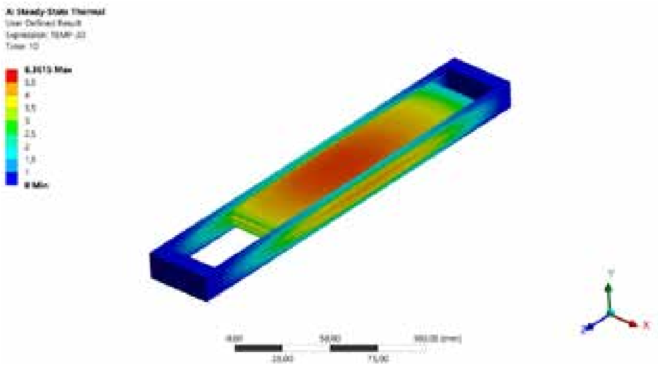


Fig. 7 - Temperature gradient on the cold plate with respect to a reference temperature of 30°C

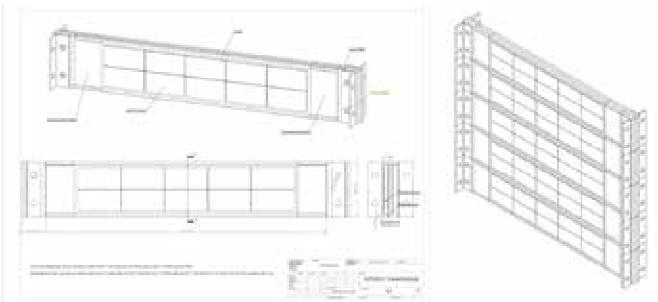


Fig. 8 - An image of the module assembly showing three staves with five mechanical units clamped by L-shaped joints

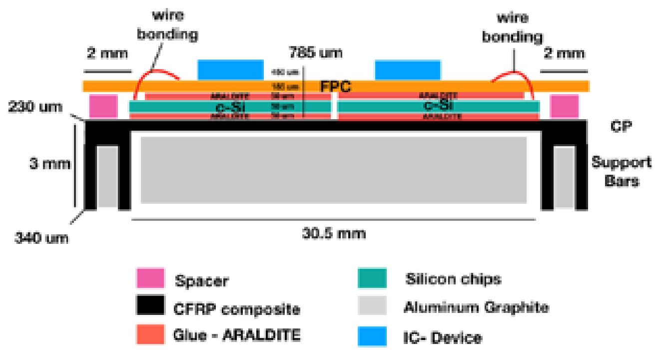


Fig. 9 - A cross-section of one stave in the mechanical unit showing the new concept with the internal structure of ply lay-up composites, the cold plate (CP), the support bars, the c-Si (ALPIDE) chips, and the FPC board containing the control electronics.

A mechanical unit consists of three layers with 5+5 chips glued onto a CFRP stave coupled to the scintillator and calorimeter to detect the particles' position and energy. The whole tracking module is created by assembling five mechanical units. The total number of chips, therefore, is 150, representing a total power dissipation of 18 W (120 mW per chip). This is a huge amount of power for a satellite and is what motivated the choice of very high conductive materials made of CFRPs. Moreover, the CFRP stave does not interact with impinging particles regardless of its thickness.

Thereafter, a passive cooling system using CFRPs and Aluminum Graphite Composites (Al-GCs) was implemented to dissipate the power produced by the ALPIDE sensors. The heat flux is collected by the stave and follows the carbon fiber orientation to reach the Al-GCs heat sink. The CFRPs components are made of ply lay-up composite and the fiber orientation is determined by the desired thermal/strength requirements. Two different configurations for optimizing the heat flow from the ALPIDE sensors were analyzed for each single layer of the mechanical unit:

1. Model 1 - a cold plate with the fiber orientation forming a 90° angle to the stave axis, and two support bars with a fiber orientation along their axes. Ten ALPIDE sensors were glued to each stave for a total of 30 device chips for each unit. The heat sinks are blocks made of Al-GCs. Two Al-GCs heat sinks clamp the ends of the plate and are connected to the frame (Figure 6)
2. Model 2 – a cold plate with the fiber orientation along the two axes, and with support bars integrated onto the plate (with the

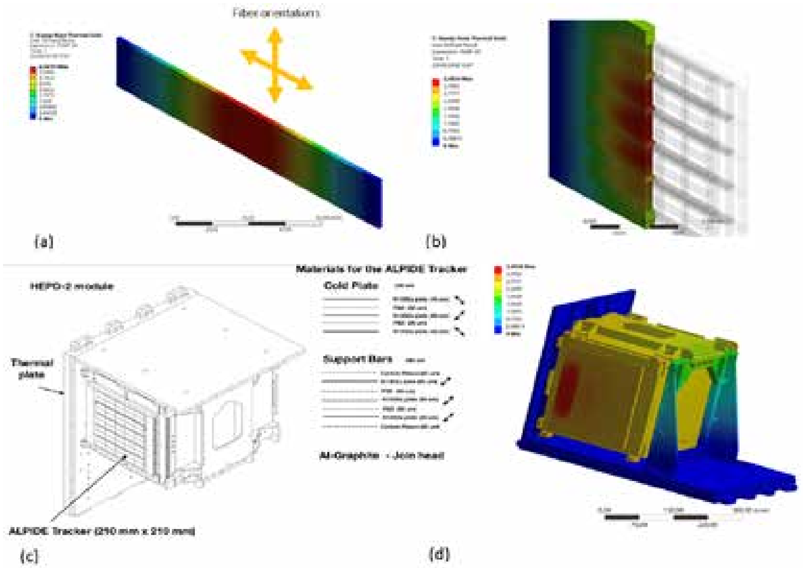


Fig. 10 - (a) One stave with 5+5 ALPIDE sensors; (b) The ALPIDE tracker with five mechanical units assembled; (c) The ALPIDE tracker connected to the structure containing the calorimeter linked to the thermal plate; (d) Simulation of the structure keeping the thermal plate's temperature at 30° C and with only one mechanical unit fed.

fibers oriented along their axes). The heat sinks are embedded into the stave terminals and connected to the frame (Figure 7).

Model 1 was developed for estimating the maximum thermal gradient inside a mechanical unit, while Model 2 was conceived as a prototype for the final tracker's geometry since it is more compliant with precision assembly. The hypotheses for the thermal analysis of models are:

- a single mechanical unit works independently without heat exchange with the other cells. No radiation or convection heat exchange is considered. Convection can be ignored because the satellite would be in an orbital trajectory in outer space and therefore subject to a hard vacuum (10-6 - 10-9 torr). While radiating surfaces may be present in the module, they can be effectively shielded by using low emissivity materials;
- the materials have no dependence on the temperature, and if there is some dependence, the small temperature gradient does not affect the performance. CFRPs are considered orthotropic and we evaluated that there is effective thermal conductivity in the ply layout;
- the contact resistance between the chips and the stave is only determined by the thermal conductivity of the glue. No thermal interfaces are considered between the staves and the Al-GCs heat sinks.

For the above reasons, the analysis is linear and can be run on a 4-processor workstation. When we performed the analysis on the mechanical unit, the two middle surfaces of the CFRPs support bars were considered adiabatic while the Al-GCs heat sink is linked to the reference temperature of 30 °C. Figure 7 reports the results for the thermal gradient < 7 °C.

In Model 2 we were able to reduce the thermal gradient as shown in Figure 10 (a), which is below 5° C in one stave; by extending the analysis to the assembly of the whole module, we get similar results to those reported in Figure 10 (b). Moreover, on performing the analysis with the tracker module clamped to the satellite structure and fixing the temperature on the thermal plate, we obtained the results shown in Figure 10 (d).

Conclusions

The ALPIDE pixel sensor was developed by CERN for the ALICE ITS Upgrade (2019-2020). Designed for high-rate events, it is a high-resolution and light particle detector that combines a high read-out speed with low power consumption. The ALPIDE detector was reconfigured for space applications. Power dissipated in the chip can be reduced to 120 mW if only the CTRL line is active (e.g. read/write operations). Moreover, we demonstrated that the dissipation of power produced by the module containing 150 sensors can be efficiently transferred from the sources to the heat sink (thermalization plate) while maintaining a thermal gradient on the cold plate at < 5°C. Two thermal models based on carbon fibers (CFRPs) implementing two schemes for power dissipation were developed and analyzed. By exploiting ANSYS Workbench's advanced toolkit for thermal analysis we were able to develop a virtual prototype of the silicon tracker that can be assembled and integrated into the future HEPD-2 satellite subsystem.

References

[1] Ambrosi Giovanni et al.: "The HEPD particle detector of the CSES satellite mission for investigating seismo-associated perturbations of the Van Allen belts", SCIENCE CHINA Technological Sciences, 61, 643 (2018)

[2] Giuseppe Osteria: "The HEPD detector on board CSES satellite: in-flight performance", presentation at 14th Pisa Meeting on Advanced Detectors - INFN, La Biodola, Isola d'Elba, Italy - 2018, 27 May - 2 June

[3] Gianluca Aglieri Rinella: "The ALPIDE pixel sensor chip for the upgrade of the ALICE Inner Tracking System, Nuclear Instruments and Methods in Physics Research (section A), Volume 845, Pages 583-587 (2017)

[4] ALICE ITS Upgrade - Space Frame and Cold Plate Characterization technical report (technical report - restricted use) (2016)

Enrico Serra, Benedetto Di Ruzza, Corrado Gragiulo, Roberto Iuppa, Christian Manea, Irina Rashevskaya and Ester Ricci

For more information, contact:
Istituto Nazionale di Fisica Nucleare (INFN), TIFPA in Trento, Italy,
email: enrico.serra@tifpa.infn.it or visit the website at:
www.tifpa.infn.it
CERN – The European Laboratory for Particle Physics,
Physics Department in Geneva, Switzerland

Super parameters massively reduce time and complexity to optimize shapes for improved fluid dynamic behaviour

FRIENDSHIP SYSTEMS

Simulation-driven design (SDD) is product development using (extensive) virtual prototyping via a tight coupling of modelling and simulation. While modelling is the definition of shapes, simulation is performance prediction via approximate models, omitting less important characteristics. Flow simulations by means of Computational Fluid Dynamics (CFD) are particularly resource-intensive. Consequently, each variant to be evaluated during a design campaign, even if virtual, is expensive. In order to understand design spaces, a reasonably high number of variants need to be studied, a typical estimate being the square of a system's degree-of-freedom. Over the past decades, high-fidelity modelling approaches have been made available in Computer Aided Design (CAD) systems in which the shape of the future product is defined (or changed) by as few geometric parameters as possible. Nevertheless, even with just ten to twenty parameters being chosen as free variables, many variants – often several hundred – need to be investigated. Hence, every free variable that can be eliminated or spared makes a difference. Very recently, a complementary approach of massive parameter reduction was proposed, based on Karhunen-Loève Expansion (KLE), which allows one to represent the original model's variability with just a few "super parameters".

Motivation

The success of many products directly depends on their fluid-dynamic behaviour. The shape of turbochargers, ship hulls, engine components etc. influences their energy efficiency, vibration excitation, noise emission, flow homogeneity, pressure drop and many more key aspects. Consequently, the optimization of complex shapes by means of simulation is being pursued with ever-increasing intensity. To this end, high-fidelity CFD codes are frequently employed. While the accuracy of the solvers has steadily improved particularly for the ranking of design variants, even the sound increase in computational resources cannot compensate for the fact that CFD simulations are often costly. Hence, in Computer Aided Engineering (CAE) a lot of effort is being put into providing sophisticated parametric models that would define shapes with as few parameters as possible. This is because in design and, ultimately, in optimization, the effort of changing geometry and reevaluating performance scales easily with the square of the system's degrees-of-freedom (DoF), i.e. the number of free variables with which to describe and control the design task.

This article is an abridged version of [1]. It introduces a method to massively reduce parameter spaces even further. The method, known as Karhunen-Loève Expansion (KLE), has been implemented within CAESES® (www.caeses.com), a versatile CAE platform with integrated capabilities for process automation and shape optimization. This enables design teams to build and analyze parametric models and, furthermore, to reduce parameter sets, utilizing the principle components for subsequent optimization campaigns.

The Karhunen-Loève Expansion

Karhunen-Loève Expansion (KLE) mathematically belongs to the family of principal component analysis. A large set of shapes, which have to be topologically identical, is generated by means of a Design-of-Experiment (DoE) in a CAD space. These statistically uncorrelated shapes are then geometrically analyzed, and a new design space is derived that is, by definition, orthogonal. The coordinate directions or modes of this so-called KLE space are interpreted as "super parameters" since, typically, a considerably lower number of KLE parameters already retains much of the variability of the original shapes.

		HVAC duct	RoPAX ferry	SWATH OSV	Compressor component
Number of free variables of the original CAD model (DoF)		14	14	27	30
Number of Sobol variants used for KLE		1000**	3000	3000	3000
Variability reached with 1st super parameter	1	83.84 %	92.38 %	72 %	54.47 %
Variability reached with 1st and 2st super parameters	2	92.05 %	**98.33 %**	86 %	89.59 %
Variability reached with the first three super parameters	3	**95.76 %**	99.34 %	94 %	94.89 %
Variability reached with the first four super parameters	4	97.44 %	99.76 %	**96 %**	**96.90 %**
Variability reached with the first five super parameters	5	98.51 %	99.93 %	98 %	97.57 %
Variability reached with the first 10 super parameters	10	99.72 %	99.99 %	99 %	99.45 %
Number of super parameters needed to reach more than 95 % variability		3	2	4	4
Ratio of number of free variables of the original CAD model and number of KLE variables needed to reach 95 % variability *[square]*		4.67 *[21.8]*	7.5 *[56.3]*	6.75 *[45.6]*	7.5 *[56.3]*

Table 1: Variability reached by super parameters for different shapes

A shape modification covariance matrix centred about the mean shape is built from the positions of all the variants' vertices. The Eigenvectors of this matrix then form the basis of the KLE space while the associated Eigenvalues represent the variance retained.
In this context, the term variability is defined as the sum of the Eigenvalues associated with each Eigenvector divided by the total geometric variance of the original CAD model. One hundred percent variability would mean that all variants considered within the DoE can be exactly reproduced. Super parameters are typically ordered according to their importance for capturing the original shapes. Interestingly, quite often 95% of the variability is already attained with very few of the most influential super parameters, see Table 1, allowing a design team to work within massively reduced design spaces which potentially reduces the number of necessary simulations by up to one order of magnitude. Details about the examples are given in [1].

Back-transformation from KLE space to CAD space

In principle, when analysing a parametric model two scenarios can be distinguished:

1. The number of CAD variables is quite similar to the number of KLE variables: This means that the free variables of the original CAD model are rather independent, and their number cannot be tangibly reduced, at least not from purely geometric considerations. It then makes sense to keep on working with the CAD model without switching to super parameters.
2. The number of CAD variables is considerably higher than the number of KLE variables: This means that it could be advantageous to replace the original CAD model with its corresponding KLE model in order to benefit from the reduced set of super parameters.

In the case of scenario 1, the classic approach of shape optimization would be followed which typically comprises the stages of selecting a set of parameters as free variables, setting their bounds and values, generating the geometry (CAD), undertaking the pre-processing (e.g. build a watertight domain), generating a mesh and performing the simulation (say a CFD analysis) and, finally, repeating this quite a few times for different values of the free variables.
In the case of scenario 2, a back transformation needs to be undertaken since the optimization takes place in the KLE space while the actual shape and flow domains are derived from a number of operations in the CAD model defined in CAD space. Again, see [1] for details.

Illustrating example: HVAC duct

The process shall be explained in detail for a design task that is complicated enough to illustrate all the necessary steps yet easy enough to follow. The example chosen is that of a Heating, Ventilation and Air Conditioning (HVAC) duct which can be found in many industries. In an HVAC duct, a fluid enters an inlet plane and is guided to an outlet plane, often undergoing either compression or diffusion while changing the flow direction and/or path. Typically, the pressure loss between the inlet and the outlet should be as small as possible while the flow homogeneity at the outlet should be high. Often, the available space is limited, meaning that not all possible shapes are feasible.

Parametric model

CAESES® was used to build a fully-parametric CAD model. Utilizing symmetries, the cross-section of the duct was modelled

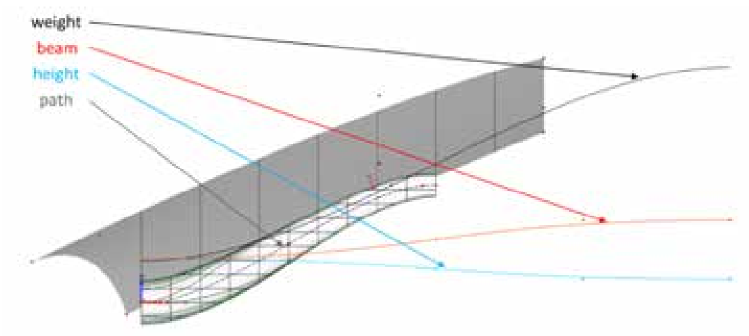


Fig. 1 - Side view of duct model within CAESES® (showing the baseline)

by means of a quadratic NURBS curve with three vertices whose middle vertex received a weight modification, depending on its position between the inlet and outlet plane.

This allowed a continuous transition from a pure circular arc to an almost rectangular cross-section. Furthermore, the path (of each cross-section's centre) as well as the maximum height and the maximum beam at each longitudinal position were described via so-called basic curves, all of which were defined as quadratic B-spline curves with five vertices (see Figure 1). The second and second-to-last vertices of the curves for path, height and beam were allowed to change in x-direction only, ensuring a perpendicular entrance and exit, respectively. The middle vertices were left free to change within the symmetry plane. The CAD model comprised 14 free variables.

An auxiliary surface reduced the available space, shown as a grey surface in Figure 1. An inequality constraint was formulated so that the smallest distance between the duct and the auxiliary surface

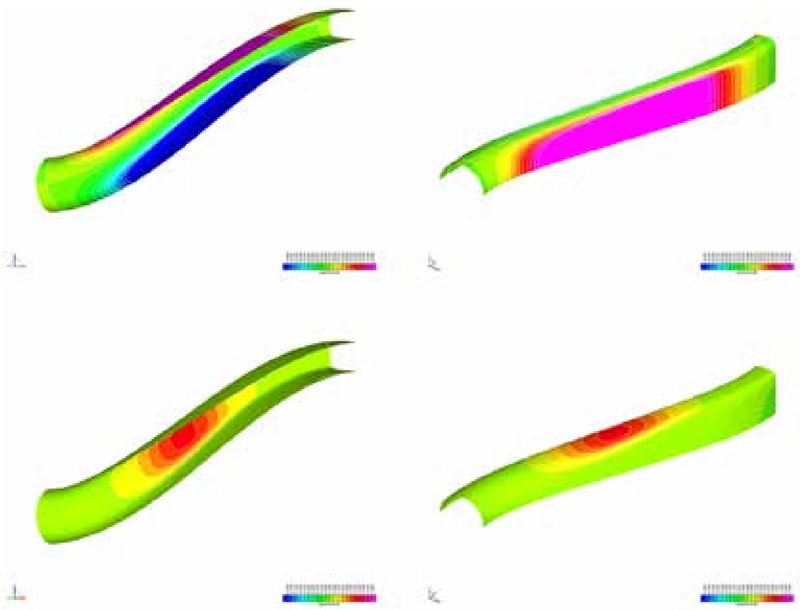


Fig. 2 - Design velocities of selected free variables in the CAD space (for the baseline)

could not go below a given limit. The baseline was set to be a smooth transition from the inlet to the outlet plane, i.e. the shape that could be considered a rather "natural" connection.

The design velocities were studied for the baseline. Design velocities are a means within CAESES® to visually check parametric models and to get an appreciation of where and how strongly each variable influences the shape. To do so, each free variable is modified by a small increment while all the other variables are kept constant. The normal displacement of each point from the original shape is then determined and mapped as a colour code onto the baseline. Figure 2 depicts the design velocities of two of the 14 CAD variables.

CFD simulation

A RANSE simulation was set up with OpenFOAM, using SnappyHexMesh for the grid generation. It should be noted that the aim of this design task was not to actually find the best shape for the HVAC duct, but to showcase the concept and approach of parameter reduction. All results for pressure drop were normalized with the baseline's pressure drop, establishing a non-dimensional objective that would have to be minimized.

KLE modes

As shown in Table 1, the Karhunen-Loève Expansion for the duct yielded a tangible potential for parameter reduction from CAD to the KLE space. The first three super parameters already provided a variability above 95% while the first five super parameters retain more than 98%. Figure 3 and 4 illustrate the influence of the first and the second super parameter, respectively, with shapes stemming from extreme values.

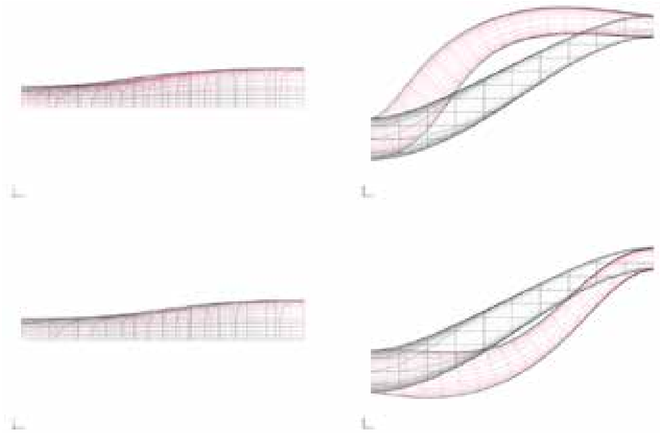


Fig. 3 - KLE mode 1

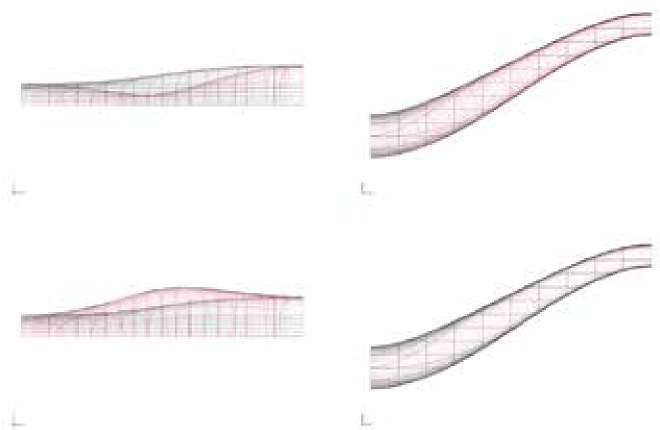


Fig. 4 - KLE mode 2

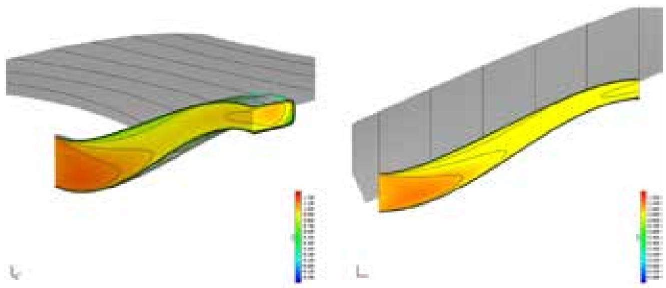


Fig. 5 - Results for the baseline

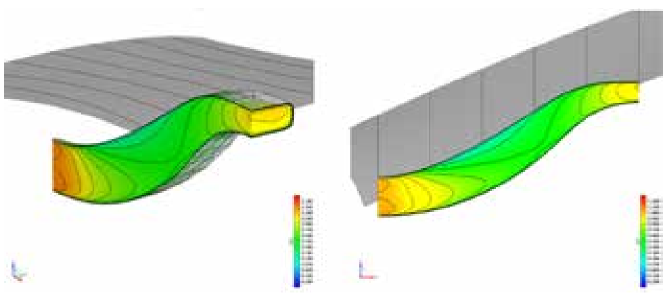


Fig. 6 - Results for variant TSearchFromBestOfNSGA2_08_des0048 (best from CAD)

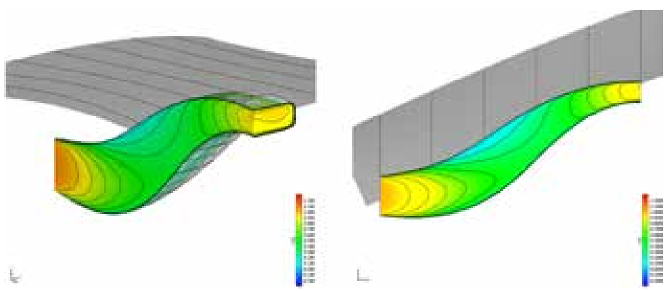


Fig. 7 - Results for variant TSearchKLE_05_des0045 (best from KLE)

Campaign	Method	Step	Number of variants studied	Best designs	Relative pressure drop	Total number of variants needed	Rank
Baseline	Fair transition	1	1	Baseline (for reference)	100.0 %	1	–
Exploration	Sobol	2	200	firstExploration_02_des0180	65.46 %	181	9
Exploitation	TSearch	3	49	TSearchFromBaseline_03_des0048	59.75 %	49	7
Exploitation	TSearch	4	49	TSearchFromBestOfSOBOL_04_des0048	53.94 %	180+49	5
Global search	DAKOTA global opt. w/ response surface	5	73	globalOptResSur_05_des0071	56.21 %	72	6
Exploitation	TSearch	6	49	TSearchFromBestOfGlobalOpt_06_des0030	51.14 %	72+30	4
Global search	NSGA 2	7	512	globalOptNSGA2_07_des0493	49.82 %	494	3
Exploitation	TSearch	8	49	**TSearchFromBestOfNSGA2_08_des0048**	**48.67 %**	**494+49**	2

Table 2: Results from CAD-based optimization with 14 free variables

Campaign	Method	Step	Number of variants studied	Best designs	Relative pressure drop	Total number of variants needed	Rank
Baseline	Fair transition	1	1	Baseline (for reference)	100.0 %	1	–
Exploitation	TSearch with 5 KLE modes	2	49	**TSearchKLE_05_des0045** (out of convex hull)	**41.44 %**	**46**	1
Exploration	Sobol with 5 KLE modes	3	25	SobolKLE_06_des0016	67.83 %	17	11
Correction	Manual	4	1	KLEresults_fromTSearchValuesSnapped (set back to original bounds of CAD variables)	59.81 %	46+1	8
Correction	Manual	5	1	KLEresults_fromSobolSnapped (set back to original bounds of CAD variables)	67.67 %	17+1	10

Table 3: Results from KLE-based optimization with 5 super parameters

Optimization campaigns

With OpenFOAM being connected to CAESES® all optimizations, both in CAD and in the KLE space, were undertaken within CAESES® directly, using the built-in strategies such as a Sobol (for a DoE), TSearch (for deterministic search), NSGA 2 (for global optimization) and, additionally, the external strategies made available via a DAKOTA plug-in (www.sandia.gov). The optimizations within the CAD space were performed according to standard practice.

Several different campaigns were run such as an exploration, various local exploitations and global searches with different strategies (including response surface techniques). The results are summarized in Table 2. The best design found in the CAD space had a drop of 48.6% of the baseline's pressure. A total of 543 variants had to be investigated to reach this since the variant (called TSearchFromBestOfNSGA2_08_des0048 in Table 2) resulted from a combination of a global search with an NSGA 2 and a subsequent deterministic and local exploitation by means of a TSearch. Another slightly less competitive variant with a pressure drop of 51.1% was reached after 102 variants.

Following the traditional campaigns in the CAD space, several optimizations were undertaken in the KLE space, see Table 3. A TSearch was started from the baseline with five super parameters. This yielded the overall best variant with a pressure drop of 41.4% after 46 iterations (called TSearchKLE_05_des0045). Interestingly, an exploration in the KLE space with a Sobol of only 25 variants led to 67.8% for the design labelled SobolKLE_06_des0016. For a comparison, a Sobol in the CAD space with 200 variants yielded a very similar result of 65.5% in pressure drop.

The results for the baseline and the two best designs are depicted in Figures 5 to 7 for comparison. Both new designs look very reasonable, meaning that they show neither any undulations nor strange flow fields. The characteristics of the new designs are somewhat similar with a widening after the entrance and a "swan-neck" exit.

Conclusions

The less free variables you can work with during an optimization campaign the better, in particular when developing products that need a lot of resources to simulate performance. Design spaces typically consist of many dimensions, since complex shapes require a lot of parameters to define them suitably. An approach has been presented with which to substantially reduce design spaces made of CAD variables by mapping them into spaces spanned by different kinds of variables, dubbed super parameters. The approach is known as Karhunen-Loève Expansion. Typical CAD variables, even though chosen with care from a diligently crafted parametric model, still influence certain regions of a shape similarly, giving rise to unwanted dependencies. KLE variables are, by definition, entirely independent from one another, i.e. each of them addresses different shape characteristics. Super parameters, in a way, elegantly combine the original CAD variables. The first super parameter excites primary control over the shape while the second super parameter takes secondary control and so forth. The more super parameters are taken into account, the more the variability of the original shapes is retained. Consequently, the number of super parameters one may pick for an optimization needs to balance the variability (as high as possible) and the dimensionality of the new design space (as low as possible). Naturally, a transformation from the CAD space to the KLE space does not come entirely for free. When running simulations, the associated codes often require the geometry in specific formats, for instance a water-tight fluid domain with a curvature-dependent discretization of its boundaries, which is easiest to derive from the original CAD model. Hence, a back-transformation from the KLE space to the original CAD space is required during an optimization. The benefits, nonetheless, are tangible (and worth the effort): a massive parameter reduction, yielding very substantial savings in the number of variants to be studied which leads, hence, to the faster optimization of shapes.

Reference

[1] E. Bergmann; C. Fütterer; S. Harries; J. Palluch: "Massive parameter reduction for faster fluid-dynamic optimization of shapes", Int. CAE Conference and Exhibition, Vicenza, Italy, October 2018; see http://proceedings2018.caeconference.com/sessions.html#trans

Stefan Harries - Friendship Systems

For more information, email: harries@friendship-systems.com at Friendship Systems AG (www.friendship-systems.com)

Fatigue Analysis using FEMFAT inside ANSYS Workbench

FEMFAT software
BY MAGNA POWERTRAIN

FEMFAT is a finite element (FE) solution for fatigue analysis. It calculates results for time and frequency domains for both high- and low-cycle fatigue. FEMFAT's assessment method considers and modifies guidelines and developments from well-known institutes. It was developed in-house based on project work with global customers as well as measurement and research by the Magna Powertrain Engineering Center Steyr GmbH & Co KG. Its analysis method is a mixture of the local stress concept and the structural stress concept and uses influence parameters to consider results from high-end simulation techniques. Manufacturing processes like sand castings, die castings, metal forming, stamping and complex material models from laminates or short fiber reinforced plastics are accounted for to ensure higher accuracy in fatigue results. Two of its unique capabilities are mean stress correction and plasticity in the frequency domain. Using FEMFAT to conduct a fatigue analysis of components requires structural and stress data to be provided in dedicated formats. These are generated automatically and are basically predefined by the user's FE pre- and post-processors.
Automated workflows enable the use of optimization tools with automatic fatigue analysis.

A FEMFAT extension was developed inside ANSYS as an interface for ANSYS Workbench (WB) to simplify fatigue analysis in terms of data management, integration of optimization software and interpretation of results.

Fatigue analysis of a cyclic loaded component is a very challenging task. Multiple factors -- from the loading to the material description including manufacturing influences and joining technologies -- must be combined and analyzed appropriately to incorporate the main influences on the results of a fatigue life analysis. Using ANSYS WB with FEMFAT is a good solution: on the one hand, ANSYS WB provides a powerful tool for complex analyses such as process simulations, frequency response analysis, heat transfer solutions, etc, while, on the other hand, the FEMFAT fatigue analysis tool takes all these influences and the ANSYS analysis results into account to generate highly sophisticated and precise fatigue life predictions. FEMFAT's open and flexible interfaces allow the integration of many FE-Solvers and preprocessors in every workflow and collaboration project. Indeed, traditionally FEMFAT and ANSYS were used in combination by transferring the data via *.cdb and *.rst files (see Figure 1).

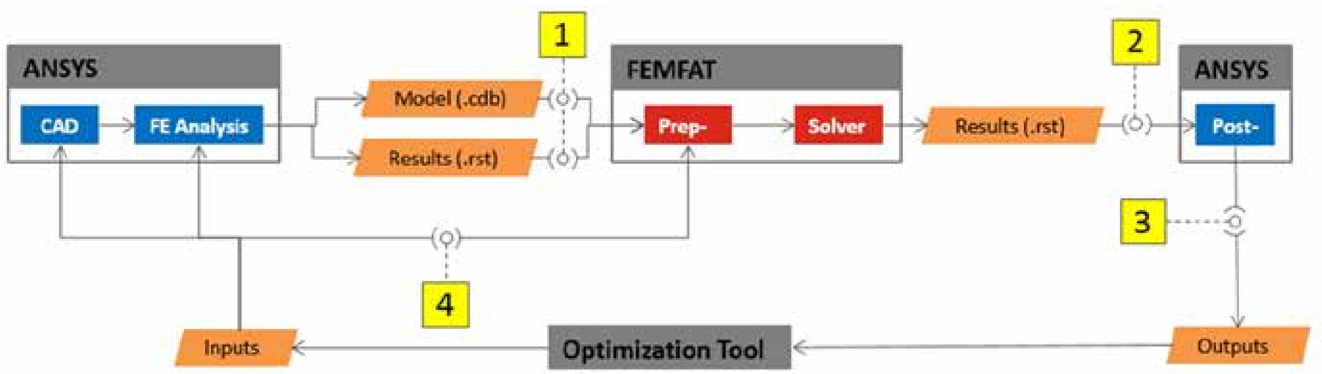


Fig. 1 - The traditional use of FEMFAT with ANSYS

In Figure 1, positions 1 to 4 indicate the traditionally required manual import/export actions. These meant that some of ANSYS WB's main advantages -- the closed workflow with the automatic update of all the design points and the possibility of parameter optimization -- were not available. The objective, therefore, was to efficiently automate both the interactions and the data handling between ANSYS and FEMFAT. Consequently, a software project was set up to conduct the requirements analysis, the implementation and the final approval of the extension.

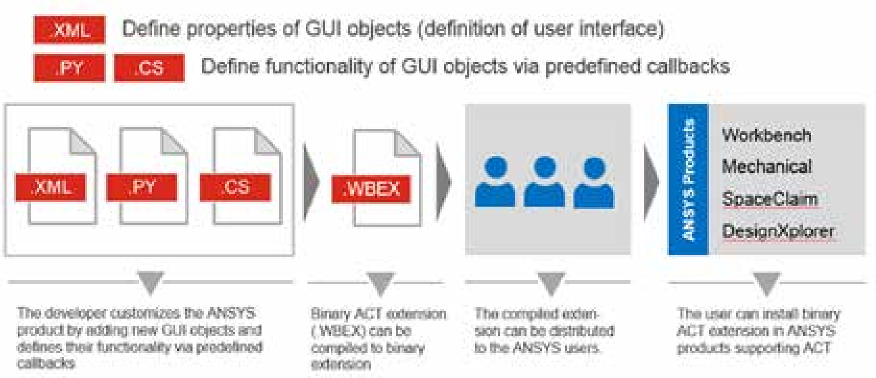


Fig. 2 - Environment description for a user-written ANSYS extension created with ACT

The software project

As a response to the demands of several customers, a software project was initiated to develop a FEMFAT extension for ANSYS WB. This project consisted of the following tasks:

Requirement analysis and specification

The ideation process started with an invitation to key customers that were using the FEMFAT-ANSYS WB combination in the traditional way. Their demands, integrated with our own experience, were noted in a specification book.

The main specifications were:

- The development of an analysis system for ANSYS WB that would utilize parts of or FEMFAT's entire graphical user interface (GUI) inside WB.
- The definition of weld seams in ANSYS WB to enable the use of optimization tools. (A redefinition should not be required as a result of a changing mesh.)
- The ability to map the fatigue analysis results directly in the Mechanical window, as is currently done for stress results.
- The ability to derive and define group and materials definitions directly in ANSYS WB.
- Documentation about the implementation and a user manual

Based on these specifications, several software companies were invited to submit quotes.

Invitation for tenders and contracting

A FEMFAT training session was held for the tendering engineering companies to give them the background knowledge necessary to prepare the requirements specification. Thereafter, these companies submitted their approach which contained the:

- Implementation concept
- Functional specs (user interface, FEMFAT modules, …)
- Delivery content
- Acceptance terms
- Milestones
- Project timeline
- Testing phase
- Help documentation
- Seminar materials

The company DesignTec was selected to realize this project.

Implementation

The implementation was undertaken using ANSYS ACT. ACT is a development environment based on XML, Python and C# which enables the developer to integrate predefined GUI objects into ANSYS products.

This toolkit was used to automate the data exchange between ANSYS and FEMFAT as described in Figure 1. The workflow of the data is described in Figure 3.

The block designated as "FEMFAT Transfer" is necessary to transfer the FE-Model and ANSYS results to FEMFAT. The FEMFAT analysis system generates a FEMFAT-Jobfile (*.ffj) that is dependent on the settings that were made in Mechanical. Subsequently, one has to start FEMFAT in batch mode and then import the FEMFAT results into ANSYS.
This workflow can be established graphically in ANSYS WB for every type of FEMFAT analysis:

A graphical representation of the FEMFAT GUI was introduced in ANSYS Mechanical. When the object <solver> is defined, the head node with the objects Analysis Settings, Solution and Solution Information are automatically added to the Tree Outline (see Figure 5). This makes it possible to specify everything in Mechanical for the majority of analysis types.

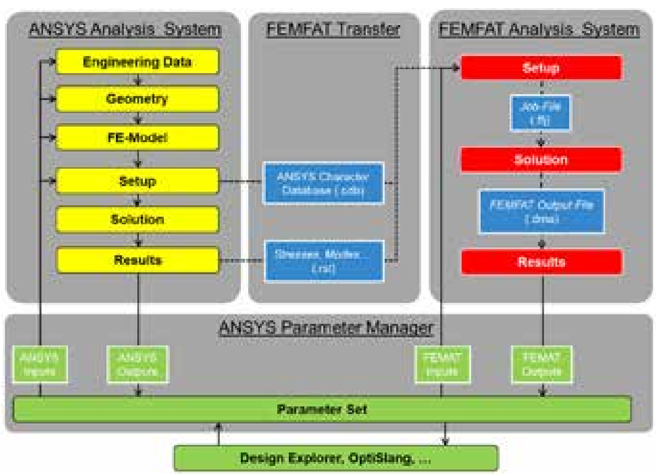


Fig. 3 - Schematic of data exchange between FEMFAT and ANSYS

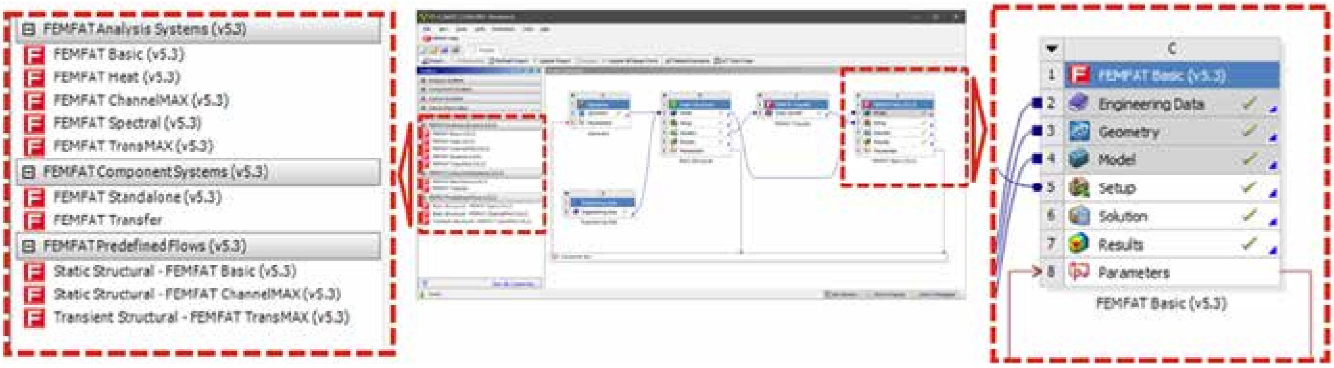


Fig. 4 - Workflow of FEMFAT analysis in ANSYS Workbench

Each object in the analysis setup has detailed view windows where the following control elements can be added:

- text boxes and entry fields,
- drop-down lists,
- tables,
- ....

With regard to post-processing, the FEMFAT analysis results can be displayed as a contour plot while the content of important ASCII files (Protocol File, Message File and Job File) can also be inspected.

Over and above this ACT development, further software developments were necessary to realize the following parts of the implementation:

- Model extraction from the ANSYS results file
  A new DLL library to extract the model data (.cdb) from the ANSYS results file (.rst) was written. This .cdb file can be directly imported to FEMFAT.
- Weld seam definition
  A new approach to weld definition based on the selection of geometrical entities was implemented. This means that redefinition as a result of a changing mesh is no longer necessary.
- Licensing
  The ECS licensing interface was integrated into the FEMFAT extension to enable all the licensing options to be offered by the FEMFAT software.

The development was completed within one year.

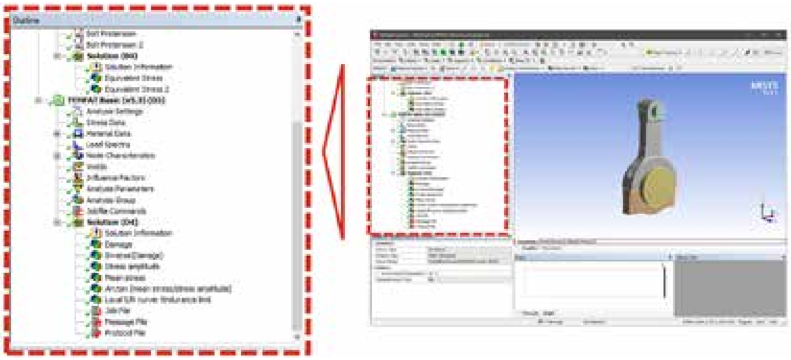


Fig. 5 - Graphical integration in Mechanical

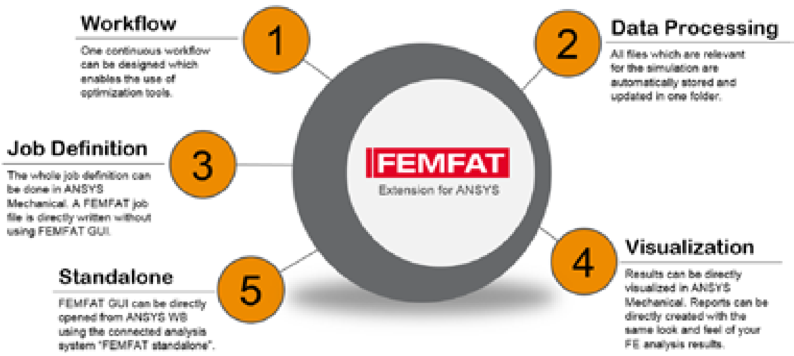


Fig. 6 - Advantages of the FEMFAT integration into ANSYS WB

The FEMFAT extension inside ANSYS Workbench

The implementation of FEMFAT inside ANSYS WB provides five advantages to the user:

1. Workflow:

One continuous workflow can be designed which enables the use of optimization tools. All FEMFAT Analysis types (Basic, Max, Spectral, Heat) are supported and can be combined.

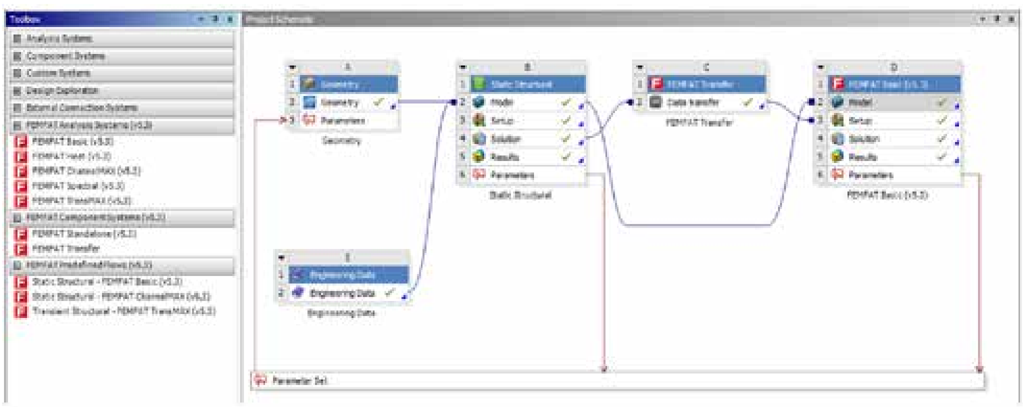


Fig. 7 - The Workflow of a FEMFAT analysis in ANSYS WB

2. Data processing:

All files which are relevant to the simulation are automatically stored and updated in one folder. As a result, when the WB project is moved or copied, all the necessary files are moved/copied, too. The file paths in the FEMFAT job file are related within the project tree to guarantee a valid analysis.

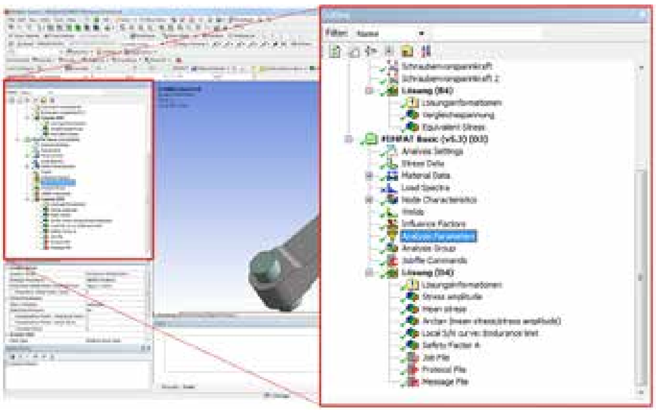


Fig. 8 - FEMFAT settings in ANSYS Mechanical

3. FEMFAT job definition:
The whole FEMFAT setup and job definition can be done in ANSYS Mechanical. A FEMFAT job file is directly written without needing to use the FEMFAT GUI.

4. Visualization:
Results can be directly visualized in ANSYS Mechanical. Reports can be directly created with the same look-and-feel of the FE analysis results. Visualization of the standard FEMFAT results is provided by default. Additional results can be generated using the Jobfile Commands menu.

5. Advanced use with Standalone and Jobfile Commands:
A Standalone FEMFAT system can be connected to the workflow so that the job can be opened from inside ANSYS WB using the FEMFAT GUI. While this system (cp. Figure 9, system E) is not available for continuous and automated workflows, it helps to cross-check the system's setup from the previous definition step (cp. Figure 9, system D).

The Jobfile Commands menu enables the use of specific FEMFAT commands once the extension has created the job file. This can be used to realize any special FEMFAT analysis and output.

Conclusions
The FEMFAT Extension enhances the current capabilities of ANSYS and enables fatigue assessment to be integrated into a continuous product development process
- The continuous workflow can be used for design point analysis and optimization
- FEMFAT jobs can be fully set-up inside ANSYS Mechanical
- Material properties and Analysis Groups can be directly defined in the Mechanical window
- The fatigue results are evaluated in ANSYS Mechanical
- A FEMFAT Standalone System can be linked to the Analysis System
- The Jobfile Commands ensure that the full capability of FEMFAT's options can be used in ANSYS
- The project demonstrates the benefits of software integration
- ACT made it possible to completely integrate FEMFAT into ANSYS WB within a reasonable time

Klaus Hofwimmer
Magna – Engineering Center Steyr GmbH and Co KG (www.femfat.com) in St. Valentin, Austria.
He can be reached at klaus.hofwimmer@magna.com

Dr Petr Vymlátil
DesignTec s.r.o. (www.designtec.info) in Brno in the Czech Republic.

For more information:
Dario D'Agostino - EnginSoft
d.dagostino@enginsoft.com

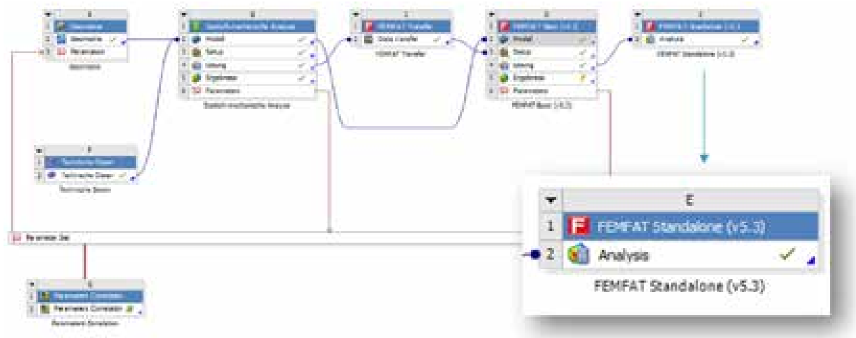


Fig. 9 - The FEMFAT standalone call in ANSYS WB

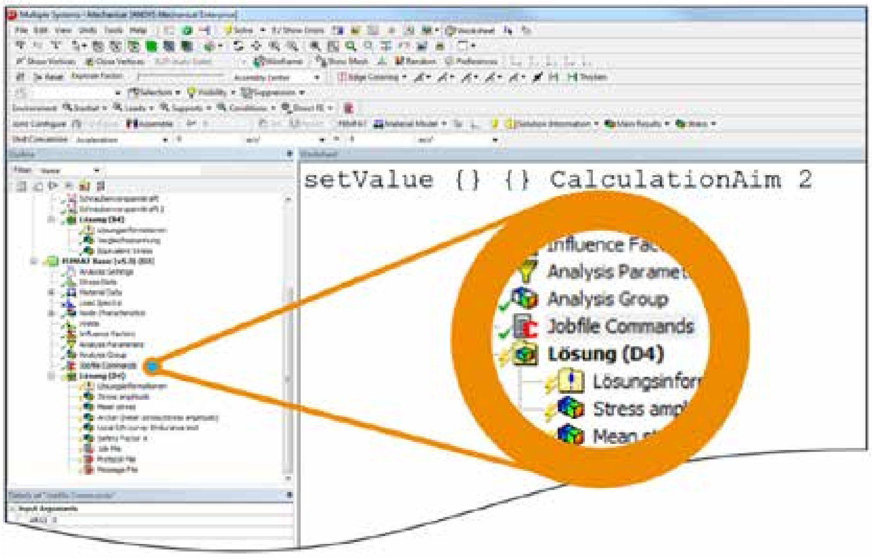


Fig. 10 - Advanced FEMFAT analysis using the Jobfile Commands option

Honda enhances pedestrian protection with modeFRONTIER

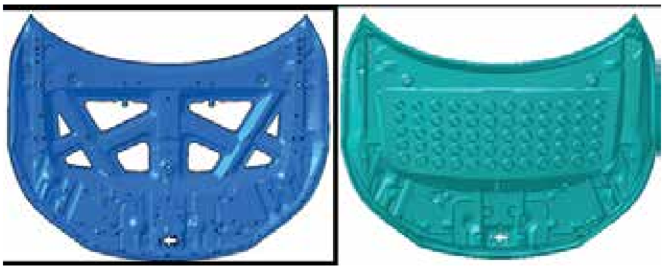
Using modeFRONTIER to minimize crash deformation of an aluminum hood

modeFRONTIER

Honda Automobile R&D Center strives to fulfil their social responsibilities as an automaker with respect to environmental conservation, safety and quality assurance.
Among these challenges, Honda employed modeFRONTIER software solution to find the optimal vehicle aluminum hood configuration in order to reduce pedestrian head injuries caused by car collisions.

CHALLENGE

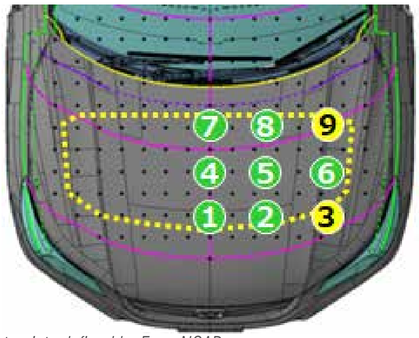
Japanese traffic accident statistics show that more than a thousand of fatalities occur every year mainly due to head injuries. The European New Car Assessment Program (Euro NCAP) is widely used to evaluate pedestrian head protection with impacts against vehicles. In addition, car manufacturers are required to reduce vehicle weight to meet CO2 emissions standard. As a result, they have increased the use of aluminum hood which guarantees 40% of weight reduction compared with steel. However, this normally demands a longer crash deformation for pedestrian protection because the energy absorption characteristics is lower than steel (low inertia and stiffness).
Accordingly, aluminum requires increased clearances under the hood together with further restrictions in terms of layout structure. Combining pedestrian protection and weight reduction became a key challenge in the car industry. Engineers at Honda, focused on building an aluminum hood capable of reducing crash deformation and achieving five-star Euro NCAP for head protection.



Embossed structure with truncated cone is chosen as conceptual structure.

SOLUTION

Starting from a conventional aluminum hood with many large holes, the panel has been filled and impressed with truncated cones to increase mass and stiffness. An optimization process was created in modeFRONTIER workflow to perfect the inner embossed aluminum hood for 9 head impact points defined by Euro-NCAP. modeFRONTIER allowed to refine 15 design parameters (mainly related to mass and stiffness) to minimize the impact deformation, and automate the interaction between different simulation solvers. CATIA was used to modify the shape, while ANSA solver generated the mesh for head impact simulation performed by LS-DYNA solver. The results were then processed in LS-PrePost to evaluate Head Injury Criterion (HIC) and deformation.

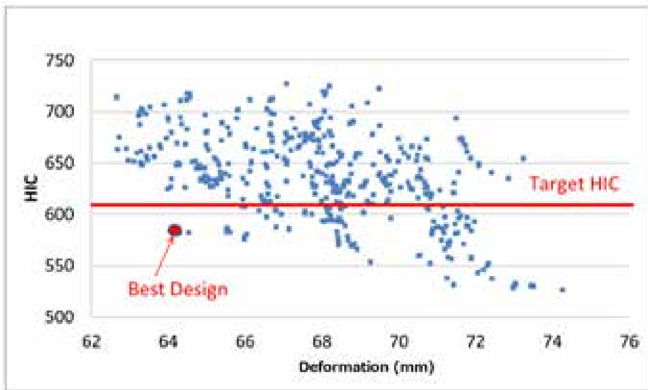


Nine head impact points defined by Euro-NCAP.

modeFRONTIER ADVANTAGES

“modeFRONTIER enabled us to save computational time when optimizing design variables for each head impact point. Design of Experiments (DOE) analysis led to identify the impact point (No. 6) which did not meet the HIC requirements. The Multi-Objective Simulated Annealing (MOSA) algorithm was used to optimize the worst impact point. This allowed to find the best designs after few evaluations. The overall optimization process allowed to reduce 6% of the crash deformation compared to the conventional aluminum hood and satisfy HIC target values” said Osamu Ito, Assistant Chief Engineer, Technology Research Division, Honda R&D Co. Ltd.

For more information:
Francesco Franchini - EnginSoft
f.franchini@enginsoft.com



Pareto Front (MOSA Algorithm).

Numerical simulations for road tunnel fire safety

UNIVERSITÀ DEL SALENTO

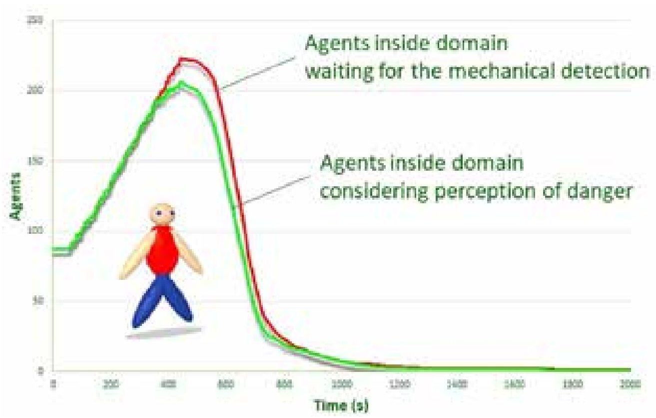
In 2012, the results of a four-year monitoring period of Italian road infrastructure found accidents occurring in tunnels to be the most severe in terms of victims and damage. There are 1,400 km of tunnels in Italy, constituting about 28% of the total tunnel distance in Europe. In addition, due to the specific aspects of their construction, the evacuation and rescue of victims from potential fire events is much more difficult. It was these considerations that laid the basis for a study focused on tunnel fire safety, in particular, for the Condó road tunnel in Lecce, Italy.

The analysis of the Condó road tunnel, which was carried out using Fire Dynamic Simulator (FDS) + Evac, evaluated smoke propagation during a potential tunnel fire and its impact on the success of an evacuation procedure. Comparisons were carried out between mechanical and human detection times, and between prescriptive-based and performance-based approaches.

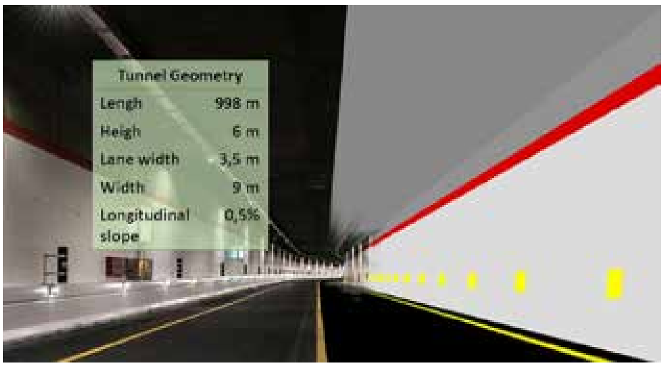
The first step in the analysis was the creation of a numerical model of the tunnel that included its geometry and the fire protection system. Particular attention was paid to the relationship between the mesh grid and the size and position of the FDS objects to improve both the accuracy of the solution and the computational time load.

In the second phase, the design fire curve of the hypothetical fire was defined by coupling the theoretical relationships and the experimental data: the fire was placed near the only emergency exit, where no smoke detectors are installed.

The fire simulation analysis found that the fire protection system takes more than six minutes to detect the presence of smoke and to activate the jet fans after a fire ignites. Meanwhile, the evacuation simulations showed that this mechanical detection time takes longer than a human-perception-of-danger time; this could result in an uncontrolled evacuation which is often chaotic and dangerous, particularly if it is aggravated by possible obstructions in the tunnel caused by abandoned vehicles. Moreover, one must consider that, while the detection time elapses, the tunnel occupants would be exposed to many toxic substances that could compromise their ability to escape. This analysis highlighted how the prescriptive approach that is used to design the tunnel underestimates important parameters which may affect evacuation dynamics in the case of a fire. The use of numerical simulations and digital models can also be considered to be game changer for maintenance procedures. In fact, their application could facilitate the evaluation of the impact of any maintenance activity or unplanned event on the fire safety of tunnels. Furthermore, numerical simulations and digital models provide the opportunity to couple and compare various aspects of the lifecycle of a tunnel, in order to build safer, more innovative and durable road infrastructures.



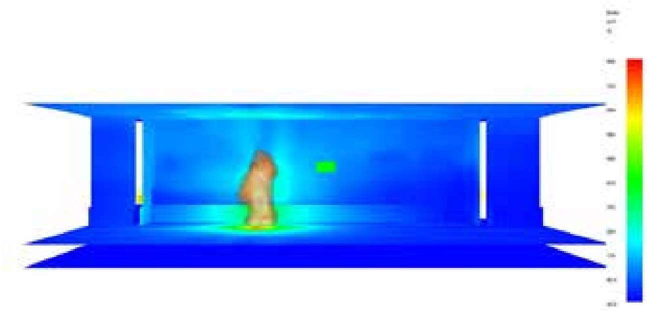
Evacuation simulation results comparing mechanical/human responses to the presence of smoke



Comparison between the real and the digital Condó road tunnel

Sandro Gori - EnginSoft
Mariarita De Rinaldis; Ada Malagnino – Università del Salento

For more information:
Sandro Gori - EnginSoft
s.gori@enginsoft.com



8 October 2018 CAE INTERNATIONAL POSTER AWARD

Wind turbine research activity at the University of Padova

Research activities

COMETES (CFD and Optimization METhods for Engineering and Science [1]) is a research group of the Industrial Engineering Department of the University of Padua. Our research involves the study of turbomachinery (transonic compressors, hydro and gas turbines, axial pumps), marine applications (waterjets, boat hulls) and wind turbines. One of our most innovative approaches involves the application of evolutive algorithms to a machine's design process. We use both commercial computer-assisted engineering (CAE) software and in-house developed codes. Our group has several years of experience in the study of wind turbine design. This article briefly presents our most important activities in this field.

The Blade Element Momentum (BEM) method

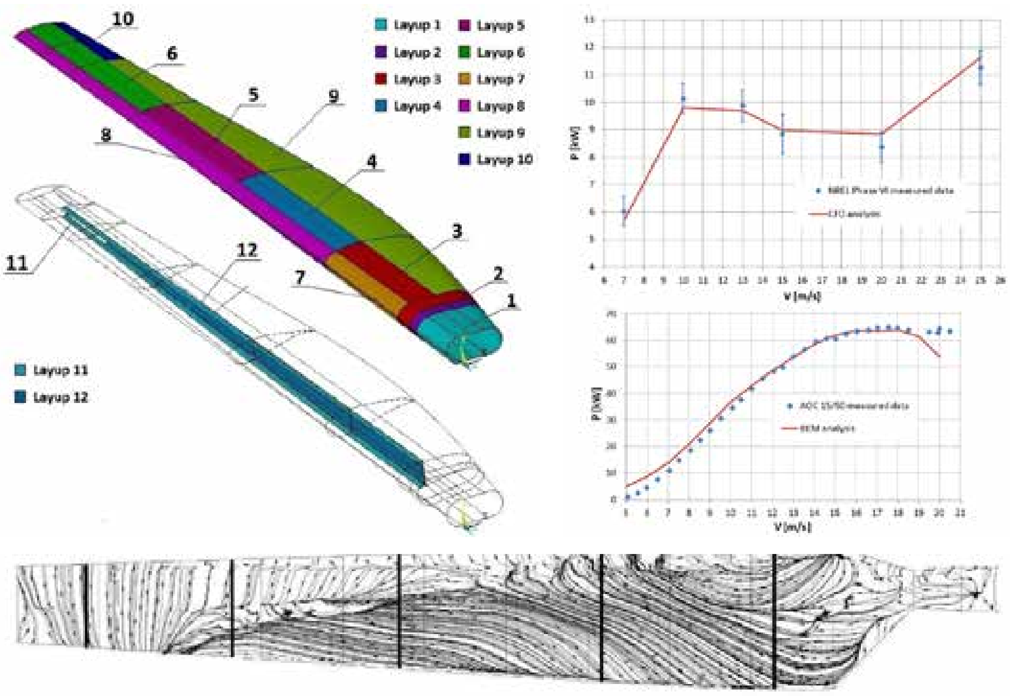
The Blade Element Momentum method allows the fast and accurate estimation of the aerodynamic performance of wind turbines. The adopted algorithm [2] is based on the well-known theory presented by Martin O.L. Hansen and has a negligible computational effort compared to a 3D Computational Fluid Dynamics (CFD) analysis. It combines two independent approaches: Momentum Theory and Blade Element Theory. The former concerns the computation of both thrust and torque by applying the conservation of the linear momentum and of the angular one to a control volume; the latter refers to the analysis of aerodynamic forces acting on each blade section, as a function of blade geometry. The method consists of dividing the rotor into a finite number of control volumes and then determining the forces acting on each element for different rotational speeds. In order to increase its accuracy, the algorithm is coupled with a panel code (Xfoil, RFOIL). The integration of the radial forces along the blades determines the net thrust of the wind turbine.

Multi-objective optimization

The optimization of the design of a wind turbine is not easy due to the presence of several optimal solutions, depending on the type of investigation. For this reason, it is usually carried out using automatic methods such as evolutive multi-objective genetic algorithms [2], [3]. Airfoil shapes are mathematically reconstructed using Bezier curve or B-Spline formulations. Then several performance objectives can be imposed to investigate the structural, aerodynamic and economic aspects. The estimation of performance is carried out through low- or high-fidelity methods, depending on the desired accuracy. The result is a Pareto Front where all the optimal solutions lie on a curve and the designer can choose the best solution based on the relative importance of the objectives. However, different applications are suited to different optimization methods; other techniques include the gradient-based methods, surrogate models and neural network algorithms.

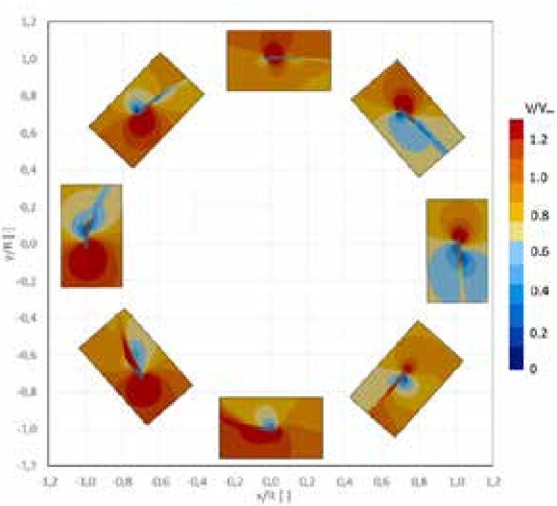
The Validation of the CFD and BEM models

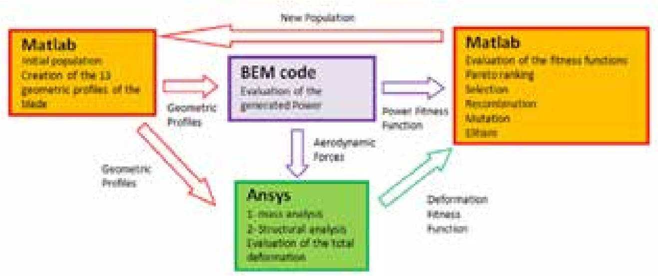
The validation of the developed algorithms was performed using different experimental data available in literature. The in-house 1D BEM, implemented in SOCRATE (Structural Optimization for Composite Rotor Air TurbinE), and a 3D steady Reynolds-Averaged Navier-Stokes (RANS) CFD model in ANSYS Fluent were validated against the power curve of the SANDIA AOC 15/50 wind turbine. The numerical power curve reproduced the experimental performance of the wind turbine at the design wind speed and in stall conditions well. Furthermore, the FEM model results were compared to the structural tests of the blade: both the experimental flap-wise and edge-wise rigidities match the analysis. Also, the NREL Phase VI wind turbine was adopted as a test case for the setup of a 3D unsteady RANS CFD model using a moving mesh strategy in ANSYS Fluent. The results obtained in terms of power curve and stall development along the blade radius are in good accordance with the measurements for all the tested wind speeds.



Optimization of Vertical Axis Wind Turbines (VAWT): an open-source toolbox

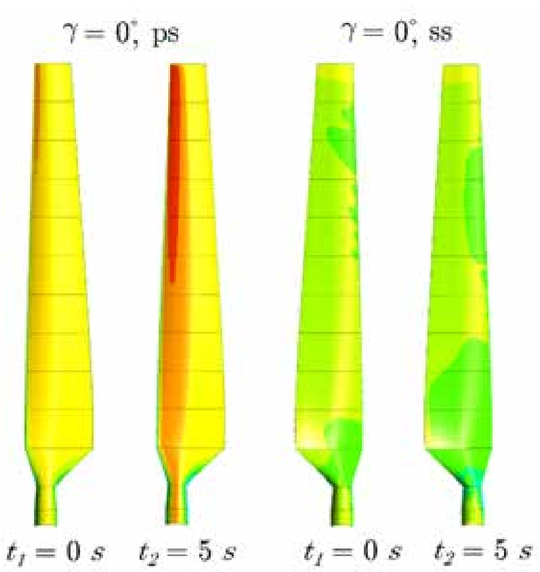
A further research field is represented by VAWT design. A fully-automated open-source environment for VAWT optimization was developed and validated [4]. Several modules are present. First of all, the optimization is led by Dakota, a toolbox from SANDIA National Laboratories that allows one to run both mono- and multi-objective optimizations. The design variables are used to generate a Python journal file for the geometry and mesh generation in SALOME, using the OpenCASCADE libraries. Then the unsteady RANS CFD is set up in OpenFOAM with a moving mesh technique and a multi-step problem approach. The results of the OpenFOAM analysis are automatically acquired and post-processed in the Paraview environment developed by Kitware. The results of the whole process were tested in cooperation with the University of Delft, using experimental measurements and Particle Image Velocimetry (PIV) data.

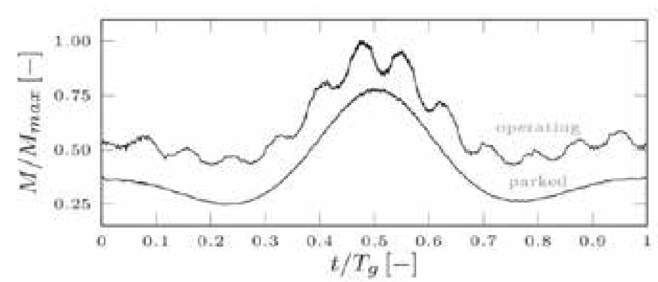




The SOCRATE algorithm

The Structural Optimization for Composite Rotor Air TurbinE (SOCRATE) algorithm is a multi-disciplinary and multi-objective optimization tool, developed to estimate the performance of Horizontal Axis Wind Turbines (HAWT) [2]. Different parameterization techniques are available for the reproduction of the airfoil sequence of the considered blade, and the effects of the selected layout of the composite blade structure in terms of materials, fiber orientations, thickness and position of reinforcements can be investigated. The coupling of different engineering modules also allows one to carry out an aero-structural analysis: the BEM code is used for aerodynamic performance, whereas the ANSYS Mechanical APDL (ANSYS Parametric Design Language) module runs the structural evaluation. The optimization module uses multi-objective genetic algorithms and focuses on both structural (maximum tip deformation, flap-wise and edge-wise rigidity), aerodynamic (Annual Energy Production) and economic (Cost of Energy) objectives.





A numerical study that aimed to analyze the effect of an extreme loading event on a small HAWT was conducted [5]. A validated 3D unsteady CFD model of the NREL Phase VI turbine was adopted, considering the Extreme Operating Gust (EOG) profile from IEC 61400-2 as the inlet condition. A wind gust is a short and strong peak in wind velocity. The aerodynamic response and the structural ultimate check, based on the IEC guidelines, were outlined for both the operating and the parked turbine, in order to underline the benefit of the safety position in terms of lowering the stress transferred to the critical root section.

References:

[1] https://www.linkedin.com/company/cometes-unipd/

[2] A. Dal Monte, S. De Betta, M. R. Castelli, E. Benini, Proposal for a coupled aerodynamic–structural wind turbine blade optimization, Composite Structures, 159, 144-156, 2017

[3] A. Dal Monte, M. R. Castelli, E. Benini, Multi-objective structural optimization of a HAWT composite blade, Composite Structures, 106, 362-373, 2016

[4] A. Dal Monte, Development of an open source environment for the aero-structural optimization of wind turbines, Ph.D. Thesis, 2017

[5] L. Menegozzo, A. Dal Monte, E. Benini, A. Benato, Small wind turbines: a numerical study for aerodynamic performance assessment under gust conditions. Renewable Energy, 121, 123 - 132, 2018

Andrea Dal Monte, Luca Menegozzo and Ernesto Benini*
University of Padova

CAE INTERNATIONAL POSTER AWARD

A healthcare digital twin using integrated imaging and finite element model

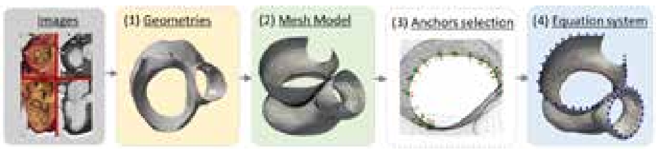
Developing a pre-planning phase to improve the performance of a Cardioband procedure for the treatment of mitral regurgitation

Mitral Valve (MV) regurgitation is a disease caused by a dilatation of the MV annulus ring. In recent years, new transcatheter techniques have been proposed to correct MV regurgitation. The Cardioband® system (Edwards Lifesciences, Irvine, USA) uses a specific polyester sleeve fastened with helicoidal metal anchors to the posterior MV annulus to contract it by pulling a sewn metal wire, to repair MV regurgitation disease. We developed an in silico tool to reproduce the Cardioband procedure by integrating image processing and Finite Element (FE) simulation. The aim of this study was to evaluate the accuracy of the tool and to analyze how different positioning of the implanted anchors influences the procedure's performance.

Finite element routine

The steps of the routine were:

(1) generation of the geometries of patient heart structures from clinical images;
(2) meshing of the geometries;
(3) selection of the anchor positions on the mesh model;
(4) simulation of the procedure.

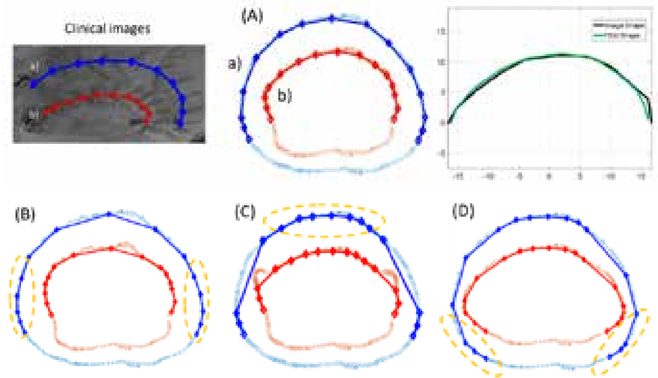


The routine solves the equation system composed by the Cardioband, FE and the compatibility equations governing the Cardioband device and the heart structure's behavior, and their interactions during the cinching phase.

Procedure simulations

Four different configurations of the anchor positions were analyzed: an equispaced (A) configuration according to the clinical procedure, and three non-equispaced configurations with anchors in the trigonal (B), the posterior (C) and the anterior (D) regions.

These configurations were compared with the clinical procedure post-cinching outcomes of the selected patient.



Results

Post-cinching comparison between anchor position (b) of the standard clinical procedure and the results of the simulation with configuration (A). It is possible to note a good overlapping of the curves.
Comparison of the final MV annulus of the different configurations simulated with the same wire-length reduction. Configuration (D) shows a healthy physiological shape.

Parameters		Pre-cinch	A	B	C	D
eccentricity		0.51	+17%	-1%	+26%	+34%
Area (mm^2)		$2.7x10^3$	-50%	-52%	-50%	-42%
Anchor Reaction Forces (N)	max	-	16	+5%	-19%	-36%
	mean	-	6	+10%	-28%	-32%
	min	-	1.6	+30%	-10%	-4%
σeq_{max} (MPa)		-	3.4	3.3	3.3	3.3

Configuration (D) showed the highest eccentricity value, and lower area reduction and anchor forces values. The stress/strain values were the same for all configurations.

Discussion

Our tool points out the feasibility of providing additional information about the annulus shape, the fields of stress/strain and the anchors forces, which is useful during a pre-planning procedure. It is possible to define the anchor positions to obtain the desired final shape of the annulus and the reduction of the MV regurgitation.

Emanuele Gasparotti, Emanuele Vignali, Katia Capellini, Benigno Marco Fanni, Alfredo Giuseppe Cerillo, Sergio Berti, Vincenzo Positano, Simona Celi
Fondazione Toscana Gabriele Monasterio, Massa

Luigi Landini
Università di Pisa

Using simulation to improve understanding about how NCT functions to facilitate its output correlation to pathologies

The non-contact tonometry (NCT) test is a non-invasive clinical test to measure intraocular pressure (IOP) and more. Nowadays, however, it is gaining importance as a tool to characterize ex-vivo corneal biomechanics.

Briefly, the eye is hit by a high-speed air-jet, which provokes the deformation of the cornea. A lot of biomarkers can be derived from the corneal profile, such as corneal hysteresis (CH) and maximum apex displacement (DA). The challenge is to correlate these outputs to some pathologies. Since the test is very complex and involves geometrical aspects, biomechanical characterization and the dynamic response of the materials, simulating it could be very useful in improving understanding about how it works. For this reason, our study aimed to simulate a general NCT using a 3D model of the eye and a fluid-structure interaction (FSI) between the cornea and a jet of air.

Material and method

The structural part was designed according to the dimensions of an average human eye (Fig.1), while the fluid domain was built to have a fully developed flow with the intention of reproducing the real boundary conditions. The mesh was built in ANSA (Beta CAE) using hexa-block for the eye (Fig.2), and triangular shell mesh for the fluid domain. The material models were linear-elastic for the lens and muscle, and hyper-elastic for the sclera, cornea and limbus. Thanks to a user-defined material sub-routine, two families of fibers were added to the cornea and one was added to the limbus to also replicate the physiological anisotropy. The vitreous and aqueoushumors were described as incompressible Newtonian fluids.

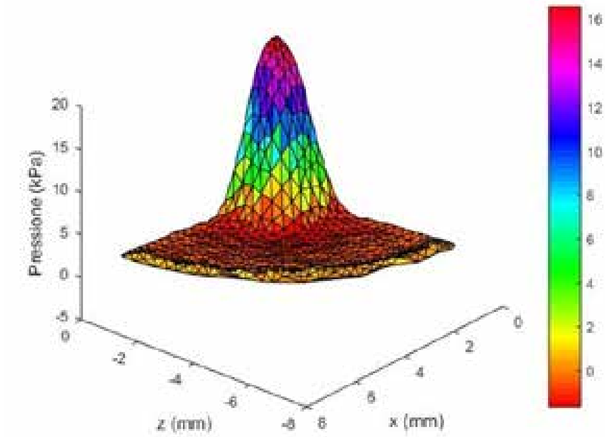


Fig. 3 - Pressure profile on corneal surface

Results

The maximum apex displacement in the cases with and without vitreous and aqueous humors differed by 38% but the DA fitted the experimental range only for the case with the vitreous and aqueous humors. The pressure inside the aqueous humor reached 22 mmHg during the running of the test, in accordance with other studies.

Discussion

The results show the importance of considering the non-linear anisotropic material of the cornea, and of incorporating the aqueous humors; indeed, neglecting these aspects in the simulation led to an overestimation of the corneal properties. Moreover, the FSI was necessary to capture the negative pressure on the corneal surface (Fig. 3) and to calibrate the mechanical material properties.

Benedetta Piva, Josè Felix Rodrigueza Matas, Giulia Luraghi e Miguel Ariza-Gracia - Politecnico di Milano

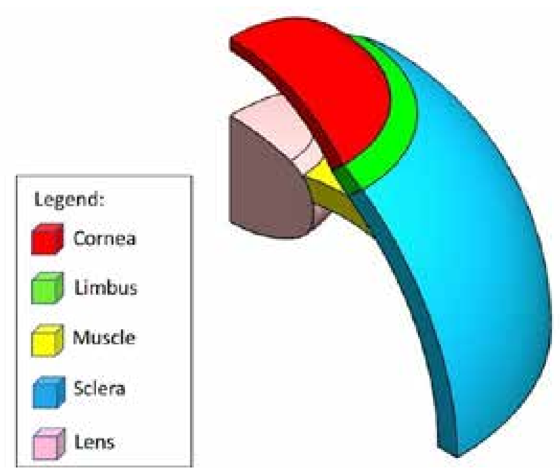


Fig. 1 - 3D model of the eye

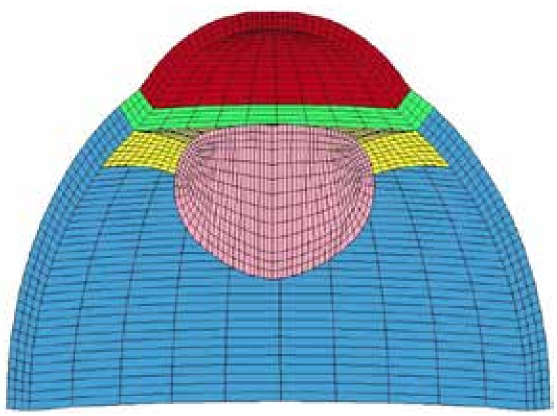


Fig. 2 - Structural mesh

Comprehensive modeling of lightweight materials in crashworthiness simulation enables more reliable virtual development

Designing a new platform or a new model in vehicle development resembles the task of “squaring the circle”. On the one hand, lightweight design is required to reduce energy consumption. On the other hand, strict requirements for occupant and pedestrian safety must be met. Different materials are available for each component: steels – in sheet form– already feature a wide range of stiffness and ductility; lightweight alloys of aluminum and magnesium are available as sheets, extrusions and castings; non-reinforced plastics as well as short- and long-fiber reinforced plastics are also available. For plastic components that require a high stiffness, thermoplastics reinforced by glass fiber fabrics – so-called organic sheets – may be used. For special applications, carbon fiber-reinforced thermoplastics are an option. The costs of materials and manufacturing are other aspects of material selection. Moreover, the choice of optimum material demands the development of hybrid material joints.

Finite Element (FE) simulation has been used for many years to virtually investigate an appropriate combination for a construction and its corresponding material selection. Right up to today, comparably simple elastoplastic material and failure models are being used in industrial practice for dimensioning. In many cases, however, the particular properties of the material are not described appropriately, and the possible impact of the production process is also not sufficiently considered.

Figure 1 shows the schematic of a simulation-based development of a component for a crash load case (here: impact of a pole into a bumper). An isotropic model for plasticity and fracture, which has been calibrated based on tensile tests only, will yield results that deviate from the physical tests with regard to force-intrusion and time-of-failure. These deviations will become apparent in the first hardware test of a prototype: fracture is found although simulation did not provide this prediction.
The parameters of the simple material model may be adapted such that the hardware test is met better. However, this iteration will not supply physically meaningful parameters, because the material model does not capture the specific properties of the material correctly. The identified parameters are only valid for the particular load case. Moreover, this iterative process is time-consuming and expensive.

The use of a more complex material model with parameters based on a comprehensive testing program of wrought material, will give a reasonable result from the first simulation, if a suitable discretization is applied in the FE model. As a consequence, time-consuming and expensive iterations at a hardware level can be omitted in most cases. The extra effort in material characterization and in preparing the material card is justified when compared to the extra costs of modifying the component geometry at a later stage of the development process.

MATFEM started developing the modular material model, MF GenYld+CrachFEM, 15 years ago. The module MF GenYld (generalized yield function) describes the elasto-visco-plastic behavior of materials. CrachFEM summarizes the different phenomena for material failure. The model has been enhanced in line with innovations in the field of materials in vehicle development.
Furthermore, the material model has been coupled to the established commercial explicit-dynamic FE solvers that are used for forming and crashworthiness simulation. This allows coordinated development with regard to material behavior between the manufacturer of semi-finished products, the manufacturer of components and the OEM, even if different FE codes are used.

This article first provides an overview of the available modules to describe metallic materials and non-reinforced and fiber-reinforced plastics in the material model MF GenYld+CrachFEM. Then the main part of the article presents example applications of materials in detail.

Overview of MF GenYld+CrachFEM

The first phase of MF GenYld+CrachFEM’s development focused on the CrachFEM module, which deals with the prediction of failure in metallic materials (mainly those used for the body-in-white). Sheets, thin-walled extrusions and thin-walled cast components are involved. In general, metallic materials fail in ductile normal fracture and ductile shear fracture (see Figure 2a and b). Brittle fracture only appears under extreme conditions (eg. very low temperatures). However, in most cases of the material failure in thin structures, a localized necking happens prior to fracture (see Figure 2c). This localized necking is an instability in the tension-tension regime and not directly a material

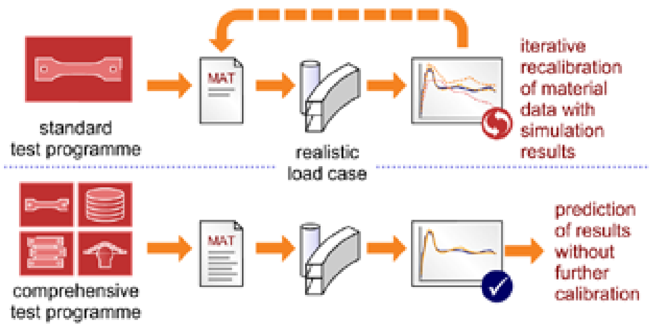


Fig. 1 - CAE development process for crashworthiness based on a simple material model (top) and based on a comprehensive material model (bottom)

failure. With a shell discretization – widely used in crashworthiness simulation – the localized necking cannot be predicted directly.

The usual criterion for sheet failure used in industrial sheet-metal-forming simulations is the classical forming-limit diagram. This criterion is based on the onset of localized necking. However, its validity is limited to linear strain paths. Therefore, it is not applicable to complex forming processes and the process chain of forming and crash. MATFEM developed the Crach algorithm to predict localized necking in linear strain paths (i.e. the forming-limit diagram), and in general non-linear strain paths. Crach is a numerical model of a quasi-3D necking problem.

The connected material model does account for isotropic-kinematic hardening, plastic orthotropy and strain-rate sensitivity. The deformation history of a shell element is transferred to the Crach algorithm for a given interval of plastic equivalent strain. A non-convergence of the algorithm indicates necking. For a given strain path prior to necking, Crach offers the necking risk as a history variable of the shell element.

For Advanced High Strength Steel (AHSS), Ultra High Strength Steel (UHSS) and aluminum sheets, fracture can also occur without prior necking, eg. in bending or during in-plane shear. This kind of sheet failure is not covered by the forming-limit diagram. The criteria for ductile fractures are necessary: they define the possible equivalent plastic strain at fracture as a function of a single stress invariant (in the case of a shell discretization with a plane stress state), or two independent stress invariants (in the case of a solid discretization with a general 3D stress state). CrachFEM uses stress-state parameters which use two stress invariants in all cases. Therefore, it is easy to swap between shell and solid discretization with MF GenYld+CrachFEM material cards.

For multi-step processes with changes in stress state, an appropriate damage accumulation law has to be used. Most available material models use a scalar description of material damage together with an integral damage accumulation. However, this does not correlate with real material behavior. After any plastic deformation of an initially non-damaged material, the damage can only be described by a tensor. The fracture risk for the stress state of the first deformation step is reduced. The ductility for other directions is not affected or may even increase. Therefore, fracture prediction with CrachFEM is based on an integral damage accumulation with a tensorial description of damage.

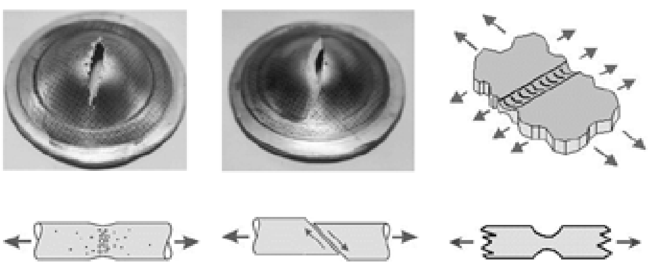


Fig. 2 - Mechanisms of ductile normal fracture, ductile shear fracture and sheet instability (localized necking)

For resume runs in multi-step forming simulations, and when mapping between forming and crash simulations with different meshes, the full membrane deformation history of an element must be transferred from one step to the next to ensure the correct prediction of necking. The deformation history must be expressed relative to a reference orientation. Therefore, the orientation of the rolling direction has to be mapped, too. For the fracture models, the cumulated values of the damage tensor have to be transferred from one process to the next.

After establishing the CrachFEM module for metallic materials, the variability of the MF GenYld module was extended. Today a wide choice of yield loci (a yield locus is a multiaxial criterion for the onset of material plasticity) can be combined with isotropic, isotropic-kinematic and general anisotropic hardening. This combination is fully modular. A correct evaluation of the strain distribution is a prerequisite for the correct prediction of material failure.

Further development steps were undertaken for non-reinforced polymers, short/long-fiber-reinforced polymers, and endless fiber-reinforced polymers (both unidirectional and fabrics). The main features of polymers in MF GenYld are visco-elastic properties and the extension to compressible plasticity (the assumption of incompressibility is the basis for all metallic materials). In CrachFEM, an orthotropic model for ductile fracture was implemented which can be used both for a general 3D orthotropy and for the transversal-isotropic case. Besides extended models for ductile fracture, models for brittle fracture of endless glass or carbon fibers were also implemented. A more detailed description will be given in the chapters on short-fiber-reinforced polymers and endless glass fiber fabrics in a thermoplastic matrix (organic sheets).

Modeling of hot-formed and quenched manganese-boron steels sheets

The criteria for passenger safety have been tightened steadily in the last few years. This has led to the wide use of quenched manganese-boron steels for passenger cab design, especially for the A-pillar and the B-pillar. Meanwhile, components with tailored tempering are also used to adapt the material properties according to local needs.

The introduction of these manganese-boron steels has introduced some new challenges for numerical simulation. On the process side, hot-forming simulation had to be established at car manufactures and their suppliers.

The use of tailored tempering also makes it necessary to predict the local hardness within the process simulation. In the field of crashworthiness simulation, it is not possible to conduct a predictive simulation of quenched manganese-boron steel components without a comprehensive assessment of material failure. On the one hand, the quenching process provides an extremely high yield strength which reduces the penetration into the passenger cab. On the other hand, the quenched steel has a low strain hardening exponent, which increases the risk of localized necking, and a reduced ductility.

Further challenges for crashworthiness simulation arise from the spot-welding of quenched manganese-boron steels. The heat-affected zone around the spot-weld has a reduced yield strength and can act as the fracture initiation point in a crash event.

The experimental characterization of a quenched manganese-boron steel is already a challenge. The classical tensile test exhibits a very low uniform elongation (only 3-4%). A meaningful true stress-true strain curve cannot be derived from this input by extrapolation to higher strains. Stack compression tests (eg. a stack with 5 or 7 circular specimens of the relevant sheet) loaded normally to the sheet plane allow one to reach true strains of 0.3-0.4. A common strain hardening curve can be derived with data from tensile tests and stack compression tests.

A further problem is the evaluation of a forming limit-diagram. Quenched manganese-boron steels with a UTS of 1300-1900 MPa cannot be tested with the classical Nakajima experiment. The specimen clamping in the die, and specimen deformation with a hemispherical punch are not possible practically and would damage the included tools. The instability module in Crach-FEM allows the forming limit curve to be predicted based on the strain hardening behavior. In addition, localized necking can be predicted in the case of non-linear strain paths.

There are also the risks of ductile normal fracture and ductile shear fracture due to the reduced ductility of quenched manganese-boron steels. The fracture limit curves were evaluated with seven different specimens, where fracture is induced by different stress states. Figure 3 shows the prediction of a ductile normal fracture during 3-point bending of a B-pillar (manganese-boron steel with a UTS of 1900 MPa).

The method of tailored tempering (patented by ThyssenKrupp Steel Europe) was introduced to achieve better adaption of the mechanical properties in different regions of the B-pillar. Tailored tempering is done by using a hot forming tool with different temperatures along its length. A high yield strength is preferable in the upper part of the B-pillar to avoid critical intrusions. Here fast quenching is used to induce a purely martensitic microstructure. The lower part of the B-pillar should allow high energy absorption with lower yield strength and higher ductility. This can be achieved by partially heating areas of the tool. The lower cooling rate (below 550°C) induces a ferritic-bainitic microstructure.

MF GenYld+CrachFEM allows the creation of a single material card where all the mechanical properties are a function of the hardness value. A comprehensive experimental characterization of the fully quenched material and of the partially quenched material is needed as a minimum. The material cards allow for an automatic interpolation of properties (yield strength, strain hardening, forming limit curve, fracture limit curves) as a function of the hardness (see example of a hardness distribution in Figure 4). Selected tests for the intermediate conditions of the quenched steel can help to improve the interpolation quality. No user interaction is necessary to define the different material properties for the different parts of the B-pillar. The hardness distribution can either come from a hot forming and quenching simulation or be derived from a microhardness measurement in the lab (only possible with a first hardware prototype). A hot forming and quenching simulation allows the smaller gradients of hardness in the fully quenched region or the partially quenched region to be represented.

Modeling of short-fiber-reinforced polymers

Stringent requirements for passenger and pedestrian safety in the automobile sector combined with the need to decrease the total weight of the structural parts has created a demand for new approaches. Short-Fiber-Reinforced Thermoplastics (SFRT) are increasingly being used to fulfill this conflicting role. Their stronger mechanical properties compared to unreinforced polymers, their flexible geometry and their short cycle-times, enable them to meet these demands.

However, their mechanical properties strongly depend on the principal orientation (eg. local filling direction) and the degree of orientation of the fibers, which result directly from the mold-injection process (see Figure 5). Dependent on the component's geometry and the parameters of the mold-injection process (such as cylinder temperature, tool temperature and holding pressure), both measures can vary strongly within a single part and this needs to be accounted for in a structural simulation.

In an integrative simulation, this information can be extracted from the orientation tensor based on a mold-injection simulation and mapped onto a comparatively coarse shell mesh for the structural FE simulation. Besides mapping the fiber orientation, the degree of

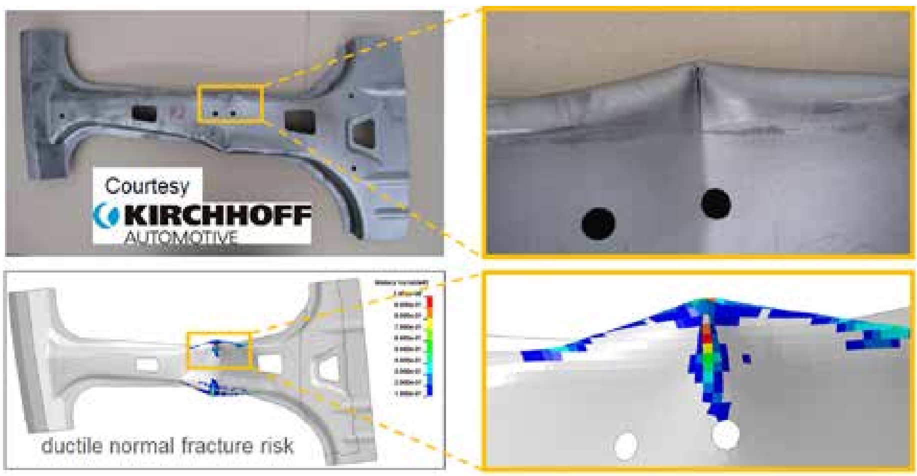


Fig. 3 - Prediction of a ductile normal fracture for a B-pillar with generic geometry in a 3-point bending rig – quenched manganese-boron steel with a UTS of 1900 MPa

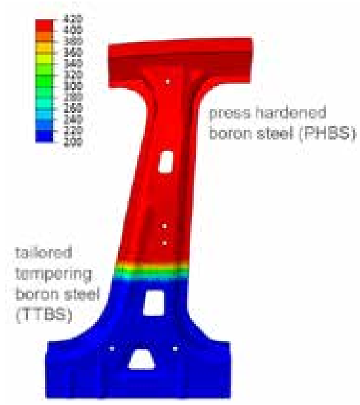


Fig. 4 - Possible microhardness distribution (Vickers hardness) in a generic manganese-boron steel B-pillar after hot forming and either full quenching (upper part) or partial quenching (lower part)

anisotropy has to be initialized at the beginning of a simulation. It is introduced as a measure to distinguish if the fibers are aligned perfectly in a unidirectional manner (highly anisotropic), or randomly distributed (quasi-isotropic).

Based on MFGenYld+CrachFEM, MATFEM has developed a method to simulate SFRTs to fully depict their complex elasto-visco-plastic material behavior including failure. This method is composed of two separate steps:

- Comprehensive material characterization for a highly oriented plate material (represented in Figure 6 right as the blue icons for a 0° and a 90° case). The material properties for arbitrary orientation can either be derived by experiments with quasi-isotropic plates, or theoretically by averaging the properties of the highly oriented state from 0° and 90°.

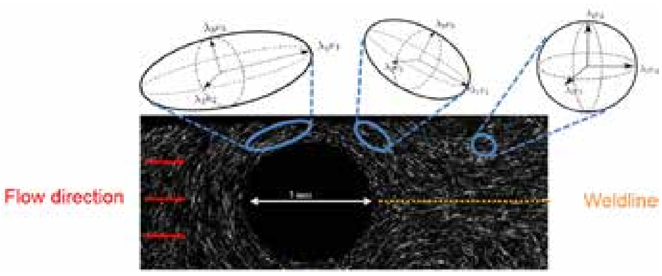


Fig. 5 - CT-Scan of a mold-injected plate with fiber orientation around an obstacle

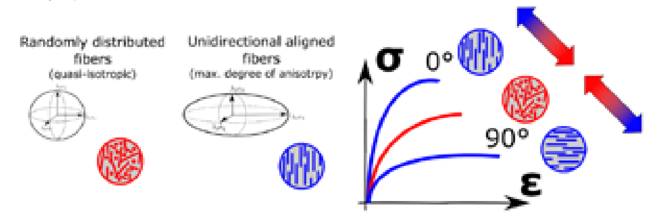


Fig. 6 - Orientation tensor for quasi-isotropic material and strongly orthotropic material (left); derivation of a flow stress curve for a given degree of anisotropy, by interpolation

- The use of a mapping tool MF Inject (a software tool under development) to map the local fiber orientation and the degree of anisotropy (see Figure 7); other commercial mapping tools can also be used.

The material card for an oriented short-fiber-reinforced polymer includes an orthotropic visco-elastic behavior, the orthotropic visco-plastic hardening behavior (with an asymmetry in tension and compression), and an orthotropic stress-state-dependent fracture. At each integration point, the properties are interpolated automatically by the material model based on the mapped principal direction of the fibers and the degree of anisotropy.

In an industrial crash simulation, the CAE engineer can now use the comprehensive material card and fully depend on the available information. In an early design phase when very little information about the mold-injection process is available, the isotropic material card with arbitrary orientation of the fibers is a first approach. With the mapping of the fiber distribution, the FE simulation using the interpolating material card can depict all relevant material properties virtually. For crashworthiness load cases as well as for pedestrian safety, it is crucial to correctly assess the initial location of the fracture

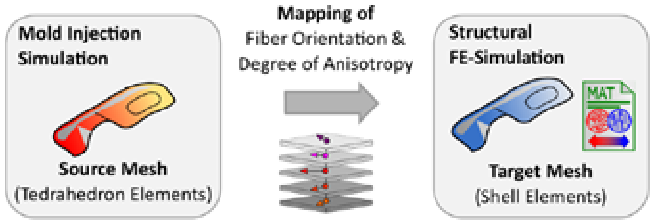


Fig. 7 - Mapping of the fiber orientation and degree of anisotropy from a mold-filling simulation on the shell model of a crashworthiness simulation

and also the overall energy absorption capability of a component or structure. The interpolating material card allows this to be done virtually in CAE system, allowing the number of needed prototypes to be reduced essentially.

Modeling of organic sheets

Endless fiber-reinforced plastics (EFRPs) have a high potential for weight reduction in the automotive area. However, these materials have very complex mechanical behavior in crash loading due to their use of comparatively soft and ductile matrix material (thermoplastic or thermoset material) combined with comparatively stiff and brittle endless fibers (glass or carbon fibers). Depending on the loading in relation to the fiber orientation, the behavior can be very stiff and brittle with no plasticity, or soft with ductile behavior and significant plasticity. Organic sheets are an example of EFRPs and are semi-finished products that may consist of multiple layers. They are composed of brittle endless glass-fibers with high strength knitted to a textile fabric (Figure 8) and embedded in a ductile, thermoplastic matrix.

Compared to SFRTs, the degree of anisotropy of organic sheets in terms of elastic, plastic and failure behavior can be even more pronounced. A comparatively small amount of plastic deformation before fracture can be seen in fiber orientation. In off-axis orientations, a highly ductile behavior with low hardening can be observed as shown in Figure 9, right.
This characteristic behavior can be stress-state dependent (eg. different for tension and compression). Also, the strain rate influence can have different trends depending on the loading direction. Figure 9, left, shows the linear-elastic behavior of the glass-fiber bundles. The fracture strength increases with increasing strain rate. In off-axis directions, the material response is dominated by the matrix material (see Figure 9, center). The thermoplast shows a viscoelastic-viscoplastic response. The fracture strain decreases with increasing strain rate.

The combination of different phenomenological modules in the material model MF GenYld+CrachFEM enables one to depict these

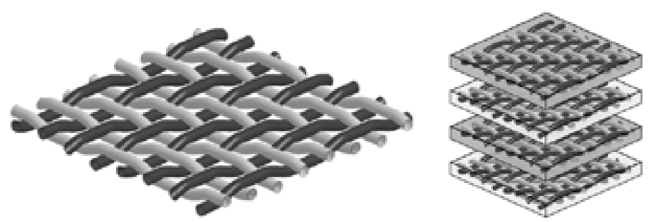


Fig. 8 - Plain woven textile structure with equally balanced fibers in warp (black) and weft (grey) direction

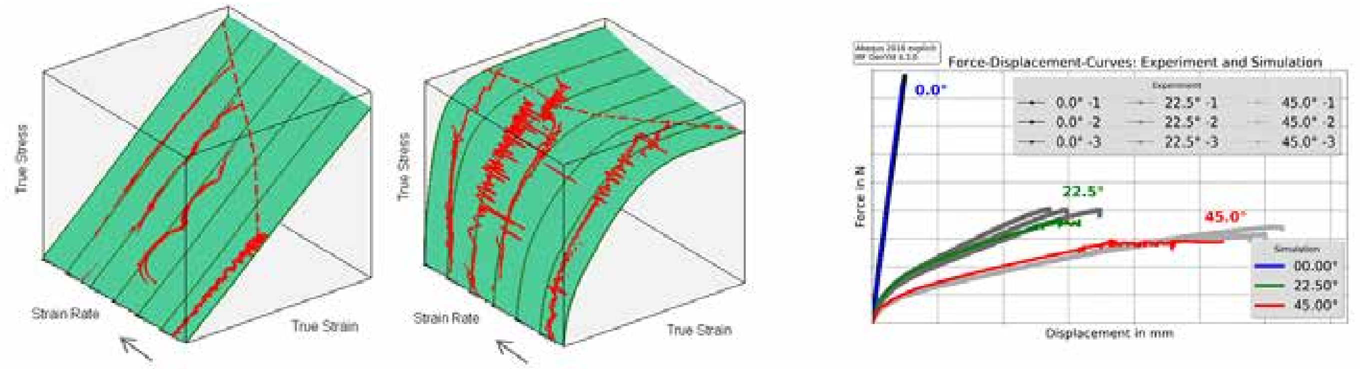


Fig. 9 - Influence of strain rate on viscoelastic/viscoplastic response and fracture limits of an organic sheet: response of the material in main fiber direction (left) and under 45° to the fiber direction (middle); in-plane orthotropy from quasi-static tensile tests (right). Source: Material tests performed by Fundacion CIDAUT, Valladolid, Spain

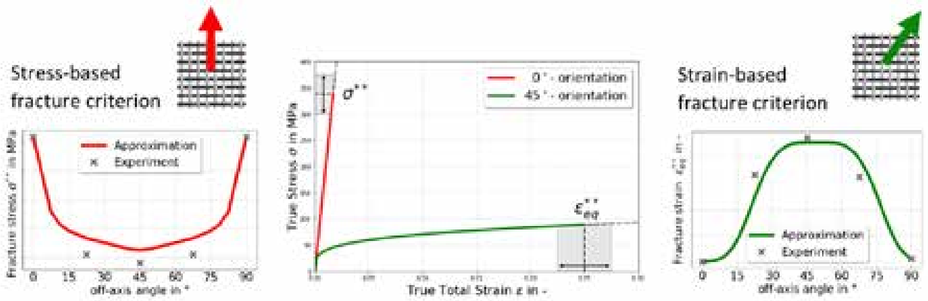


Fig. 10 - Combination of strain-based and stress-based failure criteria in MF-GenYld+CrachFEM for an organic sheet

mechanisms on a macroscopic scale, allowing engineers to keep the computational costs for a component-level or for a full-vehicle crash level within a reasonable range. For the material description, an anisotropic yield surface is used which describes the boundary of the elastic and plastic regime. After trespassing the yield surface stress-state, dependent plastic hardening of the material is assumed as a smeared description to account for the effective damage accumulation on a macroscopic scale until fracture. For the prediction of fracture two separate criteria run in parallel.

The combination of a strain- and a stress-based failure criterion allows the correct depiction of the failure mode of the fiber- and matrix-dominated orientations (0°- and 45°-orientation in Figure 10). Hence, in dependence of the orientation, the more sensitive measure will be used for the assessment of failure.

The new method has been validated on a specimen level as shown in Figure 9 and on a component level as shown in Figure 11. Overall there is a good correlation between simulation and experiment for the quasi-static and dynamic case of the 3-point bending of a generic hat profile.

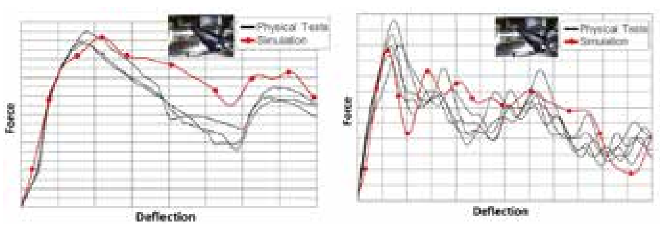


Fig. 11 - Crash simulation results vs. test results for 3-point bending of a thermo-formed hat profile; quasi-static experiments (left) and dynamic experiments (right). Source: M. Franzen, G. Oberhofer, R. Schwarzer, Improved crash simulation of continuous-fibre-reinforced thermoplastics - organic sheets; Kunststoffe im Automobilbau; Mannheim, 2017

Summary and outlook

New topics for the further development of MF GenYld+CrachFEM in the near future include fiber fabrics in thermoplastic, thermoset matrices, and additive-manufactured (AM) metallic materials.

In organic sheets, the initial alignment of fiber bundles towards each other typically shows a 90°-angle and can be described with good accuracy by using an orthotropic approach. In general, this initial structure will be partially modified during thermoforming to a more generally anisotropic structure where the alignment of fibers towards each other deviates from the 90°-orientation. Therefore, the initially orthotropic approach has been extended to a more generally anisotropic approach using a superposition of two or more transversely isotropic approaches. Based on this method, it is possible to very effectively model structures which are initially non-orthotropic. The validation of this approach is ongoing.

Another challenge is the modeling of metallic components produced by laser sintering (LS). The LS process is part of the group of additive-manufacturing methods. A possible plastic orthotropy and a fracture orthotropy in the build direction relative to the "printing" direction have to be described. The yield loci for a general 3D stress state and CrachFEM's orthotropic fracture model for a 3D stress state are a good basis for this topic.

Helmut Gese, Gernot Oberhofer, Felix Brenner, Michael Richter
MATFEM

Helmut Gese, Gernot Oberhofer, Felix Brenner and Michael Richter work for MATFEM Partner-schaft Dr. Gese & Oberhofer, in Munich, Germany.

For more information, contact:
Helmut Gese, at Matfem, Nederlinger Str. 1, 80638 Munich, Germany, or email: helmut.gese@matfem.de

MATFEM Partnerschaft Dr. Gese & Oberhofer Maschinenbauingenieure

Large system analysis: the abstraction of 3D FE models to beams and shells and more

AIES
Advanced Integrated Engineering Solutions Ltd

PART 2

This is the second part of a three-part series of papers. The first part published in the last issue of the EnginSoft Newsletter described Advanced Integrated Engineering Solutions' system analysis framework. This article, Part 2, describes our abstraction techniques to aid large system analyses. The third part, which will be published in the subsequent issue of the EnginSoft Newsletter, introduces the company's tribology solutions (or joints) for machine design, engines, gearboxes, turbomachines and motors etc.

We have developed abstraction techniques to help bridge the gap between 3D Computer-Aided Design (CAD) and Computer-Assisted Engineering (CAE) and the1D, 2D and 3D beam system analysis models and methods used by many system level software vendors. I have seen the problems first hand because, in reality, System and Control Engineers carry out so-called high level (low fidelity and low accuracy) analyses for example vehicles and they rarely collaborate or validate their assumptions and modelling methods with the real-world models used by CAD and CAE engineers. So, there is often a break in the link between system analysis and the CAD and CAE engineering groups. The System and Control Engineers conduct simulations at the vehicle concept stages with their 1D and 2D models and management makes big decisions about layout and configurations based on this work. Wouldn't it be better if the models derived for the low-fidelity simulations were abstracted from 3D CAD (solid) and CAE (Finite Element Analysis (FEA) or Computational Fluid Dynamics (CFD)) models, so they had provenance? This use of the low fidelity model has its drawbacks as it can be very risky and can lead to disastrous consequences, as I have witnessed.

We can now give the System and Control Engineers the 1D, 2D and 3D beam models with more accurate definition and provenance (based on science rather than conjecture). These models are derived from the 3D CAD (solid) and CAE (FEA and CFD) models of the vehicle and system analysis methods that CAD and CAE engineers will use downstream. The CAD and CAE Engineers can then use the vehicle and system analysis performance information, loads, cycles and environmental data in their 3D component and system models to obtain more accurate boundary conditions and durability assessments. In order to mend this break, we start from the 3D solid Fixed Element (FE) model and abstract an exact beam with reduced Degrees-of-Freedom (DOFs) for use in system analysis and MultiBody Dynamics (MBD). We have also developed techniques and methods for reducing structures and oil films to a reduced stiffness and damping representation. These will be discussed and explained in the third part of the series.

Abstraction methods developed by AIES Ltd

Firstly, we wanted to abstract a 3D beam model that closely represents the accuracy of the 3D solid from which it is derived, and that is more accurate than the 3D beam elements currently available to the CAE Analyst and Design Engineer. In addition, we wanted to improve the solution time by a factor of 100 and obtain an accuracy of better than 5% for beams with L/D ratios of 0 to10. The method does this and more. We can also make our beams smarter by including stress concentration (Kt) features at drillings and holes, etc. for more realistic stress results than those provided by standard beam theory.

We have also developed abstraction methods for 3D solids to shells which are quite promising, although we found a few issues when using them for conrod analysis that includes the assembly of the shells in the conrod and the preloading of the big end bolts. This work revealed a limitation of the shells with edge loads, and we had to find a work-around. We may well develop a new shell element for this purpose in the future.

Comparison of 3D solid sections – 3D solid to 3D beams

Below you can see the sections that were used to carry out our comparative work: 50mm x 50mm, 25mm x 50mm, 50mm x 25mm, 50mm diameter solid, and a 50mm diameter tube. This

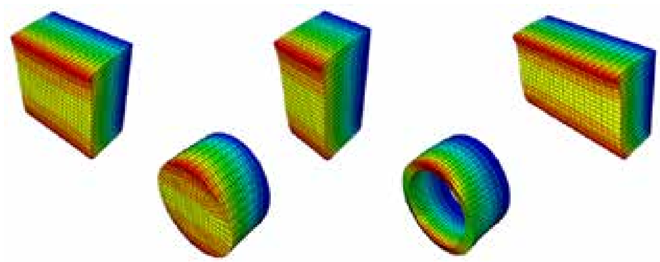


Fig. 1 - The sections investigated at L/D values of 0.25 – 10. These were then compared to the results of Timoshenko and Cowper.

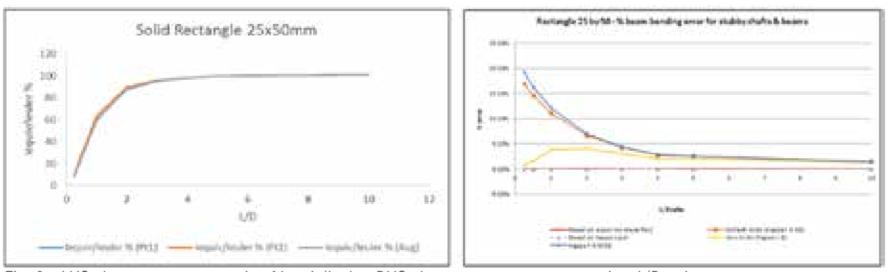


Fig. 2 - LHS shows percentage ratio of Iequiv/Ieuler. RHS shows percentage error against L/D ratio.

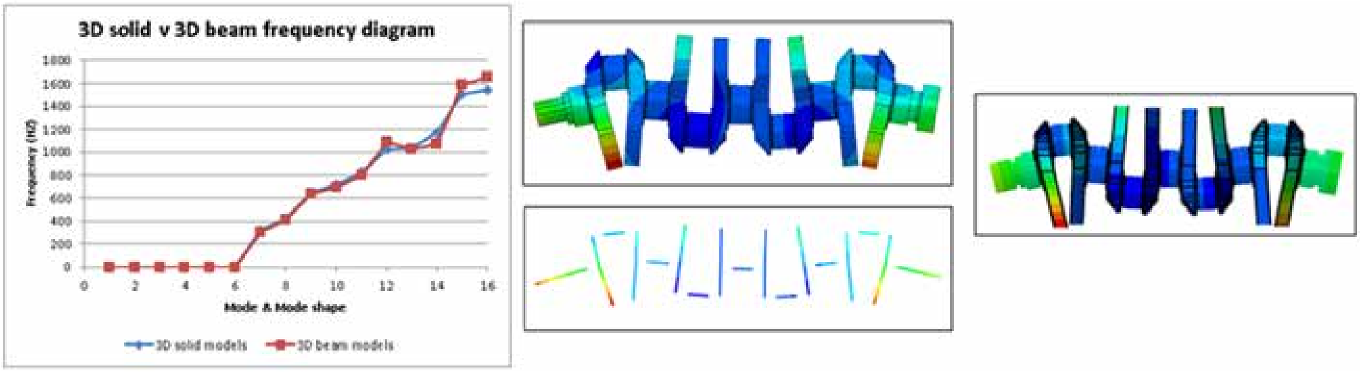


Fig. 3 - Crankshaft models – 3D, beam and shell element mode shapes and critical speeds – shells shown rendered

work was compared to the work of Timoshenko and Cowper. Figure 1 shows the hexahedra meshes generated in SystemDeveloper and the shear deformation that took place in the section. This shear deformation becomes more pronounced as the section becomes more and more stubby.

There were a lot of results from this investigation, so this paper only shows the results for the 25mm x 50mm section.

The left hand side (LHS) of Figure 2 shows the percentage variation of I (second moment of area) value compared to the Euler Bernouli beam theory. The shear coefficient graphic, if illustrated, would be quite erratic but basically provides the same results shown in the right-hand side (RHS) picture. Both our new I equivalent, and shear coefficient results, give zero error for L/D ratios of 0.25 – 10. The LHS of Figure 2 shows that as L/D tends to 10, our method is the same as Euler Bernouli theory for long beams. You can also see that Timoshenko and Cowper beams have an error of only about 2% at the point where L/D is 10. When comparing stubby beams, the RHS of Figure 2 shows that Timoshenko and Cowper's work tend towards an error of 20%, as do those of common commercial FE codes. So, in summary, our abstraction methods are far more accurate than Timoshenko and Cowper beams or those used in common commercial FE codes.

Our beam abstraction methods versus 3D FE models

This work shows our abstraction methods compared to the 3D FE models of a four-cylinder in-line crankshaft. We are looking at free-free modes of vibration. These are the sorts of results that can be validated against model testing with the crankshaft suspended on bungie cords.

But real engine crankshafts are connected to the engine via oil film bearings, which is why a natural frequency map and Campbell diagrams often need to be constructed to assess real systems.

Figure 3 below shows the comparison of the modes for a solid 3D FE versus our 3D beam abstraction (LHS). The first 6 modes are zero as shown. The middle image shows a comparison between a 3D solid and 3D beams for a crankshaft's first bending mode (mode 7). The RHS shows the results for a more accurate method of web abstraction. Bending, torsion and axial modes are considered by our methods here.

Our shell abstraction methods versus 3D solids – Conrod example

Figure 4 below shows the models used in this study. The LHS shows the 3D model. The middle image shows a shell flat model. The RHS shows the shell model with eyes (at big end and small ends). Again, modal free-free analysis was carried out.

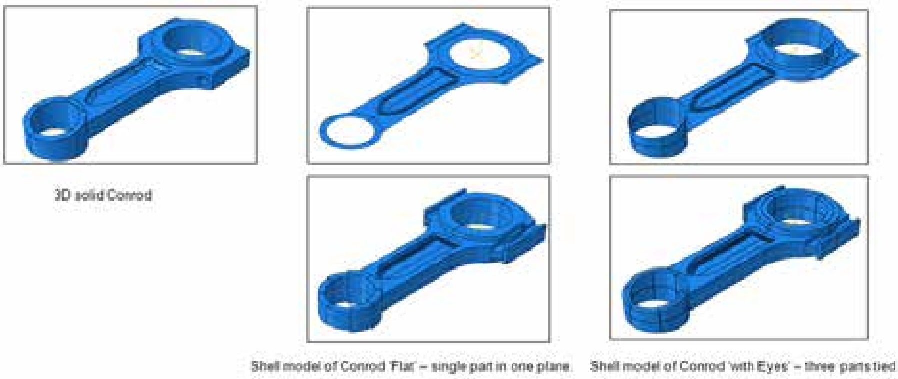


Fig. 4 - 3D and shell conrod models used in study

Conrod modal analysis

Figure 5 below shows the results for Mode 7, first out of plane bending mode, for 3D, flat shell and shell with eyes. The LHS depicts the 3D solid, mode 7 = 1203H. The middle shows a flat shell, mode 7 = 1182Hz. The RHS shows a shell plus eyes, mode

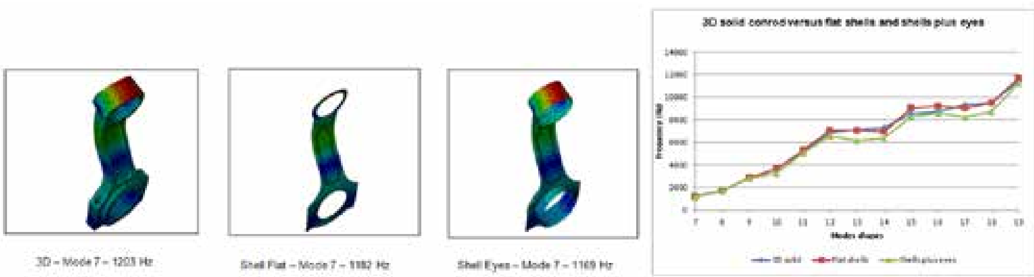


Fig. 5 - LHS model results Mode 7, first out-of-plane bending mode for 3D, flat shell and shell with eyes and the free-free modal results for all three

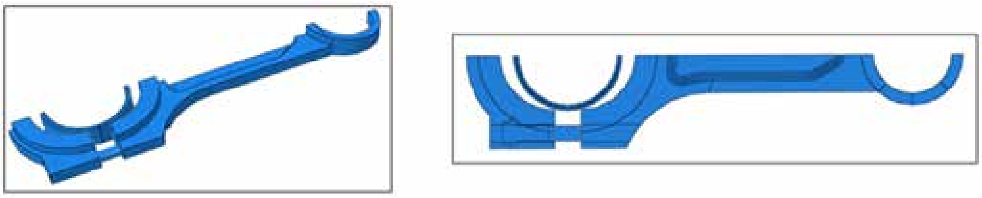


Fig. 6 - 3D solid quarter model and 2D plane stress and strain models.

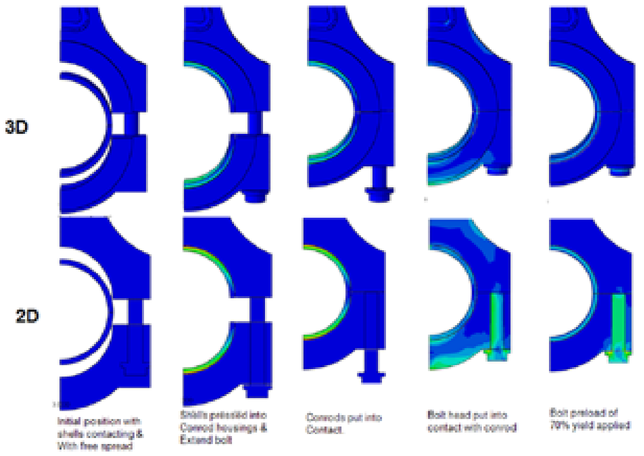


Fig. 7 - Comparison of 3D (top) and 2D (bottom) bearing fit and bolt preloading.

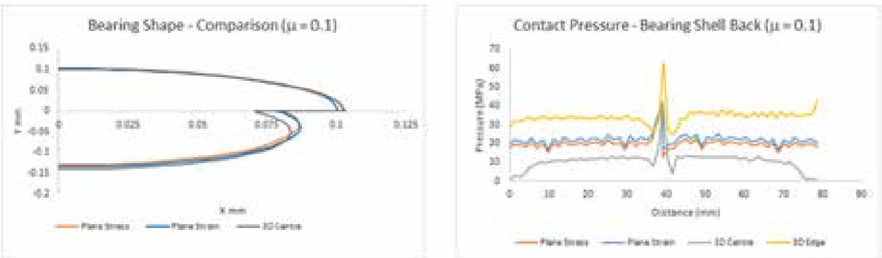


Fig. 8 - LHS -- comparisons of 3D with 2D bearing surface distortions. RHS shows contact pressure at the bearing shell back

7 = 1169Hz. So, a flat shell provides a more accurate seventh mode, however, if you also want to capture housing displacements then use shells plus eyes because you can condensate them to the bearing surfaces. The far RHS of Figure 5 above shows all the results plotted for this study, a 3D solid model versus two conrod shell representations, for all modes 7 to 19. The first six modes are rigid body modes at zero frequency, so they are not plotted. The flat shells (red line), follow the 3D solid (blue line) quite well. However, the shells with eyes (green line), do not follow the 3D solids (blue line) as well as the flat shells (red line) do. This illustrates that we can use shell abstraction methods for system analysis, replacing 3D models with shells where appropriate.

Our shell/2D abstraction versus 3D solids – fitting bearing and bolt preload

Figure 6 shows the models used in modelling the assembly process. The LHS shows the 3D quarter model and the RHS shows the 2D plane strain and plane stress model. The resulting distorted shape of the bearing shell affects film thickness, pressures profile and the stiffness damping coefficients, whilst the contact pressure affects the fatigue life of the journal and conrod. The four steps used to fit the bearings and bolt cap to the rod were as follows: bolted to nominal at 70% of yield, but can be any percentage, 3D compared to 2D plane stress and plane strain (flat models). We couldn't use shell models as we couldn't apply pressure to the edges. The objective was to see what reduction in time and accuracy could be obtained in 3D to 2D model abstraction.

Results of 2D abstraction and of 3D conrod interference study

The shells could not model the edge pressures, so we used 2D. There was not much difference at the center line displacements for 2D or 3D. But the 2D cannot model the edge effects. The LHS shows the displaced shape of the bearing shells. The RHS shows the contact pressure at the back of the bearing shell. The 2D agrees very well with the 3D. The bearing pressure at the back of the shell in 2D is an average of the 3D. The 3D pressure at the edge is 4x at the middle of shell and about 1.5x that of the 2D

Conclusions

Our abstraction methods work well but are no replacement for running a 3D simulation, especially when modeling some types of structural beams (Castellated or Porthole beams), a later paper, perhaps. The abstraction works well in the early and initial design stages when lots of iterations take place, but for the final design sign-off, it is always advisable to use a model with as much fidelity and accuracy (DOFs) as possible. However, this is not always possible in system analysis due to the large number of DOFs. Instead, large models that have been reduced for dynamic analysis provide good results for 3D beams and 2D shells.

2D shells cannot be used for connecting rod bearing shell and bolt preload calculations as they cannot take an edge pressure. A new shell element that does this is required; we may look into this in the future. The idea behind this approach is to link 3D solid modelling with the 1D, 2D and 3D models often used by system and control engineers. The 3rd paper will describe how oil films can be accurately abstracted or reduced for use with system level models, thus replacing the estimates often used by system level engineers and bringing the disciplines of system engineering closer to the designers and analysts working at the engine, gearbox and component levels.

*By Dr Ian McLuckie. Managing and Technology Director for Advanced Integrated Engineering Solutions Ltd**

*** Advanced Integrated Engineering Solutions Ltd is based in Market Harborough. LE16 7HG, Leicestershire, UK. Dr McLuckie can be contacted at email: ian.mcluckie@aiesl.co.uk, or via the web at: www.aiesl.co.uk**

New ANSYS Discovery AIM Simulation for every engineer

New easy-to-use software environment lowers barriers to access 3D Multiphysics simulations

Discovery AIM was created to be used during the design phase of a company's product development process, making Computer Assisted Engineering (CAE) analyses accessible to designer engineers with no background in numerical simulations.
ANSYS Integrated Multiphysics (AIM) is multi-purpose CAE software within a single environment that allows the definition of a wide range of numerical simulations. The user can choose between single-physics or multi-physics simulations to characterize all the aspects of a mechanical component.

The physics that can be simulated in AIM are:
- Solid-mechanics + Thermal
- Fluid-dynamics + Thermal
- Electro-magnetics
- Polymer extrusion and blow molding
- Topology optimization

Complete all-in-one software
Discovery AIM contains all the different tools required to complete the simulation workflow:
- Geometry: the package includes the ANSYS SpaceClaim environment, a feature-based geometric modeler that allows users to create 3D geometries or to import/modify/repair CAD models that have been imported from other vendors
- Meshing: AIM includes the main strategies available in Workbench Meshing to create both hexahedral and tetrahedral computational grids. These tetrahedral elements can automatically be converted into polyhedral grids using Fluent's meshing algorithms to improve quality and reduce the total number of elements.
- Physics: the software handles single-physics or Multiphysics workflows and uses ANSYS Fluent solvers for Computational Fluid Dynamics (CFD) analysis and ANSYS Mechanical solvers for structural analysis, two widely-validated and internationally-used codes.
- Post-processing: the integrated environment enables the user to visualize and export the analyses' results

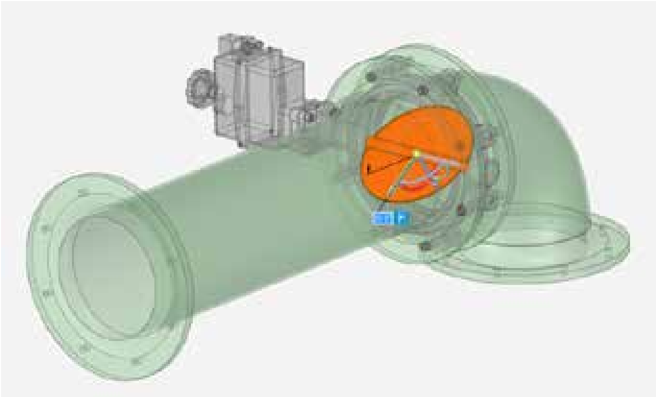
Simulation for every engineer
Discovery AIM is a complete solution for design engineers with low knowledge of CAE simulations that want to approach this discipline. All the steps from geometry creation to optimization and results generation are inserted in a single, modern, easy-to-use environment. AIM guides designers and engineers through both single-discipline or Multiphysics simulations with templates and task-based workflows. It helps non-expert users to rapidly learn the software and properly complete all the steps necessary to obtain meaningful results.

Discovery AIM can be used to solve a variety of real issues across a range of industries. Some typical industrial sectors that use Discovery AIM to design and optimize different solutions include:
- **Alternative energy:** Solar arrays, tidal power structures
- **Automotive:** External aerodynamics, ducts and air handling components, powertrain components
- **Consumer products:** Appliances, household goods and sporting equipment
- **Energy, Oil&Gas:** Pipes, valves, flow control, nozzles, and pressure vessels
- **Heating, ventilation and cooling (HVAC):** Ducts and air handling equipment, ventilation
- **Power distribution:** Bus bars, transformers and cables
- **Valve manufacturers:** Valves and flow control devices

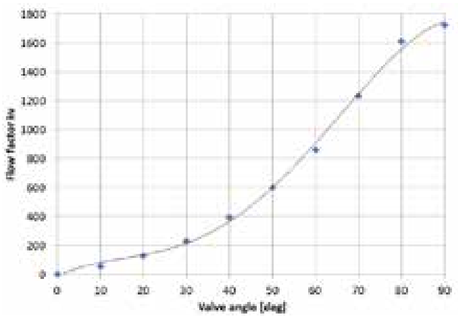
Fluid-dynamics simulations
The software's CFD template allows users to set up multiple physics analyses and provides options to simulate a large number of typical industrial fluid flows. The main simulation features available in AIM 19.2 are:
- Incompressible and compressible fluids (also supersonic regimes)
- Laminar and turbulent flows (with transition detection)
- Fluid-thermal analyses
- Conjugate heat transfer (thermal interaction with solids)
- Gravitational effects
- Particle injection models (bubbles or seeds)
- Rotational/translational periodicity
- Fan P-Q curve

While more advanced types of physics like multi-phase flows, chemical reactions or rotating frames cannot be simulated in AIM, these topics can be assessed using ANSYS flagship products.
Two examples of CFD simulations performed in AIM follow to illustrate the typical applications of this software and the typical outputs that the user can obtain.

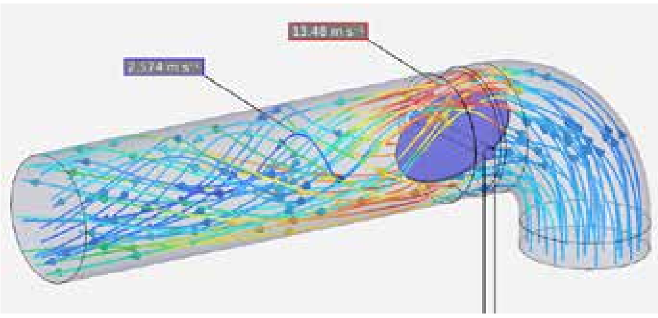
Example 1: Butterfly valve
ANSYS AIM enables users to analyze the performance and flow patterns of various type of valves.
The rotational movement of the butterfly valve can be parametrized to study the fluid flow through this duct for different valve shutter angles. The parametric study is performed automatically for different user design points. In each step, the solver changes the mesh automatically to fit the new geometrical configuration and calculates the results



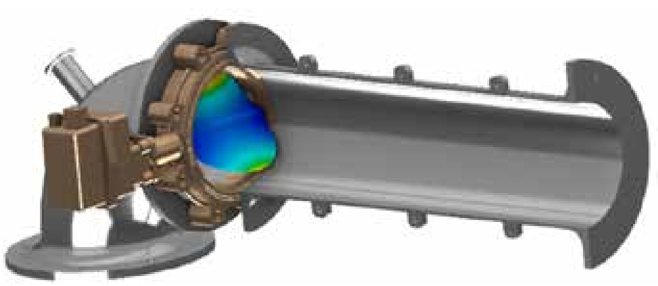
Parametric butterfly valve



Graph of flow factor for different angles



Velocity Streamlines though the valve



Von Mises stress over the valve

IT cabinet for electronic devices

based on the same or different boundary conditions. Moreover, several output parameters can be defined by the user to characterize the global performance of the component being analyzed. For example, the user can study the flow factor $k_v=Q/\sqrt{\Delta P}$ for different angles, as shown in the graph below.

The CFD analysis of this component may be coupled with a structural analysis to complete a Fluid Structure Interaction (FSI) simulation. The calculated pressure and thermal loads are taken as boundary conditions for a Finite Element Method (FEM) analysis to evaluate the valve stress and deformation. The FEM solver can evaluate the structural response due to thermal dilatation and pressure loads induced by the fluid flow.

Example 2: IT cooling systems

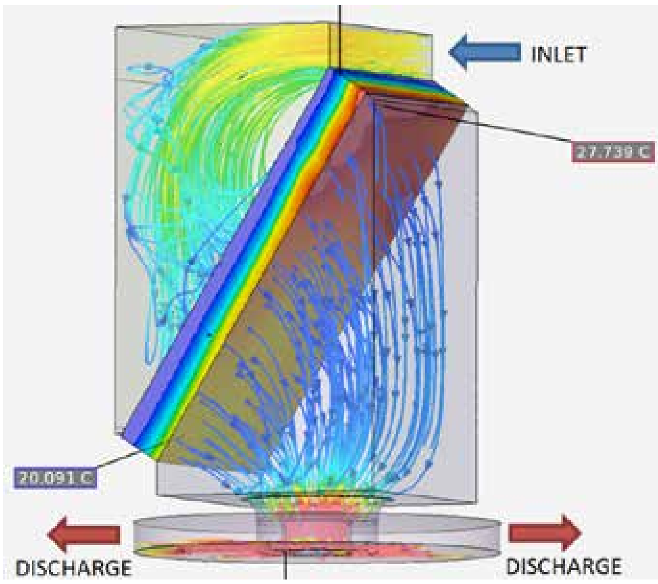
This analysis concerns the thermo-fluid-dynamics behavior of a cabinet containing heat-generating devices. The components dissipate a heat flux that varies in each location. A centrifugal fan placed in the lower part of the cabinet generates an air flow to promote cooling via air convection.

The aim of this simulation is to characterize the performance of this system to optimize its configuration. The electronic board system is modelled with a porous material that induces a pressure loss as a function of air flow; the curve is provided by the supplier. In the volume a variable heat source is assigned. The centrifugal fan is modelled with a P-Q curve that was also provided by the supplier.

The main outputs of this analysis are:

- Temperature distribution over the battery – the user can qualitatively analyze cooling efficiency using streamlines that show the critical areas.
- The fan's working point from which the pressure loss in the system can be evaluated.
- Turbulence and vortex identification to improve the geometrical configuration.

For more information:
Alessandro Cinciripini - EnginSoft
a.cinciripini@enginsoft.com



Temperature distribution over the battery and velocity streamlines

Harvesting energy to power sensors that monitor air pollution

The REM Project
The original Italian title of this project was "Recupero di Energia Meccanica da fluidi per internet delle cose e sensori remoti" from which its acronym – REM – was derived.

Final scope of the project
Recovery of Mechanical Energy from fluids to feed an Internet of Things (IoT) node that monitors diesel-engine exhaust gases.

Partners
- EnginSoft (project coordinators) https://www.enginsoft.com/
- The Center for Biomolecular Nanotechnologies of the Istituto Italiano di Tecnologia (IIT) https://www.iit.it/centers/cbn-unile
- Web Elettronica https://www.webelettronica.com/index.html

Fig. 1 – Partners' logos

The global challenge of air pollution
Poor air quality is a growing threat to public health, and air pollution contributes to more than seven million deaths annually worldwide. Governments are seeking low-cost and user-friendly solutions to monitor air quality.

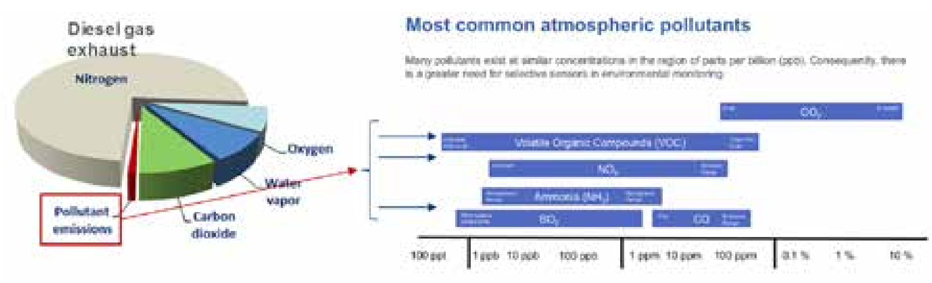


Fig. 2 – The most common atmospheric pollutants

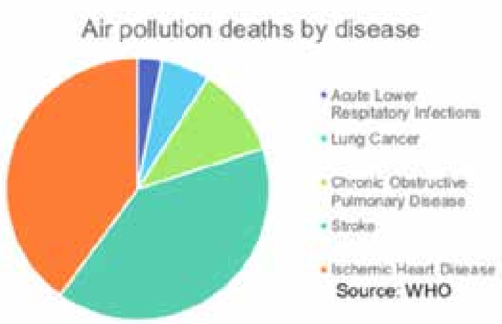


Fig. 3 – Air pollution deaths by disease

The Internet of Things (IoT): the vision of a "Smart Environment" around us
The vision of a "Smart Environment" is one of a physical world that is richly and invisibly laced with sensors, actuators, displays, and computational elements, embedded seamlessly into everyday objects and connected through a continuous network (Mark Weiser, Ubiquitous Computing).
An IoT node will be able to sense, visualize, analyze, and communicate about the environmental air pollution around us at any time.

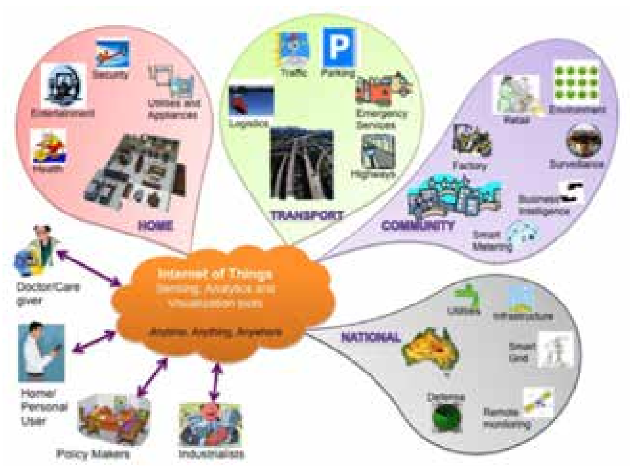


Fig. 4 – J. Gubbi et al. / Future Generation Computer Systems 29 (2013) 1645–1660

Convergence in new technologies for energy harvesting
Smart IoT nodes require energy to function. It is important to develop miniature devices with the ability to sense, visualize, compute and communicate while requiring very low energy. This requires the study of Micro-Electro-Mechanical Systems (MEMSs), Digital Signal Processing (DSP) in digital electronics, and wireless communications (Wi-Fi).

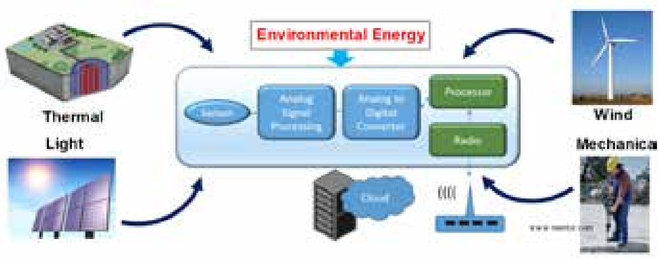


Fig. 5 – Environmental energy harvesting

Fig. 6 – Environmental energy transfer

Energy harvesters convert energy from environmental sources (light, thermal, wind, vibrations, etc.) into electrical energy to power autonomous microsystems and to extend battery life in small electronics.
The energy harvesters in MEMS are used as power sources in wireless sensor networks (WSN) which are used for structural health monitoring, building climate control and automotive sensing.

Flexible Aluminum Nitride-based piezoelectric transducers

The project is studying a piezoelectric flag transducer and is considering two technologies:

- Aluminum Nitride (AlN) thin film
    - Good piezoelectric coefficients (4-5 pm/V)
    - High temperature performance (up to 1150 °C)

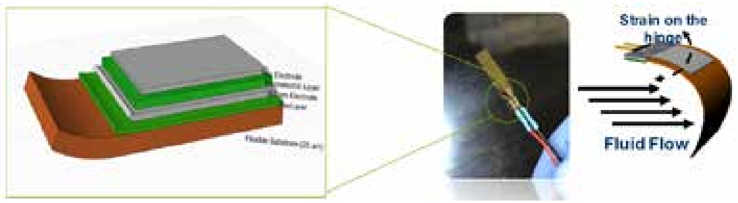


Fig. 7 – Transducer layout

	Polyimide «Structural Flag»	AlN «Piezoelectric layer»	Parylene «Water insulator»
Length (cm)	1 - 3	0.5 - 1	1 - 3
Width (cm)	0.4 - 1.3	0.3 - 1	0.4 - 1.3
Width (µm)	25	1	2
Density (kg/m³)	1420	3300	1289
Young's Modulus (GPa)	2e9	348	2.75
Poisson Coefficient	0.34	0.3	0.4

Fig. 8 – First tests and material properties

-
    - Biocompatibility
    - Growth on different flexible substrates (polyimide, kapton tape, PEN)
- Flexible Kapton substrates (polyimide)
    - Stable at a large range of temperatures [-273 to +400˚C]
    - Compatible with all the main microfabrication building blocks for MEMS.

This flexible system efficiently transforms strains of flexible substrate from fluids or vibration into piezoelectric energy. The first prototypes have been successfully created and the flag is sensitive even to very low-speed air flows, such as a breath. Directly connected to an oscilloscope, the 2.5cm-long flag generated a few hundred millivolts.

Fluid Structure Interaction (FSI) analysis

There are ongoing, detailed FSI analyses to numerically characterize the flag's oscillation.

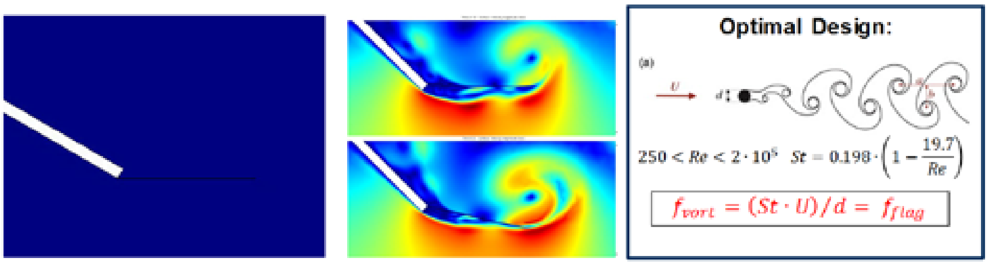


Fig. 9 – First FSI considerations (From: Petroni S., Rizzi F. et al. RSC Advances vol. 5, issue 18, pp. 14047-14052 (2015))

The optimal system design has to be found in terms of bluff body and flag geometries, the flag's orientation with respect to the flow, and source generation. To maximize the FSI interaction, the preliminary strategy will be to design the bluff body's geometry, the flag material and the geometric characteristics to select those flag resonance modes that are as close as possible to the Von Karman vortex frequencies generated by the bluff body.

Digital twin and sensor dashboard

The final physical system, consisting of a transducer and an electronic circuit for the harvesting, will be a low-cost device, accessible to and usable in any context, from private companies to industries and public organizations. The REM project's final goal is to provide a service based on a digital twin of the whole system with a dashboard accessible from both PC and mobile devices that allows the end user to check data from the sensors in real time.

For more information:
Antonio Taurisano, EnginSoft, a.taurisano@enginsoft.com
Francesco Rizzi, IIT, francesco.rizzi@iit.it
Mauro Tafuro, Web Elettronica, mauro.tafuro@webelettronica.com

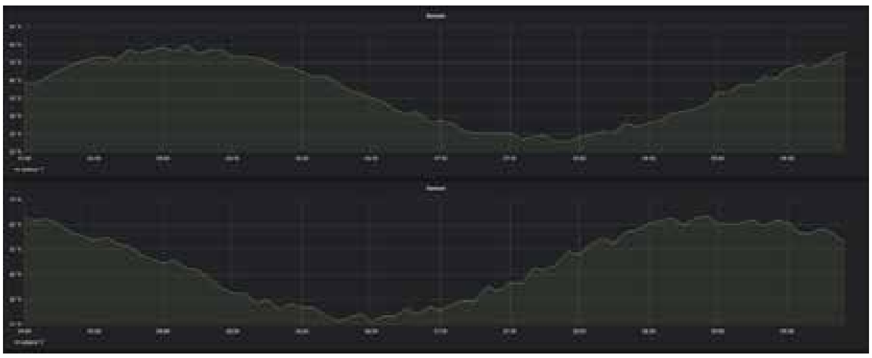


Figure 10 – Dashboard for sensor monitoring

The Aerospace Technological Cluster's Flet4.0 Project to use simulation for its new hybrid fleet management approach

EnginSoft is the lead numerical simulation and optimization partner

EnginSoft is a key partner in the Flet4.0 research and development project, which regards the development of a new approach to the predictive maintenance of aircraft engines together with Aerospace Technological Cluster (Distretto Tecnologico Aerospaziale), GE Avio and the Polytechnic University of Turin.

EnginSoft is leading the numerical simulation and optimization aspects to support GE Avio switch from their current statistical approach of maintenance management to a hybrid one using not only standard data like materials and parts availability, and the operations time charts for a group of facilities, but also and predominantly numerical simulation and optimization platforms. EnginSoft, supported by the other project partners, will develop prognostic models of engines (and their sub-systems) by collecting and manipulating flight-data gathered from on-board sensors. These models will form part of an engine health management tool that will evaluate performance and wear and will provide extremely valuable information to decision-making support algorithms.

The project's ultimate goal is the development of an optimized Removal Plan for engines that will allow GE Avio to thoroughly control the effective status of the monitored fleet, minimize maintenance costs and maximizing operational availability.

For futher information, contact:
Sandro Gori, EnginSoft
s.gori@enginsoft.com

Below is a summary of an article by Domenico Palmiotti about this project and others in the Apulian aerospace sector, which was originally published in the Italian national business newspaper, Il Sole 24 Ore, on 21 November 2018.

Italy's Aerospace Technological Cluster is growing

The Aerospace Technological Cluster has received several important injections of funding. One comes from the Italian Ministry of Instruction, University and Research (MIUR) for a District project for an "intelligent factory" to maintain aircraft, satellites and rail vehicles; another comes from the Apulian Regional government for a programme to establish a plant in San Giorgio Ionico, a town in the Apulian province of Taranto, to produce parts in advanced composite materials (eg. fiberglass, kevlar, carbon fiber) for use in the aeronautical sector.

The main portion of this second investment, worth approximately €7,5-million, comes from the Naples-based company "Lavorazione Materiali Compositi-LMC spa", active in the design, construction and testing of composite material parts for the aviation sector, as well as in the installation and assembly of parts, and complex parts, for aircraft and helicopters. According to Cosimo Borraccino, Councillor for Economic Development, the programme, was favourably evaluated by the Apulian Regional government, which will provide public financing of €3-million towards it. The programme expects to create jobs for 55 employees once fully operational.

The Apulian Region is already financing a budding aeronautical industry around Taranto through its operational program Fesr 2014-2020. For instance, not far away from San Giorgio Ionico, in the town of Grottaglie, the Leonardo plant manufactures the carbon fiber fuselages for the Boeing 787 (to date since its commencement, the plant, which has 1,200 employees, has produced 795 pairs of section and by year-end, 14 pairs will be produced monthly). There are also other companies in the same sector.

The Flet4.0 project

The FLET4.0 project, which aims to develop innovative capacities in fleet management for aircraft engines, railway vehicles and satellites, is one of the industrial research and experimental development projects in the 12 areas of specialisation identified by the National Research Plan for 2015-2020 launched by MIUR. Overall the project represents a total investment of about €8 million of which €6.6-million has been allocated to the Apulian region. The public financing amounts to €3.7-million of which €3.2-million has been assigned to Apulia. Most of the research and development activities will take place in Apulia where most of the partners are located. The project is headed by the Aerospace Technological Cluster, together with the partners the Politecnico University of Bari (in charge of the scientific aspects of the project), the University of Salento, Ge Avio, EnginSoft, Blackshape and Planetek. Among the co-proponents are the Polytechnic University of Turin, Mermec and EKA. Ge Avio has provided €3 million in resources. The objective is to acquire a greater and more accurate knowledge of the operating status of aircraft engines, railway vehicles and satellites to be able to coordinate and optimize the necessary maintenance processes. To this end, a mix of digital technologies (artificial intelligence, simulation, virtual and augmented reality, cloud systems, big data and analytics) will be used to recognize and monitor product performance decay, abnormal behaviour and any other unexpected events; identify and evaluate the causes, and then estimate the residual life of systems and their components. By integrating this information within the systems that manage the industrial production plans and the warehouses, technicians say it will be possible to provide optimized maintenance services.

2018 International CAE Conference hosts outstanding international research exhibition for the sixth year

International research was a key protagonist at the International CAE Conference for the sixth year in a row. The Research Agorà was full of outstanding European consortia that came to exhibit their cutting-edge research results in a wide range of sectors.
The entire conference community wanted to experience the tool for scanning and creating 3D digital models of real-world objects using standard smartphones. A digital twin platform designed for medical staff was presented that is capable of simulating and numerically predicting what will happen in vivo.
There was also an underground autonomous robot prototype for metropolitan environments that uses intelligent navigation. In addition, other themes like the criticality of raw materials, and new materials and technologies for aviation and aerospace came under the spotlight. Last but not least, a top-level mechatronic machine for Industry 4.0 was actively functioning in the Agorà. It was a best-in-class example of how sensors and data analytics technologies can be exploited by the manufacturing sectors.

Stay tuned to the Research Agorà to discover what International Research discovers next! In the meantime, if you were unable to spend enough time exploring the Research Agorà, we've included a selection of some of the fascinating projects in the following pages.

DEsign of components from a critical Raw MAterials Perspective (DERMAP)

DERMAP is intended to completely change the actual approach to Material Selection by properly educating and training engineers and designers.
Today, materials are selected for mechanical design without adequately considering their performance and sustainability requirements, or their supply risk in an integrated way. Many engineering materials (e.g: for aerospace and defense) use exotic elements to produce a unique set of material attributes that offer significant performance benefits but several of them are classified as "critical" from the European Community, in terms of sustainability, resource depletions and shortage.
DERMAP is an in-field training program, aimed at guiding industrial designers towards a more responsible materials-selection process that takes into account the criticality issues relating to Raw Materials. It combines dedicated training courses and properly developed and tested software tools to support the industrial Material Selection Process, as well as a clear evaluation of issues of criticality in Raw Materials. It will enable designers to identify materials with supply risks so that early mitigating action can be taken to ensure business continuity and to reduce the long-term costs of production.

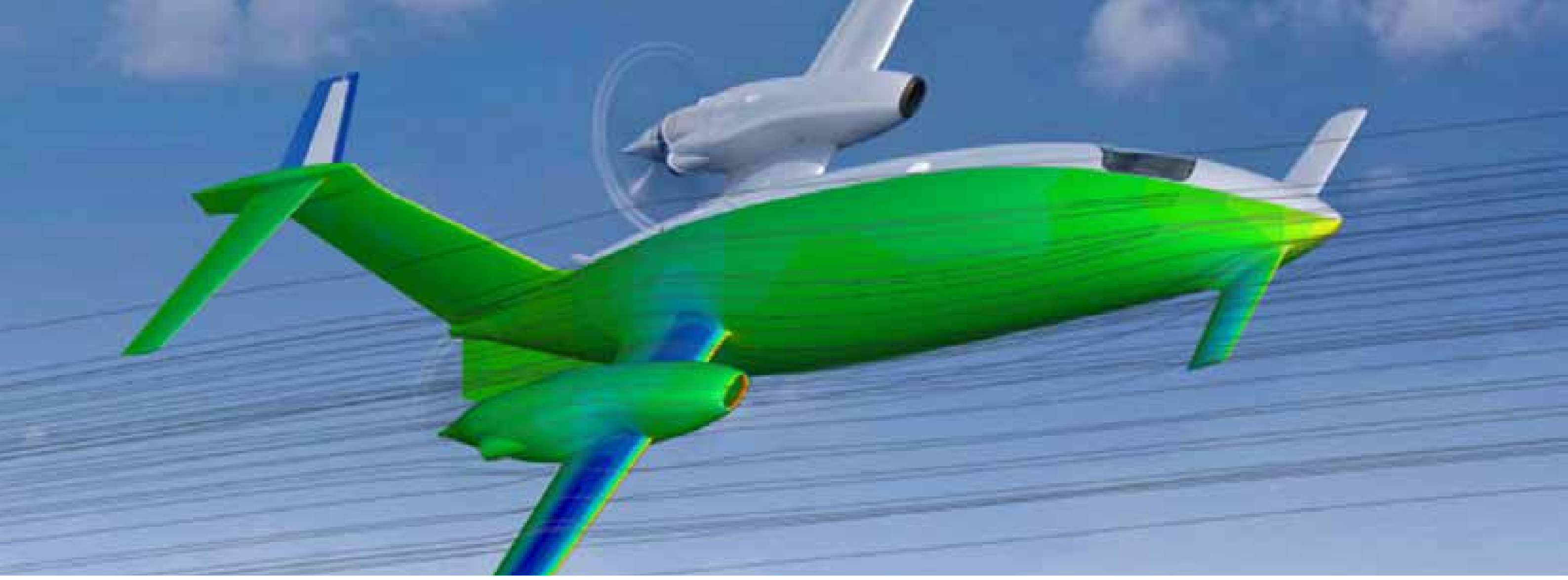
The consortium agreed to exhibit the DERMAP project at the Research Agorà and received a significant expression of interest from participants. It also provided a good opportunity to survey attendants for their opinions about the marketability of the tool and of the course.
There are six EIT-RM partners involved: the University of Padova (Italy, project coordinator); Swerea (Sweden); Zanardi (Italy); Eurecat (Spain); the AGH University of Science and Technology (Poland) and Consell General de Cambres de Comerc Industriai Navegacio de Catalunya (Spain).

There are two highly specific Task Partners and an Engineering Faculty involved: a network of 45 companies in the Foundry supply chain (Consorzio SPRING – SINFONET, Italy); Engineering & design company (EnginSoft, Italy) and the Faculty of Engineering: Mondragon Goi Eskola Politeknikoa (MGEP), Spain

DE.R.MA.P. is supported by EIT Raw Materials. This activity has received funding from the European Institute of Innovation and Technology (EIT), a body of the European Union, under the Horizon 2020, the EU Framework Programme for Research and Innovation.

DERMAP web site: www.gest.unipd.it/DERMAP/

EIT RawMaterials is supported by the EIT a body of the European Union



Fortissimo: enhancing the performance and reliability of aircrafts in the aeronautical sector

FORTISSIMO

The Project

Nowadays the aeronautical sector is very competitive, and designers are constantly urged to invest in innovative technologies to enable manufacturers to reduce aircraft development costs and delivery times.

Fortissimo received funding from the European Union'S Seventh Framework Programme under grant agreement No. 609029, and from the European Union's Horizon 2020 research and innovation program under grant agreement No. 680481 to make aircraft design more effective and efficient.

Using the RBF4AERO software platform, which was developed during the RBF4AERO project and has proven to effectively address challenging objectives such as needs for speed (time required to complete computing), accuracy (high-fidelity numerical models) and extent (different configurations tested), Fortissimo improved the cloud-based tool on high-performance computing (HPC) infrastructures.

Put simply, the platform's function is to make parametric CAE models through a meshless morphing technique based on radial basis functions (RBF). This method allows the user to enable cross-solver applications involving FSI (according to two-way and mode-superposition approaches), ice accretion simulation as well as optimizations based on the use of the evolutionary algorithm (EA) assisted by off-line trained surrogate evaluation models (metamodels) and adjoint-morphing coupling (both stochastic and gradient-based).

The Partners

The Experience at Research Agorà

The presentation of the work done on the P180 Avanti EVO model was a great success in terms of audience participation and involvement; international interest was also encouraging with participants coming from different countries.

The discussions at our booth with enthusiastic students and industrial players made our participation in the Research Agorà not only a mere dissemination activity, but a very exciting experience, too. The description of the numerical processes based on the use of RBF mesh morphing, which enabled the FSI analyses and optimizations to be done independently from the specific solver employed, was of particular interest to our visitors who wanted a deeper insight into them.

These exciting exchanges of experience, accompanied by the "virtuous contamination" of different perspectives, greatly enriched the value of the discussions and leave a pleasant memory of the 2018 International CAE Conference. We do hope to be present again next year!

RESEARCH AGORÀ

The BADGER Project: autonomous robots to facilitate underground operations in metropolitan environments

BADGER

The EnginSoft Newsletter interviewed Sandra Alvarez De Miguel of the Universidad Carlos III de Madrid and manager of the BADGER Project about their contribution to the Research Agorà

Q. What the BADGER Project about?

The BADGER project has €3.7 million in funds from the European Commission's Horizon 2020 program and has seven partners in five European countries.

Its main objective is to develop a new type of autonomous robot: specifically, underground robots with intelligent navigation for metropolitan environments.

In a nutshell, BADGER will enable access to the underground with minimum disruption, even if there are buried objects or other utilities. BADGER will expand the operational workspace from a one-dimensional one to a three-dimensional one and ease the supervision process through control and automation tools. It will also contribute significantly to reducing the time and resources necessary for underground operations, while minimizing the need for labor-intensive interventions.

Q. What does the robot look like?

The actual robot that our consortium is developing is a modular system that is 250mm in diameter and up to 3m long, depending on the specific functionalities required.

Q. What is new in this project?

The BADGER system includes various technological innovations. Through a series of sensors, geo-radars and computers, some of which are on board, the underground environment is mapped, and the robot is able to navigate autonomously, deciding where it has to go in the underground space.

Also, as the robot moves, the tunnel that it creates for the cables and pipes is reinforced with a 3D material deposition system that is located on the robot itself. This underground 3D printer is an important innovation.

Q. What is the expected impact?

The traditional approach for accessing underground spaces has been open-cut excavation. However, this technique requires the excavation of large surfaces, which involves the destruction of surface civilian infrastructure and leads to traffic congestion, noise and extended environmental interventions. This makes it very difficult to apply in areas with densely buried utilities, which is far from rare in urban environments.

BADGER's autonomous underground robotic system will drill, maneuver, localize, map and navigate in the underground space, and will be equipped with tools for constructing horizontal and vertical networks of stable bores and pipelines.

Q. What are its main applications?

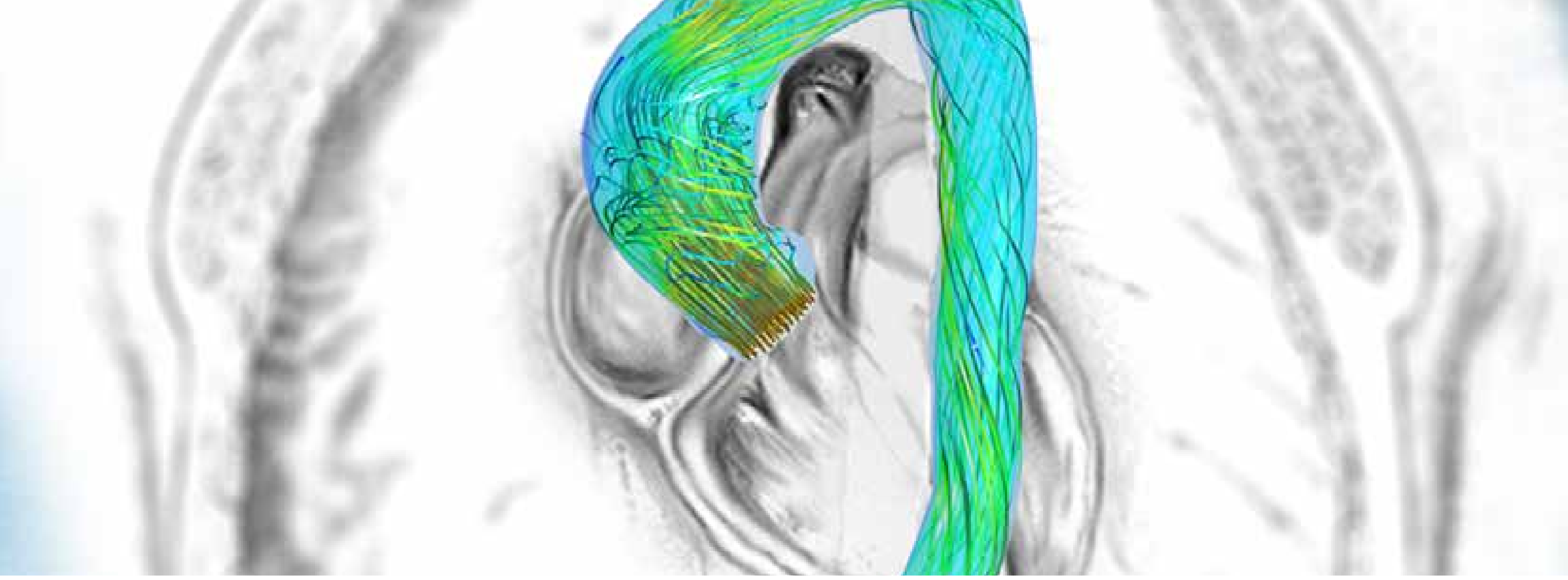
Its main application is to make pipelines in cities for telecommunications, optical fiber, small gas or water pipes, etc. without opening trenches. But the underground robot could also be used in rescue operations, for example in landslides or in mines, in natural and artificial disasters, for searching for people who have had an accident, or in other type of situations.

Q. Describe your experience at the Agorà

The Agorá provided a great environment to discuss and learn about different solutions from different sectors.

For further information on the Badger project,
visit www.badger-robotics.eu.

BADGER has received funding from the European Union's Horizon 2020 research and innovation program under grant agreement No. 731968.



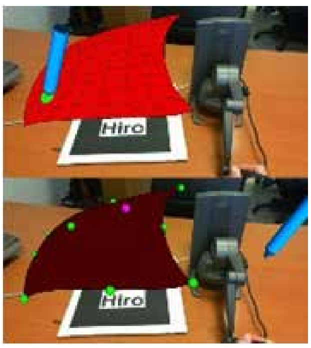
RBF4ARTIST: The medical digital twin assisted by reduced order models and mesh morphing

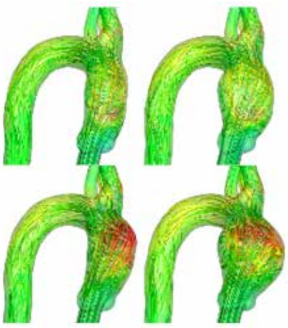
The Project

The project aims at developing a platform capable of managing a medical digital twin. This platform is developed to provide medical staff and researchers with a reliable numerical model capable of simulating what will happen in vivo. Using haptic devices and Augmented Reality (AR), the user can sculpt the model geometry interactively and adapt it to each specific patient. The shape modification can then rapidly be applied to the numerical models using Radial Basis Function (RBF) mesh morphing, a technique that has proven to be fast, reliable and efficient in several scientific fields including multi-physics simulations. The platform will quickly produce numerical predictions by exploiting the Reduced Order Models (ROMs) introduced by ANSYS which have shown to be effective for Computational Fluid Dynamics (CFD) and Computational Structure Mechanics (CSM) simulations in the medical field.

Experience at the Research Agorà

Participating in the Research Agorà allowed us to disseminate the results of our efforts to the wide audience of the 2018 International CAE Conference. Many conference attendees expressed interest in our approach to the medical digital twin, particularly in how we combined the use of haptic devices and Augmented Reality to let the user define a new model interactively on-the-fly. The use of mesh morphing in synergy with ROMs also aroused a lot of interest. The Research Agorà was a stimulating event that gave us the opportunity to disseminate our technology and work, and share in a fascinating experience.





The Partners

The REPLICATE project: creating 3D models using low-cost devices

In the past few years, 3D models have become an essential requirement in manufacturing and engineering: it is nearly impossible today to imagine designing and engineering a product without using 3D models. Product ideas can be evaluated, tested and optimized well before manufacturing the first physical prototype. In the next few years, a similar growth in the use of 3D models is also expected in the consumer and prosumer sectors. As proof of this, consider that Microsoft began including a basic tool for 3D model editing (Paint 3D) in Windows 10 in late 2016, while social networks such as Facebook are rolling out functions to support the visualization and sharing of 3D objects. Despite the increased interest and the obvious potential for 3D objects, the main barrier to wide-scale uptake has still not been fully addressed: i.e. the difficulties of 3D model creation. Creating a good 3D model is currently not only out of the reach of the average user but is also prohibitive for professionals that do not have the specific training (think about artisans, for example).

The Project

REPLICATE (cReative-asset harvEsting PipeLine to Inspire Collective AuThoring and Experimentation), co-funded by the European Commission within the Horizon 2020 programme (ICT-19-2015 call) targets this very need. The project is led by Fondazione Bruno Kessler in Trento, in partnership with ETH Zurich, Fraunhofer HHI, t2i - trasferimento tecnologico e innovazione, Wikitude, Gameware, and Animal Vegetable Mineral. The project is developing easy-to-use, real-time creativity tools that will enable individuals and groups to collaboratively scan, create and modify high-quality 3D digital models of real-world objects using standard smartphones. Target objects to scan can range in size from small desktop items, through monuments to large building complexes.

REPLICATE enables potentially anyone to quickly create their own 3D models starting from real-world objects, and then begin sharing and using them within minutes. One of the major novelties of REPLICATE is its collaborative approach: multiple users can work together to scan a large object (by simply sharing their 3D scene with others), making it very simple to work together on different details and thus to scan very large objects in very little time. This need not be done concurrently, multiple passes over many days are also possible. This add-details-to-an-existing-scan feature comes in handy if a user is not satisfied with a reconstruction result and wants to refine certain aspects of a model: simply open the app, select the scan session and press "continue".

Thanks to its simplicity, REPLICATE will open up new opportunities for users to explore and exploit 3D models. For example, think of artisans that create hand-crafted products: what if they could easily scan their products and share 3D models of them on the web (for example, on an e-commerce platform)? Or think about VR and 3D artists: what if they could easily insert real-world objects into their virtual scenes (for example, an ornament or a figurine in a virtual room), without having to model everything from scratch or use dedicated hardware? REPLICATE's collaborative approach also opens up new opportunities for creating a wide range of new interactions with objects: think for example of artistic works.

Participation in the Research Agorà

Presenting REPLICATE at the CAE Conference Research Agorà was a great opportunity for the project team to interact with professionals and experts, and to discuss possible other applications for the REPLICATE technology. In fact, it emerged that even highly-skilled 3D experts could benefit from it. For example, if Fixed Element Method (FEM) analyses or fluid dynamics analyses needed to be performed on existing objects or physical prototypes, it would be possible to very quickly generate a rough model that would enable anyone to immediately generate preliminary evaluations, leading to relevant savings in the time and resources required.

REPLICATE is a project co-funded under the European Union's Horizon 2020 Programme (Grant agreement no 687757) - Research and Innovation Action.

GASVESSEL-Themed Workshop Hosted at the 2018 CAE Conference

ESTECO demonstrated the decision support tool and design optimization platform in use by the project

Against the backdrop of the 34th CAE Conference at Vicenza in October 2018, the PNO Group (www.pnoconsultants.com) and ESTECO (www.esteco.com) organized a workshop dedicated to the GASVESSEL European Project (www.gasvessel.eu), whose main objective is to test the techno-economic feasibility of a new transportation concept for Compressed Natural Gas (CNG).

The diversification of supply routes is key to securing Europe's energy supply. This includes identifying and building new routes to unlock resources and decrease Europe's dependence on a single supplier of natural gas and other energy resources. The GASVESSEL project aims to open up new possibilities to exploit stranded, associated and flared gas in places where it is currently economically not viable to do so, and to create new, cost-efficient gas-transportation solutions. This will mainly be achieved by means of a novel onshore and offshore CNG transportation system.

One of the project outcomes is an innovative, patented solution for the manufacture of Pressure Vessels that are 70% lighter than current state-of-the-art alternatives, which is new for CNG transportation. This solution enables new ship designs that can carry much higher payloads, consequently dramatically lowering the transportation costs per cubic meter of gas.

The GASVESSEL project targets a niche market for the transportation of smaller gas volumes where the use of liquefied natural gas (LNG) concepts or pipelines is not justified. CNG does not require expensive liquefying and re-gasification plants, which are what drive up the costs of LNG. CNG is also much more flexible to transport than LNG, which requires point-to-point pipelines making it vulnerable to any associated environmental and political difficulties during construction. All these reasons make CNG the most efficient transport method for natural gas for distances up to 2,500 km.

The GASVESSEL project will open energy routes in Europe to three different oil and gas fields: the East Mediterranean gas fields, the Black Sea region, and the Barents Sea offshore oil field.

One of the steps toward the successful adaptation of the GASVESSEL concept is the optimization of gas delivery from the identified source locations to the identified markets, for different scenarios and geographical areas. This undertaking will provide indications on the optimal ship size, speed and fleet size necessary to achieve the lowest gas transportation cost. This optimization process has been managed using ESTECO's VOLTA, a web-based platform that allows engineers to orchestrate simulated data and multi-disciplinary business process optimization to enable conscious decision making and innovative product development.

Alessia Di Loreto, junior Grants and Innovation Consultant at PNO, was the first speaker in the workshop. On behalf of the consortium, she introduced the project concept and the relevant role it plays in the gas industry value chain. She stated, "GASVESSEL will completely revolutionize the midstream phase (that is the transportation and storage modes). Moreover, the innovations introduced will also have an impact upstream (on the new exploitable resources available) and downstream (for the reduction of operating costs)."

Alberto Clarich, Head of Engineering at ESTECO, presented on how the ESTECO software has been used by the GASVESSEL partners to set up a process integration/optimization workflow and share the data and process results via the web. He stated, "These activities have brought the project directly to the next steps, where the partners will design and produce ships and vessels based on the results achieved."

Finally, Luca Battaglia, Service and Support engineer at ESTECO, performed a live demonstration of how VOLTA has been used by the GASVESSEL partners.

Acknowledgments: This project has received funding from the European Union's Horizon 2020 research and innovation program under grant agreement No 723030. The authors would like to thank all the project partners for the collaboration.

How to maximize the use of Workflow Modeling

On 9 October 2018, ESTECO's staff hosted a workshop on "Workflow Modeling for Collaborative Engineering" for a very interested audience, as a collateral event of the CAE Conference that was held in Vicenza. The highlights of the presentation are presented in the following article which explains how Business Process Management can help engineers' daily activities.

Business Process Management

Over the last 20 years, modeling activities have become a regular practice within organizations, following the introduction of the Business Process Model and Notation (BPMN) standard in different industries. This standard facilitates the analysis, improvement and automation of activities within organizations.

The first step in maximizing the use of workflow modeling is to define the difference between a Project and a Process. A project takes place in a fixed timeframe, has a scope and makes use of some specific resources. It also implements change and is incorporated into the day-to-day processes of a company. A process, on the other hand, is usually defined by cycles. So, it is similar to a project, but has a start, a middle and an end, and this cycle repeats itself over an average period of time.

At a higher level, a Business Process is a series of steps performed by a group of stakeholders to achieve a concrete goal; it is finite and repeatable, it creates value and it incorporates an element of flexibility. Business Process Management (BPM), therefore, focuses on improving corporate performance by managing different business processes. BPM concentrates on five main activities: design, modeling, execution, monitoring and optimization. Process design includes the identification of existing processes, labelled 'as is', and the design of 'to-be' processes or what different stakeholders envisage as an ideal process. Modeling takes this theoretical design and introduces combinations of variables.
Execution is broadly about enacting a discovered and modelled business process and this is can be done manually, automatically or with a combination of manual and automated business tasks. Monitoring tracks individual processes, improves customer and supplier processes, and helps with process mining. As a final step, optimization retrieves process performance information, identifies bottlenecks and cost savings, and applies enhancements to the design process.

Exascale Computing Workshop at the 2018 CAE Conference

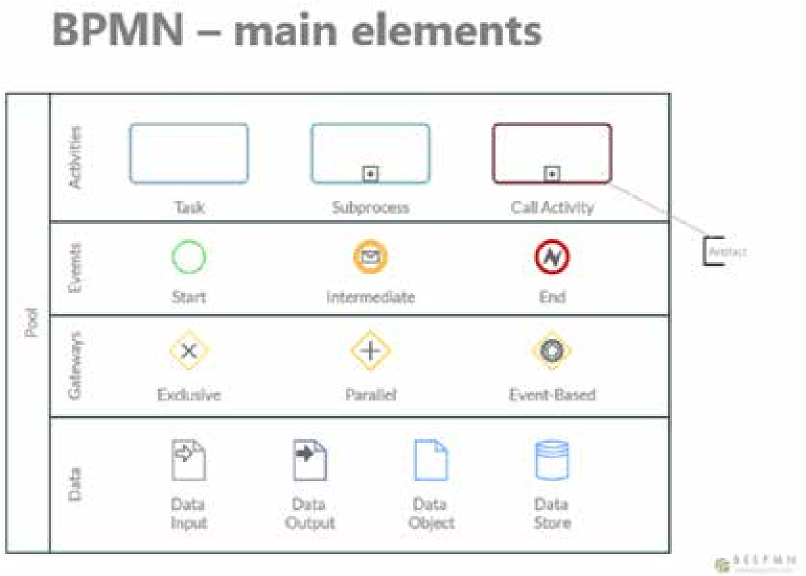
Sixteen presentations were made on the experiences and achievements of the Exascale platforms developed by the ExaNeSt, ExaNoDe, Ecoscale and EuroEXA projects over the two days of the conference. These presentations covered the full range of issues from the design of the hardware architectures and system components, to the software and the work of various application developers.

"All of this pioneering research and experience provides vital information and perspectives on the problems, challenges and potential solutions for software and hardware developers in the high-performance computing sphere. This collaboration and feedback is fundamental to assisting the efforts to make these platforms more widely available to the scientific and technological communities in the coming years, and to secure the economic, technological, scientific and industrial benefits that supercomputing offers to the global challenges facing the world today," states Gino Perna, IT & HPC services Manager at EnginSoft.

Business Process Management and Notation

BPMN is composed of three main parts: the graphical definition of the modeling elements, the transcription of the graphical model into a machine-readable XML file, and the execution semantic for process automation.

Business Analysts all over the world benefit from graphical models which help them to identify the ongoing 'as is' processes and to design optimized 'to be' operations, as mentioned earlier. BPMN is the most widely used notation standard in this field. The principal building blocks in BPMN are "Tasks", which are self-contained activities with a well-defined scope and description, and a finite duration. The standard provides for a generic task, as well as for specialized versions:



- User Task (for assigning the activity to a specific person/role)
- Manual Task (for work unrelated to any IT environment)
- Send/Receive Task (for modeling communication exchanges)
- Script Task (for invoking code execution)
- Service Task (for tasks executed by web services or automated applications)
- Business Rule Task (for automated decision making)

Tasks are represented by rectangles. Except for the generic one, they each have an icon that specifies their type.
Everything that can happen during a task or between two different tasks is an "Event", for example: errors, messages, broadcast signals, timers and similar triggers. Start and end points also fit into the Event category. Events and Tasks can interact, and this is modeled with a so-called boundary event. For instance, if a web server takes too long to respond, you may wish to stop waiting and proceed with an alternative path. In this case, you would have to use a Boundary Timer Event as the modeling element. Events are circles and they can include an identifying icon, just as the Tasks do. Gateways, instead, are diamond-shaped elements and they are responsible for governing the flow of activities. Parallel or alternative paths can exit from a gateway, depending on its marker: a "plus" sign indicates a parallel path, while the letter "x" identifies an exclusive gateway. More structured execution logics can be modeled using "Event-Based", "Inclusive" or "Complex" gateways.

The COMPOSELECTOR Project

ESTECO's online editor, BeePMN is playing a major role within the COMPOSELECTOR project, which aims to develop a Business Decision Support System (BDSS) for the complex decision processes involved in the selection and design of polymer-matrix composites. The business processes underlying the industrial partners' application cases are being deeply analyzed to uncover the business decisions that will be supported by the BDSS once it is implemented.

In the DOW application case, the BDSS is intended to help determine the optimum decision for the selection of the material and manufacturing process for a composite leaf spring at the conceptual design stage. Composite leaf springs are the elastic elements and guiding mechanism of the suspension in automotive design.

In modeling the business process underlying the DOW application case, ESTECO identified the main actors involved as well as all the activities and steps that require business decisions. This information will be used in the next phases of the project to create the BPMN models that the BDSS will run to support DOW's requirements.

BPMN applied to engineering

Workflow modeling can also be used in engineering applications such as CAE. Simulation complexity grows every day, from simple component simulations to multi-component systems. For example, a Computational Fluid Dynamics (CFD) simulation requires the generation of a CAD model, a mesh, a CFD simulation run, and a result extraction step. The CAD model requires input parameters that describe the geometry. In the CAD task, the model is generated and then passed onto the mesh task for the generation of the automatic mesh. When the meshing is completed, the file is passed to the CFD solver for the complete evaluation. After the CFD has run, the outputs are extracted.

Implementing an automatic workflow instead of performing manual tasks, allows process automation and artificial intelligence to drive the designs towards the desired goals. Optimization algorithms can be applied to automatic workflows that execute complex processes. This frees the engineer to move from a traditional approach towards a more value-added engineering design, where performance goals are set for designs. The optimization algorithm will then search the design space for the optimal solution for given a set of goals.

This makes it pretty clear that the application of BPMN to engineering can be important to maximize the capabilities of engineering process modeling, and to enable the complete automation of any project.

Additive Manufacturing wins at the 2018 PROTO Challenge

Five industrial projects developed by five companies from the Trentino and Alto Adige regions of Italy participated in the final round of the 2018 PROTO Challenge at the CLab in Trento.

For the companies, the objective of the competition was to apply the benefits of additive manufacturing and of computer-assisted engineering (CAE) to their respective projects, together with the researchers and students from four different departments of the University of Trento.
The format of the competition is very simple: on the one hand, five companies selected from the area submit a technological problem related to their production systems and processes, on the other hand, highly specialized teams of researchers and university students come together to find solutions that are shared with the business leaders in an intensive, month-long process.
The PROTO Challenge initiative is promoted by the HIT-Hub Innovazione Trentino in the context of Confindustria Trento's Regional Digital Innovation Hub and in partnership with the University of Trento and Trentino Sviluppo's ProM Facility. EnginSoft, national leaders in software and services for virtual experimentation, supported researchers and students by providing its own computer programs and ANSYS software to be used for the analysis in achieving the mechanical redesign objectives of the challenge.
The researchers and university students formed multidisciplinary teams that worked alongside the Trentino manufacturing companies to redesign plastic or metal products or elements from the companies' respective production lines using advanced software to obtain even more sustainable mechanical optimizations. The participating companies were the FAE Group (Fund), Karl Mayer (Mezzolombardo), Marangoni Meccanica (Rovereto), Vitec Imaging Solutions (which is already collaborating with Rovereto's Polo Meccatronica), and EmiControls of the TechnoAlpin Group (Bolzano).
The five projects were finally presented to professionals and researchers for evaluation, and Vitec Imaging Solutions Team Gimbal which applied topological optimization methodologies to develop a support for its top-of-the-range cameras and camcorders, was declared the winner.

Flavio Deflorian, vice rector for Support to the Production System at the University of Trento and HIT board member, stressed: "The demand for innovation and technology from Trentino companies is constantly increasing. Our territory enjoys the synergic presence of decisive elements: the research of the Bruno Kessler University and Foundation, prototyping laboratories such as the ProM Facility in Rovereto, and HIT's expertise in connecting the market with technology by providing companies with targeted innovation services. The PROTO Challenge creates an opportunity for a real exchange of expertise in a cutting-edge field such as additive manufacturing. The companies were able to discover and operationally test the solutions developed by Trentino research in the sector and see them applied to concrete cases of complex industrial innovation."
Roberto Busato, general manager of Confindustria Trento, also commented on the results of the PROTO Challenge: "Our association has always worked to ensure that the wealth of knowledge developed within university and research contexts also benefits the Italian production system. Over the years, we have stimulated the promotion of initiatives, tools and strategies to increase dialogue between companies, the university and research centers. The Challenge formula is one of the most successful because it ensures concrete, satisfactory results in a short time – for the students and research teams involved and for the companies involved."

ANSYS offers a complete simulation workflow for additive manufacturing (AM) that allows manufacturers to transition R&D efforts for metal AM into a successful manufacturing operation. Additive manufacturing (3D printing) is a technology that produces 3D parts layer by layer from a variety of materials (aluminum, steel, titanium, etc) and it has been rapidly gaining popularity as a true manufacturing process. In additive manufacturing, a digital data file is transmitted to a production machine, which translates an engineering design into a 3D-printed part.
The ANSYS AM simulation tools and EnginSoft's expertise help manufacturers to: design for AM (DfAM) utilizing topology optimization and lattice structures; conduct design validation; Improve build setup; simulate the print process and explore and better understand materials.