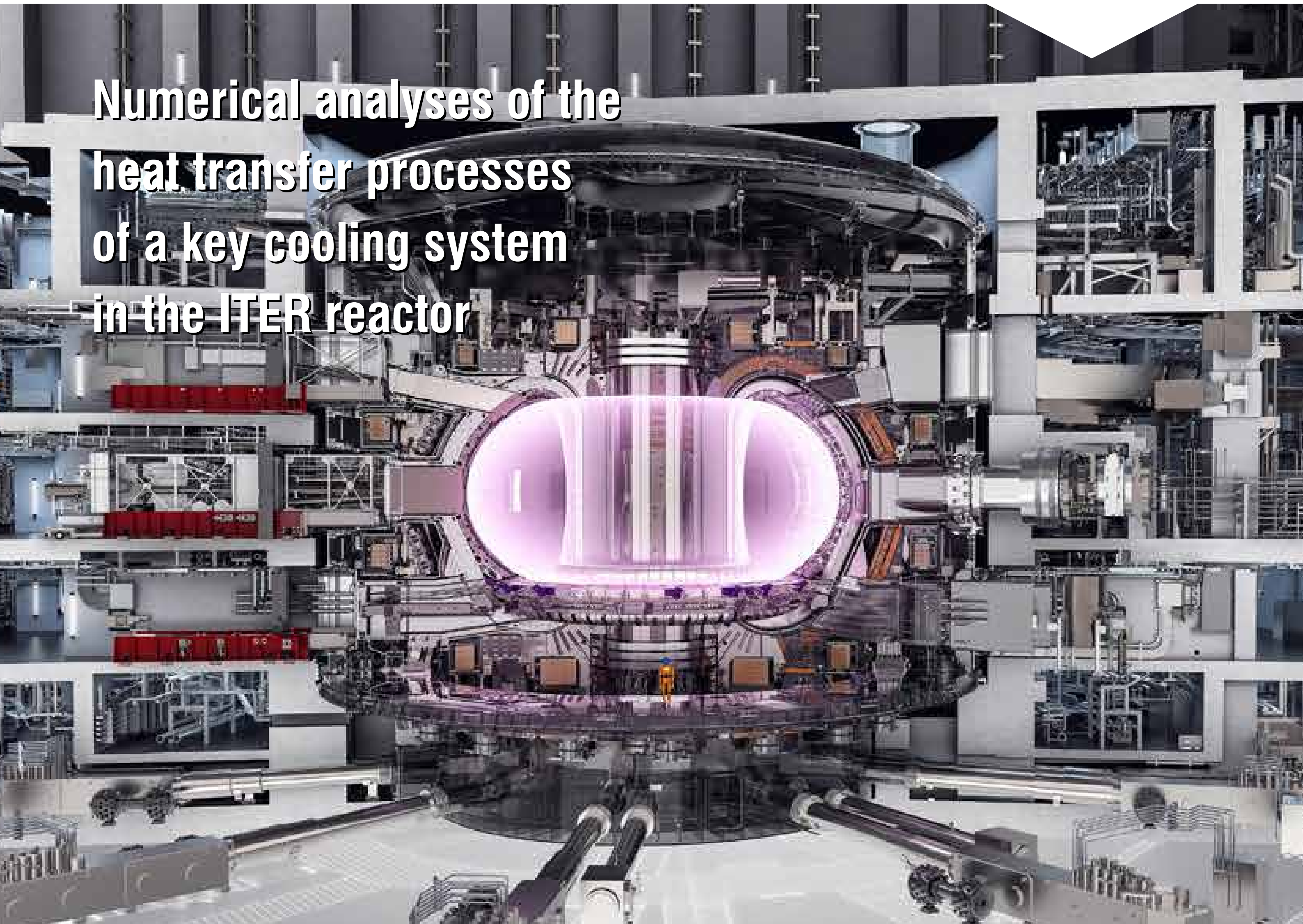
The Simulation Based Engineering & Sciences Magazine

# Newsletter

Year 17 n°4

Winter 2020



## Numerical analyses of the heat transfer processes of a key cooling system in the ITER reactor


Development and optimization of crash brackets for ECE R29 regulation compliance

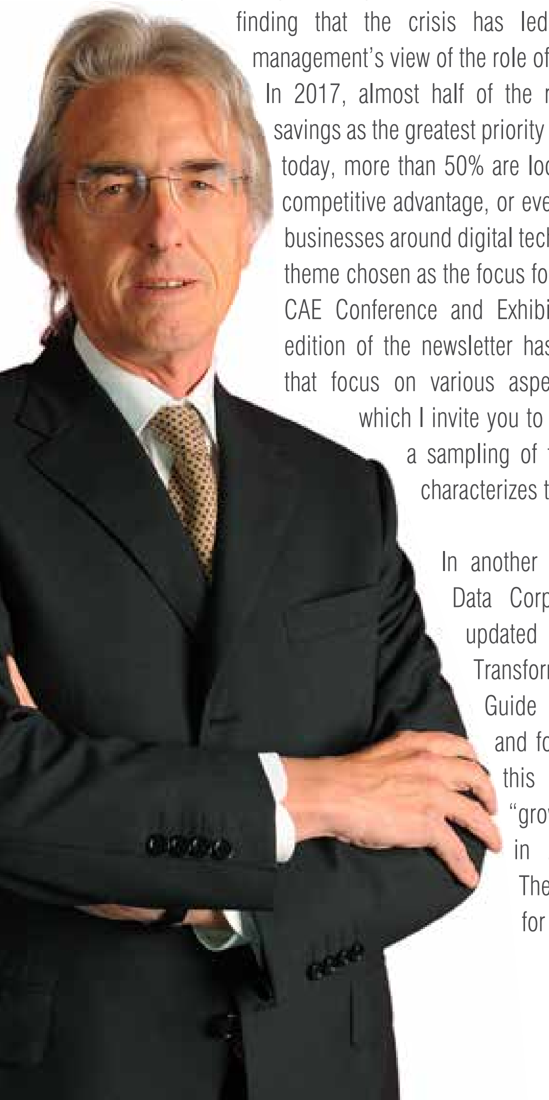
Optimizing the cable routing for a hyper-redundant inspection robot for harsh, hazardous environments

Life-safety assessment of a hotel using fire and evacuation simulations

# Flash

The end of 2020 is upon us and I am sure that most of us cannot wait to close it out, although all indications are that the effects of the Covid-19 pandemic will still be felt for much of the next year – and probably even the one after – in spite of the fragile hopes being vested in the various vaccines, all of which have been developed and are being brought to market faster than ever before in vaccine history!

This speed was also noticeable and experienced in other areas. Nine hundred C-suite executives polled by McKinsey & Company (https://mck.co/3r73jr6) in July said that COVID-19 accelerated their digital plans by as much as a decade in a matter of months, with the average acceleration of digitalization plans around the world being seven years. This digitalization concerned products, services, back office, production, and R&D. According to McKinsey, the survey respondents expect most of these changes to become permanent. Not all sectors have reported the same rates, obviously. For instance, manufacturing has been slower to implement digitalization due to the time frames involved in implementing these changes. These increases were far greater in the services related industries, and in pharma and healthcare.



Possibly of even greater significance in the long term was McKinsey's finding that the crisis has led to a sea change in management's view of the role of technology in business. In 2017, almost half of the respondents cited cost-savings as the greatest priority for digital strategy, while today, more than 50% are looking to digital to create competitive advantage, or even refocusing their entire businesses around digital technologies. Never was the theme chosen as the focus for this year's International CAE Conference and Exhibition more timely! This edition of the newsletter has a selection of articles that focus on various aspects of the conference, which I invite you to read, since they provide a sampling of the type of content that characterizes the event.

In another study, the International Data Corporation (IDC) recently updated its Worldwide Digital Transformation (DX) Spending Guide (https://bit.ly/3p42qxl) and found that investments in this area will remain solid, "growing to a forecast 10.4% in 2020 to $1.3 trillion. The specific areas of focus for these investments are autonomic operations ($51 billion), robotics manufacturing ($47 billion), and root cause ($35 billion), all driven by the manufacturing sector." Almost one-third of this spend will be in the United States, with China coming third after Western Europe.

Another aspect of industry that was dramatically impacted by the initial shutdowns in March was the supply chain and this has resulted in a serious re-think around addressing this vulnerability with regard to raw materials for production. A Gartner Inc survey (https://gtnr.it/3p2xBJu) found that more than half of supply chain management professionals will be increasing their focus on circular economy strategies to secure their access to and availability of raw materials during global disruptions.

The arguably biggest change we all experienced in our work lives this year was the pivot to remote working and the use of video-based meeting platforms and cloud meeting services. Many of these conversations are being recorded as a digital record and AI will increasingly be used in organizations to monitor their internal regulatory and legal compliance and to identify future performance and behavior among staff, according to Gartner research (https://gtnr.it/3mvft9o), which estimates that by 2025, three-quarters of work conversations will be recorded and analyzed to identify value or risk to the business, which will dramatically increase the need for policies and regulations to manage and direct the related ethical and privacy issues!

With all of its myriad challenges, many of which are still being grappled with – and have yet to be confronted – at all levels of politics, research, business, industry, and society, and in spite of the inevitable sadness that must accompany the loss of so much life and the hardship being experienced by so many, the pandemic has generated and continues to stimulate substantial innovation and collaborative activities to celebrate. While this is truer in certain sectors that in others, and many will justifiably protest that the news in their sector is worse than ever and shows no signs of improving, I believe we should look to the positive developments and remember that always in human evolution, adversity has been the incubator for great invention. As engineers, particularly, we should keep this in mind and turn our focus to how and where we can innovate in our specific business or specialization.

This edition of the Newsletter includes a series of articles that covers a selection of engineering innovation examples, which I invite you to read with as much interest as I did. All that remains is for me to wish you and those you hold dear, a peaceful and safe festive season, however you decide to celebrate it, and I look forward to working with you on innovation in 2021.

**Stefano Odorizzi,**
Editor in chief

# Contents

## Interviews



## Case Studies



## 36th International CAE Conference
## SPECIAL SUPPLEMENT

## Research & Technology Transfer



## Software Update



**Cover picture - Credit ©ITER Organization, http://www.iter.org/**

**PAST ISSUES at**
www.enginsoft.com/magazine

**Newsletter EnginSoft**
**Year 17 n°4 - Winter 2020**

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.com
e-mail: info@enginsoft.com

***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
M3E Mathematical Methods and Models for Engineering www.m3eweb.it
AnteMotion

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

**ADVERTISING**
For advertising opportunities, please contact our Marketing office at: info@enginsoft.com

**RESPONSIBLE DIRECTOR**
Stefano Odorizzi

**PRINTING**
Grafiche Dalpiaz - Trento

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

## Access to Big Data and calculation capacity

# Software development is always focused on a policy of democratization

### Engineer Roberto Gonella introduces us to the innovations in computer aided design

**By Lucilla Niccolini**
Corriere Adriatico

*In the world of digital transformation, the role of mathematical modeling, which is the foundation of the numerical simulation tools that support engineering, seems to have taken a back seat to data processing.*

**Engineer Gonella, in applying your wealth of knowledge based on 40 years of growth and transformation to the issue of digital transformation, what has changed?**

“We must not get confused: digital transformation is only an enabler, the knowledge of processes and the ability to formulate interpretative, analytical and numerical models, remain the ‘core competence[s]’. Modelling engineers of 40 years ago had to think carefully before launching a costly simulation, compared to the young digital engineers of today, who have unlimited access to computing capacity, supported by interactivity and graphics. And in the future, even by AI. And then, the computing power, the level of precision and reliability attained by the sensors, and their connectivity enable engineering forecasting models to be successfully linked to real performance.”

**How do you see predictive simulation tools and applications in the engineering world evolving in the near future?**

“The development of numerical simulation tools today is inextricably linked to the search for technically significant, historical and complete data: it is necessary to provide the mature and robust prerequisites for completing numerical analyses. The goal is to determine the specific design performance. In the past, input data belonged in the realms of expert designers; today the process is to organize BIG DATA through the development of software designed to harmonically and easily access the basic information flow of new design development. In essence, predictive analysis depends on the processing of data that is spread across different departments in individual companies.”

**Are there any particular developments that you find interesting and which deserve attention?**

“There is a common effort by large manufacturing companies to invest in the development of computer-aided design. The interesting aspect is that there are no sectors that are not affected by these needs. I have personal experience in the defense sector, Oil&Gas, aeronautics, etc… as well as in apparently less technological sectors, for example footwear manufacturing (the ‘WinShoes’ project). The concept of computer ‘service’ has certainly changed, 40 years ago it was concentrated in mainframes, then it became widespread with personal computers, subsequently it was integrated into networks. Today the concept of the centralization of data management is back in vogue: through new systems, from server to cloud, and with new problems, such as cybersecurity.”

**Beginning 40 years ago, the use of computer programs to solve engineering problems was entrusted to experts knowledgeable about the problem. Has anything changed today with the so-called "user friendly" approach?**

"The analysis of an engineering problem, using finite element models for example, consists of three mandatory phases: "pre", "solve" and "post". In 1988, when I first started working, companies had three separate offices, each one in charge of one of these operations. For several years, computer programs have simplified operations, allowing a single user to take control of the entire process stream. In short, the major players in software development have consistently focused their efforts on "democratization": making software easily accessible to all users (user friendly access). The dimension of the problem remains the determining factor: it is one thing to model the component of a car, which is certainly made easier by the tools indicated; it is quite another thing to model the evolution of an oilfield over time in 3D, which requires considerable computing power. Just think of ENI's new computer dedicated to this purpose."

**Software platforms and the ability to integrate different physics and scales, can you explain to non-experts what this means?**

"Companies' guiding principle is precisely to have a technological platform that can organize the product development workflows, starting from the basic drawing (Computer Aided Design, CAD) all the way to the final product design, and analyzing all the engineering aspects in the process. This involves focusing on the interactions between different mechanical behaviors, thermal response, the fluid-dynamic field, etc., in other words, different physics. And, last but not least, compliance with the regulations applicable to the product. Implementing such Multiphysics projects is complex and highly disruptive to traditional staff operations, and involves significant investments in key hardware, dramatically affecting budget costs. And then there is the scale: in the past it was difficult to switch from the component to the global structure. Today we can integrate the behavior of a large structure - for example, the overall superstructure of a ship - with that of a specific solder in the midship section, right down to the level of its lattice-dislocations."

## Engineering into the future

### Core competences

- knowledge of processes
- ability to formulate interpretative, analytical, and numerical models

### Enabler

- digital transformation

### Specific design performance

***In the past***

- input data based on the experience of expert designers

***Today***

- organization of BIG DATA
- software designed to easily access the basic information flow
- predictive analysis based on the processing of diffuse data

### The three phases of analyzing an engineering problem

- pre-processing
- solving
- post-processing

### Digital twin

realization of a digital replica of the component and of a complex system

## Who is Roberto Gonella?

**Director of presales and special projects at EnginSoft**

With a degree in Aeronautical Engineering from Pisa, he worked at Italcae, where he participated in the development of the "Black Shark", a state-of-the-art heavy torpedo, the detailed design of the Arane 5 boosters, and the dynamic dimensioning of the powertrain system of the 8-cylinder Ferrari engine. Once Italcae was incorporated into EnginSoft, engineer Gonella, in the role of Director of Presales and Special Projects, is responsible for all the technical activities that govern the engineering software proposition within companies, and the management of special projects, including a nuclear site in Cadarache (France) and Castorone with Saipem.

**What are the attainable potential outcomes from combining the most advanced traditional modeling with the instruments that record performance data?**

"This question contains the right path, which should be followed scrupulously, to progress successfully towards the attainment of very ambitious targets in terms of cost savings. In fact, whenever a digital system is applied to design, the time required for experimental tests will be reduced. The related costs and "time to market" will become the key elements for judging the financial rating of every single company in the third millennium. Today's key tool is the "digital twin", that is the realization of a digital replica of the physical and operational component, together with a complex system, connected to instruments on critical sub-components, which transmits their usage and performance conditions to this replicas, all aimed at greater efficiency and safety of use, whilst also reducing costs".

*A special thanks to Lucilla Niccolini and the "Corriere Adriatico" newspaper which first published the interview with Roberto Gonella in its publication on November, 11th 2020.*

# Tackling new international challenges in the product design process with a targeted CAE approach

## Haier Europe entrusts EnginSoft to conduct a technical assessment for introducing Robust Design


**By Alessandro Mellone**
EnginSoft

*Candy, now part of the Haier Group, is an international brand that offers a complete range of large appliances combining the most advanced technologies, functional and Made in Italy design, and excellent performance. Over the decades, the company's mission has remained constant: to respond to the needs of consumers with innovative products that are easy to use and affordable for everyone.*

*After graduating in Electronic Engineering from the Politecnico di Milano, Giulia Lasagna joined Magneti Marelli as Sourcing Program Manager. She subsequently moved to Whirlpool Europe as Commodity Manager after a brief parenthesis as Procurement Manager at Corning Optical Technologies. She joined Candy Group (now Haier Europe) in 2007, where she first deepened her leadership role as Global Commodity and Corporate Quality Manager before assuming her current role as Industrial Director for Kitchen Appliances in 2015.*

*In this article, she was interviewed about her experience of the assessment service provided by EnginSoft which she had requested for her team at Haier Europe and which was focused on Robust Design.*

*The EnginSoft assessment is a customizable consulting service which provides:*

- *A 360° analysis and mapping of the maturity of the client's design processes and their CAE technologies;*
- *A detailed technical feasibility analysis for Digital Twin, digitalization, and multidisciplinary projects;*
- *An analysis service enabling the introduction of simulation and collaboration platforms based on process integration technologies.*

***Alessandro Mellone (AM):*** **Engineering challenges today are becoming more and more complex, even in the household appliances sector where Haier Europe operates. Given this complexity and the different facets of simulation, what do you think, technically speaking, of EnginSoft's assessment regarding Haier Europe's requirements?**

*Giulia Lasagna (GL):* Engineering has become extraordinarily complex in the household appliances sector, too. Haier Europe faces new market challenges every day, not only nationally, but also internationally, where competitiveness is everything. Therefore, compared to the recent past, we required a completely different perspective on the product development process to be based on an efficient restructuring of our CAE systems and with a holistic approach. At the same time, we wanted to implement a more pervasive use of simulation across the entire product development process.

To this end, Haier Europe has embarked on a process of transformation by adopting the most advanced software systems and analysis methodologies. We want to implement a structured and guided approach starting with Robust Design tools and so we entrusted EnginSoft, as experts in Simulation Based Engineering Science, with conducting a dedicated technical assessment of Haier Europe's design process using robust design. In addition, we also needed to find the right mix of teamwork in the Italian-Turkish technical team. EnginSoft's assessment, while oriented towards the technical theme of Robust Design, also revealed the difficulties being faced by both the team and the individuals. We emerged from the process more united from the human point of view, and more conscious of the real difficulties to be faced, but with the understanding that EnginSoft will support us in our path to improvement.

## About Haier Europe

Candy is an Italian brand that for over 70 years has been offering products that combine accessibility, innovation and ease of use with the aim of meeting the needs of its consumers and accompany them in their everyday lives with smart solutions.
Candy is an International brand of Haier Europe, part of Haier Smart Home, the number one group globally in home appliances and among the Fortune Global 500 Companies.
Listed on the Shanghai Stock Exchange (SHA: 600690) since 1993, Haier Smart Home is present in all 5 continents with 25 industrial parks, 10 research and development centers and approximately 100,000 employees. With an operating revenue worth 25 billion Euros in 2019, the company's global sales network covers more than 160 countries.
Haier's vision is to become the global leader in IoT for smart home appliances. The company's European headquarter is located in Brugherio (MB), Italy.

**AM: What do you think about the assessment experience from a managerial point of view?**

GL: I view the assessment as positive. The EnginSoft team adapted its methodology to our specific needs in a well-structured and timely manner. Typically, these types of activities can seem pointless and are frequently boring. EnginSoft's approach, instead, not only kept the participants constantly involved, but also brought staff from different levels of the company, from the Industrial Director to the CAD designer, to the table, encouraging open dialogue without reverential barriers, and helping to bridge the cultural gaps between the members of the Italian and Turkish teams. In my opinion the team is now prepared to implement what we have learnt with EnginSoft in our procedures and is also fully cognizant of the effort that is being made by Haier Europe to empower them to work optimally and with the right tools. In conclusion, I believe EnginSoft's technical assessment has helped me to achieve some of my goals as Industrial Director, namely instilling the team with greater technical awareness and increased cohesion.

*Pictures taken during the assessment*

**AM: From a Robust Design perspective, the assessment clarified that Haier Europe needs to implement structured management of geometric product specifications (tolerances) across the entire product development cycle. Within this context, what are the most critical aspects to be addressed and what economic impacts do you expect to achieve as a result?**

GL: The main scope of Robust Design is to find the best combination of parameters to ensure minimal dispersion of a system around the required values for any combination of disturbance factor values. In industrial practice, this translates to an attempt to limit the effect of these disturbance variables by using statistical and error-search techniques directly during production or during the use of a product. Designing a product that later proves to be difficult to monitor statistically during production means having invested in poor quality, which precludes the possibility of improving the product, and increases the monitoring costs. One method for controlling product quality before going into production is to find the right combination of often-numerous design parameters to render the product and the production process relatively insensitive to variations, and to the influence of disturbing factors. I expect a Robust Design approach to help us to achieve our industrial goals more easily in the near future, by enabling us to bring to market products that can guarantee us better profits and greater customer satisfaction.

**AM: In conclusion, would you recommend this experience to your counterparts in Haier Europe and to other companies?**

GL: Honestly, I was extremely impressed by the work done by EnginSoft in only three days of close contact with the Haier Europe team involved. This short duration, condensed into just three days as I said, makes it easy to maintain a good pace during the assessment session and then easy to implement and to execute the plan afterwards. I expected a remarkably high level of assessment, and the results obtained met my expectations at a management level. In addition, the assessment also has high technical and educational content, and it is precisely the technical content of the EnginSoft assessment that makes it possible for us to structure our future work clearly and comprehensively.

Irrespective of the theme being dealt with, I would certainly recommend this experience to my counterparts at Haier Europe, and to other companies in the Italian industrial landscape. EnginSoft, as a leader in Simulation Based Engineering Science, is the right partner with which to embark on Digital Transformation, and their assessment is the ideal tool with which to begin the journey.

For more information:
Alessandro Mellone or Giulio Cenci - EnginSoft
a.mellone@enginsoft.com, g.cenci@enginsoft.com

# 36th America's Cup presented by PRADA: Luna Rossa Prada Pirelli makes extensive use of ESTECO Technology for AC75 Yacht Design

## Interview with Martin Fischer, Co-Design Coordinator Luna Rossa Prada Pirelli Team

*During the recent ESTECO International Users' Meeting 2020, we met Martin Fischer, Co-Design Coordinator for Luna Rossa Prada Pirelli Team, to learn more about how modeFRONTIER has become a valuable tool for hull and foil design, but also to optimize sailing techniques.*

**The 36th America's Cup has introduced a complete new boat class: foiling monohulls. A new concept, new challenges and little existing experience. How did modeFRONTIER help you in the preliminary design phase?**

modeFRONTIER helped us really tremendously in the preliminary design phase because these boats were completely new, we didn't have any experience. For example, we didn't know at all what the hull should look like to have an easy takeoff. We didn't get how these boats should get onto the foils. In order to explore that in the beginning, with our simulator we developed a takeoff sequence that was controlled by modeFRONTIER.

Then we used these optimized takeoff sequences to define a cross-function that represented the quality of the takeoff phase, and then we used that crossed function in the hull optimization that was also run with modeFRONTIER.

We ran that in several loops. At the beginning we explored a very wide design space because we didn't know anything about these boats. Initially with less details, then we slowly introduced more details to describe the hull shape and reduced the size of the design space.

All this took us about three months and without modeFRONTIER it would have been impossible - it would have taken ages and we never would have found the very unusual hull shape that we found in the end. A hull shape came out that none of us had expected.

I consider it a big success and it was really thanks to modeFRONTIER that we found this hull shape.

**Multi-fidelity and multi-disciplinary optimization have become a crucial aspect in the design of complex systems like the foiling monohull. How did you use modeFRONTIER's capabilities to address these topics?**

In order to address multifidelity optimization we ran optimizations on different topics in which we varied some of the input parameters that went into the optimization.

For instance, for hull shape, and also for foil design, we defined certain boat speeds and certain loads for those speeds. Since we didn't know exactly if those were the real numbers, we varied them and then ran several optimizations with different parameters.

This allowed us to take into account the uncertainties and therefore it gave us the confidence that in the end the shapes that came out of this optimization process were not just optimized into a small local minimum, but that they represented a global minimum, or as close as possible to that.

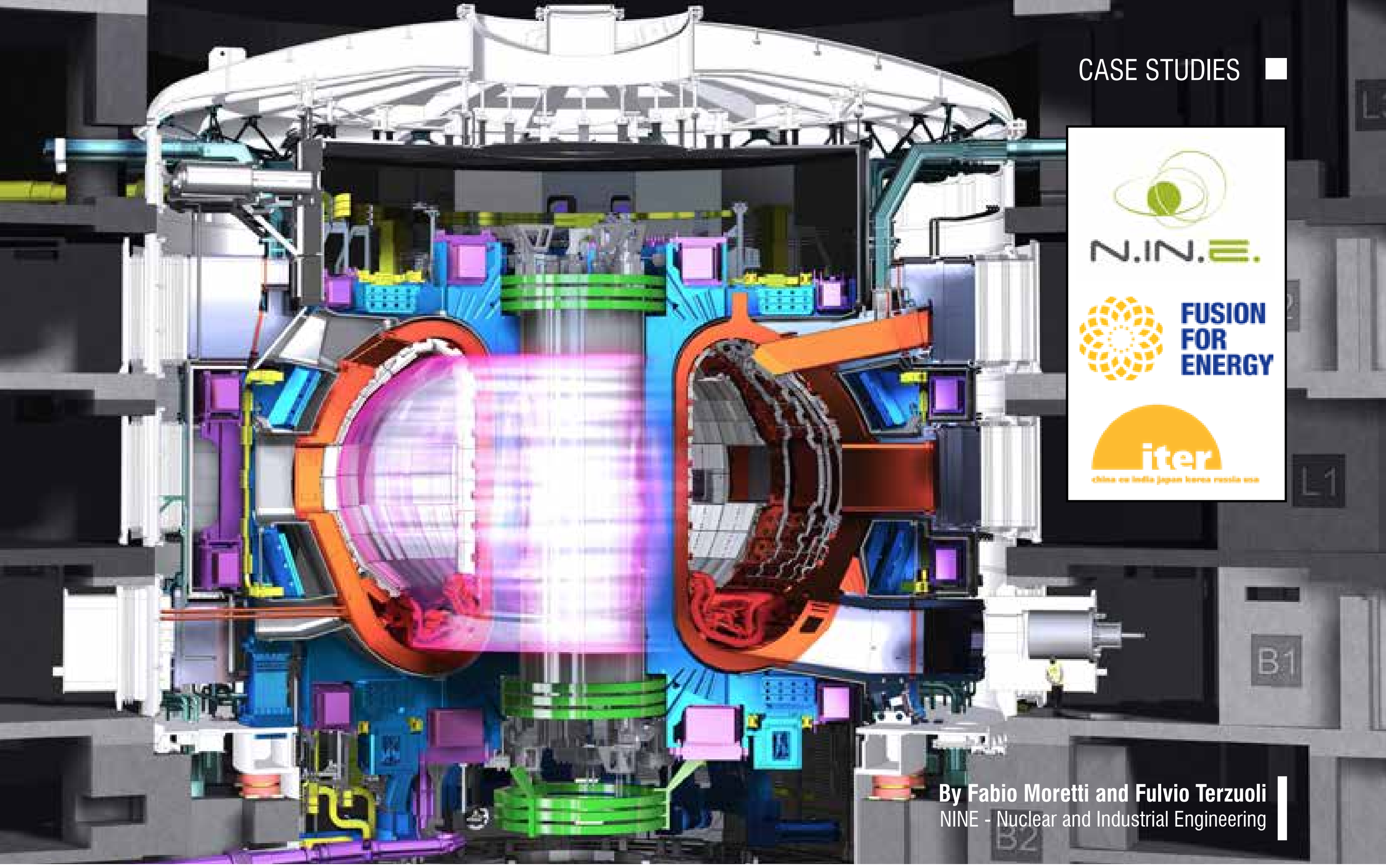
**The America's Cup is a highly competitive environment, where the time may be one of the most important factors along with the performance. Could you tell us about any time saving and performance enhancements obtained with modeFRONTIER?**

We used modeFRONTIER in the optimization of hull shapes, foil shapes, and also to optimize sailing techniques. In all those fields modeFRONTIER allowed us to explore a much wider design space in a given amount of time than we could have done without the use of such a powerful optimization tool.

Time and performance are very closely linked in the America's Cup campaign because we have a very limited amount of time. If you have a very fast tool that allows you to explore a wide design space in a given time, then for sure you will in the end get a better and faster boat.

modeFRONTIER allowed us to explore within the given timeframe that we had the biggest possible design space and therefore to get the most out of the time that we had.

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




**By Fabio Moretti and Fulvio Terzuoli**
NINE - Nuclear and Industrial Engineering

# Numerical analyses of the heat transfer processes of a key cooling system in the ITER reactor

## Supporting the design of a critical system with realistic numerical simulations of heat radiation and conjugate heat transfer

*The ITER reactor's electron cyclotron heating and current drive (EC H&CD) system launcher requires an effective cooling system due to the strong thermal loads it supports. In supporting the design of this cooling system, NINE performed several numerical studies using the Ansys simulation tools to:*

- *calculate the distribution of the heat flux on the plasma-facing materials by computing the radiation view factors;*
- *conduct a computational fluid dynamics (CFD) conjugate heat transfer analysis of the launcher's structures and cooling circuits, while accounting for thermal loads such as plasma heat radiation, microwave stray radiation, and power deposition from neutron and gamma radiation;*
- *estimate the pressure losses along the cooling circuits.*

*These analyses are part of a wider workflow carried out in a collaboration with other partners. The results of the above-mentioned simulations were then transferred to other analysts for the launcher's structural integrity checks.*

The International Thermonuclear Experimental Reactor, known as ITER, is an international project aiming to design, construct and operate a tokamak-type (i.e. based on the concept of the magnetic confinement of the plasma inside a toroidal chamber) nuclear fusion experimental reactor, in which the plasma reaches stable physical conditions for a self-sustaining fusion reaction, thus demonstrating that a larger amount of power can be generated than is required to heat up the plasma. The project is being carried out by a consortium composed of the European Union, Russia,

China, Japan, the USA, India, and South Korea. The ITER plant is being constructed near the Cadarache research facility in France. It plans to achieve the so-called first plasma by 2025.
Thousands of scientists, engineers and technicians have been contributing to this huge and ambitious project for decades, facing many scientific, technological, and organizational challenges, generating an enormous amount of progress in many areas, and producing many industrial opportunities. Italian universities, research centers, industries and engineering firms have been deeply involved in the process.

Over the past four years, NINE, a leading Italian consultancy in the design and safety analysis of nuclear installations, has been actively involved, as a third party, in a multi-year framework contract *[the Politecnico di Milano and EnginSoft are also involved in the same project as third parties]* between Fusion for Energy (F4E) *[https://fusionforenergy.europa.eu/]* and NIER Ingegneria Spa to provide support services in the areas of nuclear safety and engineering for the Electron Cyclotron (EC) and the Ion Cyclotron (IC) Heating and Current Drive (H&CD) system antennas (see Fig. 1) of the ITER nuclear fusion reactor project (https://www.iter.org/).

These large and complex devices are intended to help heat and shape the plasma by conveying powerful beams of electromagnetic waves (worth several megawatts) from external generators into the tokamak (see Fig. 2), and are thus exposed to intense thermal loads, such as the heat radiated by the plasma, the neutron and gamma radiation produced by the fusion reactions, and the power deposited by the electromagnetic beams on the transmission lines and various optical components. To give an idea: for the nuclear fusion to occur in the plasma, a temperature of 150 million degrees Kelvin must be reached, and the affected structures of the EC H&CD launcher are only a few meters away from the plasma.

In order to preserve the operational and structural integrity of the reactor and to guarantee safety, all that heat must be effectively removed by dedicated, especially designed cooling systems.

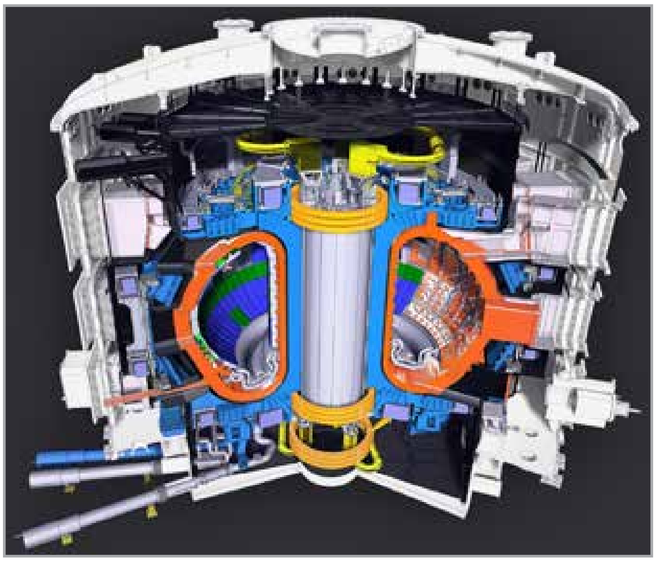


*Fig. 2 - The tokamak [Credit ©ITER Organization, http://www.iter.org/]*

NINE's focus in particular has been on the heat transfer and thermal fluid dynamic analyses necessary to support the design of these cooling systems. The Ansys simulation tools have been used systematically for this purpose.

### Characterization of the thermal loads from plasma heat radiation by calculation of View Factors

One challenging problem that has been successfully addressed is the quantitative characterization of the plasma thermal load on the structures and components of the EC H&CD antenna (or "launcher").

The plasma is contained in a toroidal vacuum chamber and is kept confined by strong magnetic fields, which prevent it from coming into contact with the chamber walls (and with any materials that face the plasma through openings on these walls, such as those belonging to the launchers). Nevertheless, all plasma facing materials are exposed to intense heat radiation, the characterization of which requires solving a rather complex and computationally expensive thermal radiation problem.

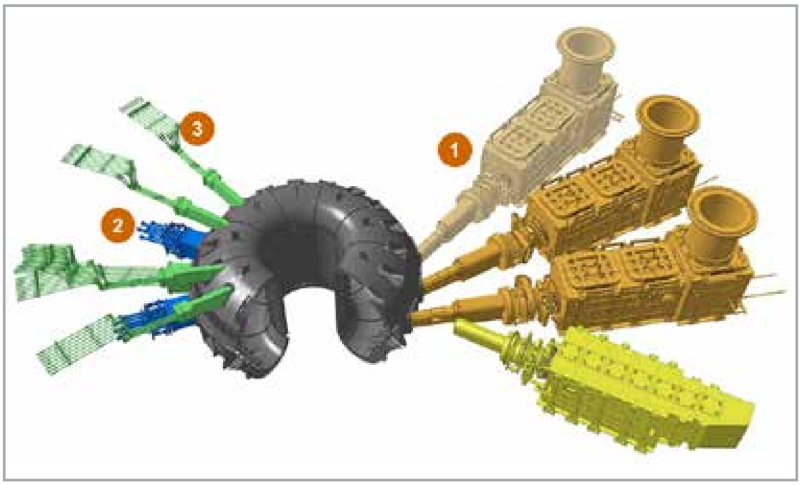



*Fig. 1 – Plasma heating external systems: 1) Neutral Beam Injection; 2) Ion Cyclotron Resonance Heating; 3) Electron Cyclotron Resonance Heating [Credit ©ITER Organization, http://www.iter.org/]*

Despite the simplifying hypothesis of uniform and isotropic thermal radiation from the plasma, the exposed surfaces are exposed to a heat flux that is distributed in a highly uneven manner, since it depends locally on the surface orientations, their distance from the radiation source, and on the presence of intermediate shielding elements. This type of radiation problem can be reduced to the calculation of the so-called view factors (VF) from the elemental source surfaces to the elemental target surfaces and requires appropriate numerical methods, such as the Radiosity Solution method (and particularly the embedded Hemicube Method) available in Ansys Mechanical APDL, which takes a discretization into shell elements of the entire set of source and target surfaces as input.

The VF calculation, using the Ansys method above, is a computationally intensive task, the cost of which is approximately proportionate to the square of the total number N of shell elements in the model. In fact, a VF factor is calculated for each shell element relative to every other shell element using a robust "brute force" numerical approach.

However, only a small subset of the N(N-1) VFs calculated is usually necessary (i.e., only the VFs from the source shell elements to the target shell elements) and extracting those VFs from the entire set requires additional post-processing steps that can be quite expensive in computational terms if executed by built-in APDL routines.

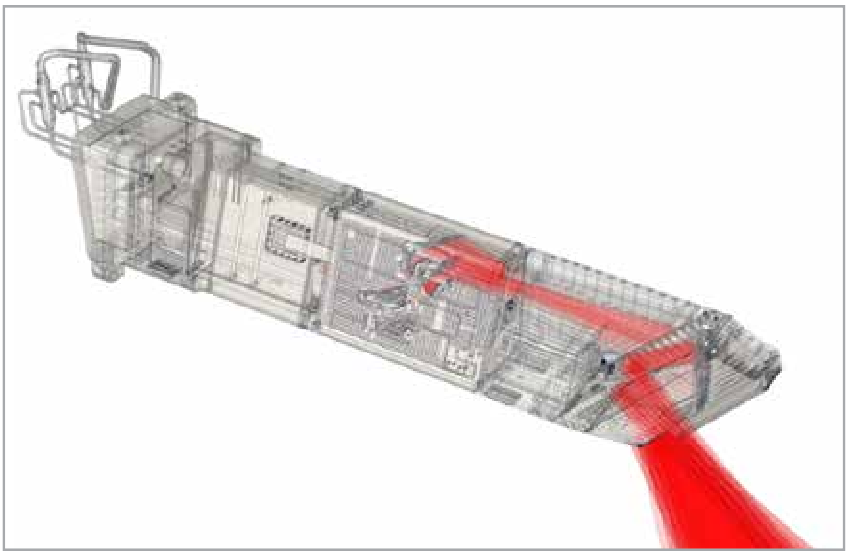


*Fig. 3 - One of the four EC H&CD upper launchers [Credit ©ITER Organization, http://www.iter.org/]*

NINE has developed its own improved methodology that combines Ansys Radiosity Method for robust VF calculation with ad-hoc Python routines to extract the relevant VFs quickly and efficiently (i.e. with a reduction of processing time by two orders of magnitude).

Once the selected VFs are available, you can obtain the heat flux distribution for the launcher's plasma-facing surfaces and use it as a boundary condition for subsequent heat transfer and structural assessment analyses.

### CFD conjugate heat transfer analysis of the port plug cooling system

Another key problem for the structural evaluation was to determine the temperature distribution over the launcher's entire structure, which then had to be transferred to the finite element analysts as input for the thermal-structural assessment.

The geometrical configuration of the launcher's cooling system is extraordinarily complex. The physical quantities that characterize the structure-coolant heat exchange locally (such as heat fluxes and heat transfer coefficients) are spread over very wide ranges, and thus it is not possible to resort to classical engineering correlations without introducing very large inaccuracies into the analysis. Instead, a state-of-the-art detailed analysis approach is necessary, taking advantage of the multi-physics modelling capabilities of CFD codes.

Specifically, a CFD conjugate heat transfer (CHT) analysis is needed, in which the thermal-fluid-dynamic problem (i.e., the solution of the mass, momentum and heat balance equations over the fluid domains that represent the cooling system) is numerically coupled to the problem of the heat conduction through the cooled structures. On the other hand, the use of a coarser two-step approach, involving segregated fluid and solid problems to be solved separately, would require the thermal boundary conditions to be defined (in terms of temperature, or heat flux, or heat transfer coefficient) respectively at the interfaces with solid and fluid domains; in complex cases it can be hard, if not impossible, to obtain reasonably accurate estimates of those boundary conditions, and an analyst may be tempted to make overly simplifying assumptions. In a CFD-CHT analysis those interfaces are handled in an implicit manner (by appropriate continuity and energy conservation conditions) and the distributions of temperature and heat flux are outcomes of the simulation rather than an arbitrary input provided by the user.

## International R&D for Carbon-free Energy: the ITER experiment

The International Thermonuclear Experimental Reactor (ITER) is the world's largest fusion experiment. Thirty-five nations are collaborating to build and operate the ITER Tokamak, a magnetic fusion device that has been designed to prove the feasibility of fusion as a large-scale, carbon-free source of energy that uses the same principle that powers our Sun and stars. The ITER reactor has been designed to produce 500MW of output power per 50MW of input.

Just a few meters separate the 4 Kelvin degrees of the cooling liquid in the superconductive coils and the 150-million degrees of the plasma where the fusion reaction takes place. Set between these two extremes, the reactor's metallic surface is exposed to a heating flux comparable to the Sun's: simulation, therefore, plays a key role in the design of the reactor. The ITER Organization and Fusion4Energy, the EU Joint Undertaking for ITER, have committed themselves and through the EnginSoft subcontract, to perform finite element method (FEM) calculations dedicated to different functionalities ranging from the electro-mechanical and magneto-static analysis of the ITER reactor blanket module to the simulation of the welding of the Toroidal Field Coil Cases.

## About NINE

NINE – Nuclear and Industrial Engineering – is an engineering company founded in 2011 and based in Lucca, Italy that provides analysis, design, and consulting services mostly in the area of nuclear installations, and to some extent also in the non-nuclear industry, always with a special focus on safety and often within highly-regulated frameworks (such as for nuclear licensing purposes).

NINE's main areas of expertise include: nuclear reactor physics; nuclear reactor system thermal hydraulics; computational fluid dynamics; structural mechanics; uncertainty analysis; fuel behavior; containment system behavior; severe accidents; waste management and radiation protection; probabilistic safety assessment; and so on. NINE's technical staff currently consists mostly of nuclear engineers with a PhD in nuclear and industrial safety. Much of NINE's work involves the use of different types of software to simulate, within a best-estimate multi-physics approach, the behavior of a plant (e.g., a nuclear power station) during various operational scenarios, ranging from normal operation to beyond design-basis accidents. Some of these simulation tools are specifically designed for nuclear applications (e.g., system thermal hydraulic codes, or reactor physics codes, or nuclear fuel performance codes, etc.), while others are general-purpose commercial tools, such as Ansys software for structural and fluid analysis. This expertise can be broadly extended from the domain of nuclear fission to the domain of nuclear fusion, as well as to non-nuclear industrial applications, while keeping the same level of attention to Verification and Validation (V&V), and Quality Assurance.

For further information, please visit NINE's website (https://www.nineeng.com/) or contact Marco Cherubini (m.cherubini@nineeng.com).

Case studies can be downloaded from:
https://www.nineeng.com/images/Documents/NINE_CaseStudies.pdf.

The first major step in the CFD-CHT analysis of the launcher was the preparation of the 3D geometrical models of the various necessary solid and fluid computational domains. The source information is constituted by the official ITER 3D CAD models, from which the relevant parts need to be exported (see, for instance, Fig. 3).

As the exported geometry was incredibly detailed and not directly usable for CFD analysis purposes, it was necessary to accurately defeature and simplify all the irrelevant details and to correct several defects. Moreover, the available CAD geometry obviously includes solid parts only, so the fluid volumes had to be "extracted" from them. All these geometry manipulation tasks were efficiently performed by Ansys SpaceClaim, within an Ansys Workbench project.

The outcome consisted of several volumes representing selected "modules" of the launcher's structure and the various sections of the cooling circuits. Those volumes are shown in Fig. 4.

The cyan-colored ones correspond to the cooling circuit for the launcher's structures and consist mostly of channels drilled through the metal bodies and shells that constitute the launcher, which form an intricate network of flow paths.

The volumes in yellow and green correspond to the cooling circuits for the optical components: they must be included in the analysis because, in addition to removing heat from the optical components, they also provide supplementary cooling to certain parts of the launcher's structure.

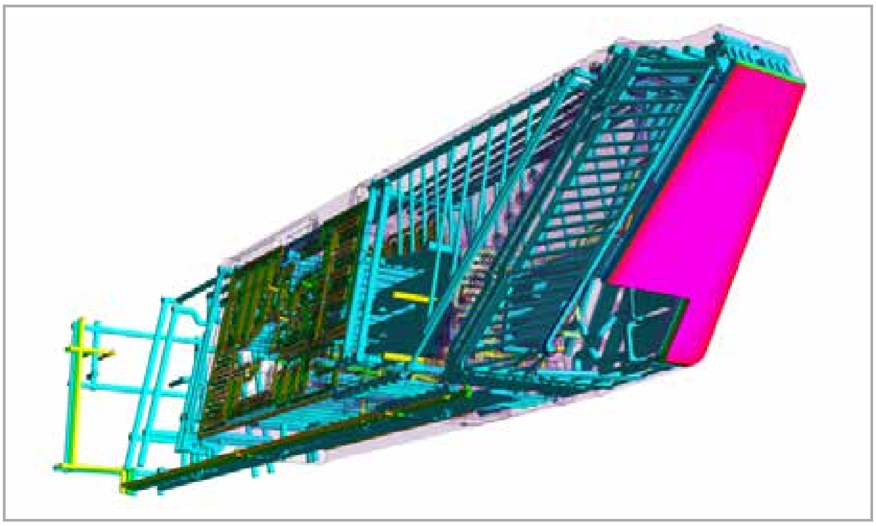


*Fig. 4 - The computational domain of the CHT CFD simulations: fluid domains in cyan, yellow and green; solid domain in transparent pink (3 modules of the launcher port plug structure). Heat flux from plasma is contour-plotted on the front end surfaces.*

The volumes depicted in transparent pink represent part of the launcher's solid structures, in particular those closest to the plasma (whereas the outermost ones do not play a significant role in the thermal problem and need not be considered in the analysis).

The next key step consisted of the generation of computational grids (or "meshes") for the above volumes with Ansys Meshing. Multiple versions were developed in some cases to allow grid sensitivity studies. The "production" grids, selected on the basis of a balance of accuracy and computational costs, were assembled into a global computational model that counted some tens of millions of nodes and approximately 100 million cells.

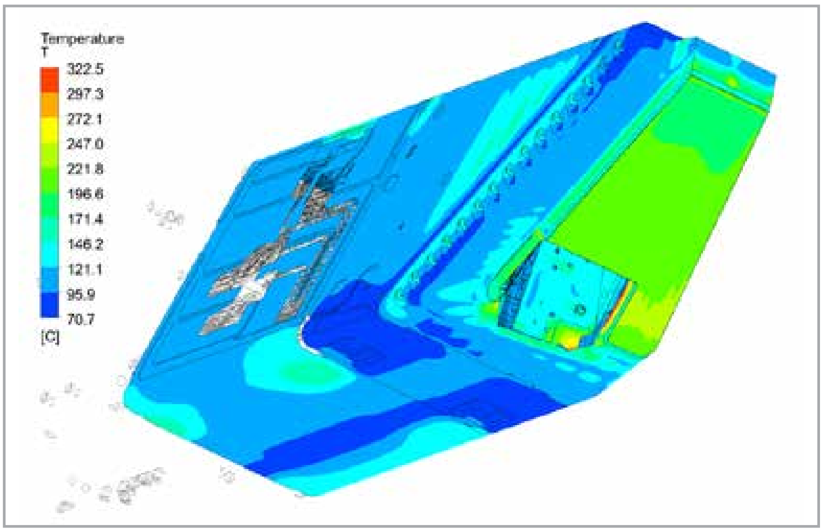



*Fig. 5 - Results of the CHT CFD simulation of the EC launcher port plug: temperature distribution over external surfaces*

CFD-CHT test and production calculations were then setup in Ansys CFX. Thermal-hydraulic boundary conditions for coolant pressure, flow rates and inlet temperatures were obtained from interface information specified by ITER and F4E and from the results of other in-house analyses.

Thermal boundary conditions for the solid domains consisted of distributions of plasma heat flux (as obtained from the VF calculation task described previously) and of heat flux due to power dissipation from the electromagnetic beams (referred to as stray radiation).

Spatial distributions of volumetric power sources were applied over all domains, to account for the energy deposition from neutron and gamma radiation; those distributions, provided by F4E in the form of "point clouds", were imported into user-defined functions. The Shear Stress Transport (SST) model was used to treat the turbulence. IAPWS-IF97 formulation of water properties, available in CFX libraries, was used.

For the solid materials in the model (stainless steel AISI 316 LN ITER Grade, and CuCrZr alloy) temperature-dependent thermo-physical properties were imported from reference tables provided by F4E. Appropriate thermal contact conductance was applied to those solid-solid interfaces where an imperfect thermal contact takes place (e.g., at bolted connections).

Although the so-called "plasma operation" of the ITER reactor is characterized by a pulsating transient behavior, with the full power conditions being maintained only for a fraction of time in a cycle (e.g., for 600 s of an 1800 s period), the main simulations were performed as steady-state, with stationary full-power thermal loads, thus providing conservative results while maintaining acceptable computational costs.

The steady-state simulations were performed to achieve the best convergence level allowed by the available meshes and the numerical setup used, i.e., with a sufficient number of iterations to minimize the residuals and the imbalances and to stabilize the locally monitored quantities.

In particular, the thermal balances over each domain were carefully checked to verify the correct application of boundary conditions and source terms.

The target results of the main simulation are the 3D distribution of temperature over all fluid and solid domains (Fig. 5), and the distribution of heat flux over all domain interfaces (both fluid-solid and solid-solid). This information was then exported for use as input for the Ansys thermo-structural models by other analysts in the work team.

Further CFD simulations were also performed to estimate the concentrated and distributed pressure losses through the cooling system, thus providing useful quantitative information to support the design and to ensure the fulfilment of the project requirements and interface specifications (such as those for total available coolant flow rate and maximum allowed total pressure loss).

## Conclusion

While the scope of these activities is very limited compared to the size, complexity and technological challenges of the whole ITER project, which relies on the collaboration of thousands of engineers and scientists from many countries all over the world, NINE has been able – and continues – to contribute effectively to the advancement of the design of a critical system, i.e. the cooling system for devices that are intended to inject very powerful electromagnetic beams into the plasma to heat it up and help keep it in a stable configuration, in order to reach the conditions for the nuclear fusion reaction to take place.

This is thanks to the skills and experience of NINE's engineers in the area of nuclear safety and design, the powerful simulation tools available in the Ansys suite, the effective collaboration created with the other partners (NIER SpA and Politecnico di Milano), and to the valuable support offered by EnginSoft.

*The work leading to this publication has been funded by Fusion for Energy under Framework Contract F4E-OFC-0800.*



For more information:
Fabio Moretti - NINE - Nuclear and Industrial Engineering
f.moretti@nineeng.com


By Enrico Gnata[1], Emiliano D'Alessandro[2], Giovanni Falcitelli[2], Enrico Boesso[2], Fabiano Maggio[2], Davide Girardi[2]
1. Bticino - 2. EnginSoft

# Simulating the multiphysics of a residual current device

## Ensuring the safety of users around electrical contacts

*Residual current devices (RCDs) are arguably the most important electrical safety devices for human beings in both residential and industrial settings. Where circuit breakers function to protect electronic devices and electrical systems from electrical overload, the residual current device is designed to cut the flow of electricity to your circuit when it detects that the flow of electricity is going to travel down an unwanted path. The speed at which the RCD cuts the electrical flow varies from 40 to 300 milliseconds after the detection of a fault. They are not failproof, however, and many factors influence the speed of response. This article examines the simulation of an RCD to determine the factors that affect its response times.*

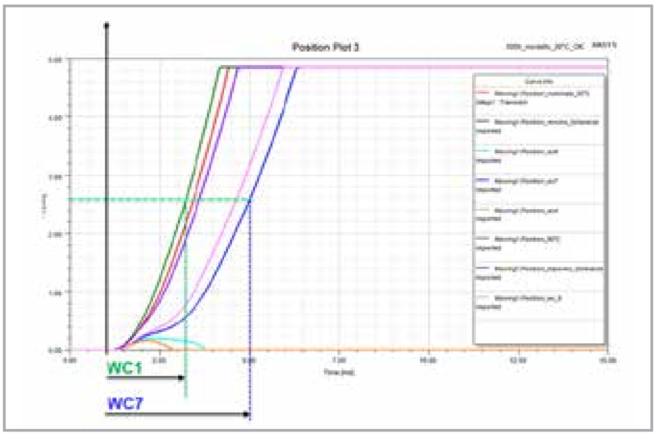



Fig. 2 – Time-Motion law of the Plunger (magnetic gap)

In both the civil and industrial fields, safety is non-negotiable in installations, but how exactly are we protected from electrical contacts? When we receive an electric shock, a portion of the current disperses from the system to the ground, passing through our bodies. This portion of the current, called "differential", is detected by appropriate systems (electrical, electronic) that trigger, by means of residual current devices (RCD), the breakers that power the system. Generally, an RCD is composed of a detection unit (electric, electronic) and an actuator (trip) that triggers the mechanical disconnection or "trip" of the power breaker.

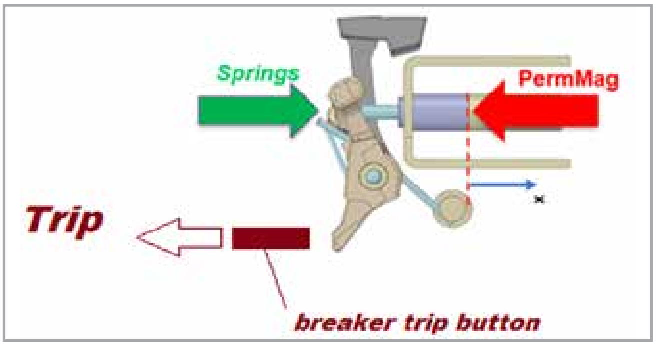



Fig. 1 – The schematic of an RCD trip switch

An RCD-trip switch (shown in the figure) activates the release button of the circuit breaker by means of a switch-lever that is operated by a loaded flexural spring held in static equilibrium by a permanent magnetic circuit. Once the trip is triggered, an electrical discharge from the capacitor powers a solenoid that depolarizes the magnetic circuit, allowing the switch-lever to move. Since it is equally governed by electromagnetic and mechanical phenomena, an RCD-trip is considered to be a Multi-physical system. The aim

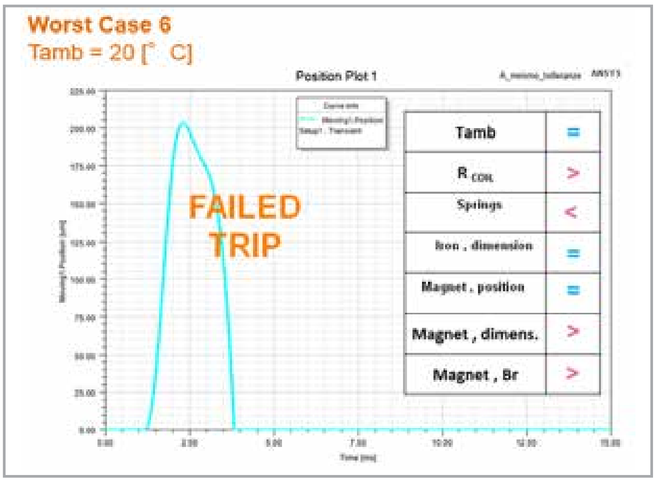



*Fig. 3 – Failed trip (motion inversion after capacitor discharge)*

## About BTicino

Founded in 1936 and an established leader in the field of Civil Electrical Breakers, Bticino today is an accredited part of the Legrand Group, where it constantly aims at developing solutions for distribution, safety, communication and control in the world of low voltage electricity.

For further information, visit: www.bticino.it

of this article is to describe how such a switch can be simulated. A preliminary analysis was conducted with Cetol6σ in order to identify the construction factors of greatest functional influence and to generate a scale of sensitivity for the geometric and dimensional elements on the drawings. The RCD's kinematic nature is clearly a 1 degree-of-failure (DoF) system, which was studied with RecurDyn based on rigid-body modeling. The aim of this part of the study was to obtain the laws of total force (elastic-friction) and of equivalent mass, both of which were transposed to the translational free coordinate of the plunger, which coincided with the magnetic gap used in the final studies with Ansys Maxwell.

Ansys Maxwell simulation software for electromagnetic fields is used to design and analyze 2- and 3-D electromagnetic and electromechanical devices, including motors, actuators, transformers, sensors and coils. It uses the finite element method to solve static, frequency-domain, and time-varying electromagnetic and electric fields. One of its key benefits is its automated solution process, which only requires the user to specify the geometry, the material properties and the desired output, after which Maxwell automatically generates an appropriate, efficient and accurate mesh for solving the problem.

The analyses with Maxwell resolved the coupled equations between the dynamic and electromagnetic fields, which enabled the governing laws in the coil, the magnetic field and the induction in space, position and speed of the plunger to be obtained.

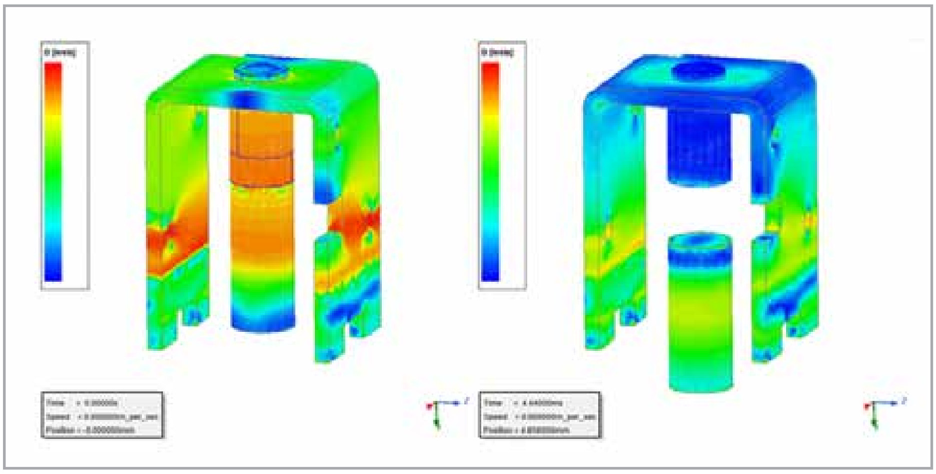


*Fig. 4 – The distribution of the induction field in the device's magnetic circuit. The field is depicted just before and immediately after the plunger motion.*


There are considerable differences in the RCD's performance depending on the set of physical-geometric factors permuted according to the possible worst-case scenarios. We note that the RCD's response time can double depending on the specific worst case, just as it can even fail if the upper limits of dimensional and flowmetric tolerance for iron and magnets are not contained. This result is in accordance with what can be observed from the working point on curves B_H, which highlight the "saturation" of the magnetic circuit (and consequently a marked increase in the forces antagonistic to the motion, in proportion to the iron and magnets).

For more information:
Emiliano D'Alessandro - EnginSoft
e.dalessandro@enginsoft.com


**By Andrea Morello, Marco Barbi**
CNH Industrial

# Development and optimization of crash brackets for ECE R29 regulation compliance

## IVECO uses modeFRONTIER to simulate results to pass type-approval tests

*This paper presents the main steps taken in the development phase of the IVECO cab suspension brackets to comply with the new ECE R29 crash regulation for Heavy Commercial Vehicles (HVC). In this study, modeFRONTIER drives and guides different CAE software (Hypermesh, Radioss, Nastran and MATLAB) to fulfil all the requirements for the component under examination. The IVECO cab suspension brackets are actually designed to deform considerably during the ECE R29 tests, while still being strong enough to survive the IVECO fatigue mission without any failures. These are two opposing goals: a typical challenge for optimization software like modeFRONTIER.*

### The ECE R29 requirements

Regulation ECE R29.03, which came into force in January 2011, applies to commercial vehicles with a separate, category N driver's cabin. It prescribes the requirements for the protection of the occupants of the cabin of the vehicle in a head-on collision or in the event of overturning. This is a TYPE-APPROVAL test: you cannot sell the vehicle if it does not comply with this regulation!

ECE R29.03 comprises three different tests after each of which the cabin must exhibit a survival space that allows the accommodation of a mannequin at the 50th percentile. Further requirements are that the doors must not open during the tests, and the cabin must remain attached to the chassis.

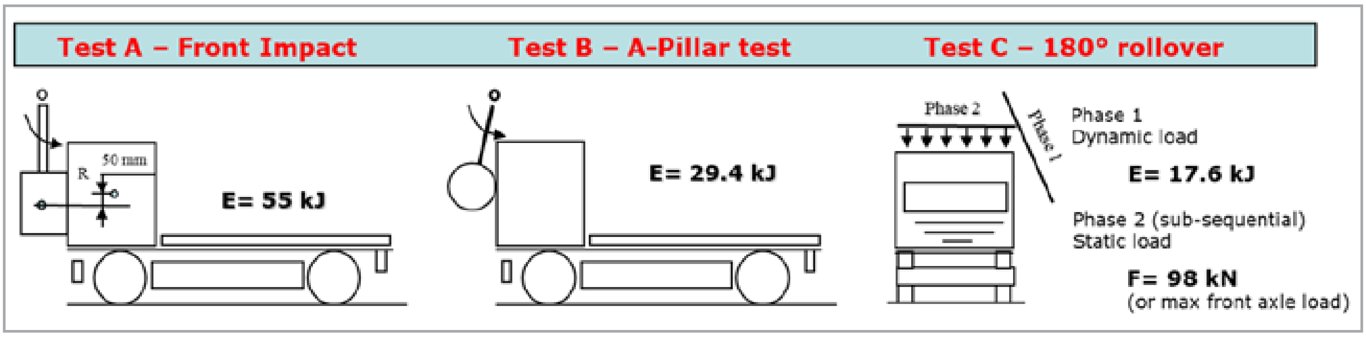



*Fig. 1 – ECE R29 regulation requirements*

Test A is a frontal impact test intended to evaluate the resistance of the cabin in a head-on collision. It was already used in the previous version of ECE R29, but the kinetic energy involved has now been increased by 25%. A rectangular impactor (0.8x2.5m) with a mass greater than 1,500kg impacts the front wall of the cabin with an initial kinetic energy of 55kJ.

Test B describes an impact to the A-pillar of the cab and is intended to evaluate the resistance of the cab in the event of overturning by 90° resulting in a subsequent impact, for example against a tree, a pole, or a wall. A cylindrical impactor (0.6x2.5m) with a mass greater than 1,000kg impacts on the centerline of the windshield with an initial kinetic energy of 29.4kJ.

Test C denotes a strength test of the cab roof intended to evaluate the resistance of the cabin in the event of overturning by 180°. It is composed of two sequential sub tests:

- A dynamic pre-test: a flat impactor, wider than the cab, and with a mass greater than 1,500kg impacts on the side of the cabin at an angle of 20° and with an initial kinetic energy of 17.9kJ.
- A static test: the roof is crushed by a rigid plane with a minimum weight of either 10 tons or the maximum load on the front axle.

## The IVECO ECE R29 simulation

IVECO performs the ECE R29.03 evaluation using virtual simulation: the FE model used is quite complex because all the details of the cab are necessary – structure, trim, suspension and so on.

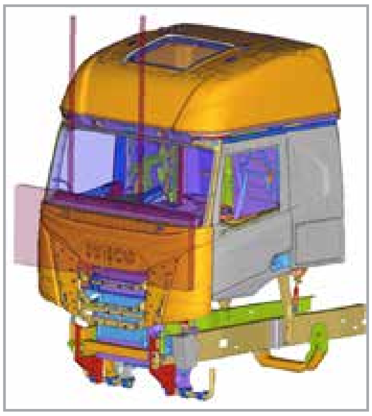


*Fig. 2 – Radioss full model*

This results in very long calculation times: about 24 hours using 48 CPUs which, of course, is not suitable for optimization tasks during the concept design phase.

During the evaluation of ECE R29.03 Test A, we noticed that the kinematics of the front suspension could help us to reach the goal: in detail, the blue bracket (Fig. 3) pushes back the green interface and, consequently, the red bracket. We observed that if the red bracket were to collapse we would be able to easily reach the objective.

We therefore designed a small model of the red bracket and its kinematics to study its collapse using a simulation of a few minutes: as a result, we obtained a force-displacement graph that helped us to identify the collapse and the level of force required to achieve it.

## The IVECO strength and durability simulation

As shown, the cabin suspension bracket should deform during the ECE R29 type-approval tests, but at the same time, it needs to be rigid and sturdy enough to survive the vehicle's fatigue mission on the road without failure due to customer use and misuse.

IVECO manages the durability test both physically and virtually; the virtual approach is time-consuming due to the tuning of the fine mesh, which has a real influence both on durability impairment and on the length of the simulation time.

During the development phase, a smarter and quicker approach was used in order to reduce response time. The bracket was evaluated using:

1. Standard static gravitational load using finite element analyses generally used for initial and preliminary dimensioning
2. Load cases obtained from the whole load time history used for durability analyses by applying some dedicated statistical analyses

Using an envelope stress map for post-processing all the load cases enables you to detect and highlight the areas of failure in the fatigue test (see Fig. 4). On a standard workstation, execution takes a few minutes instead of a few hours using the traditional procedure for durability calculations.

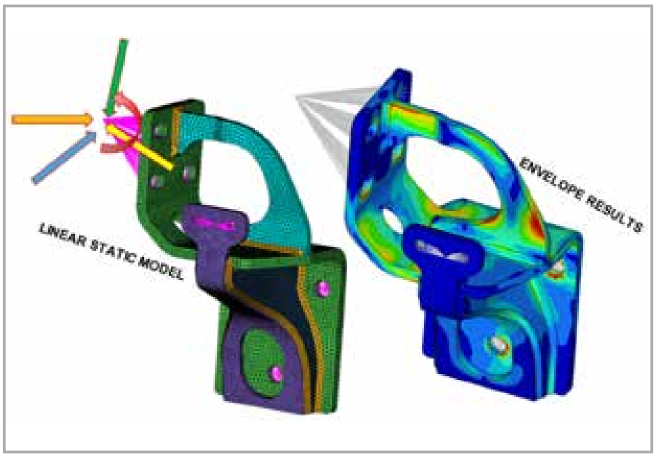



*Fig. 4 – Linear static analyses, envelope stress map*

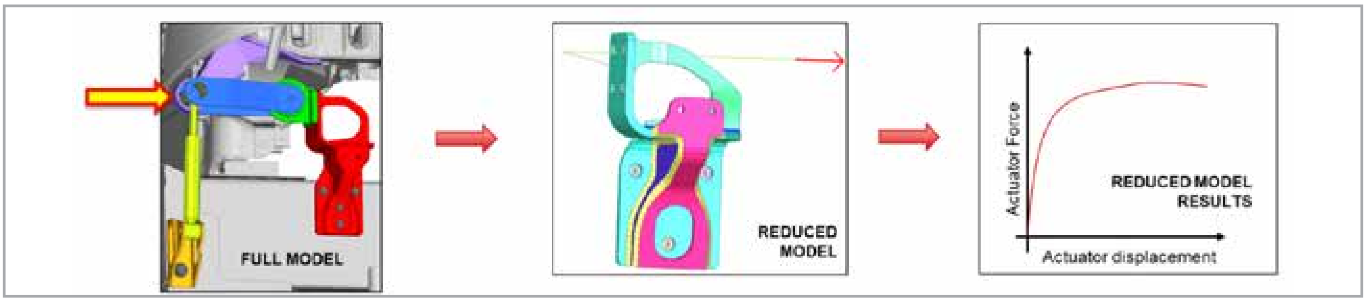



*Fig. 3 – Radioss small FE model – cabin suspension bracket*

### FE input model for modeFRONTIER

The two FE models (Radioss and Nastran) are generated from a single Hypermesh model. Mesh morphing was used in this activity: by working on the handle nodes, it is possible to stretch, deform, and enlarge the parts thus modifying the shape and geometry of the structure. The morphing is applied simultaneously to the two different FE models and then Hypermesh exports the input file for each solver (see Fig. 5).

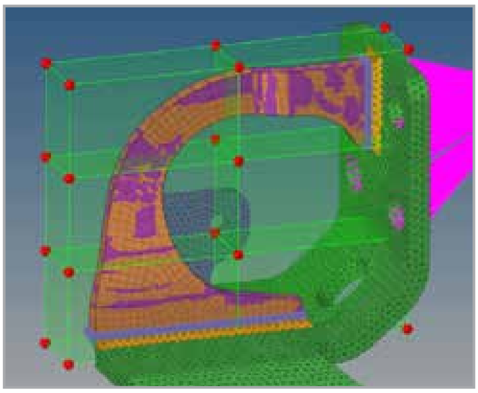


*Fig. 5 – Morphing volume on the part to modify it*

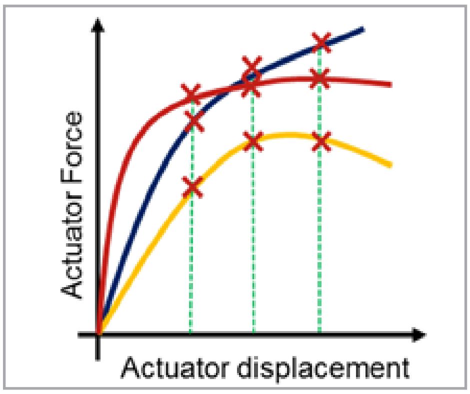
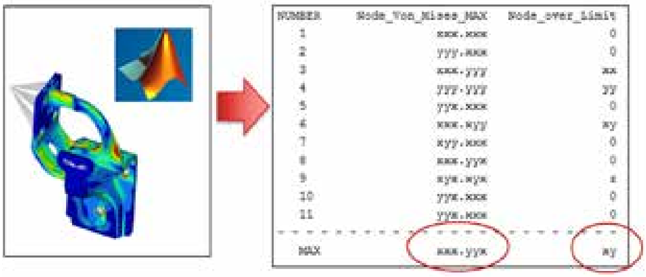



*Fig. 7 – Radioss output: force vs. displacement*

### Output results for modeFRONTIER

The outputs from the simulations are as follows:

- linear static simulation: Nastran writes the output data, like stress value, displacement and so on, into a text file with an f06 extension. A MATLAB script has been written to read the Nastran f06 output file and summarize all the information into a simple table that shows, for each load case, the maximum stress levels on the parts, and the number of nodes that exceed the stress limit. The extension area is used to understand if an elevated stress level is mainly due to a local peak as a consequence of deformed elements caused by the morphing tool (see Fig. 6).



| NUMBER | Node_Von_Mises_MAX | Node_over_Limit |
|---|---|---|
| 1 | xxx.xxx | 0 |
| 2 | yyy.xxx | 0 |
| 3 | xxx.yyy | xx |
| 4 | yyy.yyy | yy |
| 5 | yyx.xxx | 0 |
| 6 | xxx.xyy | xy |
| 7 | xyy.xxx | 0 |
| 8 | xxx.yyx | 0 |
| 9 | xyx.xyx | x |
| 10 | yyx.xxx | 0 |
| 11 | yyx.xxx | 0 |
| MAX | xxx.yyx | xy |

*Fig. 6 – MATLAB elaboration from the .f06 Nastran file*

- crash analysis: Radioss creates some output text files that report the time, actuator force and actuator displacement. This information can be used to generate the graph illustrating deformation with respect to force, as shown in Fig. 7 (three different designs have been shown in the examples). We use three different points of this graph in modeFRONTIER to identify the maximum force level.

### modeFRONTIER flow

The input parameters are the coordinates of the morphing nodes. A Hypermesh macro file collects all the inputs, then opens the model, morphs the parts, and writes the input files for the two software programs: Nastran and Radioss. Using the sincro node (see Fig. 8), you can run the two simulations in parallel: this reduces the total time for each design to the longer simulation time (and not to the sum of the two simulation times, as in the case of sequential execution). The simulation results are identified using the methods explained previously.

In order to achieve a better design than the original, the constraints for the Radioss crash force and the Nastran stress levels must be lower than for the initial design.

The optimization was performed in modeFRONTIER by moving the handle morphing nodes and requesting the reduction of the:

- Maximum Von Mises stress
- Maximum force in crash load
- Number of nodes exceeding target stress limit

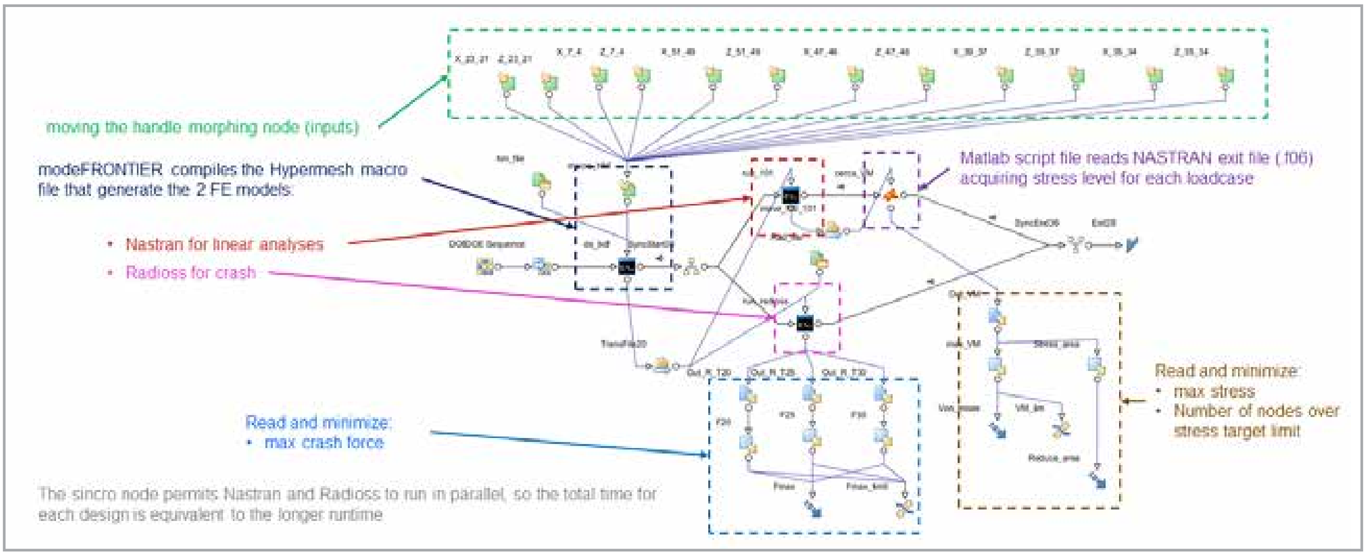



*Fig. 8 – modeFRONTIER flow*

Different solution methods were used during the optimization phase, principally multi-objective genetic algorithm 2 (MOGA2), multi-objective game theory (MOGT) algorithm and fast algorithm for the scenario technique (FAST).

## RESULTS

The following graph in Fig. 9 shows a scatter plot reporting the:

- Crash force to collapse the bracket (X axis)
- Maximum Von Mises stress on the component (Y axis)
- Number of nodes exceeding the limit (bullet diameter)

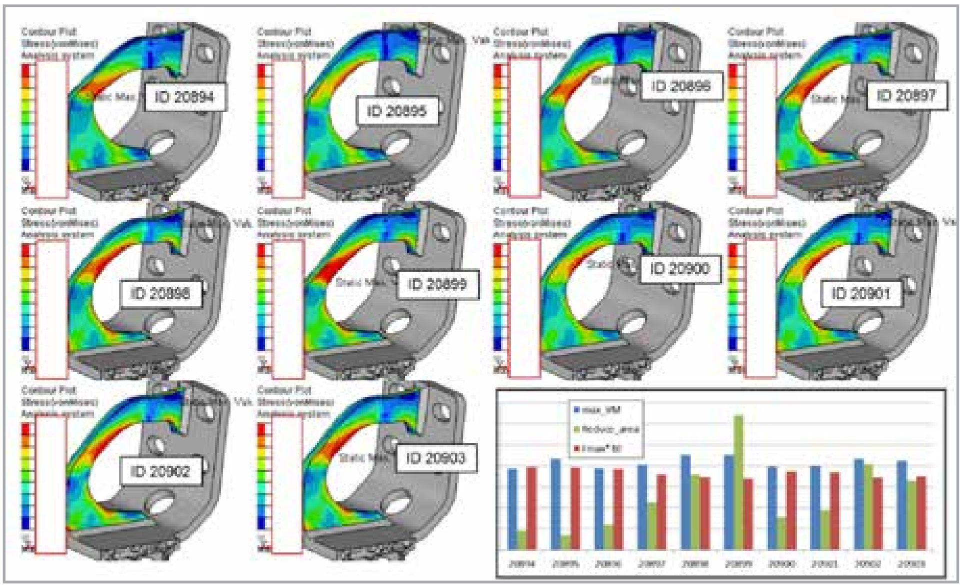



*Fig. 11 – Sample design on the Pareto frontier*

The constraint limits are indicated in the graph so that only the grey points are feasible.

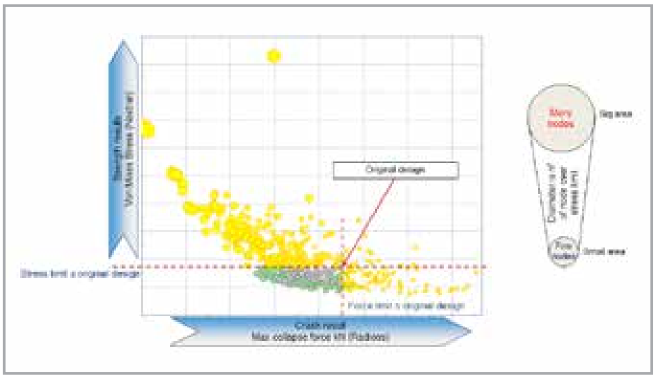


*Fig. 9 – Scatter plot of the results*

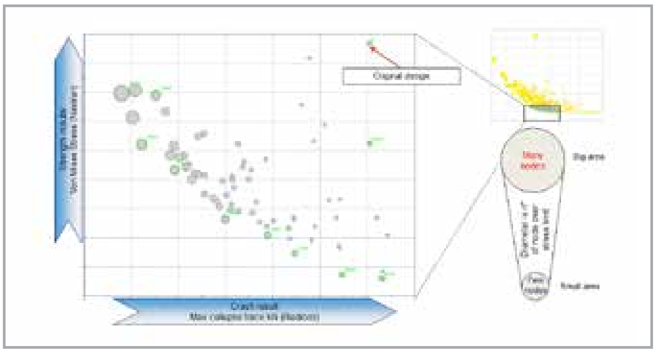


*Fig. 10 – Pareto designs*

At this point it is possible to identify the Pareto frontier (see Fig. 10) where some designs have been selected to see the shape and stress results obtained (see Fig. 11).

Since the original design already had an acceptable level of strength to achieve positive results in the crash type-approval testing, we decided to reduce the stress level as much as possible for more reliable durability results while maintaining the crash force level.

The final design used is displayed in the following photograph (see Fig. 12) showing the component before and after the successful type-approval test. It is possible to see that the simulation results match closely with the real deformation.

The durability test was also performed without creeks and breakage, so that the vehicle is now on the road.

A similar approach was also taken for the other suspension brackets due to the different layouts, different cabins, left/right side attachments, number of axles, vehicle typology (on-road, off-road), etc. all of which require a different bracket geometry. modeFRONTIER enabled us to automatically investigate a huge number of simulations, evaluating thousands of different shapes in just a few weeks. It is useful in the investigation and optimization of designs.

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

*Fig. 12 – Final design (bracket on vehicle)*

Fig. 1 - Prototype with five cable-driven joints made of 3D-printed fused deposition modelling (FDM) polycarbonate

# Optimizing the cable routing for a hyper-redundant inspection robot for harsh, hazardous environments

## A multi-objective optimization framework to reduce the actuation loads and increase the payload

**By Paolo Guardiani, Daniele Ludovico, Alessandro Pistone e Carlo Canali**
Istituto Italiano di Tecnologia (IIT)

*Robotic inspection systems are increasingly being used in both civil and industrial infrastructure or facilities to inspect hard- or dangerous-to-reach critical places. Cable-driven hyper-redundant robots, which take their design inspiration and functionality from the movement of snakes, are highly-articulated and hence very flexible robotic structures attached to a fixed base that are actuated and manipulated by intricate systems of cables and joints. This article discusses a multi-objective optimization study to determine the optimal matrix for the routing of the actuating cable system in order to minimize the cable load on the robot and maximize the robot's payload.*

The aging of infrastructure is an emerging threat in both civil and industrial environments. In the meantime, infrastructure or facilities continue to operate beyond their expected lifetime and the risk of failure increases. Moreover, infrastructure is often not designed to be inspected and this can be a significant problem when critical parts are inaccessible, difficult or dangerous for a human operator to reach. For these reasons, the use of a robotic system can represent a game-changer.

IIT is developing a new robot capable of performing inspection tasks in harsh environments and confined spaces. A first prototype is shown in Fig. 1. Inspired by a snake, it is a cable-driven hyper-redundant robot. The cable-drive makes the structure light, robust and operable via a minimal set of on-board electronics. Moreover, the use of cables is a significant advantage when the manipulator has to operate in hostile environments with high temperatures or chemical hazards, since the motors and most of the electronics can be safely positioned at the base of the robot.

At the same time, cable routing can be very complex and this can have a huge impact on system performance, in fact, the interaction between the cables and the structure can increase the cabling loads considerably, thus decreasing the payload of the robot.

This article presents a multi-objective optimization routine to address the challenges created by operating the cable in a hyper-redundant robot.

The robot under development consists of ten rigid links connected to each other by joints of one degree of freedom. The goal is for the system to be able to reach points up to 5 meters away from the base while transporting cameras and other sensors, depending on the inspection task to be performed.

## The main challenge of this study

The cables pass through the robot along a precise path and their interaction with the structure generates loads that affect the robot itself. These loads vary based on the robot's payload and the articulation angles during movement. Different cable routes change the loads on the cables significantly, so it is possible to investigate the behavior of the forces by varying the path followed by each cable.

Each joint is driven by two antagonist cables anchored at points A and B, as shown in Fig. 2. There are 20 ports on the joint, arranged in a circumference, through which the cables can pass. The cable routing matrix *T* describes the ports through which the cables pass. *T* is an upper triangular matrix and any element $t_{i,j}$ represents the port on the joint *i* through which the cable that guides joint *j* passes. The number of free variables can be reduced from 55 to 36 by considering the following design choices:

- $t_{i,i}=5$: the anchor points on the joint are aligned with port 5;
- $t_{i,i+1}=0$ or 10: the cables operating joint $i+1$ do not change direction in module *i*.

In addition, the elements of the routing matrix are subject to some constraints:

- $t_{i,j} \neq t_{i,k}$, where $i<j<n_{joint}$ and $j \neq k$. This means that within joint *i*, cable *j* cannot pass through the same port as cable *k*.
- $t_{i,j}) \neq t_{i,k}+10$, where $i<j<n_{joint}$ and $j \neq k$. Cable *k* has an antagonist cable that passes in the opposite direction inside the joint, so that port is already assigned.

Furthermore, the routings that satisfy these constraints are not always feasible because cables may intersect along the structure of the robot.
The goal of this study is to find the routing matrix that minimizes the maximum forces of the actuation cables.

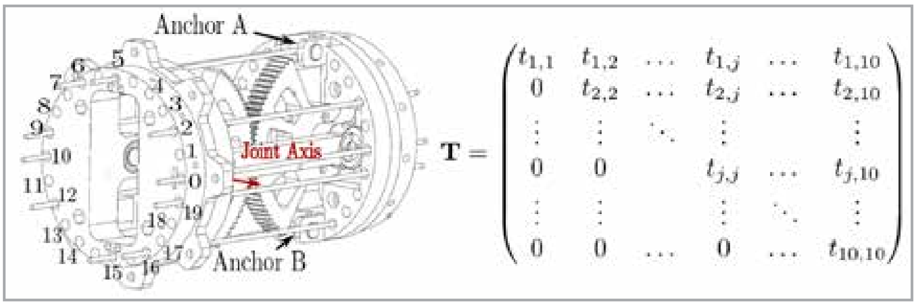



*Fig. 2 – Left: Cable routing inside a single joint of the robot. Right: Matrix representing the routing of a robot consisting of ten joints.*

## Optimization model

### *Problem formulation*

A multi-objective optimization problem was defined to minimize the maximum forces of the actuation cables. The model was developed in modeFRONTIER, which is an integration platform for process automation, optimization, and data mining. The optimization variables are a subset of the 36 elements of *T*. This subset is the result of a sensitivity analysis performed to discard the less discriminating parameters.

Both the sensitivity analysis and the optimization problem must compute the maximum cable forces for a given routing configuration.

### *Maximum cable forces of the robot*

The maximum cable forces are the solution of the static equilibrium of the system formulated as an optimization problem.

For each joint, the cable forces that ensure the equilibrium depend on the actuation forces affecting the subsequent joints, and the configuration of the joint, so starting from the last module we were able to derive the actuation forces by solving an optimization problem in which the optimization variables were the angles of the joint, and the objective function was the actuation force.

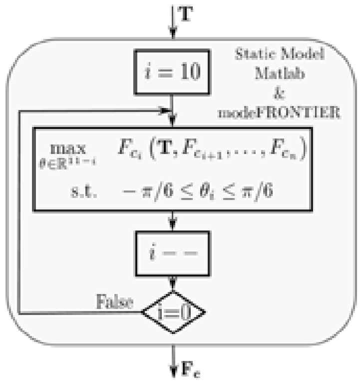



*Fig. 3 – Flowchart for evaluating the force of the static model of the robot*

The flowchart of this optimization problem is shown in Fig. 3 and the parameters used for the optimization are provided in the table below.

| | |
|---|---|
| Algorithm type | SIMPLEX |
| Maximum number of design evaluations | 500 |
| Final termination accuracy | 1e-4 |
| Initial population | ISF |
| Population size | 50 |

### *Sensitivity analysis*

Once the maximum forces on the actuation cables for a given matrix *T* were obtained, a sensitivity analysis was performed to identify the most significant parameters of the routing matrix. To meet all the above constraints, the DOE generation was implemented in MATLAB, and the scatter matrix tool was used to verify that this DOE was able to cover the entire domain of variables.

The main and interaction effects of the 36 *T* parameters on the cable forces were analyzed using SS-ANOVA. In particular, the forces of the first and second joints, which are the most stressed, and the norm of the cable force vector were selected as the performance indicators. Considering all the independent terms of *T*, this method requires a DOE table of 703 elements. This analysis made it possible to reduce the number of the optimization variables from 36 to 17.

### Optimization problem

This optimization problem aims to find the optimal routing matrix to minimize the cable forces on the first and second joints and on the norm of the cable force vector.

As shown in Fig. 4, modeFRONTIER generates the set of 17 variables that satisfy the routing constraints, then MATLAB takes these parameters as input and tries to generate a feasible $T$ matrix by verifying the cable intersection constraint. If this constraint is satisfied, the objective functions are evaluated, otherwise modeFRONTIER generates a new set of input variables.

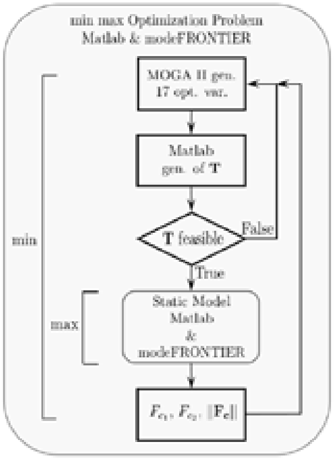



*Fig. 4 – Flowchart of the min max optimization problem*

Since the optimization variables are integers and the problem is multi-objective, the Multi-objective Genetic Algorithm II (MOGA-II) algorithm has been chosen for the optimization strategy using manual settings. The table below lists the solver parameters.

| | |
|---|---|
| Algorithm type | MOGA – Generational Evolution |
| Number of generations | 20 |
| Elitism | Enabled |
| Treat constraints | Penalizing Objectives |
| Reject Input-Unfeasible Designs | On |
| Maximum Number of Rejections | 100 |
| Initial Population | CSP + User Defined DOE |
| Population Size | 191 |

## Results

The optimization routine analyzed 3,820 designs, of which 851 were feasible (~22%). Observing each objective function, it is evident that some designs perform well in one objective, but poorly in the others. Therefore, thresholds were defined for $F_{c1}$ and $F_{c2}$ to select the best designs.

The values of the threshold were chosen by considering the pulling force of the actuators, while the norm of $F_c$ was used to rank the feasible designs.

| | $F_c$ | $F_{c1}$ | $F_{c2}$ |
|---|---|---|---|
| Best 1 | 9231 N | 5446 N | 4059 N |
| Best 2 | 9833 N | 5063 N | 3827 N |
| Worst 1 | 15253 N | 6481 N | 7583 N |
| Worst 2 | 14909 N | 7511 N | 6895 N |

The table above shows the best and the worst routing matrices obtained from the optimization problem. These results, illustrated in the bubble chart below (see Fig. 5), demonstrate that the routing matrix significantly affects the actuation forces.

## About the Istituto Italiano di Tecnologia (IIT)

The Istituto Italiano di Tecnologia (IIT), based in Genoa, Italy is a foundation financed by the State to conduct scientific research in the public interest, for the purpose of technological development. The IIT aims to promote excellence in basic and applied research and to promote the development of the national economy.

As at December 2019, the IIT had produced more than 13000 publications, participated in over 200 European projects and more than 40 European Research Council (ERC) projects, made more than 900 active patent applications, created 22 established start-ups and has more than 40 more under due diligence. It has a network of 4 hubs in Genoa that form its Central Research Laboratories, a further 11 research centres around Italy, and two outstations located in the US at MIT and Harvard.

## Conclusion

This study discusses an optimization method to minimize the load of the actuation cables and maximize the payload of a cable-driven hyper-redundant robot.

This tool can help the designer to evaluate different, non-trivial routing paths. Where an intelligent designer may conceive of a few feasible routing matrices in one day, this approach will generate hundreds of possible solutions while also performing the static force analysis.

***https://youtu.be/HDgVDRqcmTY***

For further information about this project,
contact: carlo.canali@iit.it or paolo.guardiani@iit.it

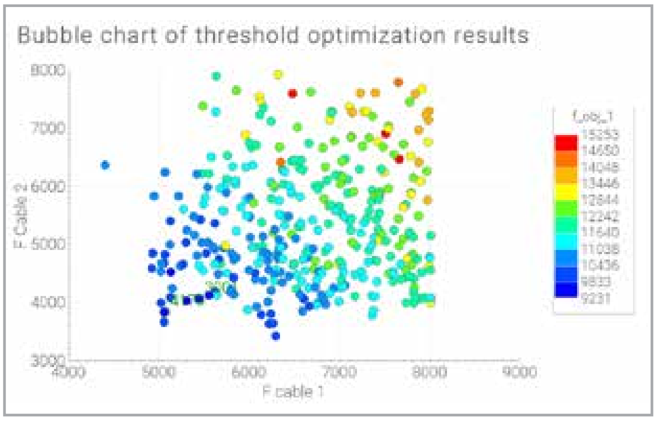



*Fig. 5 – Bubble chart illustrating the results of the optimization problem*

For further information about modeFRONTIER:
Francesco Franchini - EnginSoft
f.franchini@enginsoft.com

November 30 - December 4

2020

36th INTERNATIONAL CAE CONFERENCE AND EXHIBITION

VIRTUAL EVENT

INTERNATIONAL CAE CONFERENCE

AT THE EPICENTRE OF THE DIGITAL TRANSFORMATION OF INDUSTRY

# Online or in-person, the International CAE Conference delivers winning content that participants seek out

## 36th annual event, moved online due to COVID-19, reports high participant engagement

**In existence for almost as long as the science of engineering simulation itself, the annual International CAE Conference and Exhibition held its 36th edition this year. The event went ahead, in contrast to the findings of a survey by the US-based Professional Convention Management Association of its members which found that, by April this year, 87% of companies had already cancelled events, while two-thirds had postponed events, hoping for the COVID-19 tide to turn. On the other hand, many event organizers opted to pivot towards online events, seeking to avoid the losses of investment and time triggered by cancelling and toeing the line with WHO recommendations to reduce in-person contact and avoid large gatherings. The International CAE Conference organizers felt that the continuity of the event and the value of its role for its loyal user base, as well as the conference theme being offered were too important and too timely, respectively, to ignore. The event, therefore, was held online from November 30 to December 4, 2020. The rich and multifaceted program highlighted how engineering simulation is the lynchpin for a successful transformation project when implemented within a balanced and coherent strategy, playing a central and interdependent role among the various industry 4.0 technologies to aid the organization to achieve its business objectives within its specific industry sector. The conference program was offered live in three distinct time slots to enable attendees from around the world to participate in the time zone most convenient to them – from Asia to the Americas.**

## Registered individual participants

| | |
|---|---|
| **1572** | **Exhibition** |
| **1341** | **Conference** |
| **421** | **Research Agorà** |
| **250** | **Poster Award** |

While the live portion has officially ended at this writing, the content will remain available to existing and any interested new registrants until the end of February 2021. There were 1572 individual participants that came to the live portion of the online event, while 1,341 participants attended the conference.

This is in keeping with previous years' numbers for attendance, and amply demonstrates the loyalty that the event has built up over the years as a result of the high quality of the content and knowledge transfer it offers.

Many high-level international executives interviewed in different surveys by various organizations over the course of this year have attested to the fact that they willingly dedicate an hour or more of their time to online conference sessions where they know they will find value (see the article "Who needs live events?" at https://bit.ly/2LKoq29).

The 36th annual International CAE Conference and Exhibition placed its focus on the four key industrial sectors for engineering simulation, namely:

- automotive and transportation,
- aerospace and defense,
- energy and oil&gas, and
- manufacturing

Within these macro categories the event was organized into technical sessions, each of which was opened by a keynote speaker with expertise in that particular sector and who focused on various aspects of the digitalization there, which was followed in each case by more specific and focused technical interventions. In association with the event, many sector associations, such as the International Association for the Engineering Modelling, Analysis and Simulation Community (NAFEMS), the Agency for the Promotion of European Research (APRE) and the Italian Aerospace Research Centre (CIRA, Centro Italiano Ricerche Aerospaziali), independently organized collateral events around specific themes of interest to the audience, such as High Performance Computing, Composite Design, and the EU's massive R&D program, Horizon Europe, to name a few. Exhibitors added to the rich program with content focusing mainly on the software technologies used by engineering professionals in achieving their objectives.

In parallel with the conference, there was a virtual exhibition populated by a substantial selection of heavy weight players in the engineering simulation market. Within their virtual stands, they could make their offers known in a more conventional form by means of live interaction, albeit digitally, as well as by taking advantage of the online platform to provide presentations of application experiences, detailed product demonstrations, explicative videos and more.

Public and private research undertakings and academic institutions were also substantially represented in the online event. Numerous research consortia were able to showcase the progress of their projects and this proved to be of significant interest to the attendees, with over one-quarter of participants (421 participants) visiting the Research Agorà, the area of the CAE Conference and Exhibition that is dedicated to the divulgation of research, and 250 attendees visiting the Poster Awards, which recognizes excellence among university students in the application of computer assisted engineering to solve specific challenges.

The International CAE Conference and Exhibition offers considerable value to industry participants, who are the ones that define the requirements and make use of the specialized technologies and resources to enable their digital transformation for business competitiveness and success. This makes the event a valuable vehicle for technology manufacturers, who wish to make their product functionality, application security, and software usability known to industry users. The findings of researchers concerning simulated physics, analysis algorithms, the properties

## CAE Conference Statistics

| | |
|---|---|
| 3500+ | Visitors |
| 35+ | Events with more than 150 speakers |
| 50+ | Exhibitors |
| 35+ | Poster Award Finalist |

of magnitudes used in simulations, and their correspondence to reality in an increasingly aggregating environment are of abiding interest to engineers. Promoting the resources and support available to the industry to drive and support innovation is another key driver for participation at the event, and it is this that brings the representatives of European research agencies to the event.

Participants were offered video and audio conferencing and a live streaming experience during the live portion of the conference and exhibition. A significant and often cited benefit of virtual events is their time efficacy, with participants being able to select the specific sessions of most interest to them and then interact with the other self-selected participants and with the speakers via the chat. In fact, the ability to select the content to be consumed encourages participants to engage at a higher level in the events and this increases further if the exhibitors or speakers customize that content specifically to the needs of the audience.

## Breakdown of participants by geographic region

| | |
|---|---|
| 44% | ITALY |
| 34% | EUROPE |
| 1,5% | AFRICA |
| 2% | SOUTH AMERICA |
| 4% | USA |
| 13,5% | ASIA |
| 1% | OCEANIA |

Research conducted by the California-based US market research firm, Grand View Research, predicts that the global virtual events market was valued at US$ 77.98 billion in 2019 and expects it to show a compound annual growth rate (CAGR) of 23.2% from 2020 to 2027. The main driver for this growth is identified as being Unified Communication as a Service (UCaaS), which combines communications into a single platform for online events, to allow collaboration between participants and thereby increase productivity. Such has been the success of the online format in the general view of businesses, that most virtual event organizers believe that the future will see a more substantial online component to events, and even the morphing of traditionally in-person events into hybrid events. Almost three-quarters of the PCMA's survey participants believed that combined online and in person events will become the "new normal", with one-quarter stating that they believed that virtual events threaten the future of their in-person counterparts.

An additional benefit to virtual events, for attendees, exhibitors, and speakers, is that the online component increases the attendance by an international audience. US event organizers used to consider a 20% attendance by overseas visitors a boon, but the virtual component has seen that participation explode, by all reports, from companies as diverse as SAP, Red Hat and Adobe (see the article referenced above).

Overall, the International CAE Conference and Exhibition has always provided a number of communication channels to participants by bringing together supply and demand for the available technologies and facilitating B2B meetings between industry professionals, researchers, and trainers.

> The event and all its content will remain navigable until the end of February 2021 and registration for interested parties is still available on demand and free of charge for those people who missed the live event or who wish to access the content that was shared.
>
> Simply visit www.caeconference.com.


# Multiphase fluid dynamics simulation of electric expansion valves for refrigeration and air conditioning applications

## Evaluating the ability of numerical methods to replace experimental testing to calculate the capacities of expansion valves

**By Lorenzo Resmini**
CASTEL SRL

*Standards compliance requires expansion valve producers to supply detailed user manuals that accurately specific the refrigerant capacity of their valves. This article details a study that was undertaken by Castel, a producer of refrigeration and air conditioning components, to compare the results of an experimental method for testing expansion valves with a numerical method using Ansys.*

Expansion valves for refrigeration and air conditioning applications are used to take the refrigerant fluid from its condensation pressure to its evaporation pressure. This pressure drop is achieved by a shutter and an orifice, both of which must be appropriately designed to ensure that the whole system functions correctly, in particular to maintain a constant set-point temperature in the cold room. The other important purpose of this device relates to the superheating parameter which must be regulated and kept constant for the compressor to function correctly.

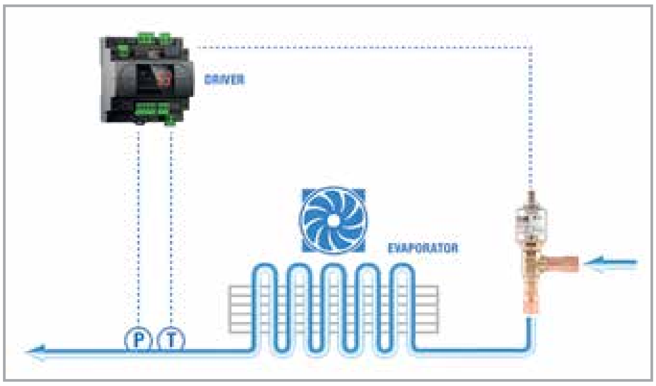



*Fig. 1 – General schematic outline of an EEV*

Electric expansion valves (EEV) are the most reliable and efficient of all expansion valves. Technologically, they are based on a stepper motor that provides precise regulation of the valve shutter position, allowing it to respond accurately to variations in the thermal load in the cold room. They require a driver and two sensors, one to control pressure and one for temperature. (Fig. 1).

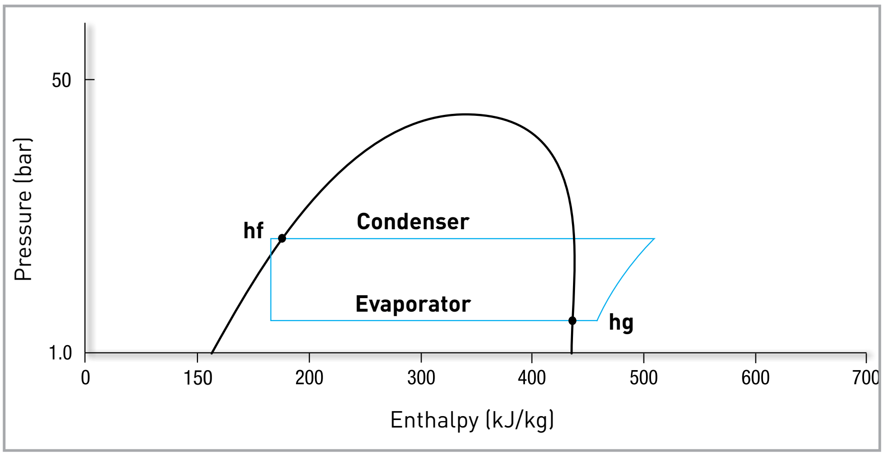



*Fig. 3 – Refrigeration cycle - ph. diagram*

All expansion valves must be supplied with a manual, an important document that must specify their refrigerant capacity. Castel's EEV portfolio includes several valve models that differ mainly in their geometrical dimensions and in the expressed mass flow rates, which depend on the pressure drop through the valve and on the refrigerant fluid, since each refrigerant has a unique capacity to transfer heat.

*Fig. 2 – EEV Models*

The company therefore needed a precise method to obtain the capacity values for all its EEV models in order to compile the EEV manuals so that refrigeration equipment manufacturers could select the most suitable components for their applications.

## Standard

In view of the fact that the EEVs were studied in the company's own R&D department, a considerable challenge lay in how to calculate the capacity of all the models designed. The ASHRAE Standard 17 offers an experimental method for testing expansion valves and provides the measurement system on a testing machine.

The capacity is calculated with a well-known formula:

$\dot{m}$ **(hg – hf)**

$\dot{m}$ = mass flow rate [kg/s]

hg = enthalpy of saturated refrigerant vapor [kJ/kg]

hf = enthalpy of saturated refrigerant liquid [kJ/kg]

It became clear at the outset that in order to implement an Ashrae-compliant experimental test it would be necessary to build a number of testing machines, each equipped with a flowmeter capable of measuring the mass flow rates from the lowest to the highest.

Given the nominal capacity of our valves, some of these testing machines would have been very large in size, requiring a significant investment and considerable time, as well as the need to produce many prototypes, resulting in a delay in commercializing the products.

## Computational fluid dynamics vs the traditional approach

Before Ansys was used, the only way to provide the refrigerant capacity for each valve (except for building the testing machines) was to calculate the flow coefficient (Cv) in combination with

| Shutter angle [deg] | Seat diameter [mm] | Rated capacity [KW] |
|---|---|---|
| 18 | 4,3 | 58 |
| 20 | 4,1 | 50 |
| 22 | 3,9 | 45 |

*Table 1 – Relationship between geometry and capacity*

## About CASTEL

Castel, a leading supplier of refrigeration and air conditioning components, is a 100% Italian-owned, family-run company which has grown and established a name for itself since 1961. Our aim is to increase our customer base by providing reliable, durable, high quality, technologically advanced products, manufactured in an environmentally friendly manner, that are supported and enhanced by service levels that exceed industry standards.

For further information, please visit: www.castel.it

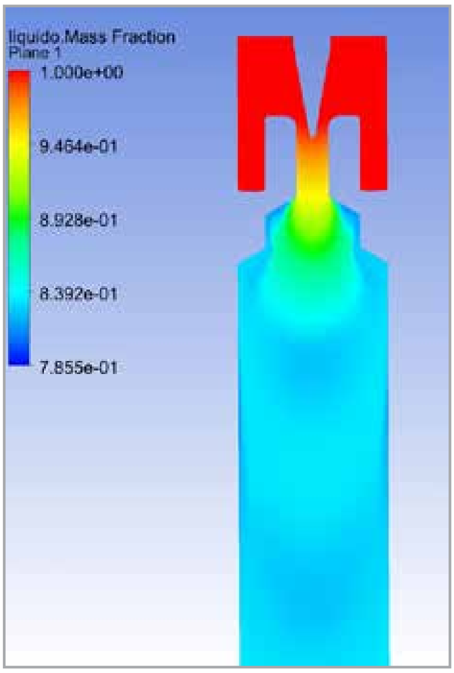



Fig. 4 – Mass fraction: seat canal

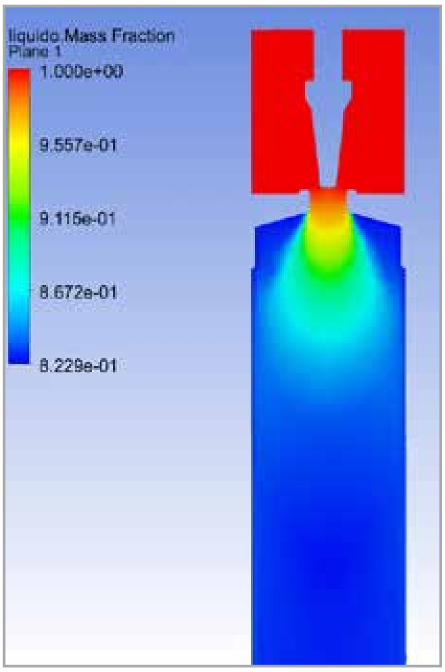



Fig. 5 – Mass fraction: reduced seat canal

the boundary conditions. More specifically, we measured the Cv experimentally in our testing room which required lengthy set-up times.

It therefore seemed very obvious to use a numerical method to solve our problem, and even though Castel's first approach was to calculate the Cv with Ansys, the objective for calculating the EEV capacity was to directly evaluate the mass flow rate for all valve models and therefore their capacity. The mass flow rate calculation, rather than the Cv, was fundamental to comply with Ashrae.

The first challenge we faced was the management of a multi-phase analysis: the refrigerant fluid at the valve inlet is subcooled liquid, while at the valve outlet there is a liquid-vapor mixture. The earliest reliable results obtained with the numerical method showed that the EEV model capacities overlapped slightly and that a few were over-estimated. It was therefore necessary to modify the internal geometries in order to standardize the capacity values; the starting point for this was the results obtained from the simulations.

This approach confirmed that there is a close correlation between the shutter angle and the seat diameter because it changes the amount of vapor at the seat's outlet; the post-processing of the initial simulations focused on these two values in order to improve the internal geometries and obtain the target capacities. Table 1 shows three cases in which the relationship between the geometry and the capacity is evident.

All simulations were conducted with one of the most common refrigerants contained in the Ansys fluid library. Moreover, by means of a specially designed conversion method, it was possible to calculate the capacities for all the refrigerant fluids at each inlet/outlet pressure condition, and so we were able to compile our manual with a variety of data very quickly.

## Results

The most important result obtained was the relationship between cavitation and internal geometry, which helped to adequately review the EEV design. Due to machining requirements, some of the models had a small canal below the valve seat (Fig. 4 vs Fig. 5) that caused the pressure to drop below the liquid's vapor pressure.

The presence of this vapor in the "seat canal" resulted in a choked liquid condition because the vapor build-up occupied additional space. As a result of this fluid dynamic phenomenon, the theoretical capacity of some valves was exceeded. This hypothesis was demonstrated by correctly setting the cavitation model.

## Conclusions

Finally, we compared the numerical results with the experimental tests in order to validate our numerical approach for calculating the mass flow. As previously noted, since mass flow meters would have been unable to measure the majority of our valve's mass flows, the smallest valve was tested with a refrigeration testing system owned by Castel's laboratory. The simulated mass flow rate values proved to be reliable (see Table 2).

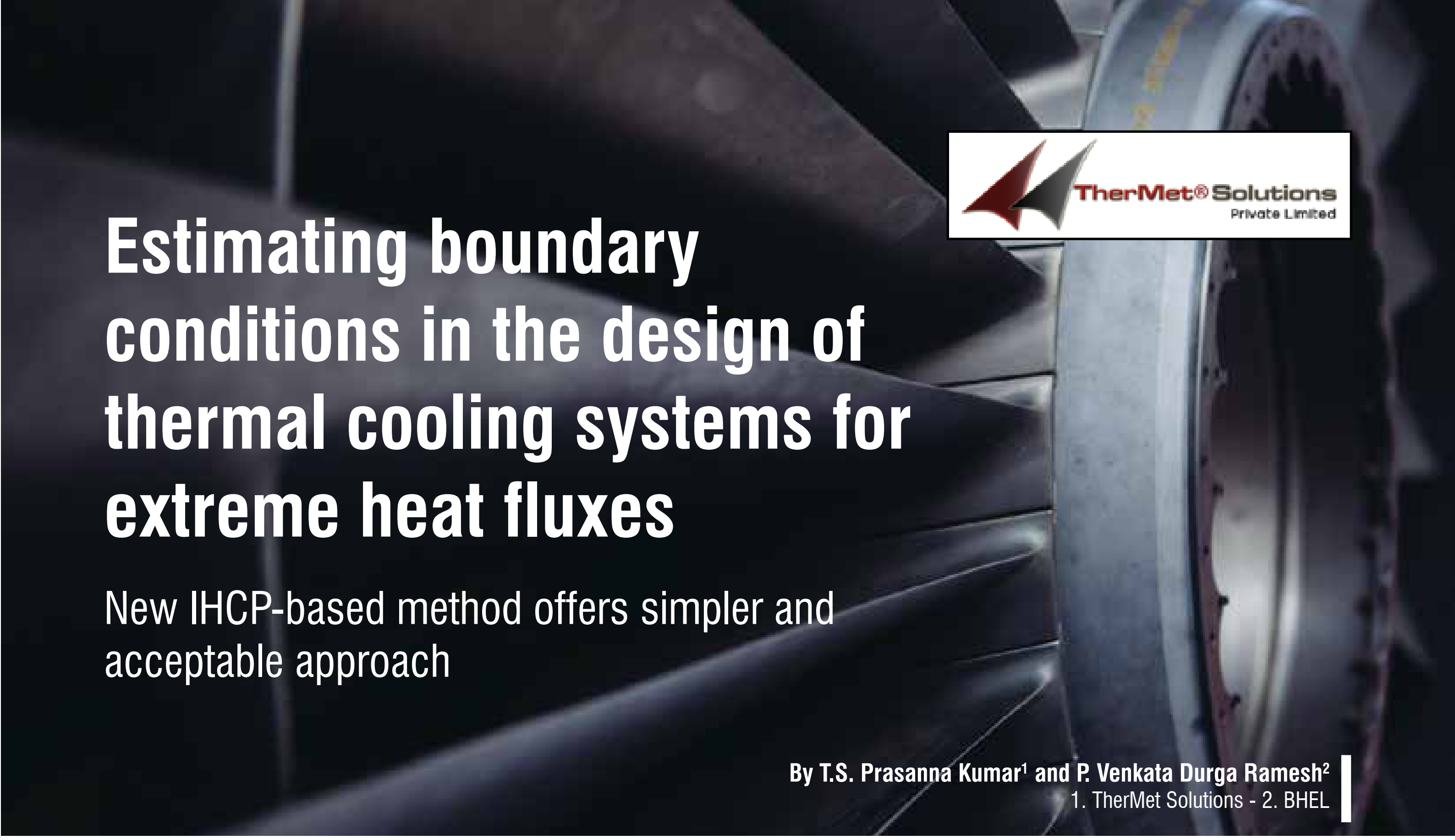
| | Temperature [°C] | Pressure [bar] | Enthalpy saturated Liq/Vap [KJ/kg) | Vapor Quality [%] | Density [Kg/m3] | Velocity [m/s] | Mass flow [kg/s] |
|---|---|---|---|---|---|---|---|
| INLET | 28 | 8,15 | 244,69 | 0 | 1177 | 1,04 | 0,095 |
| OUTLET | 5 | 3,55 | 401,67 | 0,18 | 95 | 12,65 | 0,095 |

| | |
|---|---|
| SIMULATED CAPACITY [KW] | 14,91 |
| TEST MACHINE CAPACITY [KW] | 15 |

Table 2 – Simulated vs. experimental capacity, R134a

As this case study has shown, the numerical approach is both convenient and reliable, and enabled us to predict capacity, a key feature of our valves. It is also an excellent tool for designers to predict performance changes between different design configurations.

For more information:
Paola Brambilla - EnginSoft
p.brambilla@enginsoft.com



# Estimating boundary conditions in the design of thermal cooling systems for extreme heat fluxes

## New IHCP-based method offers simpler and acceptable approach

**By T.S. Prasanna Kumar[1] and P. Venkata Durga Ramesh[2]**
1. TherMet Solutions - 2. BHEL

*Designing cooling systems for high heat environments is highly complex. The heat flux needs to be carefully calculated in order to select the correct materials and cooling fluid and to design the components. This article presents a method based on the Inverse Heat Conduction Problem for calculating the heat flux within any type of fluid-structure interaction, irrespective of the way the heat is generated.*

### Introduction

There are many challenges revolving around material selection and component design when designing cooling systems to dissipate high heat. Some examples include the design of heat shields for re-entry space vehicles, and cooling systems for the plasma facing components in the tokamak. In the "first wall" structure of the tokamak, particularly, the heat flux may reach 20 MW/m$^2$[1]. Among the many critical requirements for the design of such systems, correctly estimating the heat flux is essential because it dictates the choice of materials, the cooling fluid, and the component design.

This article presents a method based on the Inverse Heat Conduction Problem (IHCP) to estimate heat flux within any conceivable structure-fluid interaction - without needing to go into the myriad complications on the fluid side. Such heat could be generated by a burning gas, high intensity radiation, or a plasma, and the heat dissipation may be the result of the forced convection of a fluid. While computing heat flux on either side of the cooling system using computational fluid dynamics (CFD) principles is an onerous task, the IHCP-based method offers a far simpler and acceptable approach to estimating the heat fluxes on both the heating and the cooling sides.

The Gas Tungsten Arc (GTA) used in welding applications was chosen as the heat source to demonstrate IHCP's ability to estimate high heat fluxes. GTA can generate highly concentrated heat fluxes up to 40 MW/m2. Experiments were conducted to measure the transient thermal response in blocks of synthetic graphite, aluminum and stainless steel when their surfaces were exposed to a stationary GTA. Current, voltage and arc time were varied to deliver heat fluxes up to 28 MW/m$^2$. The heat flux was estimated by analyzing the time-temperature records in the solids using the commercially available software InverseSOLVER[2]. The measured and the estimated temperatures at the locations of the thermocouples were shown to be in close agreement[3].

### What is the Inverse Heat Conduction Problem and how is it solved?

The numerical solution to a heat conduction problem with specified boundary conditions and known material properties is known as Direct Heat Transfer Problem. However, if we want to obtain the solution to the heat flux (the cause) at a boundary with a known temperature history (the effect) at a point inside the solid, we have to use the Inverse Heat Conduction Problem (IHCP). IHCP requires data based on measurements from a real situation, which can be taken either from a working prototype or from a laboratory setup. Once the actual plant measurements have been taken, IHCP can be used as a diagnostic tool. If a system needs to be designed where the heat fluxes are unknown, test rigs can be set up for data collection, which is then used for scaling up.

Since the IHCP falls under the category of ill-conditioned problems, many regularization and stabilization schemes have been developed, ranging from classical algorithms for solving the IHCP[4], to purely serendipitous solutions for estimating heat fluxes that vary in time and space[5]. The serial solution has been shown to provide acceptable solutions for industrial problems like heat transfer from boiling during the quenching of steels, and heat transfer at the metal-mold interface during die casting etc.[6-8].

## Experimental procedure

Three materials with varying thermal properties were chosen for data generation, viz synthetic graphite (ATJ-S), aluminum (99% purity) and stainless steel (304L). The properties of these materials as a function of temperature are provided in Appendix 1.


*Fig. 1 - Experimental setup: (a) Data acquisition unit (b) GTAW torch (c) Specimen (d) Test rig (e) Thermocouples (f) Power source (g) Shielding gas*

Discs of 10mm-20mm thickness and 60mm diameter were used as specimens. This paper reports the results of three trials that were conducted using the stainless-steel specimens. The GT arc was focused locally over a circular area approximately 12mm in diameter on the specimen surface where it yielded heat fluxes up to 16MW/m2. The arc was maintained on the position for a maximum of 2.3s which was the upper limit to avoid any melting of the specimen.

Mineral insulated stainless steel-sheathed 'K'-type thermocouples were used to measure the temperature 2-3mm below the surface, as shown in Fig. 2.

## Mathematical Model

Heat transfer in the steel disc specimen from the stationary welding arc was assumed to be axi-symmetric and was modelled as shown below (Fig. 2). The relevant heat transfer equations with appropriate boundary conditions are provided in equations (1), (2) and (3).

$$\frac{k}{r}\frac{\partial}{\partial r}\left(r\frac{\partial T(r,z,t)}{\partial r}\right)+k\frac{\partial}{\partial z}\left(\frac{\partial T(r,z,t)}{\partial z}\right)=\rho c\frac{\partial T(r,z,t)}{\partial t}$$

with the initial condition

$$T(r,z)=T_a \quad at \quad t=0$$

and the boundary conditions:

$$-k\frac{\partial T}{\partial z}=q(t)\ on\ S_1$$

$$-k\frac{\partial T}{\partial r}n_r-k\frac{\partial T}{\partial z}n_z=h(T-T_\infty)\ \ on\ S'$$

where $n_r$, $n_z$ are the direction cosines of the outward normal vector at the domain boundary, *h* is the convective heat transfer coefficient at the boundary, *q(t)* is the unknown heat flux boundary The temperatures recorded for 10-12 seconds were then used as inputs to solve the heat conduction equation inversely.

The Finite-Element-based software, InverseSOLVER [2], was used to solve the problem. The model domain was discretized into 40x120 and 80x120 for the 10mm and 20mm thickness samples respectively, with a uniform grid using four node toroidal elements.

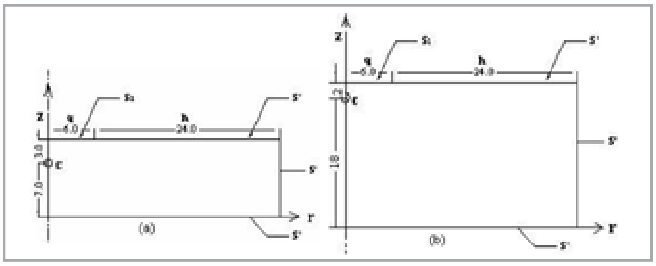



*Fig. 2 - The model domain (a) for Case 1 and (b) for Cases 2 and 3*

Starting from a nominal value of the heat flux at zero time, the equations were solved every 0.1s. The heat flux incremented based on a sensitivity analysis, and the calculations were repeated within the time step until the heat flux value converged to an acceptable value. The convergence limit was set to 10e-4 for the flux values and 10e-6 for the temperatures.

The method is computationally costly and the present problem took about 40 minutes to solve on an Intel Pentium 4 Processor. The solution algorithm has been detailed in other publications[7-9], hence it has not been repeated here.

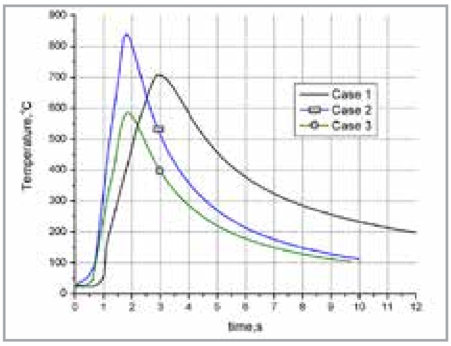



*Fig. 3 - Measured time-temperature curves for the three cases*

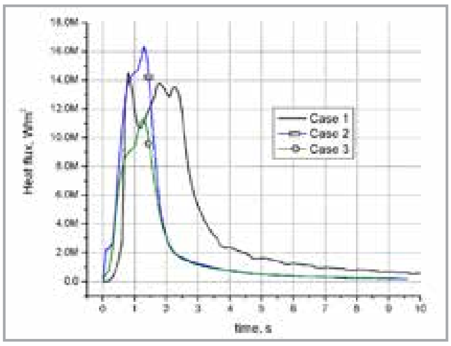



*Fig. 4 - Heat flux variation over time*

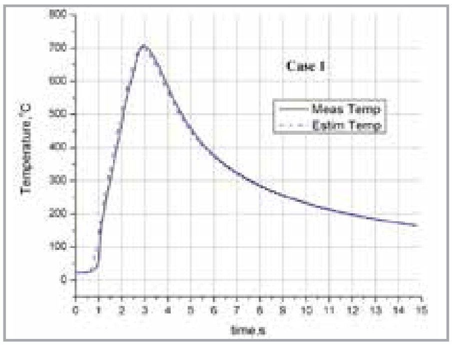



*Fig. 5 - Comparison of measured and estimated temperatures for Case 1.*

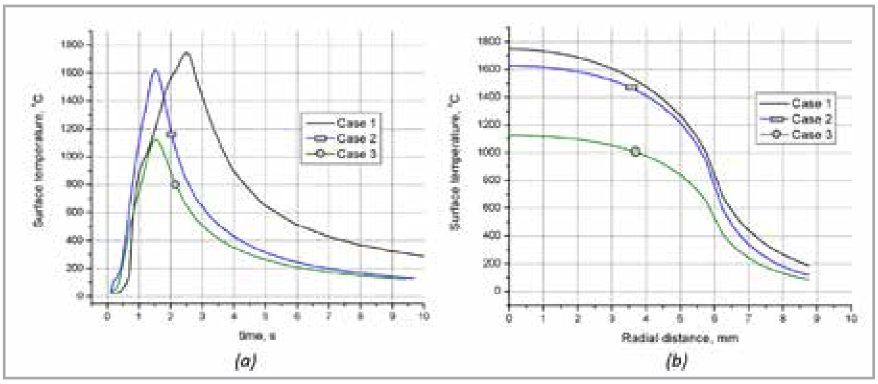



*Fig. 6 - Estimation of surface temperature by thermocouple measurement (a) over time (b) with radial distance at the time of maximum heat flux*

## Results

The temperatures recorded during the experiments are shown in Fig. 3 for all three cases. The heat flux history obtained by solving the equation (1) inversely in all three cases is shown in Fig. 4. Fig. 5 shows a comparison of the measured and estimated temperatures in a typical case, which show excellent convergence.

### *Heat flux*

In case 1, the heat flux reached a maximum of 13MW/m$^2$ within 1.7s and stayed at that level until the arc was cut-off at 2.3s. (Fig. 4). Since the mass of the solid block was half of that in cases 2 and 3, the heat dissipation took more time, as shown by the cooling curve. In cases 2 and 3, a maximum of 16.43 and 11.32MW/m$^2$ was reached within 1-1.2 seconds, although the electrical inputs to the arc were more or less the same. This shows the highly non-linear nature of the arc's properties.

### *Surface temperature*

The solution to the IHCP can also be used to estimate the surface temperature along with the heat flux, which is not possible by direct measurement. This is a distinct advantage of the IHCP since surface melting can be hazardous, particularly in nuclear fusion reactors. The estimated surface temperature at the center of the arc's contact area as a function of time is shown in Fig. 6(a) for all three cases. The temperature distribution over the arc's contact area at the instant of maximum heat flux is given in Fig. 6(b) for all three cases.

## Conclusions

A reliable method to estimate highly transient heat flux from a plasma such as a gas tungsten arc has been developed. The method is based on measuring temperatures inside a solid body by inserting thermocouples near the surface that is exposed to the heat flux. The measured time-temperature data is then used to solve the heat conduction equation inversely to obtain both the heat flux and the surface temperatures. This method can be extended to systems where the heat fluxes over time and space are unknown. By developing test rigs that replicate the real-world conditions on a smaller scale and generating thermal data, the estimated heat flux results can be used to design prototypes of highly complex thermal systems. The advantage of this procedure is that it eliminates the need for assumptions in the solution since only measured thermal data is used.

For more information:
T. S. Prasanna Kumar - Thermet Solutions Pvt. Ltd
tsp@thermetsolutions.com

## References


[1] Jochen Linke, et.al, "Challenges for plasma-facing components in nuclear fusion", Matter Radiat. Extremes 4, 056201 (2019); doi: 10.1063/1.

[2] https://thermetsolutions.com/products/inverse-solver/

[3] P. Venkata Durga Ramesh, "Heat Flux Estimation during Gas Tungsten Arc Welding using Inverse Method", M.S.Thesis, IIT Madras, India, Feb 2009

[4] Ozisik MN, Orlande HRB, Inverse heat transfer: Fundamentals and applications. Taylor & Francis, New York (2000).

[5] T.S.Prasanna Kumar "A serial solution for the 2-D inverse heat conduction problem for estimating multiple heat flux components"- Numerical Heat Transfer Part B-Fundamentals, Vol 45, (2004) 541-563.

[6] T.S.Prasanna Kumar and H.C.Kamath "Estimation of Multiple Heat Flux Components at the Metal/Mold Interface in Bar and Plate Aluminum Alloy Castings"- Metallurgical and Materials Transactions B, Vol 35 B (2004) 575-585;

[7] K. Babu, T.S. Prasanna Kumar; Effect of CNT concentration and agitation on surface heat flux during quenching in CNT nano-fluids, International Journal of Heat and Mass Transfer, Volume 54 (2011) 106-117.

[8] T.S.Prasanna Kumar, "Coupled Analysis of Surface Heat Flux, Microstructure Evolution, and Hardness during Immersion Quenching of a Medium Carbon Steel in Plant Conditions," Materials Performance and Characterization, Vol. 1, (2012)1-22, doi: 10.1520/MPC104477.ISSN 2165-3992.


| Material | Temp K | Density (kg/m^3) | Specific heat (J/kgK) | Thermal conductivity (W/mK) |
|---|---|---|---|---|
| Pure aluminium (99%) | 200 | 2702 | 798 | 237 |
| | 300 | | 903 | 237 |
| | 400 | | 949 | 240 |
| | 500 | | 996 | 236 |
| | 600 | | 1033 | 231 |
| | 800 | | 1146 | 218 |
| Synthetic graphite (ATJ-S) | 300 | 1810 | 1300 | 98 |
| | 1000 | | 1926 | 55 |
| | 2000 | | 2139 | 38 |
| | 3000 | | 2180 | 33 |
| Stainless steel (304 L) | 200 | 8000 | 402 | 13 |
| | 300 | | 477 | 15 |
| | 400 | | 515 | 17 |
| | 500 | | 539 | 18 |
| | 600 | | 557 | 20 |
| | 800 | | 582 | 23 |
| | 1000 | | 611 | 25 |
| | 1200 | | 640 | 25 |
| | 1500 | | 682 | 25 |

*Appendix 1: Thermophysical properties of materials*

# Shortcut the simulation chain with Neural Concept

## Deep Learning tools at the service of CAD/CAE

**By Thomas von Tschammer**
Neural Concept

As organizations try to move simulation earlier on in the design process, the scarce availability of simulation experts compared to the quantity of design tasks and iterations has become a major operational challenge. Moreover, with the rise of Industry 4.0, simulation has shifted very prominently from being a tool to validate mature designs to a means of exploring the product design space.

The increased availability of high-performance computing infrastructures and the progress in high-fidelity simulation methods have certainly contributed to computer assisted engineering (CAE) becoming key in reducing physical testing and in improving product performance. Yet the time and expertise required to run a simulation remain bottlenecks in the engineer's optimization loop most of the time.

In many industries, simulation is fast becoming an essential guide in design as opposed to merely a tool for final validation. Nevertheless, since complex simulation sequences must be re-run each time an engineer wishes to modify a design, the engineering process becomes slower and more expensive. While automation can sometimes make this process less manual, it remains costly in terms of time and simulation resources.

Deep Learning and artificial intelligence (AI) have revolutionized how images, sounds and natural language are processed with the result that some problems that used to be considered intractable are now easily solvable. Computer assisted design (CAD) and CAE are currently on their way to following the same track.

Neural Concept Shape (NCS) is software developed by the company Neural Concept. It empowers the engineer with a new tool, so-called "Geometric Convolutional Neural Networks", to build surrogate models of numerical solvers. NCS is the first deep learning system to understand 3D shapes (CAD) and to learn how they interact with the laws of physics (CAE). It is able to emulate fully-fledged simulators, providing predictions in approximately 30ms, versus the minutes to hours (or even days) required for classic simulators.

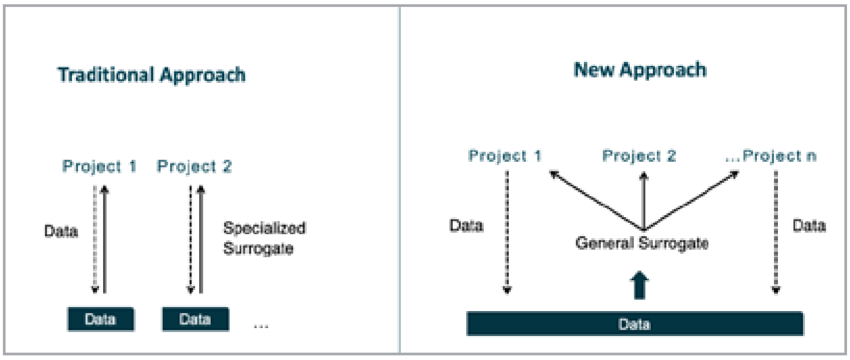



*Fig. 1 - Collapsing silos between iterations, projects, and teams*

| | Traditional "AI" for engineering<br>Learns to simulate in the parameter space | Deep-Learning<br>Learns to simulate in the physical space |
|---|---|---|
| Compute time | Milliseconds | Milliseconds |
| Optimisation | Space of designs limited by parameters | Complete freedom, exploration out of the box |
| Fields prediction | Oversmooth | Very Good / Sharp |
| Data generality | Individual design cycles are isolated | All data ever produced during all design cycles |
| Evolving performance | Fixed | Performance increases over design cycles |
| Convenience of use | Manual Parametrization Burden | Handles any raw simulation data |
| Simulation / Sensor data fusion | NA | Yes |

*Fig. 2 - Comparison between standard surrogate models and deep learning-based surrogates*

The key feature of this new method is its ability to handle raw, unprocessed, 3D data. Hence the input to the neural network is the standard geometric representation of the object that engineers are used to dealing with – the CAD format. As a result, this method suffers none of the drawbacks of the more classic surrogate or reduced-order models (ROM) that are usually used to approximate the actual response of the solver.

Indeed, NCS is agnostic to shape parameters as it directly processes the mesh representation of the design. Using the physical space as a reference for the learning makes it possible to use any raw, unprocessed data source to train a model. One can therefore use an existing database of simulation results or even reuse data across different design iterations, across projects, or across multiple teams.

This new approach is breaking down the silos between projects and clearing the way to a unified model that can be built once and used always (see Fig. 1).

NCS allows engineering teams, at all levels of expertise - including those that know nothing about Artificial Intelligence/Data Science, to include the latest and brightest deep learning-based engineering practices in their development processes. It makes it possible to shortcut any simulation chain using a predictive model that outputs post-processed results right from the CAD design. Industry is using these models to simplify processes and to emulate the expertise of simulation engineers, placing it in the hands of product or design engineers from early on in the development process.

The software platform makes it possible to reduce the number of iterations between teams and to accelerate design activities. Direct applications of faster simulation results for new designs are numerous and include interactive design exploration for engineers, real-time control in digital twins, or large-scale shape optimization.

Since the Geometric Neural Networks take the raw data as produced by the design software directly as input, it is easy and straightforward to integrate these methods into the engineer's workflow. Geometries can be exported in mesh or in CAD format straight from the software to make predictions in Neural Concept Shape, independently of how the data was originally produced. Therefore, these approximate simulations can be directly integrated with or coupled to the engineer's original design tools.

***"Neural Concept Shape enables us to be much more efficient in designing products to meet our customers' requirements. The feedback from our design iterations is so fast that Miniswys' engineers can see the evolution of the performance almost instantaneously while changing the design parameters. In other words, slow iterations are replaced by quick predictions that give us the ability to intuitively improve the performance of our actuators."***

*Raphaël Hoesli, CTO of Miniswys*

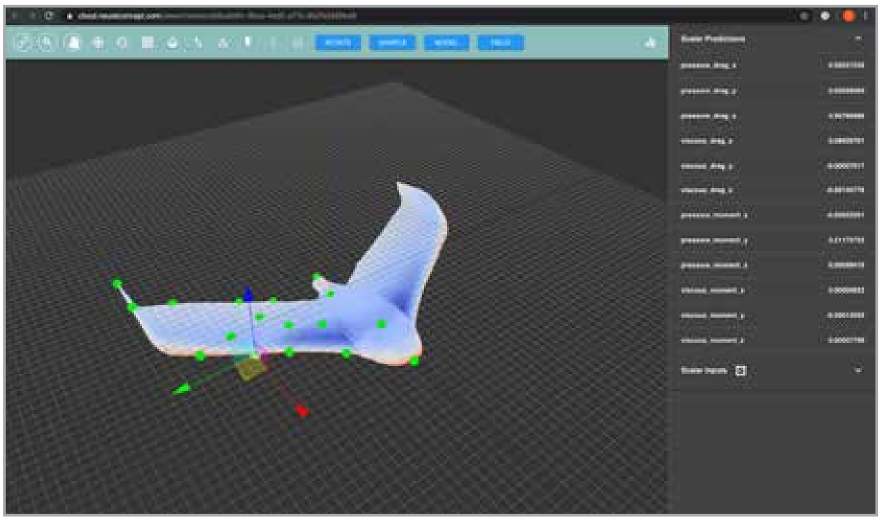


*Fig. 3 - Design exploration of a fixed-wing drone. The user is able to interactively move the green dots ("control points") to deform the geometry, after which he gets an instantaneous prediction of the aerodynamic performance of this new design.*

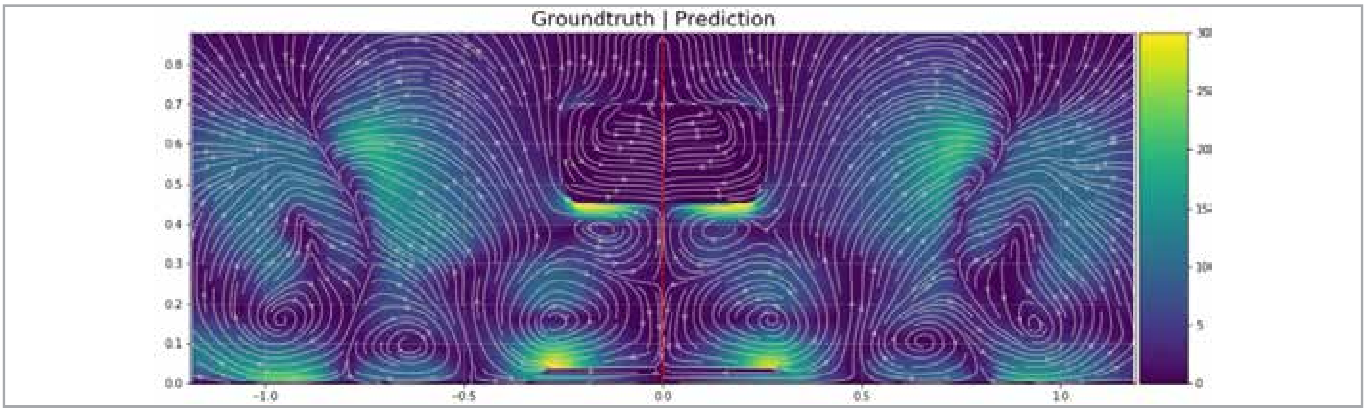



*Fig. 4 – Velocity streamlines on a 2D plane from a CFD simulation of a Formula 1 car. Comparison between the CFD simulation (left half, 3 hours of computing time) vs. the prediction of the NCS model (right half, less than 1 second).*

In addition to real-time simulation, surrogate modelling can also be used for design optimization. Since the Neural Networks are fast and accurate, they can be integrated into an optimization loop that automatically searches through thousands of different designs to converge on an optimal one.

Neural Networks are also differentiable, so one can compute a differentiated output with respect to the input and use gradient-based optimization algorithms to find the best possible design. Once again, since the surrogate is not based on a specific description of the design, the shape's parametrization and deformation is independent of the surrogate. Control points or other deformation schemes can therefore be used to deform any initial design, which can be modified or re-initialized at any point in the optimization. The optimization procedure will not be confused by a redundancy in the parameters, and the design of the deformations – which could normally be hazardous – will be more flexible and less tedious.

This opens the door to a vast unexplored landscape of opportunities in creating complex and creative design spaces for optimization. Optimization becomes less of a black box and more of a tool that the engineer can interact with, requesting algorithmic help along one or another mode of deformation, which can be created at will. Deformations that are completely free i.e. where any point is free to move independently using control points, or other CAD-based

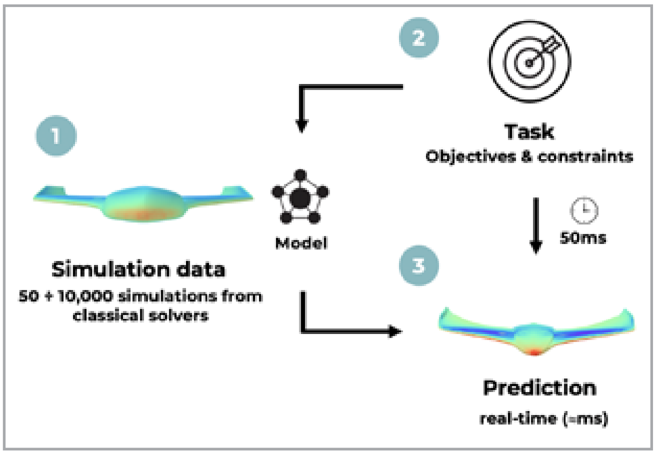



*Fig.5– Optimization workflow using Neural Concept Shape*

parametrization can be interchanged and combined within the optimization platform. Using Neural Concept Shape, even CAD-based parametrization can be integrated into a gradient-based optimization scheme to ensure that the company reaches a truly optimized design based on the given constraints and objectives.

In conclusion, the evolution in Deep Learning of recent years is providing CAE engineers and their organizations with a ubiquitous tool that can be integrated into non-CAE engineers' workflows, taking design exploration and optimization one step further.

## About Neural Concept

Neural Concept is a Swiss-based company that offers the first-ever deep learning-based software solution dedicated to Computer Aided Engineering and Design. It speeds up R&D cycles, enhances product performance, and reduces simulation costs. Neural Concept Shape (NCS) is a unique Deep Learning based software for enhanced engineering. It leverages industrial CAE data to efficiently assist engineers during the product design stage.

Starting from Computer Vision Lab EPFL's longstanding expertise in 3D surface modeling, we begun, more than four years ago, investigating a Deep Learning based approach to overcoming the limitations of existing simulation methods and speeding up by several orders of the magnitude the exploration of the space of potential 3D shapes. We submitted a patent for this new technology. Then, Neural Concept was founded in May 2018 with the goal of commercializing the technology developed by EPFL's Computer Vision Lab. The company is now active in many different industries such as aerospace, automotive, energy, marine or civil engineering, working with industry leaders across the globe, such as Airbus, Safran or Bosch. Neural Concept has now worked on more than 50 industrial projects, to design high-end products, and beat world records.

For more information:
Anthony Massobrio - Neural Concept
anthony.massobrio@neuralconcept.com

**By Francesco Di Caprio[1], Valerio Acanfora[2], Stefania Franchitti[1], Rosario Borrelli[1], Aniello Riccio[2]**
1. CIRA – Italian Aerospace Research Centre - 2. Department of Engineering, University of Campania "L. Vanvitelli"

# Numerical applications of an Ansys-based tool to analyze hybrid metal/composite lattice structures

## Computationally efficient method enables design and optimization processes of axial-symmetric lattice structures to be conducted simply and quickly

*This paper, by means of presenting three application cases, provides a brief description of the capabilities of a numerical tool developed for the design and optimization of hybrid metal/composite lattice structures created with 3D printing. The tool allows a large number of axial symmetrical structures to be generated in the ANSYS environment through the use of a macro written in APDL language. The models generated can be used for both sensitivity analyses and structural optimization processes (they are totally parametrized). The strength of the procedure is its ability to generate many unit cells by defining a matrix structure that activates specific connectivity flags. The models generated are not too expensive computationally due both to the simplification of the models (made with beam and shell elements) and to the elimination of any possible multiple point constraints (MPC) elements between the nodes of the lattice structure and the remaining solid parts. This enables the finite element (FE) models generated to also be used for more expensive analyses such as non-linear buckling ones.*

Additive manufacturing (AM) has projected the global industry into a new phase of transformation in product design and fabrication. This change involves both the manufacturing and research fields in a radical way. Indeed, the use of AM is increasing rapidly in many industrial sectors [1- 4]. The reason for this success is based on the fact that 3D printing technology uses a particular and elegant methodology known as additive layering. Specifically, this additive methodology allows an object to be created layer by layer, in a totally different way from the subtractive techniques commonly used. It enables the construction of complex geometries and easily makes use of a wide variety of materials. The preliminary phase of AM is based on a computer-aided design (CAD) of the object to be realized [5-7]. This is where the construction of elaborate shapes, for instance lattice or porous structures, is simplified, or organic constructions are generated with topological optimization [8,9]. Furthermore, the costs of the wastage of resulting materials are significantly reduced with additive technology, compared to the subtractive technique. In fact, Additive Manufacturing makes it possible to "print" a prototype by only stratifying the regions required by the CAD model with material. [10,11].

Thanks to these advantages, AM can be widely adopted in a rapidly changing sector like aerospace. The necessity to develop strategies to reduce fuel consumption and manufacturing costs and times is one of the main objectives of aerospace research. This also all leads to the significant use of hybrid metal/composite structures.

Among these hybrid combinations, the coupling of carbon fiber reinforced polymer (CFRP) composite materials with titanium alloys is the most suitable for aerospace use due to the similarities of the thermal expansion coefficients (a highly relevant aspect for composite manufacturing processes) of these materials. In addition, this characteristic reduces the issues related to galvanic corrosion.

The increasing demand for hybrid components characterized by complex shapes and high performance requires both the numerical design tools and the manufacturing processes to be upgraded. The present study presents some numerical applications of an innovative numerical design tool [12] developed for the preliminary design and numerical investigation of hybrid structures made, mainly, from a lattice structure. The developed routine allows modelling in an FE environment and the structural optimization of hybrid composite/lattice metal structures with different shapes and loading conditions. More specifically, the structures analyzed were considered to have been manufactured with Electron Beam Melting (EBM) technology for the metallic parts, and filament winding for the composite parts. In particular, the tool is dedicated to the design of axial symmetric components.

For the numerical implementation of the lattice part, there are many methodologies in the literature [13-15] that are based on tetrahedral cells in a repetitive configuration. Usually, however, these are unable to adjust to the outer borders (the cell edges are cut from the edges of the structure). Other methods, developed in commercial codes, can create other types of unitary cell structures (cubic, octahedral, etc. [16-20]), but even these are unable to adapt to complex geometries. Instead, this routine, developed in APDL, is able to overcome these limits because it allows any type of unit cell to be generated, and to be adapted if the user desires, to any type of frontier in an axial symmetrical structure. The algorithm to generate the internal lattices is very efficient, simple to manage and does not require significant computational power.

## Description of the proposed routine

The routine [12] presented in this work has been developed in APDL and tested in Ansys 16.1 and later.

As indicated in Fig. 1, the procedure consists of 12 main parts:

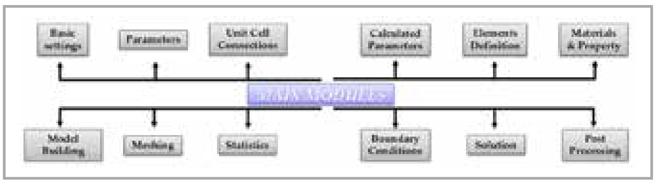



*Fig. 1 - Main modules of the routine*

The initial stage of the macro requires Basic Settings to be defined. In this phase, the user selects the desired graphics options, the file name, and the resolution of the images that will be generated during the execution of the procedure. The second stage concerns the definition of all the parameters and options to allow the geometric models to be generated, and the definition of all parameters related to the required analysis. In addition, to ensure consistent design, automated parameter checking is implemented to minimize the possibility of creating non-generable geometries.

The third and most critical module concerns the definition of the connections between the nodes of the Unit Cell (UC). It is possible to create two different Unit Cell architectures: an 8-node Unit Cell, and a 27-node Unit Cell (Fig. 3). The connection between the nodes is defined by activating a flag (1 or 0). In the fourth module (Calculated Parameters), additional parameters to correctly dimension the UC are assessed by combining all the previously defined parameters.

The fifth and sixth modules aim to define the element type and the mechanical properties of the selected materials. Another particularly delicate phase of the routine is that dedicated to the construction of the model. In this seventh module, it is possible to generate any type of hybrid axial symmetric structure, regardless of its geometric complexity. Fig. 2 shows some examples of the types of structures it is possible to realize.

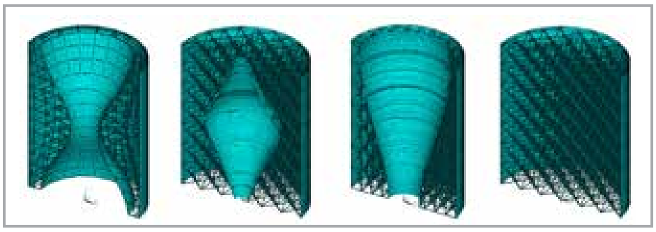


*Fig. 2 - Examples of possible structures*

In the eighth part of the routine, the structural mesh is generated, while the Statistics section evaluates some important data (total mass, mass and volume of each part, FE details). The last three modules of the procedure concern the boundary conditions to be applied, the type of analysis to be carried out, and the generation of the constraint and objective functions for the optimization phase, respectively. The routine offers the possibility of running different types of analyses: a linear structural analysis, a non-linear static analysis (non-linear buckling), and a linear buckling analysis.

### *Unit cell definition*

A significant part of the routine being presented is the Unit Cell design, which is used to generate the entire lattice structure. Different families of

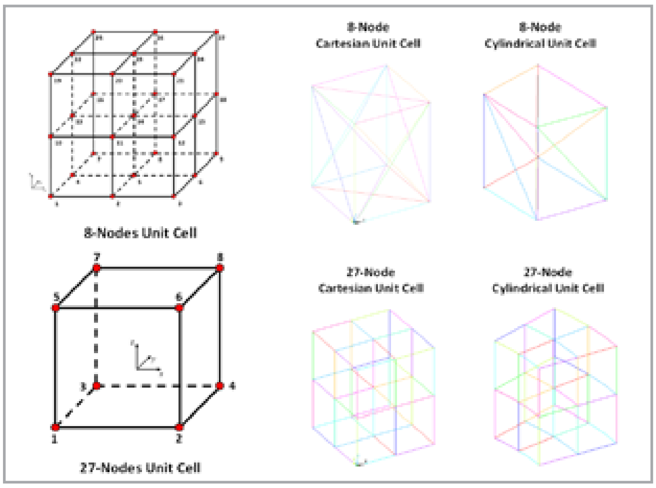



*Fig. 3 - Unit Cell classification*

unit cells have been developed for the design of the metallic lattice parts. A first classification is based on the number of nodes (no FE entities) of the unit cell. Two options are available: unit cells with 8 nodes or unit cells with 27 nodes (Fig. 3). In addition, by combining cells with 27 and 8 nodes, it is possible to realize unit cells with more than 27 nodes. Depending on the type of construction reference system being used to generate the unit cells, it is possible to obtain a second classification: unit cells with a quadrangular prism-based structure, or cylindrical sectors (Fig. 3).

Cylindrical unit cells provide additional functionalities:

- Straight or curved beams. This option allows a perfect integration of the lattice regions in axial symmetric structures.
- Bias factor. By activating the bias factor, the cells will increase in size as they move away from the axis of the structure. Bias can even be automatic, i.e. the routine automatically calculates the cell size in the radial direction in order to preserve the relationship between the circumferential and radial directions for all cells. The total number of cells that the user wishes to insert in the radial direction does not change.
- Adaptive cells. By activating this option the nodes of all cells follow the inner and outer skin (or frontiers). Therefore, this enables the basic structure of the cells to be modified, while keeping their internal connections unchanged in order to match the extreme nodes of the external cells with those of the skins.

In order to define a specific cell (or to determine the best unit cell in an optimization process), a set of flags has to be defined. By setting the flag parameters appropriately, it is possible to generate unit cells with a different topology. Fig. 4 and Fig. 5 show some examples of 8-node UC and 27-node UC.

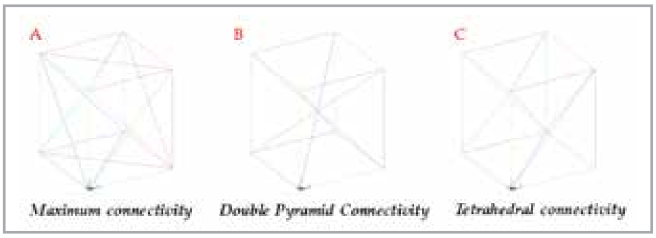



*Fig. 4 – Connections in an 8-node Unit Cell. (A) Maximum connectivity UC; (B) Double pyramid connectivity UC; (C) Tetrahedral connectivity UC.*

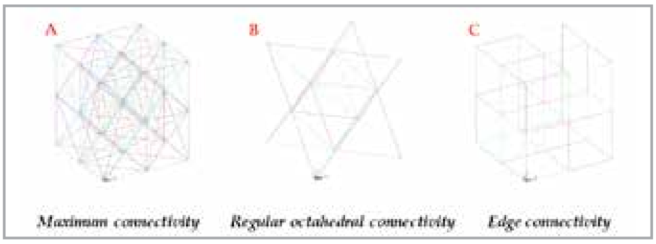



*Fig. 5 □ Connections in a 27-node Unit Cell. (A) Maximum connectivity UC; (B) Regular octahedral connectivity UC; (C) Edge connectivity UC.*

## Numerical Applications

The numerical procedure developed allows a lot of axial-symmetric structures to be realized very simply, based on the concept of lattice structures. The lattice structure and the internal skin are always assumed to have been made with additive manufacturing technology, while the external skin can be made from the same material as the other parts or from composite material.

It goes without saying that the numerical models developed can be used for any kind of numeric analysis. The tool has been designed to replicate different types of loading conditions, but others not currently implemented can easily be introduced. Furthermore, to keep the models fairly inexpensive computationally, all the nodes of all the parts are merged into each other - including the interface nodes between the lattice part and the skins. While this leads to greater complexity during model generation, it makes it possible to avoid the use of bonded contact regions. Hereafter we report some examples to demonstrate the tool's versatility.

### *Test Case 1*

The first test case concerns a truncated conical structure with an internal skin and lattice structure in titanium alloy (electron beam melting process) and an external skin in composite material with the following lamination sequence [45; -45]2. Both skins have a thickness of 1mm, while the diameter of the beams in the lattice structure is 1mm. Fig. 6 shows a schematic of the analyzed structure, the global FE model, the details of the unit cell used, and a section of the internal sub-structure.

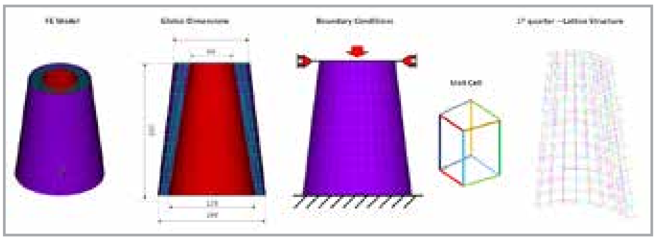


*Fig. 6 – Test Case 1: Geometry, boundary conditions, and unit cell selected*

An 8-node adaptive cell without a bias factor was used for this structure. The cell has only 12 connections, represented by the edges of the cell and neglecting the diagonal connections. The structure was subjected to a compressive loading condition to evaluate the buckling load using linear buckling analysis. The entire structure weighs 0.609kg, of which 0.246kg relates to the lattice part, 0.099kg to the external composite skin, and 0.262kg to the internal metallic skin.

Fig. 8 shows the nodal displacements (eigenvector) for the first buckling mode (eigenvalue) calculated and highlights the fact that instability involves both the skins and the internal lattice structure. Actually, for both components, local buckling modes were found without global instability that could compromise the load capacity. The critical displacement that generates the first buckling mode is 1.63mm, while the equivalent critical compressive load is 394.02kN.

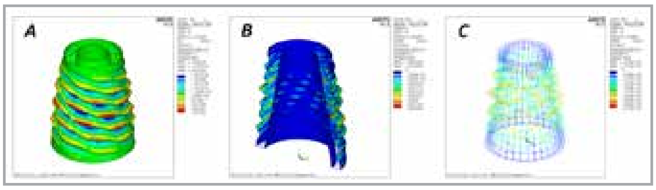



*Fig. 7 □ Test Case 1: Global displacement at 1st buckling mode. A: Cylindrical reference system; B: Section view; C: Internal lattice structure*

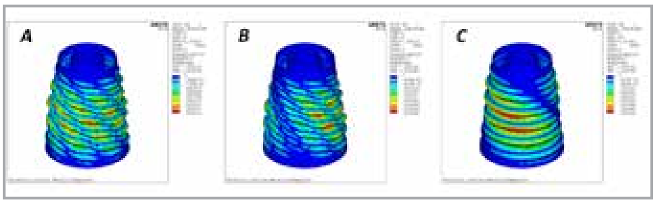



*Fig. 8 – Test Case 1: Global displacement relative to buckling modes. A: 2nd mode; B: 5th mode; C: 10th mode*

Therefore, the structure can withstand significant compressive loading before buckling occurs.

Fig. 8 shows the nodal displacements (in a cylindrical reference system) for the other buckling modes which can also be classified as local buckling modes.

The results highlight that such a structure is characterized by a large number of very closely spaced local buckling modes, which do not compromise the global loading capabilities [12].

### Test Case 2

The second test case involves a cylindrical structure with an external skin in composite material and an internal lattice structure in titanium alloy. The structure's total mass is 157g, of which the lattice structure weighs 98g, and the remaining 58g can be attributed to the external skin. The cylinder was subjected to axial compressive loading conditions.

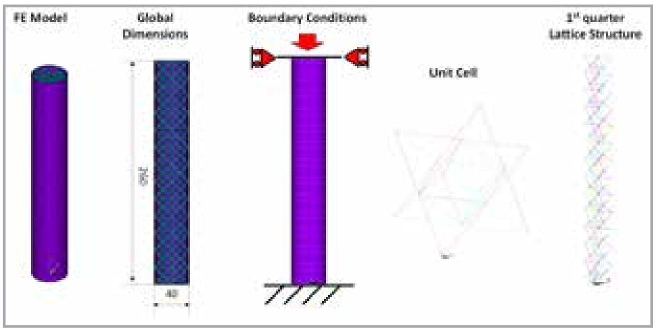



*Fig. 9 - Test Case 2: Geometry, boundary conditions, and unit cell adopted*

A 27-node Cartesian cell was used for this structure. It has 36 connections and is classified as a regular octahedral. The structure has no internal skin, while the layup of the external skin is [45;-45]3. The above figure shows the adopted elementary cell and a quarter of the global lattice structure. The numerical results reported show the instability limits, evaluated by means of a static non-linear analysis.

Fig. 10 shows the deformed shape at about 140kN, which corresponds to 10.82mm of applied axial displacement (U0). As can be seen from the images, this load value triggers local forms of buckling that involve both the skin and the internal beams. These results were also confirmed by linear buckling analyses [12], not reported here for the sake of brevity. As the load increases, more instabilities are triggered, and the structure is unable to withstand further load increments (Fig. 12).

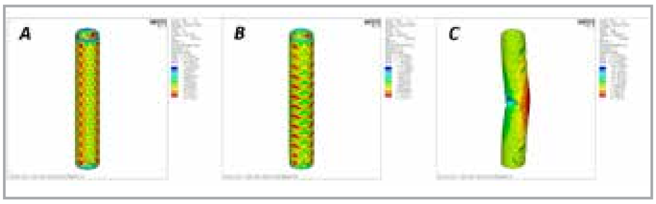



*Fig. 10 - Test Case 2: Radial displacement. A: U0 = 10.8mm; B: U0 = 13.1mm; C: U0 = 15.1mm.*

The graph in Fig. 11 shows the time history of the reaction load in the axial direction. From this graph it is evident that the global buckling is triggered at about 13mm (applied displacement), which correspond to 162kN. After the first modes of local buckling (140kN), the structure is then able to withstand an additional, fairly constant load of 15% until total collapse occurs at about U0 = 15mm.

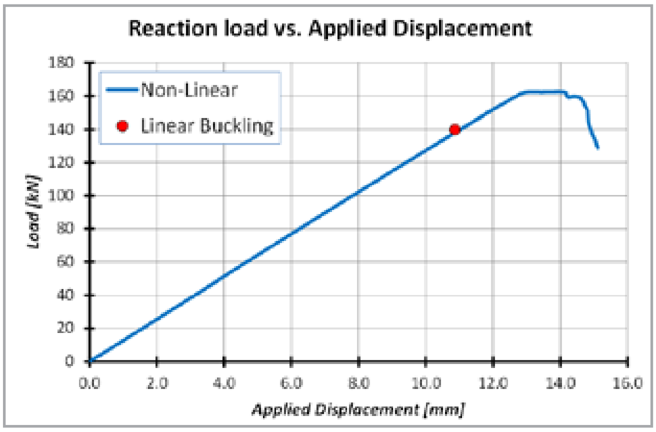



*Fig. 11 - Test Case 2: Load vs. applied displacement*

### Test Case 3

The third test case examines an axisymmetric structure, representing a convergent-divergent nozzle. The applied load distributes a uniform pressure on the inner surface of the inner skin equal to 0.1MPa. The constraints aim to eliminate the concentration of undesired stress and obtain an isostatic structure. Fig. 12 shows the analyzed structure with its dimensions and the applied boundary conditions.

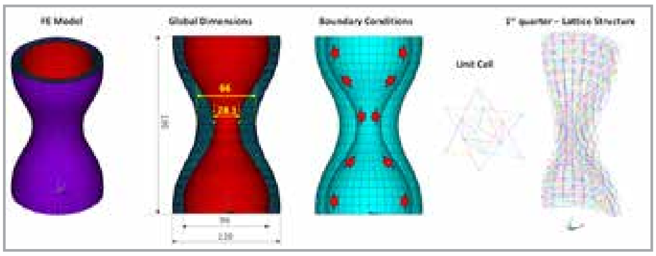


*Fig. 12 - Test Case 3: Geometry, boundary conditions, and unit cell adopted.*

An adaptive cylindrical cell with 27 nodes was used. The cell has 36 connections and is classified as a regular octahedral. The diameter of each element of the lattice structure is equal to 1.0mm.

The thickness of the metallic inner skin is 0.5mm while the layup of the external skin is [45; -45]3. The overall structural mass is 486g, consisting of 272g for the lattice structure, 113g for the composite outer skin, and 100g for the metallic inner skin. Fig. 13 shows the

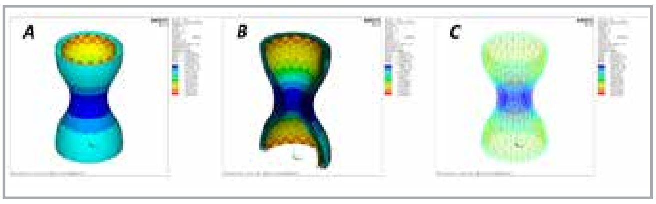


Fig. 13 - Test Case 3: Global displacement. A: Entire model; B: Section view; C: Lattice structure

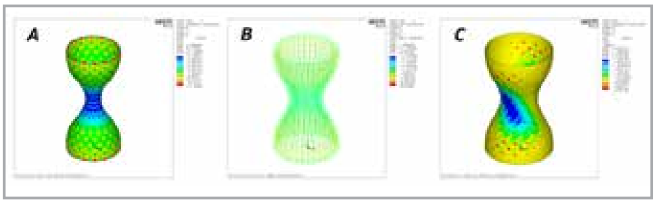


Fig. 14 - Test Case 3: Stress distribution. A: Inner metallic skin; B: Lattice structure; C: Outer composite skin

global displacements of the entire structure and the detail of the internal lattice structure.
The following figures show the stress state of the metal parts, i.e. the internal skin, the internal beams, and the external composite skin. For the internal skin, the Von Mises equivalent stress is reported, while for the beams the axial stress state is reported.

For the external composite skin, the stress in fiber direction is reported for the outer layer (similar data is obviously available for all layers in all directions).

## Conclusions

The work briefly describes an automatic procedure based on APDL macros to study complex structures made of lattice structures and solid parts. This procedure also enables the study of hybrid metal/composite structures. Given their complexity, the metal parts are intended to be created with additive manufacturing technologies.

The procedure is able to create axial-symmetric structures with different types of unit cells by simply defining a few parameters. The cells fit perfectly into the structure; therefore, no cuts are made at the boundary surfaces of the inner and outer skins. In order to reduce computational costs, various solutions were implemented: exclusive use of shell and beam elements; connections between parts made by merging the interface nodes and avoiding the use of contact algorithms with bonded option.

The use of this procedure permits many axial symmetric components to be analyzed quickly and simply. In particular, it provides valuable support for studying the performance of lattice structures with and without solid parts and with both metallic and composite material systems. Moreover, the APDL macro, being fully parametrized, enables sensitivity analyses and optimization processes to be performed.

For more information:
Francesco Di Caprio - CIRA Italian Aerospace Research Centre
f.dicaprio@cira.it

## References

1. Wei Gao, Yunbo Zhang, Devarajan Ramanujan, Karthik Ramani, Yong Chen, Christopher B. Williams, Charlie C.L. Wang, Yung C. Shin, Song Zhang, Pablo D. Zavattieri. The status, challenges, and future of additive manufacturing in engineering, Computer-Aided Design, Volume 69, 2015, Pages 65-89.
2. Tofail, S.A.; Koumoulos, E.P.; Bandyopadhyay, A.; Bose, S.; O'Donoghue, L.; Charitidis, C. Additive manufacturing: Scientific and technological challenges, market uptake and opportunities. Materials Today, 2018, 21(1), 22-37.
3. Berman, B. 3-D printing: The new industrial revolution. Business Horizons, 2012, 55(2), 155-162.
4. Huang, S.H.; Liu, P.; Mokasdar, A.; Hou, L. Additive manufacturing and its societal impact: A literature review. International Journal of Advanced Manufacturing Technology, 2013, 67(5-8), 1191-1203.
5. Gao, W.; Zhang, Y.; Ramanujan, D.; Ramani, K.; Chen, Y.; Williams, C.B.; Wang, C.C.L.; Shin, Y.C.; Zhang, S.; Zavattieri, P.D. The status, challenges, and future of additive manufacturing in engineering. CAD Computer Aided Design, 2015, 69, 65-89.
6. Thompson, M.K.; Moroni, G.; Vaneker, T.; Fadel, G.; Campbell, R.I.; Gibson, I.; Bernard, A.; Schulz, J.; Graf, P.; Ahuja, B.; Martina, F. Design for Additive Manufacturing: Trends, opportunities, considerations, and constraints. CIRP Annals - Manufacturing Technology, 2016, 65 (2), 737-760.
7. Wu, P.; Wang, J.; Wang, X. A critical review of the use of 3-D printing in the construction industry. Automation in Construction, 2016, 68, 21-31.
8. Panesar, A.; Abdi, M.; Hickman, D.; Ashcroft, I. Strategies for functionally graded lattice structures derived using topology optimisation for additive manufacturing. Additive Manufacturing, 2018, 19, 81-94.
9. Ngo, T.D.; Kashani, A.; Imbalzano, G.; Nguyen, K.T.Q.; Hui, D. Additive manufacturing (3D printing): A review of materials, methods, applications and challenges. Composites Part B: Engineering, 2018, 143, 172-196.
10. Yeh, Chin-Ching. "Trend analysis for the market and application development of 3D printing." International Journal of Automation and Smart Technology 4.1 , 2014, 1-3.
11. Dilberoglu, U.M.; Gharehpapagh, B.; Yaman, U.; Dolen, M. The role of additive manufacturing in the era of industry 4.0. Procedia Manuf, 2017, 11, 545–554.
12. Di Caprio, F., Acanfora, V., Franchitti, S., Sellitto, A., Riccio, A.; Hybrid Metal/Composite lattice structures: Design for Additive Manufacturing; 2019 Aerospace, 6 (6), art. no. 71.
13. Tomlin, Matthew, and Jonathan Meyer. "Topology optimization of an additive layer manufactured (ALM) aerospace part." Proceeding of the 7th Altair CAE technology conference, 2011.
14. Chen, W.; Zheng, X.; Liu, S. Finite-Element-Mesh Based Method for Modeling and Optimization of Lattice Structures for Additive Manufacturing. Materials, 2018, 11, 2073.
15. Ferro, C.G.; Varetti, S.; Vitti, F.; Maggiore, P.; Lombardi, M.; Biamino, S.; Manfredi, D.; Calignano, F. A Robust Multifunctional Sandwich Panel Design with Trabecular Structures by the Use of Additive Manufacturing Technology for a New De-Icing System. Technologies, 2017, 5, 35.
16. WILLIAMS, Christopher B.; COCHRAN, Joe K.; ROSEN, David W. Additive manufacturing of metallic cellular materials via three-dimensional printing. The International Journal of Advanced Manufacturing Technology, 2011, 53.1-4: 231-239.
17. ROSEN, David W. Computer-aided design for additive manufacturing of cellular structures. Computer-Aided Design and Applications, 2007, 4.5: 585-594.
18. DESHPANDE, Vikram S.; FLECK, Norman A.; ASHBY, Michael F. Effective properties of the octet-truss lattice material. Journal of the Mechanics and Physics of Solids, 2001, 49.8: 1747-1769.
19. Ferrigno, A.; Di Caprio, F.; Borrelli, R.; Auricchio, F.; Vigliotti, A. The mechanical strength of Ti-6Al-4V columns with regular octet microstructure manufactured by electron beam melting. Materialia, 2019, 5, 100232.
20. Deshpande, V.; Fleck, N.; Ashby, M. Effective properties of the octet-truss lattice material. Journal of the Mechanics and Physics of Solids, 2001, 49(8), 1747-1769.

**By Ada Malagnino[1], Simo Hostikka[2], Giorgio Zavarise[3], Angelo Corallo[1]**
1. University of Salento - 2. Aalto University - 3. Politecnico di Torino

*This paper formed the basis for one of the five winning posters in the Poster Award at the 2020 International CAE Conference and Exhibition.*

# Life-safety assessment of a hotel using fire and evacuation simulations

Improving Building Lifecycle Management by integrating technological developments and advances in knowledge of fire dynamics

Digitalization and the use of technology has been increasing in every field of engineering, yet traditional sectors like the construction industry have been slow adopters. However, the change is unavoidable as new tools are emerging and digital information management is becoming mandatory worldwide, accompanied by enablers like increasing computing power. One application of these modern technologies is using numerical fire simulations to assess the life-safety of the occupants of a building during a fire.

The possibility of studying fires numerically found application quite recently although the equations governing fluid dynamics, heat transfer, and combustion were first written down over a century ago. [Fire Dynamics Simulator: Technical Reference Guide, NIST, 2000]. The main reason for this is the complexity of the problem, followed by the required computational effort. However, advances in technology and the understanding of fire dynamics have led to changes in building design and fire codes to harness this new knowledge and apply Performance-Based Design (PBD). PBD can address the unique features and uses of a building, increasing the cost-effectiveness of the designed fire safety measures and providing a better understanding of building's behavior in the event of a fire.

Life-safety is one of the most important objectives of PBD. This objective is commonly considered to have been achieved if the occupants of the building escape the effects of the fire unharmed. Numerical simulations are often used to predict fire dynamics and


*Fig. 1 - Real building geometry and modelled geometry*

consequently the factors influencing the ability of the occupants to evacuate. Fire Risk Assessment (FRA) plays an important role in performance-based fire design as it affects decisions regarding the safety of buildings and their occupants.

The study presented in this article was performed using a Computational Fluid Dynamics (CFD) tool called Fire Dynamics Simulator (FDS) and a commercial evacuation tool called Pathfinder. It shows an example of the coupling of quantitative risk assessment techniques with simulation to support the decisions of building managers and owners regarding safety measures for occupants.

## The roles of statistics-based fire risk assessment and simulation

An important aspect of FRA is to assess the safety of life, especially for public buildings, shopping malls or hotels where there is high occupant density. However, the statistical data does not provide sufficient information for assessing the life safety risks for specific buildings and their spaces, since several factors affect the occurrence of a casualty or fatality. On the other hand, deterministic approaches such as CFD fire simulations do not provide any information on the probability of the simulated scenario occurring. To address this issue, a statistics-based FRA framework was developed to identify possible fire scenarios based on an initiating event and its associated frequency, while fire and evacuation simulations were used to evaluate the factors influencing the severity of the fire's consequences.

## Case Study

The case study is a hotel located in Lecce, in Southern Italy. The building is a medium-sized hotel, representative of the dimensions of most hotels in Italy.
In the event of a fire in a hotel, many people could be at risk regardless of the year of construction or the use of certain fire protection systems. Guests may experience difficulties in finding their way out due to lack of familiarity with the building environment, while foreign guests may respond more slowly to alarms given the communication barriers. In addition, if the fire occurs during the night, the occupants may be asleep and thus have slower reaction times. From an architectural point of view, bedrooms are often on upper floors and escape routes could be blocked by flames or smoke if the fire starts on the ground floor. The attitude of hotel operators towards fire safety, e.g. the maintenance of fire evacuation systems, is another important element contributing to fire safety. Considering these factors, fire risk assessment in hotels is a relevant issue.

## Fire risk assessment results

An event tree, constructed on the basis of information from the scientific literature and national statistical databases, was used to assess the fire risk for the different spaces in the building. The available information was used to structure the logical sequence of the event tree and to quantify the frequencies for each identified fire scenario.

In hotels, bedrooms are the category of space with the highest percentage of fires that threaten life-safety. FRA results showed that a scenario in which the fire is confined to the room of origin

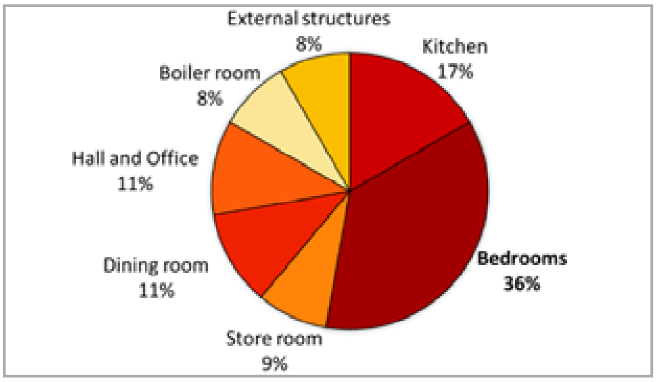



*Fig. 2 – Distribution of hotel fires that can cause casualties or fatalities. Source: UK statistics Home Officegeometry and modelled geometry*

*Fig. 3 - Section of the building showing the location of the fire and the smoke spreading*

was the most frequent of all bedroom fires, with an estimated frequency of $5{,}7 \cdot 10^{-4}$ per annum. The worst case within this series of scenarios occurs when the room door is open, so that fire and smoke can spread easily. If we combine the probability of this happening, the frequency of the scenario to be simulated becomes $5{,}7 \cdot 10^{-5}$ per annum, which is classed as extremely unlikely in the Society of Fire Prevention Engineers' (SFPE) risk classification.

## Fire and evacuation simulation results

Fire and evacuation simulations were carried out for a fire confined to the room of origin. The choice is justified by the fact that this is the most frequent scenario, and because it can be very dangerous under certain circumstances. It is assumed that the fire occurs when guests are sleeping, and the restaurant is still open during the evening/night.

The fire is on the first floor, in the room in front of the main staircase. This fire location produces the most serious consequences for the safety of life in that smoke can suddenly fill the only exit known to the hotel occupants. Occupants will typically attempt to use (or to evacuate through) the exit routes most familiar to them. Furthermore, the staircase connecting the floors will no longer be available to the occupants on the upper floors of the building.

The choice of polyurethane as a fuel is related to its high flammability and the toxicity of its combustion products, as well as its widespread use. Fires in closed environments are characterized by high levels of the two main toxic agents, carbon monoxide (CO) and hydrogen cyanide (HCN), which are produced during the under-ventilated burning phase. A fully oxidized product, such as carbon dioxide (CO2), represents another significant component that reduces the concentration of oxygen in the human body, potentially causing asphyxia. Depending on its concentration, CO toxicity corresponds to different levels of incapacitation: impairment of judgement and visual perception; dizziness, headaches, and fatigue; loss of consciousness; and, at 1000 ppm of CO, rapid death. Hydrogen Cyanide (HCN) is approximately 25 times more toxic than carbon monoxide. The effect of these toxic agents on the human body is calculated using the fractional effective dose (FED) index, which expresses the level of inability caused by toxic substances in the air.

In the event of a rapid fire, within 420 seconds of ignition (alarm time + pre-evacuation time configured for occupants who are unfamiliar with the building and asleep) the staircase is full of smoke and this evacuation route is unusable. The conditions at the two side exits on the first floor also become untenable with:

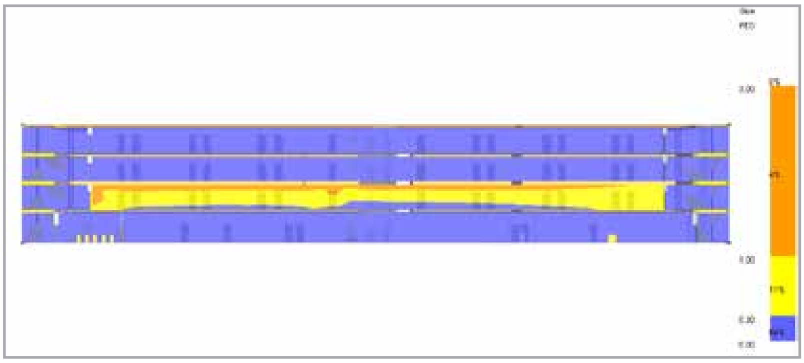


*Fig. 4 – FED index values in the building corridors at 420 seconds from ignition*

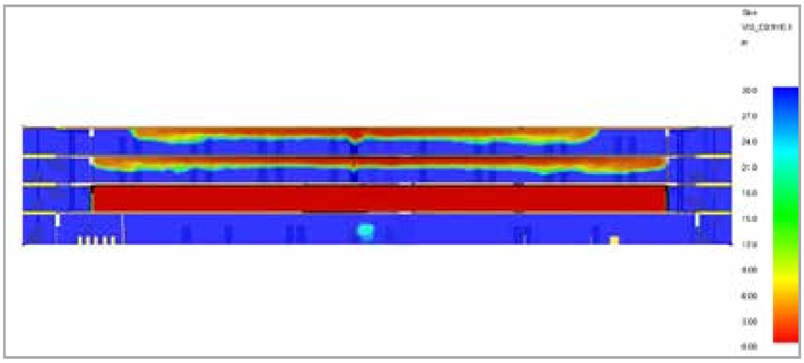


*Fig. 5 - Visibility values in the building corridors at 420 seconds from ignition*

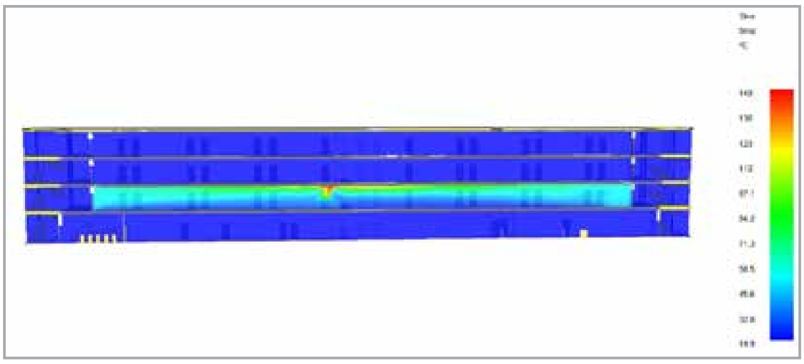


*Fig. 6 - Temperature values in the building corridors at 420 seconds from ignition*

| EVACUATION SIMULATION | | |
|---|---|---|
| Total number of people | 274 |  |
| People type distribution | Adults, 20% over 60, 1 person with disabilities per floor | |
| People gender distribution | 50% male, 50% female | |

| PROFILES | hotel guests | restaurant guests | meeting rooms guests | staff |
|---|---|---|---|---|
| type | adults, elderly, people with disabilities | adults, elderly, people with disabilities | adults, elderly, people with disabilities | adults |
| gender | male, female | male, female | male, female | male, female |
| ages | 15-60, over60 | 15-60, over60 | 15-60, over60 | 15-60 |
| people with disabilities | 1 per floor=3 | 1 | 1 | 0 |
| familiarity | not familiar | not familiar | not familiar | familiar |
| training | not trained | not trained | not trained | trained |
| awake/asleep | asleep/awake | awake | awake | awake |
| social grouping | individuals, couples | individuals, couples | individuals | individuals |

*Fig. 7 – Population distribution and occupants' profiles*

- a FED index between 0.3 and 1,
- visibility close to 0 m,
- a temperature of 57.3 °C.

A FED index of 1 is considered to cause incapacitation for 50% of the population, while a value of 0.3 is a common threshold for acceptable conditions. In the simulations, incapacitation was assumed to be lethal because the inability to escape would lead to an increased dose of toxic gases being inhaled over time. For more vulnerable occupants, Italian regulations suggest a conservative value of 0.1. Low visibility and high temperatures also reduce the possibilities of successful evacuation.

The results of the fire simulations were coupled with the evacuation simulations using Pathfinder, a commercial simulator for emergency evacuations. The behavior of the occupants is set to move towards the nearest available exit; it is not possible to use the elevators. When visibility in the internal staircase is compromised by smoke, the occupants are prevented from accessing the staircase. Occupants with disabilities move towards the lateral terraces and wait to be rescued. It is important to set the response profile of the agents according to their characteristics such as age, gender, state (awake/asleep), familiarity with the building, existence of disabilities, etc. as this affects their evacuation and response times.

*Fig. 8 - Evacuation through the lateral staircase of the building*

According to the simulation results, all the occupants on the floor where the fire originates are considered to be fatalities, because conditions become untenable before they detect the fire and begin evacuating. This does not happen with the occupants on other floors, who have sufficient time to escape. Such a severe outcome is due to the high reaction time set for occupants, considered to be asleep when the fire starts. Regulations such as the British Standard PD 7974-6, the ISO 16738, and the Italian Norme Tecniche di Prevenzione Incendi suggest a pre-evacuation time of 20 minutes, referring to one percentile of evacuees, for occupants that are asleep and unfamiliar with the building.

## Conclusion

Fire and evacuation simulations were used in combination with a statistics-based FRA to evaluate life-safety in the event of a fire in a hotel. Quantitative fire risk assessment is a tool to support the selection of fire scenarios and to associate them with an approximate probability of occurrence. The most frequent fire scenario according to statistics is simulated (a bedroom fire). The most dangerous conditions are determined according to the location in which the fire started, the spread of the fire, the products of combustion, and the compartmentalization (open room door).

The availability of innovative tools and methodologies along with the increasing knowledge of fire phenomena are compelling reasons to apply Performance-Based Design in common practice. Numerical simulations are a key aspect of the digital transformation process that should be integrated into Building Life Cycle Management.

*Ada Malagnino is a Research fellow at the University of Salento,*
*Simo Hostikka is a Professor at Aalto University,*
*Giorgio Zavarise is a Professor at the Politecnico di Torino,*
*Angelo Corallo is a Professor at the University of Salento.*

For more information:
Ada Malagnino - University of Salento
ada.malagnino@unisalento.it

# Enabling safe and efficient human-robot collaboration across Europe with the ROSSINI project

Rossini
RObot enhanced SenSing, INtelligence and actuation to Improve job quality in manufacturing

## Test cases to ensure broad adaptability and applicability across industry sectors

**By Riccardo Masiero[1], Cristian Secchi[2], Hornicak Miroslav[3], Federico Filzi[4], Alessandro Carapia[5], Giulia Di Bari[1], Matteo Zanaroli[6]**
1. Crit Srl - 2. University of Modena and Reggio Emilia (UNIMORE) - 3. Whirlpool EMA Spa 4. Schindler Elettronica SA
5. IMA Industria Macchine Automatiche Spa - 6. Datalogic IP Tech Srl

*The ROSSINI (RObot enhanced SenSing, INtelligence and actuation to Improve job quality in manufacturing) project aims to design, develop, and demonstrate a modular and scalable platform for the integration of safe human-centered robotic technologies in industrial production environments [1]. This will be achieved both by developing innovative technological components and methodologies in all fields related to collaborative robotics (sensing, control, actuation, human aspects, risk assessment methodology), and by integrating all such components in an open platform ensuring quick ramp-up and easy integration. Increasing the sensing capability, intelligence, reaction capacity and the communication skills of robots, ROSSINI will bring together the best of industrial and collaborative robots.*

Industrial robots are highly efficient at repetitive, heavy tasks but will never attain the flexibility of humans in manipulating, inspecting and quickly adapting to new situations. In some cases, the collaboration of robots and humans is necessary to optimize efficiency and job quality [2], [3]. Current collaborative robots are safe, but extremely slow and often lack the flexibility of a human co-worker [4]. In the ROSSINI H2020 project, 13 partners with diverse expertise joined together to define a scalable Human Robot Collaboration (HRC) platform in which task allocation, safety functions and working parameters can be easily adapted according to production requirements and to the needs of specific human operators. These partners are working to improve individual sensors, actuators, controls, and communications, to simplify the design of efficient applications, and to make them safer and more ergonomic.

### ENABLING TECHNOLOGIES

As reported in [5], within ROSSINI, a set of high-performance safety sensors will ensure the best detection of the working area for the robot and the best safety levels for the operator. Different technologies like 3D cameras, laser scanners and radar will be used together to suit different applications. Working together, these solutions will actualize the "ROSSINI Smart and Safe Sensing System" ($RS^4$) acronym.

Flexible, AI-based algorithms will leverage the acquired data to optimize the efficiency of the robot and dynamically adapt it to the operator (e.g., semantic map and ROSSINI controller). Quality of work parameters will be used not only in the design phase of the collaboration area, but also in the execution phase to minimize stress and improve ergonomics.

A robotic manipulator purposely designed to be innovative and collaborative ("collaborative by birth") will greatly improve the ratio of speed to safety. In addition, a set of innovative interfaces will be selected and implemented to improve human-robot communication.
New 3D tools and risk assessment procedures will simplify the working cell design.

### ADVANTAGES AND CRITICALITIES

The fusion of different sensing technologies, the introduction of advanced algorithms for control, and the development of a robot arm specifically designed for efficient collaboration will increase the number of possible applications.

The solutions developed in ROSSINI will be tested in three different use cases by March 2022. While these three use cases already cover a large range of possible HRC applications, many of the technologies developed in the project will also be applicable to other industrial fields. For example, 3D safety cameras or radars could be used to protect non-robotic industrial machinery; and AI supervision and control algorithms can be used on automatic guided vehicles (AGVs). The list of challenges to be addressed is very long: a secure, real-time reconstruction of the working environment based on sensor fusion is extremely difficult. Creating control algorithms that can use this data flexibly is no less challenging. Making this a complete, scalable system that is easy to adapt to all applications and certify for safety is an additional and ambitious undertaking.

## THE HUMAN FACTOR

The holistic approach of the ROSSINI project will enable the construction of HRC systems with significantly higher performance, output quality, and job quality for the human operators. In addition, the high flexibility of the platform and design tools will enable truly short reconfiguration times and lower costs in case of product changes.

This will help to maintain the production facilities and employment in areas with a high cost of labor, such as almost all European countries. The ability to dynamically adapt the robot's work to the operator based on the operator's skills and requirements will not only increase the quality of work but will allow more people to do that work, such as enabling female workers to do activities currently performed only by men.

## USE CASE DESCRIPTIONS

The three ROSSINI use cases are:

- domestic appliance assembly at Whirlpool;
- electronic component production at Schindler; and
- food product packaging at IMA.

These three use cases represent a heterogeneous set of applications that will allow the flexibility and wide applicability of the ROSSINI platform to be tested; the heterogeneity is described in the following paragraphs which present each use case in terms of its objectives, challenges and planned solutions.

The Whirlpool use case addresses a pick-and-place application for counterweights. The counterweights must be placed in two different types of washing machines. In the first type (Fig. 1), the counterweight (~10 kg) must be placed at the bottom of the washing machine being assembled on a "Stop & Go" production line. In the second type (Fig. 2), the counterweight (up to 14 kg) must be placed at the top of the washing machine being assembled on a "continuous" production line.

***Objectives of the use case:***

- Improving job quality and ergonomics
- Maintaining the task times of the production lines
- Realizing an adaptable solution

*Fig. 1 - Whirlpool use case: placing of counterweight in a "Stop & Go" production line*

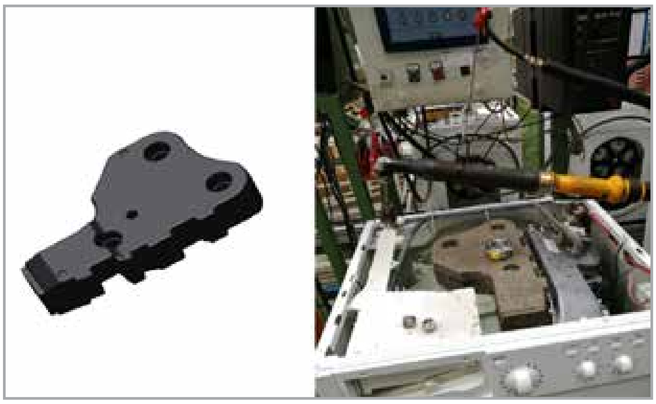


*Fig. 2 – Whirlpool use case: placing of counterweight in a "continuous" production line*

***Challenges of the use case:***

- Heavy payload (up to 40 kg with robot gripper)
  Task times to be respected (constraints)
- Redefinition and optimization of task assignment between human and robot (based on well-defined job-quality metrics and corresponding parametrization)

***Planned solution:***

Considering the robotic solution identified for this scenario, the main peculiarity of this use case is the adaptation of a non-collaborative commercial robot to a collaborative mode thanks to RS[4].

The Schindler use case addresses an optimization problem related to the kitting out and assembly of an elevator panel (Fig. 3) that is currently manufactured using a "lean certified" manual process.

*Fig. 3 - Schindler use case: elevator panel to be assembled together by human and robot (illustrated concept)*

***Objectives of the use case:***

- Optimizing the layout of the production line
- Increasing efficiency and productivity
- Improving job quality and ergonomics

***Challenges of the use case:***

- High customization of final products (wide variety of layout combinations)
- Close human-robot interaction
- Need for highly precise task assignment (between human and robot)

***Planned solution:***

In this use case, the solution envisioned is a complete redefinition of the current work method (both the physical space for the tasks and the sequencing). In this scenario, the ROSSINI partners are designing a totally new robotic arm that will be natively collaborative (i.e., a "collaborative by birth" arm) both in terms of appropriate speed and reach of the arm, as well as the safety and interaction features.

Finally, the IMA use case tackles an operational scenario involving a Tea Machine (Fig. 4). More specifically, in this use case a mobile platform supports human operators in loading and handling packaging material.

*Fig. 4 – IMA use case: example of the manual loading of packaging material into a Tea machine*

***Objectives of the use case:***

- Improving job quality and ergonomics
- Increasing efficiency and productivity
- Automatic recovery from stoppages

***Challenges of the use case:***

- Human-robot interaction in a free space
- Ensure a sufficiently high speed of the mobile robot in the moving space (including obstacles)
- Reliable human recognition (to allow the robot to improve its performance)

***Planned solution:***

In this use case the planned solution involves a pair of robots: an AGV and a collaborative off-the-shelf robotic arm. Here, the ROSSINI solution will ensure the collaborative mode of the pair of systems and increase the performance (speed, detection, efficiency) to the operative maximum achievable.

## About the Rossini Project

ROSSINI is an EU-funded project under the H2020 Program, DT-FOF-02-2018. It lasts 42 months from 10/01/18 to 04/01/22. The ROSSINI vision is the creation of a new generation of robotic applications that will be able to cooperate with humans at never before reached levels, thus contributing importantly to production sustainability in European factories.

Partners involved: Datalogic Srl (Lead partner), PILZ GmbH & Co. KG, Università degli studi di Modena e Reggio Emilia (UniMoRe), Innovacio I Recerca Industrial I Sostenible SL (IRIS), Scuole universitaria professionale della Svizzera italiana (SUPSI), Nederlandse Organisatie voor toegepast-natuurwetenschappelijk onderzoek (TNO), Fraunhofer-Gesellschaft zur Förderung der angewandten Forschung e.V. (Fraunhofer IFF), VINTIV, Crit Srl, Core Innovation and Technology OE, Whirlpool EMEA Spa, Schindler Elettronic SA, IMA Spa.

## CONCLUSIONS

The ROSSINI project aims to design, develop, and demonstrate a modular and scalable platform for the integration of human-centered robotic technologies in industrial production environments. The project targets three heterogenous use cases that will demonstrate its broad applicability in terms of industry sectors, adopted solution and end user. The final results of the ROSSINI project are expected by March 2022 and the corresponding technologies aim to be easily applicable in different industrial fields, even beyond those demonstrated within the project.

### References


[1] C. Franklin, "Take a safe approach to collaborative robots," ISA InTech Mag. 2017, 2017.

[2] OECD, "How Good is Your Job? Measuring and Assessing Job Quality." European Foundation for the Improvement of Living and Working Conditions, 2017.

[3] R. Behrens, J. Saenz, and N. Vogel, Christian Elkmann, "Upcoming Technologies and Fundamentals for Safeguarding All Forms of Human-Robot Collaboration," in 8th International Conference Safety of Industrial Automated Systems (SIAS 2015), 2015.

[4] R. Behrens, N. Elkmann, and H. J. Ottersbach, "A Contribution for Standardized Risk Assessment: Examination of Constrained and Unconstrained Human-Robot-Collisions," in IEEE/RSJ International Conference on Intelligent Robots and Systems (IROS 2012), 2012.

[5] Rossini, "Deliverable D2.1 ROSSINI State of the Art Analysis," Dec. 2018.


For further information, visit:
www.rossini-project.com or contact
Matteo Zanaroli, Fund Raising Manager at Datalogic IP Tech Srl, matteo.zanaroli@datalogic.cm

# Smart Spaces Safety and Security for All Cities

Public city spaces, such as malls, open gathering areas, and transport infrastructures are vulnerable to terrorist attacks strongly impacting the safety, prosperity and wellbeing of citizens. The digital transformation we are undergoing, despite its countless benefits, leaves us exposed to cyber-attacks. Several European cities have recently experienced both physical and cyber-attacks, and unfortunately these events are likely to continue in the foreseeable future.

The Smart Spaces Safety and Security for All Cities project (S4AllCities) is a large-scale project funded by the European Commission's Horizon 2020 Research and Innovation Programme, with the aim of revolutionizing the way smart cities become more prepared for and resilient against physical and cyber attacks on their soft targets, smart spaces and critical infrastructure. This is achieved through augmenting situation awareness in city spaces with greater intelligence, machine-encoded context knowledge, and real-time evaluation of cyber and physical security threat levels.

With a total budget of €9.7 Million and a duration of 24 months (from 1 September 2020 to 31 August 2022), the project brings together a consortium of 28 partners from nine EU countries, comprising leading European research and academic institutions, SMEs from the software and security domains, and end-users, including City authorities, Law Enforcement Agencies and Transport Operators.

The S4AllCities approach centers around three modular yet interconnected digital twin sub-systems designed to ingest large amounts of data from edge-computing sensors (such as a portable near-infrared spectrometer for detection of chemical precursors to explosives, or fiber Bragg-grating based sensors for real time monitoring of city infrastructure) deployed around the Smart Cities and which appropriately fuse the information received to establish recommended courses of action and present timely, concise, actionable information to the relevant operators. For example, real time traffic information and simulations run on the digital twins can yield recommendations on evacuation routes in the event of a fire in a train station.

The S4AllCities functionality is achieved by means of a large array of AI-based tools analyzing on the massive amounts of data received from the sensor networks, as illustrated in the figure. These networks comprise multiple innovative sensors further

advanced through the project activities and complemented by smart city legacy sensor networks. The S4AllCities System of Systems (SoS) is completed by both physical- and cyber-security shielding using appropriate safety and anonymization standard methods.

The integrated S4AllCities SoS will meet the following project objectives:

- To complement legacy monitoring systems by adapting state-of-the-art, low-cost technologies and solutions that enhance Smart City preparedness and response capacity in both the cyber and physical space.
- To design and develop an open platform for sharing and managing information, while providing intelligence with unprecedented situation awareness and decision support and enhancing European city resilience in full compliance with citizens' fundamental rights and privacy.
- To design and develop an intelligent communications architecture that ensures the interconnection and integration of city smart systems, while supporting security operators.
- To significantly impact collaboration among smart cities' stakeholders while engaging citizens in the move towards more secure and safer cities.

The S4AllCities SoS will be validated in three European Smart Cities: Trikala (GR), Bilbao (ES) and Pilsen (CZ), where the system will be installed on a pilot/demonstration basis and run continuously for three months. The effectiveness and advantages of the S4AllCities SoS will be demonstrated to a larger audience of stakeholders using different scenarios involving physical and cyber-attacks on the smart cities' soft targets (such as government buildings, public spaces, crowds or transport infrastructure).

The demonstration events will show-case the monitoring of key assets and measurements for the smart city's infrastructure (traffic, access to restricted areas, ground and airborne video streams, evacuation routes etc.); the detection of explosives, cyber-attacks, suspicious objects or activities; and the smart alerting mechanism based on context-specific criticality levels – all with a comprehensive user interface.
It is expected that the system will achieve a TRL-7 by the end of the project, and that the pilot activities will showcase the S4AllCities functionality to the relevant end-users and stakeholders.

- The Trikala (GR) pilot revolves around the protection of an autonomous bus infrastructure, the park hosting the Trikala Christmas Festival, and municipal buildings.

- The Bilbao (ES) pilot focuses on the protection of crowds during massive gatherings for festivities (such as the "Great Week" Festival) in the city center and the metro station.

- The Pilsen (CZ) pilot simulates a bomb attack followed by the evacuation of the FC Viktoria Plzen stadium.

To learn more about S4AllCities, please sign up for our newsletter on https://www.s4allcities.eu or email info-s4allcities@exus.co.uk. You can also follow the project's latest news on social media at:

- https://twitter.com/s4allcities
- https://www.linkedin.com/in/s4allcities-project/
- https://www.facebook.com/s4allcities/

*This project has received funding from the European Commission's H2020 Research and Innovation Programme, under Grant Agreement No. 883522.*

For more information:
Marisa Zanotti - EnginSoft
m.zanotti@enginsoft.com

# Ansys 2020 R2 improves collaboration and information sharing between engineering teams

## Cutting-edge CFD advancements deliver highly accurate solutions without compromise

**By Michele Andreoli, Alessandro Arcidiacono and Diana Magnabosco**
EnginSoft

*Ansys 2020 R2 is geared towards accelerating innovation in any environment by improving collaboration and information sharing between engineering teams. This article describes the major new features in the latest release with an overview of each software from SpaceClaim and Fluent Meshing, to Fluent and CFX. Every step of fluid dynamic simulation has been enriched by powerful workflows to deliver a streamlined user experience and improved functionality.*

### Ansys Space Claim Direct Modeler

Space Claim is a CAD modeler that allows the user to create, edit and repair any geometry with ease and flexibility. In this latest release, the main improvements and additions use replicable procedures to achieve automation and robustness.

A useful new feature called Block Recording allows changes to be tracked within SpaceClaim while preparing for simulation. All operations can now be registered within an intuitive tree structure, thus combining direct editing with automation. This approach supports parametric and unplanned changes, improving the bidirectional connection between SpaceClaim, Workbench and the CAD. This new capability augments the power of SpaceClaim because it now includes both a "feature-based" modeler and a "history-free" modeler.

There is a new dedicated tab for sketching that allows constraints to be added easily so that more complex sketches can be made. For example, it is possible to force curves to be vertical, horizontal, fixed, coincidental, tangential, etc.

Face splitting and naming in the Volume Extract procedure have been improved: split faces can now be retained between the solid and the fluid domains and named selections can be duplicated onto fluid bodies. This reduces the need to imprint after a modeling operation and better supports parameterization during conjugate heat transfer (CHT) simulations.

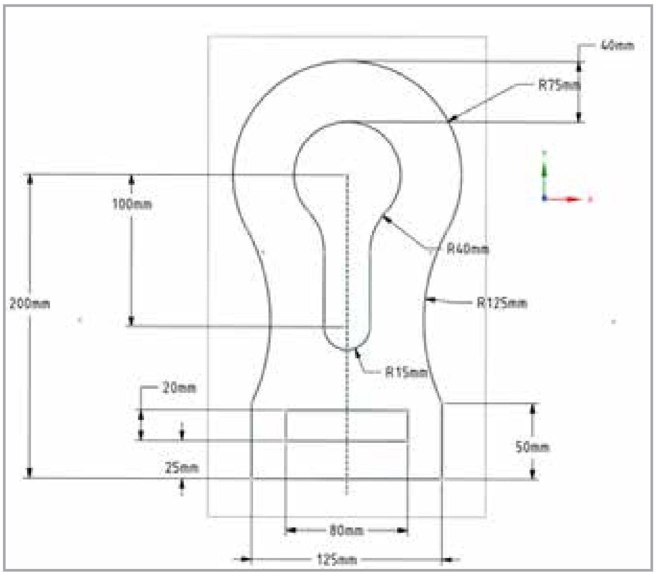



*Fig. 1 – Constraint-based sketching*

The passage of the named selection (NS) into the solver has been facilitated by the addition of two operations: detect and repair overlapping selections. If the NS overlap or are void, it is now possible to solve any discrepancy automatically.

### Ansys pre-processing with Fluent Meshing

Ansys Fluent Meshing organizes the mesh generation process with a task-based workflow that offers a streamlined end-to-end wizard to reduce the time to the solution. Depending on the starting geometry, two different workflows can be used: Watertight Geometry Workflow (for clean geometries) and Fault Tolerant

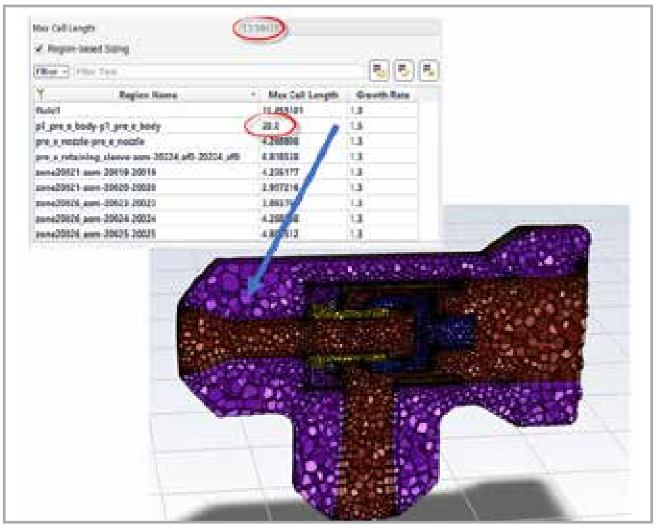


*Fig. 2 – New option to assign a region-specific size in the Generate Volume mesh panel*

Meshing Workflow (for problematic and complex geometries). In the latest release, the sequence of tasks has been enriched with more options and possibilities, increasing usability and flexibility. In addition to the Filter Text option previously available, the Meshing Workflows now have an option to use a persistent Wildcard string for selecting labels or zones. The Use WildCard option stores the wildcard string itself in recorded workflows instead of in an explicit list of locations. This means that when the workflow is played back with new geometries, the matching is performed again to pick up any matching zones or labels. Repetitive operations can thus be avoided because the same settings can be used on other geometries that have the same topology or analogous shapes.

Additional flexibility is now available in the Watertight Meshing workflow to provide more control of sizing for certain geometric features and regions. Firstly, there is the ability to specify local sizing scoped to edges that are defined in Named Selections from SpaceClaim. Region-specific Maximum Size and Growth Rates (for poly and tet) can also be specified during the Volume Meshing task. The Watertight workflow can now be started by importing an existing surface mesh instead of starting from CAD. This is useful if there is an established surface-meshing procedure, or if there is already a generated mesh (from another preprocessor or an existing Fluent case) that can be used as a starting point for Fluent Meshing.

The Fault Tolerant meshing workflow has been enhanced with the Hexcore and Poly-Hexcore methods for surface-meshed and wrapped solid regions so that it is now possible to leverage all of the available Volume meshing technologies for solids, whether they are wrapped or surface-meshed. Finally, in Fault-Tolerant Meshing Workflows the construction of prism layers within poly and poly-hexcore fluid regions can be managed with more precision. It is possible to use the stair-stepping method, which removes prism layers locally in steps to avoid collisions in the mesh, as well as poor quality meshes at sharp corners. Alternatively, you can create continuous prism layers without stair-stepping, thus avoiding undesirable behavior.

## Ansys Fluent: User Interface, Graphics and Post

Fluent 2020 R2's Preferences dialog box offers greater flexibility in assigning mouse buttons. There is a new consistent set of default assignments between Meshing and Solving, and these default assignments are also more consistent between SpaceClaim and other Ansys products. Setting up multiphase boundary conditions no longer requires the opening of multiple dialog boxes: a drop-down menu is provided to select between the Mixture and the individual phases.

The mesh display no longer renders edges by default, which improves the performance and the appearance, especially of dense industrial meshes. The editing of colormaps is also easier

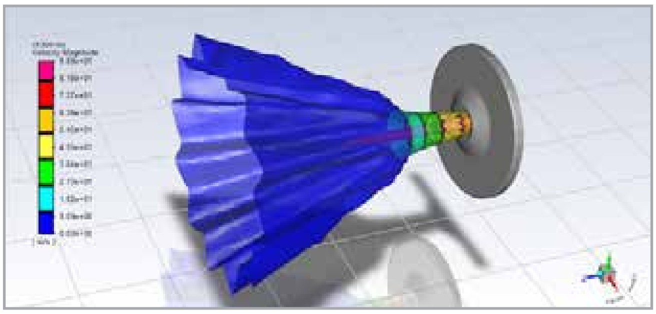


*Fig. 3 – New Colormap editor*

with clearer stop definitions and a new Color Picker dialog. Many new colormaps are installed by default, including high-contrast and banded variants, and the default values for number of contour levels and labels have changed. For unsteady simulations, a new framework is available for computing the Mean and RMS field variables.

## Ansys Fluent: Expressions and Physics Modeling

Expressions have been introduced to patch the initial field solution and they can also be used for a number of material property definitions which provides additional flexibility in defining material properties that depend on local conditions like Pressure, Temperature, Viscosity, or Strain Rate without writing User Defined Functions (UDFs). There is a new diagnostic capability in the Expressions Tab to plot the expression values for up to two independent variables. Species / Phases can be selected directly, and Locations can be selected / entered from the Locations drop-down box. A new Implicit location is supported for in-line expressions entered directly in the Boundary Zone and Cell Zone dialogs. Expressions can be marked as Output Parameters directly (without creating corresponding Report Definitions).

Fluent 2020 R2 includes a new modeling approach called GENTOP (Generalized TwO Phase), which provides a modeling solution for multiphase transitional flows when intermediate interfacial scales are present such as those that occur in slug, churn, and annular flows. In addition to a primary continuous phase and one or more polydisperse secondary phases, GENTOP includes a further phase that can behave either as continuous or dispersed depending on the phase volume fraction and a critical bubble diameter.

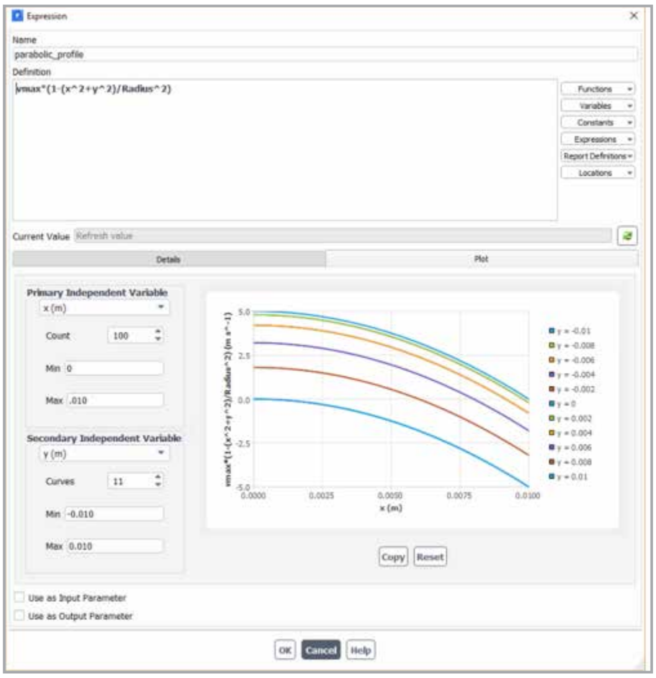


Fig. 4 – Plotting of Expressions

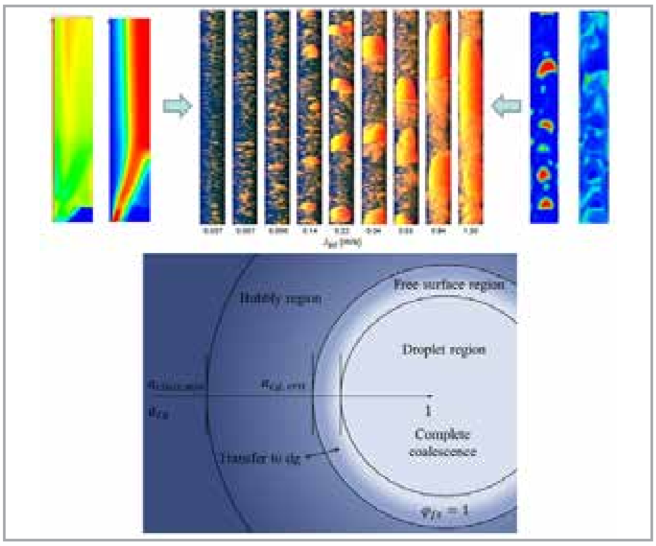



Fig. 5 – GENTOP Concept

For gas-solid flows, a Filtered Two-Fluid Model and Energy Minimization Multi-Scale (EMMS) drag model have been introduced that include sub-grid scale effects on the macroscopic flow for applications like large-scale fluidized beds.

A modified Body-Force Weighted formulation is available for Volume of Fluid (VOF) that attempts to remove some of the limitations (such as inaccuracy in high viscous and rotating flows) of standard Body-Force Weighted methods and provides better robustness and convergence.

For simulations of spray breakup, particularly like those found in gas turbine fuel injection, using the Discrete Particle Method, the Schmehl droplet breakup model (which was already available in CFX) is now also available in Fluent. This model is somewhat similar to the Madabhushi model (that was previously included in Fluent) but considers additional breakup regimes.

The General Turbo Interface that was introduced for turbomachinery simulations has been extended with support for all species models with any type of interface. In addition, Semi-Implicit Method for Pressure Linked Equations-Consistent (SIMPLEC) with segregated Pressure-Based Navier Stokes (PBNS), which is typically recommended for combustion simulations, is now available for Frozen Rotor and Transient Rotor-Stator simulations.

The Monte Carlo radiation model adds additional capabilities for internal boundary sources as well as new postprocessing fluxes. S2S files can now be written in a file format based on the Common Fluids Format.

The Ansys Granta Materials Data for Simulation (MDS) product provides material data for more than 700 solid materials. If you have a license for Ansys Granta MDS you can now load these material properties into Fluent from the Fluent materials dialog box.

In 2020 R2, a new setup dialog box for Battery Modeling provides a more straightforward and consistent method for choosing the appropriate simulation method. Fluent provides a variety of strategies for thermal analysis of battery modules and packs ranging from simple CHT-only analysis to the sophisticated Multiscale / Multidomain method. A general improvement now allows a more realistic characterization of battery performance including the effects of deterioration over time and after many use cycles.

When using the Design Tool in the Adjoint Solver, you can now examine both the initial and the optimized geometries after multiple design iteration loops. This makes it easier to see how your design has evolved through successive design optimizations. The Standard Tessellation Language (STL) export tool in the Adjoint Design tool has been extended to export iso-surfaces in addition to boundary surfaces.

## Ansys Fluent: Solver Enhancements

For simulations of electric motors, and rotating wheels in aerodynamic simulations, etc. scalability has been improved through enhancements to the wall-distance calculation in Fluent. Depending on mesh size and core count, performance can be improved by 25% or more compared to 2020 R1. Fluent 2020 R2 includes a new Incomplete Lower Upper (ILU) decomposition smoother based on a collaboration with Intel. This can yield performance improvements of 10%-15% when using the coupled solver. In the Numerics area, a new Correction form can be used to solve the pressure-based solver equations; this can improve robustness by reducing the accumulation of errors, which also allows more simulations to be run in Single Precision (thereby saving memory), and offers a modest performance improvement with the coupled solver. Furthermore, the multiple reference pressure formulation (introduced as an option in 2020 R1) is now used by default. This provides benefits in cases that involve multiple disconnected fluid regions where different reference pressures are appropriate, or where one region has a pressure boundary condition.

## Ansys CFX for turbomachinary

ANSYS is constantly developing new capabilities that help customers to accurately simulate turbomachinery. ANSYS CFX is the industry-leading computational fluid dynamics (CFD) software for turbomachinery applications, known for its outstanding accuracy, robustness, and speed. CFX is easy to use, yet extremely powerful and accurate for different types of turbomachinery including pumps, fans, compressors, and gas and hydraulic turbines. It also offers highly scalable, high-performance computing (HPC) to help solve simulations quickly and cost effectively. With 2020 R2, CFX can be coupled to GT-SUITE at flow boundaries through an intuitive workflow that allows mapping and auto-setup of boundaries, species, and materials.

The Blade Film Cooling model provides an efficient and practical way to model an array of cooling holes and the injected film cooling flow on turbine blades. The model does not require the mesh to resolve the holes and is fully compatible with periodic and moving boundaries. Recently a volumetric injection option was added in which injection is defined by a volumetric region of influence. Local flow direction angles can now also be specified as part of the injected flow quantities. Improvements in 2020 R2 focus on creating injection locators directly in CFX-Pre: in addition to reading locators from a file, injections can now be created specifying cartesian, cylindrical or turbo coordinates. Then, the locators can be visualized together with the specified injection distribution functions: uniform, quadratic polynomial or as a Common Expression Language (CEL) expression.

The 2020 R2 of CFX includes the GEneralized K-Omega (GEKO) turbulence model. GEKO is a revolutionary concept in turbulence modeling that gives you the flexibility to tailor the model to your

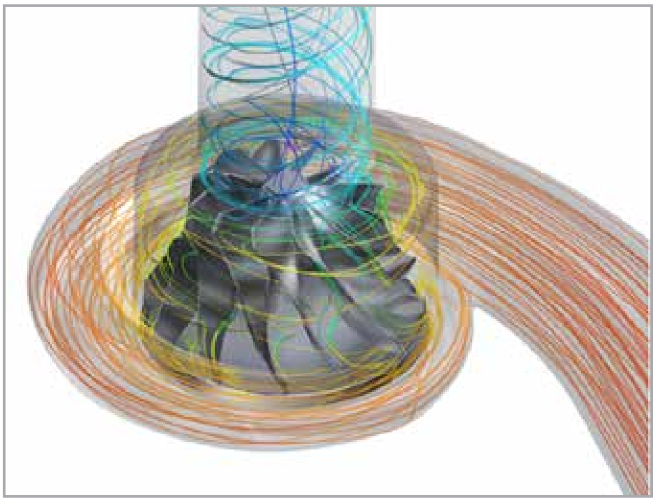


*Fig. 6 – Centrifugal compressor coupled with GT-SUITE*

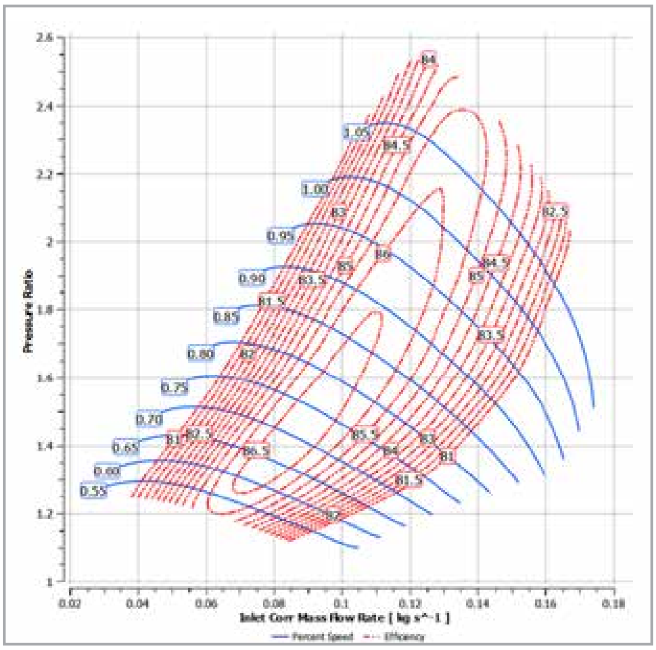



*Fig. 7 – Operating Map*

applications. Developed by turbulence expert, Dr. Florian Menter, GEKO provides four tunable coefficients — separation, near wall, mixing and jet — which can be adjusted to different flow regimes.

This latest release of CFX allows you to create complete operating maps with a simple integrated process. The performance map is the result of your analysis. Using a single performance map format file, CFX generates all necessary results as if they were a single simulation run. It then organizes them into a master folder for unified analysis.

With the GPU-accelerated animations for Transient Blade Row (TBR) results, there is no need to wait for transient files to load or for animation encoding. TBR results can now be post-processed live in Ansys CFD-Post using GPU-accelerated animations. This works for all of the following:

- Single row blade flutter and inlet disturbance cases
- Single disturbance harmonic analysis solutions
- Single-stage TBR cases using Fourier Transformation or Time Transformation pitch change

A new Droplet Condensation (Non-equilibrium Steam) Model has been implemented, with an improved surface tension formula based on the gas temperature, and a new tunable droplet growth model based on Young's equation. These models improve the average droplet size distribution and Wilson point pressure rise prediction due to nucleation.

With regard to post-processing, with the latest release, CFD-Post now supports Fluent Common File Format files, notably the cas.h5 and dat.h5 extensions.

For more information:
Diana Magnabosco - EnginSoft
d.magnabosco@enginsoft.com