ENGINSOFT

The Simulation Based Engineering & Sciences Magazine

Newsletter

Year 18 n°2

Summer 2021

Powering the next generation of off-highway suspension design

Using Simulation and Modeling for medical Radio Frequency design

Lubrication and heat dissipation in transmissions and bearings

Optimally design an artificial lung for extracorporeal life support

ON SITE & ONLINE

automotive CAE GRAND CHALLENGE

October 19 – 20, 2021
Congress Park Hanau
Germany

// BODY STIFFNESS & STRENGTH: MODELING OF CONNECTIONS

// CAE PROCESS & QUALITY: COMPARING TEST AND SIMULATION

// NOISE VIBRATION HARSHNESS: VARIABILITY ANALYSIS

// OCCUPANT SAFETY: ROBUSTNESS/SCATTER OF OCCUPANT SIMULATION – INCLUDING OOP

// DURABILITY: VIBRATION FATIGUE OF HV-BATTERIES

// CRASH: MATERIAL MODELS FOR BATTERY PACKS OF ELECTRIC VEHICLES

// FULL VEHICLE SIMULATION: LOAD CASES FOR RELEASE OF AUTOMATED DRIVING FUNCTIONS

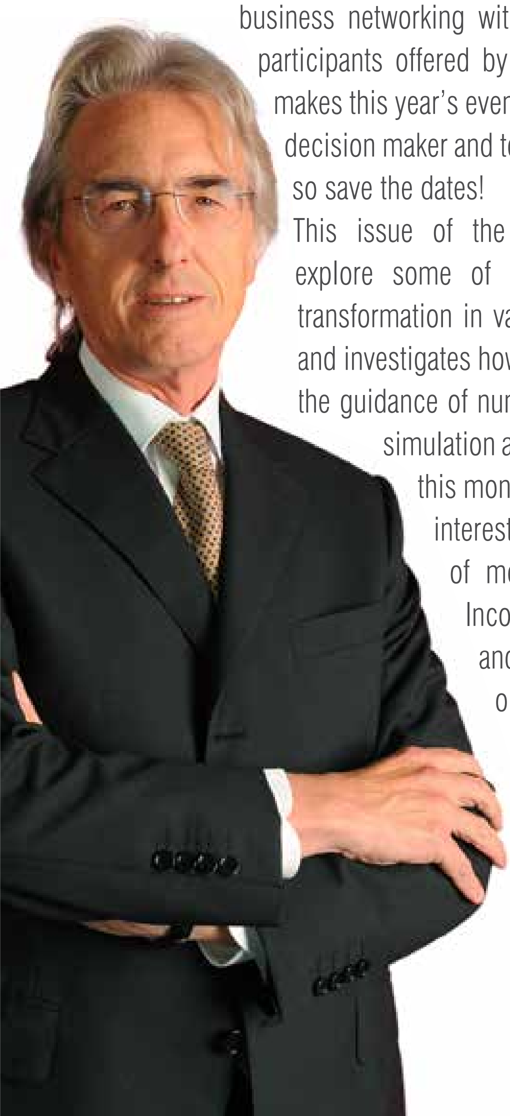
carhs
Empowering Engineers

www.carhs.de/grandchallenge

Flash

Digital transformation continues its march as the trend across all spheres of business and industry, particularly in light of the massive thrust in momentum it was given by the global pandemic lockdowns and impelling need for organisations to offer continuity in service and product delivery. As such, therefore, it will be the focus of the 2021 International CAE Conference and Exhibition, to be held as a hybrid event, ie. both online and in person at the Vicenza Fair in northern Italy from 17-19 November this year.

The event will focus on all aspects of digital transformation across industry, as well as all the interconnected technologies and software affected by and involved in an enterprise-wide digital transformation project, focusing on the key coordinating and guiding role that engineering simulation technologies can play in achieving such strategies successfully and efficiently. The unique opportunities for interaction and business networking with speakers and among participants offered by the new hybrid format makes this year's event even more essential in decision maker and technologist's calendars, so save the dates!



This issue of the Newsletter begins to explore some of the aspects of digital transformation in various industrial sectors and investigates how it is being tackled with the guidance of numerical and engineering simulation approaches. The cover of this month's edition concerns an interesting instance of the use of modeFRONTIER by Dana Incorporated to automate and validate the multi-objective optimization process for a new independent suspension axle of an off-highway vehicle. In this instance, modeFRONTIER facilitates the integration of several different tools used in Dana's engineering department.

Also not to be missed are the articles by Comer Industries concerning lubrication and heat dissipation in transmissions and bearings; an analysis by TSNE of the behavioral characteristics of detergent powder using a Rocky-Ansys Fluent two-way coupling; the study of a new approach to aerodynamic design created in collaboration between Dallara Automobili and RBF Morph that exploits the synergies between adjoint methods and mesh morphing to accelerate the optimization process; and a medical CFD study of hollow fiber membrane oxygenators (HFMO) historically used to provide temporary oxygen support to cardiopulmonary bypass patients during surgery, and today increasingly used for longer term therapies, such as acute respiratory distress syndrome and chronic obstructive pulmonary disease, to better understand the natural blood coagulation process that diminishes oxygen transfer and reduces the pressure, ultimately causing the oxygenators to fail, in order to accelerate design exploration to find an optimal solution.

As you can see, this edition highlights the breadth and depth of application of numerical and engineering simulation and the myriad and innovative ways it is being used to achieve digital transformation, both at the micro and the macro level. I hope you find it as interesting a read as I did and that it whets your appetite to attend the CAE Conference in November, where similar and other application instances, case studies, new technologies, ancillary and complementary software, and numerous fascinating research projects will be available to explore in greater detail. I look forward to seeing you there!

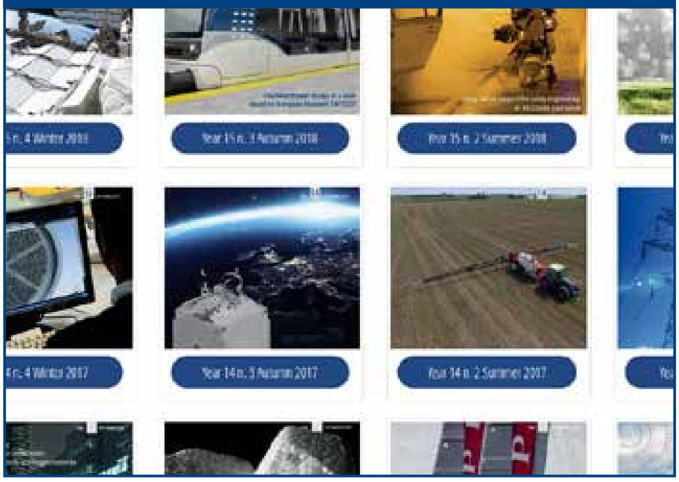
Stefano Odorizzi
Editor in chief

Contents

Case Studies

Research & Technology Transfer

PAST ISSUES at
www.enginsoft.com/magazine



Newsletter EnginSoft
Year 18 n°2 - Summer 2021

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 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

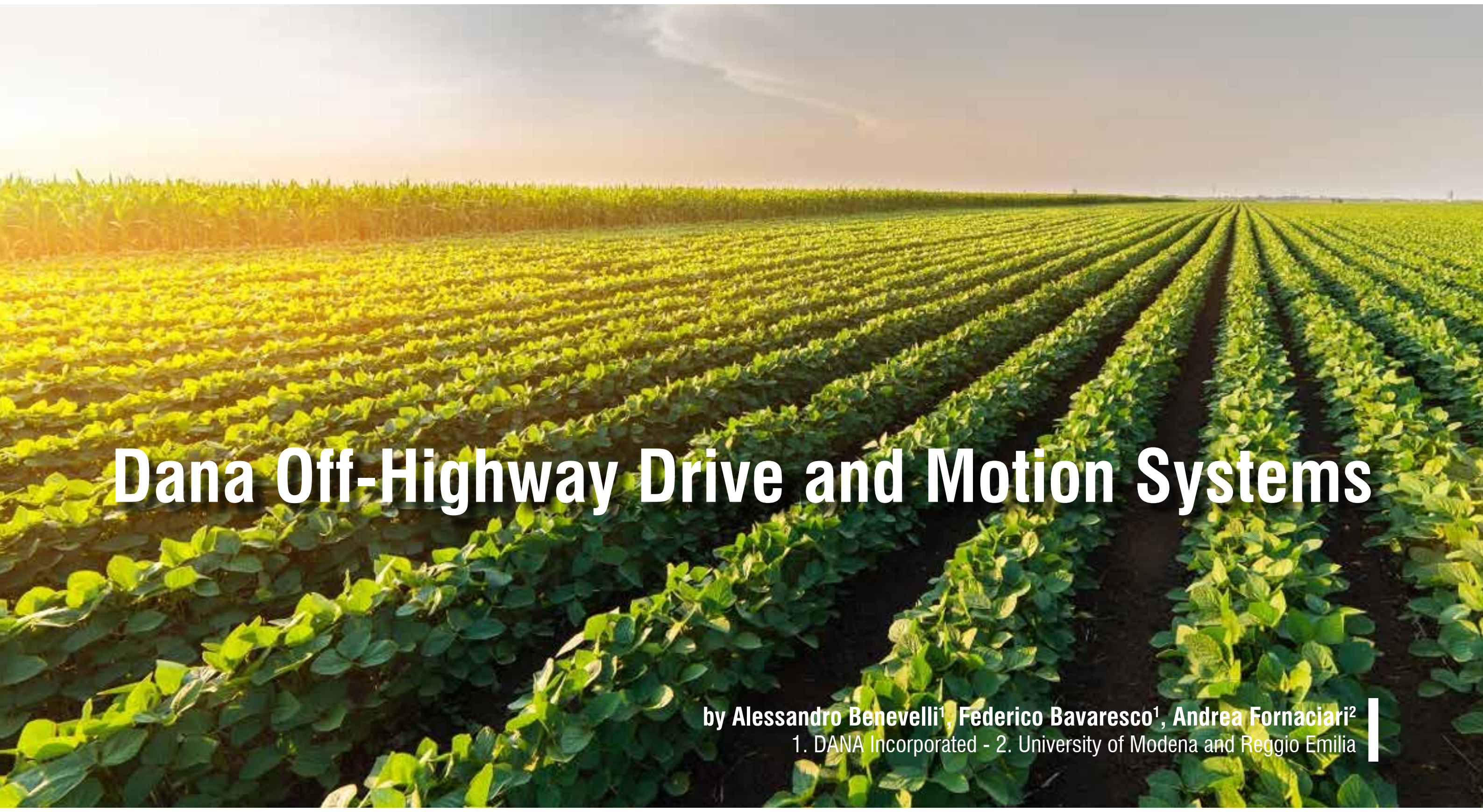
ADVERTISING

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



Dana Off-Highway Drive and Motion Systems

by Alessandro Benevelli[1], Federico Bavaresco[1], Andrea Fornaciari[2]
1. DANA Incorporated - 2. University of Modena and Reggio Emilia

Powering the next generation of off-highway suspension design

Design process integration of an independent suspension axle for off-highway vehicles

This study presents the work undertaken by Dana Incorporated to develop a new independent suspension axle for an off-highway vehicle (OHV). This multi-disciplinary simulation activity combines improvements to the kinematic and dynamic performance of the suspension while also examining the constraints of the mechanical design and the hydraulic system, as well as the cost of the suspension.
The primary goal of this study was to assess the capabilities of an automated optimization process developed using design optimization software "modeFRONTIER" which integrated several tools including Creo®, MSC Adams®, and Simcenter Amesim®. This project also served to assist Dana in successfully integrating this methodology into future workflow process enhancing independent suspension axles.

An "independent suspension" is any vehicle suspension system that allows each wheel on the same axle to move vertically, independently of the others — for example when reacting to a bump on the road. Compare this to a rigid or beam axle system in which the wheels are linked and movement on one side also affects the wheel on the other side.

Independent suspension typically offers better ride quality, traction, and handling characteristics in both automotive and off-highway vehicles. On the other hand, this kind of suspension system requires additional engineering effort and is more expensive to develop than a rigid axle.

Among these systems, hydro-pneumatic suspensions are particularly important for OHVs. One of the main reasons for this

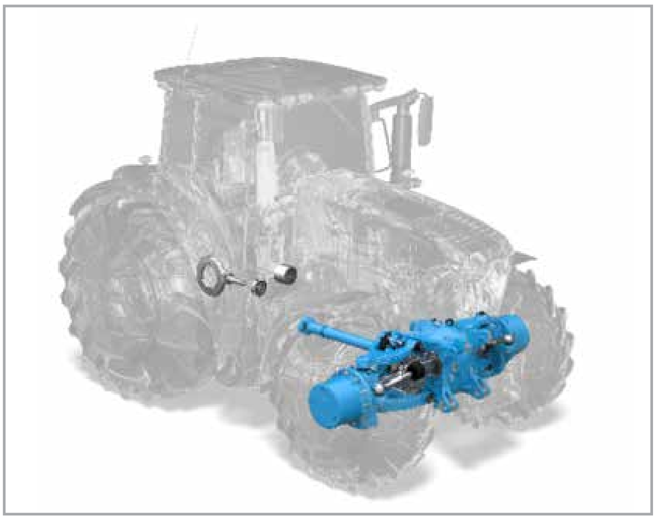


Fig. 1 - Tractor with a suspended axle (highlighted as reference)

is the hydraulic system's ability to work better with heavy loads in a limited space compared to mechanical systems. In addition, because the gas functions as an adjustable spring, the viscous friction within the hydraulic fluid is harnessed to be the damping function of the system.

This improves the ability to respond to system oscillations. Finally, a well-designed hydraulic system allows the suspension not only to adjust stiffness and damping, but also to adjust vehicle levelling and to choose between different working conditions (e.g., field or road maneuvers).

The design of this type of system involves integrating the hydraulics with the mechanical structure of the suspension, which in this case is a double wishbone suspension (see Fig. 2). This is a complicated, multi-disciplinary activity. In fact, it is necessary to combine the ideal improvements of the kinematic characteristics and the dynamic behaviour of the suspension with design constraints due to both the overall dimensions and the hydraulic system.

For this reason, a non-automated design process tends to be primarily based on experience and an iterative approach. The purpose of this activity, therefore, was to create a method for automating a multi-objective optimization process for an independent suspension design.

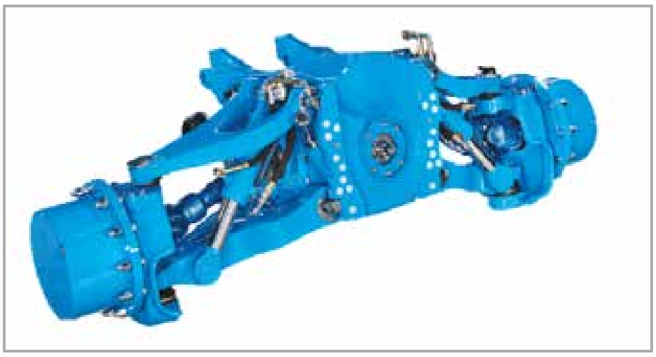


Fig. 2 - Dana's suspended axle with a double wishbone architecture

Use of modeFRONTIER allowed Dana to develop this automated process by integrating several spreadsheets and CAE tools necessary for the optimization itself. For example, an Excel spreadsheet and various MSC Adams simulations were needed to evaluate the suspension's kinematics and its dynamic effects on the vehicle, while Simcenter Amesim simulations were used to correctly size the hydraulic system. Some CAD models developed in Creo were also integrated into the process. As mentioned, they were used to ensure that no static or dynamic interference appeared in the optimized solutions.
In addition, the modeFRONTIER workflow development enabled a better understanding of the influence of certain parameters and on the overall simulation results. The results were then investigated further to identify the best solutions in terms of performance, feasibility and cost.

MSC Adams Models

The starting point for the MSC Adams and Creo models was a spreadsheet that defined the hardpoints of the suspension structure and some anti-characteristics of the system. As mentioned, many MSC Adams simulations were required to consider all the kinematic and dynamic aspects of the suspension.

These simulations were divided into three macro areas:

- Suspension kinematics testing, i.e., parallel wheel displacement, opposite wheel displacement, and steering displacement. These simulations were necessary to evaluate the kinematic characteristics and behavior of the suspension architecture.
- Modal analysis of the vehicle, which generated knowledge of the first natural frequency and the ideal stiffness and damping values for the vehicle system at different workload conditions (e.g., unladen vehicle weight, gross vehicle weight, etc.).
- Dynamic performance simulations, necessary to measure the comfort, handling, and traction performance of the vehicle. These were obtained by performing:
    - ISO5008 simulations at different vehicle speeds;
    - ramp steering and moose test simulations; and
    - traction tests on a 4-poster test bench.

Fig. 3 - Example of an MSC Adams 4-poster model

Most of the objective functions of the optimization were outputs of these simulations, such as selected kinematic characteristics of the suspension or the dynamic performance in terms of comfort, handling, and traction. Some other outputs, however, served as inputs to other models.

For example, ideal stiffness and damping values from the Adams modal analysis were considered as inputs for hydraulic circuit sizing in Simcenter Amesim. At the same time, the actual stiffness and damping force curves were evaluated in the hydraulic circuit sizing and were inputs for the dynamic simulations in Adams.

Simcenter Amesim Models

The static loads on the suspension actuators and the corresponding ideal stiffness and damping values were inputs to the sizing of the hydraulic circuit. The circuit was then sized to support these loads and achieve the target stiffness and damping values. In addition, it was sized so that the overall dimensions and cost met the targets. The main parameters to be considered were the dimensions of the hydraulic actuators, the dimensions of the hydro-pneumatic accumulators, and the damping valves and/or orifices. The correct working pressure range also had to be verified. To do this, a spreadsheet and some Amesim templates were deployed. In particular, an Excel sheet was exploited to size the stiffness part of the circuit while the Simcenter Amesim models were used to set the dimensions of the damping valves and/or the dimensions of the orifices.

These models were also used to evaluate the actual stiffness and damping force curves, which are different from the ideal ones identified in the modal analysis. These curves were needed by the Adams dynamic simulations to consider the actual behavior of the suspension system.

Creo Models

The optimization process required control so that there was no static and dynamic interference between the individual mechanical parts and the elements of the hydraulic circuit. It was therefore necessary to integrate two simplified 3D CAD models developed in Creo, one for static and one for dynamic interference.

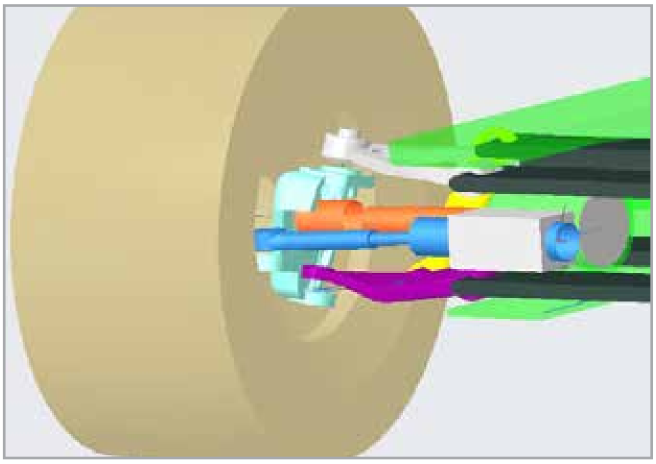


Fig. 4 - Simplified CREO model for evaluating interferences

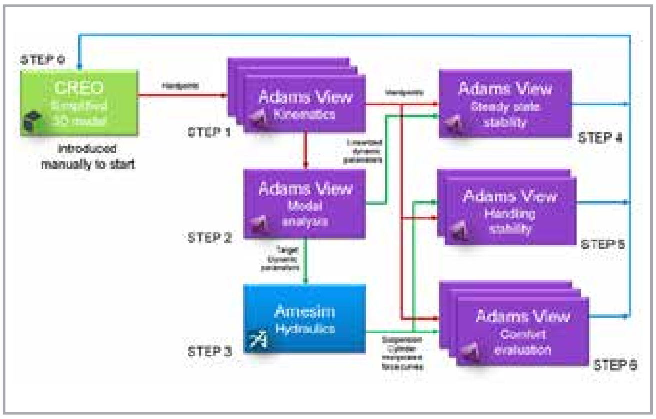


Fig. 5 - First concept of the optimization workflow

The input parameters of these models were the coordinates of the structure's hardpoints, the overall dimensions of the hydraulic circuit elements, and the maximum vertical and steering displacement of the suspension. This ensured the feasibility of the solution in each suspension configuration.

Methodology Implementation in modeFRONTIER

The first concept of the optimization workflow illustrated in Fig. 5 was characterized by a large number of input variables and objectives.

This resulted in a significant number of designs being evaluated in the optimization stage to obtain reliable and accurate results. In addition, each simulation model had to be run for each design, and some of these simulations were quite long, which would have made the process inefficient.

Therefore, we decided to split the workflow into three cycles or loops, each of which had fewer input variables, fewer objectives, and thus fewer designs to evaluate than the initial workflow. As shown in Fig. 6, each loop represented the optimization of some aspect of the entire system.

This was possible because some of these aspects were independent of each other, while we used the optimized outputs from previous loops as inputs for the dependent aspects. Finally, the workflow was structured to clearly describe the method of investigating the phenomena and to manage the strategic analysis more easily.

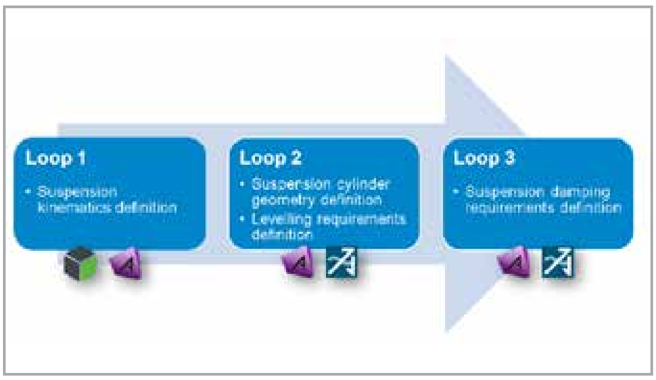


Fig. 6 - Workflow split into three loops

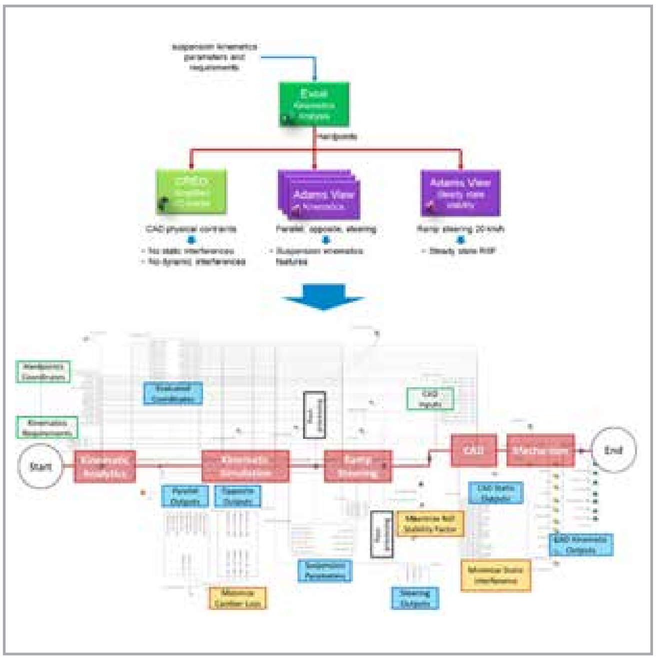


Fig. 7 - Loop 1 workflow

Loop 1

Loop 1 was used to define the kinematics of the suspension, hence the structure of the hardpoint positions that influence the main kinematic parameters. The objective of this loop was to find the best architectural solution that would guarantee the best kinematic characteristics of the suspension, while ensuring that there was no mechanical interference.

The design parameters for Loop 1 were as follows:

- Input variables - 23 variables between independent hardpoint coordinates and kinematic requirements.
- Output variables - dependent hardpoint coordinates, kinematic characteristics, steady-state roll stability factor, and possible interference.
- Objective functions - minimization of camber loss and maximization of roll stability factor.

These objectives were chosen after a sensitivity analysis. Other objectives suggested in the first optimization concept were turned into constraints.

- constraints - kinematic features have minimum values and no static/dynamic interference.

The workflow concept for Loop 1 is illustrated in Fig. 7.

Loop 2

The block diagram of Loop 2's workflow is shown in Fig. 8. The goal was to define both some remaining hardpoint locations and the suspension stiffness and damping in order to minimize the first natural frequency of the system.

The optimized coordinates from Loop 1 were held constant while the hardpoints affecting the vehicle's anti-dive were set as variables. This is because the anti-dive value affected the modal analysis of the vehicle. The modal analysis was repeated in the workflow for different load conditions.

In addition, a nested optimization of the hydraulic circuit sizing was integrated with the information from the Adams modal analysis (i.e., static cylinder loads and corresponding stiffness values). This loop focused on the stiffness aspect of the circuit while the damping aspect was studied in Loop 3.

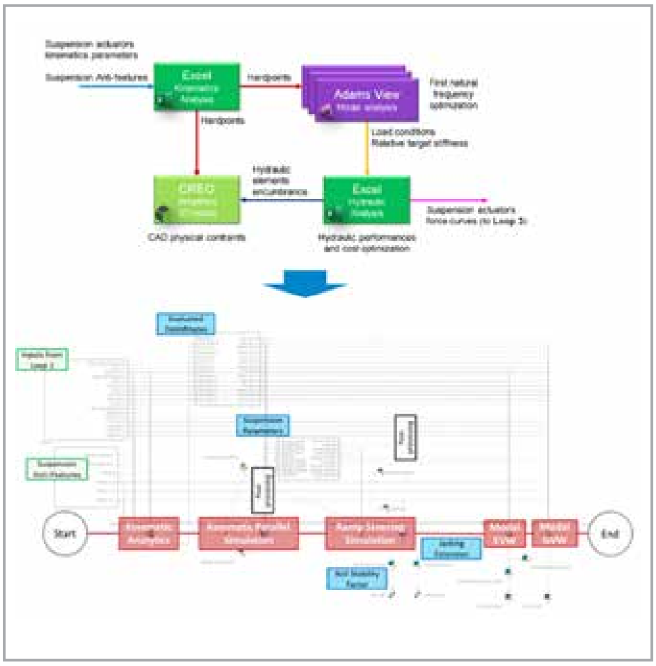
The design parameters of Loop 2 are described below:

MAIN

- Input variables - 23 variables including independent hardpoint coordinates, suspension anti-characteristics, stiffness, and damping values.
- Output variables - dependent hardpoint coordinates, modal analysis results.
- Objective functions - minimization of the first eigenmode frequencies. The first natural frequency should be as low as possible, while not descending below the motion sickness value (about 1 Hz).
- Constraints - kinematics have minimum values, motion sickness, absence of interference.

NESTED

- Input variables - 12 variables between hydraulic circuit parameters and input of constants from the modal analysis.
- Output variables - min/max effective suspension stiffness for each load condition, min/max levelling pressure for each load condition, penalty function, cylinder footprint, effective stiffness force curves.
- Objective functions - penalty function minimization, footprint and cost minimization. The penalty function was defined to reach 0 when all stiffness requirements are met. The lower the value of the penalty function value, the better the solution.
- Constraints — achieving the most important stiffness values.



Figs. 8 - Loop 2 workflow

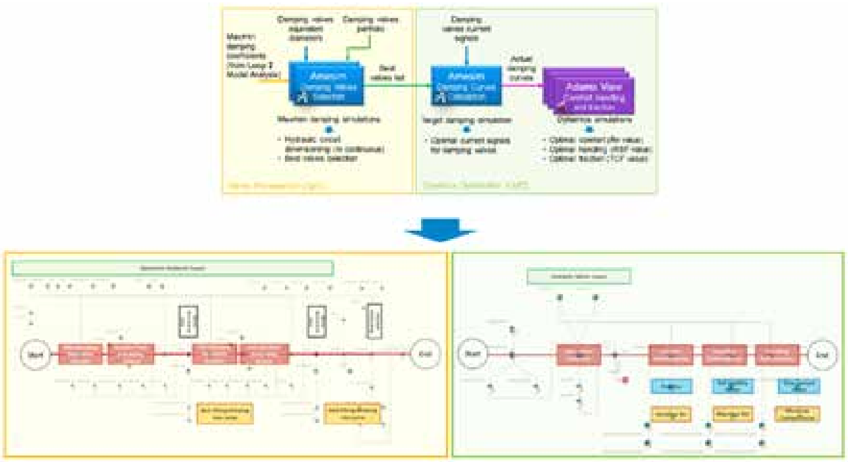


Fig. 9 - Loop 3 workflow

This made it possible to evaluate the corresponding best valve current, and hence damping value, for the three different aspects. The conceptual workflow for Loop 3 is shown in Fig. 9.

Loop 3

The optimization goal for Loop 3 was the definition of the suspension's damping requirements. The minimum and maximum damping coefficients found in the Loop 2 modal analysis were the starting points for this optimization. This workflow was divided in two sub-optimizations, a valve pre-selection and a dynamics optimization.

The first was aimed at continuous sizing of the damping part of the hydraulic circuit and subsequent selection of the most suitable valves from a portfolio. Starting from these best valves, the goal of the second sub-optimization was to improve the dynamic performance in terms of comfort, handling, and traction. The variable parameters were the input current to the damping valves, which define the damping of the entire system. Every other parameter was taken from the previous loops and set as a constant.

The design parameters for Loop 3 were:

OPT.1 – Valve pre-selection

- Input variables - 12 variables between hydraulic circuit parameters and input of constants from the modal analysis.
- Output variables - list of best damping valves.
- Objective functions - minimization of hydraulic circuit size and cost.
- Constraints - achieving the min/max damping coefficients.

OPT.2 — Dynamic optimization

- Input variables - valve index, valve input currents.
- Output variables - effective damping force curves, Adams dynamic simulation results.
- Objective functions - minimization of whole-body vibration (comfort), maximization of roll stability factor (handling), maximization of total contact factor (traction).
- Constraints - no constraints.

Each of the three dynamic simulations in Opt.2 (i.e., comfort, handling, and traction) had a dedicated project node with a nested optimization.

Optimization

Each loop presented in the previous section required a multi-objective optimization approach since there was more than one objective function per loop.

Due to the large number of input variables, the optimization strategy required a robust optimization followed by an accurate one. modeFRONTIER enabled each best-solution cluster to be identified with the first step, while refining the best values more accurately with the second.

Regarding Loop 1, an NSGA-II controlled system algorithm was selected for the robust optimization and an NSGA-II variation population size for the accurate one. The DOE of the first robust optimization was created in modeFRONTIER starting from the lower and upper bounds of the input variables and using the Incremental Space Filler and Uniform Latin Hypercube algorithm. In contrast, the DOE of the second and accurate optimization was the set of best designs from the robust optimization.

As Fig. 10 shows, the reference design was far from the Pareto frontier. This implies that one or more better solutions in terms of kinematic characteristics could easily be found among the analysed ones. With more than one best solution available, it is up to the user to choose which direction to take regarding the Pareto frontier.

Almost the same optimization strategy was chosen for Loop 2, both for the main and the nested workflows (i.e., an NSGA-II

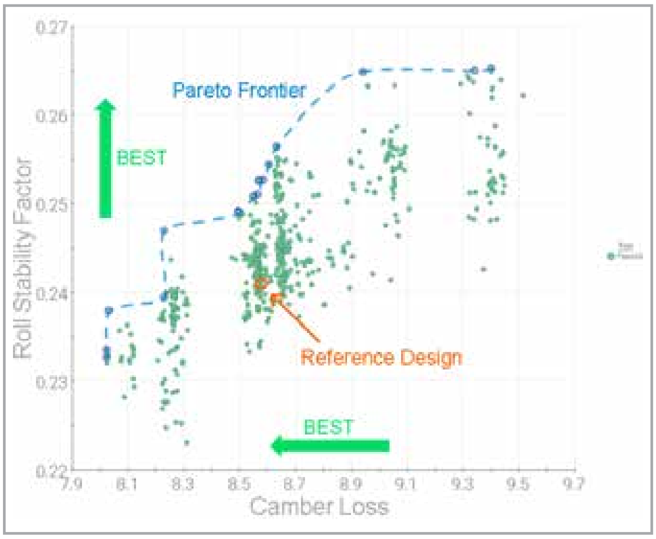


Fig. 10 - Loop 1 best solution clusters

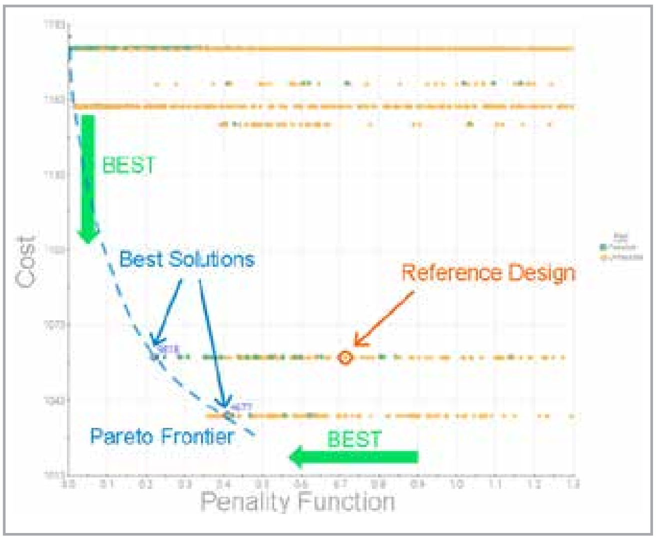


Fig. 11 - Loop 2 best solutions

controlled system for the robust optimization, an NSGA-II variation population size for the accurate optimization). The creation of the Loop 2 DOE was also similar to the previous loops.

Fig. 11 shows the results of the optimization of the hydraulic circuit from Loop 2. The best solutions highlighted are of particular interest with regard to the reference design. Both had a better penalty function value, but one had the same cost and footprint as the reference design, while the other had a lower cost. Other Pareto frontier designs were not as interesting due to their higher cost and footprint. As mentioned in the previous section, Loop 3 was divided in two sub-optimizations. The optimization strategy for Loop 3 Opt. 1 was the same as for the other loops, with a robust step and an accurate one.

The strategy for Loop 3 Opt. 2 was a little different. The goal was to optimize the valve input currents only for the valve combinations present in the best valves list obtained from Opt. 1. We therefore decided to select a DOE sequence for the main workflow of Opt. 2 while a gradient-based optimizer (B-BFGS) was chosen for the nested optimizations related to comfort, handling, and traction.

The DOE was created using an Incremental Space Filler, which ensured that all possible valve combinations were covered. An IF node was then used to check whether the current DOE design was in the list. If so, the optimization was allowed to continue, whereas the optimization would advance directly to the next design if not.

Conclusions

The development of the presented methodology in modeFRONTIER enabled not only the multi-objective optimization of the process, but also the study of the influence of some numerical parameters on the results of the simulation. This helped to increase our know-how and experience in this kind of activity. By dividing the workflow into three loops, it was possible to evaluate a larger number of designs which ensured the effectiveness and efficiency of the process.

About Dana Incorporated

Dana is a leader in the design and manufacture of highly efficient propulsion and energy-management solutions for all mobility markets across the globe. The company's conventional and clean-energy solutions support nearly every vehicle manufacturer with drive and motion systems; electrodynamic technologies, including software and controls; and thermal, sealing, and digital solutions.

Based in Maumee, Ohio, USA, the company reported sales of $7.1 billion in 2020 with 38,000 associates in 33 countries across six continents. Founded in 1904, Dana was named one of "America's Most Responsible Companies 2021" by Newsweek for its emphasis on sustainability and social responsibility. The company is driven by a high-performance culture that focuses on its people, which has earned it global recognition as a top employer, including "World's Best Employer" from Forbes magazine. Learn more at ***dana.com.***

The multi-objective approach used for each loop allowed both the Pareto frontier (i.e., the best trade-off set) to be identified and these best solutions to be compared to a reference design, which had been found manually prior to using modeFRONTIER.

As was to be expected, every loop produced one or more improvements compared to the reference solution. Finally, the modeFRONTIER workflow guaranteed complete automation of the entire analysis, maximizing the use of the hardware/software resources. Consequently, once the workflow was established, user involvement was only required to analyse the results when the optimization was complete, resulting in further cost savings.

This study succeeded in integrating the entire design process of a new independent suspension axle for an off-highway vehicle using modeFRONTIER. The multi-disciplinary and multi-objective abilities enabled the vehicle's kinematic and dynamic performance to be optimized by including the constraints from the mechanical design, the hydraulic system, and the cost in the simulation framework. Dana's engineering expertise and modeFRONTIER technology resulted in an automated optimization process with the integration of several tools as Creo, MSC Adams, and Simcenter Amesim.

Alessandro Benevelli is a System Solution Engineer at Dana Incorporated (in Reggio Emilia, Italy) and Federico Bavaresco is an Axle Lead Engineer at Dana Incorporated (in Arco, Italy).

For more information:
Stefano Carrino - EnginSoft
s.carrino@enginsoft.com

by Enrico Denna*
Flex

Using Simulation and Modeling for medical Radio Frequency design

A study demonstrating the benefits of simulation over prototyping

The growth of technology and end-user expectations is ever more important in many application areas and medical devices are at the forefront of this innovation process. The requirements of an even more connected world have had a significant impact on electronic design and specifically on the area of radio frequency (RF) which by its very nature represents a major design challenge. Modeling and simulation are valid instruments to assist in achieving this goal because performance in electromagnetic design cannot be easily predicted in advance by relying solely on the experience of the designer.

The world is changing and becoming more and more “smart”. Not more than ten years ago, the word “smartphone” was unknown, instead today it is normal to talk about smartphones, smartwatches, smart TVs and so on. Words like “connectivity”, “apps” and “data connection” have become part of our everyday lives and vocabulary. Medical devices have obviously followed this trend and have adopted connectivity as their must-have to align with market demand.

Several medical devices are becoming even smarter and more connected, storing big data, using apps, and enabling functionality that is of real benefit to users.

The design of these new features presents several challenges, one of the most critical aspects being the creation of a stable and reliable connection within a user-centric usage model. Here we are talking about a wearable model where devices are powered by smaller, sometimes non-rechargeable or non-replaceable batteries; devices that are typically either handled or directly in touch with the human body, such as insulin pens or drug delivery patch plasters; devices that are used or stored in varying environmental conditions, not only at room temperature at home, but also in the mountains, inside a car, or kept in a refrigerator.

These new design requirements have spurred the development of simulation tools to help designers refine their projects while saving time and costs on development which, until now, has always been undertaken with a trial-and-error approach. HFSS by ANSYS is one of the most powerful tools available to assist designers with their projects.

Flex is a leader in the design of medical devices for major pharmaceutical companies and its Milan office, which is its largest design center, has grown over the years by integrating all aspects of design from concept definition to mass production. In the Milan office, the electrical team that I am proud to lead has also grown and effectively adapted to the changing market by following this trend. Aiming to surmount the

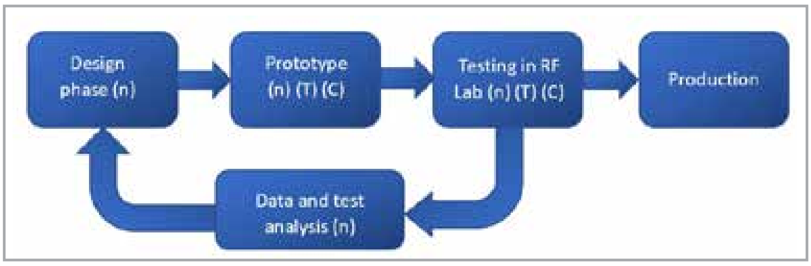


Fig. 1 – Standard Trial-and-Error Approach. Final cost of the process is calculated according to the (T)ime and (C)ost of prototyping multiplied by the number of prototyping iterations (n)

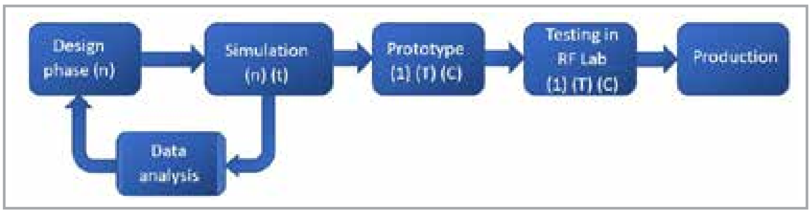


*Fig. 2 – Simulation Approach. Final cost of the process is calculated according to the (t)ime of simulation multiplied by the number of simulation iterations (n). It is clear that the cost of the first approach (n*T*C) is not comparable with that of the simulation approach (n*t + 1*T*C).*

trial-and-error approach, we improved RF knowledge with training, by accessing know-how from external experts, and with laboratory interaction, until finally introducing the simulation tool.

The benefits and advantages of the simulation approach

The comparison process flow (Fig. 1) immediately reveals that in a trial-and-error approach the number of design phase iterations (n) to fine-tune and adjust the design leads to the same number of prototypes and tests which must possibly be conducted in an external lab with the RF instrumentation and accreditation to issue a formal certification. The cost of this process (C) is calculated from the costs of the prototyping and external laboratory testing, while the time to mass production (T) is derived from the time it takes to construct the prototypes, book the external labs, conduct the tests and issue the test reports. All of these steps must be multiplied by the number of iterations (n).

In the simulation approach (Fig. 2), the number of iterations only affects the number of simulations which may take a time (t) of some hours or days depending on the complexity – this time is obviously not comparable to the weeks necessary to realize and test a prototype. Furthermore, the only person involved in a simulation is the designer, while any prototyping test involves several players (layout engineer, purchasing office, test operator, local prototype producer and financial office).

Another disadvantage of prototyping is linked to the time needed to take delivery of the prototype during which the designer is unable to work on this project. It is clear that the prototyping process is inefficient and easily justifies the investment in a simulation tool which can be amortized in just a few years of design activity.

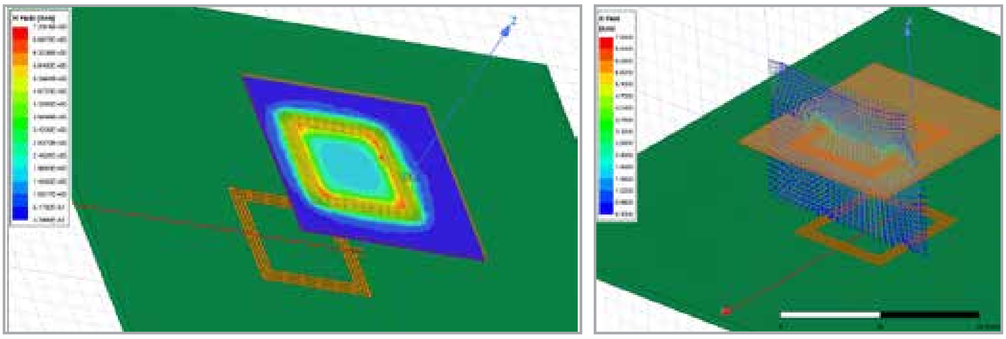


Fig. 3 (a)(b) – modeling and simulation of an RFID tag with its reader.

The improvement realized by introducing simulation is clearly evident examining some examples of design. In Flex products the connectivity designs primarily involve the Bluetooth-NFC/RFID interface typical of wearable devices, but Wi-Fi and 5G connectivity have also been analyzed. Fig. 3 (a)(b) shows the modeling of the coupling of an RFID tag and reader: the simulator enables the engineer to replicate the coils of both the tag and the reader and then to examine the magnetic field coupling between the two elements according to the spatial distance.

Typically, electrical designers begin by focusing on the PCBA and antenna design, applying their experience and radio frequency knowledge to implement the application notes and guidelines of the RF module and the integrated or chip antenna selected. This leads to a PCBA design that respects the ground planes and transmission lines and has controlled impedances, which then guides the layout engineer to design the microwave traces with the appropriate thickness and dimensions (see Fig. 4).

However, this approach may be inadequate: the mechanical parts of the device, particularly where there are multiple boards assembled together, and the metallic parts, like the battery body, motor engine or display holder, can significantly affect the electromagnetic field. All of these side effects are difficult to predict in the trial-and-error approach, resulting in an increase in the number of iterations (n). The simulator makes it possible to import complete mechanical and electrical files (such as STEP and BRD files, for instance), to obtain a complete model of the device in which it is possible to assign a material to any component using the simulator's internal library.

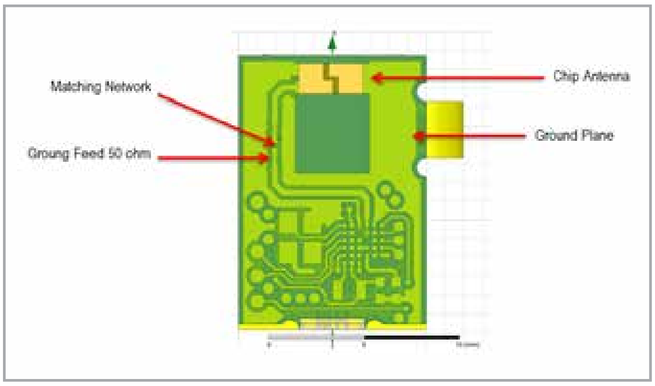


Fig. 4 – Example of PCBA with a chip antenna.

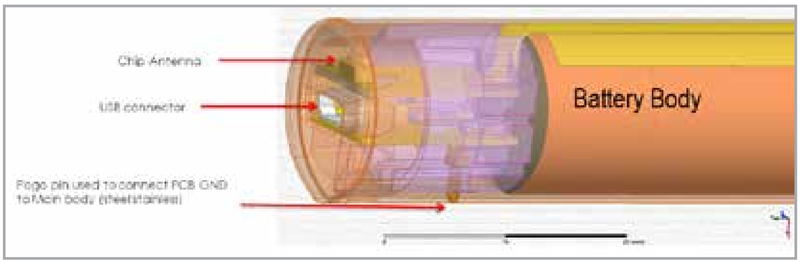


Fig. 5 – A 3D design of the final device including electronic PCBA and mechanical parts.

Fig. 5 shows part of a pen-shaped device into which the PCBA illustrated in Fig. 4 has been introduced and connected to a Micro-USB connector. The device housing is made of steel and there is a rechargeable battery on the back of the electronic location. The PCBA is connected to the housing via a pogo pin which grants a ground connection.

This setup and similar systems can dramatically affect both the RF transmission and the Bluetooth antenna. Once assembled, as seen in Fig. 5, this system has a completely different radiation pattern compared with the initial configuration visible in Fig. 4.

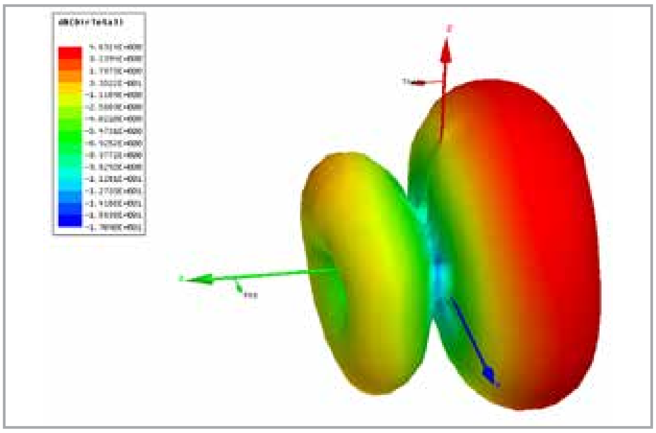


Fig 6 – 3D radiation lobe.

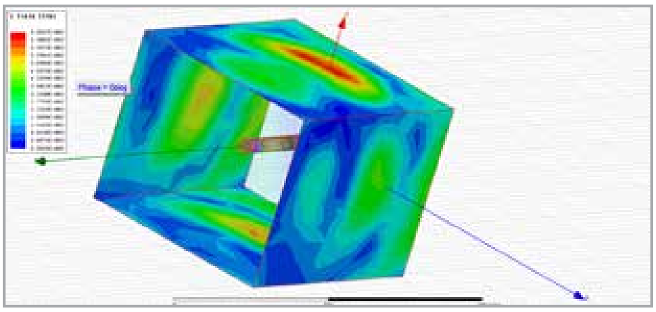


Fig. 7 – Animation of 3D phase radiation.

Fig. 6 illustrates the 3D radiation pattern generated by the simulator, which immediately informs the designer that the best direction for radiation is on the back, as opposed to on the body of the battery. This could be due to internal reflection caused by metallic parts or to the mechanical portions of the opening, both of which are compatible with this wavelength. This result is very unusual and therefore difficult to predict. The simulation result has therefore avoided a prototype round because, previously, the only way to have discovered this behavior would have been to test a prototype and then to rework the device to increase the performance in the forward direction; the simulation allows this behavior to be addressed in advance.

The simulator makes several pieces of data available both graphically and numerically: 3D radiation lobes, animations of 3D phase radiation (Fig. 7), and impedance and Smith charts suitable for S11 tuning and power transmission (Fig. 8).

Simulation makes another major contribution during the design of a PCB antenna. In the trial-and-error approach, often the application note or reference design are only indicative (and sometime ideal), which generally increases the number of iterations required because it is not always possible to rework the antenna – particularly for high radio frequencies where some tenths of a millimeter in the track dimensions can make a difference. Some challenging solutions could be accomplished using an antenna derived from the metallic parts already present in the mechanical device. This design would be easier to simulate several times before moving onto creating a prototype device once the engineer is closer to achieving the final solution.

In medical and wearable devices, the effect of the human body is not negligible: microwaves generated by Bluetooth or Wi-Fi-connections are absorbed by human tissue. Simulation plays another fundamental role here by providing a model of the human body complete with skin, bones, and internal organs into which it is possible to introduce the device to study its behavior and the effect on the radiation patterns.

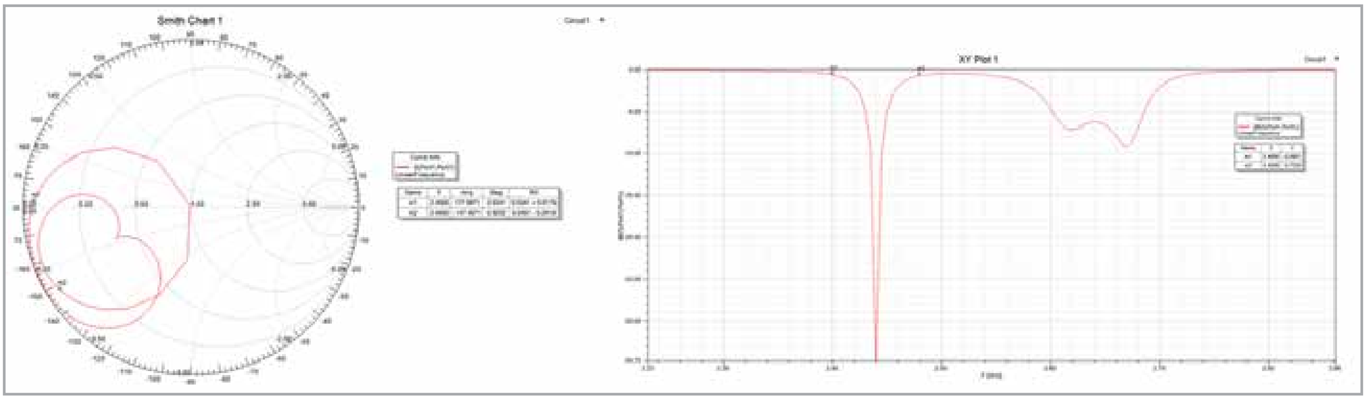


Fig. 8 – The simulator provides substantial engineering data. Here you can see the Smith Chart and the S11 parameter suitable for antenna tuning and Q factor definition.

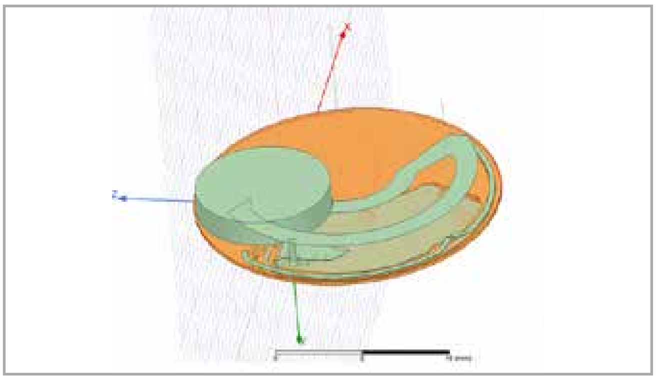


Fig. 9 – Metallic antenna derived from the battery clip.

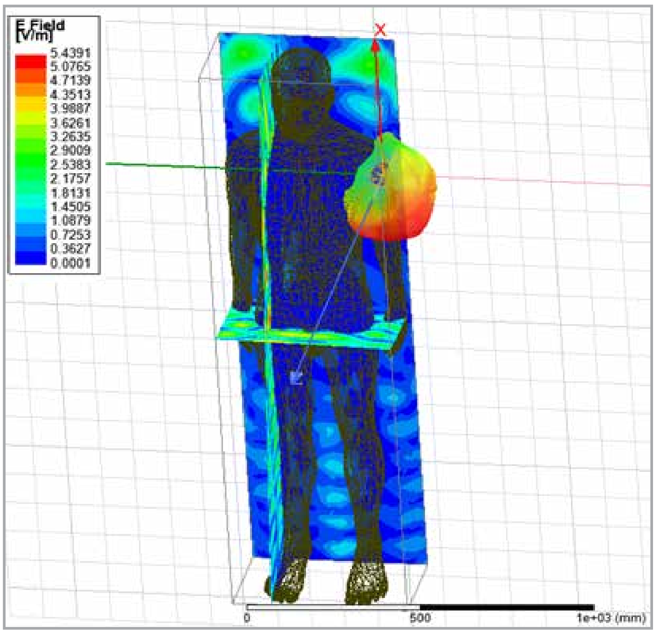


Fig. 10 – Human body model available in the simulation tool.

Fig. 10 shows the simulation of a Bluetooth device attached to the shoulder of a patient. The field strength is measured at the level of the trouser pocket where a smartphone may normally be present to receive the data. This simulation allows the designer to better understand the result and the reference direction of the radiation in the presence of a human body, making it possible to tune the design but also providing useful information for the usage model. It also provides an indication of other safety parameters, like the Specific Absorption Rate (SAR).

Fig. 12 – 3D radiation pattern measured in anechoic chamber

To optimize the confidence of the simulation result, the model can be validated during the first prototyping round. In this case, the results of tests done in an anechoic chamber using RF instrumentation and possible human dummy material (see Fig. 11 and Fig. 12) can be compared with the results obtained with the simulation allowing the simulation model to be fine-tuned by adjusting some of the parameters or corrective coefficients in its configuration. After this final step we have a powerful model that truly represents the functioning of the device inside its usage environment and that can easily be used to evaluate modifications, or as a reference for similar new devices.

The power of this tool can be supported by other sub tools, like Optimetrics which enables a parameter to be optimized or the optimal coefficient of the design to be found using automatic and iterative simulations. Other tools and add-ins can also be introduced to support thermal or impedance simulations.

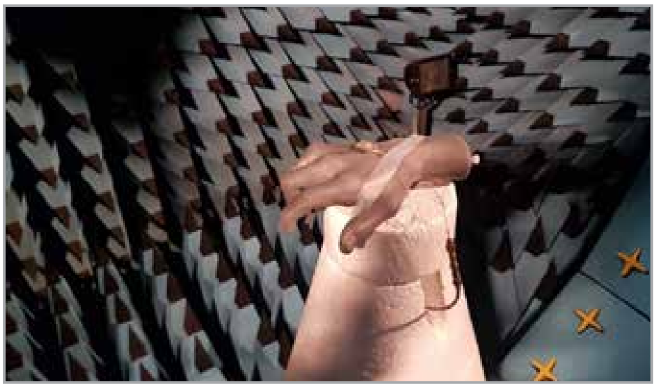


Fig. 11 – Measurement of a device in an anechoic chamber using a biological dummy

The clear advantages of the simulation approach are pushing more and more companies to consider investing in these tools to improve the design process. There are several benefits: the cost and time to production, the improvement of the designers' know-how and competence because of the ability to experiment and thereby increase their knowledge and motivation in a field that is not always intuitive or easy to master, and last but not least, simulation provides a structure to the design process that will inevitably be recognized by customers who will appreciate a more professional approach.

Enrico Denna is Hardware Engineering Manager at Flex

About Flex

Flex is the manufacturing partner of choice that helps a diverse customer base design and build products that improve the world. Through the collective strength of a global workforce across 30 countries, and responsible, sustainable operations, Flex delivers technology innovation, supply chain, and manufacturing solutions to various industries and end markets.

For more information
Andrea Serra - EnginSoft
a.serra@enginsoft.com

by Sunao Tokura, Shun Fujimoto and Akiko Kondoh
Prometech Software, Inc.

Simulating fire extinguishing equipment for historical buildings with Particleworks and Granuleworks

Studies of three methods: a water discharge gun, a drencher, and a firefighting drone

This article presents simulation examples for three types of firefighting equipment for cultural heritage buildings using Particleworks, a CFD software based on the Moving Particle Simulation (MPS) method: a water discharge gun, a drencher, and a firefighting drone. Such firefighting equipment is generally verified by means of installation standards and on-site water discharge tests, however, field tests that include all realistic fire conditions are impractical due to the cost and the physical risk to valuable buildings. Numerical simulation is therefore beneficial for the first evaluation of the firefighting equipment's effectiveness.

In recent years, some valuable cultural heritage sites, such as the Notre Dame de Paris cathedral and Shuri Castle in Okinawa, have been destroyed by fire (both in 2019). These tragic accidents and the sense of loss they provoked are still fresh in our memories. We have a mission to protect these symbols of history, tradition, and culture, and to pass them on to future generations by preventing such accidents from happening again.

In East Asia, including Japan, many historical buildings are made of wood and so there is an impelling need to prevent such fires. As a result, many facilities today are equipped with fixed and mobile water guns and drenchers that do not affect the surrounding landscape, while research and development of firefighting drones to extinguish fires in high-rise buildings continues.

The effectiveness of such firefighting equipment is generally verified by means of installation standards and on-site water discharge tests. However, conducting field tests that include all realistic fire conditions are impractical due to the cost and the physical damage to valuable buildings. It is, therefore, beneficial to evaluate the firefighting equipment's effectiveness first by using numerical simulation. The simulation considers the total amount of water applied to the building from the water discharge, the trajectory of the water discharge, and the direction of flow. Particle method computational fluid dynamics (CFD) is well-suited to effectively handle the large number of droplets and the free surface of the water as it flows over the building's exterior in these simulations.

This article presents simulation examples for three types of firefighting equipment using Particleworks, a CFD software based on the Moving Particle Simulation (MPS) method: a water discharge gun, a drencher, and a firefighting drone. Water guns and drenchers are designed to extinguish and prevent fires by spraying water, while firefighting drones use powdered extinguishing agents, which requires the powder model to be defined. Particleworks includes Granuleworks software which uses the discrete element method (DEM). By either coupling the functionality of MPS and DEM for liquid and powder interactions, or by using them individually depending on the target materials, the operation of firefighting equipment can be evaluated realistically.

Modeling airflow in the MPS method

MPS uses a formula that reproduces the incompressible flow of water and other substances and this formula is applied to the representation of water being released. During an actual fire, strong winds may be blowing which would affect the flow of the water discharged from the gun, so that the water doesn't flow in the direction of the target. Moreover, even in the absence of wind, water droplets and powder flying through the air are subject to air resistance, which gradually reduces their momentum and shortens their flight distance. To simulate a water discharge gun while taking air into account, we performed calculations using an air resistance model. One way to analyze two-phase gas-liquid flow is to use the finite volume method (FVM), also available in Particleworks, and perform the simulation using FVM-MPS coupling. However, in this case, we chose to use the air resistance model to reduce the amount of computation.

Simulation of a water discharge gun in high wind conditions

We chose a wooden structure called a Yosemunezukuri, as the target building for the water gun simulation. A Yosemunezukuri has a roof that slopes in four directions and is one of the representative architectural styles of historical buildings in Japan where there were used in many national treasures such as the Great Buddha Hall of Todaiji Temple. The main shape and dimensions of the simulation model are shown in Fig. 1. Four water discharge guns are placed diagonally across the building, and the discharge angle is set at 60 degrees from horizontal so that the water can reach the top of the roof. Note that Particleworks uses CAD data in STL format to define the shape of the structure, so meshing is not required.

For the analysis conditions, the flow rate from each water discharge gun was set to 0.15m^3/s (150l/s), and the physical properties of water and air were defined as general values. We set the particle diameter, which is an index of resolution when fluid is modeled as particles, to 20mm in consideration of the calculation time, and used the pressure explicit method to calculate the pressure explicitly.

Fig. 2 shows the result of a simulation of water discharge in windless conditions. Even in such conditions, the flying water droplets are subject to air resistance and the distance varies. Fig. 3 is a contour plot showing the total amount of water on the roof. The wet area can be seen, and the water discharge gun's angle and position can be evaluated in relation to the extent of the area covered by water.

Next, we simulated the water discharge gun with a strong, 25m/s wind blowing around the building. The airflow velocity field that combines spatial coordinates and airflow vectors is imported into Particleworks in CSV format. The air resistance is given by a 3D interpolation of the air velocity at each particle position.

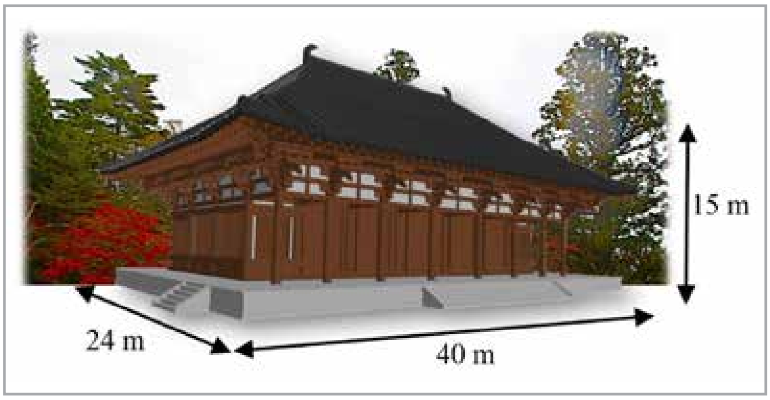


Fig. 1 - Simulation model and dimensions

Fig. 2 – Visualization of firefighting simulation using a water discharge gun in windless conditions

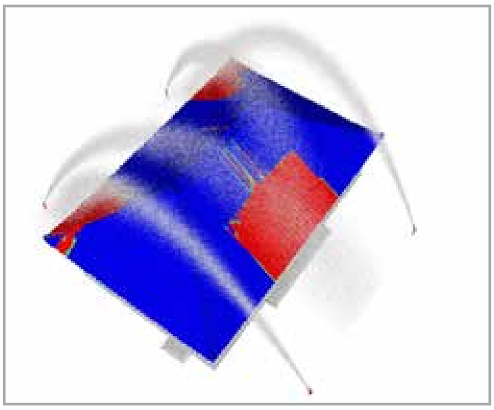


Fig. 3 – Contour plot of the total amount of water applied to the roof in windless conditions

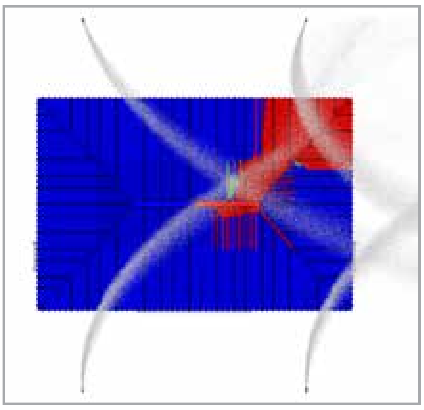


Fig. 4 - Contour plot of the total amount of water applied to the roof from a fixed water discharge gun in a 25m/s wind

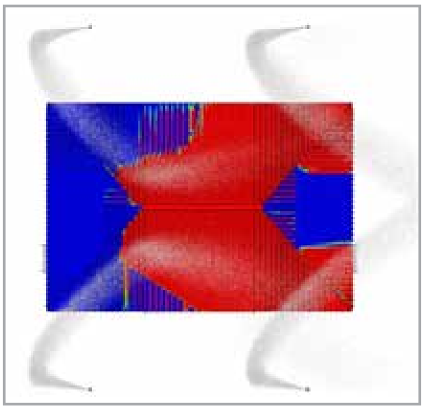


Fig. 5 - Contour plot of the total amount of water applied to the roof from a rotary water discharge gun (60° rotation) in a 25m/s wind

The result is shown in Fig. 4. Since the wind is blowing to the right in the figure, we can see that the water flow is pushed away and only a part of the roof becomes covered with water. As such, it is difficult to change the direction of the water discharge with a fixed water discharge gun according to changes in wind speed and direction. It is necessary to change the direction manually at the fire site, but this is not safe.

The next step was to simulate whether the firefighting capability could be improved by using a movable water discharge gun, using a remote control to adjust the direction of the water gun according to the situation. Fig. 5 shows the area of water coverage when the gun is rotated by 60 degrees around the vertical axis in the same 25m/s wind. We can see that the area covered by water is wider with the movable type of water discharge gun than with the fixed type. The final number of particles was approximately 350,000 and the calculation time was 30 hours using NVIDIA's GeForce GTX TITAN X GPU. If a faster GPU is dedicated to the numerical calculation, the calculation time is estimated to be several hours.

Simulation of fire prevention using a drencher

A drencher is a fire extinguishing device in which pipes are installed on the roof or exterior walls of a building, and the sprayed water forms a curtain to prevent the spread of the fire from the surrounding area. Water is also sprayed from pipes embedded in the ground surrounding the building to wrap the building in a curtain of water. Since it does not spoil the landscape, it is often used as fire-fighting equipment for cultural heritage properties.

The same building model was used for the drencher simulation. Water spray nozzles with a diameter of 100mm were configured at intervals of 5m, with a flow rate of 0.2m^3/s per nozzle and a spray angle of 6° (as shown in Fig. 6) so that the water sprayed by the nozzles would spread radially. Particle diameter was set to 25mm. To verify the drencher's effectiveness in preventing the spread of the fire, we created a situation where debris from a fire that had broken out near the building would fly in. The fire debris was modeled using DEM particles and the simulation was performed using MPS and DEM coupling. A coarse-grained DEM model was used to avoid excessive computational load, and the particle diameter was set to 50mm.

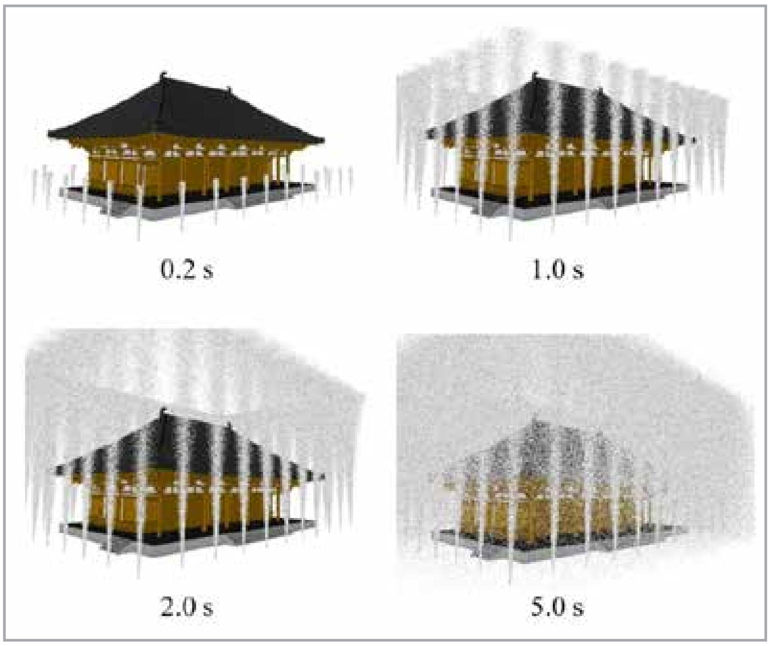


Fig. 7 – The drencher's process of forming the curtain of water

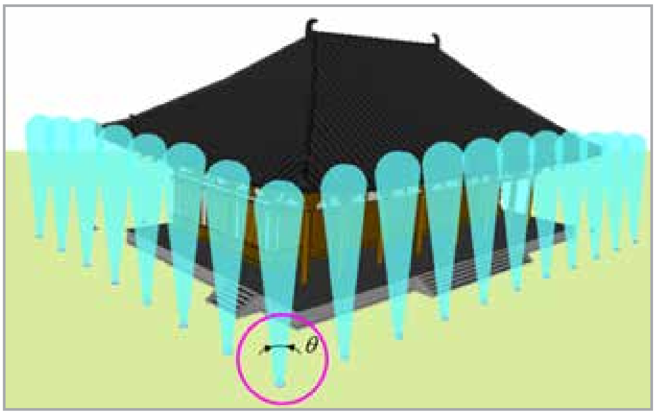


Fig. 6 – Drencher simulation model: spray nozzle angle was set at θ=6°

Fig. 7 shows the drencher's process of forming the curtain of water. The water spray reaches a height of around 27m above the ground about two seconds after the water is discharged. It then descends

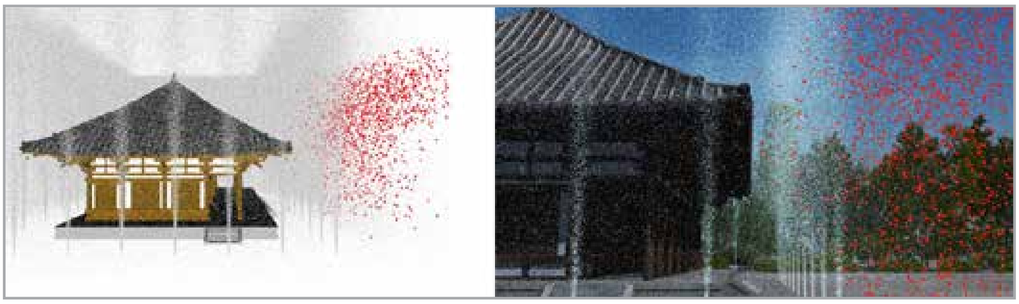


Fig. 8 – The drencher's method of preventing fire debris from reaching the building (right: CG rendering)

and, about five seconds later, the curtain of water is formed, and a dynamic steady state is reached. We tracked the behavior of the moving fire debris in this state. The simulation result showed that five seconds after the fire debris reached the curtain of water, almost all of it was blocked by the curtain of water and did not reach the building. Fig. 8 shows the results of the MPS-DEM coupling analysis of the fire debris. In the simulation, some debris from the fire reached the vicinity of the building. But the actual fire was extinguished by the water flow, so the drencher's fire extinguishing effect is deemed to be good enough. The final number of particles in this calculation was about 2 million and it took about 230 hours using an NVIDIA GV100 GPU board. In fact, further reduction of calculation time is possible by using the multi-GPU calculation capability in Particleworks.

Simulation of a firefighting drone

Research and development of firefighting drones for high-rise buildings is currently underway, some of which have reached the practical stage. This drone firefighting activity also considers the effects of wind, whether the drone can be guided to its desired destination without damaging valuable buildings, and whether a limited amount of firefighting agent can accurately reach the target. Here, we used a five-story pagoda as the model of a high-rise building, simulating the process of a drone approaching the top floor and spraying the fire extinguishing agent.

The five-story pagoda is about 31m high, and the drone has a diameter of about 1.5m. An inlet with a diameter of 60mm was defined for the nozzle at the tip from which the extinguishing agent was sprayed. The simulation assumed that the drone would rise from the ground to the height of the top floor (18m) in ten seconds, and that the fire extinguishing agent would be sprayed at rate of 30m/s immediately after the drone rose. When a liquid fire extinguishing agent is used, the drone reaches the top floor (18m) in ten seconds. However, since the use of liquid fire extinguishing agents can damage wooden structures, we used DEM to model a powder-based fire extinguishing agent that is believed to cause less damage to wooden structures. A coarse-grained model was used to reduce the computational load, and the particle diameter was set to 6mm.

Fig. 9 shows the result of the simulation and the reach of the extinguishing agent from the drone. It was evaluated in windless conditions, but the extinguishing agent's trajectory in the presence of wind can be tracked like that of the water in the example described above. Furthermore, by linking to the motion dynamics simulation, it is possible to configure more realistic conditions in which the drone flies and fights fire under the influence of wind. In this simulation, the number of DEM particles was about 90,000, and the calculation time using GeForce TITAN X was 46 hours.

This article discussed the use of Particleworks and Granuleworks to simulate fire extinguishing activities to protect important architectural buildings from fire. MPS and DEM can handle not only industrial products such as the water discharge equipment and drones discussed here, but also those used for disaster prevention in floods, tsunamis, and landslides, and in a wide range of environmental issues such as clean energy fields, which are of increasing concern in the world today. I hope that simulation technology will be used wherever possible to help protect our heritage, traditions, and culture and to realize a safe and comfortable future for everyone.

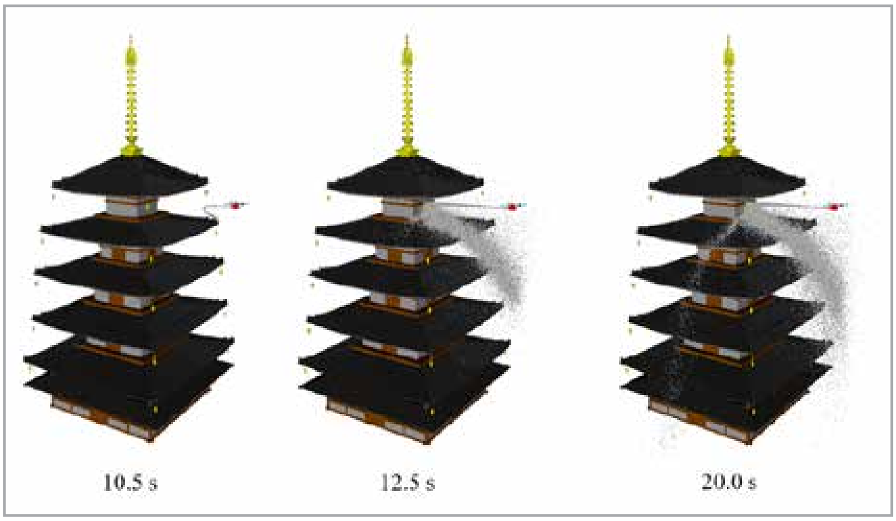


Fig. 9 – Simulation of fighting a fire on the upper floors of a wooden high-rise building with a drone

References

[1] Koshizuka, S., Shibata, K., Kondo, M., Matsunaga, T.: "Moving particle Semi-implicit Method", Academic Press, ELSEVIER, 2018

[2] Cundall, P. A., Strack, O. D. L.: "A discrete numerical model for granular assemblies", Geotechnique 29, No.1, 1979, 47-65

[3] The conference proceedings of the Japanese Society of Computational Engineering and Science, 2021

For more information:
Sunao Tokura - Prometech Software
tokura@prometech.co.jp

by Francesco Madonia
LivaNova

Optimally designing an artificial lung for extracorporeal life support

A CFD-based approach

Hollow fiber membrane oxygenators (HFMO), which have been used since the 1950s to provide patients with temporary (up to six hour) oxygen support during a cardiopulmonary bypass (CPB), are today being increasingly used for other therapies, such as acute respiratory distress syndrome (ARDS), and chronic obstructive pulmonary disease (COPB), that can last for up to 20-30 days at a time. This extended contact time between hollow fiber membranes and blood increases the complications of the natural blood biochemical process of "coagulation", which diminishes oxygen transfer and increases the pressure drop, ultimately causing the oxygenators to fail. The study described in this article was designed to obtain greater insight into the gas transfer mechanism at microscopic scale using computational fluid dynamics in order to accelerate design exploration to find the optimal solution.

Since the 1950s, the hollow fiber membrane oxygenator (HFMO) has been used to deliver oxygenated blood to, and remove carbon dioxide from, organs to support their vital functions during a cardiopulmonary bypass (CPB).

Basically, HFMOs are composed of thousands of polymeric hollow fiber membranes (usually polypropylene or polymethylpentene based), which are gathered and sealed together to form a fiber bundle. A gaseous air/oxygen mixture flows through the lumen of the hollow fiber membrane, while the outer fiber surface is in contact with the bloodstream. Based on a diffusion gradient of O_2 and CO_2 between venous blood and the ventilator gas, oxygen enters the blood from the fiber lumen while carbon dioxide travels from the blood to the fiber lumen. An oxygenator can, therefore, assist or replace lung function in critically ill patients by providing complete respiratory support. Thus, HFMOs are essentially artificial lungs that have historically been used to replace lung function for relatively short periods (up to 6 hours) during cardiac surgery procedures.

Nowadays, however, blood oxygenators are also being used for other therapies, such as acute respiratory distress syndrome (ARDS), and chronic obstructive pulmonary disease (COPB), in treatments which may last up to 20-30 days. Blood cells undergo a bio-chemical process called "coagulation" where the red blood cells and platelets aggregate to form a clot when they come into contact with foreign surfaces. Over time, this process diminishes oxygen transfer and increases the pressure drop, ultimately causing the oxygenator to fail. It is, therefore, imperative to reduce the amount of stress exerted on the blood cells. This can be done by optimizing the blood flow pathway and subsequently reducing the foreign surface area to increase the efficiency of gas transfer.

Generally, blood oxygenators for adults have approximately 2m^2 of polymer surface with which to achieve full blood saturation for up to 8l/min of blood flow, for a total oxygen gas transfer rate of 400-600ml/min. As a comparison, the lungs have 70m^2 of surface and an oxygen transfer rate of 250-400ml/min at the physiological blood flow rate at rest. Blood oxygenators are thus highly efficient because of fluid dynamics: while blood flows in a laminar and orderly manner within the pulmonary capillaries, blood oxygenators induce turbulence, increasing the gas transfer coefficient. Optimizing the local blood pathway is therefore critical to maximize the available gas exchange surface area.

In this sense, computational fluid dynamics (CFD) can be a useful and powerful tool to obtain greater insight into the gas transfer mechanism at microscopic scale and to accelerate the design exploration process to obtain the best solution based on the design inputs.

Modeling approach

Considering that a blood oxygenator is composed of thousands of tiny hollow fibers (outer diameter 300-400μm), it is virtually impossible to simulate the entire fluid domain due to the computational constraints. It is therefore imperative to define a "reduced representative volume" to represent the fiber bundle. Given that coiled oxygenators are composed of sheets of long, overlapping fibers with a relative angle "α" (generally 20°) between each layer that is then wrapped around a plastic core, one can see how the total fluid volume consists of "n" smaller volumes that are identical to each other and periodic (see Fig.1). Assuming that the distribution of the mass flow rate is uniform along the inlet surface, these smaller volumes can be considered as mini oxygenators that contribute in a "parallel/serial manner" to the overall oxygen transfer rate. This means that it is sufficient to study a smaller volume to predict the performance of the full oxygenator.

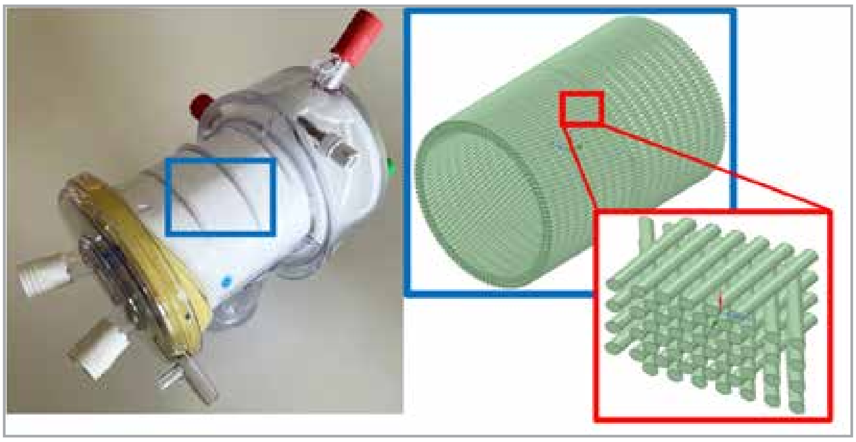


Fig.1 – Microarchitecture of the fiber bundle and its periodicity

Mass transfer model

The current work implements a classic mass transfer model first introduced by Hormes et al [1], for both oxygen and carbon dioxide, to solve the convection-diffusion equations for the species, namely O_2 and CO_2. Two user defined functions (UDF) were written to consider the diffusion mechanism of oxygen and carbon dioxide. Oxygen is found in blood either dissolved into liquid plasma or bound to hemoglobin contained in red blood cells. Carbon dioxide can be found in various forms, bicarbonate ions being the most common. The diffusion terms must therefore model different transport mechanisms to effectively mimic the physical phenomenon.

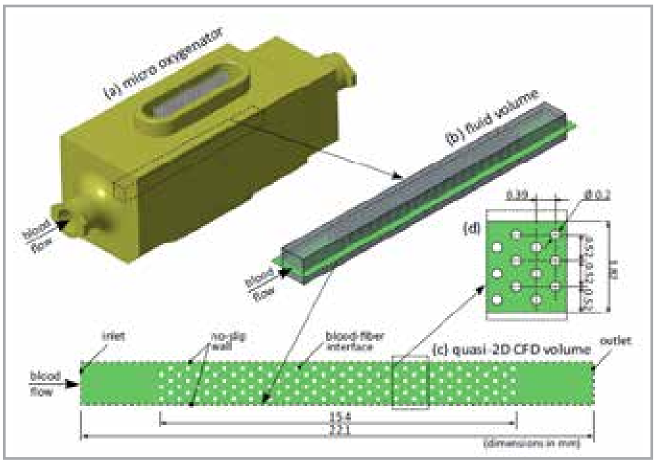


Fig.2 – Model of micro-oxygenator [2], and computational domain

In order to evaluate whether the model worked properly, it was decided to replicate the study by Kaesler et al [2] and compare the results obtained in terms of oxygen transfer. Kaesler studied a micro-oxygenator composed of 40 rows of stacked fibers. As can be seen from Fig.2, due to the periodicity of the system, a slice of the total fluid volume can be studied after appropriately scaling the mass flow rate with respect to the frontal area ratio. The solution in terms of oxygen partial pressure will be representative of the total micro-oxygenator.

The quasi-2D geometry (see Fig.3) was meshed with polyhedral elements with an adequate set of prismatic layers to resolve the boundary layer. A mesh-sensitivity analysis was performed to assess the proper size of the elements. The final mesh had 1.2 million elements. The inlet was modeled as a velocity inlet. The velocities were calculated from the mass flow rate of the entire device, which are 20, 50 and 90ml/h. The pressure outlet was set to zero-gauge pressure. The periodic faces and walls were set accordingly. Blood was chosen as the working fluid, neglecting its non-Newtonian behavior.

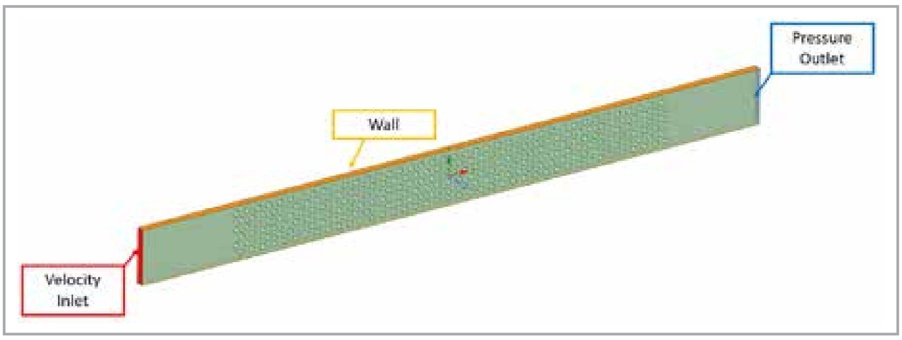


Fig.3 – Fluid domain and boundary conditions

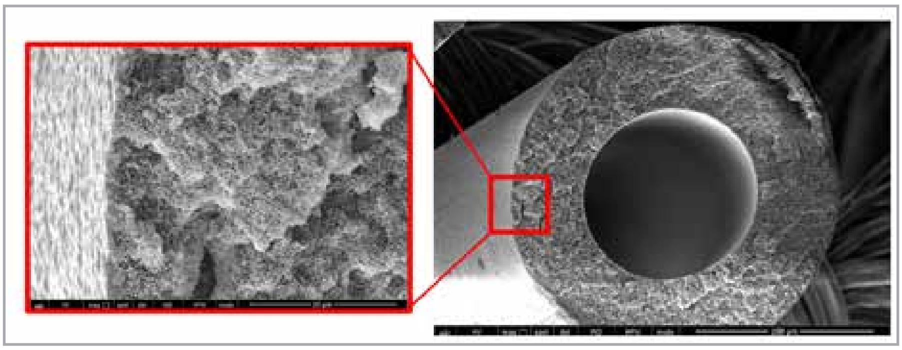


Fig.4 – Cross-section of the fiber lumen

To model the outer walls of the fibers as a source of pure oxygen, a specific value of the transported scalar (UDS) was set in these face zones. Since the ventilation gas flowing into the fiber lumen is pure oxygen at atmospheric pressure, the partial pressure of oxygen is about 1 bar, which is about 750mmHg.

Additionally, in the CAD model, the fibers are modeled as continuous surface cylinders instead of a porous membrane. Fig.4 shows a SEM cross-section of a single fiber, with a magnification window of the external surface. Note the porous sponge-like structure, which creates microchannels for oxygen diffusion.

The porosity of such a structure is in the order of 30-40%. This means that oxygen does not flow through the entire surface of the fibers, but only through the pores of the fibers, which represent 30-40% of the gross surface area in contact with the blood, as mentioned earlier.

To account for this, a UDF sub-routine was written to randomly assign a scalar value of 40mmHg and 750mmHg to the fiber surface elements. Specifically, 750mmHg was assigned to 40% of the total elements (representative of the pore area) and 40mmHg to the remaining 60%, such that

$$pO_{2_{wall}} = p \cdot 750 + (1-p) \cdot 40$$

where p is the porosity of the fiber. Looking at Fig.5, one can see how the subroutine randomly assigns the above values to the fiber wall elements.

Results and discussion

The results showed acceptable and satisfactory agreement between the values obtained with the model implemented here and the reference paper by Kaesler et al, in terms of the oxygen partial pressure at the outlet and predicted blood saturation at three different blood flow rates. It should be considered that the model implemented by Kaesler et al represents the state of the art, being the most accurate model available to be used for the study of mass transfer in blood. Looking at the fluid dynamics solution (Fig.6), the velocity streamlines showed a very well organized laminar flow path. The fluid accelerates and decelerates based on the fiber distance, as expected.

Blood enters the fluid domain with an oxygen partial pressure (pO_2) of 40mmHg, representing the venous condition. Over time, oxygen flows from the fiber wall and dissolves into the fluid, increasing its partial pressure. Fig.7 shows a stationary solution, where the fluid streamlines are colored relative to the local oxygen partial pressure. The color legend reflects a true physical phenomenon: when the oxygen content increases, the color of the blood becomes bright red.

Looking at the microscale, the blood tends to decelerate when in contact with the surface of the fibers forming the well-known boundary layer. Here the fluid is well oxygenated as can be seen from Fig.8 due to the lower velocities. In the posterior region of the fiber, you can see

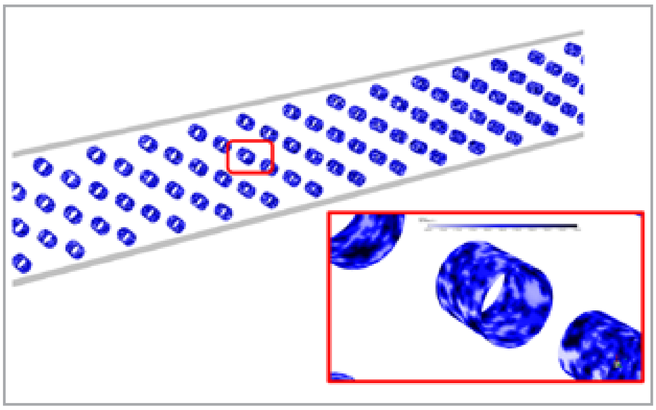


Fig.5 – Scalar source assigned to the fiber wall elements.

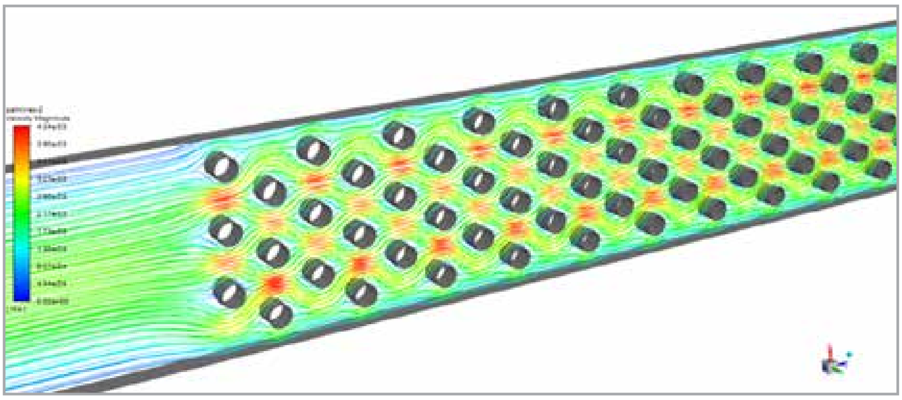


Fig.6 – Velocity streamlines

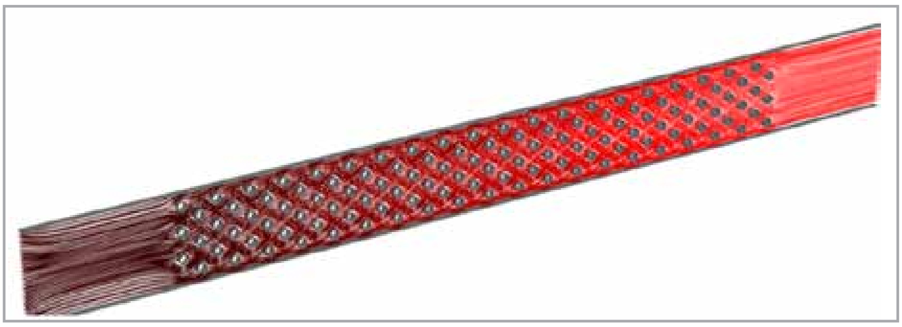


Fig.7 – Streamlines of pO_2

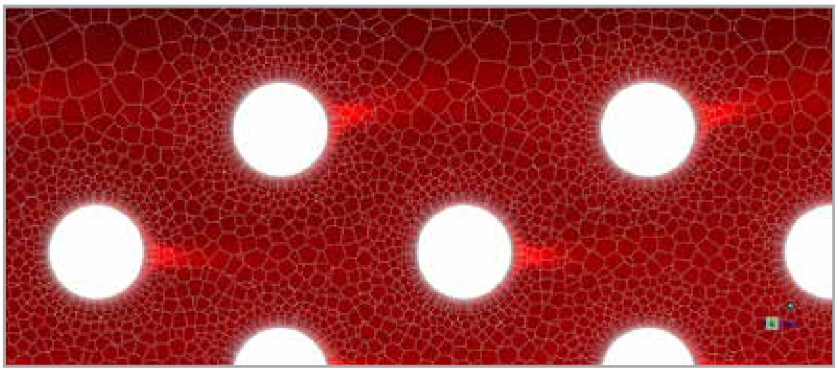


Fig.8 – pO_2 contour plot

In particular, below 100mmHg, diffusion is highly dependent on hemoglobin's reaction mechanism.

The reference model implements a two-phase fluid (plasma and red blood cells) and a more complex and complete reaction mechanism. In contrast, our model considers a simplified space-varying adjusted diffusivity. It is, therefore, clear why the predictability between the two models increases when pO_2 is less than 100mmHg.

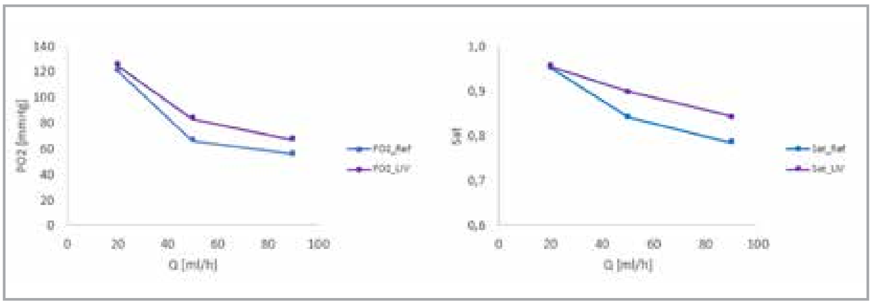


Fig.9 – Comparison of pO_2 and oxygen saturation

the presence of a "tail", underlining the transport of oxygen due to blood velocity.

Oxygen partial pressure at the fluid domain outlet and oxygen saturation were compared between our model and the reference. The results showed acceptable accuracy at the mass flow rate studied. As expected, the partial pressure of the oxygen decreases as the flow rate increases due to the reduction in contact time between the blood and the fiber surface (Fig.9).

It is worth noticing how the difference between the predictions of the two models increases as the partial pressure of the oxygen (and thus the oxygen saturation) decreases (Table.1). This is because, as explained earlier, oxygen transportation is extraordinarily complex and includes more than just the simple physical phenomenon.

	Ref	LIV	Ref	LIV
ml/h	PO2		Sat	
20	121	125	0,952	0,954
50	66	83	0,840	0,898
90	56	67	0,784	0,843

Table 1

References

[1] Hormes M, Borchardt R, Mager I, Rode TS, Behr M, Steinseifer U. A validated CFD model to predict O_2 and CO_2 transfer within hollow fiber membrane oxygenators. Int J Artif Organs. 2011 Mar;34(3):317-25. doi: 10.5301/ijao.2011.6494. PMID: 21462147.

[2] Kaesler A, Rosen M, Schmitz-Rode T, Steinseifer U, Arens J. Computational Modeling of Oxygen Transfer in Artificial Lungs. Artif Organs. 2018 Aug;42(8):786-799. doi: 10.1111/aor.13146. Epub 2018 Jul 24. PMID: 30043394.

Conclusion and future studies

This study, developed in collaboration with EnginSoft Italy, is the first step towards a CFD-based design approach for the engineering of hollow fiber membrane oxygenators. The microscale approach proposed here aims to provide numerical and specific answers to the mass transfer phenomena occurring in the blood oxygenator.

This study allows us to predict how a different microstructural arrangement affects mass transfer performance, guiding the design phase in a robust and sound manner. Similar to the heat exchanger, the local mass transfer coefficient could be accurately predicted and used for macro-scale calculations without the need for complex, experiment-based analytical calculations.

Future work will lead to a multiscale framework approach that also takes into account how fluid is directed to the fiber bundle, guiding the design of highly efficient blood collectors. This, along with predictions of pressure drop and blood activation, can really accelerate time-to-market, saving time and resources while providing innovative and optimized products to our patients.

For more information:
Fabio Villa - EnginSoft
f.villa@enginsoft.com

About LivaNova

LivaNova is a global medical technology company built on decades of experience and relentless commitment to patients. Our focus is to transform lives with products and therapies for the head and heart through health innovation that really matters. LivaNova has a diverse product portfolio, including therapeutic areas such as cardiopulmonary, advanced circulatory support and neuromodulation. Our mission is to provide hope for patients and their families through innovative medical technologies that deliver life-changing improvements for both the head and heart. LivaNova is present in more than 100 countries worldwide, with nearly 4,000 people across production sites and facilities. For further information, visit: ***www.livanova.com***

by Elisa Serioli[1], Corrado Groth[2], Simona Invernizzi[1], Marco Evangelos Biancolini[2]
1. Dallara - 2. Università di Roma Tor Vergata

Aero packs in good shape with advanced mesh morphing

How the synergy between Adjoint Methods and RBF Morph can efficiently boost Dallara Automobili's aerodynamic design process

Increasing environmental awareness has led to changes in motorsport regulations that limit the quantity of traditional and computationally intensive optimizations that can be used and encourages the use of numerical methods to improve the vehicles. This article describes a new method of aerodynamic design, the result of a collaboration between Dallara Automobili and RBF Morph, that uses adjoint methods and mesh morphing to create an innovative solution to accelerate the optimization process, reducing both time and costs.

The technological evolution we are witnessing today is redefining the boundaries of what it is possible to achieve with numerical simulations. As engineers, this constant drive mainly affects the ways we approach product testing and design in two ways: by the increase in computing power, and by the use of new and refined numerical methods. In recent years, computational fluid dynamics (CFD) has particularly taken advantage of the former, allowing complex and refined simulations – that would have required days in the past – to be completed in a matter of hours.

The increasing sensitivity towards environmental sustainability and awareness of ecological problems today, however, is leading to a new scenario in which a new balance must be found between computing resources and environmental awareness. In motorsport, this shift has been made evident by the computing restrictions introduced by the FIA rules, which place de facto limits on the ability of teams to use traditional and computationally intensive approaches to optimization, and encourage the adoption of more subtle and technologically advanced numerical methods over the use of brute force.

In this article, resulting from the cooperation between Dallara Automobili and RBF Morph, we present a new approach to aerodynamic design in which the synergy between adjoint methods and mesh morphing is exploited in order to deliver a turnkey solution to accelerate the optimization process, reducing both time and costs.

According to the proposed method, the information extracted from adjoint simulations is ingeniously employed to update the numerical grid using RBF Morph and deliver a new optimization paradigm in which the engineer can inspect – at a post processing stage, too – the influence of any given shape parameter in real time, without the need for a new CFD simulation.

Fig. 1 – Dallara is active in the production of race cars (a), consulting services for top performance vehicles (b), and recently introduced the Dallara Stradale (c) onto the market.

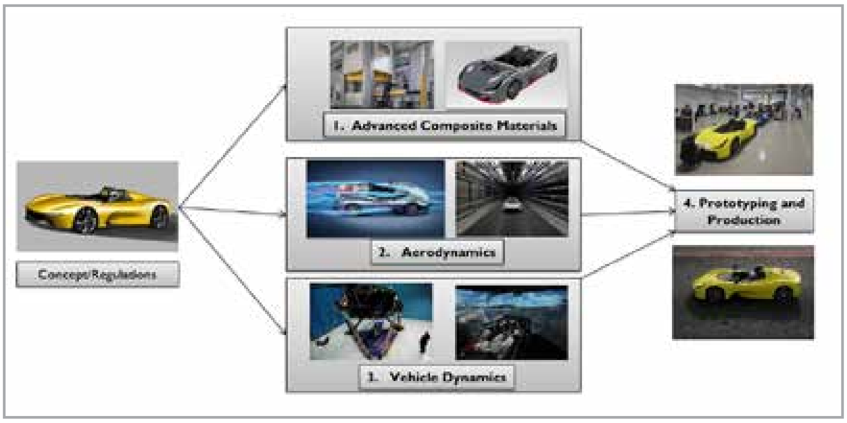


Fig. 2 – Core competencies at Dallara: from a new concept to the production of a race car built with best-in-class materials and delivering top performance, achieved by conducting accurate optimization of aerodynamics and vehicle dynamics.

To demonstrate this approach, we will show how a clever new tool, called rbf-adjoint-interactive, was applied to a Dallara Formula 3 car in order to reduce its aerodynamic drag by 0.43 drag points.

Dallara Automobili

Dallara is an Italian race car manufacturer founded in 1972 in Varano de' Melegari (Parma). The successes in Formula 3, first in Italy then all over the world, their affirmation in the American IndyCar, consultancies for important manufacturers, and constant attention to technology and innovation have all led to Dallara being recognized as one of the most important firms specialized in designing, developing and producing the fastest and safest race cars in the world. An overview of how active Dallara is in race cars and top cars is provided in Fig. 1 which summarizes its activity in motorsport (a), consultancy services in motorsport and top cars (b), and the recently launched Dallara Stradale (c) the first top car entirely conceived, produced and branded Dallara, in which "Engineer Dallara's dream has become a reality".

The excellence pursued and achieved by Dallara is the result of the core competencies (Fig. 2) the company has refined in 40+ years of experience. A good concept becomes a great prototype thanks to the adoption of best-in-class composite materials to create lightweight, durable structures; great attention to aerodynamics ensures minimum drag and the desired downforce; and advanced vehicle dynamics achieves top performance on the road and on the track. Dallara's facilities include a wind tunnel for 1:2 scale vehicle testing and a professional driving simulator.

Aero development plays a key role in conceiving and optimizing a race car. The process (Fig. 3) combines numerical simulations and wind tunnel testing. A detail of the CFD process is provided in Fig. 4 where the role of volume mesh morphing combined with a primal and adjoint flow solution is highlighted. In the study presented here, a detailed example is given that shows how advanced mesh morphing provided by RBF Morph is combined within the aero development of a Formula 3 car.

RBF Morph

RBF Morph is a pioneer in providing reliable and high-performance mesh-morphing-based technology for CAE multi-physics modelling and optimization, with more than 10 years of experience in industrial applications of Radial Basis Functions (RBF). At the core of its business there is a line of best-in-class products crafted to deal with challenging CFD and CSM applications. RBFs are recognized as one of the best mathematical tools for mesh morphing, able to continuously interpolate or extrapolate values defined at discrete

Fig. 3 – The aerodynamic process at Dallara: wind tunnel and CFD are combined to design and optimize aero shapes.

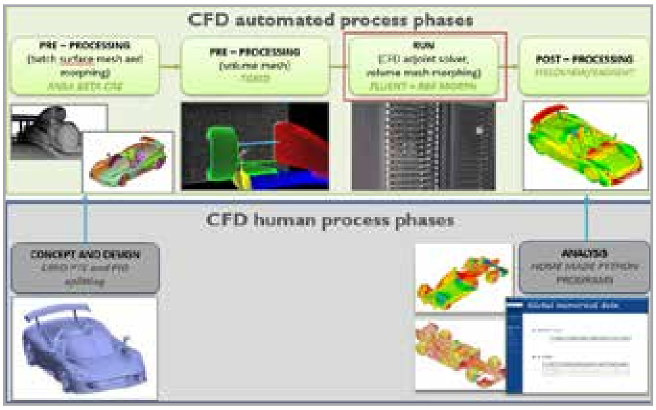


Fig. 4 – The CFD process in detail. High quality surface mesh and volume mesh are prepared for the CFD solver from CAD models. Volume mesh morphing happens in combination with a CFD solution of primal and adjoint flow.

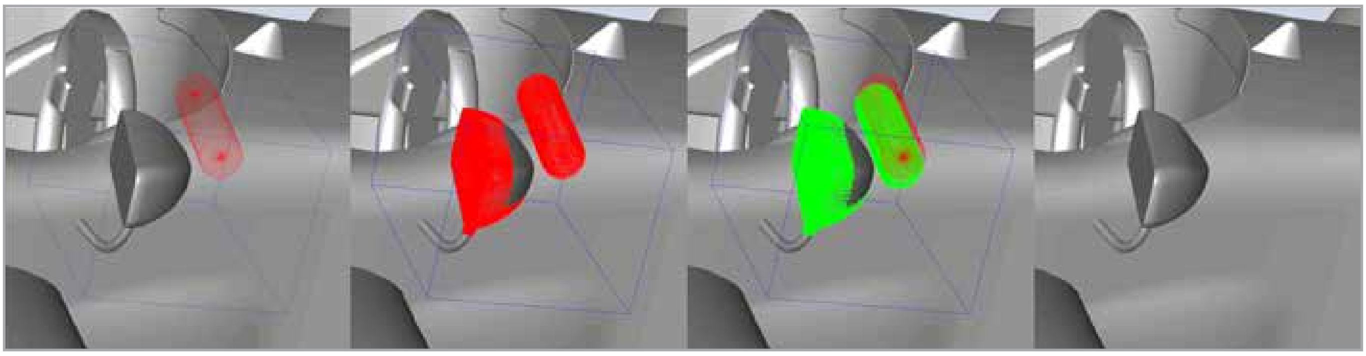


Fig. 5 – Example of RBF Morph set-up.

points. RBF makes it possible to define mesh morphing problems from a list of points and their displacements in space, since it can propagate volume deformations that have been carefully and precisely prescribed at the boundaries. RBF-based mesh morphing has a number of advantages over traditional methods. It is generally faster than remeshing and allows the shape of a numerical mesh to be parameterized while preserving its original topology. An example of how RBF Morph can control the shape using RBF points is provided in Fig. 5.

The remeshing noise, introduced by a new computational grid, is avoided and the shape of a validated CAE model can be updated without rebuilding a new mesh. New shapes can be investigated even if the underlying CAD geometry is missing, and the mesh can be updated to measured shapes (i.e. to take manufacturing tolerances into account).
Thanks to the mesh-independent property typical of RBF methods, the same mesh deformation problem can be applied to different grids without added overhead and, since the numerical geometry can be deformed directly at the solving stage, it is possible to obtain an impressive acceleration in optimization (usually reduced by a factor of five) compared to traditional approaches, since calculations on new design variations can be restarted using converged solutions obtained at the previous configuration.

In this study, the RBF Morph Fluent add-on was used, leveraging its integration with the Fluent Graphical User Interface and its ability to be controlled and steered with TUI commands. Several shape variations can be configured independently and saved using the graphical interface, and later mixed together – each shape with its own amplification factor – in order to obtain complex geometries resulting from their linear superimposition.

Beyond optimization: advanced adjoint based post-processing

The power of adjoint formulation lies in its ability to obtain, for each objective function ϕ, its variation with respect to any given parameter at the cost of a single added evaluation. This approach is particularly powerful when dealing with freeform optimizations, in which each nodal displacement x_k is a parameter and traditional optimization is not feasible (10k surface nodes would translate into 30k parameters).

On the other hand, RBF Morph provides the deformation velocity, namely the displacement of each node function of the shape amplification factor b, for each shape variation. By coupling an Adjoint solution in the form of a shape sensitivity map ($^{\delta\phi}/_{\delta xk}$) with the shape parameters generated using RBF Morph ($^{\delta xk}/_{\delta b}$), it is possible to efficiently compute the influence of a given shape parameter on the objective function ($^{\delta\phi}/_{\delta b}$) at the cost of a single multiplication.

This calculation can be carried for any given shape parameter and automatically used in a gradient-based optimization, for example together with a gradient descent algorithm.

In this article, which uses the high-fidelity CAE solver Ansys Fluent (CFD + adjoint), we present a variation of this approach using rbf-adjoint-interactive, a brand new interactive custom feature defined to quickly explore new shapes without any additional solver calculations.

This approach involves four steps:

- first, the flow and adjoint solutions are inspected; then, the areas to be modified can be decided from the sensitivity map in order to maximize the impact on performance by modifying the most influent surfaces;
- at this point a set of shape modifications can be created using RBF Morph, using selected design parameters, FEA deflections or even shapes sculpted directly using the adjoint solution as input;
- the rbf-adjoint-interactive tool allows the influence of each parameter to be inspected and a performance estimation to be interactively achieved in real-time by manually tweaking the amplification of each shape parameter.

Drag analysis of a Formula 3 car

Dallara used the rbf-adjoint-interactive functionality to optimize the performance of an F3 car. In particular, the drag was evaluated by adopting a half-car model mosaic mesh comprised of about 50 million cells.

The drag sensitivity was calculated with the adjoint solver (Fig. 6) and three specific regions of interest were identified: the rearview mirror, the bargeboard, and the front wing end plate.

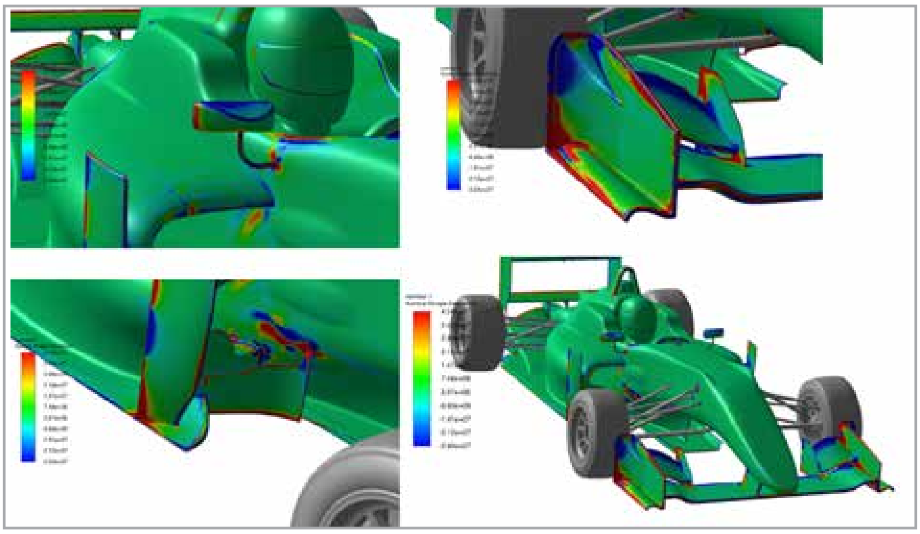


Fig. 6 – Sensitivity map of the car; details of the rearview mirror, the bargeboard, and the front wing end plate.

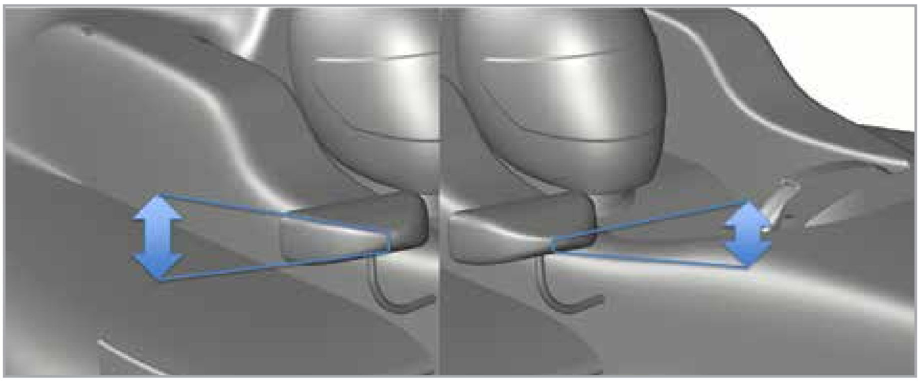


Fig. 7 – Example of mesh morphing, the width of the mirror nose is reduced.

The mesh morphing configuration explained in Fig. 5 was applied to the three regions of interest and 14 shape parameters were generated to control the rearview mirror (four parameters), the bargeboard (six parameters) and the front wing end plate (four parameters).

An example of how mesh morphing affects the shape is given in Fig. 7 where the effect of one of the four shape modifications is demonstrated: the size of the nose of the mirror is a geometrical parameter controlled by mesh morphing.

As explained in the introduction, in this specific study parameterization was not used at it usually would have been for a complete optimization which foresees as certain number of loops. The scenario discussed here was inspired by the limitations to the number of allowable shape variations that can be explored as imposed by the FIA rules for Formula 1.

The rbf-adjoint-interactive tool is, in fact, intended to best understand what can be achieved in a single shot after inspecting the adjoint sensitivities.

By inspecting the quantitative estimation of the individual effect of each shape parameter, the engineer can decide how to combine them to define a new, hopefully improved, shape. It is important to note that the sensitivities are only valid around the baseline. This means that too much variation could be risky, while a small variation is safer… but represents a small gain.

We challenged ourselves on a Friday afternoon in order to run a new simulation over the weekend. We defined the new shape by combining eight parameters (four on the mirror, and four on the front wing end plate) that showed the most promising results. A 0.6% reduction of the drag force was obtained, corresponding to a reduction of 0.43 drag points. A comparison between the original shape and the optimized one is provided in Fig. 8. It is worth noticing how small shape variations, applied to the most sensitive regions as identified by inspecting the adjoint results in Fig. 6, produce an important variation in the drag force of the car.

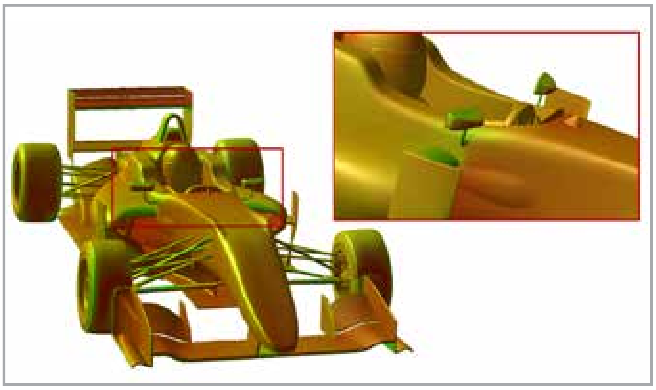


Fig. 8 - Baseline configuration (left side of the car) compared with the optimized one (right side of the car) and a detail of optimal design with the outline of the original one superimposed

Conclusions

Advanced tools are necessary to obtain as much information as possible from high-fidelity CFD. When shape sensitivities are available (adjoint solution), we can calculate the derivatives in performance with respect to parameters.

In this study, we presented rbf-adjoint-interactive, a new tool based on Ansys Fluent and RBF Morph that predicts the effect of shape on performance without the need for a new CFD computation. The proposed method was applied to reduce the aerodynamic drag of a Dallara Formula 3 Car by modifying the mirror and the front wing end plate to gain 0.43 drag points.

For more information:
Marco Evangelos Biancolini
Università di Roma Tor Vergata
biancolini@ing.uniroma2.it

by Elisabetta Fava[1] and Massimo Galbiati[2]
1. Comer Industries S.p.A - 2. EnginSoft

Lubrication and heat dissipation in transmissions and bearings

Lubrication and heat dissipation in transmissions and bearings are critical to both the performance and the life of these systems.

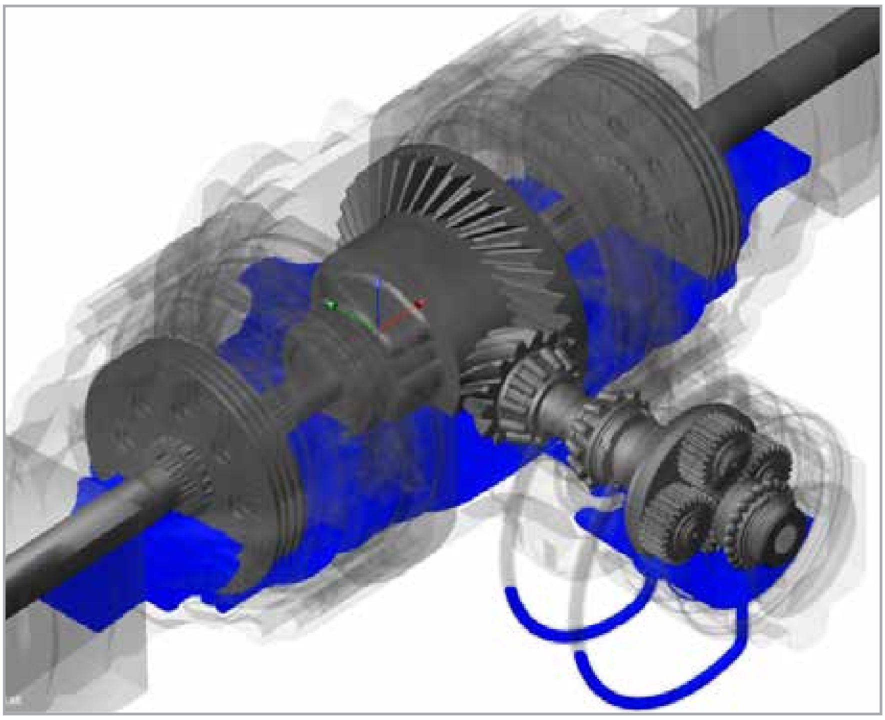
Transmission design is mainly based on the mechanical aspects of the transmission and lubrication is an aspect that is verified, and eventually corrected, based on bench testing, i.e. once the design phase has been completed and a physical prototype is available.

The use of transparent prototypes or windows in specific regions of the transmission makes it possible to visualize, at least partially, the flows and distribution of the lubricant within the transmission, and to understand whether it reaches the various components.

Similarly, by means of a physical prototype and bench tests, it is possible to verify the correct positioning of the breather ducts, which must be adequately protected in order to prevent the escape of the lubricant, and the functioning of the transmission under different operating conditions, for example by changing inclination, number of revolutions, or direction of rotation.

These are some of the issues faced daily by those who design and build transmissions of all kinds, from the automotive and industrial sectors to the aeronautical sector; from small transmissions to those for the naval and wind-power sectors.

However, waiting for an advanced stage of the project (i.e. once you have a prototype on the bench) to address the issue of lubrication can present surprises that can significantly impact both the development time of the transmission and its cost.



production of advanced engineering systems and mechatronics solutions for power transmission. The company operates in the fields of agricultural machinery, construction and forestry equipment, energy and industry.

Founded in 1970 by the Storchi family, today the company is led by the second generation. The president and CEO is Matteo Storchi.

Comer Industries has 1400 employees, exports to all 5 continents and has 7 offices in Europe, 1 in the United States, 1 in Brazil, 2 in China and 1 in India, including production plants and commercial branches it reaches 54 countries with its products and in 2020 it registered has a net turnover of 396 million euros. In March 2019, the company opened to external investors by listing on the Milan Stock Exchange's AIM market.

Discovering that some vital components, such as bearings, are not properly lubricated, or that there are oil leaks from the vents may require design changes, which can be costly at this late stage. In addition, there are some machine operating conditions that cannot be or are difficult to test on the bench, such asdynamic braking, acceleration, or particular temperature conditions.

To address these issues and to reduce the risks, costs and development times of transmissions, more and more companies are shifting the issue of "good lubrication" from the experimental verification phase to the actual design phase.

This has come about thanks to the availability of new numerical simulation technologies that enable the use of a virtual bench to test different operating and lubrication conditions quickly, and especially before building a physical prototype.

Models of this type complement and complete experimentation and, if used in the preliminary stages of the design, allow the project to be directed correctly and prevent lubrication or overheating problems.

One of the companies that has adopted these numerical simulation methods is Comer Industries.

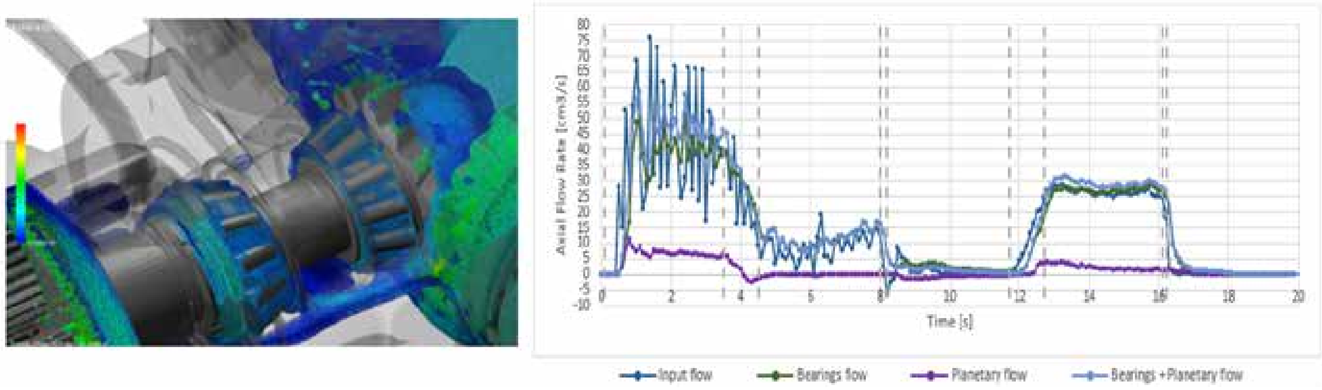
Comer Industries, based in Reggiolo in the province of Reggio Emilia, is the leading global player in the design and

Comer Industries is one of most important suppliers of the most relevant world players in the construction of agricultural, industrial and renewable energy machinery. The mechanical components made by Comer Industries represent crucial elements for the correct functioning of combine harvesters, tractors, plows, mowers, round balers, excavators, bulldozers and wind generators.

The market on which Comer Industries works is perfectly globalized and competition has imposed the ability to ensure customers products and assistance services with unique standards of excellence in every part of the world.
One of Comer Industries' numerical simulations for lubrication concerned predicting the path of the oil in an axle with an integrated planetary gear system input stage for a compactor.

During its operations, the compactor frequently travels uphill and downhill. The lubrication of the planetary gears is critical in these phases and must be guaranteed, as must the correct exchange of oil between the planetary gears and the axle to avoid dangerous increases in temperature.

Since the planetary gear system and the axle communicate via two oil passages, the analysis aimed to optimize the geometry of these two passages by recreating a meaningful operating condition in a single simulation consisting consecutively of a horizontal machine path phase, an uphill, a downhill, and a horizontal phase to return to the starting point.



All internal components of the axle and planetary gear system were included in the model along with two variants of the housing: the first with the existing oil passage geometry and the second with some proposed optimized oil passages. Finally, the oil was modeled at its working temperature properties: the model consisted of about six million particles.

The results obtained were very interesting and made it possible to evaluate:

- the oil redistribution between the two oil passages both qualitatively and quantitatively in terms of oil flow rates
- how this redistribution changes in the various phases - horizontal, uphill and downhill
- how the previous stage of the path affects the next stages, and how the oil behaves during transitions

It was found that, with the same oil quantity, the geometry of the current passages does not allow the oil to reach the planetary gears in all the configurations, thereby the oil exchange between the two environments is insufficient.

On the other hand, by enlarging and shaping the passages differently, it was possible to guarantee a greater flow of oil to the planetary gears and to ensure an adequate exchange of oil between the axle and the planetary gears that can avoid temperature raise.

This analysis was validated experimentally with some specific tests on a test bench: a high correspondence was obtained during the same work cycle (with the same inclinations) when comparing the simulated dynamic oil level with the experimental one.

This made it possible to implement the modifications to the oil passages without carrying out multiple experimental iterations which, since they affect the casting models, would have resulted in excessive time and costs related to potentially numerous remakes of equipment and components. In addition, there was no need to increase the oil level, so efficiency could be preserved.

About Comer Industries

Comer Industries, based in Reggiolo in the province of Reggio Emilia, is the leading global player in the design and production of advanced engineering systems and mechatronics solutions for power transmission. The company operates in the fields of agricultural machinery, construction and forestry equipment, energy and industry.

Founded in 1970 by the Storchi family, today the company is led by the second generation. The president and CEO is Matteo Storchi. Comer Industries has 1400 employees, exports to all 5 continents and has 7 offices in Europe, 1 in the United States, 1 in Brazil, 2 in China and 1 in India, including production plants and commercial branches it reaches 54 countries with its products and in 2020 it registered has a net turnover of 396 million euros. In March 2019, the company opened to external investors by listing on the Milan Stock Exchange's AIM market.

Comer Industries is one of most important suppliers of the most relevant world players in the construction of agricultural, industrial and renewable energy machinery.

The mechanical components made by Comer Industries represent crucial elements for the correct functioning of combine harvesters, tractors, plows, mowers, round balers, excavators, bulldozers and wind generators. The market on which Comer Industries works is perfectly globalized and competition has imposed the ability to ensure customers products and assistance services with unique standards of excellence in every part of the world.

EnginSoft wishes to acknowledge "Il Progettista Industriale" which first published this article in its magazine in April 2021.

For more information
Massimo Galbiati - EnginSoft
m.galbiati@enginsoft.com

American Magic perfects AC75 design for the 36th America's Cup

ESTECO technology plays a key role in foil and sail design

The America's Cup isn't just the first sailing competition in history, it's also the first when it comes to innovation. Learn how American Magic engineers partnered with ESTECO to prepare for their next challenge. Using modeFRONTIER in different phases of the design process, they integrated geometry changes, performed hydrodynamic CFD simulations and explored different optimization strategies.

That fly

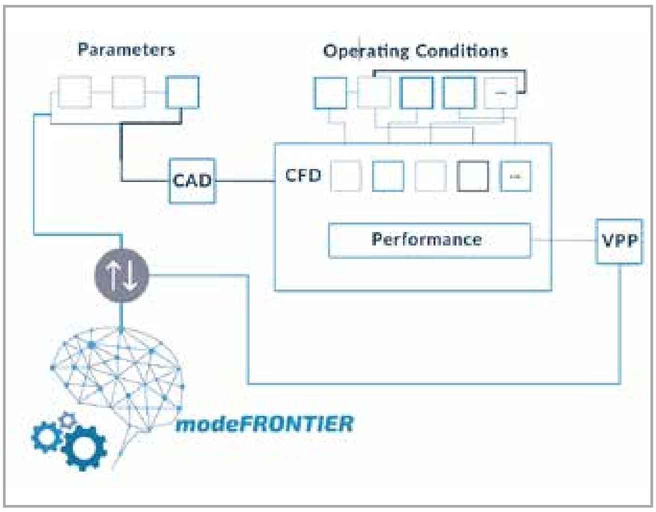
The America's Cup is the oldest and most important trophy in sailing. What makes it unique is that the reigning champion gets to decide the rules for the next edition, like the date and the location. More importantly, it defines boat class and design rules. The 33rd America's cup in 2010 pushed the boundaries of boat design by introducing new technologies, design concepts and materials. When BMW Oracle pitted its 34-meter trimaran and 55-meter high rigid wing against Alinghi in the first regatta, it won by 15 minutes, sailing at more than 18 knots in 8 knots of wind.

The 36th America's Cup builds on changes to previous edition class rules with a new boat concept: a monohull racing boat that doesn't sacrifice the concept of flying boats. Two t-shaped side foils guarantee that the boat flies above the water. Engineers can't rely on previous experience and now find themselves having to design a completely new boat. Moreover, competition rules allow teams to design only certain parts of the boat, like foil wings, sails, hulls and systems, while others must be designed by third-parties. Whereas the foil wing structure and profile can be designed by teams, the arm structure is determined, designed, and built by a supplier company.

American Magic is using ESTECO technology in each phase of the design process, from concept to the refinement of foils and sails. Specifically it uses modeFRONTIER. Its process automation, intelligent algorithms and advanced post-processing capabilities enable engineers to deliver optimized solutions faster.

Complex simulation studies on foil and mainsal geometries

The AC75 has two T-shaped side foils. The arm is attached to the hull with a moving joint which allows the crew to move the foils in and out of the water according to the sailing mode. On the other end of the arm is the foil itself. The foil has a main airfoil profile section coupled with moving flaps. Internally, enough space must be guaranteed for the systems to operate the flaps.



Foil design is challenging because it involves the simulation of myriad geometries in different configurations and under multiple operating conditions - all of which determine boat performance. Foils need to create low drag but generate enough force to enable the boat to lift at the start and fly during the race. Righting moment is required to balance the heeling moment of the sails. At higher speeds, cavitation can cause significant loss in performance. Stability is fundamental, especially during maneuvers.

In the first stage, hydrodynamic performance is computed using a low-fidelity solver that takes into account the different operating conditions - namely speed, sailing mode and position in the water. In the second stage, the full 3D geometry is designed and evaluated using CFD simulations. High-fidelity simulations can't be used directly in the first stage of development due to lengthy computational times.

The sail plan is composed of the mainsail and a jib or code zero, which are interchangeable. All dimensions are restricted by rules, so engineers can optimize the shape within specific limits. The mainsail is a twin skin sail that acts like an airfoil. By adjusting mast rotation, twist and boom position, the 3D geometry of the mainsail can be tuned for different wind speeds and sailing modes. As for foils, the aerodynamic forces are tuned to generate lift, maintaining momentum and low drag.

Getting foil and sail design just right

American Magic engineers are using several simulation software for the foil and sail design which consists of three steps: geometry definition, force computation and boat speed estimation. The entire process is automated in modeFRONTIER. Multiple workflows handle input modifications, the execution of different tools and file and data exchange.

Mares, developed by Airbus, handles the geometry generation of the airfoil and the flap, considering different combinations. Hydrodynamic forces generated by the foil, lift, drag and momentum, as well as cavitation speed are obtained through CFD simulations. The designers use a low fidelity 1D potential-based code in the initial phases to evaluate multiple configurations in a small amount of time. American Magic uses a RANS-based tool to perform high fidelity 3D simulations in the final refinement and optimization phase. Both Mares and the CFD tools are coupled with modeFRONTIER using Easydriver nodes. This enables them to couple their in-house tools using input and output files and customize execution scripts. Geometry consistency is guaranteed by constraints which filter out weird shapes and meet requirements for internal cabling and mechanisms as well as cavitation limits.

> *"Using modeFRONTIER process automation, intelligent algorithms and decision making capabilities enables us to decrease foil optimization time from 3 weeks to 4 days."*
>
> **Paolo Motta - American Magic**
> Performance Prediction Engineer

The main goal of the optimization is to minimize drag for specific lift values. It isn't enough to understand the efficacy of a foil shape. It's important to understand how the boat behaves. An in-house Velocity Prediction Program (VPP) software estimates the overall boat performance at different wind speeds and sailing modes. The software uses forces generated by foils and sails to find the overall boat equilibrium and predict boat velocity.

A nested modeFRONTIER workflow handles different operating conditions. These are sequentially run, using internal loops to compute the global performance of the design. Multiple operations are handled in parallel to make the most of computational resources. Once the forces are solved, these are passed on to the VPP calculation.

> *"Working side-by-side with ESTECO engineers enables us to pool our respective expertise to get the most out of modeFRONTIER."*
>
> **Giorgio Provinciali - American Magic**
> Velocity Prediction Program (VPP) Lead

Foil and sail design share most of the process and simulation tools but defining the geometry is more complex. The mainsail is divided into several sections where each section can have a different shape based on input values. On top of this, optimal adjustments for every shape need to be calculated. This results in large numbers of configurations which are run to find the best design. It's fundamental to formulate constraints based on maneuverability, considering adjustments that are feasible for the crew - optimal solutions have no meaning if they are too complex to be performed during the race.

Each design phase requires a different optimization strategy. In early stages, genetic algorithms guarantee robustness to find the global optimum in a large design domain. In the last phase, it's important to cut optimization time - conventional techniques aren't feasible. Therefore, multi-strategy algorithms are used in combination with advanced initialization techniques to speed up the whole optimization process.

American Magic and ESTECO - Partners in Innovation

The American Magic design team relied on ESTECO technology in the design and optimization of the boat. Paolo Motta, Performance Prediction Engineer says, "The AC75 is a complex racing boat with interacting subsystems. This makes the design process a challenging and time-consuming task. Using modeFRONTIER process automation, intelligent algorithms and decision making capabilities enables us to decrease foil optimization time from 3 weeks to 4 days. This gives us time to discuss and think about present challenges and develop new solutions".

According to Arthur Rozand, Performance Prediction Engineer, "The key benefit of using modeFRONTIER is to have a suite of tools in one place. In this way it's easy to manage design and optimization from the exploratory phase to post-processing and decision making. Time is a constraint in development. With modeFRONTIER we have the flexibility to tailor the strategy.
For example, in the early stages of development, DOE strategies and the sensitivity analysis tool help us understand which design variables are the most important. In the final stage of development we use multi-strategy algorithms and advanced charts to select the best design."

"Our partnership with ESTECO is bringing in great results," says Giorgio Provinciali, Velocity Prediction Program (VPP) Lead, "Working side-by-side with ESTECO engineers enables us to pool our respective expertise to get the most out of modeFRONTIER".

For more information:
info@esteco.com

About Esteco

ESTECO is an independent software provider, highly specialized in numerical optimization and simulation data management with a sound scientific foundation and a flexible approach to customer needs. With 20 years' experience, the company supports leading organizations in designing the products of the future, today.
esteco.com

About American Magic

Formed in October 2017 by Bella Mente Racing, Quantum Racing and the New York Yacht Club, American Magic represents a joint vision to win the America's Cup, the highest prize in sailing and the oldest trophy in international sports. American Magic brings together two highly successful racing programs with one of the foremost yacht clubs in the world, united by a campaign to win back the Cup, reconnect the American sailing base with the premier event in the sport and elevate the quality of competitive sailing in the United States. The name, American Magic, is a nod to the New York Yacht Club's storied America's Cup history; a combination of the boat the trophy is named for, and the first boat to defend it.
americanmagic.americascup.com

Azimut Benetti propels cruise speed with modeFRONTIER

Using modeFRONTIER to perform multi-objective cavitating propeller optimization

Azimut Benetti Group is the world's largest network producing megayachts and the leading private group in the luxury yacht industry. Azimut-Benetti's R&D Centre develops unique technologies, for an effortless and safe navigating experience. The Naval Architecture and Marine Engineering Unit (DITEN Department) of Genoa University work jointly with DETRA Custom Propellers and Azimut Benetti's R&D Centre, using modeFRONTIER to optimize the design of a custom propeller for a high-speed Azimut Benetti 95 RPH yacht.

CHALLENGE

The design of a propeller is always a trade-off between competing objectives and constraints: maximizing the propulsion efficiency and ship speed while avoiding cavitation and maintaining a sufficient blade strength. The traditional lifting line/surface methodologies define the propeller shape by including simplified geometric assumptions that make them unsuitable for modern fast propeller design. The application of more accurate flow solvers and the automatic investigation, possible through the parametric description of the geometry (unconventional combinations of pitch, camber, or, for instance, local hydrofoil shapes), proves to be a successful design alternative for a high-speed propeller.

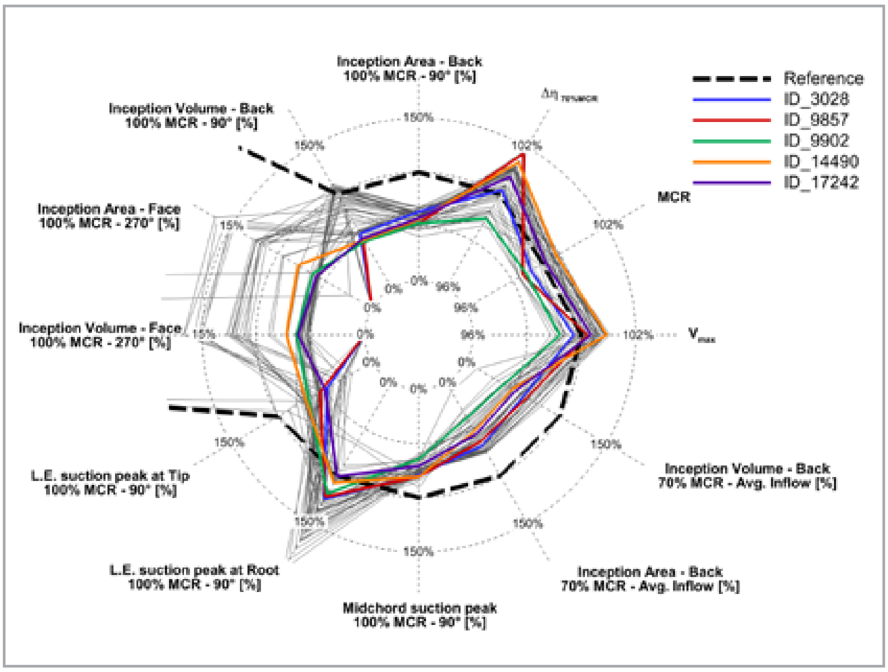
> *"modeFRONTIER helped increase cruise speed and reduce cavitation in marine propellers"*

SOLUTION

Following this new approach, the optimization of a reference propeller with modified rake distribution was driven by the MOGA-II, the genetic algorithm included in the automation workflow in modeFRONTIER. The experimental data collected at the cavitation tunnel confirmed the

About Azimut Benetti

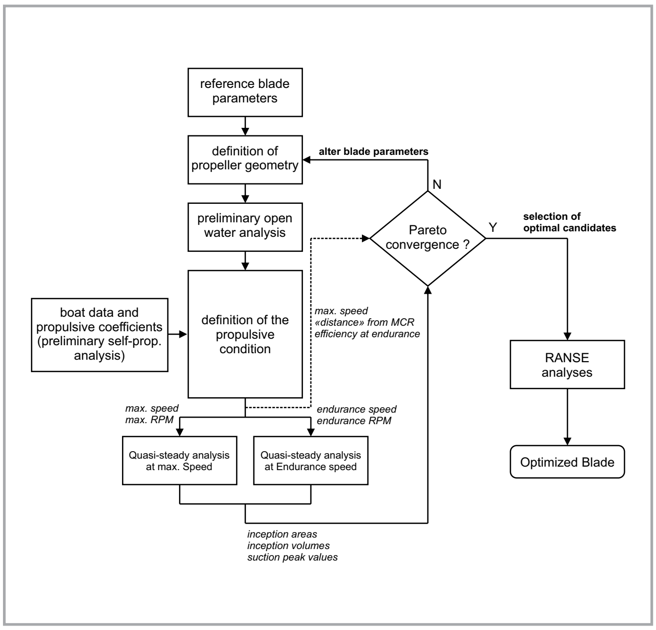
Azimut Benetti Group is the largest network producing megayachts and the world's leading private group in the luxury yachting industry. The company operates and produces in its 6 shipyards and with more than 40 models in production, the two brands Azimut and Benetti offer the broadest range in the world. The group accompanies its manufacturing excellence with a particular focus on services for its customers offering financial services, designing, planning and building new tourist marinas, providing customers with a luxury showroom, and training crews.
azimutbenetti.it



The modeFRONTIER optimization environment was selected to build the optimization flow reliability of both the Boundary Elements Method and RANSE numerical approaches. A dedicated full-scale sea trial, performed with propellers manufactured by Detra, showed that the cruise speed achieved with the optimized propeller is 1 kn higher than the baseline propeller speed geometry while the cavitating behavior was also significantly enhanced. "The result is remarkable, especially keeping in mind that the increase of cruise speed, together with the enhancement of comfort onboard, is crucial to the perception of luxury yacht customers," said Francesco Serra, R&D Office, Azimut Benetti Group.

modeFRONTIER ADVANTAGES

modeFRONTIER helped build an optimization framework to interact with the parametric description of the geometry to define each new blade shape and employ flow solvers to quantify how each propeller fulfills the constraints and the objectives of the design. "Starting from a set of 48 blade parameters to alter the reference propeller geometry, the use of MOGA-II algorithm allowed to compute and test 50,000 different geometries in about 5 days to achieve a satisfactory Pareto convergence and choose optimal candidates (one for any rake distribution) for RANSE analyses," said Michele Viviani, Associated Professor at DITEN Department, Genoa University.



Pareto diagram. Optimization with the modified rake distribution

For more information:
info@esteco.com

Supporting customers gearing up for digital transformation

EnginSoft reorganizes and broadens its training packages

The industrial world is in a very "operational" phase regarding the articulation of Digital Transformation – a phase that also requires the conscious application of numerical methodologies that are instrumental to its implementation. This "consciousness" requires adequate training in the numerical tools to be applied, which is why EnginSoft has decided to renew and enrich its training courses for the modeFRONTIER platform, a tool that is particularly well suited to the intelligent integration of technologies, processes, and digital know-how.
The modeFRONTIER training package has been updated both in content and form, in an effort to incorporate all the various potentialities of the modeFRONTIER platform:

- In terms of content, the training solutions have been increased from the four currently available (ie. Basic, Advanced, Model-Based Process Integration, and Robust Design) to a total of seven, with three new courses being introduced on specific topics, namely Data Analytics, Optimization Algorithms and Metamodeling.
- In terms of form, a graphic solution was found that allows an immediate and intuitive understanding of how the different courses are structured in terms of content. Each petal in the multi-petalled flower represents a thematic area while the petal's width indicates the level of detail covered in each specific topic. Some courses (eg. Basic, and Advanced), covering multiple topics, engage more "petals", while others, being very specialized in one topic, are "single petal".

The courses have been structured to guide the user along a learning path that allows them to increase their degree of proficiency in the application of various techniques. To this end, the courses provide a series of practical exercises supported by the discussion and application of "best practices" developed by the instructors in the field during their work in industrial and non-industrial activities, in addition to the discussion of basic theory.

www.enginsoft.com/training/corsi-a-calendario-ottimizzazione.html

by Sunggeun Park
TSNE

Analysis of the behavioral characteristics of detergent powder using Rocky-Ansys Fluent two-way coupling

Ansys Fluent has a variety of particle analysis methods, but sticky particles are particularly difficult to analyze. Rocky, a specialized particle analysis program that can analyze various shapes and sticky particles, and ductile analysis are needed. This analysis attempts to understand the behavior of sticky detergent powders according to flow velocity using Rocky-Ansys Fluent two-way coupling.

When powdered washing detergent is used, water enters the detergent drawer and injects the detergent into the washing machine. As shown in Fig. 1, as the detergent powder gets wet and adheres, the adhesion increases, and the powder remains in the detergent drawer.

While Ansys Fluent has several particles analysis methods, most particle analysis methods are not suitable for this case. Therefore, a method that uses Rocky-Ansys Fluent coupling, a specialized particle analysis program, is required. Rocky can implement both spherical and non-spherical particles, as well as particle stacking and sticky particles. In addition, Ansys Fluent coupling can be implemented very easily, and the flow analysis grid can be greatly reduced. If many particles are present, the GPU can be used to dramatically shorten the analysis time.

In this article, we delve deeper into the analysis of the behavioral characteristics of detergent powder according to water flow velocity using Rocky-Ansys Fluent two-way coupling. This analysis was conducted to determine the possibility of interpretation without precise conditions.

Fig. 1 – Phenomenon of residual detergent powder

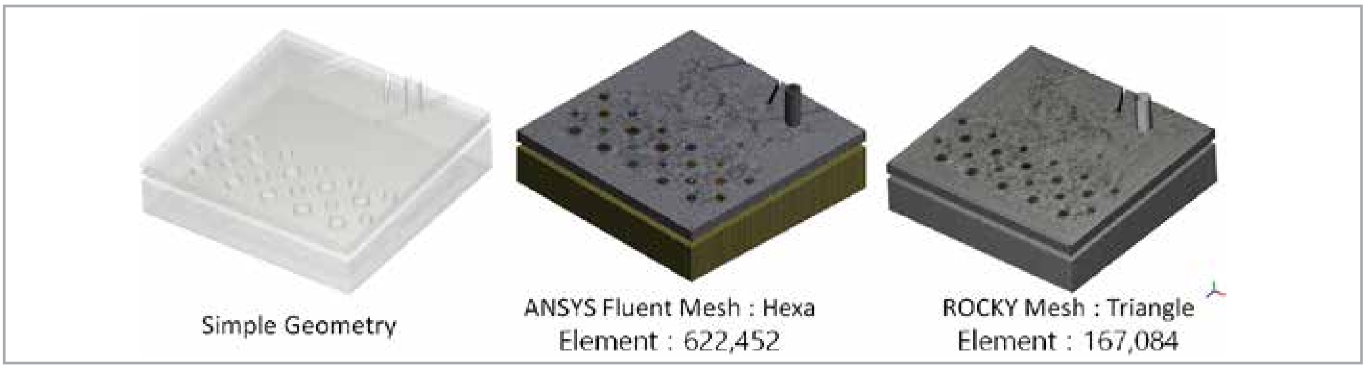


Fig. 2 – Simple geometry and mesh

Geometry and mesh

As shown in Fig. 2, the behavior of detergent powder is determined using simple geometry. By creating an Ansys Fluent case file, Rocky can use a boundary in the Fluent case. The grid is a full hexahedral type and about 620,000 elements. In Rocky, the surface grid is about 170,000 elements and the quadrilateral grid is automatically converted to a triangular one.

Ansys Fluent configuration for coupling analysis

There are two ways to conduct Rocky coupling. If only one fluid is present, it is set to a single phase and treated with porous using volume fractions of the particles. If two or more fluids are present, a multiphase model must be used, and only the Eulerian model is possible. This case requires a three-phase analysis using the Eulerian model since water, air, and detergent particles are present. The realizable k-e model and scalable wall function were used for the turbulence model, and the turbulence multiphase model was set to dispersed.

Configure Ansys Fluent as follows for two-way coupling:

1. 3D double precision
2. Transient
3. Gravity
4. Dispersed turbulence model for multiphase (using the Eulerian model)

The properties of each fluid use Ansys Fluent's default values for water and air. The properties of the particles are set by Rocky. Particle density is set the same way and viscosity is not required, so use the default value or set it to 0.

The input sets the velocity inlet condition where the velocity is 2m/s and 5m/s, respectively. The output sets the relative gauge pressure to 0Pa in the pressure outlet condition.

The wall is set to stationary and non-slip conditions. In solution method, pressure is set to body force weighted, transient formulation is changed to bounded second order implicit, and the default values are used for the rest. Standard initialization is used, and air volume fraction is set to 1. Finally, save the case file. This case file is available to Rocky, so it is not necessary to create a separate surface mesh for Rocky.

Rocky configuration for coupling analysis

Rocky settings for stacking of detergent powder

After stacking the detergent powder, water is added into the washing machine. First, it is necessary to stack the detergent powder using Rocky: the stacked state is the initial condition. After coupling with Ansys Fluent, the behavior of the detergent is analyzed when water enters.

The normal force hysteretic linear spring model is used by adding the Leeds Contact Model and the tangential force uses the Linear Spring Coulomb Limit Model. To consider the sticky properties of the particles, use the Leeds Contact Model among the adhesive models. To consider rolling resistance, select Type C: Linear Spring Rolling Limit. Gravity (-9.81m/s^2) was set in the Y direction, and heat transfer was not considered. The detergent powder particles are spherical and have a size of 0.3mm. The total mass input is 5g, and the rolling resistance is set to 0.2.

The density of detergent powder is 2,100kg/m^3, and the bulk density is 1,260kg/m^3. Bulk Young's modulus is 0.1GPa, and Poisson's ratio is 0.2.

The detergent drawer is plastic, the density is 1,100kg/m^3, Young's modulus is 3GPa, and Poisson's ratio is 0.2. It establishes particle-particle interactions and particle-wall interactions. The static and dynamic friction of the particle-particle interactions is 0.55, and the coefficient of restitution is 0.1.

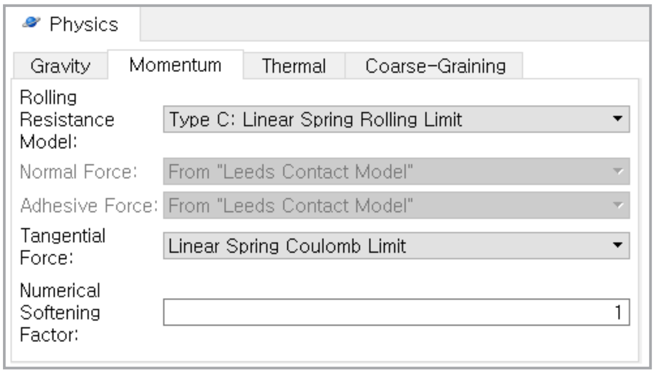


Fig. 3 Momentum of Physics

For the adhesive force, the damping ratio was set to 0 and the surface energy was set to 0.0003j/m^2. The static and dynamic friction between the particle-wall interactions is 0.2, and the

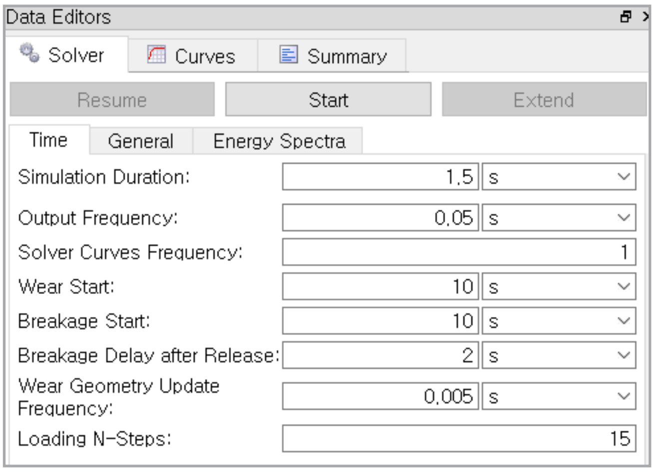


Fig. 4 Solver setting

coefficient of restitution is 0.1. The analysis time is 1.5 seconds, and the analysis is performed using the GPU (Tesla P100).

During the analysis, stacking particles can be checked for each time as shown in Fig. 5. When the stacking is complete, save the 1.5 second result as an initial condition, as shown in Fig. 6. We configure the coupling with Ansys Fluent from the initial conditions saved.

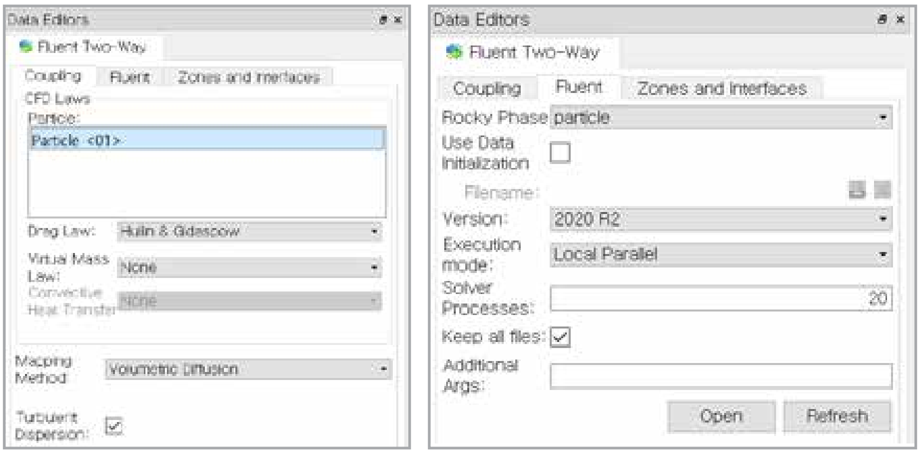


Fig. 7 – Settings for the Rocky-Ansys Fluent two-way coupling

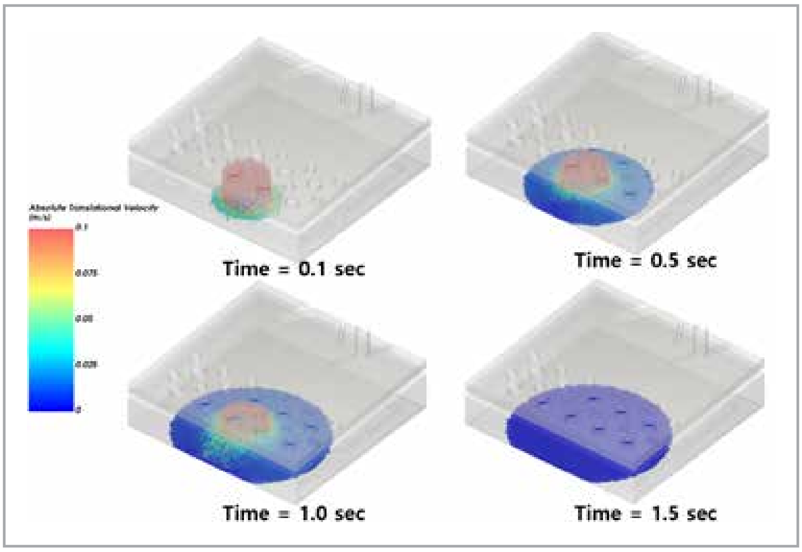


Fig. 5 Stacking particles over time

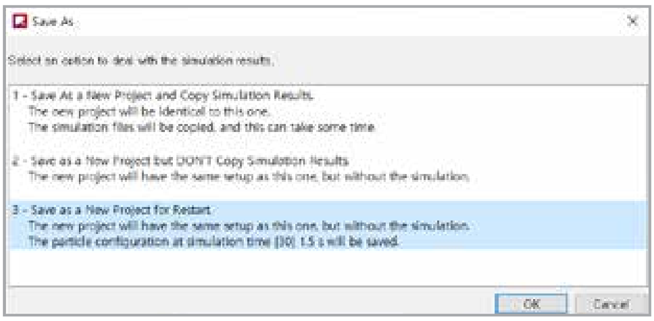


Fig. 6 Save as a new project for restart

Rocky settings for Ansys Fluent coupling

All other conditions are the same as before, so only the CFD coupling needs to be configured. In CFD coupling, select coupling mode as Fluent two way and then select the Ansys Fluent case file. Select drag and virtual mass law from the CFD law and Huilin & Gidaspow from drag law (see Fig. 7).

Turbulence dispersion is similar to discrete random cloud in Ansys Fluent. In the Fluent tab, Rocky phase selects particle phase from secondary. Then select the version of Ansys Fluent to use and enter the number of parallel cores. Configuring Rocky-Ansys Fluent two-way coupling is remarkably simple. Rocky 4.4.3 supports from Ansys R19.2 to Ansys 2021 R1.

Determine the total analysis time and storage period in the solver and press start. Rocky automatically executes Ansys Fluent (stand-alone) to load the saved Fluent case file, and then exchanges data between them each time.

Results

In Rocky you can only visualize the particle behavior. Ansys Fluent only allows you to view the flow analysis results with the volume fraction of the particles.

EnSight was used to visualize the flow results and particle behaviors together. The particle position over time was exported from Rocky and the Ansys Fluent analysis results were imported into EnSight.

Fig. 8 shows the behavior of the detergent and volume fraction of the water according to the water flow rate over time. When the water flow rate is 2m/s, after 0.5 seconds the water is slow, so the detergent is not washed away, and most of the detergent remains after 5 seconds.

After a little more time, you can see that the water supply has been built up. Because of the low flow rate, the sticky particles cannot be washed down.

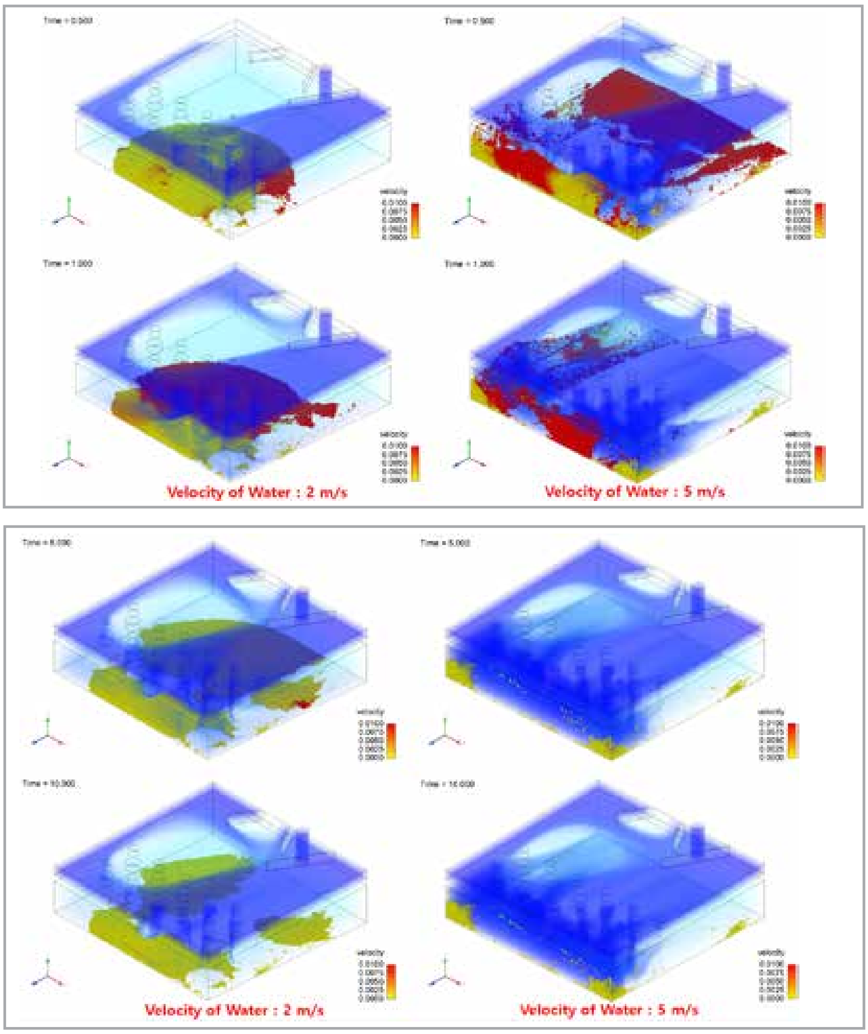


Fig. 8 – The behavior of the detergent powder according to the water flow rate

However, when the water flow rate is 5m/s, you can see that after 1 second most of the detergent is washed away and the remaining detergent particles have collected in the corner.

It can be seen that the main parameter for introducing the detergent powder into the washing machine is the flow rate. In addition, the initial position of the detergent powder is especially important.

Conclusion

Rocky-Ansys Fluent two-way coupling was used to confirm the behavior of sticky particles of detergent powder according to the flow rate of water.

In this way, Rocky enables sticky particle analysis for different particle shape, size and adhesion. Therefore, Rocky-Fluent coupling can be used to overcome the limitations of Ansys Fluent, making fluid-particle analysis available for use in various industries.

For more information
Sunggeun Park - TSNE
sgpark@tsne.co.kr

About TSNE

Since its establishment in 1988, TSNE has specialized in CAE, providing engineering programs and services to Korean customers. Tae Sung S&E (TSNE) aims to be the “One Stop Total CAE Solution Provider” (OSTS) both in domestic and global markets.

TSNE leverages its large base of business capabilities and its team of CAE experts to provide services to customers in various industries (aerospace, automotive, civil engineering, biomedical, shipbuilding, electrical and electronics, energy, defense, chemical industries, etc.) and is expanding its business scope to research innovative technologies and apply them in the field.

The company is striving to become a global engineering company and increase its potential to become a sustainable engineering company. Tae Sung S&E is partner to all engineers who strive to solve challenges. Tae Sung S&E will work with you to achieve “NO PROBLEM, BE HAPPY”.

Addressing the issue of composite modeling in CAE analyses

Multiscale.Sim accurately predicts the material properties of composites

by Koji Yamamoto
CYBERNET SYSTEMS CO., LTD.

This paper introduces Multiscale.Sim, an analysis tool for predicting the material properties of composites, which addresses a common issue in CAE analysis. A simple analytical example is provided to show how this product works. While the examples presented here are limited, the product continues to be vigorously developed to make it more versatile and enable it to deal with a variety of problems.

A recent NAFEMS survey [1] revealed that the biggest challenge in CAE analyses that involve composite materials is the lack of a materials database. This is largely due to two of the characteristics of composites: they are anisotropic and can be freely mixed. Their anisotropic nature means there are a variable number of constants which makes the material testing process very cumbersome.

For instance, if we consider the most common type of anisotropy – orthogonal anisotropy – the CAE analysis would require a total of nine material constants including the modulus of elasticity, the shear modulus and Poisson's ratio for three directions. By comparison, isotropy only has two constants – Young's modulus and Poisson's ratio.

Another complicating factor is that, unlike metal materials, composites are both manufactured and molded at the same time, so the material properties may vary depending on various conditions such as the combination of raw materials used, or the molding history and so on.

This means that composites have a great variety of material properties, which makes it impractical to evaluate all of them in actual material testing. As a result, virtual material testing methodologies, particularly those using the finite element (FE) method, have been attracting attention. CYBERNET SYSTEMS CO., LTD. has developed Multiscale.Sim, an analysis tool for efficiently executing virtual material testing.

What is Multiscale.Sim?

Multiscale.Sim is the outcome of a collaboration project between industry and academia that started in Japan in 2007. The tool is an add-in for Ansys, one of the most popular general-purpose CAE tools, and therefore is easy to use. Fig. 1 provides an overview of how a virtual material test functions in Multiscale.Sim.

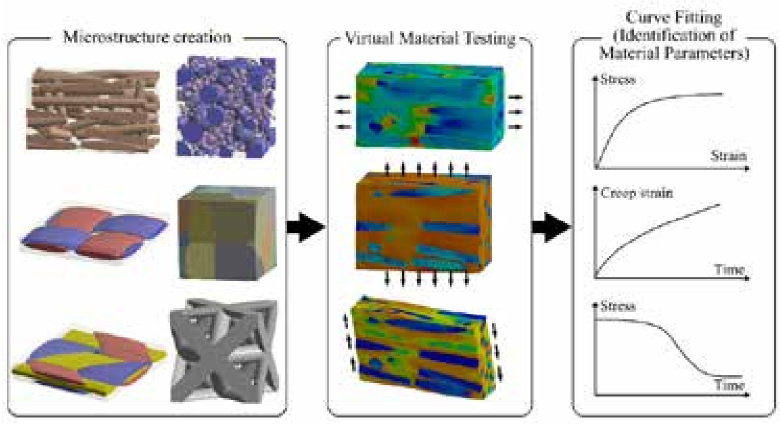


Fig. 1 – The three steps of virtual material testing in Multiscale.Sim

In virtual material testing, the non-homogeneous microstructure of a composite material is the target of the analysis. The first step therefore, is the creation of a unit cell FE model that represents a microstructure.

Since microstructures are generally very complex, which makes it difficult to create the model manually, Multiscale.Sim enables them to be generated automatically based on their geometric information, such as the volume fraction of the fiber, or the fiber orientation and so on.

Virtual material testing is then done using the model created. Test loads can be arbitrarily assigned to achieve any mode of deformation, taking into account cyclic symmetry. Like an actual test, the user defines the ideal stress state without creating a dumbbell-shaped specimen.

The results of the analysis provide the macroscopic material response for each deformation mode as an output. Next, these results are fitted to the curve to identify the material constants (e.g. Young's modulus from the tangential slope if the material exhibits elastic properties). In Multiscale.Sim, all these steps are performed automatically. The software can be set to generate a case study to evaluate the material constants in the composites for all conditions overnight while you sleep.

Advantages of Multiscale.Sim

These Multiscale.Sim features offer three advantages:

- ***The material properties of composites can be evaluated without performing an actual test.***
  All material properties of composites can be evaluated by virtual material testing as long as the microstructure and material constants of the base material are available. Recently, so-called material informatics, which efficiently determines the optimal material for a product using material information accumulated as big data, has been in the spotlight. A combination of virtual material testing is essential to achieve this goal.
- ***It significantly improves the computational efficiency of CAE analysis.***
  Directly representing these non-homogeneous microstructures in product-scale analysis models would be very expensive computationally. Evaluating the equivalent material constants in Multiscale.Sim is almost the same as simplifying the FE model, without compromising the accuracy of the analysis. For example, it is often time consuming to analyze non-homogeneous structures, such as perforated metals and lattice structures; Multiscale.Sim helps to overcome this problem.
- ***The results in the microstructure can be observed in situ.***
  Virtual material testing allows the response inside a non-homogeneous microstructure to easily be observed in situ. While digital image correlation (DIC), for example, allows distributions to be evaluated, the resolution is not high enough to observe the details of the microstructure and only the surface can be observed.

Example analysis of composites of cross-woven glass-fiber reinforced polymer (GFRP)

We conducted an example analysis to evaluate the material properties of the common plain weave. The analysis was conducted on a composite material composed of glass fibers and epoxy resin commonly known as FR4, which is used as a base for electric circuit boards (PCB). Fig. 2 shows an overview of the analysis model.

First, we observed a cross-section of the material using an electron microscope to source geometric information such as the volume fraction in the fiber bundle and the pitch between the adjacent bundles. These results were used to create a model of the microstructure, as shown in Fig. 2 (b) and (c).

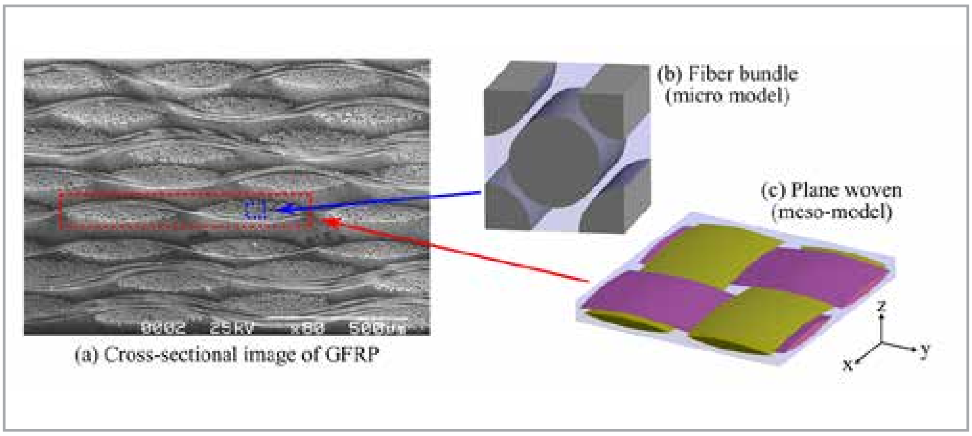


Fig. 2 - Overview of analysis example

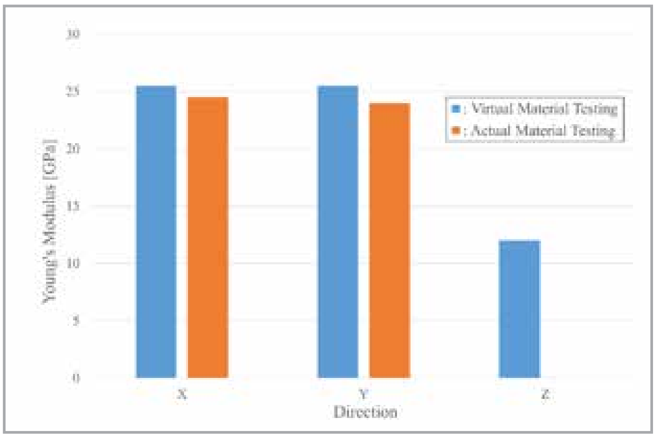


Fig. 3 – Elasticity measurements from actual and virtual tests

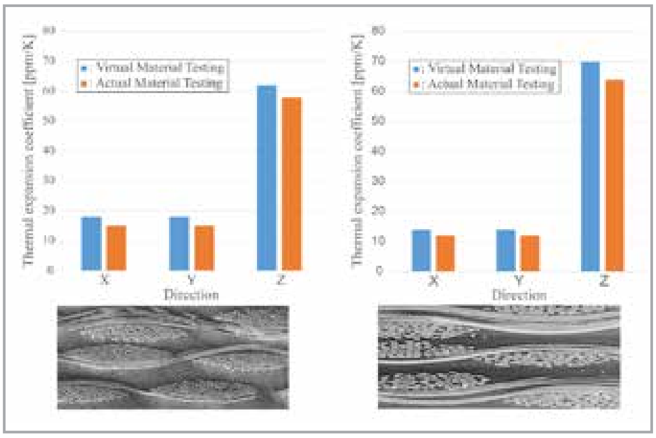


Fig. 4 - Thermal expansion coefficient measurements from actual and virtual tests

We refer to them as the micro-model and meso-model respectively in this article.

Modeling a monofilament to a meso-model is not practical. The fiber bundle parts are, therefore, modeled as homogenized parts in the meso-model, for which the anisotropic properties are obtained from virtual material tests on the micro-model.

The equivalent material properties obtained from the tests on the meso-model are validated against the results of the actual tests. Fig. 3 summarizes these results.

Only the longitudinal modulus of elasticity and the coefficient of linear expansion in the three normal directions are described in a representative manner. Axis z corresponds to the out of plane direction of the thin plate shape. While it was impossible to conduct an actual tensile test for this direction, virtual material testing easily obtains the results.

The composite material in the example has a strong anisotropic character because the gap in material properties between the glass fibers and the epoxy resin is so large. The close correlation of the two results confirms the validity of the analysis results.

Fig. 4 shows the results of a similar analysis using a slightly different microstructure. Here, the cross-sectional shape of the fibers is modelled as a circle and a quadrangle. In the actual test, we were able to confirm that the material properties of GFRP woven materials depends on the cross-sectional shape of the fibers. Changing the cross-sectional shape of the glass fibers from a common round shape to a square lowers the coefficient of thermal expansion (CTE) in the in-plane direction and increases the CTE in the out-of-plane direction. These trends are well captured analytically as well.

Conclusion

Multiscale.Sim addresses an important issue in CAE analysis of products based on composite materials. While the analyses provided in this article were quite simple, the software continues to be developed to increase its versatility and its ability to address a variety of problems.

CYBERNET believes that Multiscale.Sim can help engineers in their R&D of products using composite material by simplifying the material modelling necessary for accurate CAE analyses.

A special thanks to EnginSoft S.p.A. for its collaboration. EnginSoft is Cybernet's Channel Partner in the distribution of Multiscale.Sim in the EU.

EnginSoft wishes to acknowledge "Compositi" which first published this article in its magazine in March 2021.

For more details visit:
www.enginsoft.com/solutions/multiscale.html

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

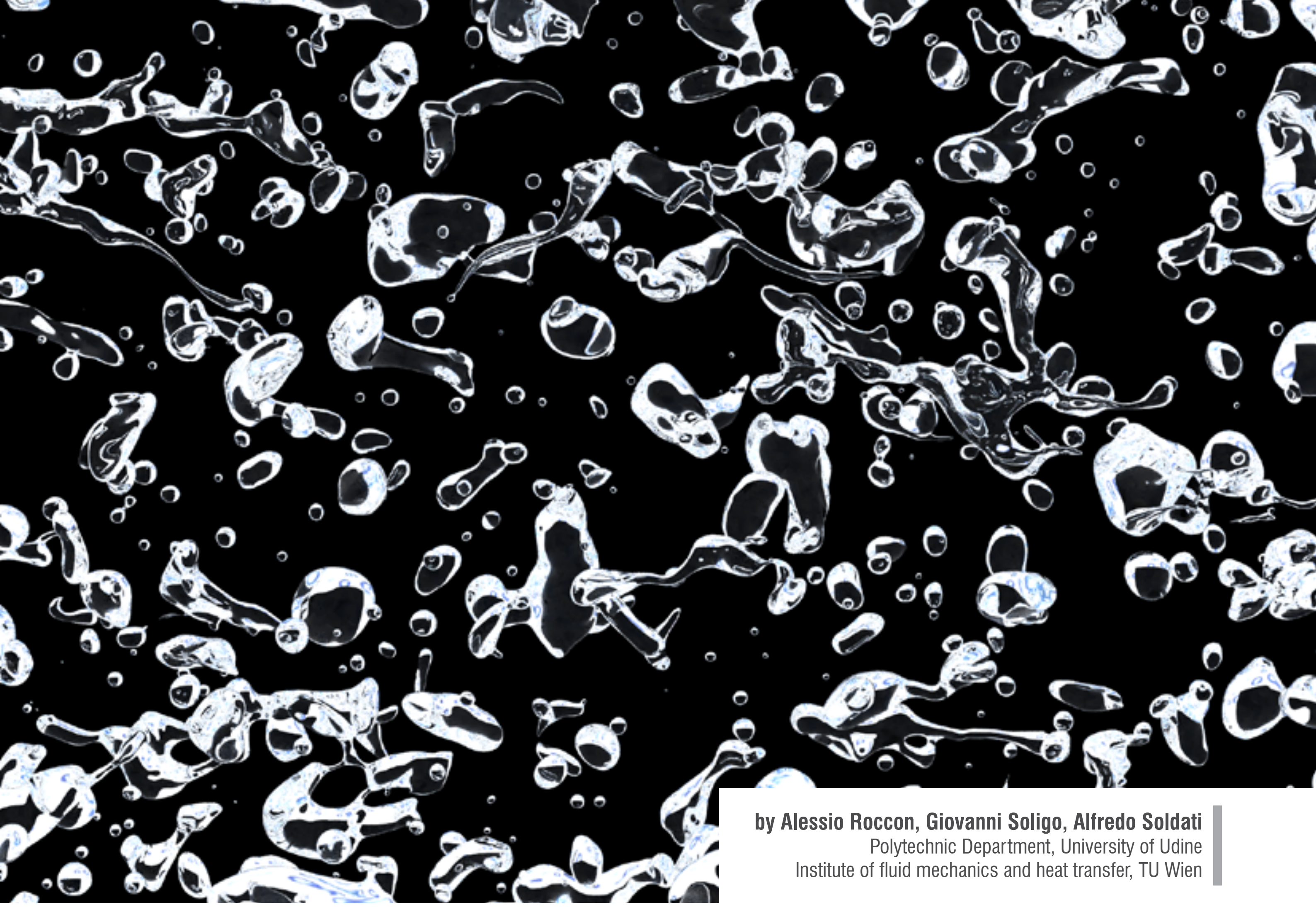
About Cybernet Systems

CYBERNET SYSTEMS CO., LTD., an Ansys software reseller and a solution provider for the past 35 years, provides various analysis solutions based on techniques that have been developed over time. CYBERNET has also been recognized as an Elite Channel Partner and a Solution Partner (Software) by Ansys.

For further information, visit:
www.cybernet.co.jp/ansys/product/lineup/multiscale/en/
or email: cmas@cybernet.co.jp

Reference

[1] NAFEMS, Composite Survey 2014 (www.nafems.org/publications/resource_center/r0112/)



by Alessio Roccon, Giovanni Soligo, Alfredo Soldati
Polytechnic Department, University of Udine
Institute of fluid mechanics and heat transfer, TU Wien

Studying the size distribution of surfactant-laden droplets

Improving the accuracy of numerical tools to accurately determine transfer rates across deformable interfaces

Accurate prediction of momentum, heat, and mass transfer across a deformable interface is a fundamental problem that plays a key role in a wide range of environmental and industrial applications, from the formation of raindrops in the atmosphere to fragmentation and evaporation processes in combustion chambers. Mathematical models and empirical correlations based on macroscopic observables, such as interface shape and extent, are often used to estimate these transfer rates; these correlations, although mostly tuned to experimental data, can often provide an initial estimate of transfer rates. To improve the accuracy of these predictions and gain a better understanding of the underlying physics, the development of numerical tools capable of performing high-fidelity simulations is highly desirable.

Multiphase flow simulations are extremely challenging since the temporal evolution of a deformable, topologically changing, and continuously moving interface must be described. In recent years, several numerical techniques have been developed with this goal in mind.

These methods can be classified as interface tracking methods, which explicitly track the interface using a set of marker points; and interface capture methods, which implicitly define the interface using a constant value of a color function.

Here, to describe the dynamics of a swarm of surfactant-laden droplets, we use an interface capture method known as the phase field method. In the phase field method, a scalar variable (the so-called order parameter) is used to describe the local concentration

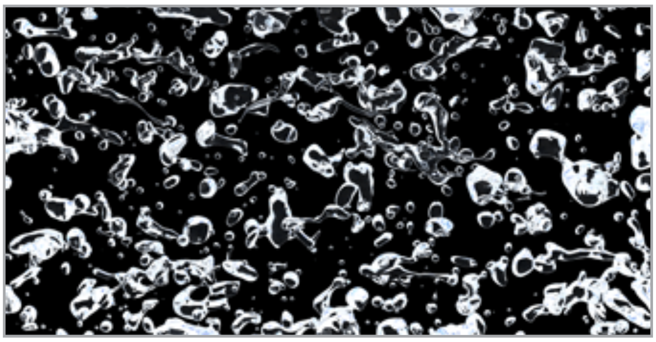


Fig. 1 – Snapshot of droplet shape (zero level of phase field) for the lowest value of surface tension considered (Weber number, We = 3.00) and for the strongest surfactant (elasticity number, β_s = 4.00).

of the two phases; the order parameter has a constant value in most of each phase and undergoes a smooth transition through the interfacial layer. In this work, we also consider the presence of surface-active agents (surfactants) to better mimic real-world applications.

Surfactant molecules are amphiphilic (they have polar heads and non-polar tails) and tend to accumulate at the interface, where they produce a local decrease in surface tension. Within the phase field method, the presence of surfactants is accounted for by introducing an additional order parameter that describes the surfactant concentration in the entire domain. This additional variable is uniform across most of the two phases and reaches its maximum value at the interface (where surfactants preferentially gather).

In this work, we numerically investigate the breakup, coalescence and ultimately the size distribution of surfactant-laden droplets in wall-bounded turbulence. We combine direct numerical simulations of turbulence with a phase field method used in a two order parameter formulation (phase field and surfactant concentration). We consider droplets characterized by two different values of surface tension, fixed by Weber number (the ratio of inertial to surface tension forces), and four different surfactant strengths, fixed by elasticity number (the parameter quantifying the magnitude of surface tension reduction produced by the surfactant).

Methodology

Multiphase turbulent flow dynamics are described by performing a direct numerical simulation of the Navier-Stokes equations coupled with a two order parameter phase field method (Soligo et al. 2019). The Navier-Stokes equations, suitably modified to account for the presence of a surfactant-laden interface, describe the flow field while the phase field method is used to calculate the dispersed phase morphology and surfactant concentration.
The resulting system of equations is solved in a closed-channel geometry using a velocity-vorticity formula. The equations are discretized using Fourier series along the streamwise and spanwise directions while Chebyshev polynomials are used along the direction normal to the wall. All unknowns (three components of the velocity vector and the two order parameters) and their governing equations are Eulerian and are solved on the same Cartesian grid.

Breakage and coalescence

Once released into the turbulent flow, surfactant-laden droplets can interact with neighboring droplets and background turbulence (see Fig. 1). The result of the droplet-droplet and droplet-turbulence interactions can be classified from a topological point of view. In particular, we can distinguish two types of events: coalescence (merging of two interfaces) and breakup (creation of new interfaces).

A breakup event can be divided into four phases (Fig. 2, top): i) deformation – under the action of shear stresses a droplet is

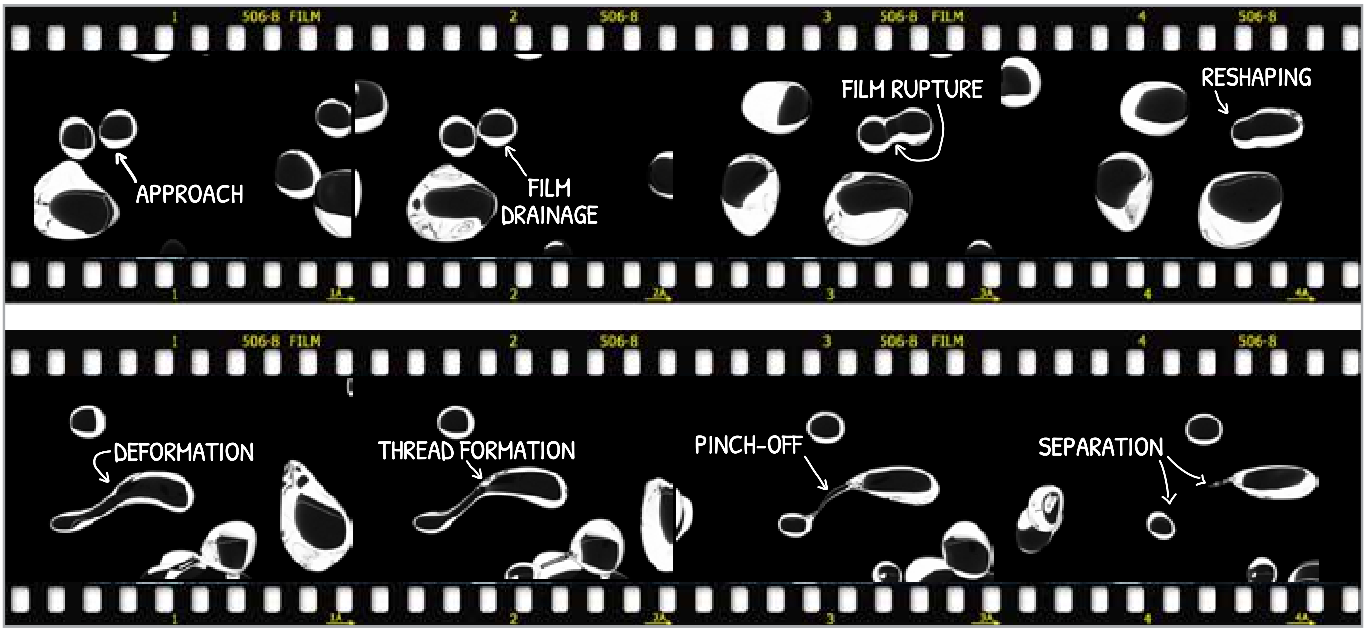


Fig. 2 - Top row shows the time sequence of a coalescence event: approach, film drainage, film rupture and reshaping, while the bottom row shows the time sequence of a breakage event: deformation, thread formation, pinch-off and separation.

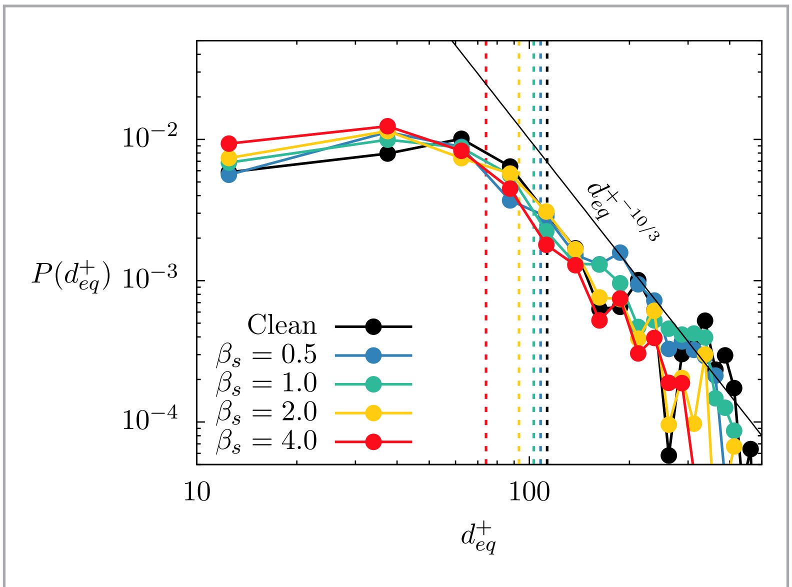


Fig. 3 - Droplet size distributions (DSD) reported in log-log scale. The results refer to We = 3.00 and colors are used to identify the different cases black (surfactant-free), β_s = 0.50 (blue), β_s = 1.00 (green), β_s = 2.00 (yellow) and β_s = 4.00 (red). The theoretical scaling d+ −10/3 is reported with a thin continuous white line. The Hinze inviscid scales for each case are reported with vertical dashed lines (same color codes as the distributions).

deformed; ii) thread formation – the droplet is further stretched and a ligament is formed; iii) pinch-off – the ligament is further stretched and capillary instabilities pinch off the liquid thread; iv) thread breakage – the liquid thread breaks, and the newly formed droplets separate.

Likewise, a coalescence event can be divided into four phases (Fig. 2, bottom): i) approach – the two droplets approach each other; ii) film drainage – the thin liquid film between the droplets begins to drain; iii) film rupture – molecular forces lead to the rupture of the thin liquid film separating the two interfaces, and the coalescence bridge is formed; (iv) reshaping – surface tension forces an attempt to restore the quasi-spherical shape of the droplet.

To study the effects of surfactants on dispersed phase morphology, we consider the droplet size distribution (DSD). The physical mechanisms that generate DSD are complex and difficult to untangle; however, in general, the droplet size distribution can be viewed as the result of the ultimate competition between coalescence and breakup events. Based on physical considerations, we can assume that larger drops are more likely to break into smaller drops, while smaller drops are less likely to be broken and coalesce more frequently. In fact, whether or not a droplet breaks is determined by the balance between shear forces, which seek to break the drop, and surface tension forces, which seek to restore the spherical shape of the droplet. Using this balance of forces, we can calculate the critical size of a droplet that will not break. This critical diameter is commonly identified as the Hinze inviscid scale and can be used to distinguish between two different regions of the droplet size distribution: the coalescence-dominated regime (for droplets smaller than the Hinze scale) and the breakage-dominated regime (for droplets larger than the Hinze scale).

The focus here is on the breakage-dominated regime; for this regime, Garrett et al. proposed a power law scaling based on the droplet diameter with the exponent 10/3. The results obtained from the simulations at We = 3.00 (lowest surface tension value) are shown in Fig. 3. The different cases are identified with different colors: black (surfactant-free), β_s = 0.50 (blue), β_s = 1.00 (green), β_s = 2.00 (yellow) and β_s = 4.00 (red). The theoretical scale d+ −10/3 is shown with a thin continuous black line. Finally, the Hinze inviscid scale for each case is shown with a vertical dashed line (same color codes as the distributions). As can be observed from Fig. 3, the DSDs obtained are in good agreement with the theoretical scaling. The quality of agreement increases for larger elasticity numbers (larger number of samples).

References

[1] Soligo, G., Roccon, A. and Soldati, A., 2019. Breakage, coalescence and size distribution of surfactant-laden droplets in turbulent flow. J. Fluid Mech., 881, 244–282.

[2] Garrett, C., Li, M. and Farmer, D., 2000. The connection between bubble size spectra and energy dissipation rates in the upper ocean. J. Phys. Oceanogr., 30(9), 2163–2171.

[3] Deane, G. and Stokes, M., 2002. Scale dependence of bubble creation mechanisms in breaking waves. Nature, 418(6900).

[4] Skartlien, R., Sollum, E. and Schumann, H., 2013. Droplet size distributions in turbulent emulsions: Breakup criteria and surfactant effects from direct numerical simulations. J. Chem. Phys., 139(17).

Conclusions

We studied the dynamics of a swarm of surfactant-laden droplets in turbulence using direct numerical turbulence simulations coupled with a phase field method. We considered two different surface tension values (Weber numbers) and four different surfactant strengths (elasticity numbers).

To characterize the effects of surfactant on the droplet morphology, we studied the resulting droplet size distribution. We found that the resulting droplet size distribution scales approximately with the inverse of the droplet volume. Specifically, the calculated droplet size distributions follow the theoretical scaling of the 10/3 power law proposed by Garrett et al. (2000), which has also been confirmed by previous experimental (Deane & Stokes 2002) and numerical (Skartlien et al. 2013) investigations.

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

For more information:
Alessio Roccon - Università di Udine
roccon.alessio@spes.uniud.it

Digital Twin to improve vibro-acoustic comfort in industrial work environments

FORSAL project to develop and validate a new prototype of digital twin

by Nicola Gramegna
EnginSoft

This article discusses the use of an innovative digital approach to study vibro-acoustic comfort in which simulation models are integrated with real data sources in the work environment (Digital Twin) and are then validated in the field with data collected in the grinding department of SAFAS, a specialist steel foundry in northern Italy. This novel digital system for Vibro-Acoustic Comfort falls under the Italian Industry 4.0 National Plan (Industry 4.0 is an international program, Piano nazionale 4.0 is referred to Italy) and the digitalization trajectories expressed in the Veneto Region's RIS3 objectives. The first prototype developed in this project by EnginSoft, while tested and validated in a foundry, has wide transversal value for all sectors with interactions between humans and machines where the study of vibro-acoustic comfort is strategically important in selecting machines and robots that may impact human health and environmental sustainability. The system also has other advantages with excellent market potential that arise from monitoring the stability of vibro-acoustic safety indicators for machine operators and reviewing and maintaining equipment that may tend to emit elevated levels of noise due to wear and/or breakage.

The industrial case: the grinding department of a steel foundry

SAFAS is a specialized steel foundry based in the Veneto region of northern Italy. Its mechanical workshop was used as a reference scenario to create a real-time environmental noise detection system integrated with simulation tools for sound pressure, in order to improve the vibro-acoustic comfort of users. Fig. 1 shows the plan of the workshop floor, highlighting the main sources of noise, which are located in three adjacent and communicating cabins (Box1, Box2 and Box3), and consist of two types of grinding wheels (pneumatic and electric), where metal components are processed. The grinding wheels in the cabins can work both simultaneously and individually. The workshop environment is also affected by diffuse background noise due to the presence of overhead bridge cranes used to move metal parts around the foundry.

The diagram in Fig. 1 shows that there are three cabins (boxes) specifically for grinding and finishing operations with electric and pneumatic tools. Two microphones were placed in these work areas and another three were placed outside the cabins at different distances from the noise sources.

The noise monitoring system has two components:

- one based on real-time measurements taken either with smartphones that allow both geolocation and measurement of sound pressure at different points, or with advanced microphones that can be located in strategic points of the building. Accuracy of localization and in signal transformation has motivated the adoption of the latter solution, which also guarantees compliance with current legislation on privacy and the protection of personal data;
- one to model and simulate the sound waveforms produced during processing in order to design environments that are safer and more comfortable, in terms of workplace regulations. The sound pressure simulation system requires the noise sources to be characterized both in intensity and frequency and the modeling of the wave type based on the tools in use and the materials involved.

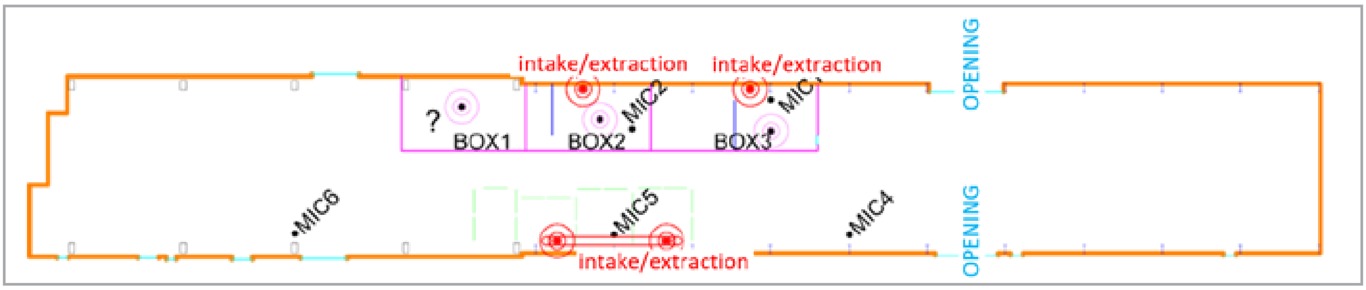


Fig. 1 – Floorplan of the mechanical workshop: cabins and microphones location

As with any industry 4.0 application, the tools used for work in the designated areas were monitored to understand timing and any malfunctions that affect production efficiency at this stage of the production chain. The duration of measurement acquisition remains limited in time and does not represent continuous "monitoring" of work unless otherwise required by health and safety requirements. The development of a system to simulate the noise perceived by workers was considered to be of considerable interest and necessary to create safer and more comfortable working environments. The project therefore focused on defining a system and a scalable and systematic process to model and simulate levels of sound pressure in different scenarios efficiently and quickly.

The physics of the problem

As is well known, the decibel [dB] is used in all fields of audio technology to express signal levels, and differences between levels in sound pressure, power, voltage, etc. The decibel is useful because it allows a narrow range of numbers to be used comparatively to express large quantities that are often difficult to handle. It also makes sense from a psychoacoustic point of view because it is directly proportional to the effect of auditory sensory stimuli.

This study of the physical phenomenon and its expression in the numerical simulation model took into account the pressure, power, and intensity of the sound as well as its diffusion and absorption by the materials present that absorb energy according to their own characteristics.

When a pressure wave encounters a body, part of the incident wave is absorbed by that field and part is reflected. The reflected pressure must be added to the incident pressure; this is called a " diffuse" or "reverberant" field. When the source stops emitting sound waves, there is a so-called sound tail, which means that the reflected field persists for a short period and then progressively attenuates until disappearing completely. Reverberation time, or the time that elapses from when the sound source is turned off to when the sound level becomes negligible, was introduced to evaluate this phenomenon. Finally, it should be noted that, in a diffuse field, one measures the acoustic pressure.

The frequency composition of sounds can be analyzed using different methods based on subdividing (filtering) the content of sound energy into bands, i.e. into predefined frequency intervals. Octave and one-third octave filters are widely used because they best approximate the sound sensation caused in critical bands, so the field measurements were processed and converted into one-third octave frequencies.

In psychoacoustics, a critical band is defined as the range of frequencies within which two pure tones occurring simultaneously cannot be perceived as distinct due to masking. This implies that as the frequency increases, the ratio between the amplitude of the affected region and the range of frequencies that can be perceived in that interval is drastically reduced. This explains very naturally why critical bandwidth grows as the frequency increases.

Actual measurements in the work environment

It is important to note that employers are required to take into account the attenuation provided by hearing protection devices when assessing their efficiency, as well as compliance with the Exposure Limit Value, as required by Art. 193 of Italian Legislative Decree no. 81/08. This assessment must be conducted using the UNI EN 458:2005 technical standard, which contains recommendations for the selection, use, care, and maintenance of ear protectors.

According to preventive best practices, individual preventive devices must be used once it has been ascertained that, despite having implemented all the technical, procedural, or reorganizational preventive measures (such as reducing risk at the noise source, or modifying work methods), the worker still faces unavoidable "residual risk". Individual hearing protection devices must comply with the provisions of Italian Legislative Decree no. 475/1992, must be adequate to the task and avoid "overprotection" for instance, and should not themselves be the cause of new risks, all the while taking into account the ergonomic and health requirements of the worker, as well as the nature of the work performed.

After the measurements were taken, it was confirmed that the FORSAL Project's system for collecting noise data in factories did not conflict with regulations protecting worker privacy. The measurements taken for this research lasted only those few seconds necessary in order to establish the appropriate time span to design a workplace detection system capable of monitoring changes instantaneously using new 4.0 technologies. Consequently, this system cannot in any way be used to remotely monitor or profile workers. Moreover, the measurements taken, which were used to produce an acoustic map of the working environment and to simulate the propagation of noise in that environment, were confirmed as useful for the preparation of the Company Program for the Reduction of Exposure (P.A.R.E.) and, more generally, for planning the technical and organizational measures to contain the risk (see UNI 1134 7:2015).

The sound surveys were conducted with UMIK-1 miniDSP microphones connected via USB to a laptop PC and controlled using open-source software, RoomEQWizard (REW), version 5.19. For each survey, the measuring apparatus produced two text files generated by the operator following a manual procedure. This procedure consisted of capturing approximately 10s of sound using the REW/RealTimeAnalyzer (RTA) in one-third octave mode, uploading the analysis, and then exporting the measurements in table format to a text file.

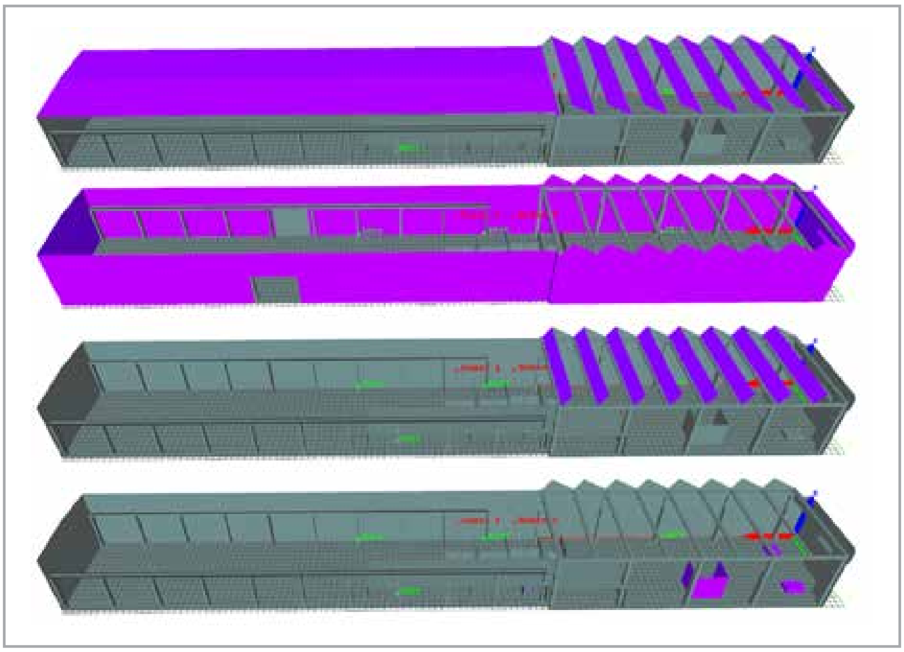


Fig. 2 – Definition of work environment or machining shopfloor and materials: insulating panels, PVC tarpaulins, beams, roofing, flooring, walls, ceiling, windows, tables and partitions, and openings to the outside.mechanical workshop being studied

The necessary measurements were taken inside the cabin workstation identified as BOX3. The difference between an outdoor measurement and an indoor measurement basically implies an alteration in the measured values. In the latter case, in fact, the sound field at each point includes both the sound emanating directly from the source, and the reverberation resulting from the interaction with the surrounding environment, which is absent or negligible outdoors.
The proportion of the two components depends on the relative positions of the detection point and the noise source, the position of the noise source within the enclosed space, and lastly, the geometry of the enclosed space and the acoustic characteristics of the surrounding surfaces.

Digital twin of vibro-acoustic comfort

The study consists of a comparison between some measurements of noise levels detected in an industrial environment and the results obtained from the elaboration of the corresponding predictive particle-tracing model using the open-source code I-SIMPA/SPPS.

Like all other software that implements predictive models based on the light analogy, I-SIMPA/SPPS only allows the simulation of the impulse response with an energy approach. To correlate the simulated results with the measured sound levels, which are considered stationary, it is assumed that the integral in time of the impulse response equals the sound level of the respective stationary source, according to the backward integration method devised by Schroeder in 1975, according to which there is no direct relationship but there is an identity between the mean square decay of the noise and the integral of the square of an impulse response in the same spectrum.

In particular, the I-SIMPA/SPPS forecasting model was used for an industrial cabin used for the grinding of steel castings, where operators are exposed to high levels of high-frequency noise. An approximate virtual model was therefore defined which takes into account the main sound sources (background noise and tools) located inside the cabin and considers the sound absorption and soundproofing capacity of the structures inside the cabin, for the purpose of validating the numerical model by comparing it with some measurements of the typical noise generated in the actual conditions.

The first step in defining the virtual model consists of modeling the geometry of the inside of the cabin. The simulator selected allows geometric files to be imported directly in the most common and versatile formats such as STL, so the geometry can be defined starting from the floor plan of the cabin using any program for 3D computer graphics.

Creating a project with I-SIMPA requires the following preprocessing steps (Fig. 2):

1) import the 3D geometry of the scenario in a compatible triangularized format;
2) assign acoustic material properties to each surface element of the imported scenario. Data can be imported from an internal or external database, or from one defined by the user;
3) define the characteristics of one or more sound source points;
4) define sound receptors that can be either points or surfaces;
5) characterize some of the environmental data, such as atmospheric conditions and meteorological parameters, and possibly define noise dispersion zones called fitting zones.

Once the model has been configured, just a few other parameters related to the generation of the calculation grid and the SPPS solver's selected calculation approach need to be assigned to start the calculation. The postprocessor for analyzing the results is also integrated into I-SIMPA. At present, since the interface was initially developed for energy models, there are acoustic parameters available for indoor environments, such as reverberation time and others (as

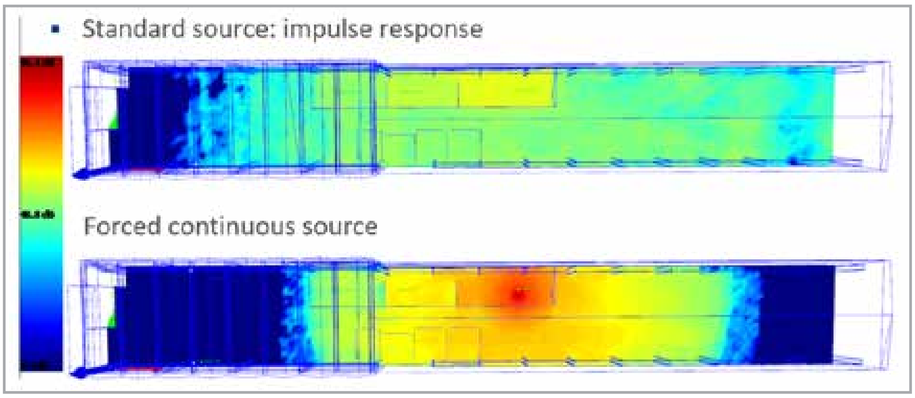


Fig. 3 – Pulse and continuous source results of the sound map [dB] inside the cabin

walls also implies the estimate of the soundproofing capacity in addition to the absorption coefficient (which describes the soundproofing performance). Therefore, in the numerical model, the panels that form the boxes of the grinding area were considered transmissive, and the spectral distribution of soundproofing power was estimated from similar products, in order to obtain the same variation in sound pressure level (SPL) as simulated and to verify that the abatement is close to the stated level of 20 [dB(A)].

defined in ISO 3382-1). Also included are the echogram, Schroeder integration curve, intensity vector field, and time and cumulative sound maps. All results can be viewed in integrated spreadsheets, exportable in CSV format.

The simulation model succeeds in replicating and mapping situations with a standard pulse source and a forced continuous source (Fig. 3).

For the simulation, five configurations were defined from the prototype model, each of which corresponded to one of the scenarios for which measurements were actually taken. The acoustic characterization of the materials was based on direct observation and by estimating typical values from widely shared databases on the web or from data for similar products. In particular, the sound-insulating panels that separate the grinding area from the rest of the shed are made of modules for which the declared sound pressure level reduction is 20 [dB(A)]

As far as the characterization of the sound sources is concerned, a distinction must be made between the tools being studied (Fig. 4) and the background noise (Fig. 5) characteristic of the environment, when no tools are operating. The tools consist of electric and pneumatic grinding wheels for which the manufacturers' technical data sheets declare a sound power of 94±3dB(A) and 95dB(A) respectively, without however mentioning the emission spectrum.

Five numerical models were developed corresponding to each of the measured scenarios. The characterization of the internal

Model calibration and validation

Each of the five simulations corresponding to the respective scenarios measured required little computing time (a few minutes) to obtain realistic results (i.e., considering a sufficient number of sound particles), especially when compared to the hours of machine time that would have been required using a classical CFD approach.

Although the measurements were affected by notable disturbances, the comparison between the different scenarios showed that the method is very reliable since the spectral distribution values of the real and simulated SPL are comparable. A good correspondence is found in particular for all the scenarios measured with an active grinding wheel in the cabin both for the measurements at the microphones in the box and for MIC4.

The environmental (background) noise scenarios measured showed a sufficient correspondence with the measurements at the microphones outside the boxes, but a poor correspondence at the microphones in the boxes considering the transmissivity of the soundproofing panels, probably due to excessive disturbances during the measurements.

In conclusion, the I-SIMPA/SPPS predictive model enabled the SPL distribution of the noise inside the shed to be adequately represented, making it possible to consider both the reverberant field and the effect of dispersion, absorption, and isolation of the various devices inside the shed, thus obviating the poor reliability of the measurements taken. It is therefore possible to extract quantitative information from the validated model in order to define the transfer function that

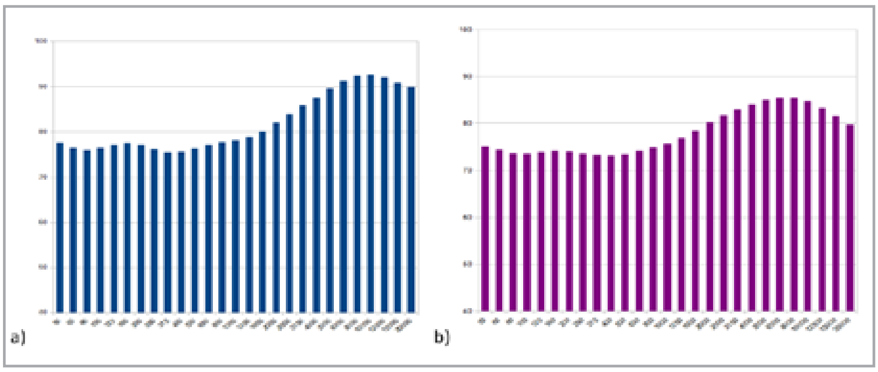


Fig. 4 – Characterization of the electric (a) and pneumatic (b) grinding wheels in BOX3

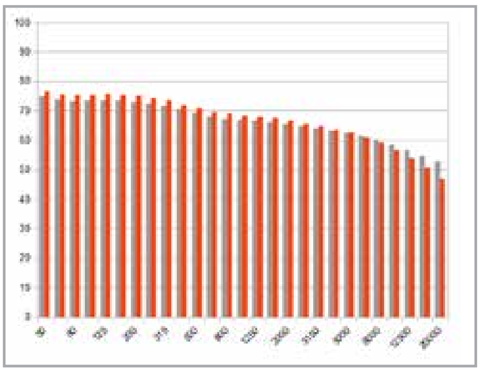


Fig. 5 – Example of background noise calibration with spectral distribution detected by MIC5: real (gray) and simulated (red)

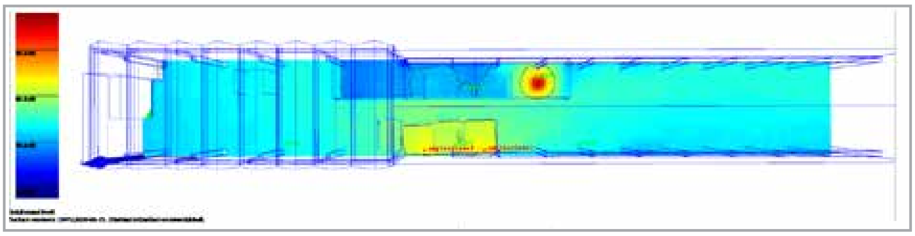


Fig. 6 – Scenario 2: electric grinder active in BOX3

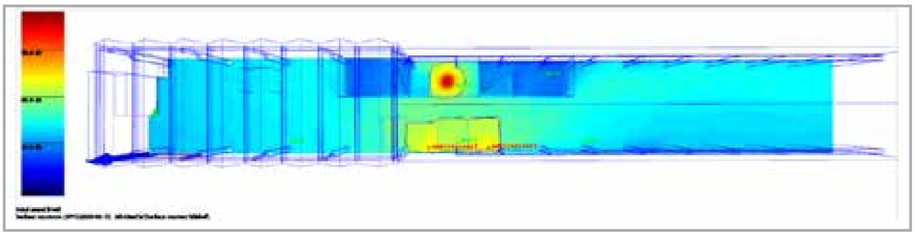


Fig. 7 – Scenario 5: pneumatic grinder active in BOX2

represents the acoustic impedance of the shed in question, allowing the noise in the working environment to be transduced in real time as a function of the distribution of the noise sources considered.

Conclusions and future prospects

The FORSAL project has conducted some investigations and the preliminary study of the innovative Vibro-Acoustic Comfort digital platform to understand how simulation models can be integrated with real sources in a working environment.

Although it was developed, tested, and validated in a foundry context, the first prototype of the digital platform developed in this project has a wide cross-sectional value in work environments where human-machine interactions and vibro-acoustic comfort are of strategic importance in the adoption of machines and robots that impact human health and environmental sustainability. The advantages of using such a platform can be found both in the design phase of the factory layout and in the testing and periodic verification phases for authorities according to the health and safety regulations for workers.

In the first case, as frequently happens with the use of numerical modelers, the design offers the benefit of evaluating several different scenarios and configurations before implementing the optimal solution. Materials and factory layouts can be selected and implemented to meet all production requirements with a significant time and cost advantage. In this case, the reference person will be the plant design manager.

Testing and periodic verifications required by law become continuous and streamlined if sound detectors are deployed and connected to a simulator that provides 3D maps in near-real time (Digital Twin). The point signal detected by a microphone will no longer be the only signal available to warn of risky situations.

The planned distribution throughout the work environment captures the combined effect of different sources, even anomalous ones, for the benefit of both workers and visitors safety. Archiving data, maps and reports can streamline verification procedures and improve the comfort levels which can also be promptly restored in the event of an alert.

For EnginSoft, the innovative Vibro-Acoustic Comfort digital system, which integrates simulation models (Digital Twin) with real sources in the workplace represents a new Digital Twin application, useful for providing consulting services to its customers during the design phase, or as a support for a continuous monitoring service and periodic checks on workplace vibro-acoustics.

About Forsal

The FORSAL project was approved by the Veneto region of Italy as part of the POR-FESR 2014-2020 program with reference to Axis 1 and Action 1.1.4 "Support for collaborative R&D activities for the development of new sustainable technologies, new products and services". The two Regional Innovative Networks (RIRs), SINFONET (http://www.retesinfonet.org/) and IMPROVENET, have brought together a number of industrial partners and university departments to research, develop, and implement a series of innovative solutions aimed at improving the working conditions in foundries where the interactions between humans and machines affect health and well-being. As a result, the FORSAL project was successfully concluded in November 2020 and its outcomes have significant industrial, economic, and social value. The FORSAL project falls within the Smart Manufacturing sphere, making use of micro/nano-electronic enabling technologies (interactive robotic management systems for equipment), Advanced Materials (high-performance light alloys for the creation of a prototype and materials for ad hoc tools made for the aforementioned processes), and Advanced Production Systems (introducing robotic systems into the foundry). The main driver for the innovation is Active Ageing and the problem of occupational diseases caused by the absence of vibro-acoustic comfort in the case of a medium-large foundry and its ancillary activities. One of the many project objectives specifically studied grinding and finishing operations, which are conducted manually by skilled workers, to mitigate the potential risk of occupational diseases that can be caused by ergonomic problems and poor vibro-acoustic comfort.

Acknowledgements

Special thanks to the SPRING Consortium, leader of the FORSAL "Robotized Workshop for Workers' Health" project, and to all the project partners, in particular SAFAS and the University of Padua's DTG department for their fruitful collaboration.

For more information:
Nicola Gramegna - EnginSoft
n.gramegna@enginsoft.com

by Annalisa Chiocchetti
Università del Piemonte Orientale (UPO)

FLAMIN-GO: Joint-on-Chip (JoC) to optimize the management of rheumatoid arthritis (RA)

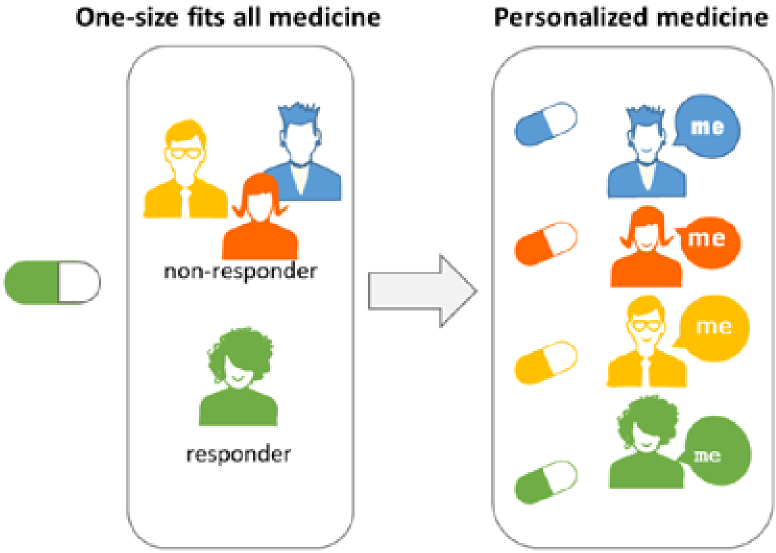
A migratory journey led by FLAMIN-GO for all RA patients

While most people can walk or bike to work, recharge at the weekend with a run, or participate in outdoor activities with family and friends, patients with rheumatoid arthritis (RA) have physical limitations that make activities of daily living difficult and, therefore, they may also have impaired emotional and social functioning. RA is a chronic, autoimmune disease that involves the joints and affects up to 1% of the total population. Despite its deleterious effects on patients' quality of life and related social and economic impacts, to date, the management of RA follows conventional treatment strategies that revolve around a trial-and-error approach. To counter the latter, ***FLAMIN****-GO plans to develop an innovative synovia-on-chip (SoC) model that will help fine-tune the management of RA using available treatments and also help discover new treatments.*

Have you ever wondered why you can move your hands, knees, and feet freely, but can only move your spine slightly? Well, although the joints allow movement, they are not all created equally in terms of the degree of movement.

The "hero behind the scenes" of maximum body movements is the synovial membrane that lines some joints and provides protection and lubrication, which explains why freely moving joints are also known as "synovial joints".

In light of the essential role of synovial membranes, the consequences of any damage they may experience are self-explanatory and mainly include pain and immobility. One major type of joint disease involving the synovial membrane is rheumatoid arthritis (RA), which affects up to 1% of the total population and is caused when the immune system perceives the synovial membrane as a foreign body, which triggers autoinflammatory and autoimmune reactions. RA is characterized by its chronic, systemic nature and manifests as joint pain, lack of balance, and joint deformity. Advanced stages of the disease are marked by loss of physical function, including the inability to perform daily activities, and may also be associated with other health risks, such as cardiovascular disease and osteoporosis.



In addition to being a serious condition that adversely affects patients' quality of life, RA poses a challenge to healthcare providers for several reasons. First, the direct etiology remains unknown, making prevention of RA virtually impossible. Second, it is difficult to diagnose, especially in the early stages, because the signs and symptoms are not distinguishable from those of other inflammatory conditions affecting the joints. Third, the management of RA is not simple, rather it is practically challenging.

Management of RA follows the treat-to-target strategy to achieve a state of minimal disease activity or remission and ultimately prevent irreversible joint damage. Although the treatment outcome is well-defined, the treatment strategy is not. Several immune (T cells, B cells, dendritic cells, macrophages, and neutrophils) and non-immune pathways (fibroblasts and chondrocytes) cells are involved in the pathophysiology of RA and, therefore, patients respond differently to currently available treatments.

Some respond to first-line treatment, which is either one or a combination of disease-modifying antirheumatic drugs (DMARDs), with or without adjunctive anti-inflammatory agents; whereas others exhibit resistance and require second-line treatment, which is one of the targeted DMARDs, and even then, not all patients respond to treatment. However, assignment of treatment(s) is completely random and occurs through trial and error, resulting in delayed initiation of the right treatment, low levels of improvement, high risk of disability, and unnecessary exposure to potentially toxic drugs, as well as significant social and economic implications.

All of the above, highlights the need to understand the heterogeneity of the pathophysiological pathways associated with RA, to detect biomarkers predictive of clinical safety and efficacy of RA treatment strategies, and to discover new therapeutic targets. ***FLAMIN***-GO (***F***rom pathobio***L***ogy to synovi***A***-on-chip: driving rheu***M***atoid arthritis to the precision medicine ***GO***al) is not only working to resolve these three issues but aims to do so with an innovative, personalized technology that skips in vivo testing.

The current habitat of *FLAMIN*-GO

Reviewing "old" and "recent" scientific literature on RA, it is possible to clearly identify an urgent need to better understand the pathophysiology of RA, and to find new treatment options. These two unmet needs account for the largely empirical nature of RA management, and call for a shift from "one size fits all" to precision medicine. Applying precision medicine to RA will uncover new general and individual factors associated with its pathophysiology, and thus help tailor treatment strategies to the specific needs of each patient.

The main aim of this research project is to develop a personalized, next-generation synovia-on-chip (SoC) using patients' biopsies to replicate the personalized and complex nature of RA joints, which will enable the development of patient-specific clinical trials on a chip (CToC), with clear optimization of RA management.

The migration of *FLAMIN*-GO

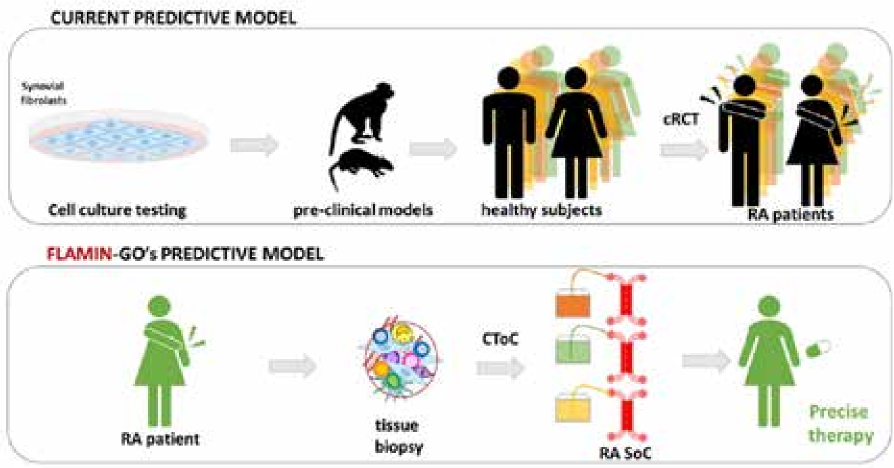
Spread your wings: a fantastic journey is about to begin

Current research and development (R&D) methodologies are out of step with technological advances. The available in vitro and preclinical in vivo tests are time-consuming and do not accurately replicate the complex environments and interactions of the human body. In addition to this, increased interest in animal welfare calls to minimize the number of animals used for scientific experimentation and research.

Consequently, in vitro models, such as micro-engineered chips that replicate human organ systems are considered a reliable alternative. These chips are called organ-on-chip (OoC) and are made from living human cells with microfluidic channels. They recreate the natural physiology of the human body and, as a result, allow scientists to examine changes at the cellular and molecular levels in normal and pathological conditions, as well as in response to triggers, such as drugs, chemicals, or toxins.

Keep on flapping: halfway through the journey

In order to apply OoC technology in RA, ***FLAMIN***-GO will design a multi-compartment model to mimic RA joints using 3D cell cultures, among other innovative technologies that will be developed during the research project. This model will coculture all relevant joint tissues into the appropriate extracellular matrix (ECM). It should be noted that the final OoC model will be developed and validated incrementally using tissues collected from RA patients, in order to ensure that the building blocks of the new in vitro system are individually functional and adaptable.



Prepare for landing: the final destination is in sight

Finally, we come to the final and most exciting part of ***FLAMIN***-GO's journey, the clinical validation of the SoC. To do this, the SoC will be tested for disease relevance, biomarker/target discovery, and drug testing, and the results will be compared to those obtained in concurrent randomized controlled trials (RCTs). Using the data collected, it will be possible to detect correlations between variations in the evaluated biomarkers and response to treatments, which will help healthcare providers understand the basis of the significant heterogeneity of response to treatments among RA patients and make more informed treatment decisions, and thereby begin the right treatment earlier in the course of the disease, improve prognosis, and reduce healthcare costs. In addition, new potential targets for the treatment of RA may be discovered, which may lead to the development of new drugs.

The glories of FLAMIN-GO's final destination

The approach briefly described above helps solve the main difficulties associated with conventional RA treatment strategies by simulating the interfaces and interactions among the tissues, cells, and ECM of RA joints. Therefore, ***FLAMIN***-GO will assist in:

- Capturing significant differences among RA patients due to various factors, such as genetic diversity, ethnicity, gender, and age, as well as better understanding disease-specific pathways and discovering new ones
- Testing the safety and efficacy of currently available drugs for RA using the patients' actual biopsies which, in combination with data on pathophysiological differences, helps uncover the underlying factors that drive variations in drug responses
- Detecting potential drug repurposing (DR) as a result of multi-omics analysis of the secretome, particularly of extracellular vesicles, which will help increase the number of evidence-based treatment options for RA
- Respect animal welfare and the 3Rs (replacement, reduction, and refinement) in research and testing, and reduce the number of human participants in clinical trials.

As we move through our challenging journey, we hope to help remove barriers to OoC technologies, boosting European competitiveness in the biomedical and healthcare sectors, as well as improving awareness of medical regulatory policies and requirements for developing and commercializing medical products. However, the ultimate goal will be to improve the quality of life of RA patients and to reduce public and private healthcare costs.

Acknowledgement

This project has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No 953121. This presentation reflects only the author's views and the Commission is not responsible for any use that may be made of the information contained herein.

For more information:
Annalisa Chiocchetti
Università del Piemonte Orientale (UPO)
annalisa.chiocchetti@med.uniupo.it

Developing innovative equipment to detect volatile hazardous substances

Using technology, IOT, AI and big data to pre-emptively thwart unforeseen deadly terrorist attacks

by Elisabetta Pasqualotto and Matteo Scaramuzza
EXA s.r.l.

Security can be regarded as both a need and a necessity, and these feelings are found whenever we perceive that something – or someone – we consider valuable is threatened. Since this is ubiquitous, security from risks has emerged as a social value to be pursued and accomplished by every possible means, irrespective of whether what is endangered is property, people, or the status quo. In fact, the main purpose of security activities is to prevent those acts that may impair the perceived value of threatened objects, independent of their inherent nature.

This is a guiding principle that has led to the development of numerous national guidelines and transnational treaties in past years, making preventive security activities a common responsibility among countries that share efforts to address this critical task through cutting-edge technological advancements.
The VOLPE project aims to be a response to the growing demand for innovative security solutions to react preventively to imminent and invisible threats. The project goal is to design, develop and optimize a completely contactless explosive trace detector to continuously monitor risky and hazardous volatile compounds in the air to safeguard both citizens and critical infrastructure such as embassies, airports, museums, freight ports, concerts, post offices, and any other location or event that is likely to present a significant possibility of deadly terrorist attacks. These unforeseen deadly actions are particularly difficult to monitor because the tactics they use are constantly changing and improving, often building on precursors, i.e., common substances that only became harmful when mixed e.g., fertilizers, fuel, and bleach. Therefore, there is a growing need for smart detection devices that can simultaneously monitor multiple chemical agents, and process complex correlation analyses from large raw datasets.

The challenge of early detection of "dirty" attacks is being tackled by VOLPE with an innovative trace detector optimized to continuously monitor air samples collected from luggage, parcels, and even human hands, without any mechanical interaction with those surfaces, i.e., no swipes or swabs for sample collection. The air samples are analyzed with an advanced optoelectronic system designed to sensitively and accurately query a matrix sensor in which several different polymers are deposited using state-of-the-art, low-volume manipulation equipment to achieve parallel and simultaneous detection of different target molecules within the same sample. When the air samples come into contact with the matrix sensor, each different spot in the matrix selectively reacts with a single target molecule, thus creating a unique impression or "fingerprint" of the sample. Each fingerprint is then analyzed by

2014/2020 POR FESR / REGIONE DEL VENETO
UNIONE EUROPEA
REGIONE DEL VENETO

neural network algorithms and compared to a reference database to identify and quantify the concentration of hazardous materials in the air.

The VOLPE project is funded by the Veneto region of Italy in the program "POR FESR 2014/2020 Asse 1 Azione 1.1.4" and is being developed in partnership by three companies:

- EXA, a SME located in Padova (Italy) and focused on research, development, and innovation in civil security. EXA is the project leader and owns the authorship of the idea for the innovative contactless trace detector. Its role in the project is the design and characterization of the polymer-based matrix sensor, performance evaluation of the various prototypes, and business development strategy.
- EnginSoft, one of the leading technology transfer companies in the field of Simulation Based Engineering Science. Its role in the project is the modelling and optimization of the thermo-fluid dynamic behavior of the mechanical front-end of the equipment, to improve both the energy transferred to the analyzed surface and the volume of gas collected for the creation of reliable samples.
- Buson, a company located in Pernumia (Padova, Italy) and specialized in high precision mechanics and automated manufacturing. Its role in the project is the fabrication of the mechanical frontend for gas sampling, the design and implementation of the automatic control of the equipment, and the design of the entire machine (www.buson.it).

A first prototype has already been developed and tested at Marco Polo airport in Venice where it has been integrated into a regular X-ray checkpoint to boost the detection capability of standard instruments: this was enabled by the specific design of the prototype, which focused on improving the conventional security scanner with complementary detection capabilities, e.g., for explosives and biological threats.
During these real-world tests, the technology was shown to meet important requirements for explosive trace detection, namely:

- high sensitivity, due to the specific engineering of both the matrix sensor and the optoelectronic readout system, which are carefully calibrated to detect low quantities of target explosives on the baggage surface, i.e., less than 10ng/mm^2;
- high selectivity, which signifies a high rejection of possible interfering molecules such as airplane fuels or perfumes, with a corresponding reduction in the false positive rate;
- short analysis time, given that the system can scan and analyze an entire bag in less than 20 seconds.

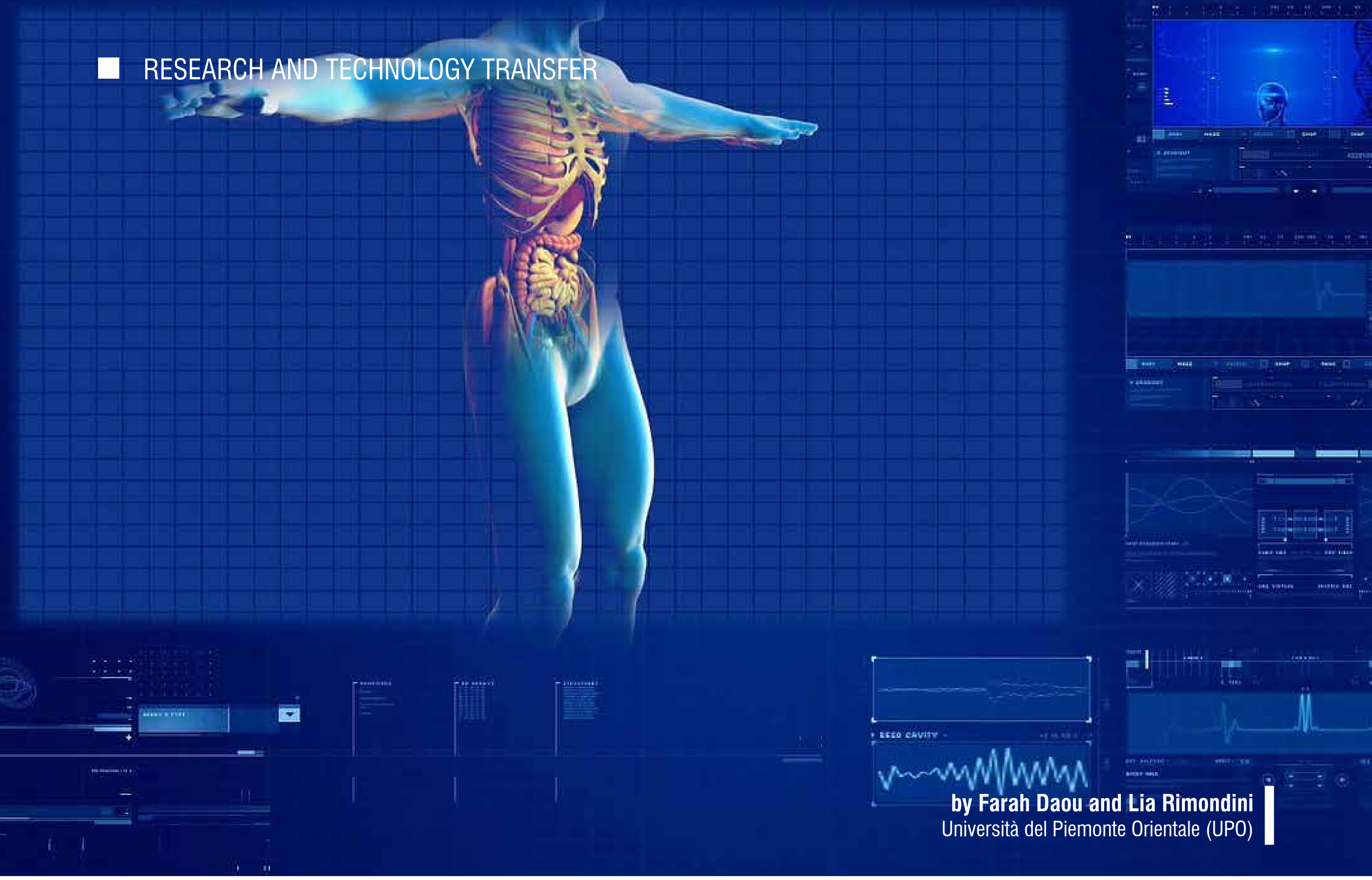
With these excellent preliminary results, the VOLPE project's goal is the further engineering and industrialization of this technology.

This requires additional phases of performance optimization and component miniaturization to achieve other fundamental criteria for the development of a new instrument that can overcome all the technical and commercial barriers associated with the use of conventional screening techniques, e.g.:

- a stand-alone system for continuous monitoring of gas samples without the need for skilled user supervision or direct operator control;
- a compact, cost-effective solution that can be used to improve the detection capabilities of existing security checkpoints;
- digitalization and IOT management in accordance with the Industry 4.0 program.

The expertise of the VOLPE project partners is critical to achieve these challenging goals, and with all the shared efforts, optimized pilot instruments could be installed and tested in several other critical infrastructures supported by a promotion and marketing phase for the upcoming market launch of the product.

For more information:
Elisabetta Pasqualotto or Matteo Scaramuzza
amministrazione@exa-project.it



by Farah Daou and Lia Rimondini
Università del Piemonte Orientale (UPO)

PREMUROSA: "To take care with care"

A new chapter begins for people with musculoskeletal conditions

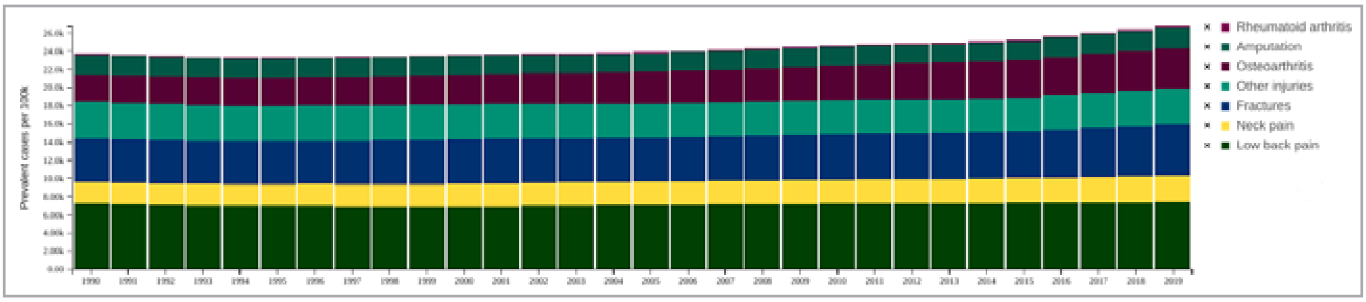
How many times have you looked up a novel therapy that captured the headlines? And how many of those have reached clinics? By answering these two questions and reading through this article, you will undoubtedly notice that PREMUROSA is different from all other research projects on many fronts, from combining regenerative medicine, precision medicine, and multi-omics technologies, to being intersectoral and multinational. This latter characteristic makes this research project a life-changing turning point for more than 22% of the worldwide population.

Musculoskeletal conditions include a broad range of acute and chronic conditions affecting approximately 1.71 billion people globally. They are a major contributor to low physical functioning, reduced ability to work, years lived with disability (YLDs), and the need for rehabilitation, and thus represent a substantial social and economic burden. Based on recent data, low back pain, neck pain, fractures, and osteoarthritis are the most prevalent conditions, and have been showing an upward trend over the last 29 years (see Fig. 1).

Initially, treatment of musculoskeletal conditions was limited to pain and surgical management, in addition to physical and occupational therapy. However, in recent years, "regenerative medicine" has been added to the management options and is currently paving the way not only for "treating" the symptoms of musculoskeletal conditions, but also for "curing" them. Regenerative medicine reverses tissue damage caused by musculoskeletal conditions by utilizing a combination of cells, growth factors, and/or biomaterials that repair or replace tissue function.

Despite the tremendous advancements in regenerative medicine, there still is a long way to go to improve the efficacy, safety, scalability, and contextualization of the regenerative technologies applied to musculoskeletal conditions, and this explains why only a limited number of therapies have been approved for clinical use in musculoskeletal regeneration by the Food and Drug Administration (FDA) and the European Medicines Agency (EMA).

This is where PREMUROSA comes onto the scene with an innovative two-tiered approach, in an attempt to translate the



Fig, 1 – Global prevalence of musculoskeletal conditions per 100k across all ages for both genders from 1990 until 2019

discoveries in regenerative technologies into clinical practice. Firstly, it will model “precision regenerative medicine” for musculoskeletal conditions, whereby the addition of “precision medicine” using genetic, lifestyle, and environmental data allows both the personalization of regenerative technologies and the prevention of musculoskeletal conditions. Secondly, by going beyond “genomics” and incorporating “multi-omics” technologies to impart an immense quantity of data, it will provide a comprehensive understanding of the pathophysiology of musculoskeletal conditions and improve clinical decision-making and public health policy-making processes.

The scope of the project

The principal aims of PREMUROSA (an acronym for PREcision medicine for MUsculoskeletal Regeneration, prOstheticS, and active Ageing) are to advance the use of regenerative medicine, precision medicine, and multi-omics technologies as a single entity in the prevention, diagnosis, and treatment of musculoskeletal conditions, and to train a new generation of scientists through an unequaled, intersectoral collaboration between academia, industry, and hospitals.

PREMUROSA builds on the findings of two previous projects (MP1005 NAMABIO and MP1301 NEWGEN COST) that included most PREMUROSA participants, and that highlighted the significance of optimizing and personalizing the treatment of musculoskeletal conditions in order to achieve four major scientific objectives, namely:

1. To gain a deep scientific understanding of the role of the extracellular matrix (ECM) in musculoskeletal regeneration, and of the interactions between biomaterials, ECM, cells, and tissues, while taking into account patient morbidity, comorbidity, immunological status, age, and multi-omics data;
2. To develop substantially new in vitro assays to test biomaterials in conditions that closely mimic clinical reality, and to forecast personalized healing and regenerative response;
3. To develop new in silico methods and a decision support system (DSS) to assist healthcare providers in preventing, diagnosing, and treating musculoskeletal conditions in a personalized manner, while reducing the use of animal models;
4. To generate a biologically integrated, multi-omic “signature” to predict the patients’ healing and regenerative response.

Project activities

To fulfill its aims, PREMUROSA created a workflow with structured and predefined activities that are taking place in parallel, as discussed below and summarized in Fig. 2.

A. Biomaterial/tissue interactions:

To understand the role of biomaterial characteristics in musculoskeletal regeneration, and to produce reference biomaterials, several biomaterials with numerous physical and chemical properties are being evaluated together with human adult mesenchymal stem cells (MSCs) and released extracellular vesicles (EVs) by means of cell stress analysis, transcriptomics, and proteomics.

The angiogenic and neurogenic role of EVs is also being tested. Moreover, the role of protein interactions in the extracellular matrix (ECM) with biomaterials and MSCs in the regenerative process, and the role of glycoprotein interactions in the ECM with cells in low back pain (LBP) are being studied. In addition, cartilage-engineered microtissues used to regenerate the cartilage of inflamed, injured, and arthritic joints are being evaluated in terms of pathological innervation as it is directly linked to joint pain, and in particular, the effect of a pro-inflammatory environment on innervation is being evaluated to identify candidate molecules capable of modulating the innervation of cartilage-engineered microtissues. Finally, the biocompatibility of the biomaterials developed by PREMUROSA will be evaluated by studying their interaction with immune system cells.

B. New in vitro assays:

Current in vitro assays fail to simulate biological processes, and this is more pronounced for musculoskeletal regeneration, which involves interactions among cells, the components of the extracellular matrix (ECM), and biomaterials. To bridge the gap between in vitro and in vivo assays, PREMUROSA aims to develop a portfolio of advanced 3D in vitro assays. Four different 3D in vitro assays are being developed, namely: vascularized bone organ models; osteoblast/osteoclast co-cultures; osteoblast/chondrocyte co-cultures; and osteoblast/mesenchymal stem cell bone models. All of these will be validated using in vivo pre-clinical

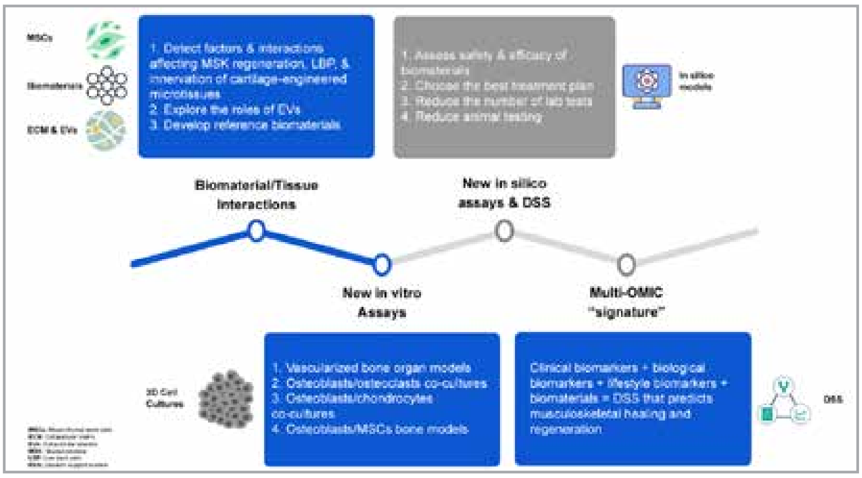


Fig. 2 – PREMUROSA's four scientific objectives and their components

and clinical data, and they will then be used to test biomaterials to improve risk analysis and regeneration. PREMUROSA is also developing the bioreactors required to test and validate the new 3D assays, and the 3D microfluidic assays to evaluate the ability of cartilage-engineered microtissue for joints to promote or avoid peripheral tissue innervation.

New in silico assays and the decision support systems (DSS):

Although healthcare innovation is accelerating at an unprecedented rate, the research and development (R&D) processes are falling behind. In an attempt to replace animal testing in research and reduce the R&D cycle time and cost, in silico methods will represent a substantial part of the work done by PREMUROSA. Data from experiments will be added iteratively to build mathematical models, and then to build a decision support system (DSS) to guide healthcare providers in assessing the safety and efficacy of the biomaterials and medical devices used in musculoskeletal conditions, and therefore, in choosing the most appropriate treatment plan and reducing the amount of unnecessary laboratory tests. The DSS so created will be based on "right data" and not "big data", tested and validated with clinically relevant 3D in vitro cultures, and optimized by machine learning.

Multi-OMIC "signature" for musculoskeletal healing and regeneration:

This final step is the essence of the project, since all the data acquired in the previous parts of the project will be gathered together to define a multi-omic "signature" to predict patients' healing and regenerative abilities. To this end, patients will be stratified according to clinical, biological, and lifestyle biomarkers into different groups, after which these biomarkers will be tested with biomaterials of differing physical and chemical properties. Data will be added iteratively, and those biomarkers that most accurately predict the musculoskeletal healing and regeneration of every group of patients will be selected. This will then be translated into the decision support system (DSS) to guide healthcare providers in selecting the right biomaterials, thereby moving precision medicine from the lab into the clinic.

In addition to its scientific objectives, PREMUROSA is also transferring essential soft and transferable skills, including but not limited to work ethic, research design, business development, leadership, teamwork, and critical thinking, to 13 early-stage researchers (ESRs).

When, who and where?

Launched in January 2020, the four-year program will conclude in December 2024. Work is currently underway in seven European countries: Finland, Ireland, Italy, Latvia, Portugal, Serbia, and Switzerland, across various academic, industry, and medical institutions (see Fig. 3).

Acknowledgement

This project has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No 860462. This presentation reflects only the author's views and the Commission is not responsible for any use that may be made of the information contained herein.

For more information:
Lia Rimondini - Università del Piemonte Orientale
lia.rimondini@med.uniupo.it

Fig. 3 - PREMUROSA's beneficiaries and partner organizations

ENGINSOFT

The Simulation Based Engineering & Sciences Magazine

Newsletter

Special Issue

Choose the **best way**
to gain **knowledge** today

INTERNATIONAL
CAE
CONFERENCE

SAVE THE DATE

37th INTERNATIONAL CAE CONFERENCE AND EXHIBITION

ORCHESTRATING DIGITAL TRANSFORMATION THROUGH SIMULATION

Channelling Technology, Processes and People
for Sustainable Competitive Advantage

VICENZA, ITALY
17-19 **NOVEMBER**
2021

HYBRID EVENT
www.caeconference.com